Journal of Engineering and Applied Technology
Vol. 5, No. 2, August 2024, pp. 72-84

Online ISSN 2716-2265
Printed ISSN 2716-2257

Design analysis of the baseplate of a stack module using computer-aided engineering

Muhammad Iqbal[1], Etik Puspitasari[2,*]

[1,2] Departement of Mechanical Engineering, State Polytechnic of Malang, Indonesia
E-mail: etik.puspitasari@polinema.ac.id *
* Corresponding Author

ABSTRACT

A PWM rectifier is a power converter that functions to convert alternating current into direct current using pulse width modulation (PWM). The baseplate is an important part of the PWM rectifier box, which acts as a stack module holder and has an important role in maintaining the stack module and increasing the power conversion efficiency and reliability of the train traction system. This study aims to determine the effect of variations in material and plate thickness used on Von Mises stress, deformation, and safety factors as well as the mechanical properties of the designed baseplate. The research method used is finite element analysis (FEA) by doing 3D modelling using CAD software and a simulation process using computer-aided engineering (CAE) software with material and plate thickness variations. The results obtained the best baseplate based on the value of Von Mises stress and minimum deformation, as well as the maximum safety factor, namely the baseplate with SS400 material with a plate thickness of 5 mm, with a minimum Von Mises stress value of 76.374 MPa, a minimum deformation value of 0.13611 mm, and a maximum safety factor value of 3.1424. It indicates the effect of material and plate thickness on the value of Von Mises stress, deformation and the resulting safety factor, thus affecting the mechanical properties of a good and safe baseplate in maintaining the performance of the stack module.

ARTICLE INFO

Article history
Received:
26 July 2024
Revised:
19 August 2024
Accepted:
20 August 2024

Keywords
Baseplate
CAE
FEA
Rectifier
Safety factor
stack module

1. Introduction

Hybrid trains are used to transport passengers and goods on certain rail lines. Hybrid trains use a diesel engine as the main power source and an electric drive system as an additional power source. Diesel-electric rail trains are commonly used in various countries as part of the rail transportation system. These rail trains are essential to meet the needs of logistics and mass transportation [1].

In this modern transportation system, hybrid trains have become one of the most efficient and environmentally friendly solutions. Hybrid trains combine the power of a diesel engine with an electric motor to generate the necessary power to move the most trains [1]. In the hybrid train system, there is a stack module component used in the rectifier. The stack module functions as a converter of alternating current (AC) into direct current (DC), responsible for controlling the output voltage and frequency produced to suit the needs of the train traction system [2]. The stack module component certainly requires a place or bracket on the pulse width modulation (PWM) rectifier box. Based on these needs, it is necessary to have an additional component in the form of a baseplate as a place for the stack module, which has an important role in maintaining the performance and reliability of the rectifier system. The design with good mechanical properties on the baseplate is expected to prevent the hot temperature from rising on the stack module components and can withstand the load of the stack module [3].

The use of computer-aided design (CAD) and computer-aided engineering (CAE) software in 3D modelling and simulation of design results is very helpful in the 3D modelling process and provides an efficient approach and accuracy in the simulation and analysis process of the baseplate design. By using this software, 3D modelling and simulation under various operating conditions can be performed to accurately determine the Von Mises stress and loading-induced deformation values, as well as the safety factor value, which determines the baseplate design's safety value [4].

The stack module component used today has a smaller size compared to the stack module in the previous PWM rectifier box, so it is necessary to adjust the PWM rectifier box design with the addition of a baseplate component that functions as a stack module holder. Based on these problems, this research was conducted to analyze the selection of a new stack module baseplate structure with variations in material and plate thickness used against the capabilities of the baseplate, focusing on the stress and deformation of the structure that occurs due to loading and the value of the safety factor produced by the baseplate design. By simulating and analyzing the existing design, it is expected to know the effect of material variations and plate thickness, so that it can be taken into consideration in determining the material and thickness of a good baseplate that will be used in the PWM rectifier box production.

Tan Ai, et al. conducted research that focused on the design and optimization of an adaptive robotic gripper. The research was motivated by the need to overcome the limitations of traditional robotic grippers, which often struggle to effectively handle objects of different shapes and sizes. This research aimed to develop a two-finger adaptive robotic gripper with a rigid body capable of maintaining strength and safety while accommodating different sizes and gripping forces. The method used was 3D modelling in CAD software and performing simulations under various loading conditions using generative design and finite element analysis (FEA). The results showed that the mass of the gripper was optimized by 79% while maintaining a safety factor of 2, which demonstrated the success of the optimization and highlighted the potential for increased adaptability in various applications [5].

. Another research conducted by Alaneme, et al focused on optimizing the selection and design of Ti-based biometallic alloys for fracture and tissue rehabilitation through a review of the Finite Element Analysis (FEA) method. The research that has been conducted is based on the need to develop biocompatible materials with high mechanical strength and superior corrosion properties. The research that has been conducted aims to summarize recent developments in the application of FEA to Ti alloys as medical materials. The method used is a literature review of previous studies that used FEA to assess the performance of Ti alloys. The research results show that FEA contributes significantly to the selection of optimal material composition and design parameters for orthopaedic applications [6].

In addition, research by Kamaruzzaman, et al. focused on designing and simulating an adjustable three-axis gimbal structure using the Finite Element Analysis (FEA) method. The research that has been done is motivated by the need for a more stable and efficient gimbal system to support photography and remote sensing devices, especially in dynamic conditions. The purpose of this research is to design an optimal gimbal structure by considering factors such as stability, load, and dynamic response. The research method used is FEA-based simulation to analyze the strength and performance of the designed gimbal structure. The results show that the proposed gimbal structure performs well in terms of stability and durability and can withstand significant dynamic loads, making it a potential choice for applications that require high precision [7].

2. Method

The research method was modelling the baseplate using CAD software in Fig. 1 and Fig. 2 based on the rectifier box and simulating the baseplate design using CAE software with the FEA method to determine the effect of the material and plate thickness on the mechanical properties of the baseplate. Using FEA which divides a complex design into small parts, it is expected to visualize the stress, deflection, and material characteristics of the entire baseplate design [8].

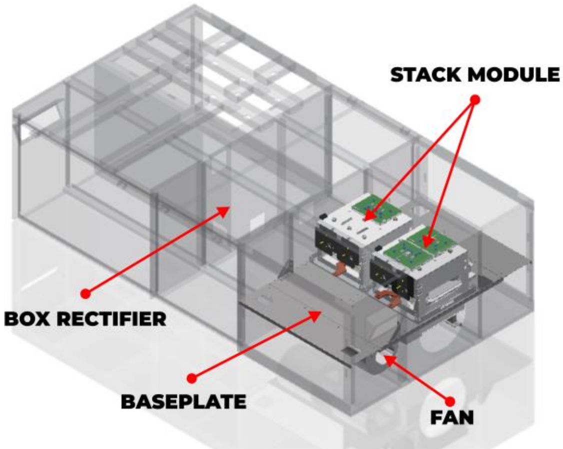

Fig. 1. PWM rectifier box

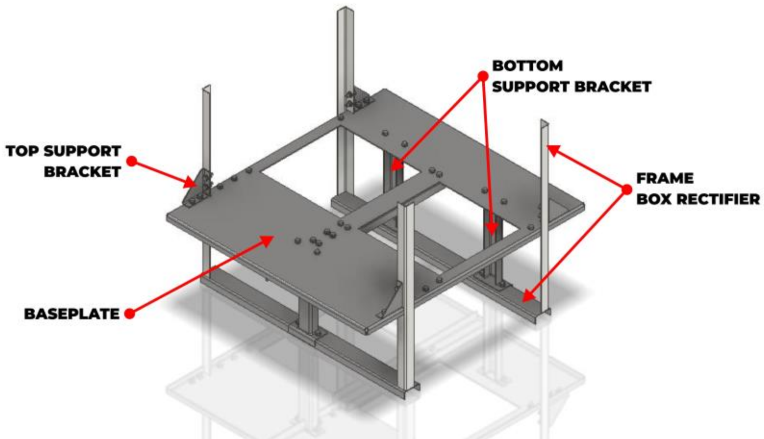

Fig. 2. 3D Design of baseplate

The type of meshing in the finite element analysis method is very important because it affects the accuracy and efficiency of the simulation. The meshing type used as shown in Fig. 3 is tetrahedral with small tetrahedron-shaped elements that form an irregular three-dimensional space structure. Tetrahedral meshing was chosen because of its ability to handle complex geometries well and allows flexibility in mesh formation on geometries that are not symmetrical or have small details [9]. Although it often requires more elements than hexahedral meshing to achieve the same accuracy, the mesh size used in this analysis is 5 mm.

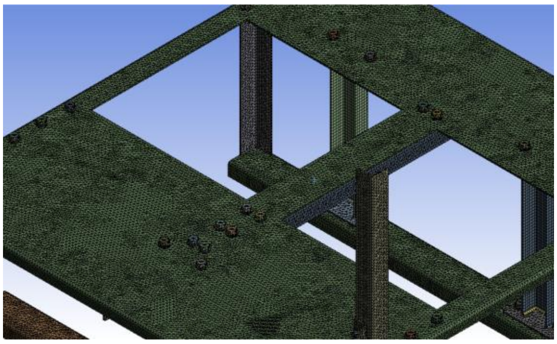

Fig. 3. Process meshing on baseplate model

Before running the analysis simulation process, the necessary focus and load settings are required so that the simulation can run. The focus setting is shown in Fig. 4, with the frame box rectifier surface as fixed support. As shown in Fig. 5, the force adjustment is on the components of the fasteners, in particular on the surface of the ring of the myrtle plate and the beam that focuses the load directly from the fan component with a load of 125 N downward. Fig. 6 shows the force setting on the baseplate component in particular on the surface concentrates the direct load of the stack module component with a load of 775 N downward. Fig. 7 shows the loading styles of the fan and stack modules defined by the downward arrow direction with each working load of 125 N and 775 N.

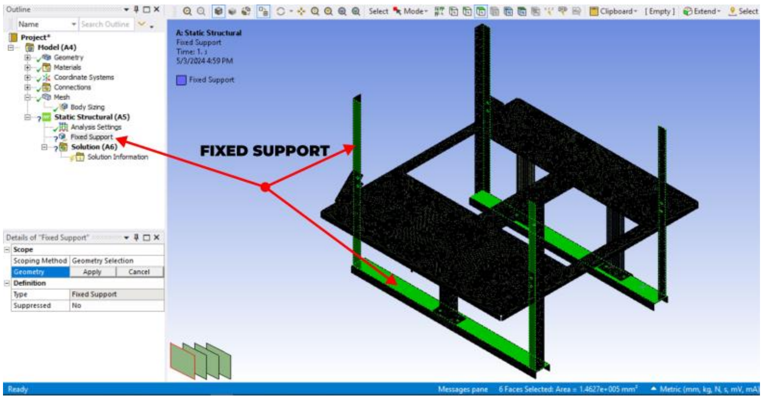

Fig. 4. Fix support settings

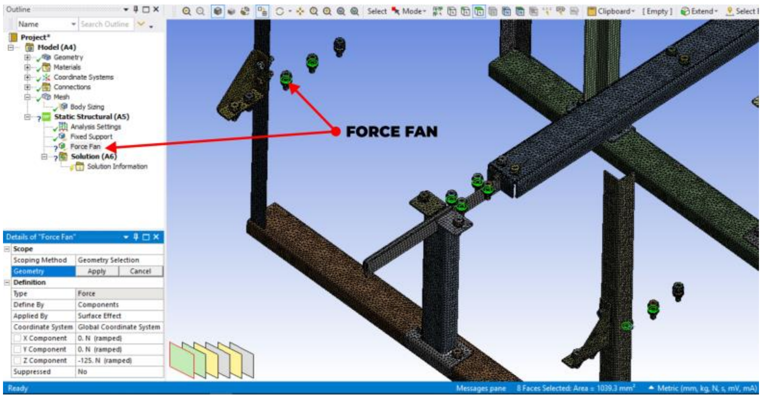

Fig. 5. Force settings for fan load

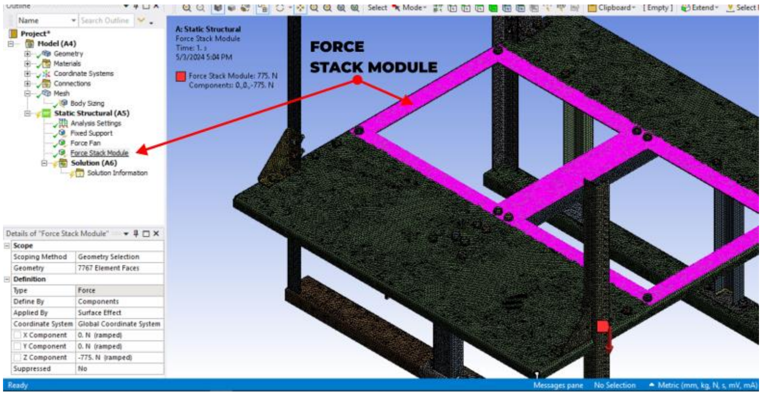

Fig. 6. Force settings for stack module load

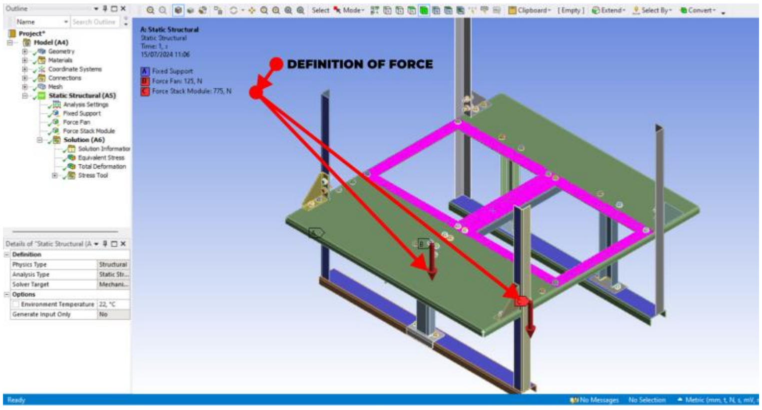

Fig. 7. Load definition

The analysis method used in this study involved a two-way ANOVA to evaluate the effect of material variation and plate thickness on the Von Mises stress, deformation, and factor of safety in the baseplate. Two-way ANOVA allows the analysis of the interaction between two independent factors, in this case, material and plate thickness, on the dependent variable being analyzed. The statistical hypothesis for this study is **H0** if there is no significant effect of variation in material and plate thickness on the values of Von Mises stress, deformation, and safety factor. **H1** if there is a significant effect of material variation and plate thickness on the values of Von Mises stress, deformation, and safety factors. If the ANOVA results show that the P-value is less than the set significance level, then H0 is rejected, and it can be concluded that the variation of material and plate thickness has a significant effect on the tested variables. Otherwise, H0 is accepted, and it can be concluded that these variations have no significant effect.

2.1. Material

The material selection of AISI 316 stainless steel, JIS G3101 SS400, and aluminium 6061-T6 for the baseplate production is based on the mechanical properties of each material that are relevant for the baseplate structural performance analysis. AISI 316 was chosen due to its high strength and excellent corrosion resistance, especially in aggressive environments. SS400 was chosen due to its wide availability, economical cost, and sufficient mechanical strength for general applications. Meanwhile, 6061-T6 aluminium offers a high strength-to-weight ratio and ease of fabrication, making it ideal for applications where weight reduction is a priority. Analysis using these three materials will provide insight into the influence of mechanical properties and resistance to stress, deformation, and safety on the baseplate. The baseplate design uses several materials such as AISI 316 stainless steel, JIS G3101 SS400, and aluminium 6061-T6, with mechanical properties that can be seen in Table 1.

Table 1. Mechanical properties of the material [10]-[12]

Property	Material			Units
	AISI 316 Stainless Steel	JIS G3101 SS400	Aluminium 6061-T6	
Density	8	7.85	2.7	g/cm^3
Yield Strength	205	245	276	MPa
Tensile Strength	515	510	310	MPa
Specific heat capacity	500	465	900	J/kg.K
Thermal conductivity	18.9	25	170	W/m.K
Modulus of Elasticity	193	206	68.9	GPa
Poisson's ratio	0.28	0.3	0.33	

3. Results and Discussion

3.1. Analysis of Von Mises Stress

The simulation results of the baseplate using CAE software with variations in material and plate thickness were used to obtain Von Mises stress as shown in Fig. 8, Fig. 9 and Fig.10.

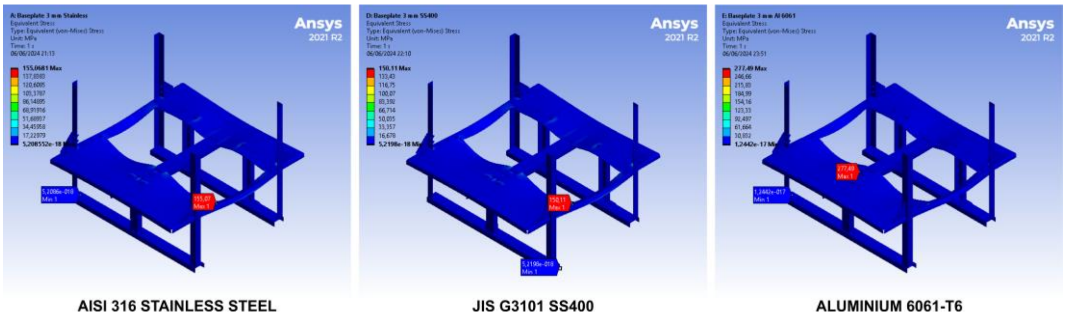

Fig. 8. Von Mises stress simulation results of 3 mm-thick plate

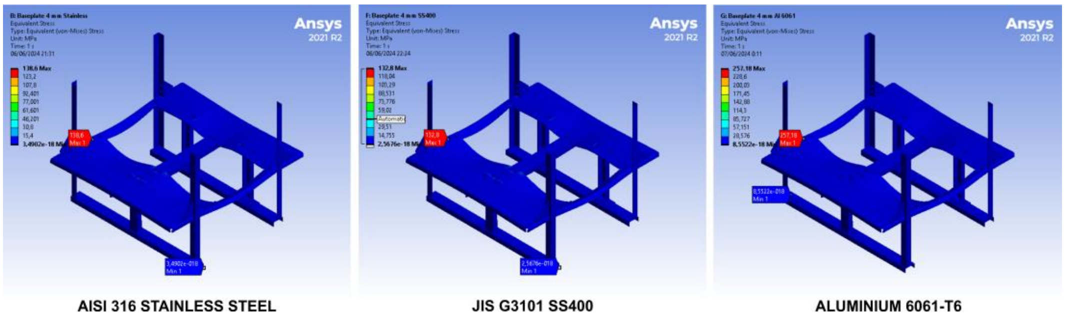

Fig. 9 Von Mises stress simulation results 4 mm-thick plate

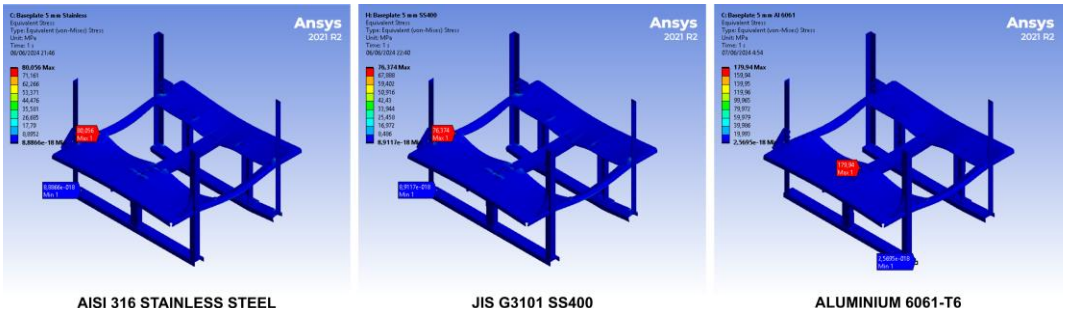

Fig. 10. Von Mises stress simulation results of 5 mm-thick plate

Table 2 and Fig. 11 show the maximum stress from the simulation results of the baseplate design, the use of material and plate thickness that shows a significant effect on the maximum Von Mises stress on the baseplate. So, it can be concluded that the SS400 material has the best ability to withstand the load so that the stress that occurs has a low value compared to aluminium and stainless steel.

Table 2. Maximum Von Mises stress simulation result

Maximum Von Mises stress (MPa)			
Thickness	**Material**		
	Alumunium 6061-T6	**JIS G3101 SS400**	**AISI 316 Stainless Steel**
3 mm	277.49	150.11	155.07
4 mm	257.18	132.8	138.6
5 mm	179.94	76.374	80.056

According to Von Mises theory, materials that experience stresses exceeding yield strength will experience plastic deformation [13], so choosing the right material and thickness is very important to ensure the safety and efficiency of the baseplate. Thus, the Von Mises stress analysis results show that plates with stainless steel and SS400 materials at all thicknesses and aluminium plates with thicknesses of 4 mm and 5 mm meet the Von Mises stress criteria. However, after considering the safety factor with a minimum safety factor of 1.5, only the 5 mm stainless steel, SS400 (3 mm, 4 mm, 5 mm), and 5 mm aluminium plates meet the Von Mises criteria.

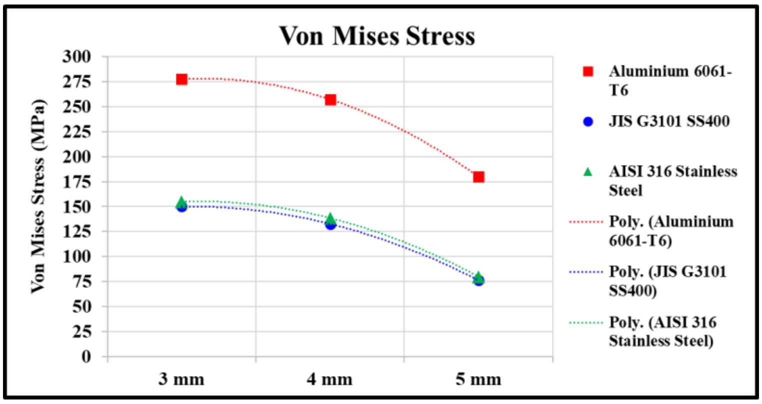

Fig. 11. Maximum Von Mises stress simulation result

Based on the data analysis, the baseplate with SS400 material with a thickness of 3 mm is recommended for the baseplate design, in addition to the Von Mises stress that occurs within the criteria as well as the production cost factor which is lower than the 4 mm and 5 mm thickness plates.

Table 3. ANOVA of Von Mises stress results

Source	DF	Adj SS	Adj MS	F-Value	P-value
Material	2	26963.0	13481.5	257.02	0.000
Thickness	2	11170.5	5585.2	106.48	0.000
Error	4	209.8	52.5		
Total	8	38343.3			

Based on the analysis of variance presented in Table 3, the P-values for material variation and plate thickness are less than the significance level $\alpha = 0.05$. The results of this two-way ANOVA indicate that material variation and plate thickness significantly affect the Von Mises stress on the base plate. Therefore, the null hypothesis (H0), stating there is no significant effect of variation in material and plate thickness on the values of Von Mises stress, is rejected and the alternative hypothesis (H1), stating that there is a significant effect of material variation and plate thickness on the values of Von Mises stress, is accepted.

3.2. Analysis of Deformation

The simulation results of the baseplate using CAE software with variations in material and plate thickness were used to obtain deformation values as shown in Fig. 12, Fig.13 and Fig. 14. As shown in Table 4 and Fig. 15, the maximum deformation that occurs with the FEA method from the simulation of the baseplate shows that material variations and plate thickness have a significant effect on the maximum deformation value of the baseplate. Therefore, the SS400 material shows the best performance in its ability to withstand loads to reduce the value of deformation that occurs compared to aluminium and stainless steel.

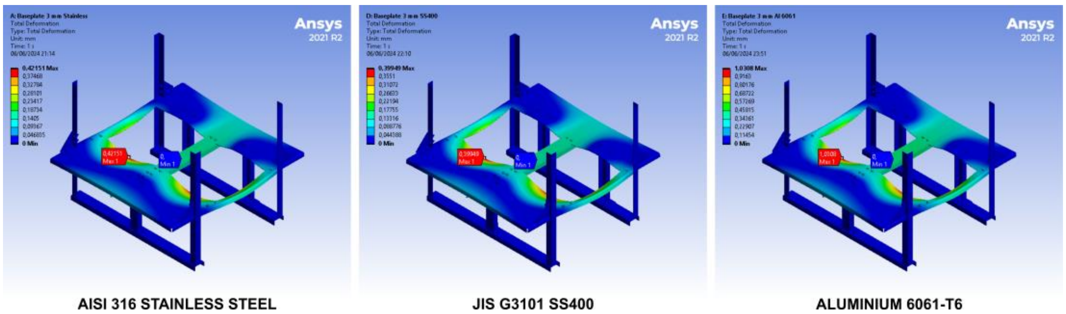

Fig. 12. Deformation simulation results of 3 mm-thick plate

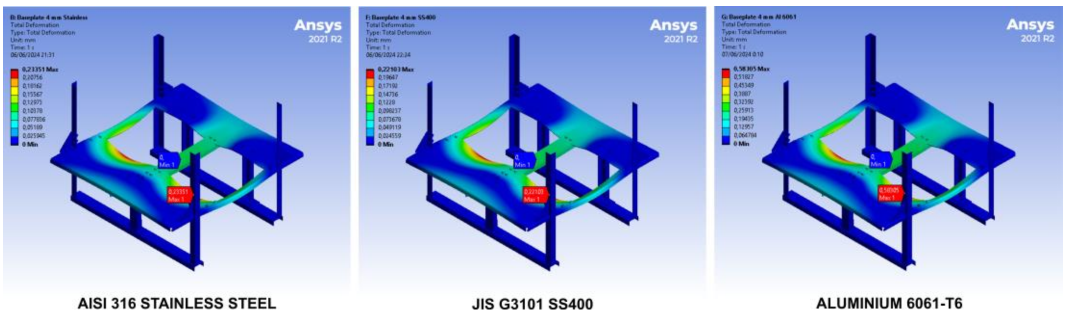

Fig. 13. Deformation simulation results of 4 mm-thick plate

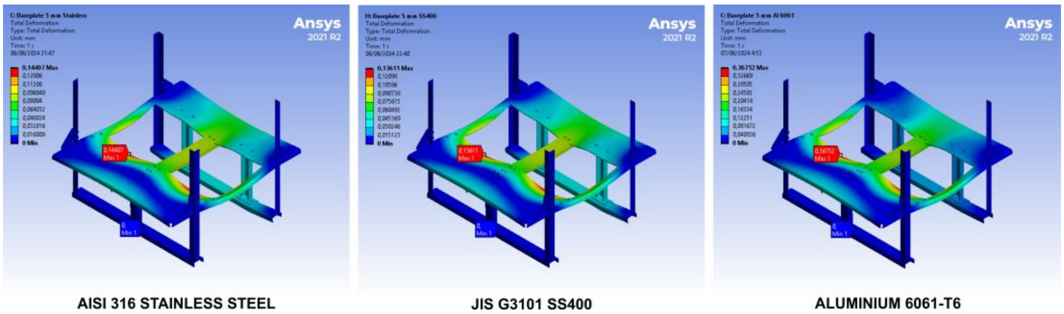

Fig. 14. Deformation simulation results of 5 mm-thick plate

Table 4. Maximum deformation simulation result

Maximum Deformation (mm)			
Thickness	**Material**		
	Alumunium 6061-T6	**JIS G3101 SS400**	**AISI 316 Stainless Steel**
3 mm	1.0308	0.39949	0.42151
4 mm	0.58305	0.22103	0.23351
5 mm	0.36752	0.13611	0.14407

Materials with higher modulus of elasticity and more thickness will experience smaller deformations under the same load [14]. In line with the Von Mises theory supporting the analysis results, it is stated that materials with lower stress will experience smaller deformation, which ensures the reliability and safety of the structure. Therefore, the material with more thickness is proven to be optimum in reducing deformation and increasing the safety factor of the stack module baseplate based on FEA.

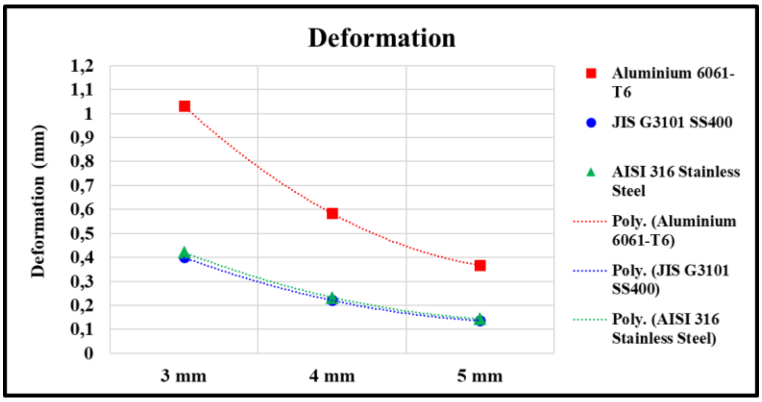

Fig. 15. Maximum deformation simulation result

Table 5. ANOVA of deformation results

Source	DF	Adj SS	Adj MS	F-Value	P-value
Material	2	0.32218	0.16109	12.03	0.020
Thickness	2	0.25165	0.12582	9.40	0.031
Error	4	0.05356	0.01339		
Total	8	0.62738			

Based on the analysis of variance in Table 5, the P-values for material variation and plate thickness are less than the significance level $\alpha = 0.05$. The results of this two-way ANOVA indicate that material variation and plate thickness significantly affect the deformation of the baseplate. Therefore, the null hypothesis (H0), stating there is no significant effect of variation in material and plate thickness on the values of deformation, is rejected, and the alternative hypothesis (H1), stating that there is a significant effect of material variation and plate thickness on the values of deformation, is accepted.

3.3. Analysis of Safety Factor

The simulation results of the baseplate using CAE software with variations in material and plate thickness were used to obtain safety factors as shown in Fig. 16, Fig.17 and Fig. 18.

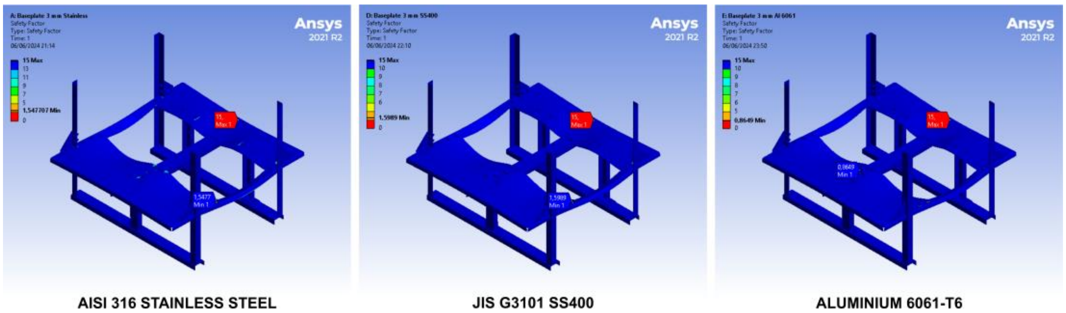

Fig. 16. Safety factor simulation results of 3 mm-thick plate

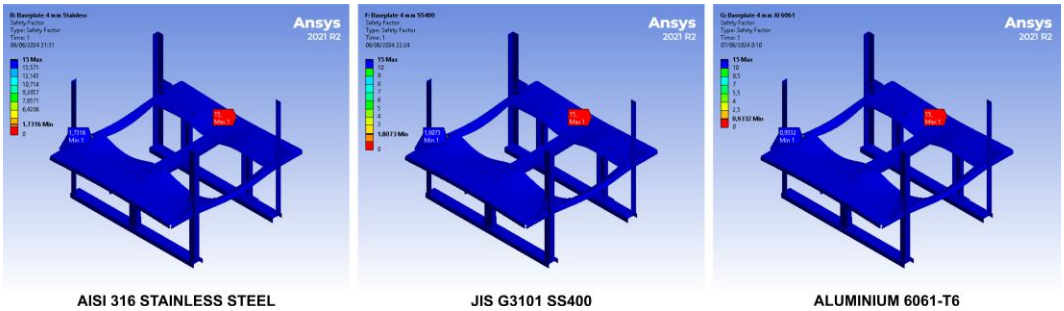

Fig. 17. Safety factor simulation results of 4 mm-thick plate

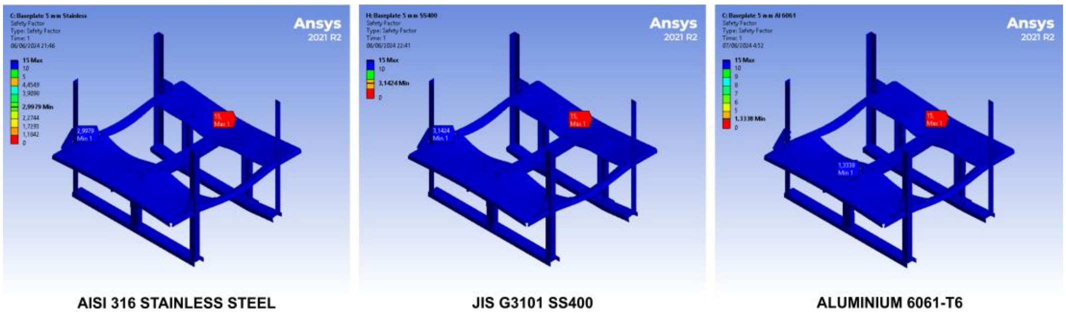

Fig. 18. Safety factor simulation results of 5 mm-thick plate

Table 6 and Fig. 15 present the minimum safety factor from the simulation results of the baseplate design, the material variation and the thickness of the plate that has a significant effect on the safety factor of the baseplate. Fig. 19 shows that the SS400 material with a thickness of 5 mm has the highest safety factor of 3.1424, which indicates that this material is the best that can withstand the load without failing. On the other hand, aluminium with a thickness of 3 mm shows the lowest safety factor of 0.8649, which means this material is most prone to failure under the same load.

Table 6. Simulation result of minimum safety factor

Minimum Safety Factor			
Thickness	**Material**		
	Alumunium 6061-T6	**JIS G3101 SS400**	**AISI 316 Stainless Steel**
3 mm	0.8649	1.5989	1.5477
4 mm	0.9332	1.8073	1.7316
5 mm	1.3338	3.1424	2.9979

In line with Von Mises theory and the FEA method, materials with higher elastic modulus and yield strength will have a greater safety factor. SS400, with a higher yield strength than aluminium and stainless steel, showed a greater safety factor at various thicknesses. Stainless steel also performed well, with the safety factor increasing with thickness, reaching 2.9979 at 5 mm thickness.

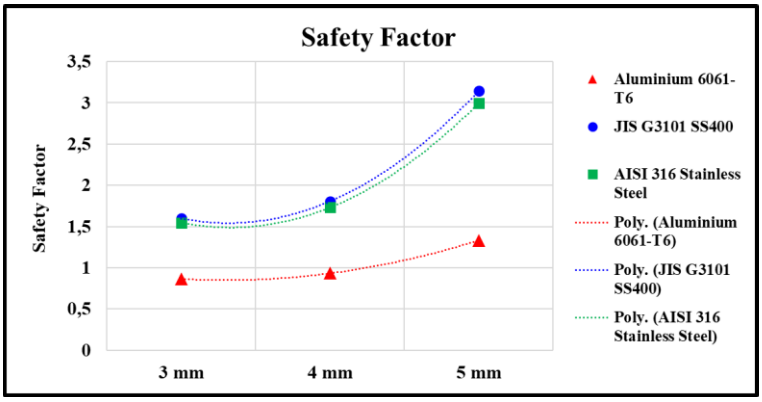

Fig. 19. Minimum safety factor simulation result

The increase in plate thickness also contributes significantly to the increase in the safety factor. This is in line with the deformation theory which states that thicker structures tend to be stiffer and have a greater ability to withstand loads without undergoing significant deformation [15]. Thus, for applications that require high resistance to load acceptance, SS400 material with more thickness is an optimal choice to ensure a high safety factor and prevent structural failure of the baseplate.

Table 7. ANOVA of safety factor results

Source	DF	Adj SS	Adj MS	F-Value	P-value
Material	2	2.4045	1.2022	11.43	0.022
Thickness	2	2.3571	1.1785	11.20	0.023
Error	4	0.4208	0.1052		
Total	8	5.1824			

Based on the analysis of variance presented in Table 7, the P-values for material variation and plate thickness are less than the significance level $\alpha = 0.05$. The results of this two-way ANOVA indicate that material variation and plate thickness significantly affect the safety factor on the base plate. Therefore, the null hypothesis (H0), stating there is no significant effect of variation in material and plate thickness on the values of the safety factor, is rejected, and the alternative hypothesis (H1), stating that there is a significant effect of material variation and plate thickness on the values of the safety factor, is accepted.

4. Conclusion

Overall, SS400 material showed the best performance in stress reduction, deformation minimisation, and safety factor improvement, making it the most reliable and safe material to use compared to aluminium and stainless steel, especially at larger plate thicknesses. From the results, the best baseplate based on the minimum Von Mises stress value, minimum deformation, and maximum factor of safety is the baseplate with JIS G3101 SS400 material with a thickness of 5 mm, which has a minimum Von Mises stress value of 76.374 MPa, a minimum deformation value of 0.13611 mm, and a maximum factor of safety value of 3.1424.

Acknowledgement

The authors acknowledge, appreciate, and are grateful for the support of PT. X for providing the necessary resources and facilities to conduct this research and the Department of Mechanical Engineering, State Polytechnic of Malang, for their support throughout the research process.

References

[1] M. Kapetanović, M. Vajihi, and R. M. P. Goverde, “Analysis of hybrid and plug-in hybrid alternative propulsion systems for regional diesel-electric multiple unit trains,” Energies (Basel), vol. 14, no. 18, 2021.

[2] J. Boukhenfouf et al., “Modular Multilevel DC Converter: Impact of the Control on the Design and Efficiency,” in 2023 25th European Conference on Power Electronics and Applications, EPE 2023 ECCE Europe, 2023.

[3] F. D. Wijaya, Y. F. Sidik, R. Irnawan, A. I. Cahyadi, and M. B. Frasetyo, “Droop Control Strategy of Controlled Three-Phase PWM Rectifier for Hybrid Train Propulsion,” in 2023 15th International Conference on Information Technology and Electrical Engineering, ICITEE 2023, 2023.

[4] M. T. H. Khan and S. Rezwana, “A review of CAD to CAE integration with a hierarchical data format (HDF)-based solution,” Journal of King Saud University - Engineering Sciences, vol. 33, no. 4, 2021.

[5] R. J. C. Tan Ai, M. C. S. Pilapil, R. A. A. Shi, J. L. D. Badugas, and S. M. T. Cheng, “Design and Optimization of an Adaptive Robotic Gripper using Finite Element Analysis and Generative Design,” Comput Aided Des Appl, vol. 20, no. 1, 2023.

[6] K. K. Alaneme, S. A. Kareem, B. N. Ozah, H. A. Alshahrani, and O. A. Ajibuwa, “Application of finite element analysis for optimizing selection and design of Ti-based biometallic alloys for fractures and tissues rehabilitation: a review,” 2022.

[7] M. A. Kamaruzzaman, H. H. Basri, and M. N. Ayob, “Design and Simulation of a Customize Three-axis Gimbal Structure using Finite Element Analysis Method,” Journal of Advanced Research in Applied Sciences and Engineering Technology, vol. 30, no. 1, 2023.

[8] Z. Zhang, "Application of finite element analysis in structural analysis and computer simulation," Applied Mathematics and Nonlinear Sciences, vol. 0, no. 0, May 2023.

[9] T. Schneider, Y. Hu, X. Gao, J. Dumas, D. Zorin, and D. Panozzo, "A Large-Scale Comparison of Tetrahedral and Hexahedral Elements for Solving Elliptic PDEs with the Finite Element Method," ACM Trans Graph, vol. 41, no. 3, 2022.

[10] M. H. Nubly and H. Yudo, "Strength analysis of propeller shafting on orca class fisheries inspection boat using finite element method," *International Journal of Civil Engineering and Technology*, vol. 8, no. 10, 2017.

[11] H. Pranoto, B. Darmono, and G. Widyaputra, "Strength Analysis of the Frame Structure with the Impact Load Between the ASTM A36 And JIS G3101 Materials in the Electric Car E-Falco," *International Journal of Advanced Technology in Mechanical, Mechatronics and Materials*, vol. 3, no. 1, 2022.

[12] M. Srinivasa, Y. Rammohan, M. Sadashiva, and N. Santhosh, "Effect of Shock Waves on the Hardness of Graphene Reinforced Aluminium Composites," *Undefined*, no. April, 2020.

[13] R. F. Silva, P. G. Coelho, F. M. Conde, B. R. Santos, and J. P. Oliveira, "Minimizing the maximum von Mises stress of elastic continuum structures using topology optimization and additively manufactured functionally graded materials," *Comput. Struct.*, vol. 301, no. September 2023, p. 107469, 2024, doi: 10.1016/j.compstruc.2024.107469.

[14] F. Qin, S. Du, Y. Li, X. Zhao, and Y. Xu, "Effects of strain rate on dynamic deformation behavior and microstructure evolution of Fe-Mn-Cr-N austenitic stainless steel," *Mater. Charact.*, vol. 212, no. March, p. 113975, 2024, doi: 10.1016/j.matchar.2024.113975.

[15] J. H. Fu, K. P. Zhu, J. G. Gong, F. H. Gao, and F. Z. Xuan, "Correlation between reliability and safety factor based methods in characterizing uncertainty of creep rupture properties," *Int. J. Press. Vessel. Pip.*, vol. 206, no. September, p. 105068, 2023, doi: 10.1016/j.ijpvp.2023.105068.