JOURNAL OF
MECHANICAL ENGINEERING,
SCIENCE, AND INNOVATION
ITATS
Research Article

Comparison of Manual, AI Generative, and Hybrid Design on Structural Performance and Manufacturing Aspects of Truck Wheel Mounting Aid Frame using FEA in Fusion 360

Journal of Mechanical Engineering,
Science, and Innovation
e-ISSN: 2776-3536
2026, Vol. 6, No. 1
DOI: 10.31284/j.jmesi.2026.v6i1.8749
ejurnal.itats.ac.id/jmesi

Muhammad Najmul Aulia[1]* and Dony Satriyo Nugroho[1]

[1]Faculty of Engineering, Dian Nuswantoro University, Indonesia

Corresponding author:
Muhammad Najmul Aulia
Faculty of Engineering, Dian Nuswantoro University, Indonesia
Email: 512202201654@mhs.dinus.ac.id

Abstract

This study analyzes the design of a truck wheel mounting fixture, using three different approaches: manual design, AI-driven generative design, and hybrid design, employing Finite Element Analysis (FEA) in Autodesk Fusion 360. The objective is to identify the most effective design in terms of structural strength, mass efficiency, and manufacturability. All models were analyzed under identical static loading conditions using AISI 304 stainless steel with a distributed load of 2,500 N. A mesh independence study was conducted using element sizes of 10%, 5%, and 1% of the model size to ensure numerical reliability. The results show that the manual model has a stress of 103.01 MPa, a deformation of 2.6 mm, a safety factor of 2.08, and a mass of 17.76 kg. The AI generative model achieved the lowest mass of 5.38 kg but exhibited excessive stress of 1,844.6 MPa, deformation of 66.7 mm, and a safety factor of 0.11, indicating structural inadequacy under the applied load. The hybrid model demonstrated the best balance, with the lowest stress of 82.01 MPa, minimum deformation of 1.6 mm, highest safety factor of 2.62, and a reduced mass of 8.42 kg. These findings suggest that combining generative load path optimization with manual reinforcement can significantly improve structural efficiency while maintaining practical manufacturability. Therefore, the hybrid approach is considered the most suitable design for workshop-based truck wheel mounting applications.

Keywords: *Manual Design, Ai Generative Design, Hybrid Design, Structural Performance, FEA.*

Received: February 18, 2026; Received in revised: April 27, 2026; Accepted: April 30, 2026
Handling Editor: Afira Ainur Rosidah

INTRODUCTION

The wheel installation process in heavy-duty vehicles often requires significant manual effort due to the large mass of truck wheels, which typically ranges from 70 to 130 kg. Improper handling during installation may increase operator fatigue, reduce alignment accuracy, and create safety risks in workshop environments. Therefore, mechanical assistive devices are needed to support safer and more efficient wheel mounting operations. The structural frame is the main load-bearing component of a wheel mounting aid. It must be capable of supporting wheel loads, maintaining alignment stability, and resisting deformation during operation. In engineering practice, frame structures are commonly developed using conventional manual design methods based on empirical experience and simplified calculations. Although such designs are generally robust and easy to manufacture, they often use excessive material, resulting in higher mass and lower efficiency.

The development of design and engineering technology in the manufacturing industry has experienced rapid growth along with the integration of digital design systems and computational-based simulations. The use of Computer-Aided Design (CAD) and Computer-Aided Engineering (CAE) software not only serves to visualize models but also allows for analysis of strength, deformation, and structural bearing capacity before components are manufactured. Autodesk Fusion 360 is one such platform that combines 3D modeling, finite element-based numerical simulation, and an artificial intelligence-powered shape optimization system through the Generative Design feature [1].

Generative Design operates on the principle of topology optimization, which involves reducing material in areas that do not significantly contribute to structural stiffness while maintaining load-bearing capacity [2]. This approach aims to achieve optimal material distribution based on compliance and stiffness criteria; however, previous studies have shown that topology optimization often produces complex geometries with thin members and irregular contours that require further modification to meet manufacturability requirements when using conventional fabrication processes [3]. This process utilizes a load-based shape-finding algorithm to generate a variety of geometric alternatives with a high strength-to-mass ratio. However, the resulting shapes are generally organic, with complex branching and non-uniform contours. Although computationally optimal, these shapes are often incompatible with conventional manufacturing methods such as plate cutting, steel profile forming, and linear jig-based welding. Furthermore, organic structures tend to potentially produce stress concentrations on thin branches, requiring geometric modifications before production [4].

On the other hand, manual design is still widely used in workshops and small industries due to its simplicity, ease of operator understanding, and suitability to available manufacturing capabilities [5]. However, manual design tends to result in larger structural masses and uneven load distribution [6]. Therefore, the Hybrid Design approach was born, which combines the basic forms resulting from Generative Design and then manually simplified to suit fabrication capabilities, without losing the characteristics of mass efficiency and structural strength [7], [8]. This approach serves as a bridge between computational-based design and the needs of manufacturing realization in the field.

The application of this approach is relevant in the design of truck wheel mounting aids. Truck wheels and rims can weigh up to 70–130 kg, making repetitive manual handling potentially harmful to workers, leading to spinal injuries and musculoskeletal disorders [9]. ISO 11228-1, on manual handling ergonomics, states that repetitive lifting loads should be supported by mechanical equipment. Furthermore, SNI ISO 14121-1:2018 requires that work aids have a safety factor of at least twice the working load and must not undergo permanent deformation during use. Therefore, reverse engineering was performed on the tools used in the workshop to obtain geometric data, fulcrum points, and loading conditions as a basis for design [10].

The material used is AISI 304 Stainless Steel because it has good corrosion resistance, ease of cutting and welding processes, and mechanical properties suitable for structures with static loading. This material has a yield stress of around 215 MPa and a maximum tensile strength of around 515 MPa, so it is able to withstand repeated workloads in a workshop environment (Perancang, 2025). Design performance evaluation was carried out using Finite Element Analysis (FEA) to assess the von Mises stress distribution, total deformation, and safety factor in each design model [6].

Previous research in industrial engineering has focused on work ergonomics, production aids, and fixture design. Several studies, such as those by Siska & Gunawan [11], have highlighted issues of physical workload, unergonomic postures, and the design of safe aids for workers. Other studies, such as those by Putra et al., [12] have focused more on the Axiomatic Design and House of Quality methods for designing aids that meet operator needs. Although both studies contribute to the context of work aids and design, they do not explore aspects of structural optimization or comparisons with modern design methods such as Generative Design. Most research on manufacturing aid design still uses conventional approaches (manual design) or ergonomic approaches alone, without involving computational evaluations such as Finite Element Analysis (FEA), so the structural performance of components is not assessed quantitatively. There has not been any research that applies the three design approaches (Manual, Generative, Hybrid) to truck wheel mounting aid components with integrated analysis using FEA in Fusion 360 which requires a combination of high structural strength, minimal mass, and ease of fabrication factors that are very suitable for comparison through multi-method design studies.

Based on the description, this study aims to compare the structural performance of three truck wheel mounting tool frame design approaches, namely manual design, Generative Design-based design, and Hybrid Design, through numerical analysis using FEA by considering mass efficiency, structural strength, minimum safety factor 2, and suitability to workshop manufacturing capabilities.

Recent studies show that while generative design and topology optimization have matured into powerful tools for producing lightweight and high-performance structures, challenges remain in translating computationally optimal geometries into manufacturable parts. Hybrid methods that integrate generative design with other optimization strategies, including manual simplification or additive manufacturing constraints, have been proposed to address these gaps. For example, new hybrid generative design frameworks combine multiple design-for-manufacturing methods with optimization algorithms to produce functionally efficient yet producible geometries, particularly for lightweight structural applications in industrial contexts. These methods demonstrate significant material reduction while maintaining mechanical integrity and performance targets, underscoring the value of integrating design creativity with practical manufacturing constraints [13].

In parallel, the literature emphasizes the increasing role of *Finite Element Analysis (FEA)* as an essential tool for validating and comparing structural performance across design alternatives. Recent reviews of topology optimization methods highlight how FEA remains central within optimization loops to predict stress distribution, deformation, and overall structural behavior with high accuracy, especially when combined with machine learning or surrogate models to accelerate computations. Additionally, comparative studies between generative design, topology optimization, and hybrid approaches reinforce the need to quantify performance metrics such as stiffness, safety factors, and manufacturability through numerical analysis, rather than relying solely on conceptual or ergonomic evaluations. Collectively, these developments support the need for systematic multi-method design assessments in engineering research, particularly when balancing structural efficiency with real-world fabrication capabilities [14].

METHODS AND ANALYSIS

This study employs a numerical experimental approach based on Finite Element Analysis (FEA) using Autodesk Fusion 360 software. The research steps are systematically outlined as shown in Figure 1.

The research steps are shown in Figure 1, beginning with a literature review, field identification, framework modeling, parameter tuning, numerical simulation, mesh independence study, and results analysis. Design methodologies in mechanical engineering are now shifting from manual approaches to more efficient computational techniques. Manual design relies on experience and empirical calculations, producing safe components but often resulting in material waste. In response, Generative Design and Topology Optimization have been developed to generate lightweight and efficient design alternatives by minimizing mass without compromising structural integrity [15].

The load configuration applied to the frame of the truck wheel mounting aid is illustrated in Figure 1, showing the load locations, directions, distribution, and support constraints that represent actual operating conditions. The detailed boundary conditions and simulation assumptions are summarized in Tables 1 and 2. The next stage is meshing,

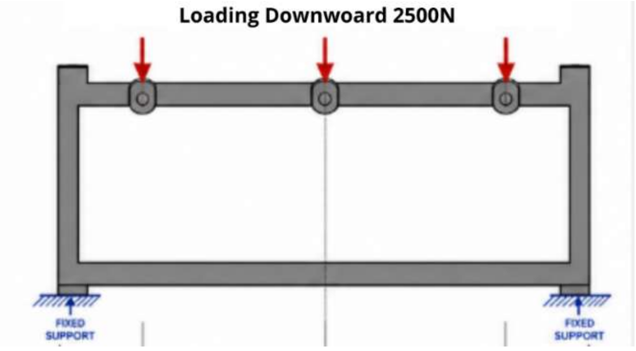


Figure 1. Loading diagram

Table 1. Boundary Condition parameters

Parameter	Specification	Description
Support Type	Fixed Support	Applied at the lower frame connection to the base
Degree of Freedom (DOF)	All DOF constrained (UX, UY, UZ, RX, RY, RZ = 0)	No translation or rotation occurs
Load Type	Distributed Load	Load applied at the wheel support bracket area
Load Magnitude	2500 N	Vertical downward load
Load Direction	-Z (vertical downward)	Consistent with gravitational force
Load Location	Upper beam	Wheel contact area or supporting bracket

Table 2. Material Properties

Property	Value
Material	AISI 304 Stainless Steel
Densitas Material	8000 kg/m^3
Modulus Elastisitas	193 GPa
Yield Strength	215 MPa

Table 3. Meshing Parameter

Parameter	Value
Average Element Size	1%, 5% and 10% of model size
Maximum Aspect Ratio	10
Element Type	Parabolic
Mesh Type	Solid

which involves discretizing the model into smaller elements with the parameters shown in Table 3.

The simulation was performed using average element sizes in Fusion 360 with variations of 1%, 5%, and 10% of the model size. The finite element model was discretized using three-dimensional solid elements. Parabolic element formulations were employed to improve the accuracy of stress predictions, particularly in regions with high stress gradients. A maximum model size and aspect ratio of 10 were used to control element distortion. The use of solid elements ensures an accurate representation of the three-dimensional geometry.

All design models were analyzed using Finite Element Analysis (FEA) in the Autodesk Fusion 360 Static Stress Simulation module, with evaluation parameters including von Mises stress, total deformation, and the safety factor relative to the material's yield strength. Boundary conditions were defined by applying fixed constraints at the base of the frame to simulate ground contact. A static vertical load was applied to the wheel support surface to represent operational loading conditions. All structural joints were assumed to be fixed, representing welded joints in the shop fabrication process. The evaluation was conducted to assess the structural capacity to withstand the applied load and distribute forces uniformly [17].

Mesh independence studies were conducted to ensure that the numerical results were not significantly influenced by mesh size. Three average element size settings 1%, 5%, and 10% of the model size were evaluated for all design configurations. The final stage of the study involved a comparative performance assessment of the three design approaches based on mass efficiency, maximum stress, deformation, minimum safety factor, and manufacturability in a workshop environment. The optimal design was selected based on a balance between structural strength and manufacturability.

RESULTS AND DISCUSSIONS

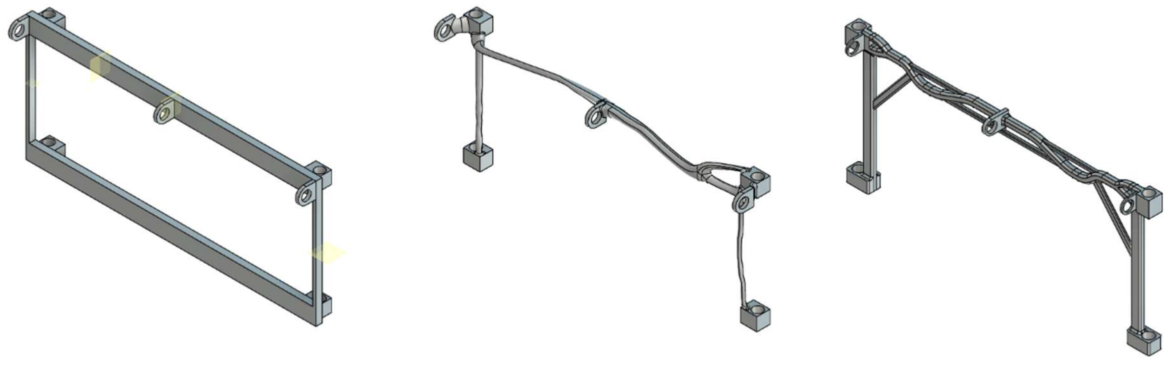
This study analyzes truck wheel mounting frames using three different design models: manual, generative AI, and hybrid. These three models have distinct shape and structural characteristics depending on the design approach used. The manual design is based on conventional engineering considerations, using straight bars and simple joints that are easy to weld. The generative AI design is automatically generated by a Generative Design algorithm to minimize mass. The resulting shape is organic, with some very slender sections. The hybrid design combines the form efficiency of generative design with reinforcement at weak points to facilitate conventional production [18].

A wheel mounting aid is a mechanical device designed to facilitate the mounting of truck wheels using a hydraulic jack as the primary lifting mechanism. During operation, the wheel is placed on the support frame, and the jack is activated to lift the frame vertically, aligning the wheel with the vehicle's hub. Once the wheel is aligned with the axle, the operator can perform alignment and installation without needing to manually lift the load. The frame of this device serves as the primary support structure for the wheel and as a load-transfer mechanism from the wheel to the hydraulic jack. In addition to supporting the vertical load, the frame maintains the stability of the wheel's position to ensure safer and more precise installation. Therefore, the frame must possess sufficient strength and stiffness to withstand the working load without experiencing excessive deformation or structural failure.

The physical properties used for the three models were obtained from the Properties feature in Autodesk Fusion 360; the material used was AISI 304 stainless steel. The simulation was performed under static loading conditions using three-dimensional parabolic solid elements to improve accuracy in capturing stress distribution, particularly in areas with complex geometries. The external load applied was 2,500 N at the wheel mounting location, while the supporting areas were fully considered fixed. This load is equivalent to 255 kg and was intentionally selected to represent a conservative loading scenario. This value exceeds the wheel weight, which ranges from 70 to 130 kg, and is applied to account for dynamic amplification, uneven load distribution, and safety considerations during operation.

The physical properties used for the three models were obtained from the Properties feature in Autodesk Fusion 360; the material used was AISI 304 stainless steel. The simulation was performed under static loading conditions using three-dimensional parabolic solid elements to improve accuracy in capturing stress distribution, particularly in areas with complex geometries. The external load applied was 2,500 N at the wheel mounting location, while the supporting areas were fully considered fixed. This load is equivalent to 255 kg and was intentionally selected to represent a conservative loading scenario. This value exceeds the wheel weight, which ranges from 70 to 130 kg, and is applied to account for dynamic amplification, uneven load distribution, and safety considerations during operation.

The manual design model was created using beam elements with uniform cross-sections and a simple geometric arrangement. The relatively even distribution of material



(a) (b) (c)

Figure 2. (a) Results of manual model properties (b) Result of AI Generative model properties (c) Result of hybrid model properties

Table 4. Properties result for each design model

Parameter	Manual	AI Generative	Hybrid
Mass	17.76 Kg	5.38 Kg	8.42 Kg
Volume	2.220.000 $mm3$	672.897 $mm3$	1.052.000 $mm3$
Mass Reduction to Manual	-	69.7 % lighter	52.6 % lighter
Geometric Composition	Stem straight	Branched organic contour	Combination of straight bars and optimized contours
Material Deployment	Flat	Concentrated on the load path	Selective at critical points
Manufacturing Readiness	Very high	Very low	High

(Source: Processed by researchers)

in the manual model provides good structural stiffness but results in a relatively large total mass. This indicates that material usage is not entirely efficient in sections not subjected to stress concentrations. Although less than optimal in terms of weight, the manual design offers the advantage of ease of fabrication, as all elements can be produced using cutting and welding methods commonly employed in heavy-duty vehicle workshops. This straight-bar frame design approach is known to maintain a stable flow of structural forces [19].

Quantitatively, the AI generative model has the lowest mass of ±5.38 kg, resulting in a 69.7% reduction compared to the manual design, which has a mass of ±17.76 kg. The hybrid model shows a mass of ±8.42 kg, or 52.6% lighter than the manual design, which means that geometric modifications based on the force flow pattern of the generative design can significantly reduce material usage without sacrificing the main structural shape. However, the very high mass efficiency of the generative model is inversely proportional to structural stability, because the extreme reduction in cross-section results in stress concentrations at the joints and transition points of the cross-section thickness. This causes the generative model to have a tendency to experience excessive deformation when subjected to operational loads [20]. On the other hand, the manual design model provides good structural stiffness because the material distribution is evenly distributed throughout the component, so that the load is distributed stably and not concentrated at a particular point. However, this character makes material use less efficient, because some parts receive more material than their actual structural contribution [19]. This condition is the basis that manual designs are generally strong, but heavy and uneconomical.

The organic contours in the generative AI model are visible at the joints between the main beam and the connecting elements, indicating a topology optimization process that directs material only along the main load paths, resulting in a gradually tapering cross-section. While this approach is effective in significantly reducing mass, variations in cross-section thickness and non-uniform shape transitions can potentially lead to stress concentrations when the structure is loaded. Furthermore, such organic shapes are incompatible with conventional cutting and welding processes, requiring additive manufacturing, which is not commonly available in heavy vehicle workshops [20].

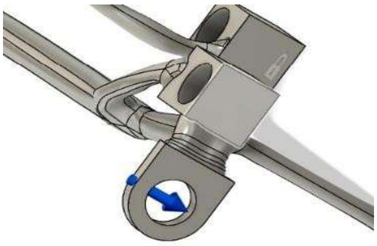


Figure 3. Organic parts of the AI Generative model

The hybrid design model was developed by adopting the primary load path structure from the generative model and then simplifying the shape and thickening certain elements to accommodate conventional manufacturing processes. The hybrid model demonstrates a significant reduction in mass compared to manual designs, while retaining load-bearing structural elements along the primary load paths. Local reinforcement is applied to areas with previously minimal cross-sections, resulting in a more even stress distribution and increased structural stiffness without excessive mass gain. Additionally, the modified shape remains producible using cutting, bending, and welding processes available in heavy-duty vehicle workshops. This approach aligns with the principles of vehicle frame structural optimization, where local cross-sectional adjustments can improve mechanical performance without altering the primary geometric configuration [21]. Before determining the most appropriate design, a comparative analysis of the three models was required based on physical parameters and manufacturability. This comparison aimed to assess the relationship between the design approach and material efficiency, structural stability, and the design's manufacturability using available workshop facilities.

A mesh independence study was conducted using three mesh sizes 10%, 5%, and 1%, resulting in different numbers of elements, as shown in Table 5. Based on the model size to ensure that the simulation results are not sensitive to discretization. The graph of the simulation results based on the number of mesh elements is shown in Figure 5.

Table 5. Mesh Statistics of Manual Design

Average Element Size	Node	Total Element
1%	32341	16791
5%	5022	2019
10%	3967	1600

(Source: Processed by researchers)

Table 6. Mesh Statistics of Generative Design

Average Element Size	Node	Total Element
1%	36714	20744
5%	24739	13424
10%	9620	5190

(Source: Processed by researchers)

Table 7. Mesh Statistics of Hybrid Design

Average Element Size	Node	Total Element
1%	96891	57843
5%	94416	55238
10%	94023	55083

(Source: Processed by researchers)

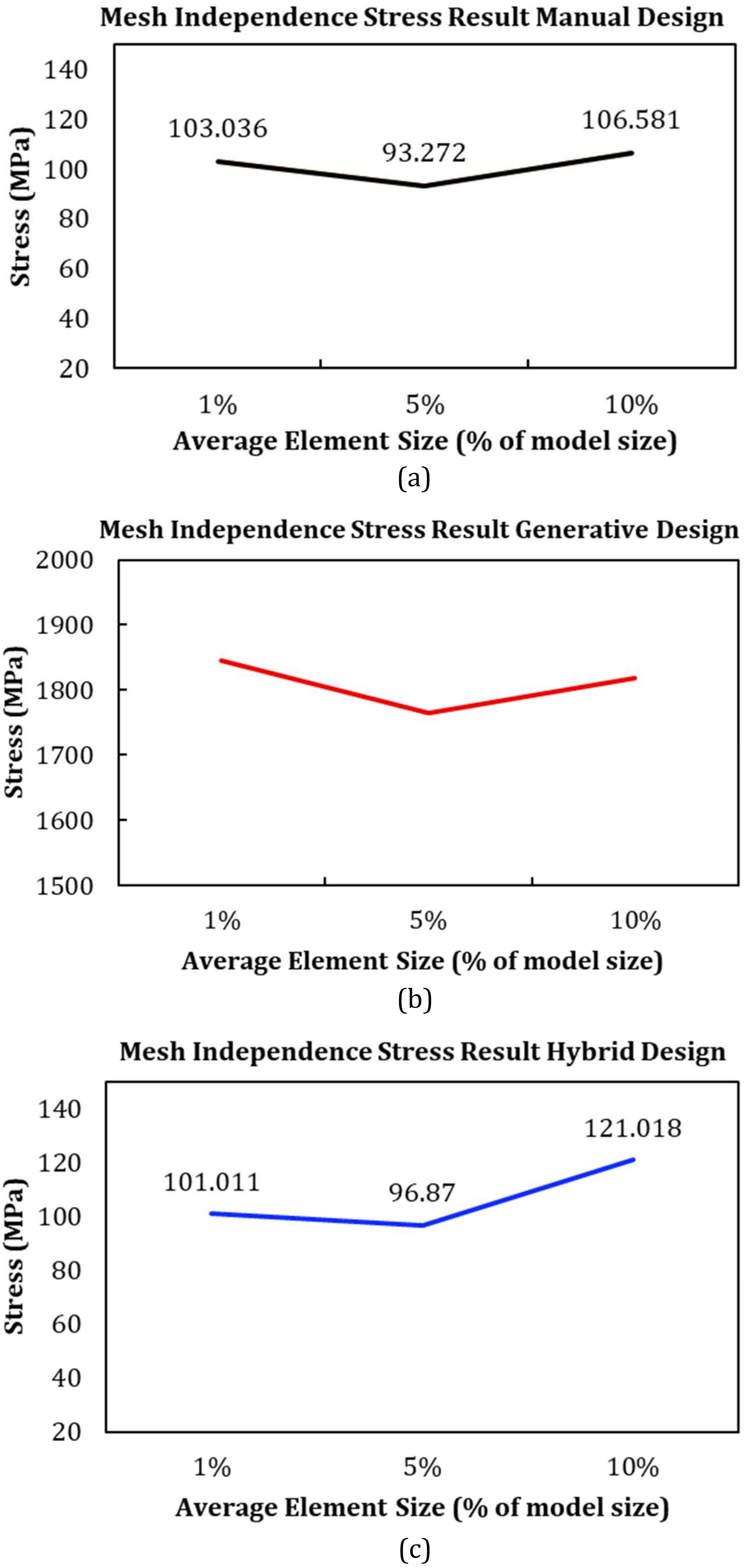
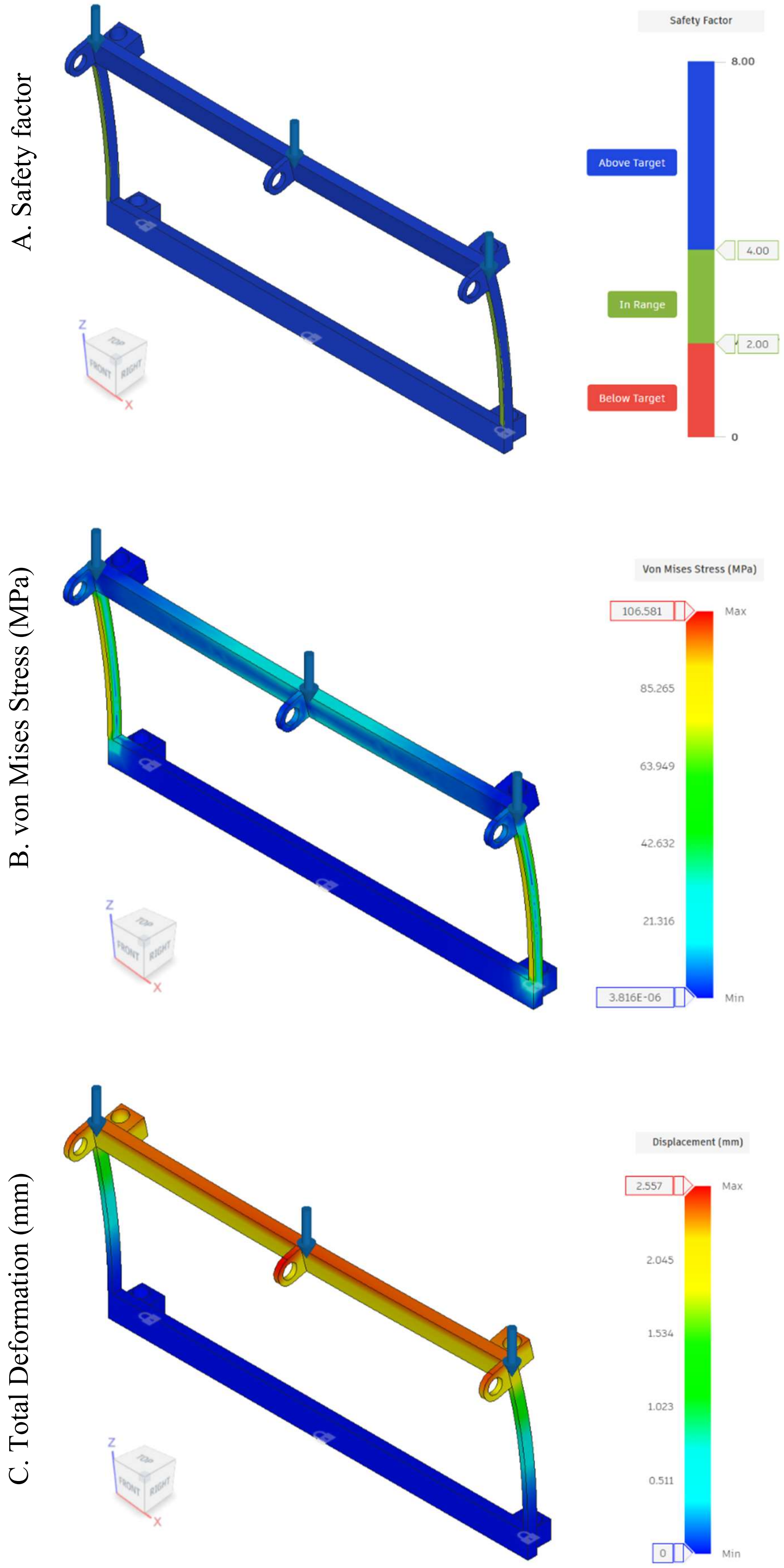
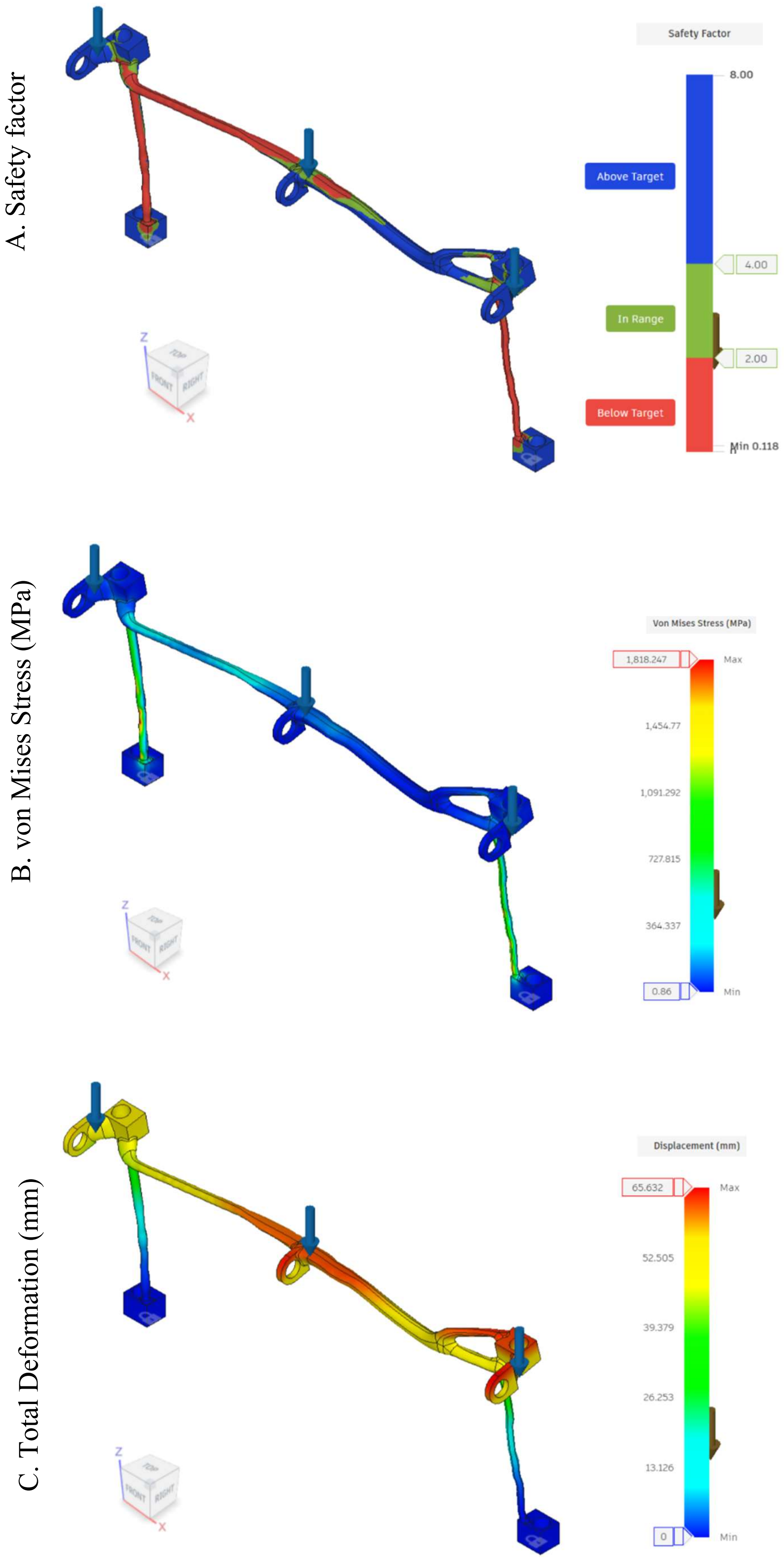


Figure 4. (a) Mesh independence stress result manual design (b) Mesh independence stress result *generative* design (c) Mesh independence stress result *hybrid* design

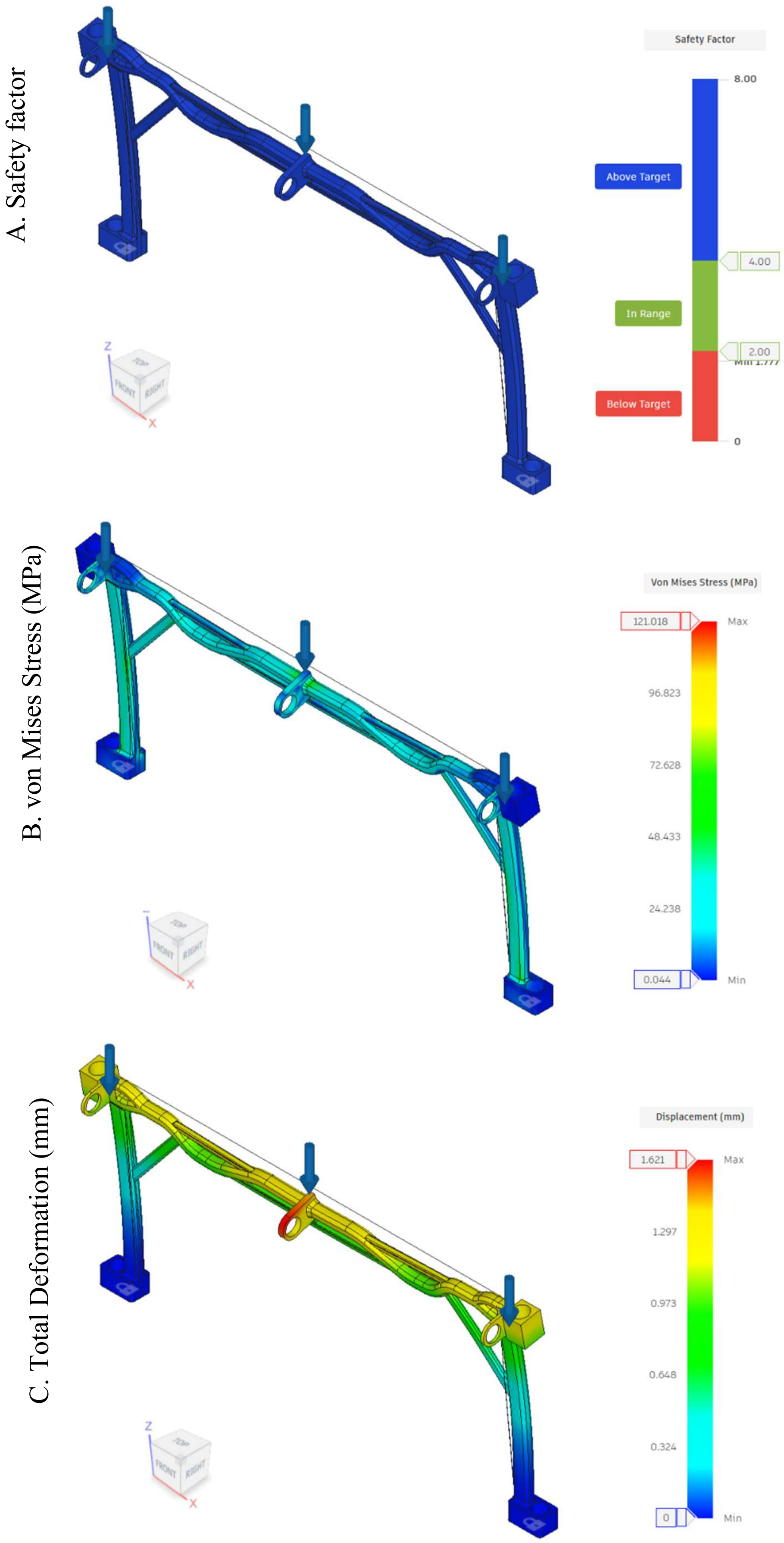
The results of the FEA simulation contour using Fusion 360 are shown in the following figure.


A. Safety factor
Safety Factor
8.00
Above Target
4.00
In Range
2.00
Below Target
0
Z
TOP
FRONT
RIGHT
X
B. von Mises Stress (MPa)
Von Mises Stress (MPa)
106.581
Max
85.265
63.949
42.632
21.316
3.816E-06
Min
C. Total Deformation (mm)
Displacement (mm)
2.557
Max
2.045
1.534
1.023
0.511
0
Min

(a). Manual



(b). AI Generative



(c). Hybrid

Figure 5. Simulation results of the three design models: (a) Manual, (b) AI Generative, (c) Hybrid. Each model shows (A) Safety Factor, (B) von Mises Stress (MPa), and (C) Total Deformation (mm).

Table 8. Simulation results for each design model

Model	Safety Factor	Von Mises Stress (Mpa)	Deformation (mm)
Manual	2.08	103.01	2.6
AI Generative	0.11	1844.6	66.7
Hybrid	2.62	82.01	1.6

(Source: Processed by researchers)

For the manual design, the stress values using a 10% mesh variation range from 106.581 MPa, the results for a 5% mesh variation are 93.272 MPa, and for a 1% mesh variation, approximately 103.036 MPa, indicating stable numerical behavior. The results of the hybrid design show that the stress decreases from 121.018 MPa at a 10% mesh variation to 82.011 MPa at a 1% mesh variation. In contrast, the generative design showed significant variation in stress values ranging from 1,818 to 2,490 MPa. Based on the mesh independence study, a 1% mesh size was selected because it produced convergent results with lower numerical variation compared to coarser meshes, ensuring reliable stress and displacement predictions without excessive computational cost.

Fusion 360 element analysis simulation results were used to assess the frame structure's ability to withstand loading based on three main parameters: von Mises stress, total deformation, and safety factor. The von Mises stress is used to determine whether the material is approaching its yield point, while deformation indicates the structure's stiffness level during operation. The safety factor is used as an indicator of the safety margin, with a value of ≥ 2 defined as the limit of structural suitability for vehicle frame applications [23]. The results of the Fusion 360 simulation can be seen in the following Figure 6 and Table 8:

The manual design yields a stress value of 103.01 MPa with a safety factor of 2.08 and a strain of 2.6 mm. From a mechanical perspective, this indicates that the applied load is distributed over a relatively large effective cross-sectional area, resulting in moderate stress levels well below the material's yield strength. The relatively low stress and moderate strain indicate that the structure is operating within the elastic range, where stress and strain maintain a linear relationship. The higher mass of this design implies that the excess material contributes to increased stiffness and reduced stress concentration, leading to stable structural behavior [24].

The generative design yields a stress value of 1844.6 MPa, which significantly exceeds the yield strength of AISI 304. This indicates that the structure will undergo plastic deformation or immediate failure under the applied load. This behavior can be explained by the drastic reduction in the effective load-bearing area due to the optimization of the geometry, which becomes flatter. According to stress-strain relationships, a smaller cross-sectional area causes a sharp increase in stress. Additionally, the irregular and highly optimized geometry creates geometric discontinuities, which act as stress concentration zones. The deformation in the optimized geometry is 66.7 mm, which is significantly larger compared to the manual and hybrid design methods. High stress leads to excessive strain. A very low safety factor of 0.11 indicates that the structure is unable to withstand the applied load. However, it is important to note that these extreme stress values may also be influenced by numerical effects such as stress singularities or inadequate geometric constraints, which are common in highly optimized geometries [25].

In the hybrid model, the stress of 82.01 MPa is lower than that of the generative AI and manual models, indicating a good force distribution without localization at specific points. The deformation of 1.6 mm is the lowest among all models, indicating that the hybrid structure has the highest stiffness. The safety factor of 2.62 confirms that this model is safe and capable of withstanding the load. The advantage of this model stems from local reinforcement in critical areas and the restoration of cross-sectional thickness that was previously thin in the generative model, without excessively increasing the mass [26].

The performance differences among the three models are heavily influenced by material distribution strategies. The manual model uses excess material, making it strong and stable but less efficient. Generative models are very lightweight but sacrifice stiffness and durability, leading to structural failure. This behavior is consistent with previous research on generative design, where aggressive mass reduction often results in extremely thin structural elements, causing high stress concentrations and excessive deformation when manufacturing constraints are not adequately considered. The hybrid model is able to balance mass efficiency while maintaining strength and stiffness through targeted local reinforcement, making it the most suitable and implementable design [23]

CONCLUSIONS

Based on the analysis and discussion, it can be concluded that the design approach has a significant influence on the structural performance of the truck wheel mounting frame. The manual design demonstrates good stiffness and strength with a Safety Factor that meets safe values, but the use of evenly distributed material results in a large total mass and reduced efficiency. The AI generative design is capable of significantly reducing mass, but it produces very thin cross-sections, resulting in high stress concentrations, excessive deformation, and a Safety Factor far below the acceptability limit; therefore, this design is deemed structurally unfeasible. The hybrid design combines the strengths of both approaches: it maintains the efficient force flow paths from the generative model while reinforcing critical areas through cross-sectional adjustments. This results in uniform stress distribution, minimal deformation, and a Safety Factor above the safe limit. Therefore, the hybrid design is the most optimal solution to implement, as it offers a balance between mass efficiency, structural strength, and readiness for the manufacturing process in a workshop environment. Similar findings have been reported in studies of hybrid structures, which show that combining the optimized load paths from generative designs with geometric simplification and local reinforcement significantly improves structural reliability and manufacturability.

ACKNOWLEDGEMENTS

DECLARATION OF CONFLICTING INTERESTS

The authors declare that they have no potential conflicts of interest regarding the research, authorship, and/or publication of this article.

FUNDING

The author(s) disclosed receipt of financial support for the research, authorship, and/or publication of this article.

REFERENCES

[1] V. Andries and S. M. Zaharia, “Generative Design, Simulation, and 3D Printing of the Quadcopter Drone Frame,” *Journal of Applied Sciences*, vol. 15, no. 17, 2025, doi: 10.3390/app15179647.

[2] M. Watson, M. Leary, D. Downing, et al., “Generative design of space frames for additive manufacturing technology,” *The International Journal of Advanced Manufacturing Technology*, pp. 4619–4639, 2023, doi: 10.1007/s00170-023-11691-9.

[3] C. Zhu, “Topology Optimization of Finite Periodic Structures with Compliance and Frequency Criteria,” Journal of Metallic Material Research, vol. 07, no. 01, pp. 22–31, 2024.

[4] H. Gu, H. Smith, and J. A. Norato, “Manufacturing-Cost-Driven Topology Optimization of Welded Frame Structures,” *Journal of Mechanical Design*, vol. 145, no. 8, pp. 1–11, 2023, doi: 10.1115/1.4062394.

[5] G. E. Pramono, A. Hidayat, and R. Waluyo, “Perancangan dan Simulasi Desain Rangka Sepeda Motor Listrik Tipe Trellis Menggunakan Finite Element Analysis,” *JTERA (Jurnal Teknol. Rekayasa)*, vol. 5, no. 2, p. 319, 2020, doi: 10.31544/jtera.v5.i2.2020.319-326.

[6] Reza Vannessa Valentina, F. Bisono, and Fais Hamzah, “Perancangan Mesin Loading-Unloading untuk Mesin TruPunch 5000,” *Jurnal Ilmiah Teknik Mesin, Elektro dan Komputer*, vol. 3, no. 2, pp. 311–324, 2023, doi: 10.51903/juritek.v3i2.1801.

[7] I. Dumyati and S. Nurhaji, “Modeling dan Simulasi Finite Element Analysis pada Segitiga T Sepeda Motor Menggunakan Sofware Ansys 2023,” *Quantum Teknik: Jurnal Teknik Mesin Terapan,* vol. 5, no. 1, pp. 26–30, 2023, doi: 10.18196/jqt.v5i1.19012.

[8] Z. Masood, M. Usama, S. Khan, et al., “Generative vs . Non-Generative Models in Engineering Shape Optimization,” *Journal of Marine Science and Engineering*, vol. 12, no. 4, p. 566, 2024, doi: 10.3390/jmse12040566.

[9] Supandi, A. Gamayel, Kasum, et al., “Analisis Cetakan Plastik Injeksi Spesimen Uji Tarik Melalui Simulasi Plastik Injeksi Menggunakan Autodesk Fusion 360,” *Scientific Journal of Mechanical Engineering Kinematika*, vol. 9, no. 1, pp. 46–54, 2024, doi: 10.20527/sjmekinematika.v9i1.278.

[10] J. Waworuntu, A. Halim, and A. P. Irawan, “Analisis Kekokohan Rangka Multi-Speed Pneumatic Linear Transfer System Pick-And-Place Dalam Keadaan Static Menggunakan Metode Fea,” *Jurnal Serina Sains, Teknologi dan Kedokteran*, vol. 2, no. 1, 2024, doi: https://doi.org/10.24912/jsstk.v2i1.33551.

[11] M. Siska and A. Gunawan, “Perancangan Alat Bantu Las Listrik untuk Mengurangi Keluhan Musculoskeletal Disorder Menggunakan Metode Loading on the Upper Body Assesment (LUBA),” *Jurnal Teknik Industri*, vol. 9, no. 3, pp. 212–219, 2019, doi: 10.25105/jti.v9i3.6651.

[12] H. R. P. Yongki and R. Fitriani, “Analisis Perancangan Alat Bantu pada Mesin Slitter Menggunakan Metode AHOQ,” *Jurnal Teknik Industri*, vol. 12, no. 2, pp. 117–124, 2022, doi: 10.25105/jti.v12i2.15634.

[13] Z. Wang, Y. Zhang, M. Orquera, et al., “ScienceDirect A New Hybrid Generative Design Method for Functional & Lightweight Structure Generation in Additive Manufacturing,” *Procedia CIRP*, vol. 119, pp. 66–71, 2023, doi: 10.1016/j.procir.2023.02.127.

[14] G. M. Azanaw, “Blending Data-Driven Surrogates with Physics - Based Topology Optimization : A Critical Review of Machine Learning - Accelerated Design in Fibre - Reinforced Polymer and Concrete Structures,” American Journal of Science, Engineering and Technology, vol. 10, no. 3, pp. 80–93, 2025.

[15] M. Masy’ari, E. Karyadi, and R. Rusadi, “Penerapan teknik digital prototyping dalam perancangan dan pembuatan model produk berbasis generative design,” *Turbo J. Progr. Stud. Tek. Mesin*, vol. 9, no. 1, 2020, doi: 10.24127/trb.v9i1.1171.

[16] E. O. Ogunnowo, M. A. Adewoyin, J. E. Fiemotongha, et al., “A Conceptual Framework for Reliability-Centered Design of Mechanical Components Using FEA and DFMEA Integration,” *J. Front. Multidiscip. Res.*, vol. 4, no. 1, pp. 342–361, 2023, doi: 10.54660/.jfmr.2023.4.1.342-361.

[17] J. Bright, R. Suryaprakash, S. Akash, et al., “Optimization of quadcopter frame using generative design and comparison with DJI F450 drone frame,” *IOP Conf. Ser. Mater.*

Sci. Eng., vol. 1012, no. 1, p. 012019, 2021, doi: 10.1088/1757-899x/1012/1/012019.

[18] K-Y. Li, C-K. Huang, Q-W. Chen, et al., “Generative AI and CAD automation for diverse and novel mechanical component designs under data constraints,” *Discover Applied Sciences,* no. 57, 2025, doi: 10.1007/s42452-025-06833-5.

[19] N. Razavi, M. R. Ayatollahi, and F. Berto, “ScienceDirect ScienceDirect Assessment of mixed mode fatigue crack growth under biaxial loading using an iterative technique,” *PROSTR*, vol. 26, no. 2019, pp. 240–245, 2020, doi: 10.1016/j.prostr.2020.06.029.

[20] S. R. Gunjawate and S. B. Solapur, “Structural Analysis and Topology Optimization of Leaf Spring Bracket,” *Int. J. Eng. Res. Technol.*, vol. 9, no. 07, pp. 1488–1494, 2020.

[21] J. Wang, N. Wei, and Q. Xu, “Research on Mechanical Performance Analysis and Topology Optimization Design of New Automobile Frame Structures,” vol. 9, pp. 15–22, 2024, doi: 10.23977/jemm.2024.090303.

[22] M. Çallı, E. İ. Albak, and F. Öztürk, “Prediction and Optimization of the Design and Process Parameters of a Hybrid DED Product Using Artificial Intelligence,” *Journal of Applied Sciences*, vol. 12, no. 10, p. 5027, 2022, doi: 10.3390/app12105027.

[23] O. Zamzam, M. Abdelaziz, T. Elnady, et al., “Structural performance evaluation of electric vehicle chassis under static and dynamic loads,” *Research Square (Preprint)*, 2025, doi: 10.21203/rs.3.rs-4258581/v1.

[24] F. Budi, “ScienceDirect ScienceDirect Structural Assessment of Ladder Frame Chassis using FE Analysis : A Designed Construction referring to Ford AC Cobra,” *Procedia Struct. Integr.*, vol. 33, pp. 35–42, 2021, doi: 10.1016/j.prostr.2021.10.006.

[25] A. K. Faizin, R. Rizaldi, W. A. S. Putra, et al., “Structural Analysis of Chassis Frame of a Prototype Car : A Finite Element Method,” *Proceedings of NSTP*, vol. 2023, pp. 252–256, 2023, doi: 10.11594/nstp.2023.3636.

[26] Y. Liu, C. Liu, X. Gao, et al., “Multiphysics Finite Element Analysis and Optimization of Load-Bearing Frame for Pure Electric SUVs,” *Symmetry*, vol. 17, p. 1143, 2025, doi: 10.3390/sym17071143.

[27] T. Tuswan, M. Andrian, W. Amiruddin, et al., “Design Improvement Using Topology Optimization for the Structural Frame Design of a 40 Ft LNG ISO Container Tank,” *Designs*, vol. 8, no. 2, p. 21, 2024, doi: 10.3390/designs8020021.