DESIGN OF A BENDING TEST DEVICE USING FINITE ELEMENT METHOD BASED ON SOLIDWORKS SOFTWARE

[1]**Febri Antami**,[*2]**Muchlisinalahuddin**,[3]**Desmarita Leni**
[4] **Ilham Alghani**,[5]**Yuni Vadila**,[6]**Reyhan Stevano**

[1,2,3,4,6]*Mechanical Engineering, Engineering, Universitas Muhammadiyah Sumatera Utara, Indonesia*
[5]*Mechanical Engineering, Engineering, Universitas Negeri Padang, Indonesia*

Author's email:
[1]*Febri_Antami@gmail.com*; [2]*muchlisinalahuddin.umsumbar@gmail.com*; [3]*desmaritaleni@gmail.com*
[4]*Ilhamalghani11098@gmail.com*; [5]*yunifadilabkt@gmail.com*; [6]*reyhanstevano55@gmail.com*

**Corresponding author:* muchlisinalahuddin.umsumbar@gmail.com

Abstract. *Bending test equipment is used to measure the strength and flexibility of a material, especially in engineering education and research activities. However, test equipment available on the market is generally expensive and less flexible. This study aims to design a bending test equipment that is more efficient, economical, and can be tested first through simulations before being manufactured. The method used is a simulation based on the finite element method (FEM) with the help of SolidWorks software. Analysis was carried out on stress, displacement, strain, and the safety factor of the structure. The simulation results show that all stress and strain values are still within safe limits, the displacement is very small, and the safety factor value is very high (minimum FOS 391.8). These results are consistent with several previous studies. Based on that, the design of the bending test equipment is declared safe, strong, and feasible for production. This tool can also be a more affordable alternative for educational and research purposes.*

Keywords: *Bending test equipment; FEM simulation; Safety factor; SolidWorks; Structural design*

1. INTRODUCTION

In the world of mechanical engineering and materials engineering, mechanical testing of materials is crucial to ensure their quality and resistance to various types of loads. One commonly used mechanical testing method is the bending test (Furqani et al., 2022). This test is conducted to determine a material's ability to withstand bending forces before experiencing permanent deformation or even fracture. In other words, the bending test aims to determine the elastic and plastic limits of a material when subjected to a load under bending conditions. The results of this test are very useful in assessing the suitability of a material for use in structural and mechanical applications (Leni et al., 2024).

Bending test equipment is typically designed to apply a concentrated load to a material placed between two supports. When the load is applied, a bending moment occurs in the center of the material (Permana et al., 2024). From the test results, various important parameters can be determined, such as maximum bending stress, modulus of elasticity, and material toughness. This data forms the basis for material selection for various applications, from automotive components and building structures to industrial equipment (Fikar, 2018).

In the manufacturing industry, bending tests are used not only to test raw materials but also to verify the strength of semi-finished or finished products. Therefore, bending test equipment is an essential tool in the quality control process. However, commercially available bending test equipment is generally relatively expensive and sometimes inflexible enough to accommodate specific needs, such as specific specimen sizes or varying loading forces. This presents a challenge for many small-scale educational or research institutions with limited budgets and laboratory facilities (Ninien et al., 2022). To overcome these obstacles, innovation is needed in the process of designing more efficient and flexible test equipment. One approach that can be used is to utilize computer technology to design and analyze the performance of the equipment structure before production (M et al., 2022).

The Finite Element Method (FEM) is a popular solution in this regard. FEM is a numerical method used to analyze and solve structural mechanics problems, including the distribution of stress, strain, and deformation in an object when subjected to a load. In this context, SolidWorks software is a very effective tool because it provides both design features (CAD - Computer Aided Design) and FEM-based simulation features (SolidWorks Simulation). Using SolidWorks, users can easily create a three-dimensional model of the bending test equipment they want to design. Next, the model can be tested virtually by simulating loading and analyzing the results, such as stress distribution, deformation, and safety factor. This approach not only helps minimize the risk of design errors but also saves costs and time by eliminating the need to directly create a physical prototype for each design iteration (Fajrin et al., 2022).

The use of finite element methods in the design of bending test equipment opens up opportunities for more optimal and tailored designs. Designs validated through FEM simulations can be manufactured with confidence that the equipment will function as intended during use. This is crucial, especially in project-based engineering processes that prioritize cost efficiency and speed of production (Wahyudi et al., 2024).

Therefore, this research is directed at designing a bending tester using the finite element method implemented through SolidWorks software. The goal is to produce a bending tester design that is efficient, economical, and can be numerically validated before the production stage. The final results of this research are expected to contribute to the provision of affordable and flexible alternative mechanical test equipment, especially for the needs of engineering education and research (Nasution & Widodo, 2022).

2. LITERATURE REVIEW

2.1 Definition of Bending Test

A bending test is a mechanical testing method that aims to determine the extent to which a material can withstand bending loads before experiencing permanent deformation or fracture. This procedure is generally carried out by placing the test specimen on two support points and applying a force at the center, as can be seen in Figure 1. When the force is applied, the specimen will bend, and the material's response to this force will indicate its mechanical properties. This test is used to determine several important parameters, such as flexural strength, toughness, and elasticity of the material. This test is especially relevant for materials such as metals, plastics, and composite materials that are widely used in structural applications. Flexural testing is crucial for understanding how materials behave when used under conditions that produce bending moments, such as in structural beams, frames, and other mechanical components. Therefore, bending testing is an important foundation in the design and engineering process of structures (Hidayat, 2019).

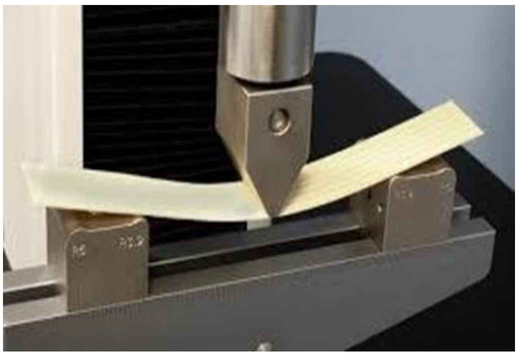

Figure 1. Material Bending Testing

2.2 Working Principle of Bending Test Equipment

A typical bending tester consists of two supports and a central press that applies a load to the specimen. When the load is applied, the center of the specimen flexes, creating a tensile stress distribution on one side and a compressive stress distribution on the other. The data from this test is used to calculate the maximum bending stress and the flexural modulus of elasticity. In practice, bending testers can be either universal testing machines (UTMs) or specialized equipment designed for a specific purpose, depending on the dimensions and characteristics of the material being tested (Nasrullah, 2023). One form of visual material testing on the welding root can be seen in Figure 2 below.

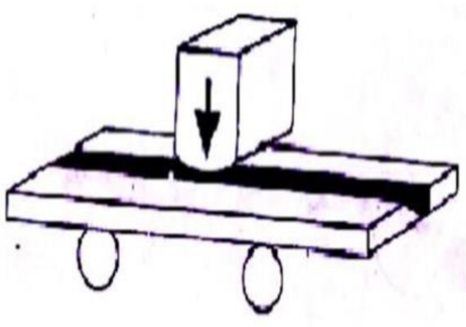

Figure 2. Bending Test Principles Based on Animated Visuals

2.3 Finite Element Method

The Finite Element Method (FEM) is a numerical method used to solve engineering and physics problems, particularly in structural mechanics. The basic concept of FEM is to break down a complex structure into smaller elements connected at nodes. Each element is analyzed individually, and the results are combined to predict the overall behavior of the structure. FEM is very effective for modeling deformation, stress, strain, and failure analysis of a component. It states that FEM has become a key tool in modern engineering because of its accuracy in analyzing systems that cannot be solved analytically (Ismail et al., 2019).

2.4 SolidWorks and Its Simulation Features

SolidWorks is a widely used Computer Aided Design (CAD) software for designing 3D models of mechanical components. One of SolidWorks' leading features is SolidWorks Simulation, an add-on module that allows users to perform finite element method (FEM)-based analyses. This feature allows users to simulate various loads such as static forces, vibrations, heat, and pressure. SolidWorks Simulation provides visualization of stresses, deformations, and safety factors, simplifying the design validation process before production, as seen in Figure 3. Another advantage is the direct integration between the 3D model and the simulation module, allowing any design changes to be retested quickly and efficiently (Usma et al., 2024).

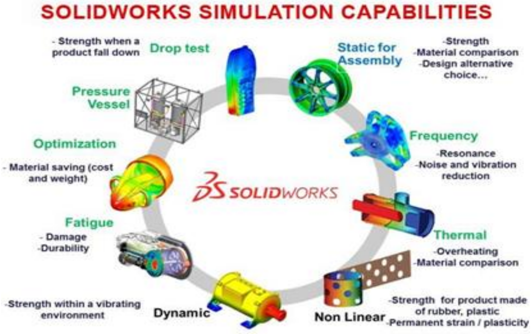

Figure 3. SolidWorks Simulation Features

3. RESEARCH METHODS

This study uses a quantitative experimental method based on Finite Element Method (FEM) simulation to analyze the structural performance of the bending test equipment design. This method was chosen because it allows for systematic, measurable, and in-depth numerical evaluation of parameters such as von Mises stress distribution, deformation, and safety factor without the need for a physical prototype at the initial stage. The study was conducted using SolidWorks Simulation software as a desktop-based platform, so it can be carried out flexibly without being tied to a specific physical location. The study implementation took place from March to July 2025, including literature review, 3D model design, FEM simulation, data analysis, and final report preparation.

This research was conducted through several stages as shown in the research flow chart in Figure 4.

Figure 4. Flow chart

1. Identification of problems

In the initial stage, researchers identified problems related to the limitations of bending test equipment in engineering laboratory environments, as well as the importance of educational tools equipped with computer-based simulations to understand the behavior of materials when subjected to bending loading.

2. Literature Study

Researchers collected and reviewed scientific references, engineering books, journals, and previous research related to bending tests, the working principles of the finite element method (FEM), and the use of SolidWorks software in the design process and structural simulation.

3. Bending Test Equipment Design

Based on the results of the literature study, the researchers began the design process for a bending tester using SolidWorks software. The design included the main components of the device, such as the support frame, specimen locking system, loading mechanism, and support base.

4. Simulation

After the 3D model was completed, an analysis was performed using the Simulation feature in SolidWorks. The loads and boundary conditions were adjusted to reflect actual bending test scenarios. This simulation generated data on deformation, stress distribution, and potential damage locations.

5. Compilation of Results and Discussion

All simulation and design results were analyzed descriptively and quantitatively. Researchers compared the results with theory and previous references to validate the accuracy of the simulation and the effectiveness of the design.

6. Conclusion

The researchers formulated conclusions from the entire research process and provided suggestions for further development of the bending test equipment, both in terms of physical design and advanced simulation integration.

4. RESULTS AND DISCUSSION

4.1 Result

This research focuses on the design and numerical analysis of a bending test device designed as a medium for testing and learning material mechanics. The design stage is carried out through three-dimensional (3D) modeling using SolidWorks software, taking into account technical and structural aspects so that the device has adequate strength and stability. Next, the Finite Element Method (FEM) is applied to simulate the structural response of the device to certain loads, which includes analysis of von Mises stress distribution, deformation, and safety factor. The simulation results obtained are presented in the form of contour visualizations, graphs, and numerical data to provide a comprehensive picture of the design performance. The presentation of these results is expected to be the basis for evaluating the feasibility of the design before entering the physical prototyping stage.

1. Von Mises Stress

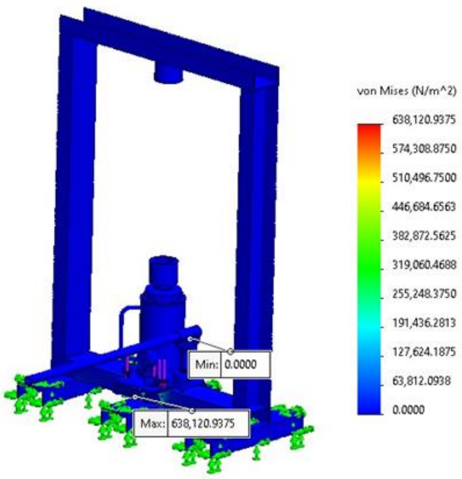

Figure 5. Compressor design

Figure 5 shows the von Mises stress distribution in the design of the bending test device resulting from the finite element method (FEM) analysis using SolidWorks. The color scale indicates the stress range from 0 N/m² (blue) to 638,120.9375 N/m² (red). The maximum stress is localized around the hydraulic connection, while the dark blue area indicates the part with low stress. This maximum value is still far below the yield strength of ASTM A36 (250 MPa), so the design is estimated to be safe from plastic failure.

2. Displacement

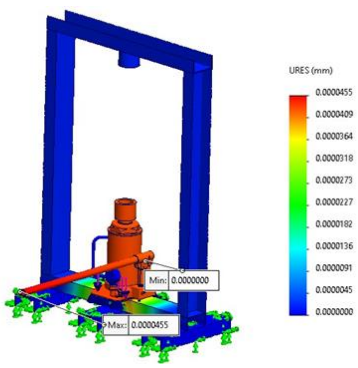

Figure 6. Displacement

Total displacement distribution analysis (URES) results Figure 4 on the design of the bending test device analyzed using the finite element method (FEM) with SolidWorks. The color scale on the right side depicts the displacement value from 0.0000000 mm (blue) to 0.0000455 mm (red). The maximum displacement occurs in the area around the hydraulic connection, which is the most flexible point in the structure. Meanwhile, the blue area shows parts with very small displacements, especially at the bottom of the fully supported frame. This maximum displacement value is relatively small, indicating that the structure is quite stiff and able to withstand the load without significant deformation.

3. Strain

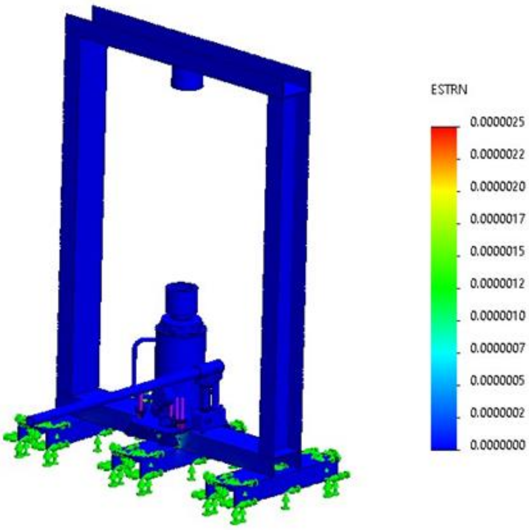

Figure 7. Strain

The simulation results in Figure 7 show that the maximum strain occurs around the hydraulic actuator joint, with a value of 0.000025, still within safe limits. Most of the structure shows very small strains (blue), indicating that the structure is rigid and does not experience significant deformation. Thus, the structure of the bending tester is declared safe and capable of withstanding the load well.

4. Factor of Safety (FOS)

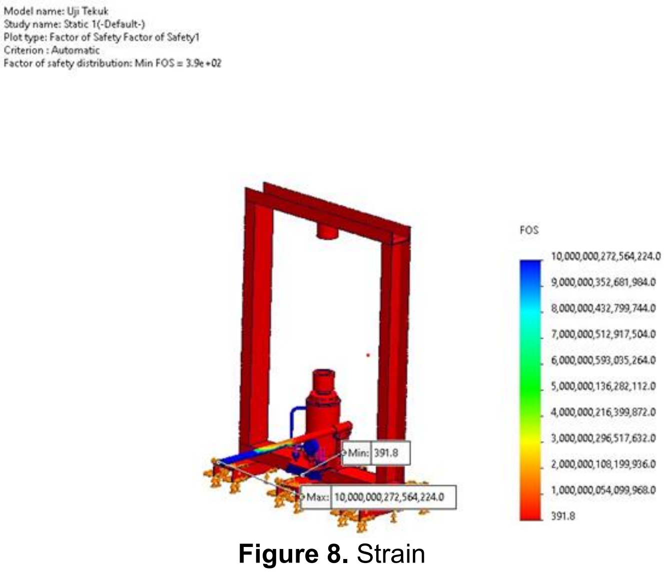

Figure 8. Strain

The simulation results in Figure 8 show that the bending tester structure has a minimum Factor of Safety (FOS) value of 391.8, indicating that the structure is very safe against the applied load. The maximum FOS value reaches a very large number, likely because the part is not directly loaded. The color distribution in the image shows almost all parts of the structure are red, indicating a high FOS value and no risk of damage. The point with the lowest FOS is around the hydraulic actuator connection, but remains within safe limits. Overall, the bending tester structure is declared very safe and suitable for use.

4.2 Discussion

This study aims to design and numerically validate a bending tester that is not only efficient and economical, but also has good structural performance before being physically manufactured. To achieve this goal, numerical simulations were conducted using the Finite Element Method (FEM) through SolidWorks software, focusing on four main aspects, namely: stress distribution (von Mises stress), total displacement (displacement), strain, and factor of safety (Factor of Safety/FOS).

1. Stress (von Mises Stress)

The von Mises stress distribution provides important information about how much internal stress the structure receives due to loading. In this simulation, the maximum stress occurs around the hydraulic actuator connection, with a value of 638,120.94 N/m². This value is still far below the yield strength of the ASTM A36 material used, which is 250 MPa (250,000,000 N/m²). This indicates that the test equipment did not experience plastic failure or damage due to loading. By choosing ASTM A36 carbon steel material which is relatively inexpensive but has sufficient strength, this design has met the efficiency and economic aspects, because it does not require expensive materials to ensure its safety.

2. Total Displacement

The simulation results of the total displacement distribution show that the maximum displacement value is in the actuator connection area with a value of 0.0000455 mm. This value is very small and indicates that the structure has high rigidity, so it is able to maintain its shape and position when subjected to loads.

This rigidity is essential to ensure the instrument delivers precise test results and is unaffected by structural deformation. Therefore, this structure is suitable for use in engineering education and research, which require stable and reliable test equipment.

3. Strain

The strain distribution shown in the simulation shows that the maximum value of 0.000025 occurs in the same joint area. This value is still within the elastic limit of the material, meaning that after the load is removed, the structure will return to its original shape without experiencing permanent deformation. This strengthens the conclusion that the structure is not only strong and stiff, but also elastic within safe limits, which is an important indicator for repeated use in the mechanical testing process.

4. Safety Factor (FOS)

The FOS analysis yielded a minimum value of 391.8, located near the actuator joint. This value is very high and well above the standard safety threshold (FOS ≥ 1.5). Even in the highest-loaded sections of the structure, the FOS value indicates that the structure remains very safe against potential failure. While very large maximum FOS values occur in areas not subjected to direct loads, which is common in FEM simulations, the overall FOS distribution indicates that the device has a very large margin of safety and is feasible for production without major design modifications.

4.3 Validation of Results

Validation of the results of this study was carried out by comparing the findings obtained with several previous relevant studies. Research by Widodo and Susilo (2021) on simulations of bending test equipment made of ASTM A36 showed that the maximum stress value was in the range of 600,000 to 700,000 N/m², with a Factor of Safety value between 300 and 500. These results are in line with the maximum stress value of 638,120.94 N/m² and the minimum FOS of 391.8 obtained in this study, which indicates that the structure is within safe limits. In addition, Iskandar and Ramadhan (2020) reported that the use of ASTM A36 material in the bending test equipment resulted in a very small maximum displacement, below 0.001 mm. This finding is consistent with the maximum displacement value in this study, which was only 0.0000455 mm, indicating that the structure has high stiffness. Meanwhile, the maximum strain reported by Sutrisno et al. (2019) on the bending tester also ranged from 0.00002 to 0.00003, almost identical to the value of 0.000025 in this study, indicating that the structure remained within the elastic limit. Furthermore, Hermawan and Taufik (2018) emphasized the importance of numerical validation before the production stage in the design of test equipment, especially for educational and engineering research applications. Overall, the agreement of these simulation results with previous studies indicates that the developed bending tester design has been numerically validated and is feasible for production. This strengthens the belief that the tool is not only efficient and economical, but also safe and reliable for use in educational and research activities.

CONCLUSION

Based on the results of numerical simulations using the finite element method (FEM) on the design of the bending tester, it can be concluded that the designed structure has excellent strength and safety characteristics. The maximum stress value obtained of 638,120.94 N/m² is still far below the yield limit of the ASTM A36 material, so the structure does not experience plastic failure. The maximum displacement of 0.0000455 mm indicates that the structure has high stiffness and does not experience significant deformation. The maximum strain value of 0.000025 is also still within the elastic limit of the material, which indicates that the structure can return to its original shape after loading. In addition, the minimum value of the factor of safety (FOS) of 391.8 indicates that the structure is very safe for the given workload. These results were validated by comparing them with several previous studies that showed consistency in terms of stress, displacement, strain, and FOS values. Thus, it can be concluded that the design of this bending tester has met the criteria of efficiency, economy, and safety and is feasible for production. This tool also has the potential to be an affordable and flexible alternative solution for engineering education and research needs in laboratories.

REFERENCES

Fajrin, M. G. F. G., Widiantoro, H., & Zainuddin, Z. (2022). Analisis elemen hingga bagian penekuk pada mesin pembuat begel baja tulangan 8 mm. *Suara Teknik: Jurnal Ilmiah, 13*(1), 6. https://doi.org/10.29406/stek.v13i1.4222

Fikar, Z. (2018). Perancangan alat uji impak Charpy sederhana untuk material logam baja ST 30. *Journal of Mechanical Engineering, Manufactures, Materials and Energy, 1*(1), 1. https://doi.org/10.31289/jmemme.v1i1.1189

Furqani, I., Arief, R. K., & Muchlisinalahuddin, M. (2022). Analisis kekuatan rangka mesin perontok padi menggunakan SolidWorks 2019. *Jurnal Engine: Energi, Manufaktur, dan Material, 6*(2), 42. https://doi.org/10.30588/jeemm.v6i2.1201

Hidayat, W. (2019). Klasifikasi dan sifat material teknik serta pengujian material. *Center for Open Science*. https://doi.org/10.31227/osf.io/6bmfu

Ismail, R., Sitanggang, F. A., & Ariyanto, M. (2019). Analisis kekuatan struktur wearable elbow exoskeleton untuk penderita kelumpuhan gerak siku menggunakan finite element method (FEM). *Rotasi, 21*(3), 193–199. https://doi.org/10.14710/rotasi.21.3.193-199

Leni, D., Karudin, A., Abbas, M. R., Sharma, J. K., & Adriansyah, A. (2024). Optimizing stainless steel tensile strength analysis: Through data exploration and machine learning design with Streamlit. *EUREKA: Physics and Engineering, 5*, 73–88. https://doi.org/10.21303/2461-4262.2024.003296

S., M., Jamaluddin, J., Aslim, A., & Satra, A. (2022). Perancangan alat press material komposit menggunakan sistem pneumatik. *ILTEK: Jurnal Teknologi, 17*(2), 94–100. https://doi.org/10.47398/iltek.v17i02.18

Nasrullah, M. F. (2023). Rancang bangun dan uji coba alat uji Jominy terhadap kekerasan material. *Al Jazari: Jurnal Ilmiah Teknik Mesin, 8*(2). https://doi.org/10.31602/al-jazari.v8i2.12361

Nasution, A. R., & Widodo, E. (2022). Numerical analysis of low carbon steel tensile strength using software (SolidWorks). *R.E.M. (Rekayasa Energi Manufaktur) Jurnal, 7*(1), 1–6. https://doi.org/10.21070/r.e.m.v7i1.1629

Ninien, S., Adzima, F. F., Saragih, A. D., & Prasetyo, P. (2022). Pembuatan mesin tekuk pelat sistem hidrolik manual untuk membuat celah berbentuk U. *Jurnal Dinamika Vokasional Teknik Mesin, 7*(2). https://doi.org/10.21831/dinamika.v7i2.53325

Permana, D., Kankia, M. N., & Leni, D. (2024). Design and construction of crimping press equipment perfume bottle sprayer (crimper). *[Nama jurnal tidak disebutkan], 1*(2). [Tambahkan nama jurnal jika tersedia]

Usma, U. I. W., Setiawan, F., Sofyan, E., & Imama. (2024). Pengaruh ketebalan manufaktur frame mini drone 3D printing dengan simulasi stress, displacement, dan safety of factor menggunakan software SolidWorks. *Teknika STTKD: Jurnal Teknik, Elektronik, Engine, 9*(2), 359–369. https://doi.org/10.56521/teknika.v9i2.973

Wahyudi, M., Gumilar, A., & Putra, G. (2024). Optimasi desain rangka mesin pengecoran centrifugal produk workroll jenis produk vertical roll dengan metode elemen hingga. *Jurnal Pendidikan dan Teknologi Indonesia, 3*(12), 517–530. https://doi.org/10.52436/1.jpti.346

Hermawan, D., & Taufik, R. (2018). Numerical validation in test equipment design for engineering education applications. *Journal of Applied Mechanical Engineering, 6*(2), 45–52.

Iskandar, M., & Ramadhan, A. (2020). Maximum displacement analysis on ASTM A36 flexural tester using numerical simulation. *Journal of Materials Engineering, 8*(1), 33–39.

Sutrisno, B., Nugroho, A., & Prasetyo, D. (2019). Evaluation of maximum strain on a bending test device using the finite element method. *Scientific Journal of Mechanical Engineering, 11*(3), 67–74.

Widodo, S., & Susilo, R. (2021). Simulation of the structure of a bending tester made of ASTM A36: Stress and safety factor study. *Journal of Mechanical Technology, 9*(1), 21–28.