JISO: Journal Of Industrial And Systems Optimization
Volume 8, Nomor 1, Juni 2025, 34-42
ISSN 2622-8971 online
ISSN 2522-898X print

End-to-End Engineering Workflow for Turmeric (Kunyit) Slicing Machines: CAD, FEM, and 3D Simulation

Tri Andi Setiawan[1], Anda Iviana Juniani[1,*], Pranidiya Otaviya Ningrum[1], Dhika Adithya Purnomo[1], Habib Ngumar Faruq[1], Khoirul Amin[2]
*E-mail Korespondensi: andaiviana@ppns.ac.id
[1]Engineering Design and Manufacture Study Program, Department of Marine Mechanical Engineering, Politeknik Perkapalan Negeri Surabaya, Indonesia
[2]Green Smart Manufacturing Study Program, Department of Industrial Management, National Formosa University, Taiwan

ABSTRACT

This study details an end-to-end engineering workflow for the development of a turmeric slicing machine, encompassing computer-aided design (CAD), finite element method (FEM) analysis, and physical testing. This research aims to design and analyze the structure of a slicing-type cable-cutting machine intended to meet industrial-scale cable-cutting demands. The machine design was developed using Autodesk Fusion 360 software, which supports comprehensive 3D modelling, stress analysis, and structural simulation. The study was conducted using the Finite Element Method (FEM) to evaluate strength, deformation, and stress distribution on the machine frame structure. Simulation results reveal a high safety factor, with minimum values exceeding safe operational thresholds, very low maximum deformation, and evenly distributed stress. This simulation indicates that the frame can withstand operational loads without structural failure. The workflow demonstrates the effectiveness of combining 3D CAD, FEM simulation, safety design, and experimental validation in producing reliable turmeric slicing machinery. It addresses a pronounced research gap identified in existing studies. Additionally, the design emphasizes material efficiency, ease of assembly, and stability during the cutting process. The study also includes mechanical calculations for key components such as transmission belts and shaft torque to ensure optimal real-world performance. This research successfully delivers a technically reliable machine design suitable for modern cable manufacturing applications by combining integrated design and simulation tools. The results demonstrate that simulation-based design significantly enhances performance prediction, minimizes errors, and accelerates product development.

Keywords: CAD, FEA *simulation, Fusion* 360, *turmeric slicing machine*, *structural analysis*

ABSTRAK

Penelitian ini memaparkan alur kerja teknik end-to-end untuk pengembangan mesin pengiris kunyit, yang meliputi desain berbantuan komputer (CAD), analisis metode elemen hingga (FEM), dan pengujian fisik. Penelitian ini bertujuan untuk merancang dan menganalisis struktur mesin pemotong kabel tipe pengiris yang dimaksudkan untuk memenuhi kebutuhan pemotongan kabel skala industri. Desain mesin dikembangkan dengan menggunakan perangkat lunak Autodesk Fusion 360, yang mendukung pemodelan 3D yang komprehensif, analisis tegangan, dan simulasi struktur. Penelitian dilakukan dengan menggunakan Finite Element Method (FEM) untuk mengevaluasi kekuatan, deformasi, dan distribusi tegangan pada struktur rangka mesin. Hasil simulasi menunjukkan faktor keamanan yang tinggi, dengan nilai minimum yang melebihi ambang batas operasional yang aman, deformasi maksimum yang sangat rendah, dan tegangan yang terdistribusi secara merata. Simulasi ini menunjukkan bahwa rangka dapat menahan beban operasional tanpa mengalami kegagalan struktural. Alur kerja ini menunjukkan keefektifan penggabungan CAD 3D, simulasi FEM, desain keselamatan, dan validasi eksperimental dalam menghasilkan mesin pengiris kunyit yang andal. Hal ini mengatasi kesenjangan penelitian yang diidentifikasi dalam penelitian yang sudah ada.

Desain ini menekankan efisiensi material, kemudahan perakitan, dan stabilitas selama proses pemotongan. Penelitian ini juga mencakup perhitungan mekanis untuk komponen-komponen utama seperti sabuk transmisi dan torsi poros untuk memastikan kinerja dunia nyata yang optimal. Output desain menunjukkan bahwa desain berbasis simulasi secara signifikan meningkatkan prediksi kinerja, meminimalkan kesalahan, dan mempercepat pengembangan produk. Simulasi menunjukkan bahwa desain memiliki faktor keamanan tinggi dengan nilai minimum melebihi batas aman, deformasi maksimum yang sangat kecil, serta distribusi tegangan yang merata. Hal ini menunjukkan bahwa rangka mesin mampu menahan beban operasional tanpa mengalami kegagalan struktural. Selain itu, desain memperhatikan efisiensi penggunaan bahan, kemudahan perakitan, dan stabilitas selama proses pemotongan. Studi ini juga menguraikan perhitungan mekanis komponen utama, seperti sabuk transmisi dan torsi poros, untuk memastikan kinerja optimal dalam operasional nyata. Dengan pendekatan desain terintegrasi dan simulasi berbasis perangkat lunak.

Keywords: analisis struktur, CAD Fusion 360, mesin perajang rempah, simulasi FEA,

INTRODUCTION

Advances in computer-aided design (CAD) and integrated finite element method (FEM) simulation tools are transforming mechanical design workflows by unifying 3D modeling, topology optimization, and engineering analysis within a single software environment. Autodesk Fusion 360 combines parametric CAD and an embedded FEA engine, enabling users to conduct iterative design and analysis cycles using a single model history. This literature review surveys recent research that applies Fusion 360 for both 3D design and finite element analysis (FEA) in mechanical engineering contexts, with a focus on works that explicitly leverage this integrated workflow.

Several studies demonstrate the use of Fusion 360's full platform for end-to-end mechanical design—beginning with 3D modeling, progressing through topology or generative optimization, and concluding with static stress simulation using the built-in FEA module. For example, Galon and Ang (Galon & Ang, 2024) present a case study in which a gripper jaw for a durian dehusking machine is reverse-engineered, weight-optimized via Fusion 360's Generative Design functionality, and then validated using its FEA simulation tools before fabrication by additive manufacturing. Similar approaches are observed in studies of robotic grippers, carabiners(Rizki et al., 2024), prosthetic fingers (Chaerunisa et al., 2024), tricycle frames (Syahid et al., 2024), and consumer products such as clothes hangers. In these cases, the studies report parametric or iterative workflows, where changes in CAD geometry propagate directly to updated FEA simulations.

A recurring theme is the application of generative design and topology optimization modules within Fusion 360, followed by FEA validation to ensure mechanical performance standards such as safety factor and maximum stress are met (Galon & Ang, 2024)(Azhagan M. et al., 2023). This approach is applied to a diverse set of mechanical systems, including drone frames (Azhagan M. et al., 2023), robot manipulators, and mechanical joints . The studies emphasize metrics such as von Mises stress, displacement, and safety factors, predominantly under linear static loading conditions.

Some papers make use of Fusion 360 alongside other design or simulation software, or focus primarily on parts of the workflow such as generative design, without explicitly documenting iterative CAD-to-FEA processes within the same file (Leen, 2017). However, the majority of recent literature utilizes Fusion 360's native design and simulation capabilities, reflecting the tool's maturity and accessibility for mechanical engineering applications. Typical practice involves iterative design modification, automatic re-solving of FEA studies, and, in some cases, experimental validation through prototype fabrication (Galon & Ang, 2024; Chaerunisa et al., 2024).

Collectively, these works illustrate that Fusion 360 is widely adopted as a unified platform for both 3D mechanical design and integrated FEM simulation, with its CAD–simulation loop being applied across a range of domains and mechanical components. The literature also reveals a focus on static stress analysis and weight reduction optimization, while advanced simulation features and cross-validation with external FEA tools remain less reported.

Setiawan et al (2023) discusses the development of generative design concepts and the importance of product redesign to improve functionality, usability, and manufacturing efficiency for products that have been on the market for a long time (Setiawan et al., 2023). The redesign process often faces communication challenges between designers and engineers due to differences in software and strategies. Autodesk Fusion 360 is used as an integrated CAD/CAM/CAE solution to strengthen product redesign simulations, covering industrial design, mechanical design, rendering, animation, and manufacturing (Song et al., 2018).

Fusion 360 combines engineering and analysis tools in a single platform, increasing efficiency and ease in collaborative product development(Song et al., 2018)(Onery Andy Saputra & Anwar Nurharyanto, 2023). Finite Element Analysis (FEA) simulation using Fusion 360 enables visualization and evaluation of material strength and stress. A case study on the frame of a three-wheeled cargo motorcycle involves size optimization by considering the weight of the rider and cargo, as well as straight and turning motion conditions. The structural analysis results show that increasing the bracket thickness through optimization significantly enhances the longitudinal bending and torsional strength of the high-load motorcycle frame (Setiawan et al., 2023). This approach allows for effective and safe evaluation of product durability and mechanical resistance prior to fabrication, minimizing risks to users.

This research aims to design a spice slicing machine using Fusion 360 software as the primary tool in the 3D design process and structural strength analysis. The main research question of this study is how CAD modeling and FEM simulation can be used to evaluate design performance and support decision-making in the mechanical engineering process for product redesign and development.

MATERIALS AND METHOD

This section will discuss the primary literature review supporting this article, as well as the research methods.

Finite Element Analysis

Finite Element Analysis (FEA) is a widely used and effective method for modeling in the field of biomechanics. Specific challenges in biomechanics include irregular geometries, complex microstructures, and loading conditions of biological tissues, all of which remain difficult to model accurately. However, using CT scan data to create knee joint models can simplify the analysis process. Prior to implantation, various aspects of the knee implant must be evaluated, including material properties, dimensions, shape, surface treatment, load resistance, and potential failure. In this study, FEA is applied to two different types of materials (Salimi, 2023).

Autodesk Fusion 360

Autodesk Fusion 360 is a comprehensive software platform that integrates 3D computer-aided design (CAD), engineering (CAE), and manufacturing (CAM). It is widely utilized by engineers, designers, and manufacturers for developing and prototyping products (Rahman et al., 2020; Sholeh et al., 2018). The software offers a wide range of features that support efficient product redesign. One of its key strengths is the ability to create parametric 3D models, allowing users to easily modify dimensions, angles, and features, which facilitates rapid and flexible design changes. Fusion 360 also supports cloud-based collaboration, enabling multiple team members to work on a project simultaneously and track design changes through built-in version control tools. This collaborative environment enhances teamwork and improves efficiency during the redesign process. Additionally, the platform includes advanced simulation and analysis tools that help engineers evaluate the structural integrity and performance of a redesigned product under various conditions, allowing them to detect potential flaws before manufacturing. Realistic rendering and visualization capabilities enable the creation of high-quality images that help clients and stakeholders better understand and evaluate proposed design changes. Furthermore, Fusion 360 is equipped with integrated CAM tools that streamline the transition from design to manufacturing, providing functions such as toolpath generation, machining simulation, and CNC code output. Overall, Autodesk Fusion 360 delivers a robust and efficient workflow that supports every stage of the product redesign process, from conceptualization to final production.

RESEARCH METHOD

This research begins with a literature study to identify knowledge gaps related to the discussed topic. After the research problem is defined, the research objectives are clearly formulated. Next, the design and modeling process of the spice slicing machine is carried out using Fusion 360 software as a tool for design optimization. The initial stage involves redesigning the product in the form of a CAD model. Additionally, analysis using the Finite Element Method (FEM) requires a meshing process to numerically model the machine frame structure after all necessary data is entered into the solver. The subsequent stages include defining the loading and boundary conditions before running the FEM simulation. The simulation results are then verified to ensure the accuracy of the calculations using the FEM approach. Figure 1 illustrates the research methodology, as elaborated in the subsequent section. Figure 2 presents a comprehensive overview of the inputs, outputs, tools, and the sequential workflow employed in this study.

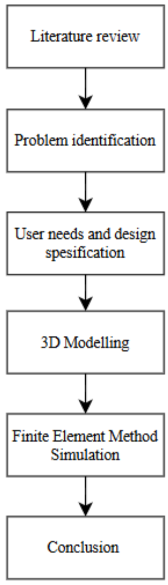

Figure 1. Research Methodology

RESULT AND DISCUSSION

Before presenting the results and discussion, it is important to outline the design and analysis framework applied in this research. The development of the spice slicing machine aimed to produce a reliable, efficient, and manufacturable

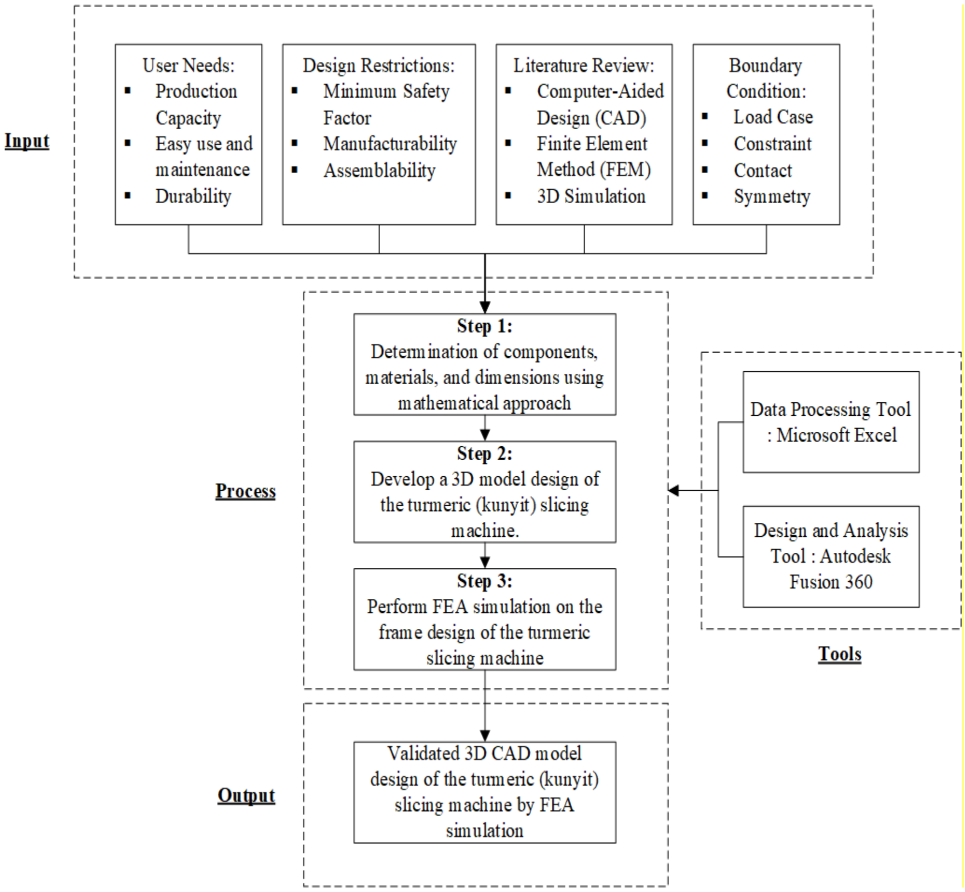

Figure 2. Research Framework Visualization

design that meets the operational requirements for large-scale spice processing. The machine was modeled using computer-aided design (CAD) tools to ensure dimensional accuracy and structural feasibility. After the modeling stage, structural analysis was conducted using the Finite Element Method (FEM) to evaluate the machine frame's response to various mechanical loads. This preparatory step was crucial in validating the structural integrity of the design before fabrication. The following sections detail the analysis results and provide an in-depth discussion regarding the machine's performance, durability, and suitability for real working conditions.

3D Design of Spice Slicing Machine

The 3D design of the spice slicing machine developed in this research was created using Autodesk Fusion 360 software. This design is intended to meet the needs of large-scale home industries to medium-scale industries that require an automatic, high-capacity spice cutting process. In addition to considering efficiency and productivity, the design also takes into account structural strength, user safety, as well as ease of assembly and maintenance.

The machine frame is constructed from square steel hollow profiles to provide rigidity and stability during operation. The structure is designed in a cubic form with horizontal and vertical supports on each side, intended to withstand the dynamic loads from shaft rotation and reaction forces during the cutting process. The motor is positioned at the bottom, allowing the machine's center of mass to remain low to maintain stability. This frame design also allows open access from multiple sides, making maintenance and internal component replacement easier.

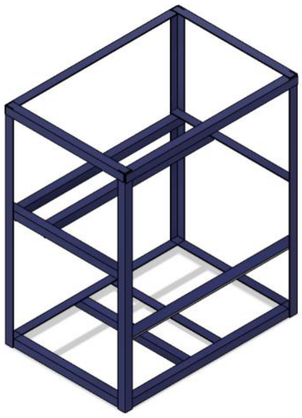

Figure 3. 3D Model of the Machine Frame

This machine uses an electric motor as the main drive, which is connected to the blade rotation shaft through a belt-pulley transmission system. This transmission system was chosen for its ability to dampen vibrations, ease of installation, and flexibility in adjusting rotational speed. The diameters of the driving pulley and the pulley mounted on the shaft have been calculated to achieve a speed ratio that matches the spice cutting capacity. The motor is positioned coaxially with the main shaft direction but located beneath it, allowing the transmission belt to form a specific angle to avoid direct interference from cutting debris.

P = F × V

P = F × ω × r

P = F × $(\frac{2\,\pi\, n}{60} \times r)$

P = 210 $(\frac{2.3{,}14\,.\,641}{60} \times 0{,}075)$

P = 1056 W

Then,

Pd $= \frac{1056}{0.96 \times 0.9}$

Pd = 1222,2 W

Based on these calculations, an electric motor with an actual power of 2200 watts (2.2 kW) and a rotational speed of 3000 rpm was selected.

The next step, main shaft is designed to transmit torque from the transmission system to the blade disc. It is mounted using two bearing housings embedded in the frame. These bearings serve to reduce rotational friction and ensure that the shaft remains precisely aligned. The shaft diameter was selected based on the results of torsional and bending load analysis, ensuring that no significant deformation occurs during the cutting process.

Pulley Selection

$$i = \frac{n1}{n2}$$

$$i = \frac{2800}{6410}$$

i = 0,4

Then,

$$i \;=\; \frac{Dp2}{Dp1}\,(1 - \frac{0{,}3}{100})$$

$$0{,}4 = \frac{x}{6}\,(1 - \frac{0{,}3}{100})$$

x = 2.5 in

If the planned diameter of pulley 1 is 2.5 inches, then the second pulley has a diameter of 6 inches.

The next step is the calculation and selection of the actual V-belt.
C' = 76 × 1,5 = 228.6 mm

L' = C'.2 + 1,57 (Dp2 + Dp1) + $\frac{(Dp2-Dp1)^2}{4C'}$

L' = 228.6× 2 + 1,57 (152.4+63,5) + $\frac{(152.4-63{,}5)^2}{4x228{,}6}$

L' = 804,8 mm

From the calculation above, the actual V-belt used is V-belt A32 with an internal length of 813 mm.

The next step is the calculation of the torsional moment to determine the shaft diameter.

T = 9,74 × $10^5\,\frac{pd}{n2}$

T = 9,74 × 10^5 × $\frac{0{,}114}{2564}$

T = 43,305 kg.mm

Next is the calculation of shaft shear stress.

$$\tau\alpha = \frac{\sigma b}{Sf1\ \times Sf2}$$

$$\tau\alpha = \frac{65{,}89}{5{,}6\ \times 1{,}3}$$

$$\tau\alpha = 9\ kg/mm2$$

The next step is to calculate the actual shaft diameter.

$$D_s \geq [\frac{5{,}1}{\tau\alpha} \times KtCbT]^{\frac{1}{3}}$$

$$D_s \geq \left[\frac{5{,}1}{9} \times 1 \times 1 \times 43{,}305\right]^{\frac{1}{3}}$$

$D_s \geq 2.9$ mm

The shaft diameter calculation results show that the diameter must be greater than 2.9 mm. Therefore, the chosen shaft diameter is 20 mm. This size is considered safe because it is larger than the minimum limit.

The main cutting component is a metal disc equipped with slots or blade holders on several sides. This disc is rigidly mounted on the main shaft and rotates at a speed corresponding to the motor's rotation. The slots are positioned in such a way that the resulting cuts have uniform thickness. The number of blades on the disc is determined by considering cutting speed, the force exerted on the material, and the maximum centrifugal force that can be safely withstood by the mechanical joints between components.

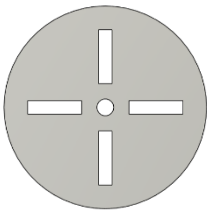

Figure 4. 3D Model of the Machine Blade

The spices are fed into the top part of the machine and directed toward the cutting blade path by gravity or a feeder system. When the machine is turned on, the motor rotates the shaft and cutting disc through the belt-pulley system. As the material contacts the rotating blades, it is quickly sliced and falls downward in uniform pieces. This design ensures a continuous and automatic process with a high and consistent cutting rate. The machine is also capable of long-duration operation because the mechanical system has been designed to efficiently withstand repetitive forces.

Finite Element Simulation

In the machine design development process, strength and structural behavior analysis is a crucial step to ensure the design functions properly and safely under real working conditions. One common method used to analyze the mechanical response of a component is the Finite Element Method (FEM). This method allows detailed modeling of stress, strain, and deformation distribution on objects with complex geometries.

Fusion 360, as an integrated Computer-Aided Design (CAD) software, includes a FEM simulation module that facilitates efficient analysis. With Fusion 360, users can perform structural analysis directly on the created 3D model without the need to switch between complicated software. This accelerates design iterations and minimizes potential errors that may occur during manual evaluation.

This section explains the FEM analysis process using Fusion 360, starting from 3D model preparation, assigning materials and boundary conditions, to interpreting simulation results. Figure 5 illustrates the boundary conditions applied for FEM analysis on the spice slicing machine frame.

The frame, modeled using Autodesk Fusion 360, is constructed from square hollow steel profiles arranged in a cubic structure to ensure rigidity and stability. The blue arrows indicate vertical loads applied at the base, representing the weight of machine components and operational forces during cutting process.

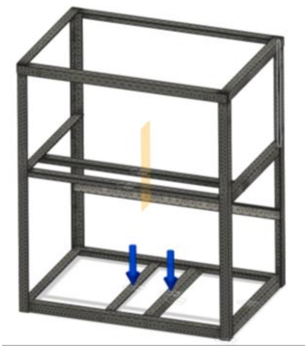

Figure 5. Boundary Conditions for FEM Analysis

The purpose is to analyze how the applied loads are distributed throughout the frame, to assess potential deformation, and to evaluate the structural strength and stability under actual operating conditions. Such analysis is essential to ensure that the frame can withstand the operational stresses without experiencing structural failure.

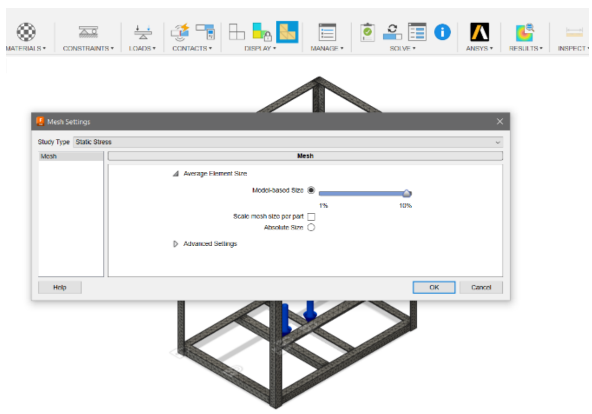

Figure 6. Mesh Specifications for FEM Analysis

A safety factor analysis was conducted to determine the structural strength of the machine frame in withstanding operational loads. The simulation was performed using the Finite Element Method (FEM) in Fusion 360 software. The simulation results showed the distribution of safety factor values across the entire machine frame based on the applied loads and specified support points. The minimum safety factor obtained was 15, indicating that the frame design has strength far exceeding the imposed working loads. This value demonstrates that the frame structure is very safe and carries no risk of failure under normal operating conditions. Critical areas in the design showed no significant stress concentrations. This simulation serves as evidence that the spice slicing machine frame design meets structural safety requirements well and is ready for manufacturing or further testing (see Figure 6).

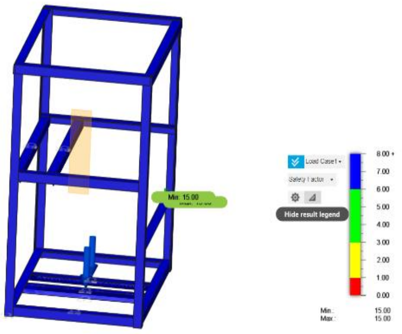

Figure 7. Safety Factor Results from FEM Simulation

Displacement analysis was conducted to determine the extent of movement or positional change of structural elements under load. The figure shows the total displacement simulation results of the frame structure using the Finite Element Method (FEM) in Fusion 360 software. The simulation revealed a maximum displacement value of 1.73×10^{-8} inches and a minimum displacement of 0.00 inches. These very small values indicate that the frame structure is highly rigid and does not undergo significant deformation when subjected to load.

The color distribution in the image indicates that the largest displacement occurs at the lower area of the frame, while most of the structure remains stable. This demonstrates that the frame design can maintain its shape and stability during use and is suitable to withstand the working loads of the spice shredding machine components.

This simulation reinforces the conclusion that the frame design has good structural performance and is safe for use under normal operating conditions (see Figure 7).

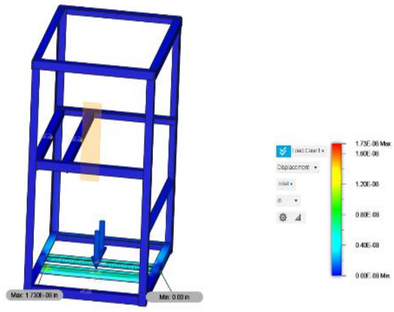

Figure 8. Displacement Results from FEM Simulation

Stress analysis was conducted to determine the magnitude of internal forces acting on structural elements due to applied loads. This simulation used the von Mises stress criterion, commonly applied in evaluating material failure under combined loading conditions.

The simulation results showed a maximum stress value of 1,566 psi and a minimum stress of 0.00 psi. This maximum stress is very low compared to the yield strength of carbon steel (approximately 36,000 psi), indicating that the structure operates well below its maximum capacity.

The stress distribution, depicted through color gradients, shows the highest stress concentration at the lower part of the structure, precisely where the load is applied. Meanwhile, most of the remaining structure experiences very low or even zero stress, indicating that the frame design is very safe from risks of plastic deformation or structural failure.

These results indicate that the frame design has a high safety factor and is reliable for use under normal working loads. Thus, from the material strength perspective, this structure is suitable for realization (see Figure Z).

Figure 9. Maximum Stress Results from FEM Simulation

CONCLUSION

This research utilizes Autodesk Fusion 360 to strengthen product design simulations. The argument for using Autodesk Fusion 360 for CAD/CAM modelling is that this tool can combine engineering and analysis tools on a single platform, increasing efficiency and simplicity in collaborative product design development. CAE studies were performed on the bracket using Autodesk Fusion 360 to simulate its performance under realistic loading conditions and constraints, enabling the determination of stresses, displacements, and safety factors. Autodesk Inventor is extremely useful for product design, tooling, mechanical design, and simulation.

Using the 3D design model of the frame of a three-wheeled cargo motorcycle, a size optimization design was performed to enhance frame strength, and the following results were obtained. The loading and boundary conditions for the frame of a three-wheeled cargo motorcycle took into account the weight of the driver and cargo, as well as the motion conditions that may occur during straight-driving and curve-driving. Structural analysis during the design stage results the assessment of three-wheeled cargo motorcycle in terms of their durability and mechanical resistance is possible without putting the user at risk. Increasing the thickness of the bracket through size optimization had the greatest impact on enhancing the longitudinal bending and torsional strength of the frame of a high-load three-wheeled cargo motorcycle.

Notwithstanding the useful insights derived from this study, certain limitations must be recognised. The prototype has not undergone experimental validation, perhaps restricting the generalisability of the findings to various operational scenarios. Moreover, future study is required to utilise prototypes to substantiate that simulation-based design greatly improves performance prediction, reduces errors, and expedites product development.

REFERENCE

Azhagan M., T., Shanmugam, R., Khan, S., & Lata, S. (2023). *Design Optimization of Hexacopter Frame Using Generative Design and Additive Manufacturing*. *Volume 3: Advanced Manufacturing*, V003T03A048. https://doi.org/10.1115/IMECE2023-111791

Chaerunisa, S. T., Kumarajati, D. Y. H., & Sugianto, W. (2024). *PEMODELAN DAN ANALISIS DESAIN PROSTETIK JARI MENGGUNAKAN METODE ELEMEN HINGGA (FEM)*. *7*(3), 1–7.

Galon, M. A. T., & Ang, E. B. (2024). Development, fabrication, and assessment of durian dehusking machine's gripper by direct metal laser sintering-powder bed fusion using AlSi10Mg. In H. Tsukamoto (Ed.), *Seventh International Conference on Mechanical Manufacturing and Industrial Engineering (MMIE 2024)* (Vol. 13290, p. 1329007). SPIE. https://doi.org/10.1117/12.3051363

Leen, R. (2017). Shape optimisation of a snowboard binding highback. A case study of generative design process comparison. *KnE Engineering*, *2*(2), 73. https://doi.org/10.18502/keg.v2i2.598

Onery Andy Saputra, & Anwar Nurharyanto. (2023). Pelatihan Desain Dan Manufaktur Menggunakan Software Autodesk Fusion 360 Untuk Guru Produktif Dan Siswa Program Studi Teknik Dan Bisnis Sepeda Motor Smk Negeri 1 Selo. *Proficio*, *5*(1), 79–84. https://doi.org/10.36728/jpf.v5i1.2927

Rahman, M. F. F., Juniani, A. I., & Setiawan, T. A. (2020). Perancangan Jok Ergonomis Dalam Fabrikasi Mobil Minimalis Roda Tiga. *Proceedings Conference on Design Manufacture Engineering and Its Application*, *2654*.

Rizki, M., Gamayel, A., & Zaenudin, M. (2024). Simulation on the influence of the shape of the carabiner as a hanging accessory on stress distribution using Autodesk Fusion 360. *JTTM : Jurnal Terapan Teknik Mesin*, *5*(1), 33–40. https://doi.org/10.37373/jttm.v5i1.779

Salimi, A. A. (2023). Simulasi Finite Element Analysis (Fea) Dan Fabrikasi Implan Untuk Perbandingan Kinerja Implan Kovensional Dan Custom Cruciate Retaining (Cr) Untuk Penderita Osteoarthritis Berdasarkan Rekonstruksi Data Mri/Ct Scan Tulang Lutut. *Simulasi Finite Element Analysis (Fea) Dan Fabrikasi Implan Untuk Perbandingan Kinerja Implan Kovensional Dan Custom Cruciate Retaining (Cr) Untuk Penderita Osteoarthritis Berdasarkan Rekonstruksi Data Mri/Ct Scan Tulang Lutut*, 1–115.

Setiawan, T. A., Juniani, A. I., Purnomo, D. A., Rinanto, N., & Faruq, H. N. (2023). How To Utilize Autodesk Fusion 360 That Reinforces Product Redesign Simulation? *JISO : Journal of Industrial and Systems Optimization*, *6*(1), 48. https://doi.org/10.51804/jiso.v6i1.48-54

Sholeh, A., Iviana Juniani, A., & Novrita Devi, Y. (2018). Analisis dan Perancangan Sepeda Statis untuk Rehabilitasi Penderita Stroke. *Politeknik Perkapalan Negeri Surabaya*, 11–16.

Song, P. P., Qi, Y. M., & Cai, D. C. (2018). Research and Application of Autodesk Fusion360 in Industrial Design. *IOP Conference Series:*

Materials Science and Engineering, *359*(1). https://doi.org/10.1088/1757-899X/359/1/012037.

Syahid, M., Rudi, A., Hayat, A., Kasim, L., Mochtar, A. A., & Ansar, M. (2024). Simulation of Static Loading of Electric Tricycle Frame. *Journal of Physics: Conference Series*, *2739*(1). https://doi.org/10.1088/1742-6596/2739/1/012039