

ISSN (online): 2581-3048 Volume 9, Issue 4, pp 261-268, April-2025 https://doi.org/10.47001/IRJIET/2025.904036

Evaluation of Front Impact Response on the Front Tubular Frame of a Maung 4x4 Tactical Vehicle: A Design Improvement Perspective

¹*Norman Iskandar, ²Sulardjaka, ³Faza Hadiyan

^{1,2,3}Mechanical Engineering Department, Diponegoro University, Semarang, Indonesia *Corresponding Author's E-mail: <u>normaniskandar@lecturer.undip.ac.id</u>

Abstract - The vehicle chassis functions as a structural frame that supports loads such as engines, passengers, and cargo while maintaining the rigidity of the vehicle. Combat vehicles and cruise vehicles usually add additional frames to the front and rear areas of the vehicle as additional protection for the vehicle. This study analyzes the strength of the additional frame design installed in the front area of the Maung 4X4 tactical vehicle. The material used is STKM 13-B pipe. Two design models were tested using the Finite Element Method (FEM) approach through Solidworks simulation. The results show that the second design (modified) has a higher safety factor than the initial design.

Keywords: Tubular Frame, STKM 13B, Safety Factor, FEM, Solidworks.

I. INTRODUCTION

The Maung 4x4 Tactical Vehicle, developed by PT Pindad, is designed for close combat support, mobility, and all-terrain operations. It features a 6-speed manual transmission, a top safe speed of 120 km/h, and weighs between 1,200–1,300 kg. With dimensions of 4,931 mm (L) \times 1,640 mm (W) \times 1,820 mm (H), it offers a range of up to 800 km and agile, reliable maneuverability. The Maung can be equipped with a 7.62 mm weapon bracket, SS2-V4 console, GPS, tracker, and other mission-specific gear. Its appearance is shown in Figure 1.



Figure 1: Maung 4x4 tactical vehicle

One of the key components of the Maung 4x4 Tactical Vehicle is its chassis, which serves as the structural support for the vehicle's weight, including the engine, passengers, and cargo. Typically made from steel, the chassis provides rigidity and prevents bending [1]. During manufacturing, the vehicle body is moulded to fit this frame. Chassis materials—usually metal or composites—must possess high strength to sustain structural loads. Although the Maung's chassis design is already robust, an additional front protective frame is recommended. A tubular frame is ideal for this purpose, offering excellent torsional stiffness, heavy load resistance, and impact protection. The area designated for this enhancement is shown in Figure 2.



Figure 2: Frame addition description

The Maung 4x4 Tactical Vehicle has been enhanced with a tubular space frame added to the front, extending from the lower front end to the sides, and an additional frame along the side, connecting the front and rear. These design improvements, based on observational analysis, offer both functional and aesthetic benefits.

The frame used is a tubular space frame—an assembly of steel pipes shaped to match the vehicle's construction. Its strength depends largely on the quality of the welds. This type of frame provides excellent yield strength, particularly in torsional stiffness, heavy load resistance, and impact absorption [2].

The frame utilized in this study is constructed from STKM 13-B, a low-carbon steel known for its toughness and potential for enhanced mechanical strength. This material is widely applied across various industries, including machinery,

International Research Journal of Innovations in Engineering and Technology (IRJIET)



automotive components, bicycles, furniture, tools, and other structural parts [3-6].

This research employs the Finite Element Analysis (FEA) method. The fundamental principle of FEA involves discretizing a continuous structure into a collection of small interconnected elements joined at nodal points located at the element boundaries. Each element comprises several nodes, and each node possesses multiple degrees of freedom [7-10].

The analysis is conducted using SolidWorks 2024, a widely adopted computer-aided design (CAD) software used in product, mechanical, and structural engineering design. SolidWorks 2024 is equipped with simulation tools capable of evaluating structural responses such as stress, strain, and thermal effects. As a parametric, feature-based modelling software, it allows modifications to geometric relationships and features even after the model has been fully developed, significantly facilitating the design and optimization process [11,12].

The primary objective of this study is to perform a structural analysis using SolidWorks 2024 to obtain key mechanical indicators including displacement, stress, and safety factor. Furthermore, the research aims to compare the displacement values between the original and the optimized frame designs.

II. METHODOLOGY

This study uses the Finite Element Analysis approach by utilizing Solidworks 2024 software as a tool. The study of how much displacement, Safety of Factor, and stress values occur in the frame models will be the main study point.

2.1 Troubleshooting Procedure

In SolidWorks 2024, analysis using Finite Element Analysis (FEA) involves three main stages: pre-processing, processing, and post-processing. In the pre-processing stage, the user prepares the model by creating 3D geometry and specifying materials, loads, and boundary conditions. This stage also includes meshing, which divides the model into smaller elements for more detailed analysis. The processing stage is when the simulation is based on the input from preprocessing. The simulation is used to solve the equations that govern the physical behavior of the model, generating data related to stress distribution, displacement, and safety factors. In the post-processing stage, the results of the calculations are analyzed and visualized.

To start the static loading analysis in Solidworks software, click "Simulation">"New Study" and select "Static".

Volume 9, Issue 4, pp 261-268, April-2025

ISSN (online): 2581-3048

https://doi.org/10.47001/IRJIET/2025.904036

Study	0	D
✓ × ×		
Message	^	
Study stresses, displacements, strains and factor of safety for components with linear material		
Name	^	
Static Design Actual		
General Simulation	^	
💸 Static		

Figure 3: Create new simulation

In this section, material selection will be carried out to determine the material specifications of the front tubular frame for analysis. The following are the steps to apply material in Solidworks simulation. Click "Apply Material" > "Material Dollies" and then select the material "STKM 13-B" > "Apply" > "Ok".

Properties	Tables	& Curves	Appearance	CrossHatch	Custom	Application Data	•
Material p Materials to a custo	oropert in the om libra	ies default lib ary to edit i	rary can not b t.	e edited. You	must first (copy the material	
Model Typ	oe:	Linear Ela	stic Isotropic	~ (Save mo	del type in library	
Units:		SI - N/mm	^2 (MPa)	~			
Category:		Fame Tub	ullar Depan				
Name:		STKM 13-	В				
Default fa	ilure	Max von M	Mises Stress	~			
Description:	on:	STKM 13-	в				
Source:							
Sustainab	oility:	Undefine	d		s	elect	
Property			Value		Units	5	
Elastic Mo	dulus		215000		N/mr	n^2	- 1
Poisson's P	Ratio		0.29		N/A		
Shear Mod	lulus		82000		N/mr	n^2	11.
Mass Dens	ity		7900		kg/m	1^3	11.
Tensile Str	ength		30.5		N/mr	n^2	11.
Compressi	ve Stre	ngth	30.5		N/mr	n^2	
Yield Stren	gth		30.5		N/mr	n^2	
Thermal Ex	pansio	n Coefficie	nt 1.38e-05		/K		
Thermal Co	onduct	ivity	25		W/(m	η·K)	
			Save	Config	Apply	Close Help	•

Figure 4: Material properties

In this section, we will select the fulcrum when given a load. Press "Fixture">"Fixed Geometry" and then select the joint that should be the fulcrum, choosing the joint at the end of the frame because that part is the fulcrum of the frame when given a load. After determining the joint that becomes the fixed geometry, then click "Ok".

International Research Journal of Innovations in Engineering and Technology (IRJIET)



	Fixture	?
✓ :	× +	
Exam	ple	^
Stand	lard(Fixed Geometry)	^
Ľ	Fixed Geometry	
Ľ	Immovable (No translation)	
	Use Reference Geometry	
₩	Joint<40, 1>	
	Joint<91, 1>	
	Joint<73, 1>	

Figure 5: Fixed geometry

In this section, we will add the force that will be received at the front of the frame by determining the position using the front plane. The force that will be received by the frame is with the variations that have been provided. The following are the steps to add the force that will be received by the frame.

The first step is to click "External Loads">"Force". Next is to determine the position that will be subjected to the load and enter the force value to be received by the frame, determine the position using the front plane and given a force variation. Next, to set the loading point, namely on the part of the frame that has a curve and on the part of the frame that has a meeting point on the other frame connection which can be called a joint, set it in the Z direction.



Figure 6: Determining the external load and setting the impact position

ISSN (online): 2581-3048 Volume 9, Issue 4, pp 261-268, April-2025

https://doi.org/10.47001/IRJIET/2025.904036

In this section the frame will be divided into several small objects with a certain shape and connected between points and nodes. Here are the steps to create meshing in the simulation using Solidworks application. Press "Mesh", then click "Create Mesh".

S N	/lesh		
• f	\$	Mesh Advisor	
8 .	8 ₀	Simplify Model for Meshing	
	6	C <u>r</u> eate Mesh	
		Mesh and <u>R</u> un	
		Apply Mesh Control	
		De <u>t</u> ails	
	4	Create <u>M</u> esh Quality Plot	
		Collapse Tree Items	
	▼ [Mesh Mesh Advisor Simplify Model for Meshing Create Mesh Mesh and Run Apply Mesh Control Details Create Mesh Quality Plot Create Mesh Quality Plot Create Mesh Quality Plot

Figure 7: Meshing

After all the steps above have been carried out, the next step is to simulate and see the results that will be obtained. The trick is to click "Run Study".



Figure 8: Run study

2.2 Calculation Formula

Basically, stress can be defined as the amount of force acting on a unit area. Voltage or stress can be found by the force divided by the cross-sectional area, Voltage is denoted by σ , Force (N) is denoted by F, and Area (m²) is denoted by A. The stress value can be found by Equation 1.

Stress
$$(\sigma) = \frac{F}{A}$$
 (1)

Strain can be found by the total length increase divided by the initial length, Strain is denoted by ε , total length increase is denoted by δ , and the initial length is denoted by *L*. Strain values can be found with Equation 2.

Strain
$$(\varepsilon) = \frac{\delta}{L}$$
 (2)

Structure-failure, the actual strength of a material must exceed the required strength. The ratio of the actual strength to the required strength is called the factor of safety. Factor of safety can be found with the maximum permissible stress value ($\sigma_{allowable}$) divided by the value of the stress imposed on the structure ($\sigma_{applied}$). The factor of safety value can be found by Equation 3.



(3)

This impact analysis can be simplified with static analysis using the relationship of the forces acting at a short time and the change in momentum of the vehicle. Time of vehicle impact can be found by Vehicle mass times Vehicle speed minus After collision Vehicle speed. The force on the Front Crash Impact can be found by the Time of vehicle impact divided by the vehicle stopped after collision. Force on Front Crash Impact (N) is denoted by F, Time of vehicle collision (s) is denoted by ΔP , Vehicle mass (Kg) is denoted by m, Vehicle speed after collision (s) is denoted by V_{stop}, Vehicle speed (m/s) is denoted by V, the time the vehicle stops after collision (s) is denoted by t. The value of Time of vehicle collision and Force on Front Crash Impact can be found by Equation 4 and Equation 5.

$$\Delta P = m x (V_{stop} - V)$$
(4)
$$F = \frac{\Delta P}{t}$$
(5)

2.3 Validation

Z

The test validation process uses a case study in the book Strength Mechanics of Materials page 11 by Amrinsyah, 2013) [13]. In this case study, a 2m long perforated cylinder has an outer diameter of 50 mm and an inner diameter of 30 mm. If the cylinder bears a load of 25 kN, find the stress in the cylinder. Also find the deformation that occurs in the cylinder if the modulus of elasticity of the cylinder material is 100 GPa. It is known that the length of the cylinder (1) is 2 meters or 2000 mm, with an outer diameter (D) of 50 mm and an inner diameter (d) of 30 mm. The applied load (P) is 25 kN or 25 x 103 N, and the modulus of elasticity (E) of the cylinder material is 100 GPa or 100 x 103 N/mm². The results of manual calculation analysis obtained with the stress that occurs in the cylinder is 19.9 MPa and the deformation that occurs is 0.4 mm.

To validate the analysis, namely by comparing the results of the case study calculations with the results of the finite element method simulation analysis in the Solidworks 2024 application. The following simulation results are shown in Figure 9.



(a)



ISSN (online): 2581-3048

Volume 9, Issue 4, pp 261-268, April-2025 https://doi.org/10.47001/IRJIET/2025.904036

Figure 9: Simulated Stress Value Results (a) and Simulated Deformation Value Results (b)

The Solidworks simulation results with a maximum stress of 20,350 MPa and a displacement of 0.4 mm show a very good agreement with more than 2% of the simulation results with manual calculations. This shows that the finite element method analysis used is accurate enough to predict the stress in tubular pipes subjected to static loading.

III. RESULTS AND DISCUSSION

3.1 Data

The research data needed to perform the analysis are the actual design of the front tubular frame, the design improvement of the front tubular frame, the material characteristics of STKM 13-B, and the value of the force variation that will be given to the simulation. The actual design and design improvement of the front tubular frame of the Maung 4x4 tactical vehicle and the addition of support to the tubular frame. Improvements to the frame with the addition of a tubular frame at the front connected from the lower end to the side end of the front, this improvement has the aim of strengthening the front structure and as a buffer so that the front frame is sturdy. The second addition is on the side, the addition is located on the front to rear link, this has the aim that in the event of stress or displacement the frame will withstand a significant tilt displacement. The following is the actual design and design improvement shown in Figure 9.



(a)

ISSN (online): 2581-3048 Volume 9, Issue 4, pp 261-268, April-2025 https://doi.org/10.47001/IRJIET/2025.904036



Figure 10: Isometric view of actual design (a), and isometric view of improved design (b)

Characteristics or material properties used for simulation in Solidworks 2024 software. The following are the Material Characteristics of STKM 13-B shown in Table 1.

Property	Value	Units
Elastic Modulus	215000	N/mm^2
Poisson's Ratio	0.29	N/A
Shear Modulus	82000	N/mm^2
Mass Density	7900	kg/m^3
Tensile Strength	30.5	N/mm^2
Compressive Strength	30.5	N/mm^2
Yield Strength	30.5	N/mm^2
Thermal Expansion Coefficient	1.38e-05	/К
Thermal Conductivity	25	W/(m·K)
Specific Heat	465	J/(kg·K)

Table 1: Material characteristics of STKM 13-B

Front Crash Impact is assumed that the vehicle is traveling at varying speeds of 30 Km/h, 60 Km/h, 90 Km/h and hits a stationary object such as a parapet or stationary object. Collision conditions with these speeds are assumed that the vehicle stops after 0.2 seconds from the collision. Finite element modeling (FEM) simulation based on the actual condition of the vehicle when driving is used to find the loading value in each condition. The amount of loading in the finite element modeling based on the actual condition approach of the vehicle when driving is used to find the front impact loading value, each speed and force variation is contained in Table 2.

Table 2: Speed a	and Force	values
------------------	-----------	--------

Speed (km/h)	Force (N)
30	49800
60	99600
90	150000

3.2 Result Analysis and Discussion Chart

The results of the analysis on both designs with static simulation of the chassis frame with Solidworks 2024 software produce stress, displacement, and factor of safety with the material used in both designs is STKM 13-B.

In the Finite Element Method (FEM) analysis using the Solidworks 2024 application, the stages carried out to find stress, displacement, and safety factors can be divided into three parts, namely pre-processing, processing, and postprocessing. Pre-processing involves the initial stages such as drawing and modeling the geometry to be analyzed. This stage ensures that the geometry created is accurate and in accordance with the actual conditions. After the geometry has been created, the processing stage is carried out. In this stage, perform numerical analysis of the structural model that has been made. This analysis includes the use of methods such as the Finite Element Method (FEM) to predict stress and displacement in the structure. The post-processing stage is carried out after the analysis is complete. In this stage, it displays the results of the analysis that has been carried out, such as stress, displacement, and safety factor. The following is a view of the applied load and a view of the meshing geometry shown in Figure 11.



Figure 11: View of applied load (a) and View of meshing geometry (b)

International Research Journal of Innovations in Engineering and Technology (IRJIET)



ISSN (online): 2581-3048

Volume 9, Issue 4, pp 261-268, April-2025

https://doi.org/10.47001/IRJIET/2025.904036

Display of actual design simulation results that obtain stress, displacement, and factor of safety values. The following is a display of the results of the stress, displacement and factor of safety simulation on the actual design shown in Figure 12 to Figure 14.



Figure 12: Display of stress simulation results on actual design



Figure 13: Display of displacement simulation results on actual design



Figure 14: Display of factor of safety simulation results on actual design

Display of improvement design simulation results that obtain stress, displacement, and factor of safety values. The following is a display of the results of the stress, displacement and factor of safety simulation on the actual design shown in Figure 15 to Figure 17.



Figure 15: Display of stress simulation results on design Improvement



Figure 16: Display of displacement simulation results on design improvement



Figure 17: Display of factor of safety simulation results on design improvement

From the analysis results, we can observe the comparison of stress, displacement, and factor of safety results against the speed variation between the actual design and design improvement. The following are the simulation results on the actual design and design improvement in Table 3 and Table 4.



 Table 3: Simulation Results on Actual Design

Design Actual			
Speed(km/h)	Max Stress (MPa)	displacement (mm)	Factor of Safety
30	57.934	4.944	0,00052
60	115.869	9.888	0,00026
90	174.502	14.892	0,00017

Table 4: Simulation Results on Design Improvement

Design Improvement			
Speed(km/h)	Max Stress (MPa)	displacement (mm)	Factor of Safety
30	26.367	1.045	0,020
60	52.735	2.091	0,010
90	79.420	3.150	0,007

Based on the results of stress analysis on the frame, the actual design has a higher stress level than the improvement design. This shows that the actual design has the potential to experience damage or structural failure faster than the improvement design. The improvement design is more recommended for the chassis frame because it has a lower stress level and less potential damage. The graphical results can be seen in Figure 18.



Figure 18: Stress result graph of the two designs

Based on the analysis results, the actual design of the frame produces a higher displacement than the Improvement design. This shows that the actual design is more prone to deformation when given a load. Stress on the improvement design is lower, indicating that the design has a higher resistance to the load acting on it. The graphical results can be seen in Figure 19.

ISSN (online): 2581-3048

Volume 9, Issue 4, pp 261-268, April-2025

https://doi.org/10.47001/IRJIET/2025.904036



Figure 19: Displacement results from both designs

Based on the analysis, the actual design of the frame results in a lower factor of safety compared to the Improvement design. This indicates that the actual design has a smaller safety margin and is more at risk of failure. Stress in the improvement design is higher, but the factor of safety is higher. This indicates that the improved design has a higher ability to withstand greater loads without failing. The graphical results can be seen in Figure 20.



Figure 20: Safety of factor results from both designs

IV.CONCLUSION

Based on the analysis of stress, displacement, and factor of safety, it can be concluded that the modified and improved frame design demonstrates clear advantages over the original tubular frame. These improvements were achieved through the addition of relatively simple components, strategically positioned to produce a significant impact on the overall structural performance. This highlights the importance of thoughtful design modifications in enhancing mechanical efficiency without the need for complex interventions.



ISSN (online): 2581-3048

Volume 9, Issue 4, pp 261-268, April-2025 https://doi.org/10.47001/IRJIET/2025.904036

REFERENCES

- Fadila, Ary, 2012, Analisis Simulasi Struktur Chassis Mobil Mesin, USU Berbahan Besi Struktur Terhadap Beban Statik Dengan Menggunakan Perangkat Lunak Ansys 14, (ISSN 2338-1035), Universitas Sumatera Utara.
- [2] Keith J. Wakeham, 2009, Introduction to Chassis Design, Newfoundland and Labrador:Memorial University.
- [3] Simamora J, Gamayel A, Indra IB, Zaenudin M. Pengaruh pengaturan voltase terhadap kekuatan tarik pada gas metal arc welding antara pipa STKM 13B dan pelat SPH 440. Jurnal Teknologi Terapan Mesin. 2023;7(2):67–74.
- [4] Okokpujie IP, Tartibu LK. Investigating the effect of hot and cold working on the microstructure and mechanical properties of low-carbon-steel for sustainable manufacturing. Journal of Applied Mechanics and Materials. 2024;9(1):1–12.
- [5] Karyanik K. Effect of treatment of low carbon steel tensile test specimen on tensile strength for tractor wheel axle. Jurnal Pengabdian dan Teknologi Teknik. 2023;5(1):45–50.
- [6] Lee J, Kim Y, Park J, Choi H, Lee S. Effect of microstructure on low-temperature fracture toughness of a submerged-arc-welded low-carbon and low-alloy steel plate. Metals. 2021;11(11):1839.

- [7] Zienkiewicz OC, Taylor RL, Zhu JZ. The Finite Element Method: Its Basis and Fundamentals. 7th ed. Elsevier; 2013.
- [8] Cook RD, Malkus DS, Plesha ME, Witt RJ. Concepts and Applications of Finite Element Analysis. 4th ed. Wiley; 2002.
- [9] Touze C, Thomas O, Berlioz A. Finite element computation of nonlinear modes and frequency response of geometrically exact beam structures. J Sound Vib. 2023;548:117534. doi:10.1016/j.jsv.2022.117534.
- [10] Yunus SM, Saigal S, Cook RD. On improved hybrid finite elements with rotational degrees of freedom. Int J Numer Methods Eng. 1989;28(4):785–800. doi:10.1002/nme.162028040.
- [11] Camba JD, Contero M, Company P. Parametric CAD modeling: An analysis of strategies for design reusability. Comput Aided Des. 2016;74:1-17. doi:10.1016/j.cad.2016.01.003.
- [12] Agus Adi, I. N., Dantes, K. R., & Nugraha, I. N. P. (2018). Analisis Tegangan Statik Pada Rancangan Frame Mobil Listrik Ganesha Sakti (Gaski) Menggunakan Software Solidworks 2014. Jurnal Pendidikan Teknik Mesin Undiksha, 6(2), 113. Https://Doi.Org/10.23887/Jjtm.V6i2.13046.
- [13] Amrinsyah, M.M. (2013). Mekanika Kekuatan Material. Program Studi Teknik mesin, Fakultas Teknik, Universitas Medan Area.

Citation of this Article:

Norman Iskandar, Sulardjaka, & Faza Hadiyan. (2025). Evaluation of Front Impact Response on the Front Tubular Frame of a Maung 4x4 Tactical Vehicle: A Design Improvement Perspective. *International Research Journal of Innovations in Engineering and Technology - IRJIET*, 9(4), 261-268. Article DOI <u>https://doi.org/10.47001/IRJIET/2025.904036</u>
