Finite Element Simulation of Power Weeder Machine Frame

Angger Bagus Prasetiyo, Kartinas...

Submission Review Copyediting

Production

Round 1

Round 1 Status

Submission accepted.

Notifications

[IJoCED] Editor Decision	2022-08-22 08:46 AM
[IJoCED] Editor Decision	2022-09-13 05:33 PM

Reviewer's Attachments

Search

930-1 Reviewer, 291-99Z_Article Text-929-1-4-20220802 (1) Indonesia.docx

Revisions

Search Upload File

961-2	Mohon izin salah unggah (2)	Article Text
962-1	Mohon izin salah unggah	Article Text
963-1	Mohon izin salah unggah	Article Text

964-1 Author, 291-930-1-5-20220804 Revisi Baru.docx Article Text

Review Discussions

Add discussion

Name	From	Last Reply	Replies	Closed

No Items

IJoCED

Indonesian Journal of Computing, Engineering, and Design

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

Design ~~dan~~ and Analysis of the Power Weeder Machine Frame Using Solidwork 2022

ABSTRACT

One of the agricultural technologies that have evolved recently is the weeding machine. This device was ~~created~~devloped 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 will model the frame of a power weeder machine utilizing finite element analysis using the SolidWorks 2022 program. A total load of 120N is applied to the AISI 316L used for the frame. The analysis's findings indicate that the maximum von Mises stress value is 2.51e+7N/m^2, the greatest deformation is 0.556mm, and the safety factor is 6.8. The power weeder machine frame in this study is designed in a way that makes it safe to bear dynamic loads with AISI 316L material.

ARTICLE INFO

Article History:
Received dd mm yyyy
Revised dd mm yyyy
Accepted dd mm yyyy
Available online dd mm yyyy

Keywords:
Frame,
Power Weeder,
FEA,
Solidwork.

Commented [u1]: D

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 and the necessity to design and work to gain higher productivity, increase comfort, and increase safety (Upendar et al. 2018). 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. Before designing the weed weeding machine, many things 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 the inertia of forces acting on the entire frame if it is moving quickly and has no mass (Soden et al. 1986). The original axis deviated from the inertial force's reaction. Because of its flexibility, the power weeder machine's frame will revert to its initial position. If the frame exhibits strong strength and stiffness qualities, this will occur (Baihaki and Bintoro 2021).

The procedures that need to be followed before beginning the machine ~~manufacturing~~ development process, namely creating designs using ~~the advancement of design computer technology (~~ the computer Aided Design and Manufacturing (CAD ~~and~~/ CAM) tools, can reduce considerable expenses (Chirende, Li, and Vheremu 2019; Cekus et al. 2019). ~~resulting from design flaws.~~ By optimizing the design, one can reduce manufacturing errors while simultaneously lengthening the product's service life (Vegad and Yadav 2018).

Advanced businesses working in mechanics and other types of buildings have used ~~this~~ ?(Gheorghe et al. 2018). Software like SOLIDWORK 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 stress, von misses, displacement, and safety factors (Lu et al. 2019), optimize a design, and carry out theoretical calculations (Lai, Yu, and Dong 2019; Kešner et al. 2021), predict a material's strength, and verify the loading force with numerical simulations (Azimi-Nejadian et al. 2019), among other things (Yin and Xu 2018).

The goal of this study is to engineer the design of the power weeder machine frame model using? a finite element analysis approach using? SolidWorks 2022 software to assess the stress and strength of the frame. This is based on the description of the problem given above.?? *Do you have any free body diagram to show all the forcess on the frame?* Von Mises stress, displacement, and safety factor are the findings of this investigation.

2. RESEARCH METHODOLOGY

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

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

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

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

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

Finite Element Analysis

The software can now be used to address technical issues, and the results can

Commented [u4]: Avoid repititive words in a sentence.You may re-write it.

Commented [u2]: WHat do you mean?... this will occur?please make it clear.

Formatted: Font: Italic

Commented [u3]:

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, Solidwork 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, see accurate mass properties, and see 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).

Model and Materials

Generally speaking, SolidWorks 2022 software is used to assist in the design of the power weeder machine frame model. To make it simpler for farmers to operate, the frame design was designed under 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. According to (Kelly, 2017), AISI 316L was the material used in the construction of the power weeder machine frame for this investigation.

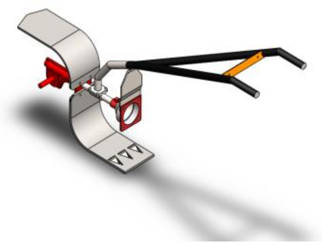


Figure 1. Rangka Power Weeder Machine

This material was chosen because it is heat- and corrosion-resistant. Table 1 provides an overview of the parameters used in the simulation.

Tabel 1. Parameter Simulation

Description	Value
Name Material	AISI Type 316L
Model type	Isotropic
Yield Strength	1,7e+08 N/m^2
Tensile Strength	4,85e+08 N/m^2
Elastic Modulus	2e+11 N/m^2

Poison's Ratio	0,265
Mass Density	8.027 kg/mˆ
Shear Modulus	8,2e+10 N/mˆ2
Thermal Expansion	1,65e-5 /Kelvin
Load Machine	80 N
Load Cultivator Tiller	40 N
Load	20 N

~~Pembagian Domain Komputasi~~ ? The language you use should be uniform

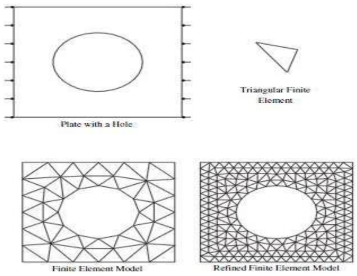
Domain division or meshing affects computational modelling utilizing the Finite Element Analysis (FEA) approach (Doustdar and Kazemi 2019). In a simulation, the partition of the computational domain (gird) is a procedure with high complexity (Sosnowski et al. 2018a). The division of the computational domain (mesh) results has a significant impact on the simulation's convergence results (Sosnowski, Krzywanski, and Scurek 2019; Prasetiyo et al. 2019). When the computational domain is divided incorrectly, the simulation will fail and you will have to restart the mesh building process, which is time-consuming. The results will be more accurate and the simulation process will run more slowly with a smaller mesh (García Pérez and Vakkilainen 2019; Prasetiyo et al. 2019).

Hexahedral mesh, polyhedral mesh, and tetrahedral mesh are some of the mesh types utilized in CFD simulation (Chen et al. 2021; Sosnowski et al. 2018b). The tetrahedral mesh was employed in this study. Tetrahedral mesh is employed because it is more effective for simulating stress distributions (Vutton D. V. 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.

Tabel 2. Information Mesh

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

Calculations utilizing the finite element approach must be performed using a computer since there are so many equations that require it. This will save money and time 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 will take the shape of a triangle, with each element being a linear quadrilateral connected by a node (node). In Figure 2, you can see more information.



make the Figure more clear

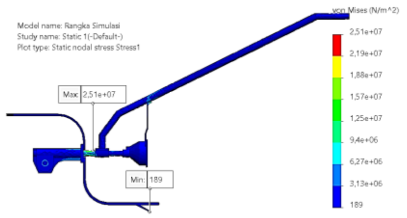
Figure 2. Mesh pada Metode Elemen Hingga????The language should be uniform.

Formatted: Font: Italic

3. RESULTS AND DISCUSSION

Von Mises Stress

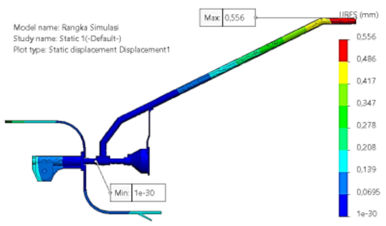
Because the material choice and frame design must be based on the maximum value of working stress, analysis of the stresses occurring on a structure is crucial. From the findings of the uniaxial tensile test, the von Mises stress is utilized to forecast the yield of materials subjected to complicated 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 Solidwork 2022.



Make the Figure more clear.

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 189N/m2, while the connection section between the tiller cultivator and the connecting pipe to the frame and engine has the highest value at 2,51e+7N/m2. 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



Make the Figure more clear.

Figure 4. Power Weeder Machine Frame Deformation Simulation Results

Safety Factor

yield strength, a material is said to begin yielding.

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 (Robert C. 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 (Robert C. 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 displays the results of the material's deformation value against the load. The weeding section has the smallest deformation value at 1e-30mm, while the frame holder has the largest deformation value at 0.556mm. This demonstrates that material is stronger the less it deforms (Richard W. Hertzberg, Richard P. Vinci 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.

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 whether or not the tool is designed to be operated, the safety factor is important. 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 displays the results of the modelling simulation. The frame design that has been created is extremely suited

Formatted: Font: Italic

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).

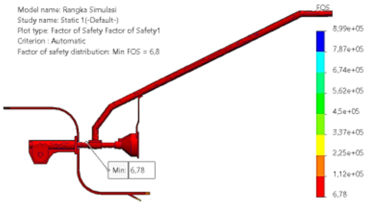


Figure 5. Safety Factor Simulation Results for Power Weeder Machine

4. CONCLUSION

~~Conclusion~~ The following conclusions are reached as a consequence of frame analysis performed using Solidwork 2022 software on a power weeder machine frame made of stainless steel 304 material:

1. The power weeder machine frame simulation yielded a maximum von mises stress of 2.51e+7n/m2.
2. In the simulation of the power weeder machine frame, the maximum deformation result is 0.556mm.
3. The power weeder machine frame's minimal safety factor simulation yielded a 6.8 with an AISI 316L material type result.
4. The power weeder machine's structure, which is made of AISI 316L material, may be deemed safe to bear dynamic loads.??

REFERENCES

Al-Shammari, Mohsin Abdullah, Qasim H. Bader, Muhannad Al-Waily, and A. M. Hasson. 2020. "Fatigue Behavior of Steel Beam Coated with Nanoparticles under High Temperature." *Journal of Mechanical Engineering Research and Developments* 43 (4): 287–98.

Al-Waily, Muhannad, and Zaman Abud Almalik Abud Ali. 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 (April): 26–34.

Ansys Release Documentation, Ansys. 2005. "ANSYS Workbench Release 10.0." *ANSYS Workbench Release 10.0*. http://kashanu.ac.ir/Files/Content/ANSYS Workbench.pdf.

Awwaluddin, Muhammad. 2019. "Analisa Kekuatan Rangka Sepeda Listrik Menggunakan Software SolidWorks." *Piston: Journal of Technical Engineering* 3 (1): 5–16. https://doi.org/http://dx.doi.org/10.32493/pjte.v3i1.7486.

Azimi-Nejadian, Hadi, Seyed Hossein Karparvarfard, Mojtaba Naderi-Boldaji, and Hossein Rahmanian-Koushkaki. 2019. "Combined Finite Element and Statistical Models for Predicting Force Components on a Cylindrical Mouldboard Plough." *Biosystems Engineering* 186: 168–81. https://doi.org/10.1016/j.biosystemseng.2019.07.007.

Baihaki, Muhammad Ilham, and Carolus Bintoro. 2021. "Aplikasi Desain Generatif Pada Rangka Sepeda Dengan Menggunakan Perangkat Lunak Solidworks Dan Matlab

Simulink." *Jurnal INOVTEK Seri Mesin* 2 (1): 2–5. http://ejournal.polbeng.ac.id/index.php/ISM/article/view/2329.

Cekus, Dawid, Renata Gnatowska, Paweł Kwiatoń, and Michal Šofer. 2019. "Simulation Research of a Wind Turbine Using SolidWorks Software." *Journal of Physics: Conference Series* 1398 (1). https://doi.org/10.1088/1742-6596/1398/1/012001.

Chen, Haofu, Xiaoqing Zhou, Zhuangbo Feng, and Shi Jie Cao. 2021. "Application of Polyhedral Meshing Strategy in Indoor Environment Simulation: Model Accuracy and Computing Time." *Indoor and Built Environment* 0 (0): 1–13. https://doi.org/10.1177/1420326X211027620.

Chiad, Jumaa S., Muhannad Al-Waily, and Mohsin Abdullah Al-Shammari. 2018. "Buckling Investigation of Isotropic Composite Plate Reinforced by Different Types of Powders." *International Journal of Mechanical Engineering and Technology* 9 (9): 305–17.

Chirende, Benard, Jian Qiao Li, and Wonder Vheremu. 2019. "Application of Finite Element Analysis in Modeling of Bionic Harrowing Discs." *Biomimetics* 4 (3): 1–11. https://doi.org/10.3390/biomimetics4030061.

Dantulwar, N B, R G Maske, and J T Patel. 2017. "Finite Element Analysis of Ball Valve Assembly for Earthquakes." *International Conference on Ideas, Impact and Innovation in Mechanical Engineering* 5 (6): 1460–67.

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

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

García Pérez, Manuel, and Esa Vakkilainen. 2019. "A Comparison of Turbulence Models and Two and Three Dimensional Meshes for Unsteady CFD Ash Deposition Tools." *Fuel* 237 (September 2018): 806–11. https://doi.org/10.1016/j.fuel.2018.10.066.

Gheorghe, Gabriel Valentin, Catalin Persu, Iuliana Gageanu, and Dan Cujbescu. 2018. "Structural and Modal Analysis in Solidworks of Basic Structure of Equipment to Prepare Germinative Bed in Strips." *Engineering for Rural Development* 17 (May): 818–26. https://doi.org/10.22616/ERDev2018.17.N064.

~~James Kelly. 2017. "The Rise of 3-D Printing: The Advantages of Additive Manufacturing over Traditional Manufacturing." *Business Horizons* 60 (5): 677–88. https://doi.org/10.1016/j.bushor.2017.05.011.~~

Jweeg, Muhsin J., H. A. Hamzah, Muhannad Al-Waily, and Mohsin Abdullah Al-Shammari. 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. https://doi.org/10.1088/1757-899x/1067/1/012141.

K. Z. V. Dobrovolsky. 1973. *Machine Elements : A Textbook*. Moscow: Peace.

Karmankar, Rahul 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): 1367–74. https://doi.org/10.13140/RG.2.2.26447.79520.

Kešner, Adam, Rostislav Chotěborský, Miloslav Linda, Monika Hromasová, Egidijus Katinas, and Hadi Sutanto. 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: 125–38. https://doi.org/10.1016/j.biosystemseng.2021.06.012.

Lai, Qinghui, Qingxu Yu, and Jiayu Dong. 2019. "Dynamic Analysis of Rotary Tiller Gearbox Based on EDEM, ADAMS and ANSYS." *Journal of Intelligent and Fuzzy Systems* 36 (2): 1153–60. https://doi.org/10.3233/JIFS-169889.

Lu, Shiqing, Hui Jin, Miao He, and Zhe Xu. 2019. "Analysis System of Power Tiller's General Machine Components Based on VB and ANSYS." *Journal of Physics: Conference Series* 1237 (4): 1–6. https://doi.org/10.1088/1742-6596/1237/4/042054.

Popa, Mădălina, Ivanciu Sorin- Ștefan, B Nicoleta, and George Ipate. 2021. "F EM Model for the Study of Agricultural Soil Compaction under the Action of Two-Wheel Tractor" 40 (1): 38–40.

Pranoto, Sigiet Haryo, and Muslim Mahardika. 2018. "Design and Finite Element Analysis of Micro Punch CNC Machine Modeling for Medical Devices." *AIP Conference Proceedings* 1941 (March). https://doi.org/10.1063/1.5028079.

Prasetiyo, Angger Bagus, Azhim Asyratul Azmi, Didit Setyo Pamuji, and Rizqi Yaqin. 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.

Pratama, Juan, and Muslim Mahardika. 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. https://doi.org/10.1063/1.5028080.

Richard W. Hertzberg, Richard P. Vinci, Jason L. Hertzberg. 2012. *Deformation and Fracture Mechanics of Engineering Materials*. 5th ed. Wiley. https://en.id1lib.org/book/3364788/eaccde.

Robert C. Juvinall. 1967. *Engineering Considerations of Stress, Strain, and Strength*. 1 th. Amerika Serikat: McGraw-Hill College.

Robert C. Juvinall, Kurt M. Marshek. 2011. *Fundamentals of Machine Component Design*. Edited by Wendy Ashenberg. 5 th. Amerika Serikat: Jhon Willey & Sons, Inc.

Soden, P. D., B. A. Adeyefa, Y. S. Wong, and M. A. Millar. 1986. "Loads, Stresses, and Deflections in Bicycle Frames." *The Journal of Strain Analysis for Engineering Design* 21 (4): 185–95. https://doi.org/10.1243/03093247V214185.

Sosnowski, Marcin, Jaroslaw Krzywanski, Karolina Grabowska, and Renata Gnatowska. 2018a. "Polyhedral Meshing in Numerical Analysis of Conjugate Heat Transfer." *EPJ Web of Conferences* 180 (March 2019): 02096. https://doi.org/10.1051/epjconf/201818002096.

———. 2018b. "Polyhedral Meshing in Numerical Analysis of Conjugate Heat Transfer." *EPJ*

Web of Conferences 180: 4–9. https://doi.org/10.1051/epjconf/201817002096.

Sosnowski, Marcin, Jaroslaw Krzywanski, and Radomir Scurek. 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). https://doi.org/10.3390/e21111047.

Stolarski, T.; Nakasone, T.; Yoshimoto, S. 2006. *Engineering Analysis With Ansys Software*. *Nuevos Sistemas de Comunicación e Información*.

Suprapto, Ready Kresna Nanda, and Lasinta Ari Nendra Wibawa. 2021. "Desain Dan Analisis Tegangan Rangka Alat Simulasi Pergerakan Kendali Terbang Menggunakan Metode Elemen Hingga." *Jurnal Teknik Mesin ITI* 5 (1): 19. https://doi.org/10.31543/jtm.v5i1.559.

Upendar, K., R.C. Dash, D. Behera, and A.K. Goel. 2018. "Ergonomical Evaluation of Power Weeder in Wetland Paddy Condition." *International Journal of Current Microbiology and Applied Sciences* 7 (11): 855–62. https://doi.org/10.20546/ijcmas.2018.711.101.

Vegad, G. M., and R. Yadav. 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. https://www.cabdirect.org/cabdirect/abstract/20183168236.

Vutton D. V. 2003. *Fundamentals of Finite Element Analysis*. Edited by David Hutton. *McGraw-Hill Science/Engineering/Math*. 1st ed. McGraw-Hill Science/Engineering/Math. https://en.id1lib.org/book/445247/341616.

Wang, Xiaojun, Qinghe Shi, Weichao Fan, Ruixing Wang, and Lei Wang. 2019. "Comparison of the Reliability-Based and Safety Factor Methods for Structural Design." *Applied Mathematical Modelling* 72: 68–84. https://doi.org/10.1016/j.apm.2019.03.018.

Wibawa, Lasinta Ari Nendra, Kuncoro Diharjo, Wijang Wisnu Raharjo, and Bagus H. Jihad. 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): 106–15. https://doi.org/10.37934/ARFMTS.70.2.106115.

Yin, Zhiqin, and Lianjiang Xu. 2018. "Finite Element Analysis and Optimization Design of Paper Cutter Cutting Blade Based on ANSYS." *Proceedings - 2018 International Conference on Robots and Intelligent System, ICRIS 2018*, 475–78. https://doi.org/10.1109/ICRIS.2018.00125.