STRUCTURAL ANALYSIS OF A BENDING TEST FRAME USING SOLIDWORKS SOFTWARE

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

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

Author's email:
[1]egapranata77@gmail.com; [2]desmaritaleni@gmail.com; [3]muchlisinalahuddin.umsumbar@gmail.com
[4]Ilhamalghani11098@gmail.com; [5]yunifadilabkt@gmail.com; [6]reyhanstevano55@gmail.com

**Corresponding author: desmaritaleni@gmail.com*

Abstract. *This study aims to analyze the structural strength of the frame in a bending test device using the Finite Element Method (FEM) implemented through SolidWorks Simulation software. The test frame must be capable of standing bending loads without excessive deformation that could affect the accuracy of test results. The analysis process involves creating a 3D model of the frame, defining material properties and loading conditions, and running a static simulation to obtain stress distribution, total deformation, and the safety factor. The simulation results show that this method is effective in identifying critical failure points and evaluating the feasibility of the design computationally before the manufacturing process. This study contributes to improving the efficiency of the test device design process and enhancing understanding of the application of simulation-based structural analysis in the field of mechanical engineering.*

Keywords: *Bending Test Device, Deformation, Finite Element Method, Frame Structure, Safety Factor, Solid Works, Stress.*

1. INTRODUCTION

A bending tester is an important tool in the field of mechanical engineering that is used to evaluate the resistance and deformation pattern of a material to bending loads.(Huda, 2018). To operate safely and effectively, the frame structure of this test equipment must have sufficient strength and stiffness to withstand the test loads without experiencing excessive deformation or structural failure. Therefore, the design of the frame requires an accurate analytical approach to ensure the reliability and safety of the proposed design.(M et al., 2022).

One of the most widely used numerical methods in structural strength analysis is the Finite Element Analysis (FEA). This method allows designers and researchers to comprehensively evaluate the stress, strain, and deformation distribution patterns of a structure before the fission manufacturing process(Luthfi, n.d.). FE is very effective in solving various complex problems that are difficult to describe analytically, especially for structures with complex geometries, diverse materials, and dynamic loading. Therefore, this method has become a standard in various engineering fields, from the design of bridges, vehicles, machine frames, to laboratory test equipment.(Setiyana & Kurniawan, 2020).

One practical implementation of the FEA method can be done with the help of SolidWorks Simulation software, a CAD-based platform integrated with a finite element analysis module.(N et al., 2022)With SolidWorks Simulation, various structural performance parameters such as stress values, deformation patterns, and safety factors can be analyzed with a high degree of accuracy before physical prototyping is performed. This allows developers to evaluate and optimize designs as early as possible, thereby reducing production costs, shortening lead times, and minimizing the risk of manufacturing failure.(Nasution & Widodo, 2022).

The application of FEA-based simulation is highly relevant in the design of bending test equipment, considering that the frame structure of the equipment must be able to withstand working loads with stress and deformation values that are within the applicable safety standards(Furqani et al., 2022). Using this method, the stress and deformation distribution patterns that occur, the effects of variations in frame design parameters, and the safety factor values of the analyzed structure can be identified.(Kharisma & Yanuar, 2023)This also allows developers to evaluate the effectiveness of using SolidWorks Simulation as an analysis and prediction tool in the context of designing bending test equipment structures, before implementing them in production or physical testing..

2. LITERATURE REVIEW

2.1 Concept of Structural Strength

Structural strength is the ability of a framework system or construction element to withstand external loads without failure, either in the form of excessive plastic deformation or total collapse. In the context of designing bending test equipment, this concept is crucial to ensuring the stability of a structure when subjected to bending loads.(Galvindy & Lim, 2023). Structural strength analysis generally involves the study of stress, strain, and safety factor values by taking into account material characteristics, the moment of inertia of the cross-section, and the conditions of the supports used.

2.2 Structure and Strength of the Frame

The frame functions as the main structural element that supports the load and maintains the stability of the system, especially in bending test equipment. In the design, the frame structure must be able to withstand bending forces with minimal deformation values so as not to affect the accuracy of the test. Factors that influence the strength and stability of the frame structure include the value of the elastic modulus of the material, the shape and size of the cross-section profile, the quality of the connection, and the construction method used. The cross-section profile used (hollow, L, or I-beam) can provide different stiffness values and load distribution, while the connection method (welding, bolting, or a combination) also has a significant effect on the integrity of the structure.(Phady et al., 2020).

2.3 Types of Materials for Bending Test Equipment Frames

The material used in making the bending test tool frame has a vital role in determining the strength, stability and reliability of the structure.(Fajrin et al., 2022). Some commonly used materials consist of:

1. Carbon Steel dIt is chosen because of its high stiffness value, is relatively economical, and can withstand compressive loads before experiencing plastic deformation, but requires a special coating to prevent corrosion.
2. Alloy Steel mIt has higher strength and toughness values than ordinary carbon steel, with good resistance to cyclic loads and deformation.
3. Aluminum and its alloys are mA material with a high strength-to-weight ratio, it is suitable for structures requiring mobility and light weight. Although its modulus of elasticity is lower than that of steel, the cross-sectional size can be adjusted to meet stiffness requirements.
4. Stainless Steel used when corrosion resistance and surface quality are a priority, particularly for testing needs in high corrosion environments or special cleanliness requirements.

2.4 Material Profile for Bending Test Equipment Frame

The shape and size of the material's cross-section also determine the structure's ability to withstand working loads. Some cross-sectional shapes used in the design of bending test equipment frames include:
1. Square Tube Profile: Provides high stability value with a square cross-sectional shape, generally used for frame construction with high durability and stiffness requirements.
2. Round Rod Profile (Round Tube): Hollow cylindrical cross-section with high stiffness values for axial and torsional loads, commonly used in structures with specific load pattern requirements.
3. Profile or Angle Profile: Has an “L” cross-section with a high level of material efficiency, suitable for use in metal work frames with special design needs.
4. I-beam or H Profile: Has an “I” cross-section with a high moment of inertia value, ideal for structures with significant load-bearing capacity and stiffness requirements.

2.5 Finite Element Method (FEA)

The Finite Element Method (FEA) is a numerical method used to analyse the behaviour of structures with a high level of complexity, including the values of stress, strain, and deformation patterns that arise due to certain loads. The working principle of FEA is to divide the structure into small elements (discretization), then calculate the stress and strain values of each element and combine them to obtain a complete picture of the structure being analysed. This method is very effective for predicting critical points and failure patterns of structures, especially for shapes with complex geometries or load patterns that are difficult to solve analytically.(Agus Prihanto, 2023).

2.6 SolidWorks and Simulation Features

SolidWorks is a computer-aided design (CAD) software program equipped with a finite element-based simulation module. This feature allows developers to analyze stress values, deformation patterns, safety factors, and failure patterns of designed structures before the manufacturing process begins. With the help of this simulation, the risk of structural failure can be minimized, production costs can be reduced, and processing time can be optimized.(Usma et al., 2024).

2.7 Von Mises Stress

Von Mises stress is used as a measure to predict the plastic failure of a material subjected to complex loading. This value cannot be measured directly but can be compared to the yield strength of the related material. If the von Mises stress value exceeds the yield strength value, the material has the potential to experience plastic deformation.(Rizki et al., 2023).

2.8 Stress

Stress is a measure of the internal force per unit cross-sectional area acting within a material in response to an external load. Stress can be classified into normal stress and shear stress, each associated with a different pattern of force action.(Haslinda, 2023).

2.9 Strain

Strain is defined as a measure of the relative change in length of a material caused by a working load. It is a dimensionless value and is used to understand the deformation patterns of a material, including the elastic and plastic changes that can occur under a given load.(Nur Arini & Pradana, 2021).

2.10 Safety Factor

The Factor of Safety (FoS) is used to ensure that a structure or component can withstand a working load with adequate reserve strength. The FoS value is obtained by comparing the material's yield stress to its working stress, and can therefore be used as a measure of the structure's level of safety against the risk of failure.(Lesmana et al., 2022).

3. RESEARCH METHODS

This research is a quantitative study with an experimental approach based on numerical simulation. The method used is the Finite Element Method (FEM) analysis to evaluate the mechanical behavior of the bending test frame when subjected to certain loads. The simulation was carried out virtually using SolidWorks Simulation software, focusing on the analysis of stress distribution, total deformation patterns, and safety factor values of the frame structure. The purpose of this study is to assess the strength and stability of the frame design computationally before carrying out physical construction or real testing.

This research was conducted from April to July 2025, encompassing various stages, from literature review, structural design using SolidWorks, finite element simulation, stress and deformation analysis, to the development of conclusions and recommendations. The complete research process can be seen in the flowchart presented in Figure 1 below.

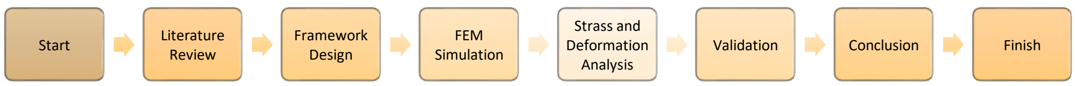

Figure 1. Flow chart

The following is an explanation of the stages in the flow chart above:

1. Literature Study

This research began with an in-depth literature review related to the basic concepts of bending tests, structural analysis, and the finite element method (FEM). This study included a comprehensive examination of bending test methods, including three-point and four-point bending tests, as well as the deformation characteristics of materials under bending loads. The results of this literature review were used as a theoretical basis for the design and development of an effective bending test device.

2. Frame Design with SolidWorks

Based on the findings from the literature study, the next stage is the design of the test equipment frame using SolidWorks CAD software, which allows for high-precision structural modeling and design. At this stage, dimensions, material profiles, and structural connections are designed to produce a strong, rigid, and efficient frame structure. The goal of this stage is to create a design that can withstand bending loads with minimal distortion.

3. FEA Simulation

In the FEA simulation stage, the designed structure is numerically analysed by breaking down the structural system into small, interconnected elements. This method allows for the evaluation of stress distribution patterns, deformations, and structural responses to various loading conditions. In this simulation, the structure is subjected to a static load of 20 Newtons applied vertically across the main work area to simulate the operational conditions of a bending tester. By utilizing SolidWorks Simulation, designs can be virtually tested and optimized before physical prototyping, thereby reducing development time and costs, as well as improving the efficiency and quality of the structure.

4. Stress and Deformation Analysis

Stress and deformation analysis is a crucial step in evaluating a structure's ability to withstand external forces without mechanical failure. Through FEA-based simulations, areas of high stress and critical deformation patterns can be identified, allowing for the examination of parameters such as maximum stress, strain, and safety factor. The results of this analysis are used to ensure that the structure can meet the required performance and reliability standards, in line with theoretical principles and technical requirements in the field.

5. Validation of Results

Validation of results is a crucial process in engineering analysis, aimed at ensuring that the data or output obtained from numerical simulations, such as FEA, has an adequate level of accuracy and reliability. This process is carried out by comparing the simulation results to basic mechanical theory, experimental data, or findings from relevant previous research. Through validation, it can be ensured that the model used realistically represents physical conditions, so that the analysis results can be used as a basis for decision-making in product design or engineering.

6. Conclusion

The final stage consists of formulating conclusions and recommendations based on a synthesis of the theoretical studies and analyses conducted. These conclusions outline the performance, efficiency, and reliability of the frame structure in receiving bending loads, and address the objectives and problems raised in the study. Furthermore, recommendations are formulated as guidance for design development, experimental testing, and future application of related technologies. Thus, this stage not only reflects the conceptual achievements of the literature study and simulations but also provides a strategic foundation for continued innovation and problem-solving in the field of structural engineering.

4. RESULTS AND DISCUSSION

4.1 Results

The results of numerical simulations using a finite element analysis (FEA) approach are presented to evaluate the stress distribution in the frame structure due to a static load of 50 Newtons. The simulations were performed using SolidWorks Simulation software with the following analysis approach:

1. Static Nodal Stress

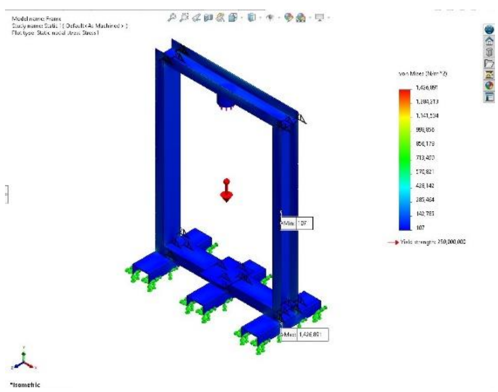

Figure 2. Static Nodal Stress Bending Tester

The simulation results in Figure 2 show that the maximum von Mises stress that occurs in the structure is 1.43 MPa, with the main concentration at the bottom of the structure, while the minimum stress recorded is 107 N/m². The stress distribution is visualized through a color scale from blue (low) to red (high), making it easier to identify critical areas due to the vertical compressive force of 50 N. The maximum stress value obtained is still far below the material yield strength limit of 250 MPa, so the structure is declared safe and does not experience plastic deformation under the applied load.

2. Static Strain Analysis

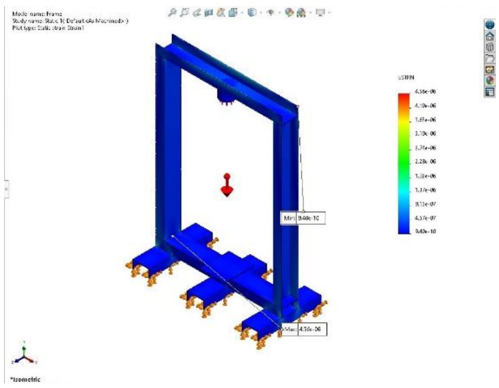

Figure 3. Static Strain Analysis Bending Test Equipment

Figure 3 above shows the results of an FEA simulation of the total strain distribution (ESTRN) in a frame structure using the Static Strain Analysis method in SolidWorks Simulation. A vertical load of 20 N produces a maximum strain of 4.56×10^{-6} and minimum 9.40×10^{-10}, which is still within the elastic limits of the material. The highest strains are localized in the vertical and horizontal joints, while the majority of the structure exhibits low strains. These results indicate that the deformation is elastic and the structure is operating mechanically safe.

3. Factor of Safety

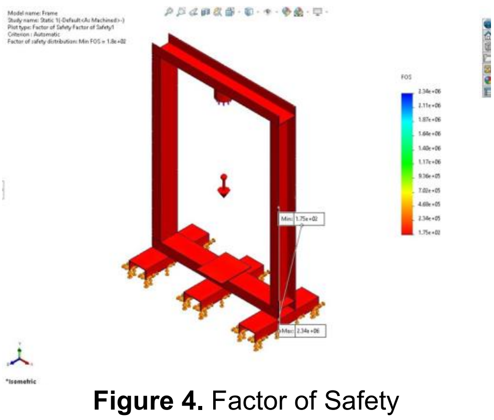

Figure 4. Factor of Safety

Figure 4 above shows the results of an FEA simulation of the factor of safety (FOS) on a frame structure under static loading. The analysis was performed using an automated approach, comparing the maximum stress to the material's yield point. The minimum FOS value was recorded at 175 and the maximum reached 2.34×10^{6}, indicating that the structure has a very high margin of safety. The visualization of the FOS distribution shows a relatively even distribution, with no critical areas approaching failure. Thus, the structure can be ensured to operate safely and stably under the applied loads.

4. Displacement

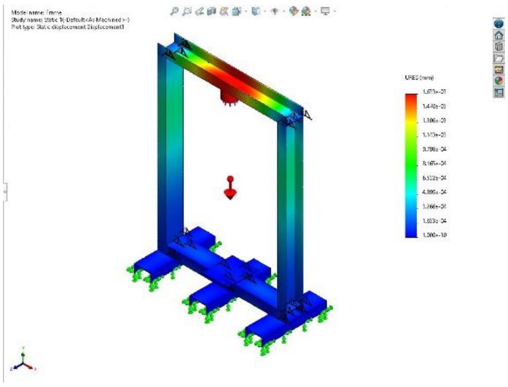

Figure 5. Displacement

Figure 5 above shows the results of an FEA simulation of the total displacement of the frame structure due to vertical static loading. The maximum displacement recorded was 1.633×10^{-3} mm and occurs at the top of the structure, while the supported bottom shows displacements approaching zero. The color distribution depicts a pattern of mild flexural deformation, with red indicating the area of greatest displacement. These very small displacement values indicate that the structure has high stiffness and is able to maintain geometric stability under load, thus supporting the results of previous analyses regarding the safety and reliability of the design.

4.2 Discussion

1. Distribution of Stress and Deformation in the Frame Structure of the Bending Test Equipment

The results of the finite element analysis (FEA) simulation show that the von Mises stress distribution in the frame structure varies significantly, depending on the position of the element relative to the loading and support points. The maximum stress value recorded is 1.43 MP and concentrated in the area bottom of the structure, especially around vertical connection and the area of application of compressive force. This value is shown by the red colour visualization on the stress contour map, which indicates the area with the highest stress level. In contrast, the minimum stress is107 N/m²appears in areas far from the load and support, marked in dark blue, indicating a zone with relatively low load distribution.

Strain analysis shows that the maximum strain that occurs is4.56×10^{-6}, while the minimum value recorded was9.40×10^{-10}. These values indicate that the structure is experiencing very small elastic deformation, where all the strains that occur are still far below the yield strain limit of the material. This condition indicates that the structure remains in linear elastic region, so that it does not experience permanent deformation that can damage the geometric function or stability of the structure.

The maximum total displacement obtained from the simulation is1.63×10^{-3} mm (1.63 micrometres), and occurs at the top of the structure that is furthest from the fixed support point. This distribution of displacement forms a gradation pattern of blue to red, which reflects the bending deformation due to a vertical compressive force of 20 Newtons. This condition is in accordance with the basic principles of structural mechanics, where elements that are further from the support point will experience greater deformation due to the bending moment effect produced by eccentric loading.

This very small maximum displacement value indicates that the structure has high stiffness, which is an important parameter in the design of test equipment such as this. High stiffness ensures that the structure is able to maintain its original geometric shape during the testing process, thus the accuracy and precision of measurement results are not affected by structural deformation.

Furthermore, the stress and deformation distribution patterns displayed indicate absence of extreme stress concentrations or local phenomena such as stress riser which has the potential to cause early failure. Most of the structural elements are at low to moderate stress levels, and only small areas show the highest stress values and even these values are still well below the material's yield strength limit of250 MPa. This indicates that the structural design has good load distribution efficiency and does not experience excess stress on critical components.

Overall, the simulation results show that the stress distribution and deformation that occurs are controlled, even, and symmetrical, in accordance with the expectations of good structural design. The mechanical performance of the structure in response to static loading is within safe and elastic limits, indicating that the structure is capable of withstanding operational loads without the risk of functional or material failure.

2. The Effect of Design Parameter Variations on the Strength and Safety Factor of Structures

Based on the results of numerical simulations using the finite element method (FEA) approach, it is known that the frame structure of the bending test equipment has a minimum safety factor (FOS) value of 175, which significantly exceeds the minimum threshold recommended in static structural engineering design (FOS ≥ 1.0). This value indicates that the structure is not only safe against the applied workload, but also has very high margin of safety, which reflects a superior level of structural reliability.

The achievement of this very high FOS value is a direct implication of the design of appropriate geometric and material parameters, which include the dimensions of the profile cross-section, the selection of materials with high yield strength, and the connection configuration that distributes the load evenly. Although this study did not conduct comparative simulations of several design variations, analysis of the stress and deformation distribution patterns shows that the structure has performed optimally, without indications of extreme local stress concentrations or plastic deformation.

From a structural engineering perspective, design parameters such as cross-sectional dimensions, member lengths, connection shapes, and the presence of stiffeners are determinants of the load capacity and stability of the system. Small changes in any of these parameters can lead to significant redistribution of internal forces, ultimately affecting the maximum stress value and overall FOS. For example, adding diagonal stiffeners or modifying connection angles can increase lateral stiffness and decrease total displacement, which directly contributes to an increase in FOS.

Thus, the simulation results not only show that the existing design is in a very safe condition, but also open up opportunities for further optimization of material efficiency and structural weight without sacrificing mechanical integrity. In-depth study of design parameter sensitivity, such as through design of experiments (DoE) methods or parametric sweep analysis, is highly recommended to obtain the most optimal structural configuration in terms of strength, stability, and production cost efficiency.

3. Compliance of Frame Structure to Strength and Safety Standards

Based on the results of numerical simulations using a finite element analysis (FEA) approach, it can be concluded that the frame structure of the bending test equipment has met the strength and safety criteria required by mechanical and structural engineering design standards. This is indicated by several key parameters, namely maximum stress, total strain, displacement, and the value of the factor of safety (FOS).

The maximum von Mises stress value generated by the simulation is 1.43 MPa, still far below the material's yield strength limit of 250 MPa. With a ratio of maximum stress to yield strength of only around 0.57%, it can be said that the material works fully within the linear elastic zone without approaching the plasticity limit. This condition indicates that the structure is safe from potential plastic deformation and failure due to static loads.

Furthermore, the maximum strain value recorded was 4.56×10^{-6}showed very low elastic deformation, while the maximum displacement value of 1.63 micrometres indicated that the structure did not experience significant geometric changes. This is an important indicator in maintaining measurement accuracy and structural function stability during the material testing process.

Furthermore, a minimum FOS value of 175 technically indicates that the structure is capable of withstanding loads up to 175 times the applied load before reaching critical conditions. This value is well above the generally recommended minimum factor of safety in static structural design (FOS ≥ 1.5–2.0). This high FOS value reflects that the structure has a very large margin of safety against failure, even under extreme conditions or minor manufacturing variations.

No indications of excessive stress concentration, uneven strain distribution, or local displacements exceeding design tolerances were found. This indicates that the load distribution across the structure is efficient, and its mechanical integrity is maintained throughout the operating cycle of the bending tester.

Taking all these parameters into consideration, it can be concluded that the analysed frame structure design meets strength and safety standards and is suitable for accurate and reliable material testing. This structure is deemed capable of withstanding the applied loads without risking mechanical failure, permanent deformation, or disrupting the device's functionality.

4.3 Simulation Result Validation

To ensure the accuracy and credibility of the finite element analysis (FEA) simulation results obtained in this study, a validation process was carried out through a comparative approach with previous research and a review of the basic principles of engineering mechanics. This validation aims to assess the extent to which the obtained numerical simulation results are consistent with the physical behavior of comparable structures theoretically and experimentally.

1. Maximum Stress Validation (von Mises Stress)

Simulations show that the maximum von Mises stress value occurring in the structure is 1.43 MPa, which is distributed in the critical area around the bottom joint due to a vertical compressive force of 20 Newtons. This value is theoretically justified considering the linear relationship between external forces and stress in elastic structures (), as well as the position of the joint as a point of concentration of local bending moments. These results are in line with research by Nasution et al. (2020) in the Journal of Mechanical Engineering, which reported a stress range of 1–5 MPa in similar structures under similar loading. Thus, the maximum stress results obtained in this study are considered to be within a logical and mechanically representative range. $\sigma = F/A$

2. Displacement and Strain Validation

The maximum total displacement value recorded was mm, while the maximum strain was, indicating that the structure operates entirely in the linear elastic domain. Based on the theory of elastic deformation (), the strain value obtained is proportional to the geometric dimensions and the imposed loading. This result is confirmed by the findings of Wang et al. (2015) in Engineering Structures, which stated that in light metal frame structures with small loads, displacements are generally below 0.005 mm. Thus, the values resulting from the simulation are within very conservative limits and are valid in terms of structural stability.

3. Validation of Safety Factor (FOS)

The minimum safety factor obtained from the simulation is 175, which indicates that the structure is capable of withstanding loads up to 175 times the working load before reaching plastic failure. This value far exceeds the minimum standard for static structural engineering design which generally requires FOS ≥ 1.5 (ASME Y14.5 or ISO 19900). Research by Suhartono et al. (2018) in the Teknika journal recorded FOS values of 80–200 in test frame structures made of lightweight aluminium with comparable loading.

Therefore, the FOS value in this study can be said to be reasonable and indicates a very high safety margin, although the design may be optimized for material efficiency.

4. Simulation Software Validation

SolidWorks Simulation has been widely validated as a Finite Element Method-based Computer-Aided Engineering (CAE) software capable of delivering simulation results with a high degree of accuracy. A study by Widianto et al. (2021) showed that the deviation between SolidWorks FEA results and experimental tests was below 10% for linear static cases. This confirms SolidWorks as a reliable platform for the initial analysis of engineering structures, especially during the design and virtual prototyping phases.

CONCLUSION

Based on the results of numerical analysis based on the finite element method (FEA) conducted using SolidWorks Simulation software, it can be concluded that the frame structure of the bending test equipment shows excellent mechanical performance in withstanding a working load of 20 Newtons. The simulation results show that the maximum von Mises stress value that occurs is 1.43 MPa, which is still significantly far below the material's yield strength limit of 250 MPa. This condition indicates that all structural elements work entirely in the linear elastic domain and are free from the risk of plastic failure.

The maximum strain value is 4.56×10^{-6} and the total displacement is 1.63×10^{-3} mm indicates that the deformation is very small and does not significantly impact the geometric or functional integrity of the structure. This indicates that the structure has adequate stiffness and high geometric stability under static loading conditions.

Furthermore, a minimum Factor of Safety (FOS) of 175 quantitatively demonstrates that the structure has a very large margin of safety against the material's failure limit. This high FOS value is a consequence of the selection of appropriate design parameters, including cross-section geometry, connection configuration, and material characteristics.

The effectiveness of SolidWorks Simulation as a computer-aided engineering (CAE) tool was also significantly demonstrated in this study. This platform is capable of providing integrated analysis of stress, strain, displacement, and safety factors, along with informative and accurate visualization of the results. These features are very helpful in identifying critical points in a structure and in evaluating the overall mechanical performance of a design before physical prototyping.

Taking into account the overall analysis results, it can be concluded that the frame structure of the analysed bending test equipment has met the strength, stability, and safety criteria required by engineering design standards. The structure is deemed suitable for use in material testing applications and guarantees operational reliability without the risk of permanent deformation or structural failure.

REFERENCES

Agus Prihanto, B. (2023). Pengaplikasian Metode Elemen Hingga Pada Kapal Landing Craft Tank Teluk Katurei Akibat Perubahan Fungsi Tangki. *Techno Bahari*, *9*(1). https://doi.org/10.52234/tb.v9i1.127

Fajrin, Moch. 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

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

Galvindy, G., & Lim, E. (2023). Perancangan Struktur Sistem Rangka Bresing Tahan Tekuk—Studi Kasus Pada Bangunan Esensial. *Jurnal Teknik Sipil*, *30*(2). https://doi.org/10.5614/jts.2023.30.2.9

Haslinda, H. (2023). ANALISIS KEKUATAN TEGANGAN GESER PADA BAJA KARBON SEDANG DARI HASIL SAMBUNGAN LAS TITIK. *JNSTA ADPERTISI JOURNAL*, *3*(2), 29–35. https://doi.org/10.62728/jnsta.v3i2.464

Huda, M. K. (2018). RANCANG BANGUN ALAT UJI IMPACT METODE CHARPY. *Mechonversio: Mechanical Engineering Journal*, *1*(1), 7. https://doi.org/10.51804/mmej.v1i1.348

Kharisma, A. A., & Yanuar, D. M. (2023). SIMULASI KEKUATAN RANGKA GANTRY CRANE SINGLE GIRDER MENGGUNAKAN METODE ELEMEN HINGGA (FINITE ELEMENT ANALYSIS). *Jurnal Ilmiah Teknologi Dan Rekayasa*, *28*(2), 146–158. https://doi.org/10.35760/tr.2023.v28i2.4606

Lesmana, S., Muvariz, M. F., Fadilah, N., Saputra, H., & Satoto, S. W. (2022). Studi Kekuatan Support Structure (Leg) pada Konstruksi FPSO Bulk Water Separation Module. *Jurnal Teknologi Dan Riset Terapan (JATRA)*, *4*(2), 45–51. https://doi.org/10.30871/jatra.v4i2.3650

Luthfi, M. (n.d.). *Failure Analysis of Drive Motor Bracket on Mini CNC Machine Using Finite Element Analysis Software*.

M, S., Jamaluddin, J., Aslim, A., & Satra, A. (2022). PERANCANGAN ALAT PRESS MATERIAL KOMPOSIT MENGGUNAKAN SISTEM PNEUMATIK. *ILTEK : Jurnal Teknologi*, *17*(02), 94–100. https://doi.org/10.47398/iltek.v17i02.18

N, R. P., Anggoro, P. W., & Yuniarto, A. T. (2022). Redesain Mesin Spinner pada UMKM Putri 21 untuk Mengurangi Getaran pada Foundational Bolt Menggunakan Finite Element Analysis Software SOLIDWORK 2018. *Jurnal Rekayasa Mesin*, *17*(1), 21. https://doi.org/10.32497/jrm.v17i1.2897

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

Nur Arini, R., & Pradana, R. (2021). ANALISA TEGANGAN REGANGAN PADA BALOK DENGAN MENGGUNAKAN SOFTWARE ABAQUS CAE V6.14. *Jurnal ARTESIS*, *1*(2), 193–198. https://doi.org/10.35814/artesis.v1i2.3227

Phady, A., Rajmi, A., Ramadhani, F., Andalan, M. T. P., Aski, S., & Alie, M. Z. M. (2020). PENGARUH OPTIMASI BEBAN RANGKA TUBULAR TERHADAP ANALISIS KEKUATAN TEKUK DAN KELELAHAN PADA FIXED OFFSHORE PLATFORM. *Riset Sains Dan Teknologi Kelautan*, 8–13. https://doi.org/10.62012/sensistek.v3i1.13233

Rizki, M. N., Fikri, A., Faisal, F., & Nanda, R. A. (2023). ANALISIS VON-MISES STRESS, STRAIN, DAN TOTAL DEFORMASI PADA PELAT IMPLAN METATARSOPHALANGEAL (MTP) DENGAN MATERIAL TI-6AL-4V MENGGUNAKAN FINITE ELEMENT METHOD. *Jurnal Teknologi Kimia Unimal*, *12*(2), 178–189. https://doi.org/10.29103/jtku.v12i2.12971

Setiyana, B., & Kurniawan, A. J. (2020). INVESTIGASI NUMERIK KEKUATAN STRUKTUR CHECK VALVE ½" 9K Psi MENGGUNAKAN FINITE ELEMENT SOFTWARE. *JURNAL ILMIAH MOMENTUM*, *16*(2). https://doi.org/10.36499/mim.v16i2.3760

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