Vol. 12, No. 1 (2023) : 174 - 185
DOI : http://dx.doi.org/10.23960/jtep-l.v12i1.174-185

JURNAL TEKNIK PERTANIAN LAMPUNG

ISSN 2302-559X (print) / 2549-0818 (online)
Journal homepage : https://jurnal.fp.unila.ac.id/index.php/JTP

Design and Simulation of Chassis for Electric Cultivator

Dwi Santoso[1]✉, Abdul Waris[2], Iqbal[2], Saat Egra[3]

[1]Faculty of Agriculture, Universitas Borneo Tarakan, Tarakan, INDONESIA
[2]Faculty of Agriculture, Universitas Hasanuddin, Makassar, INDONESIA
[3]Faculty of Agriculture, Gifu University, Gifu, JAPAN

Article History :

Received : 1 November 2022
Received in revised form : 21 January 2023
Accepted : 12 February 2023

Keywords :
Design,
Electric cultivator,
Simulation .

✉Corresponding Author:
dwisantoso@borneo.ac.id

ABSTRACT

Building or designing an electric cultivator requires a comprehensive study by paying attention to each main component, namely the chassis. The chassis serves as a place to attach the constituent components and holds the weight of the overall components contained in the tool. A good machine frame will increase the workability of the machine because the components that make up the cultivator are in the right layout. A good chassis design is needed to improve the performance of the electric cultivator. This study aims to design and simulate the strength analysis of the electric cultivator frame. This research consists of several stages, namely literature review, frame design, chassis strength simulation and chassis cultivator capability analysis. From the results of the analysis concluded that technically this tool is classified as safe with a loading condition of 18 kg and a support on its axis. However, it is still recommended that before production, the tool design must be re -optimized. Especially at the joints of the upper and lower frames near the pillow block. This is because this cross section is a critical area because it has a large equivalent stress value, has a small life cycle, and has a low safety factor.

1. INTRODUCTION

Farmers still use simple equipment, especially in the process of tillage. Cultivation is also included in the activities that require the most energy in plant cultivation activities (Santoso *et al.*, 2020). The process of tillage using simple equipment can cause the cultivation process to be slow and the quality of the soil to decline (Nawaz *et al.*, 2013). Soil tillage is carried out using simple tools that utilize human power, such as hoes, and equipment that uses animal power such as the plow, to equipment that uses machine or tractor power, such as rotary plows and harrows (Rakshith, 2017).

The process of plowing and other activities for land preparation by manual, can take quite a long time, and cannot plow land on a large scale (Grünbühel *et al.*, 2003). This is not effective and will only make crop production low. At this time technological advances in agriculture are increasingly rapid, the use of tools and machines in the production process aims to improve efficiency, effectiveness, productivity, quality of results, and to be able to ease the workload of farmers (Santoso & Murdianto, 2022). One of the concepts of agricultural modernization is the use of electricity-based power sources to drive agricultural machinery (Lal, 2008). The concept of developing electric-based agricultural machines has not been widely developed, this is because the plowing process requires a large amount of power and torque to suppress and tear the soil, this of course can be done on a combustion engine (Eli-Chukwu, 2019).

Along with the times, technological advances and agricultural machinery are increasingly expensive, making it increasingly difficult for farmers to get optimal harvest (Yahya, 2018). Therefore, this electric cultivator is a new breakthrough in increasing production output that is expected to help people in Indonesia, especially farmers. Building or designing an electric cultivator requires a comprehensive study by paying attention to every main component, namely the chassis. The chassis serves as a place for the attachment of the constituent components and holds the overall weight of the components contained in the tool. A good machine frame will increase the workability of the machine because the components that make up the cultivator are in the right layout (Riyaz, 2021). There needs to be a good chassis design to improve the performance of the electric cultivator. This study aims to design and simulate the strength analysis of the electric cultivator chassis

2. MATERIALS AND METHODS

The research was conducted out from March to September 2022. Located at the Mechanization Laboratory, Faculty of Agriculture, University of Borneo Tarakan. The material used in this research were hollow iron, round iron pipe, aluminum plate, ANSYS APDL software and Solid Work 3D software.

2.1. Research Procedure

The main activities in this research was as follows:

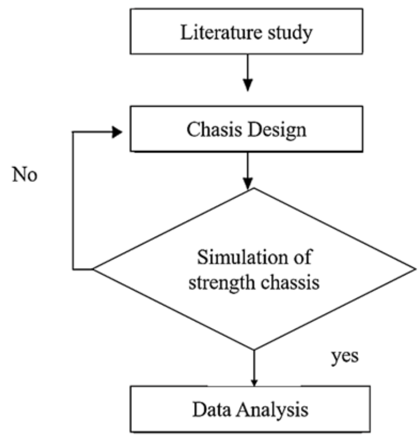

Figure 1. Flow chart of research

2.1.1. Chassis Design

A mini rotary cultivator that is planned to have a dynamo engine which will be transmitted via a gear box with a power of 5 hp with a torque of 60-75 Nm, engine dimensions of 25x34x45cm and a weight of about 18kg, this cultivator is expected to be used in rice fields and dry land so it requires a strong and sturdy frame so that the process of strengthening the soil remains stable even though it is used on land that is quite extreme. The type of frame chosen in this research is the type of trellis frame in the form of a series of welded turbular pipes one by one, this type of frame has not been widely used in tractors or cultivators. This type of frame is widely used on Italian motorcycle manufacturers such as the Ducati Multistrada motorcycle (Barbagallo *et al*., 2018). The more pipes that cross, the chassis will be sturdy and rigid (Gkioxari & Malik, 2015).

The chassis is designed using Solid Work 3D software, the frame in the design will use hollow steel as the engine seat frame, while for the handle bars use round iron pipes. Based on the background and existing research, the author is motivated to design a chassis for an electric cultivator with changes in dimensions using hollow iron material so that the mass of the frame is not large so that the total weight of the cultivator can be minimized.

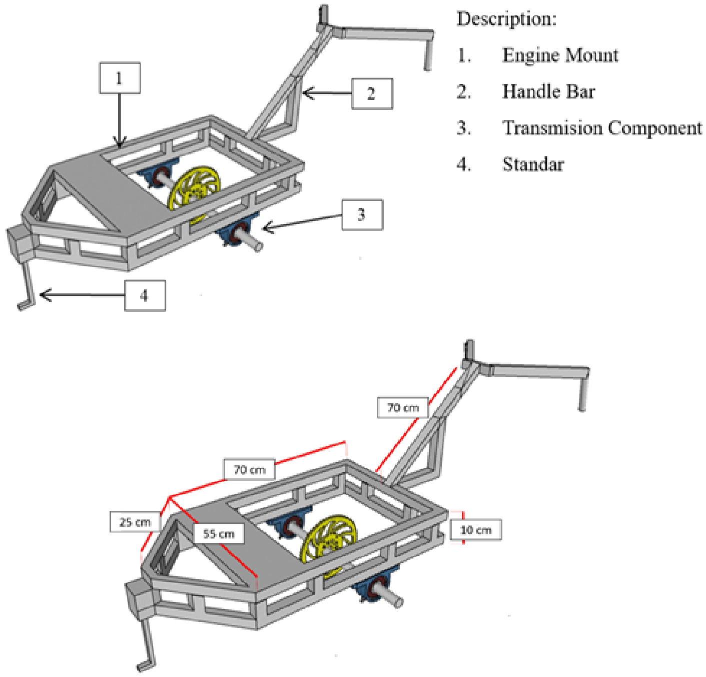

Figure 2. Chassis parts of cultivator (upper) along with important dimensions (bottom)

Appropriate technology that will be developed in this research is part of a series of ongoing research with the ultimate goal of creating food security and self-sufficiency in border areas, especially the province of North Kalimantan.

2.1.3.Chassis Strength Simulation and Data Analysis

The simulation steps using ANSYS APDL 15.0 (Figure 3) are as follows: (1) the preference process is a preliminary step to determine the analysis model for the existing material conditions, (2) determine the nature of the material element (Element type), which is to determine the nature of the test element to be used, (3) determining the Real Constant is to define the type of test element, (4) material properties is to define the type of material to be used with the aim of knowing the stress and strain properties of the material, (5) the modeling process is the process of making a model of the object to be used, (6) meshing is the process of dividing the model into small elements, (7) displacment is determining the location of the fulcrum, (8) force is the application of Load (load) with the application of force to be given, (9) solve is a step work to calculate all the variables that have been entered, (10) general Postproc serves to view the calculation results from simulations in various types shapes such as contours, paths, and lists.

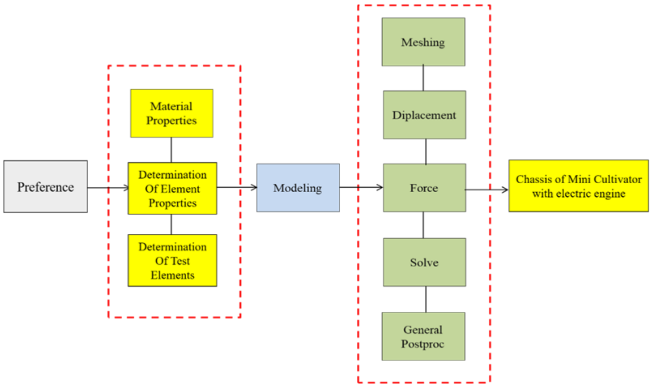

Figure 3. Simulation step with software Ansys

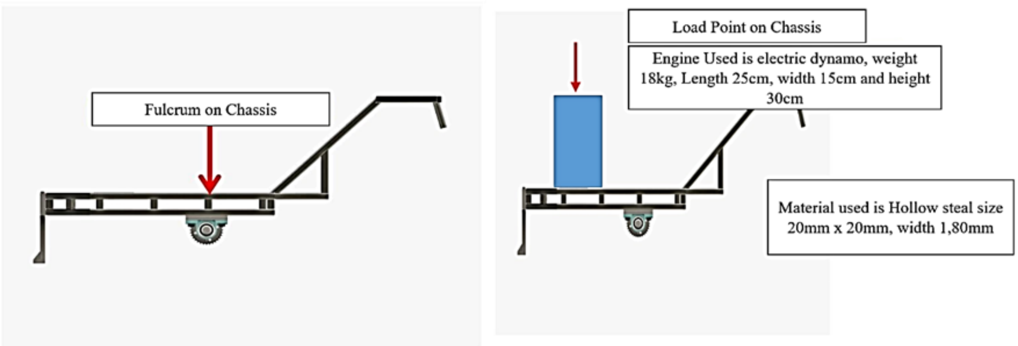

Figure 4. Input process of material type

The type of material used is described in figure 4. the fulcrum is parallel to the bearing. while the location of the engine is at the front of the chassis. The material used is SHS (square hollow section) which is widely available in the market with the ST37 or ASTM A36 type. The analysis module used is Static Structural to see the effect of static loading on the chassis and also Eigenvalue Buckling to see possible buckling conditions on the chassis.

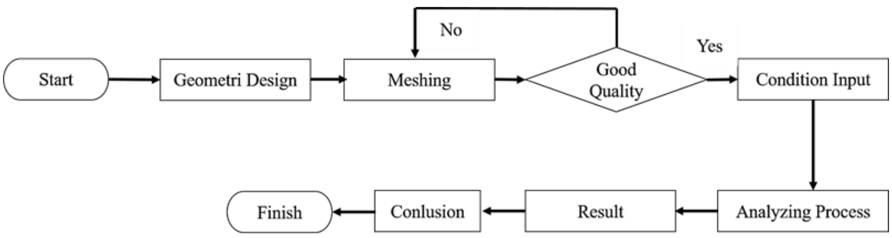

Figure 5. Flow chart of analysis Ansys

Flow chart of analysis Ansys described in figure 5. Simulations are carried out using Ansys software. Chassis strength testing uses the static test method, which is a technique that allows machine designers to determine defects in the components of the tool or machine that are designed without operating them on a real system (Bray & Stanley, 2018).

A. Design of Geometri Analysis (Cleaning Process)
The geometry cleaning process is a basic step for simulation using Ansys software. The purpose of this cleaning process is so that the simulation does not become heavy due to non-functional constituent components and unnecessary contours in the design that do not affect the analysis process. Figure 6 is the geometry for the analysis of the cleaning results based on the master design in Solid work software.

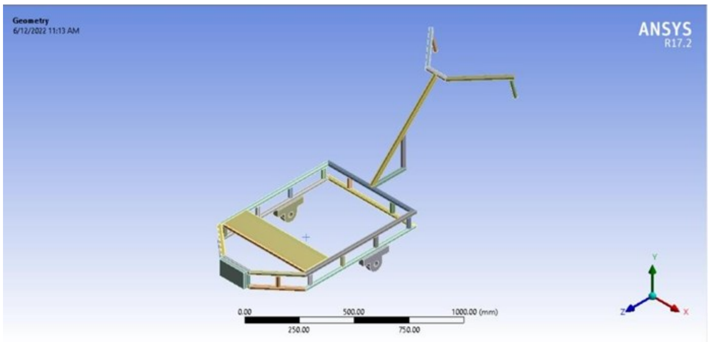

Figure 6. Cleaning process of geometry design

B. Input Condition
Figure 7 shows the position of the load and support, where the pedestal is on the bearing shaft B, the load is on the red point on the engine mount of 18 kg.

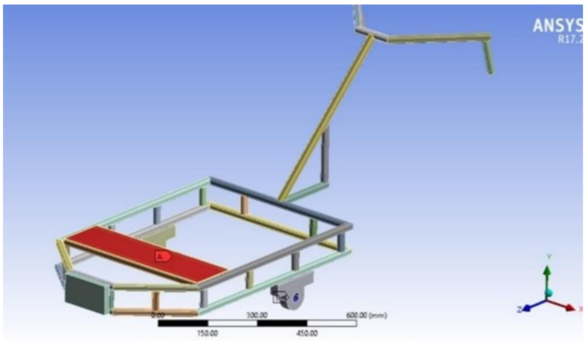

Figure 7. Input condition

3. RESULT AND DISCUSSION

After the design process with solid work software is complete, the results of the analysis and simulation can be known, namely the maximum and minimum values that can be seen directly on the autodesk inventor display. Simulations carried out using Ansys software obtained several test results, namely Equivalent Stress (Von Mises), Deformation, Fatigue, safety factor and buckling.

3.1. Equivalent Stress (Von Mises)
To determine the spread of stress that occurs in the material, a simulation is carried out by showing the value of Equivalent Stress (Von Mises) (Rakshith, 2017). Sugiyanto *et al*. (2022) and Wu *et al*. (2022) explained that Equivalent Stress is an indicator that

measures material failure by analyzing the resultant of 3 main stresses or commonly called Principal Stress. Failure is predicted if the Equivalent Stress value is greater than the yield stress of the material.

The simulation of the calculation of Equivalent Stress (Von Mises) is shown in Figure 8 that the stress point is in the center of the chassis, which is above the bearing holder. Von Mises effective stress is defined as a uniaxial tensile stress that can produce a distortion energy equal to that produced by a combination of applied stresses (Scott-Emuakpor *et al*., 2012). The main frame that is given an external load will result in stress on the frame. The critical section will occur at a point with a non-uniform geometric shape (Lohar *et al*., 2022). Stress in the frame will also occur at the point where the load will be transmitted to other components.

From the analysis results, the minimum equivalent stress value is 0.0008893 MPa and the maximum is 118.48 MPa. The critical area is indicated by a red circle where this area is located at the connection of the upper and lower frames near the pillow block (Figure 8). The critical area is an area that has a small lifetime compared to other parts. When the chassis is given a load for each loading position, the central point of the chassis will experience a bending load, where the bearing holder will be the support (Imran, 2017). Figure 9 shows that the maximum equivalent stress value of the tool is still smaller than the maximum yield stress of the material, which is 220 MPa. So it can be said that the framework is still in the safe category.

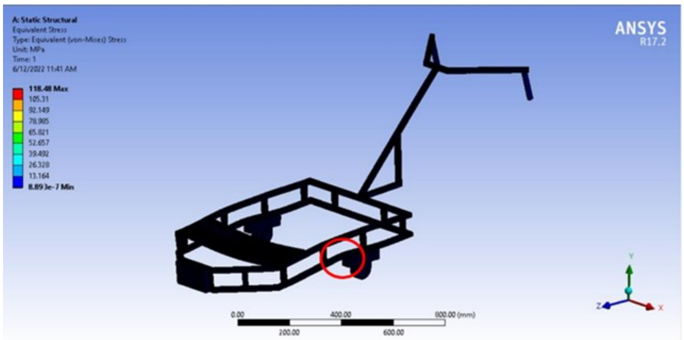

Figure 8. Simulation of equivalent stress (Von Mises) calculation

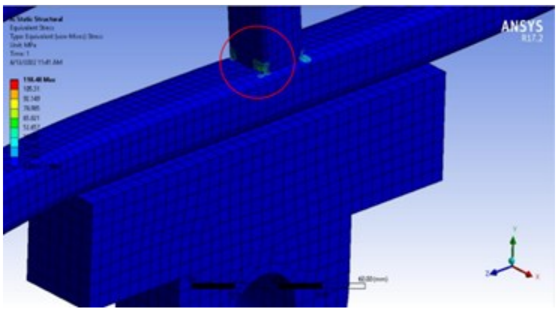

Figure 9. Chassis breaking point based on simulation results

Figure 10 explains that the deformation value is at a value of 0.2 with a maximum yield stress of 220 MPa. This shows that for all loading positions when the chassis is loaded, the chassis structure will only experience elastic deformation where the shape of the chassis will return to its original shape when the load is released.

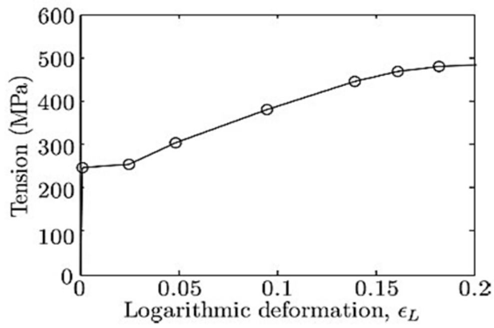

Figure 10. Stress strain relatiomship of ASTM A36 Steel

A. Meshing

Figure 11 is the value input process for the meshing test. The results of meshing in structural analysis are very important, where the better the results of the mesh from the geometry, the more valid the output of the analysis (Cottrell et al., 2007). One way to measure the mesh value is to use the skewness method where the smaller the skewness value, the better the mesh results. Figure 12 shows the results of the mesh of the cultivator frame generally having a value of 0.5 which is in GOOD condition. With the number of nodes 386514 and the number of elements 69043.

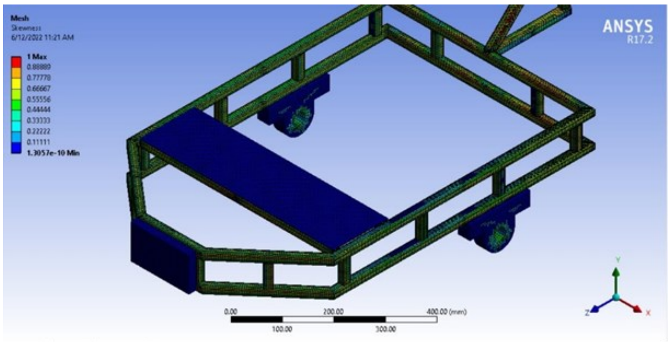

Figure 11. Input Process of meshing value

Skewness mesh metrics spectrum:

Excellent	Very good	Good	Acceptable	Bad	Unacceptable
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

Orthogonal Quality mesh metrics spectrum:

Unacceptable	Bad	Acceptable	Good	Very good	Excellent
0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00

Figure 12. Meshing spectrum

B. Deformation

Deformation is the process of changing the shape or distortion of a component that occurs due to a load (force) or pressure (pressure). Deformation is one indicator to determine the strength of the material (Wibawa, 2020). The stronger the material, the smaller the deformation value resulting from the loading process. The weaker a material is, the greater the deformation value resulting from the loading process. From the results of the analysis, it is shown that the minimum total deformation value is 0 mm and the maximum total deformation value is 0.20834 mm, the area that has the red contour is the area that has the maximum deformation value. However, this maximum deformation value is still safe when viewed from the deformation curve in Figure 13 because the value of 0.20834 mm is still in the elastic region, so that if the load = 0 the frame can return to its original position or the deformation is 0.

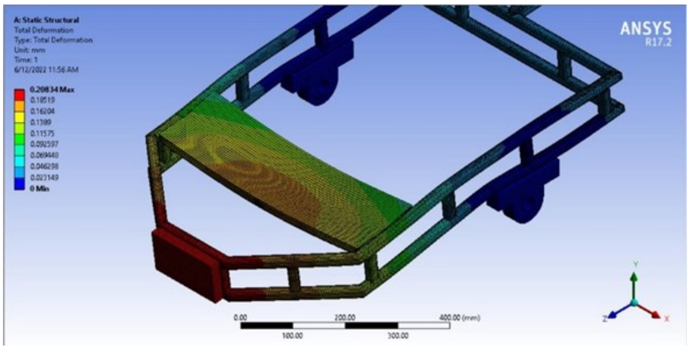

Figure 13. Deformation Test

C. Fatigue

Fatigue is damage to the structure (especially welded joints) that occurs due to cyclical loads from the environment that work continuously during the operation of the structure. Structural damage can be in the form of initial cracks (crack initiation/ fatigue), followed by crack propagation (crack growth) resulting in fractures in the structure (Arsić *et al*., 2019). Structural conditions due to environmental influences such as corrosion and extreme work operations will increase the susceptibility to structural damage due to fatigue (Phady *et al*., 2020). Fatigue analysis (figure 14) was carried out using the Gerber mean stress correction theory method. The following results were obtained:

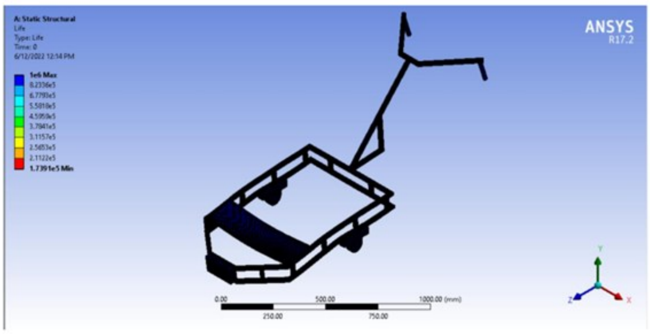

Figure 14. Fatigue test

With a frame fatigue test using a load of 18 kg, it has a maximum yield of 1000000 times. Where the minimum result is 173910, the area where the first failure occurs is shown in Figure 13. The chassis failure point against the load is influenced by several factors including external load, chassis material and welding strength.

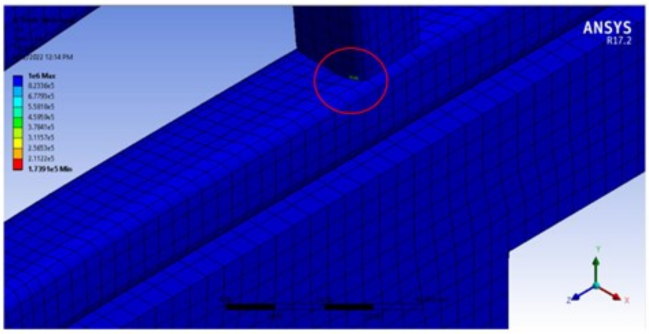

Figure 15. Chassis failure point

Baker (2011) and (Arsić et al., 2019) explained that fatigue strength analysis is applied to all machine structures that are subjected to high external loads and predominantly to cyclical loads, to ensure structural integrity and to assess possible fatigue damage as a basis for efficient inspection methods. Wave loads are a source of fatigue cracking (Rizky *et al*., 2016). However, other cyclical loads also affect fatigue failure and must be taken into account. Chassis stresses against loads often occur in welding parts (James *et al*., 2007).

D. Safety Factor

The safety factor is a factor that is analyzed to evaluate the safety standards of a structure in the designed system, where the strength of a material must exceed the actual strength (Osgood, 2013). The greater the value of the safety factor, the better and stronger the construction is made to accept loads outside of the calculation. The safety factor is also an indicator of the strength of the chassis (Spanaki *et al*., 2021). The factor of safety is used to evaluate the safety of a component or structure even though the minimum dimensions are used. The minimum factor of safety using the Ansys Workbench software is calculated as the yield strength of the material divided by the maximum von Mises stress (Wibawa, 2020). A safety factor value of less than 1 (one) indicates a permanent failure of a design.

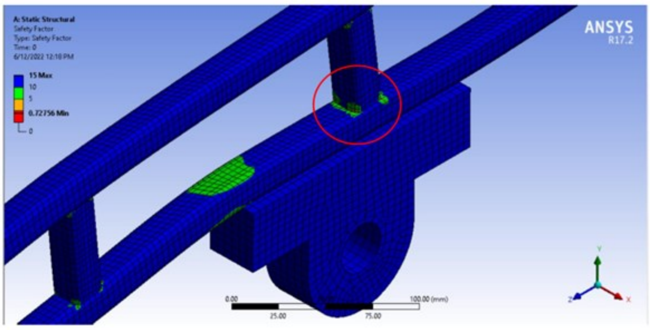

Figure 16. Result of safety factor analysis

From the results of the analysis it can be seen that in general the safety factor has a very good value of 17 as shown in the figure 16. There are several areas that have very poor SF values, namely the area circled in red. In that area SF is worth 0.72756 on the X axis. In the structure, the range of safety factors that must be owned is 1.0 to 10. The higher the safety factor value, the higher the level of safety possessed by the structure when receiving loads, the better (Singh *et al*., 2022). If the safety factor is less than 1.0 then a structure can undergo plastic deformation which will have an impact on the inability of a structure to maintain its function in accordance with the initial design, as a result or the condition of the structure will experience stress or fracture when given an excessive load (Dhir, 2018) and (Lohar *et al*., 2022) explained that the value of this safety factor is important to maintain when the system receives a load outside of the calculation, the goal is that the design that has been made does not fail when experiencing these conditions.

E. Buckling

A structure that has small or thin dimensions will experience buckling or buckling instability problems when stressed. Buckling is a process where a structure is unable to maintain its original shape, in such a way that it changes shape in order to find a new balance (Huang *et al*., 2020). The consequence of buckling is basically a geometric problem, where there is a large convex shape that will change the shape of the structure.

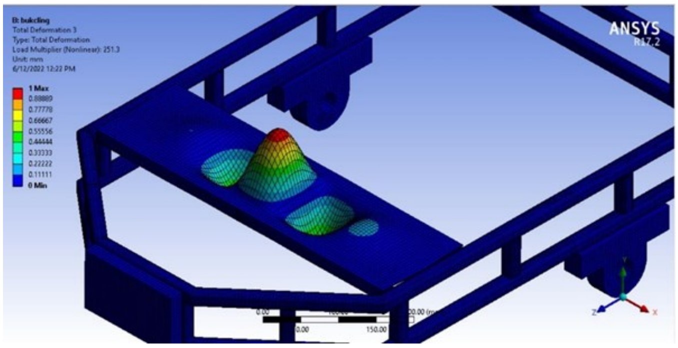

Figure 17. Simulation result of buckling test

From the results of the analysis using the Ansys software described in figure 17, it is shown that the plate for mounting the motor has the potential for buckling with a maximum value of 1mm. The maximum value is still in the safe category because the plate used uses an iron plate with a thickness of 20mm. The simulation of the buckling test on the cultivator chassis is carried out by pressing the mounting plate of the cultivator machine. The simulation is assumed to use a load on a deformable or flexible frame with a rigid object, with a deformable material the frame can be investigated for deformation and with a rigid material it will not deform when impact testing is carried out (Kumar *et al*., 2019). The amount of deformation that occurs in the frame from the simulation results when it is influenced by the speed and mass and kinetic energy that occurs is the basis of the compressive test analysis carried out (Wu *et al*., 2022).

4. CONCLUSION

From the results of the analysis concluded that technically the tool is classified as safe with a loading condition of 18 kg and a support on its axis. However, it is still advisable, before production, the tool design must be optimized again. Especially at the connection of the upper and lower frames near the pillow block. This is because this section is a critical area because it has a large equivalent stress value, has a small life cycle, and has a low safety factor.

REFERENCES

Arsić, D., Gnjatović, N., Sedmak, S., Arsić, A., & Uhričik, M. (2019). Integrity assessment and determination of residual fatigue life of vital parts of bucket-wheel excavator operating under dynamic loads. *Engineering Failure Analysis*, ***105***, 182–195.

Baker, A. A. (2011). A proposed approach for certification of bonded composite repairs to flight-critical airframe structure. *Applied Composite Materials*, ***18***(4), 337–369.

Barbagallo, R., Sequenzia, G., Cammarata, A., Oliveri, S. M., & Fatuzzo, G. (2018). Redesign and multibody simulation of a motorcycle rear suspension with eccentric mechanism. *International Journal On Interactive Design And Manufacturing (Ijidem)*, ***12***(2), 517–524.

Bray, D.E., & Stanley, R.K. (2018). *Nondestructive Evaluation: A Tool in Design, Manufacturing, and Service*. CRC Press.

Cottrell, J.A., Hughes, T.J.R., & Reali, A. (2007). Studies of refinement and continuity in isogeometric structural analysis. *Computer Methods In Applied Mechanics And Engineering*, ***196***(41–44), 4160–4183.

Dhir, D. K. (2018). Thermo-mechanical performance of automotive disc brakes. *Materials Today: Proceedings*, *5*(1), 1864–1871.

Eli-Chukwu, N.C. (2019). Applications of artificial intelligence in agriculture: A review. *Engineering, Technology & Applied Science Research*, ***9***(4), 4377–4383. https://doi.org/10.48084/Etasr.2756

Gkioxari, G., & Malik, J. (2015). Finding action tubes. *Proceedings of The IEEE Conference on Computer Vision and Pattern Recognition*, 759–768.

Grünbühel, C.M., Haberl, H., Schandl, H., & Winiwarter, V. (2003). Socioeconomic metabolism and colonization of natural processes in sangsaeng village: Material and energy flows, land use, and cultural change in Northeast Thailand. *Human Ecology*, ***31***(1), 53–86.

Huang, W., Xie, L., Li, C., Jia, D., Chai, B., Mu, Y., Hou, C., & Dai, W. (2020). Ananalytical and experimental study of t-shaped composite stiffened panels: effect of 90 plies in stringers on curing and buckling performance. *Applied Composite Materials*, ***27*** (5), 597–618.

Imran, A.I., & Kadir, K. (2017). Simulasi tegangan von mises dan analisa safety factor gantry crane kapasitas 3 ton. Dinamika: Jurnal Ilmiah Teknik Mesin, ***8***(2), 1-4.

James, M.N., Hughes, D.J., Chen, Z., Lombard, H., Hattingh, D.G., Asquith, D., Yates, J.R., & Webster, P.J. (2007). Residual stresses and fatigue performance. *Engineering Failure Analysis*, ***14***(2), 384–395.

Kumar, S.D, Karthik, D., Parida, S.K., & Jha, S.K. (2019). Optimization of semi-solid-forging parameters of a356–5tib 2 in situ composites using ANSYS and DEFORM Simulation. In *Innovations in Soft Computing and Information Technology*. Springer, 279–286.

Lal, R. (2008). Soils and sustainable agriculture. A review. *Agronomy for Sustainable Development*, ***28***(1), 57–64.

Lohar, S., Patil, V., Save, S., & Thakur, R. (2022). Design and analysis of independent suspension system of a FSAE Vehicle. In *Recent Advances in Manufacturing Modelling and Optimization*. Springer, 439–452.

Nawaz, M. F., Bourrie, G., & Trolard, F. (2013). Soil compaction impact and modelling. A review. *Agronomy for Sustainable Development*, ***33***(2), 291–309.

Osgood, C.C. (2013). *Fatigue Design: International Series on The Strength and Fracture of Materials and Structures*. Elsevier.

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. *SENSISTEK: Riset Sains dan Teknologi Kelautan*, 13–18.

Rakshith, N. (2017). *Design & Fabrication of Mini Tiller*. ATMECE.

Riyaz, S. (2021). Multipurpose Agriculture Cultivator. *Scienceopen Preprints*.

Rizky, A. P., Mulyatno, I. P., & Jokosisworo, S. (2016). Analisa fatigue kontruksi main deck sebagai penumpu towing hook akibat beban tarik pada kapal tug boat 2 x 800 hp dengan metode elemen hingga. *Jurnal Teknik Perkapalan*, *4*(1).

Santoso, D., & Murdianto, D. (2022). Artificial intelligence in the perspectives of agricultural technology development in Indonesia. *Budapest International Research and Critics Institute (BIRCI-Journal): Humanities and Social Sciences*, ***5***(1), 4348–4354.

Santoso, D., Rahajeng, G.Y., & Wijaya, R. (2020). Identifikasi kebutuhan alsintan tanaman pangan (padi dan jagung) di Kota Tarakan. *Jurnal Ilmiah Inovasi*, *20*(3).

Scott-Emuakpor, O., George, T., Cross, C., Wertz, J., & Shen, M.-H.H. (2012). A new distortion energy-based equivalent stress for multiaxial fatigue life prediction. *International Journal of Non-Linear Mechanics*, ***47***(3), 29–37.

Singh, S. K., Kumar, D., Jha, G. K., & Nain, P. K. S. (2022). Design and analysis of overhead ambulance. In *Advances in Mechanical Engineering and Technology*. Springer, 531–543.

Spanaki, K., Sivarajah, U., Fakhimi, M., Despoudi, S., & Irani, Z. (2021). Disruptive technologies in agricultural operations: A systematic review of ai-driven agritech research. *Annals Of Operations Research*, 1–34.

Sugiyanto, D., Pangestu, R.A., Chan, Y., & Uyun, A.S. (2022). Design and performance testing of semi-automatic machine for potato peeler-cutter. *Jurnal Teknik Pertanian Lampung*, ***11***(2), 325–338.

Wibawa, L.A.N. (2020). Simulasi umur fatik rangka main landing gear menggunakan metode elemen hingga. *Dinamika dan Teknik Mesin: Jurnal Keilmuan dan Terapan Teknik Mesin*, ***10***(2), 120–126.

Wu, X., Chen, Q., Zhao, B., Zhang, K., & Wang, P. (2022). Safety assessment of aircraft panel under the impact load by tire fragment based on thermal–mechanical effect. *Journal of Materials Engineering and Performance*, ***32***, 1119–1132.

Yahya, N. (2018). Agricultural 4.0: Its implementation toward future sustainability. In *Green Urea*. Springer, 125–145.