Finite Element Simulation of Power Weeder Machine Frame

by Teknologi Industri

Submission date: 12-Aug-2023 08:02PM (UTC-0400)
Submission ID: 2144877519
File name: 291-37-1019-2-10-20221004_2_1.pdf (544.71K)
Word count: 4396
Character count: 23636

IJoCED

Indonesian Journal of Computing, Engineering, and Design

Journal homepage: http://ojs.sampoernauniversity.ac.id/index.php/IJOCED

Finite Element Simulation of Power Weeder Machine Frame

Angger Bagus Prasetiyo[1], Kartinasari Ayuhikmatin Sekarjati[2]*

[1] Department of Mechanical Engineering, Faculty of Industrial Technology, Institut Teknologi Nasional Yogyakarta, Indonesia

[2] Department of Industrial Engineering, Faculty of Industrial Technology, Institut Sains & Teknologi AKPRIND Yogyakarta, Indonesia

Correspondence: E-mail: angger.bagus@itny.ac.id

ABSTRACT

One of the agricultural technologies that have evolved recently is the weeding machine. This device was developed to aid farmers in the weed-control procedure. The engine frame is one of the primary elements of the weed wacker. The most crucial component of agricultural equipment is the frame or chassis since it serves as the foundation for mounting other parts. To examine the stress and strength of the frame, this study models the frame of a power weeder machine utilizing finite element analysis using the SolidWorks 2022 program. A total load of 120 N is applied to the AISI 316L used for the frame. The analysis's findings indicate that the maximum von Mises stress value is about 2.51 $10^7 N/m^2$, the greatest deformation is 0.556 mm, and the safety factor is 6.8. With AISI 316L material, the frame design of the power weeder machine in this study is safe to endure up to 140 N.

ARTICLE INFO

Article History:
Received 17 Jul 2022
Revised 22 Aug 2022
Accepted 13 Sep 2022
Available online 03 Oct 2022

Keywords:
Finite element analysis,
Frame,
Power weeder,
SolidWorks.

1. INTRODUCTION

Many ergonomic studies have been conducted on operator comfort and safety during field operations due to the significant importance of agricultural work in a nation. In addition, there is necessity to have a better design in order to gain higher productivity, to increase comfort, and to increase safety (Upendar et al., 2018). Machines become more and more prevalent in daily life, which makes human labor easier and more productive (Oktaviandri and Paramasivam, 2020). One of the agricultural technologies currently being explored is the weeding machine. This device was created to aid farmers in the weed-control procedure. The engine frame is one of the primary elements of the weed wacker.

The most crucial component of agricultural equipment is the frame or chassis since it serves as the foundation for mounting other parts. To support the majority of the load brought on by other components attached to the frame, the frame must be constructed robustly. Prior designing the weeding machine, many factors need to be taken into account, including aesthetics, safety, convenience, ease of use while in operation, and component safety factors, notably the weed weeding machine structure (Awwaluddin, 2019). The machine's combination of components, which can be bought separately on the market, still serves its intended purpose.

Determining the load on the frame must also be taken into account when constructing the design of the power weeder machine. Knowing the forces affecting the power weeder machine from this is helpful. As a result, the load application needs to be taken into account when designing the power weeder machine.

The power weeder tool is made to remove weeds from the muddy ground. This could result in an excessive push and load. There will be inertia of forces acting on the entire frame if it moves faster and has no mass (Soden et al., 1986). If the power weeder machine's frame has high flexibility, it will revert to its initial position (Baihaki and Bintoro, 2021).

Prior to manufacture the machine, a design must be created using computer design technology (CAD/CAM). This technology can reduce major expenses associated with design flaws (Chirende, Li, and Vheremu 2019; Cekus et al., 2019). Design optimization can reduce manufacturing errors and can lengthen the product's service life (Vegad and Yadav, 2018). Technology-based design optimization has been used by cutting-edge businesses working in mechanics and other types of structures. (Gheorghe et al., 2018). Software like SolidWorks and ANSYS (Al-Shammari and Al-Waily, 2018; Al-Shammari and Abdullah, 2018) uses finite element analysis techniques as an efficient tool to discover solutions to challenging problems and can solve many engineering difficulties effectively (Al-Shammari et al., 2020).

The power weeder machine frame needs to be analyzed using the finite element analysis method to assess its strength. This approach has been widely used by designers and engineers to analyze Von Misses stress, displacement, and safety factors (Lu et al., 2019), optimize a design, carry out theoretical calculations (Lai, Yu, and Dong, 2019; Kešner et al., 2021), and numerical simulation for forecasting a material's strength and confirming the loading force (Azimi-Nejadian et al., 2019; Yin and Xu, 2018). Numerical analysis can also be used to determine the impact of deformation and safety factor of a design (Prasetiyo, Sekarjati, and Haryo, 2022).

The results from earlier research that used the finite element approach to assess the level of strength of bicycle frame constructions consisting of composites and steel revealed that while the von Misses stress was higher, the deformation and strain values were lower (Kubasad, 2018). The strength of the frame of the weed-eating machine was next examined in more detail using the finite element approach and various load changes. The findings indicate that the safety factor's value rises as the load increased (Suprapto and Wibawa, 2021).

Based on the aforementioned description of the issue, the goal of this study is to use SolidWorks 2022 software and the finite element analysis method to examine the weed-weeding machine's frame construction. The minimum value of the safety factor is one of the references used as a gauge of a design's safety (Elishakoff, 2004). Engineers frequently utilize an object's safety factor as a reference metric when determining how much stress it can withstand (Wang et al., 2019). A material's capacity to sustain dynamic loads and shock loads determines how safe it is (K. Z. V. Dobrovolsky, 1973). This study's results include von Mises stress, displacement, and safety factor.

2. RESEARCH METHODOLOGY

Von Misses stress, displacement, and safety factor are among the simulation outcome parameters. The following equation can be used to calculate theoretically the value of strain and stress:

$$\begin{Bmatrix}\sigma_{xx}\\ \sigma_{yy}\\ \sigma_{zz}\end{Bmatrix} = \frac{E}{(1+v)(1-2v)}\begin{Bmatrix}(1-v)\varepsilon_x + v\varepsilon_y + v\varepsilon_z\\ v\varepsilon_x + (1-v)\varepsilon_y + v\varepsilon_z\\ v\varepsilon_z + \varepsilon_y + (1-v)\varepsilon_z\end{Bmatrix}$$

where σ is stress, ε is strain, v is poison ratio dan E is modulus young of material. The following equation can be used to calculate the safety factor's value:

$$SF = \frac{\sigma_{max}}{\sigma_{max\ material}}$$

where SF is a safety factor, σ_{max} is allowable material stress, $\sigma_{max\ material}$ is stress on the material.

2.1. Finite Element Analysis

The software can now be used to address technical issues, and the results can be used to demonstrate if a product is damaged or worn, or even whether it performs as intended (Jweeg et al., 2021). By breaking down the resulting item into elements using the finite element analysis approach, SolidWorks is one of the programs used to solve engineering issues that are described by partial differential equations (Al-Waily and Ali, 2015; Chiad, Al-Waily, and Al-Shammari, 2018). Three linked file types are produced by SolidWorks' use of parametric design principles: components, assemblies, and drawings.

The other two files will therefore be updated if any changes are made to one of these three files. SolidWorks can examine issues to find the best design so that products don't need to be manufactured or manufactured before an error occurs, saving time and money and lowering the number of prototypes needed. SolidWorks can show every component of the design in detail, to check accurate mass properties, and to check problems (Suprapto and Wibawa, 2021).

Differential equations are typically challenging to solve analytically, necessitating help to complete these tasks. Partial differential equations can be converted into linear algebraic equations using the finite element approach (Stolarski, T.; Nakasone, T.; Yoshimoto, 2006). The finite element approach can be used to tackle a variety of issues, including issues with structural analysis, buckling (buckling), and vibration analysis (Dantulwar, Maske, and Patel, 2017; Popa et al., 2021). Structural analysis is the finite element analysis technique that is most widely employed.

The term "structure" in this context refers to mechanical, aeronautical, and naval structures as well as buildings and bridges. The static structural analysis ignores the effects of inertia and damping while accounting for displacements, stresses, strains, and forces on the structure as a result of loading. Both linear and nonlinear analyses of static structures are possible (Ansys Release 2013; Ansys Release Documentation, 2005). It is possible to examine a structure's failure to reduce the fault function, produce reliable solutions, and address technical issues (Doustdar and Kazemi, 2019).

2.2. Model and Materials

SolidWorks 2022 software was used to assist in the design of the power weeder machine frame model. To make it simpler for farmers to operate, the frame was designed in accordance with the conditions in the field. The frame's design also allows for simple construction and mobility. **Figure 1** depicts the power weeder machine's frame design.

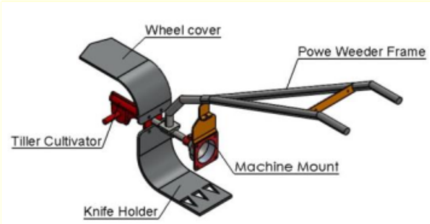

Figure 1. Power weeder machine frame

AISI 316L was the material utilized in the construction of the power weeder machine frame in this study. It was chosen because it is heat- and corrosion-resistant (Kelly, 2015). **Table 1** provides an overview of the parameters used in the simulation.

Table 1. AISI 316L simulation parameters

Description	Value
Name Material	AISI Type 316L
Model type	Isotropic
Yield Strength	$1{,}7x10^{8}$ N/m^2
Tensile Strength	$4{,}85x10^{8}$ N/m^2
Elastic Modulus	$2x10^{11}$ N/m^2
Poison's Ratio	0,265
Mass Density	8.027 kg/m^2
Shear Modulus	$8{,}2x10^{10}$ N/m^2
Thermal Expansion	$1{,}65x10^{-5}$ /Kelvin
Load Machine	80 N
Load Cultivator Tiller	40 N
Load	20 N

2.3. Mesh simulation modeling

Mesh has an impact on computational modelling utilizing the Finite Element Analysis (FEA) technique (Doustdar and Kazemi, 2019). In a simulation, mesh is a process that has a high level of complexity (Sosnowski et al., 2018). Mesh results have a significant impact on the simulation's convergence outcomes (Sosnowski, Krzywanski, and Scurek, 2019; Prasetiyo et al., 2019). The simulation may fail as a result of an error made during creating the mesh, which means that mesh generation process must be reperformed, which is time-consuming. The findings is more accurate with smaller meshes, but the simulation procedure takes longer. (García Pérez and Vakkilainen, 2019; Prasetiyo et al., 2019).

Table 2. Mesh distribution information

Description	Value
Mesh Type	Tetrahedral
Mesher Used	Curvature
Maximum Element	2,69609 cm
Minimum Element	0,539218 cm
Mesh Quality	Draft
Total Nodes	7808
Total Elements	25599
Maximum Aspect Ratio	2.558,3
Percentage Aspect Ratio <3	91,6 %
Percentage Aspect Ratio >10	0,43 %

Hexahedral mesh, polyhedral mesh, and tetrahedral mesh are some of the mesh types utilized in computational fluid dynamics (CFD) simulation (Chen et al., 2021; Sosnowski et al., 2018b). The tetrahedral mesh was employed in this study because it is more effective for simulating stress distributions (Hutton, 2003) and CFD simulations are frequently used in irregular geometries (Chen et al., 2021). Overall, **Table 2** provides thorough information on the mesh distribution used in this investigation.

Calculations utilizing the finite element approach must be performed using a computer due to many equations involved. This approach is cost and time efficient while also ensuring the accuracy of the results. The fundamental idea behind the finite element approach is to discretize an item into a finite number of parts. This section takes the shape of a triangle, with each element being a linear quadrilateral connected by a node (node). Further information on this is shown on **Figure 2**.

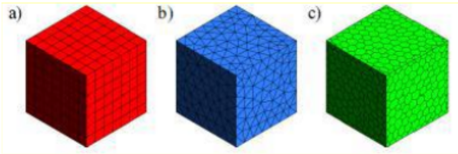

Figure 2. (a) Mesh hexahedral (b) Mesh tetrahedral (c) Mesh polyhedral on finite element method (Sosnowski, 2018)

3. RESULTS AND DISCUSSION

3.1. Von Mises Stress

The material choice and frame design must be based on the maximum value of working stress, analysis of the stresses occurring on a structure is crucial. Based on the findings of the uniaxial tensile test, the von Mises stress is utilized to forecast the yield of materials subjected to complicated loading (Suprapto and Wibawa, 2021). The Von Mises stress determines whether a material will be safe or fail (Karmankar, 2017). If the stress value exceeds the material strength, von Mises can fail (Vutton D. V., 2003). **Figure 3** displays the outcomes of the simulation analysis of the power weeder machine frame model created using the software SolidWorks 2022.

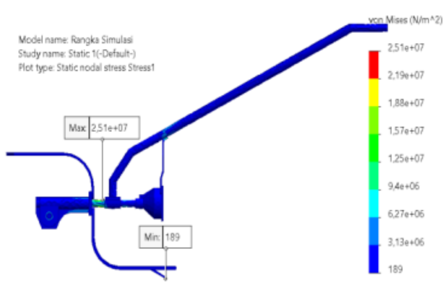

Figure 3. Von Mises stress simulation results for power weeder machine

Figure 3 demonstrates how the stress is spread equally throughout the frame, tiller cultivator, and engine support. The weeding portion has the lowest value of von Mises stress at 189 N/m^2, while the connection section between the tiller cultivator and the connecting pipe to the frame and engine has the highest value at 2.51 x 107 N/m^2. The amount of force applied to an object will immediately affect the amount of stress (Pranoto and Mahardika, 2018). When the von Mises stress reaches a certain level, known as the yield strength, a material is said to begin yielding.

3.2. Deformation

Deformation is a physical alteration to an object brought on by a load or force. Elastic deformation and plastic deformation are the two categories into which deformation is further separated (Juvinall, 2011). When an object undergoes elastic deformation, which is a physical change brought on by a force or load, it will revert to its original shape (Juvinall, 1967). Naturally, elastic deformation is used while developing tools since the maximum stress is constrained below the yield strength (K. Z. V. Dobrovolsky, 1973).

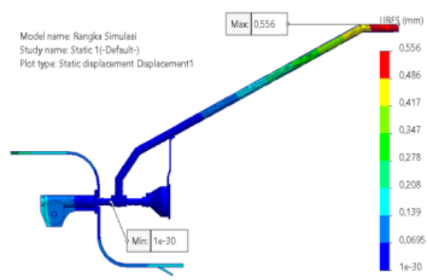

Figure 4. Power weeder machine frame deformation simulation results

Figure 4 displays the results of the material's deformation value against the load. The weeding section has the smallest deformation value at 1 x 10^{-30} mm, while the frame holder has the largest deformation value at 0.556 mm. This demonstrates that material is stronger the less it deforms (Hertzberg et al, 2012). The simulation findings demonstrate that even under high loads, the component does not experience significant deformation. The component is damaged if it cannot resist the applied load.

3.3. Safety Factor

When performing stress testing on a model of an object, one of the parameters used as a reference is the safety factor (Wang et al. 2019). To prevent a failure and establish operability of the tool design, considering safety factor is necessary. The review procedure uses the safety factor, which ensures the proposed design is secure and serves as a gauge for an element's strength (Wibawa et al., 2020).

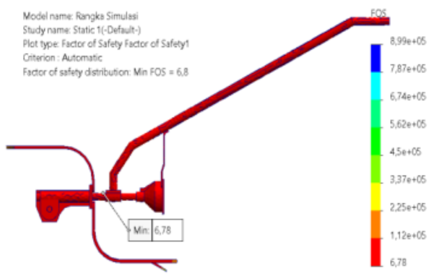

Figure 5. Safety Factor Simulation Results for Power Weeder Machine

Figure 5 displays the results of the modelling simulation. The frame design that has been created is extremely suited to withstand a load of 160 N since the result of the safety factor value in the simulation of the loading of the power weeder machine frame surpasses the necessary numerical value. A good model's safety factor has a value closer to 1. (Elishakoff, 2004; Pratama and Mahardika, 2018). While the safety factor for a material that can bear dynamic loads is between two and three (K. Z. V. Dobrovolsky, 1973).

4. CONCLUSION

The finite element study of power weeder machine frame made of AISI 316L was successfully conducted using SolidWorks 2022 software. The power weeder machine frame simulation yielded a maximum von mises stress of 2.51 x 10^7 N/m^2. In the simulation of the power weeder machine frame, the maximum deformation result is 0.556 mm. The power weeder machine frame's minimal safety factor simulation yielded a 6.8 with an AISI 316L material type result. Using AISI 316L steel, the suggested power weeder machine frame design can be deemed safe to bear loads up to 140N.

REFERENCES

Al-Shammari, M. A., Qasim , H. B., Al-Waily, M., & Hasson, A. (2020). Fatigue Behavior of Steel Beam Coated with Nanoparticles under High Temperature. Journal of Mechanical Engineering Research and Developments, 43(4), 287-298.

Al-Waily, M., & Abud Almalik Abud Ali, Z. (2015). A Suggested Analytical Solution of Powder Reinforcement Effect on Buckling Load for Isotropic Mat and Short Hyper Composite Materials Plate. International Journal of Mechanical and Mechatronics Engineering, 15(4), 80–95.

Ansys Release. (2013). ANSYS Mechanical APDL Thermal Analysis Guide. Analysis 16.1, 26–34. Retrieved from Ansys Release.

Ansys Release Documentation. (2005). ANSYS Workbench Release 10.0. ANSYS Workbench Release 10.0. Retrieved from http://kashanu.ac.ir/Files/Content/ANSYS Workbench.pdf.

Awwaluddin, M. (2019). Analisa Kekuatan Rangka Sepeda Listrik Menggunakan Software

SolidWorks. Journal of Technical Engineering, 3(1), 5-16. doi:https://doi.org/http://dx.doi.org/10.32493/pjte.v3i1.7486

Baihaki, M. I., & Bintoro, C. (2021). Aplikasi Desain Generatif Pada Rangka Sepeda Dengan Menggunakan Perangkat Lunak Solidworks Dan Matlab Simulink. Jurnal INOVTEK Seri Mesin, 2(1), 2-5. Retrieved from http://ejournal.polbeng.ac.id/index.php/ISM/article/view/2329

Cekus, D., Gnatowska, R., Paweł, K., & Šofer, M. (2019). Simulation Research of a Wind Turbine Using SolidWorks Software. Journal of Physics: Conference Series, 1398(1). doi:https://doi.org/10.1088/1742-6596/1398/1/012001

Chen, H., Zhou, X., Feng, Z., & Cao, S. J. (2021). Application of polyhedral meshing strategy in indoor environment simulation: Model accuracy and computing time. Indoor and Built Environment, 1-13. doi:https://doi.org/10.1177/1420326X211027620

Chiad, J. S., Al-Waily, M., & Al-Shammari, M. A. (2018). Buckling Investigation of Isotropic Composite Plate Reinforced by Different Types of Powders. International Journal of Mechanical Engineering and Technology, 9(9), 305-317.

Chirende, B., Li, J. Q., & Vheremu, W. (2019). Application of Finite Element Analysis in Modeling of Bionic Harrowing Discs. Biomimetics, 4(3), 1-11. doi:https://doi.org/10.3390/biomimetics4030061

Dantulwar, N. B., Maske, R. G., & Patel, J. T. (2017). Finite Element Analysis of Ball Valve Assembly for Earthquakes. International Conference on Ideas, Impact and Innovation in Mechanical Engineering, 5(6), 1460–1467.

Dobrovolsky, K. Z. (1973). Machine Elements : A Textbook. Moscow: Peace.

Doustdar, Mehdi, M., & Kazemi, H. (2019). Effects of Fixed and Dynamic Mesh Methods on Simulation of Stepped Planning Craft. Journal of Ocean Engineering and Science, 4(1), 33-48. doi:https://doi.org/10.1016/j.joes.2018.12.005

Elishakoff, I. (2004). Safety Factors and Reliability: Friends or Foes. 1st ed. Springer Netherlands. doi:https://doi.org/10.1007/978-1-4020-2131-2

García Pérez, M., & Vakkilainen, E. (2019). A Comparison of Turbulence Models and Two and Three Dimensional Meshes for Unsteady CFD Ash Deposition Tools. Fuel 237 (September 2018), 806-811. doi:https://doi.org/10.1016/j.fuel.2018.10.066

Gheorghe, G. V., Persu, C., Gageanu, I., & Cujbescu, D. (n.d.). Structural and Modal Analysis in Solidworks of Basic Structure of Equipment to Prepare Germinative Bed in Strips. Engineering for Rural Development 17 (May), 818–826. doi:https://doi.org/10.22616/ERDev2018.17.N064

Hertzberg, R. W., Vinci, R. P., & Hertzberg, J. L. (2012). Deformation and Fracture Mechanics of Engineering Materials (5th ed.). Wiley. Retrieved from https://en.id1lib.org/book/3364788/eaccde

Hutton, D. V. (2003). Fundamentals of Finite Element Analysis (1st ed.). McGraw-Hill Science/Engineering/Math. Retrieved from https://en.id1lib.org/book/445247/341616

Juvinall, R. C. (1967). Engineering Considerations of Stress, Strain, and Strength (1st ed.).

United States: McGraw-Hill College.

Juvinall, R. C., & Marshek, K. M. (2011). Fundamentals of Machine Component Design (5th ed.). (5th, Ed.) United States: Jhon Willey & Sons inc.

Jweeg, M. J., Hamzah, H. A., Al-Waily, M., & Al-Shammari, M. A. (2021). A Finite Element Simulation of Nano Effects on Stress Distribution in a Below Knee Prosthetic. IOP Conference Series: Materials Science and Engineering, 1067(1), 012141. doi:https://doi.org/10.1088/1757-899x/1067/1/012141

Karmankar, R. G. (2017). Analysis of Von- Mises-Stress for Interference Fit and Pull- Out States By Using Finite Element Method. International Research Journal of Engineering and Technology, 4(11), 1367-1374. doi:https://doi.org/10.13140/RG.2.2.26447.79520

Kelly, J. (2015). Stainless Steels, In: Mechanical Engineers' Handbook – Materials and Mechanical Design (Vol. 7). (M. Kutz, Ed.) United States of America, United States of America: John Wiley & Sons: John Wiley & Sons. doi:https://doi.org/https://doi.org/10.1002/0471777447.ch2

Kešner, A., Chotěborský, R., Linda, M., Hromasová, M., Katinas, E., & Sutanto, H. (2021). Stress Distribution on a Soil Tillage Machine Frame Segment with a Chisel Shank Simulated Using Discrete Element and Finite Element Methods and Validate by Experiment. Biosystems Engineering, 209, 125–138. doi:https://doi.org/10.1016/j.biosystemseng.2021.06.012

Kubasad, P. R. (2018). Numerical Analysis for a Bicycle Frame Made of Mild Steel and Composite. International Journal for Research Trends and Innovation, 3(4), 40–46. Retrieved from https://ijrti.org/papers/IJRTI1804008.pdf

Lai, Q., Yu, Q., & Dong, J. (2019). Dynamic Analysis of Rotary Tiller Gearbox Based on EDEM, ADAMS and ANSYS. Journal of Intelligent and Fuzzy Systems, 36(2), 1153–1160. doi:https://doi.org/10.3233/JIFS-169889

Lu, S., Jin, H., He, M., & Xu, Z. (2019). Analysis System of Power Tiller's General Machine Components Based on VB and ANSYS. Journal of Physics: Conference Series, 1237(4), 1-6. doi:https://doi.org/10.1088/1742-6596/1237/4/042054

Nejadian, H. A., Karparvarfard, S. H., Boldaji, M. N., & Koushkaki, H. R. (2019). Combined Finite Element and Statistical Models for Predicting Force Components on a Cylindrical Mouldboard Plough. Biosystems Engineering, 186, 168-181. doi:https://doi.org/10.1016/j.biosystemseng.2019.07.007

Oktaviandri, K & Paramasivam, D. K. A. V., (2020). Design and Fabrication of Customized Ais Kacang Vending Machine. Indonesian Journal of Computing, Engineering, and Design, 2(1). 24-31. doi:https://doi.org/10/35806/ijocedv2i1.100

Popa, M., Ștefan, I. S., Nicoleta, B., & Ipate, G. (2021). FEM Model for the Study of Agricultural Soil Compaction under the Action of Two-Wheel Tractor. INTERNATIONAL SCIENTIFIC JOURNAL "MECHANIZATION IN AGRICULTURE & CONSERVING OF THE RESOURCES, 40(1), 38-40.

Pranoto, S. H., & Mahardika, M. (2018). Design and Finite Element Analysis of Micro Punch CNC Machine Modeling for Medical Devices. AIP Conference Proceedings 1941(March).

doi:https://doi.org/10.1063/1.5028079

Prasetiyo, A. B., Azmi, A. A., Pamuji, D. S., & Yaqin, R. (2019). Pengaruh Perbedaan Mesh Terstruktur Dan Mesh Tidak Terstruktur Pada Simulasi Sistem Pendinginan Mold Injeksi Produk Plastik. Prosiding Nasional Rekayasa Teknologi Industri Dan Informasi XIV Tahun 2019 (ReTII) 2019 (November), 400-406.

Prasetiyo, A. B., Sekarjati, K. A., Pranoto, S. H., Sutrisna. (2022) Design and Analysis of The Effect of Variation of Compression Force on Allen Key Using Finite Element Analysis Method. SJME Kinematika, 7(1). 39-52. doi: https://doi.org/10.20527/sjmekinematika.v7i.

Pratama, J., & Mahardika, M. (2018). Finite Element Analysis to Determine the Stress Distribution, Displacement and Safety Factor on a Microplate for the Fractured Jaw Case. AIP Conference Proceedings 1941, 1–7. doi:https://doi.org/10.1063/1.5028080

Soden, P. D., Adeyefa, B. A., Wong, Y. S., & Millar, M. A. (1986). Loads, Stresses, and Deflections in Bicycle Frames. The Journal of Strain Analysis for Engineering Design, 21(4), 185–195. doi:https://doi.org/10.1243/03093247V214185

Sosnowski, M., Krzywanski, J., & Scurek, R. (2019). A Fuzzy Logic Approach for the Reduction of Mesh-Induced Error in CFD Analysis: A Case Study of an Impinging Jet. Entropy, 21(11). doi:https://doi.org/10.3390/e21111047

Sosnowski, M., Krzywanski, J., Grabowska, K., & Gnatowska, R. (2018). Polyhedral Meshing in Numerical Analysis of Conjugate Heat Transfer. EPJ Web of Conferences 180 (March 2019), 02096. doi:https://doi.org/10.1051/epjconf/201818002096

Sosnowski, M., (2018). The Influence of Computational Domain Discretization on CFD Results Concerning Aerodynamics of Vehicle. Journal of Applied Mathematics and Computational Mechanics, 17(1), 79-88, doi:10.17512/jamcm2018.1.08.

Stolarski, T., Nakasone, T., & Yoshimoto, S. (2006). Engineering Analysis With Ansys Software. Nuevos Sistemas de Comunicación e Información.

Suprapto, R. K., & Wibawa, L. A. (2021). Desain dan Analisis Tegangan Rangka Alat Simulasi Pergerakan Kendali Terbang Menggunakan Metode Elemen Hingga. Jurnal Teknik Mesin ITI, 5(1), 19-28. doi:10.31543/jtm.v5i1.559

Upendar, K., Dash, R., Behera, D., & Goel, A. (2018). Ergonomical Evaluation of Power Weeder in Wetland Paddy Condition. International Journal of Current Microbiology and Applied Sciences, 7(11), 855–862. doi:https://doi.org/10.20546/ijcmas.2018.711.101

Vegad, G. M., & Yadav, R. (2018). Design Analysis and Optimization of Rotary Tiller Blades Using Computer Software. AMA, Agricultural Mechanization in Asia, Africa and Latin America, 49(1), 43–49. Retrieved from https://www.cabdirect.org/cabdirect/abstract/20183168236

Wang, X., Shi, Q., Fan, W., Wang, R., & Wang, L. (2019). Comparison of the Reliability-Based and Safety Factor Methods for Structural Design. Applied Mathematical Modelling, 72, 68–84. doi:https://doi.org/10.1016/j.apm.2019.03.018

Wibawa, L. A., Diharjo, K., Raharjo, W. W., & Jihad, B. H. (2020). Stress Analysis of Thick-

Walled Cylinder for Rocket Motor Case under Internal Pressure. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, 70(2), 106–115. doi:https://doi.org/10.37934/ARFMTS.70.2.106115

Yin, Z., & Xu, L. (2018). Finite Element Analysis and Optimization Design of Paper Cutter Cutting Blade Based on ANSYS. Proceedings - 2018 International Conference on Robots and Intelligent System, 475–478. doi:https://doi.org/10.1109/ICRIS.2018.00125

Finite Element Simulation of Power Weeder Machine Frame

ORIGINALITY REPORT

14%	14%	0%	0%
SIMILARITY INDEX	INTERNET SOURCES	PUBLICATIONS	STUDENT PAPERS

PRIMARY SOURCES

1	publikasi.mercubuana.ac.id Internet Source	14%

Exclude quotes	Off	Exclude matches	< 14%
Exclude bibliography	Off		