Journal of Railway Transportation and Technology
Vol. 4 No. 1 (2025) 44-51
eSSN: 2830-6880

Structural Modeling and Analysis of Flat Wagon Underframe Based on Finite Element Method

Ilham Satrio Utomo[1], Edi Nyoto Marsusiadi[1], Sulis Tri Wibowo[1]

[1], Railway Mechanic Technology
Tirta Raya Street, Madiun, 63132, INDONESIA

Article Info

Article history:

Received 2025-05-21
Revised 2025-06-04
Accepted 2025-06-10

Keywords:

Flat Wagon
Ansys
Finite Element Method

ABSTRACT

This research analyzes the structural strength of the 50-foot Flat Wagon underframe using the finite element method. Modeling was done in Autodesk Inventor and load simulation using ANSYS Workbench with a mesh size of 25 mm. The analysis refers to the UIC 571-4 and EN 12663 standards. The results showed that the maximum stress of 347.56 MPa and maximum deformation of 15.305 mm occurred under combined loading. This value is still below the yield strength of the material (355 MPa), but exceeds 75% of the allowable stress according to KM No. 43 Year 2010 (266.25 MPa). The highest safety factor is 1.82 (compression loading), the lowest is 0.76 (combination loading). This simulation shows the need for design optimization to meet national safety standards..

Corresponding Author:

Ilham Satrio Utomo
Department of Mechanical Engineering, Indonesian Railway Polytechnic
Jl. Tirta Raya, Pojok, Nambangan Lor, Manguharjo, Madiun, Jawa Timur 63161, Indonesia
Email: kontak@ppi.ac.id

1. INTRODUCTION

Railway transportation is a crucial mode of transportation globally for transporting passengers and goods. In Indonesia, the contribution of rail-based transportation is still lower than land transportation. To increase the volume of freight transportation, the operation of double-stacked trains is one way to increase the amount of cargo transported. Flat carriage is one type of carriage used to transport logistics in the form of containers.

The design of the train structure refers to the applicable technical regulations. Railway component structures, such as flat car frames, are exposed to complex loading conditions in the real world. These loads include static loads and dynamic loads. In addition, the structure is also subjected to repetitive loading that can lead to fatigue failure. Determining the stress distribution and vertical deformation due to dynamic loads is important. Therefore, structural integrity, strength, impact resistance, safety factor, stress, and deformation must be considered in the design.

Šťastniak made a study to assess the strength conditions of a newly developed railway bogie frame structure and proved that the new design meets the strength requirements through calculations and prototype testing [1]. Analytical and numerical analysis was also used to show that the interference size affects the contact pressure distribution at the press fit joint of the railway wheel axle, with larger interference increasing the contact pressure allowing higher torque and load transfer capability [2]. In addition to this, FEM was also used to evaluate the fatigue design of electric railroad bogie frames to, static testing, full-scale fatigue testing, and testing on rail tracks. The study showed that the bogie frame has adequate strength against static and fatigue loads [3].

Simulation results using ANSYS and MATLAB show that the maximum dynamic response of bridges or vehicles does not always occur under certain working conditions, and the interaction of the combined load of trains and vehicles is nonlinear, so it cannot be represented as a linear superposition of each load [4]. Pradhana concluded that the greater the initial crack length value on the UIC 54 rail with the same crack height, the greater the mode 1 stress intensity factor (SIF) and J-Integral values that occur due to repeated loading [5]. Analysis of Box Girder using ANSYS software-based finite element analysis to evaluate the static response of the bridge under various load combinations of Indian railways [6]. Through 3D finite element analysis and RING software, Costa found that infill materials significantly affected the failure modes of stone arch bridges, especially in relation to the structural response of the spandrel walls [7]. Vibration analysis using analytical and finite element models showed that damage to the third rail can cause low-frequency resonance (<10 Hz), which increases deflection and contact force of the collector shoe, making it useful for predictive maintenance [8].

Structural simulations using ANSYS showed that a 1435 mm track width resulted in lower deformations, stresses and strains, and a longer service life than a 1067 mm track under double-stack train loads. Operating speed also has a significant effect on dynamic loads and service life, making the selection of track width and maximum speed key factors in track design for these loads [9]. Structural and fatigue analysis of railway vehicle axles using CATIA and ANSYS showed that Model A had lower maximum deformation and stress than Model B, thus demonstrating durability [10].

Although extensive FEM studies have been conducted on various railway components such as bogies and axles that are part of flat cars, as well as on railroad and bridge structures that interact with the cars, specific analysis using FEM on flat car frames, especially in the context of transporting containers or double-stacked loads, is important to ensure their integrity and safety under complex operational conditions. This analysis can help identify critical areas and evaluate the structural behavior of the flat car frame against the acting loads.

Therefore, this study aims to conduct a structural analysis of a railway flat car frame using the Finite Element Method to evaluate the stress, strain, and deformation responses under loading scenarios, and identify critical areas for design or maintenance improvements.

2. RESEARCH METHOD

This research uses a numerical approach based on finite element simulation (FEM) with ANSYS Workbench software. The steps of simulation implementation are divided into several main stages, namely the creation of geometry models, determination of material properties, mesh creation, application of loading and constraints, solving process, and analysis of results.

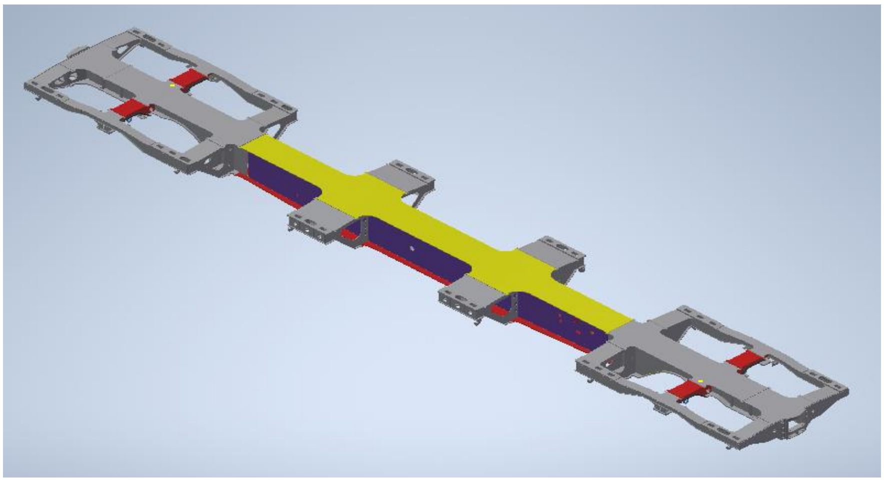

Figure 1. Modeling Underframe

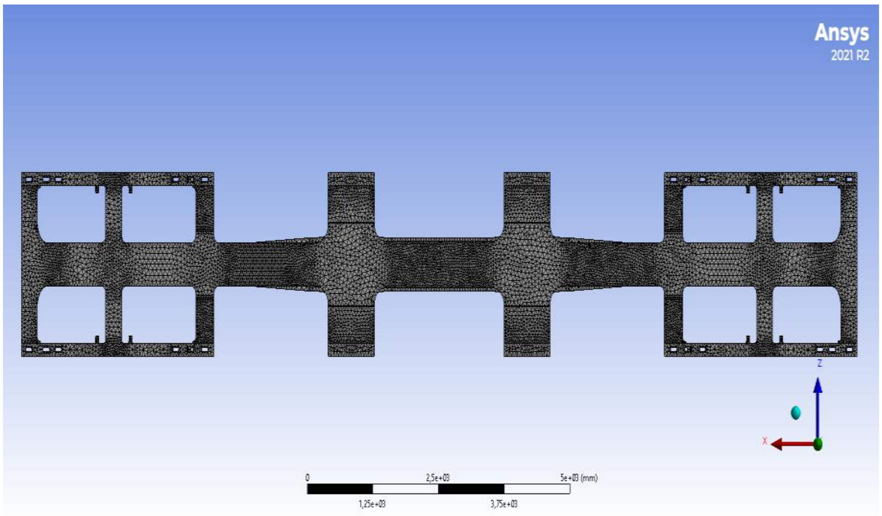

Figure 2. Meshing

Result is the result obtained through simulation using finite element-based software. The result is displayed as a visual display of the force distribution on the simulated object structure. This visual display is in the form of outlines with different colors, where each color defines a different value. The red color in the simulation results indicates that the maximum output value is on the contour, while the red, orange, yellow, green and blue sequential color contours indicate that the output value obtained is decreasing. The results obtained are in the form of maximum stress, maximum strain and safety factor. These results will be validated to determine whether the simulation meets the acceptability criteria.

The simulation is carried out by selecting the location of the loading and the location of the fulcrum, then given input data in the form of the amount of load force applied. In this study, the load is applied evenly on the top surface of the underframe vertically with the maximum load. There are types of loading carried out, namely vertical, compression and combination loading.

Figure 3 show the vertical loading can be assumed under evenly distributed load conditions with a dynamic coefficient of 1.3 as required by KM. 43 of 2010 concerning standard technical specifications of wagons.

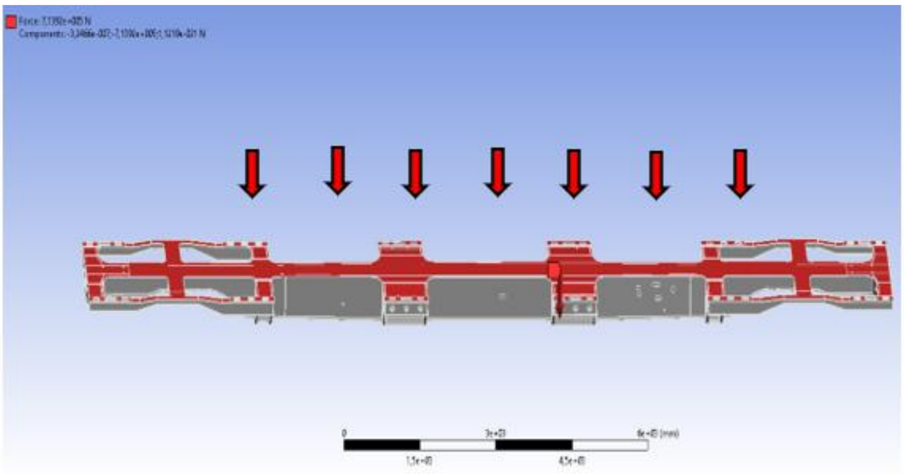

Figure 3. Vertical Loading Location Determination

Figure 4 show the compression load is a load in the longitudinal direction imposed on the coupling device with a weight of 100 tons.

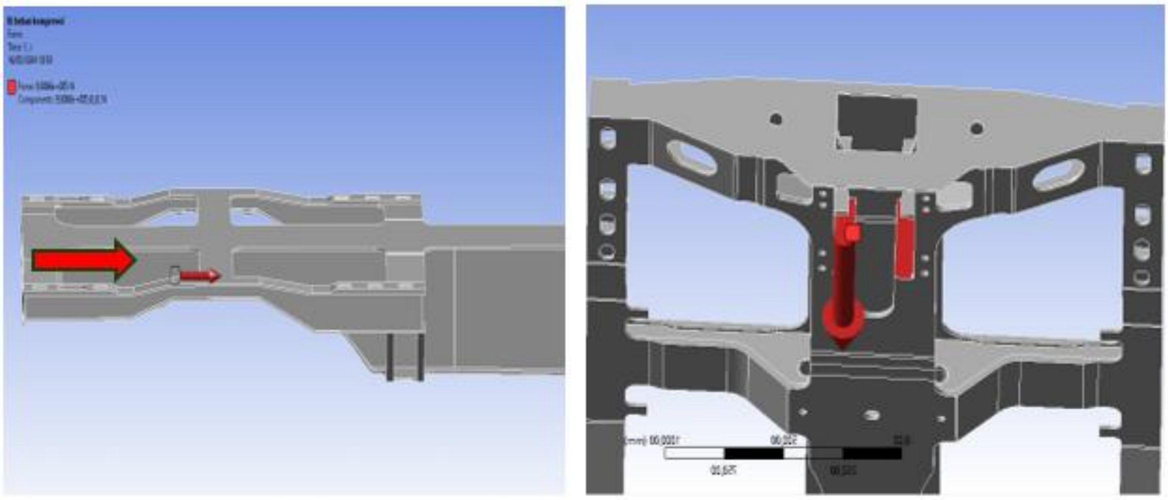

Figure 4. Compression Load Determination

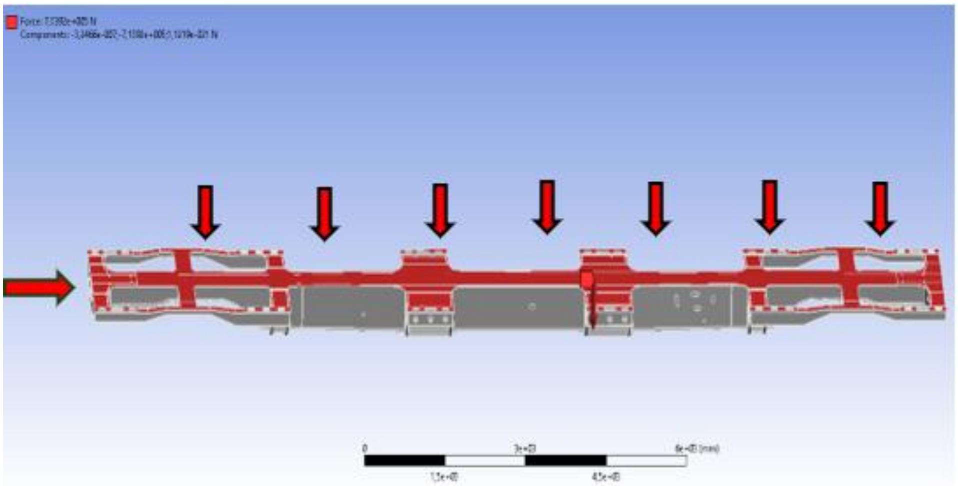

Figure 5. Combination Loads

Figure 5 show the combination loads refer to situations where several different types of loads, vertical and compression loads are applied to a structure simultaneously. These load combinations need to be analyzed to ensure that the structure can withstand all the loads acting at the same time without suffering damage or failure.

3. RESULTS AND DISCUSSION

Simulation is a technique used to present an object as its real condition, this simulation is used to study or analyze the object being observed.

3.1. Vertical Static Loading

When simulating a vertical static loading of 56000 kg. produces a maximum stress (von mises) of 152.81 Mpa with the loading considered evenly distributed, of course it is still very safe from the material allowance. The maximum stress point is located at the bottom of the headstock next to the fixed support.

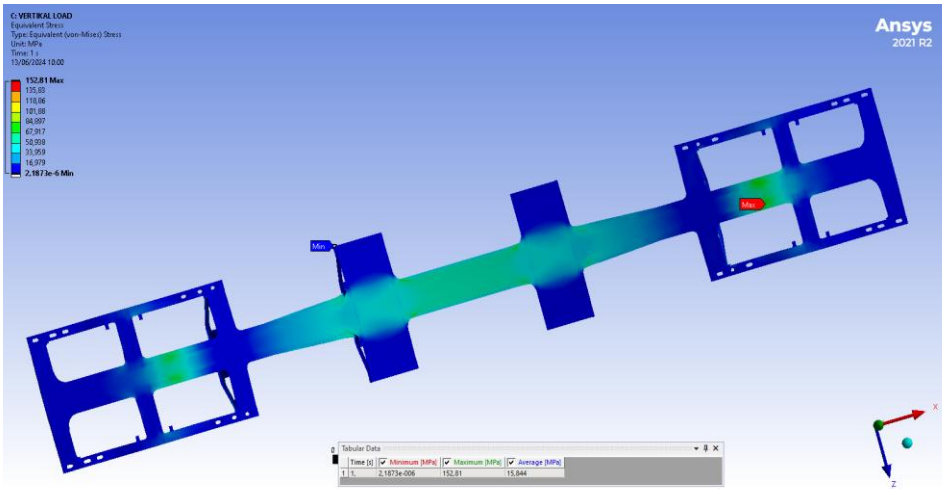

Figure 6. **Vertical Static Loading**

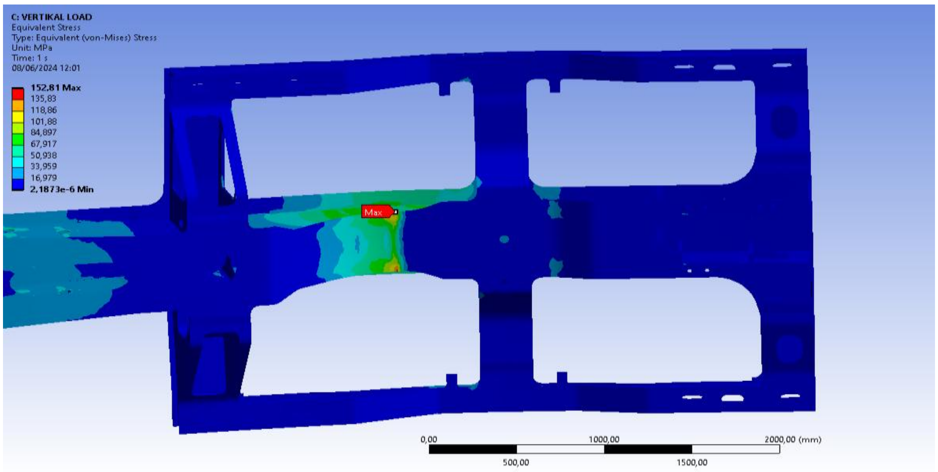

Figure 7 Critical Stress Point

Figure 6 show the vertical static loading simulation of 56,000 kg resulted in a maximum Von Mises stress of 152.81 MPa, assuming the load was evenly distributed. This stress value is still below the material allowable stress limit, so the structure is declared safe. The maximum stress concentration point was identified at the bottom headstock area on the fixed support side.

Based on Figure 4.8, the simulation results with a maximum loading of 56 tons acting in the vertical direction show that the maximum stress occurs in the center pivot area, which coincides with the position of the fixed support.

3.2. Compression Loading

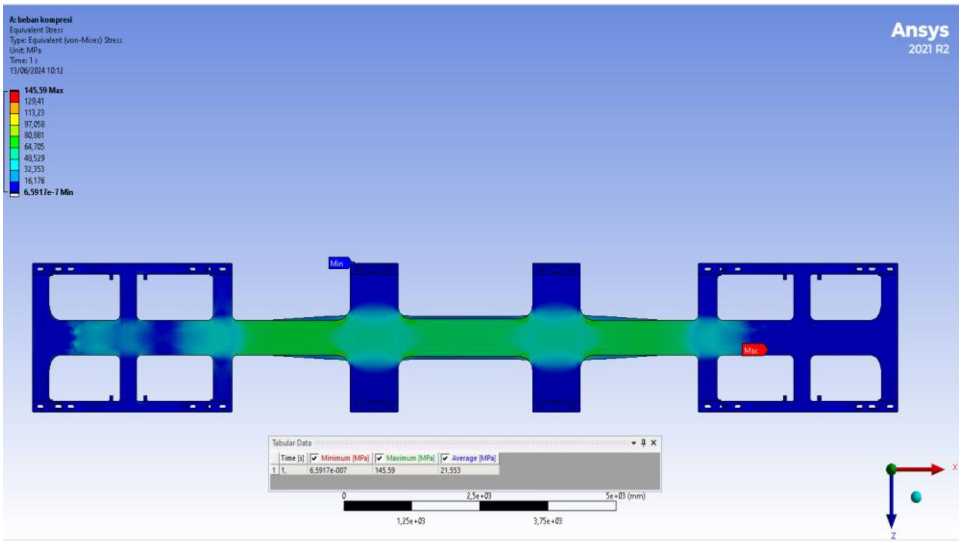

Figure 8 Compression Loading with 100 Tons

Figure 8 show the compression loading in the longitudinal direction imposed on the coupler with a load of 100 tons. The maximum stress simulation (von mises) obtained a value of 145.59 Mpa. This value is still safe from the allowable stress value of 266.25 Mpa**.**

3.3. Combination Loading

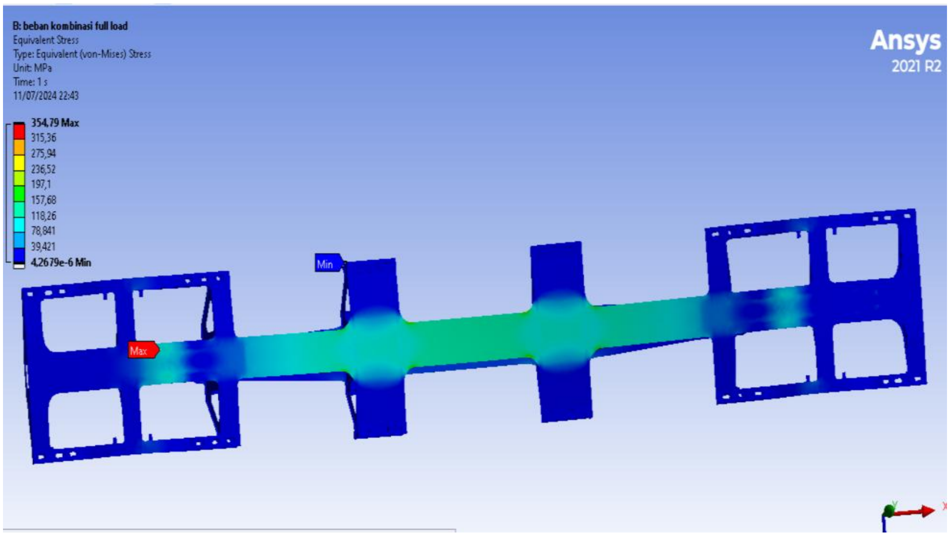

Figure 9 Combination Loading

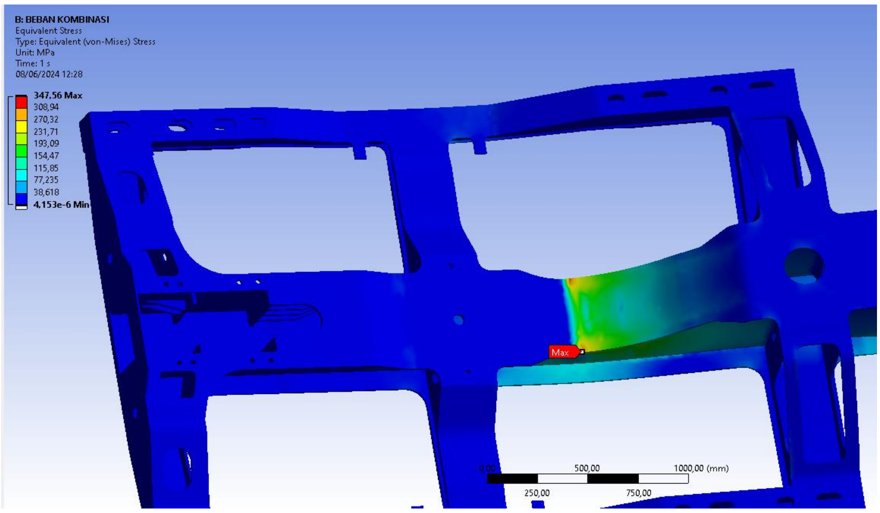

Figure 10 Maximum Stress Point

Figure 9 show the combination loading is carried out with 2 loading cases simultaneously between vertical dynamic loads and compression loads. For the simulation at maximum stress (von mises), a stress of 347.56 Mpa was obtained. This simulation result value is close to 75% of the material allowable stress of 266.25 but this value is still within the safe limits of the material.

At the time of combination loading in Figure 10, the maximum stress point is at the headstok bottom plate, namely in the fixed support area or where the bogie will be assembled with the underframe.

The calculation of the safety factor value refers to a formula in which the material allowable stress is set at 75% of the yield stress value, which is 266.25 MPa. This value is then divided by the yield stress result obtained from the simulation. In general, a design is considered safe if the safety factor value is greater than 1.

Table 1 Safety Factor

No	Type of Loading	Material Allowable Stress 75% (Mpa)	*Yield strength* (Mpa)	Result (Mpa)
1.	Vertikal Statis	266,25	152,81	1,74
2.	Compression	266,25	145,59	1,82
3.	Combination	266,25	0,76	0,76

This safety factor calculation aims to ensure that a system, structure or component has a sufficient level of safety against failure. In the three types of loading, the results of loading that are said to be safe are static vertical load 1.74, compression 1.82 and combination load 0.76.

4. CONCLUSION

The simulation results of the flat wagon underframe structure have a maximum stress value (von mises) in the static vertical loading simulation of 152.81 Mpa, compression loading of 145.55. This value is said to be safe because it is still below 75% of the material allowable stress value of 266.25 Mpa. But not in combination loading with a value of 347.56 Mpa. From the calculation of the safety factor in accordance with the formula that has been determined that the flat wagon underframe construction is safe in accordance with the predetermined limit value, which exceeds the number 1. But for combination loading there is a value of 0.76 this value is not safe and is still below the predetermined standard.

ACKNOWLEDGEMENTS

The authors would like to thank the Research and Community Service Center of the Indonesian Railway Polytechnic Madiun for making this research possible.

REFERENCES

[1] P. Šťastniak, M. Moravčík, P. Baran, and L. Smetanka, "Computer aided structural analysis of newly developed railway bogie frame."

[2] K. A. Irena, H. G. Lemu, and Y. A. Kedir, "Effect of Interference Size on Contact Pressure Distribution of Railway Wheel Axle Press Fitting," *Designs (Basel)*, vol. 7, no. 5, Oct. 2023, doi: 10.3390/designs7050119.

[3] J. W. Seo, H. M. Hur, H. K. Jun, S. J. Kwon, and D. H. Lee, "Fatigue Design Evaluation of Railway Bogie with Full-Scale Fatigue Test," *Advances in Materials Science and Engineering*, vol. 2017, 2017, doi: 10.1155/2017/5656497.

[4] P. Wan, X. Lei, W. Wu, Q. Hu, L. Wan, and G. Li, "Simulation Analysis of Coupled Structural Vibration of Highway Railway Combined Bridge Induced by Overlapping Action of Vehicle and Train," *Geofluids*, vol. 2022, 2022, doi: 10.1155/2022/1986593.

[5] P. Kurniawan, D. Darto, A. Iswantoko, and F. Sutrisno, "Simulation of Stress Intensity Factor and J-Integral on UIC 54 Railway Failure Using A Fracture Mechanics Approach," *Engineering Innovations*, vol. 11, pp. 1–8, Jun. 2024, doi: 10.4028/p-qvaf50.

[6] M. F. Shaikh and K. Nallasivam, "Static analysis of box-girder bridge under the influence of Indian railway vehicle loading using ANSYS finite element model," *Advances in Bridge Engineering*, vol. 3, no. 1, Dec. 2022, doi: 10.1186/s43251-022-00076-9.

[7] R. Silva, C. Costa, A. Arêde, R. Calçada, and D. Oliveira, "Structural analysis of a stone arch bridge under incremental railway static loading."

[8] M. Paudel, L. J. Lim, F. F. Yap, and K. Kho, "Vibration analysis of the third rail structure of a mass rapid transit system with structural defects," *Applied Sciences (Switzerland)*, vol. 11, no. 18, Sep. 2021, doi: 10.3390/app11188410.

[9] H. I. Prabawa, "Analysis of Indonesia's Railway Track Structure Behavior Under Double-Stack Freight Train Loading," *International Journal of Sustainable Construction Engineering and Technology*, vol. 14, no. 4, pp. 412–425, Oct. 2023, doi: 10.30880/ijscet.2023.14.04.030.

[10] J.-U. Cho, "Study on Convergence Technique through Structural Analysis on the Axle of Railway Vehicle," *Journal of the Korea Convergence Society*, vol. 6, no. 1, pp. 93–101, Feb. 2015, doi: 10.15207/jkcs.2015.6.1.093.