Jurnal Sainstech Nusantara
https://nusantarajournal.id/sainstech
Vol.1 No.4 (2024) 21-29
ISSN: 3063-0657

Modeling and Simulation of Static Strength Railway Bogie Bolster Using Ansys Workbench

Willy Artha Wirawan[1], Akbar Zulkarnain[2], Ilham Satrio Utomo[2], Muhammad Sabilil Rouf[2], Muhammad Aizuddin Bin Azari[3], Joewone Prasetijo[3]

[1]Department Aircraft Engineering Politeknik Penerbangan Surabaya, Jalan Jemur Andayani I No 73, Surabaya, 60236, INDONESIA
[2]Indonesian Railway Polytechnic, Tirta Raya street, Manguharjo, Madiun, 63161, INDONESIA
[3]Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia Parit Raja, Batu Pahat, Johor, 86400, MALAYSIA

Article Info

Article history:

Received 01 January 2024
Revised 2 February, 2024
Accepted 16 February, 2024

Abstract

This study aimed to ensure that plastic deformation does not occur on the structure of bolster bogie 1722 when it accommodates burdens or loads based on previously determined overload limit. The simulation in this study used the finite element method with the help of Ansys Workbench 2020. The standard of testing UIC 515-4 was employed as a reference for the simulation, notably in determining the load case. The result of static simulation on the overload showed that the maximum stress on the vertical load case was 72.764 MPa, and the maximum stress on the transverse and vertical load case (combination) was 109.9 MPa. The result of maximum vertical deflection was obtained from the case of vertical load as 0.553 mm and in the case of transverse and vertical load (combination) as 1.799 mm. The structure of the bolster design is considered safe because the maximum score of stress was under the yield strength score of the used material, namely SS400 as 245 Mpa.

Keywords: Bolster Bogie, Static Strength, Railway, Finite Element Method

Corresponding Author:

Name: Willy Artha Wirawan
Email: willyartha@poltekbang.ac.id

1. Introduction

In the field of engineering, accurate modeling and simulation play a crucial role in ensuring the strength and reliability of various components. One application of this is the modeling and simulation of static strength in railway bogie bolsters. By using ANSYS Workbench, engineers can accurately analyze and evaluate the static strength of railway bogie bolsters [1]. This helps in improving the design and performance of these components, ensuring their safety and efficiency. In today's modern world, efficient and reliable railway systems are of paramount importance for transportation networks. Therefore, the development and optimization of railway bogie components, such as bolsters, are essential. Through the use of ANSYS Workbench, engineers can simulate and analyze the static strength of railway bogie

DOI: https://doi.org/10.71225/jstn.v1i4.54

bolsters, allowing for a better understanding of their performance under various loading conditions and potentially identifying areas for improvement [2][3] [4].

Predictions and simulations of bogie frames have been carried out by researchers in Indonesia. Budiwantoro et al. [5] investigated the simulation of frame loads on the bolster of a capsule metro train. In the study, the UIC 615-1 and UIC 615-4 standards were used as a reference in providing load conditions. The results of the study show that the rolling frame meets the UIC standards and the strength standards that have been allowed under the Regulation of the Minister of Transportation of the Republic of Indonesia No. 175 of 2015. Evaluation of bogie rollers on Indian Railway (IR) railway wagons was also conducted by Pokkula & Gupta [6] using the finite element method for new design evaluation or otherwise called design optimization. The simulation design was designed using Solidworks software and simulation using Ansys Workbench. The simulation standard used is AAR-M-202, with the result that the modified bolster has increased stiffness and reduced maximum stress values.

This research investigates the application of ANSYS Workbench in modeling the static strength characteristics of rolling bogies. By utilizing finite element analysis techniques, the complex geometry, material properties, and loading conditions inherent in bogie bumpers can be represented and analyzed with high accuracy. It is possible, through thorough simulation, to evaluate stress distributions, deformation models, and potential failure points, therefore facilitating appropriate design decisions and optimization strategies. The significance of this research lies in its contribution to improving the structural performance and safety of bogie bolster in transport systems. By using ANSYS Workbench for modeling and simulation, it is possible to perfect design parameters, reduce potential weaknesses, and ultimately ensure the robustness and reliability of the bogie bolster system under static loading conditions.

2. Research Method

This research analyses the 1722 bogie bolster using the simulation-based finite element method. The simulation of bogie bolster 1722 uses Ansys Workbench software. The 1722 bogie bolster will be modeled and divided into very small parts, commonly referred to as meshing. The boundary conditions and loading refer to the UIC 515-4 standard which regulates Passenger Rolling Stock - Trailer Bogies - Running Gear - Bogie Running Gear - Bogie Frame Structure Tests. A bogie is required to accommodate the load it is supporting without plastic deformation occurring International Union of Railways [7]. In the UIC 515-4 standard, overloading consists of vertical and transverse loading, which describes the condition of the bogie when it is under extreme circumstances. Bogie 1722 is designed to support the load of the empty wagon body weight combined with the passenger load, with each passenger assumed to weigh 75 kg.

2.1. Materials

The materials required for bogies are in the Minister of Transportation Regulation Number: KM. 41 the Year 2010 in article 12, paragraph 1, which contains bogies made of steel with high strength and stiffness against loading without fixed deformation, is equipped with loading-resistant construction and can reduce vibration [8]. Based on the applicable standards, the material used in the 1722 bogie construction is SS400 steel (JIS G3101) for all 1722 bogie bolster constructions with yield strength 245 Mpa and tensile strength 400 Mpa [9]

DOI: https://doi.org/10.71225/jstn.v1i4.54

Table 1. Mechanical Properties

Elastic Modulus	Poisson Ratio	Density	Yield Strength	Ultimate Strength
2.1×10^5 (Mpa)	0,3	7850 (Kg/m)^3	245 (Mpa)	400 (Mpa)

2.2. Geometry Modeling

The 1722 bogie bolster geometry modeling in this simulation uses Autodesk Inventor software. The designed geometry model starts from the arrangement of constituent parts and is then put together, or called the Assembly stage. The design that comes from Autodesk Inventor is stored in STEP type (.stp), which is then imported into Ansys Workbench.

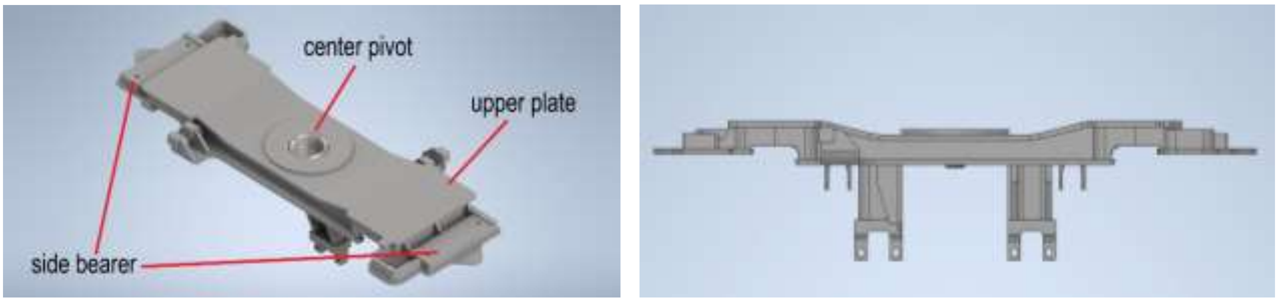

Figure 2. Bolster Bogie (a) Model in Autodesk Inventor Top view (b) Model in Autodesk Inventor Front view

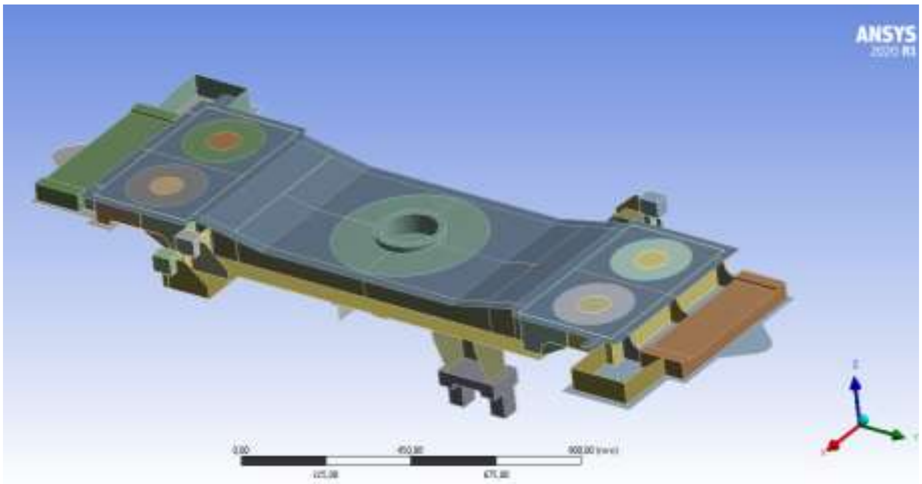

Figure 3. Model Simplification in Ansys Workbench

2.3. Meshing

Mesh is the division of objects into smaller or finite parts [10]. In this 1722 bogie bolster simulation, the part or component modeled as a shell uses a hybrid mesh type because it can adjust to the shape of the geometry. The element type used is predominantly quadrilateral, so it is neater and has better mesh quality on parts modeled as solid using the tetrahedron type. The smaller the meshing, the more accurate the calculation results, but it requires large computing power [11] [12]. In the meshing stage, 48526 nodes and 15793 elements were obtained.

DOI: https://doi.org/10.71225/jstn.v1i4.54

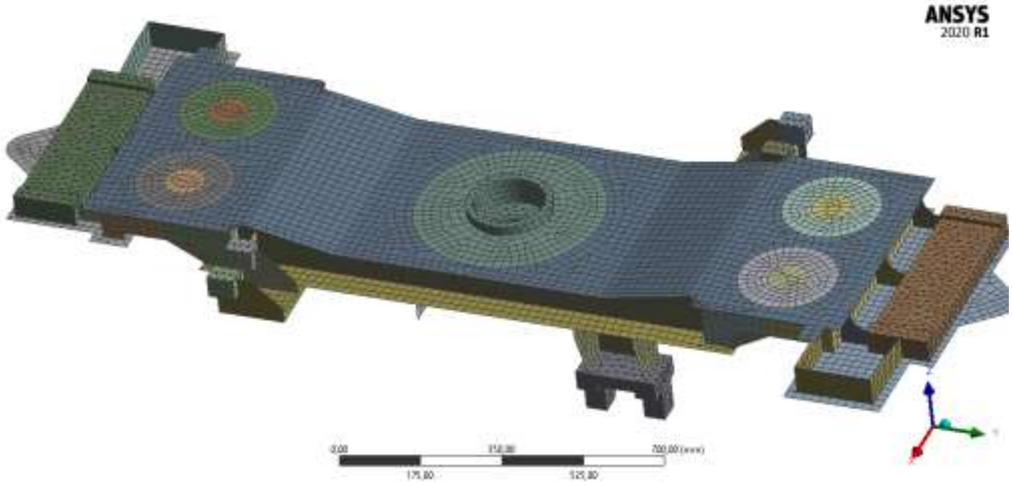

Figure 4. Meshing 1722 bogie bolster mesh

The determination of mesh size is done by conducting a convergence test. The convergence test is carried out by varying the mesh size and then observing whether the results of the maximum stress of each mesh size variation that occurs will converge to a certain value. In addition to determining the mesh size, the convergence test on finite element modeling is carried out to verify the simulation results regarding the mesh size used. According to Naik et al. [13], finite element analysis results are declared convergent if the percentage error between element sizes is 6%-8%. Based on the convergence test results table, the mesh size with the smallest error value is selected, namely the mesh size of 20 mm. In the hybrid mesh with a size of 20 mm, 47,224 nodes, and 16,629 elements are obtained.

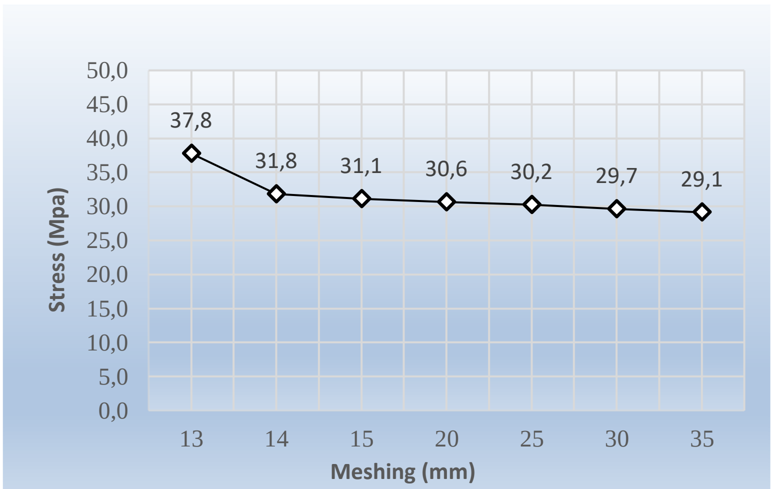

Figure 5. Meshing Convergence test

2.4. Determination of Boundary Conditions and Load

The fulcrum location is selected in the secondary suspension on the *bogie bolster* that rests on the *bogie frame*. Determination of boundary conditions is done by providing support at several predetermined fulcrums. The support used in this analysis is *remote* displacement. The static loading

DOI: https://doi.org/10.71225/jstn.v1i4.54

used is based on the UIC 515-4 standard. Static loading with overload has two types: vertical loading and combination loading (transverse and vertical).

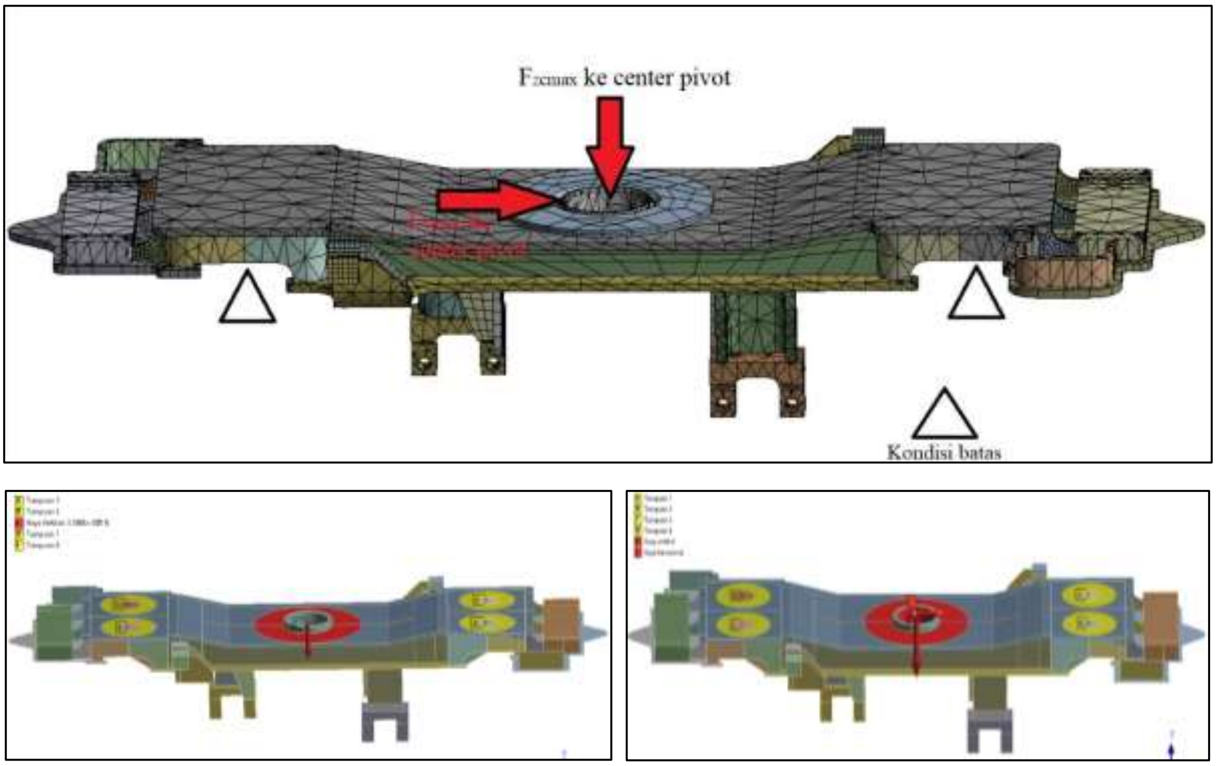

Figure 5. (a) Boundary conditions (b) Vertical loading (c) Combination loading

The static loading that occurs on bogie bolster 1722 is assumed to support the load of the 34.750 kg K1 executive train and the total passenger load of 3.750 kg. The load received is located at the top of the center pivot for vertical loads, while for transverse loads, it is on the inside of the center pivot, pointing towards the side. The loading support in both cases is located on the *secondary suspension bogie*. The loads supported by the 1722 bogie bolster based on the UIC 515-4 standard can be seen in Table 2.

Table 2. Load

Type of Loading Force	Value	Load Location
Vertikal Load ($Fz1max$ &$Fz2max$)	188.650 N	Center Pivot
Transversal Load ($Fymax$)	82.883 N	Center Pivot

3. Results And Discussion

3.1. Vertical Loading

The results obtained from this calculation-based simulation are the maximum stress value and maximum deflection of the 1722 bogie bolster. Vertical loading is done by applying an even force on the center pivot of the bolster in the direction of the Z-axis (assumed to be the downward direction of the

DOI: https://doi.org/10.71225/jstn.v1i4.54

train) of 188,650 N. This case is drawn when the train passes through a rail joint which causes an extreme downward load on the bogie bolster. The vertical loading simulation results can be seen in the following figure 6.

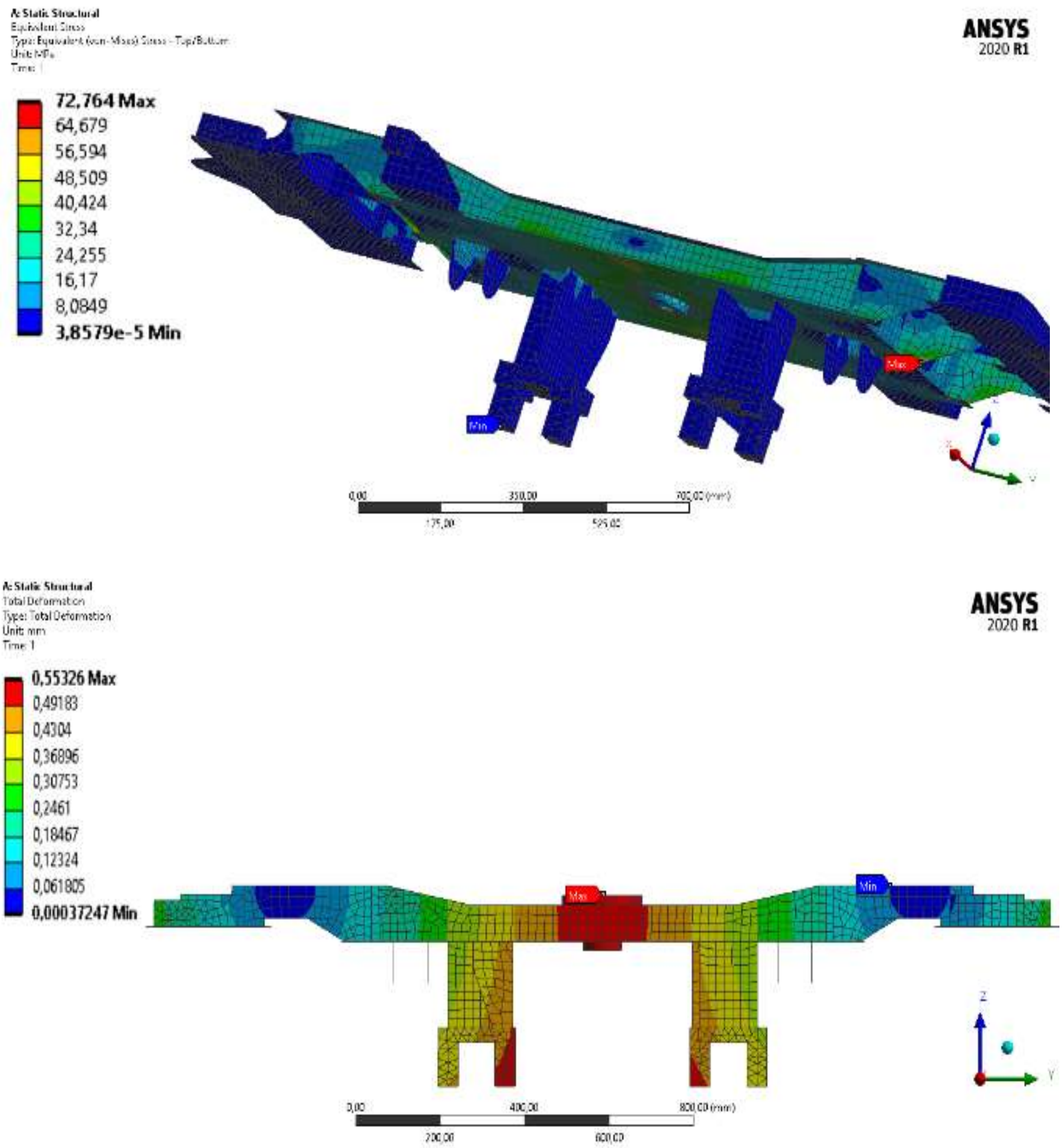

Figure 6. Vertical loading (a) Maximum Stress (b) Maximum Deflection

The results obtained in vertical loading are that the 1722 bogie bolster experiences a maximum stress of 72.764 MPa on the side stiffener due to the supported load. At the center pivot, the maximum deflection is downward or to the Z-axis of 0.553 mm.

DOI: https://doi.org/10.71225/jstn.v1i4.54

3.2. Combination Loading

In the case of combination loading, a transverse force is applied to the inside of the center pivot bolster to the Y-axis (assumed to be side direction) and a vertical force to the Z-axis (assumed to be downward direction). This case illustrates when the train passes through a bend or uneven railroad. The results of the maximum stress and maximum deflection in the combination loading case can be seen in the following figure:

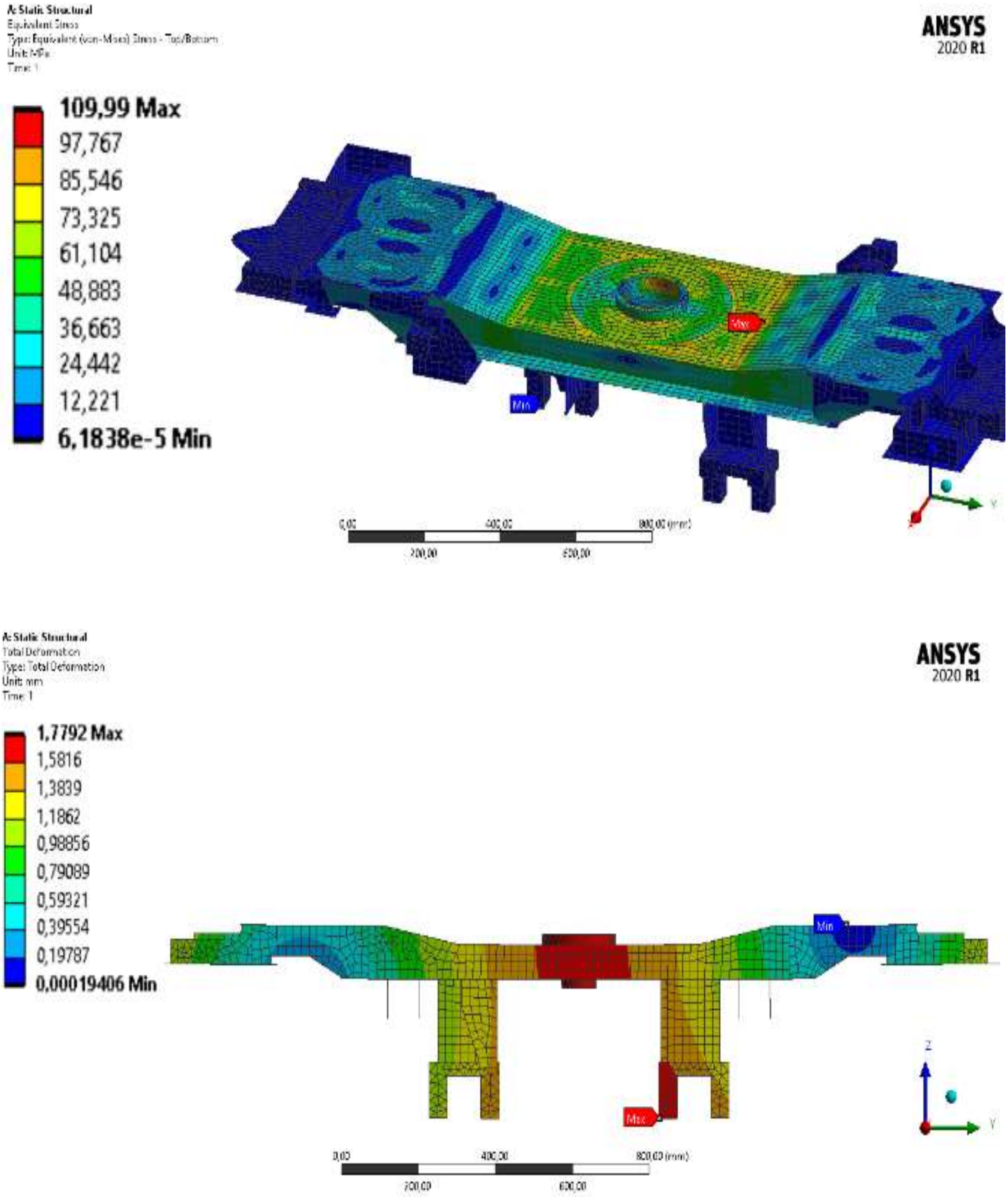

Figure 7. Combination Loading (a) Maximum Stress (b) Maximum Deflection

DOI: https://doi.org/10.71225/jstn.v1i4.54

The results obtained in transverse and vertical loading are that the 1722 bogie bolster experiences a maximum stress of 109.99 MPa on the upper plate due to the load it supports. The maximum deflection occurs downward or to the Z-axis of 1.799 mm on the traction dumper. Simulation-based measurements were carried out with loading cases referring to the UIC 515-4 standard to ensure this. From the two loading cases in the UIC 515-4 standard, the results are obtained as maximum stress, which is compared with the strength of the material used, namely SS400. If the maximum stress value is lower than the yield strength value, it can be concluded that the structure is safe and no fixed plastic deformation occurs. The following are the results of simulation-based measurements on the 1722 bogie bolster structure [14].

<table>
<tr><th>Loading Case</th><th>Maximum Stress (MPa)</th><th>Maximum Deflection(mm)</th><th>Yield Strength Value (MPa)</th><th>Description</th></tr>
<tr><td>Vertical</td><td>72,764</td><td>0,553</td><td rowspan="2">245</td><td>σ max < σ clearance</td></tr>
<tr><td>Transverse & vertical (combination)</td><td>109,9</td><td>1,799</td><td>σ max < σ clearance</td></tr>
</table>

Table 2. Simulation Result of bogie bolster

Based on the two loading cases carried out, all the results show that the stress values that occur in the simulation due to the loading of the two cases are still below the maximum allowable stress values. The deflection that occurs can be used as additional information for design at the component integration stage. If the deflection that occurs is very large, it is assumed that it can interfere with the performance of the bogie structure, and then design adjustments are made.

4. Conclusion

Based on the simulation of the two variations of loading cases that have been carried out using Ansys 2020 software, it is obtained that the maximum stress value occurs in the case of combined loading (transverse and vertical) of 109.9 MPa on the upper plate of the bogie bolster with the SS400 material permit stress of 245 MPa. This result is considered safe because the maximum stress value in both loading cases is 44.89% below the material's allowable stress. The maximum deflection value occurs when combined loading (transverse and vertical) is 1.799 mm.

Acknowledgments

I would like to express my sincere gratitude to the Department of Aircraft Engineering, Politeknik Penerbangan Surabaya, Indonesian Railway Polytechnic, and Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia, for their support, resources, and opportunities throughout this research. The collaboration and guidance from these institutions have greatly enriched my knowledge and perspective in the fields of aircraft engineering, railway systems, and engineering technology in general.

DOI: https://doi.org/10.71225/jstn.v1i4.54

References

[1] H. Claus and W. Schiehlen, "Modeling and Simulation of Railway Bogie Structural Vibrations," *Veh. Syst. Dyn.*, vol. 29, no. SUPPL., pp. 538–552, 1998, doi: 10.1080/00423119808969585

[2] Menteri Perhubungan RI, "Peraturan menteri perhubungan republik Indonesia nomor 16 tahun 2022 tentang rancang bangun dan rekayasa sarana." Kementerian Perhubungan Republik Indonesia, 2022.

[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," Adv. Mater. Sci. Eng., vol. 2017, 2017, doi: 10.1155/2017/5656497.

[4] S. N. Wang, "Static analysis and size optimization for the bed of gantry milling machine based on ANSYS workbench," Curr. Trends Comput. Sci. Mech. Autom., vol. 2, pp. 298–304, Jan. 2018, doi: 10.1515/9783110584998-033/MACHINEREADABLECITATION/RIS.

[5] B. Budiwantoro, A. H. Masyhur, and K. Mukti, "Static load analysis of bolster frame component in metro kapsul train using finite element method," Pros. SNTTM, vol. XVII, no. 25, 2018.

[6] R. Pokkula and T. V. K. Gupta, "Finite element method based evaluation of bogie bolster design," Int. J. Veh. Struct. Syst., vol. 13, no. 2, pp. 160–163, 2021, doi: 10.4273/ijvss.13.02.05.

[7] International Union of Railways, "UIC 515-4." pp. 1–26, 1993.

[8] Menteri Perhubungan RI, "Peraturan Menteri Perhubungan nomor: KM 41 tahun 2010 tentang standar spesifikasi teknis kereta yang ditarik lokomotif." Kementerian Perhubungan Republik Indonesia, Indonesia, 2010.

[9] F. P. W. W. A. Wirawan, D. S. Atmaja, and F. X. P. P. Fadli Rozaq, Aji Satria Bagaskara, Modelling and Structural Analysis of Tram Railway Vehicle Body with Finite Element, vol. 1. Atlantis Press International BV, 2023. doi: 10.2991/978-94-6463-126-5.

[10] S. Widi Astuti, W. Artha Wirawan, A. Zulkarnain, and D. Tri Istiantara, "Comparison of Energy Absorption and Pattern of Deformation Material Crash Box of Three Segments with Bilinear and Johnson Cook Approach," J. Phys. Conf. Ser., vol. 1273, no. 1, 2019, doi: 10.1088/1742-6596/1273/1/012078.

[11] J. D. Chung and J. S. Pyun, "A study on analysis method considering camber for the carbody deflection of the urban transit vehicles," Adv. Mater. Res., vol. 871, pp. 323–329, 2014, doi: 10.4028/WWW.SCIENTIFIC.NET/AMR.871.323.

[12] W. Artha Wirawan, A. Zulkarnain, H. Boedi Wahjono, Jamaludin, and A. Tyas Damayanti, "The Effect of Material Exposure Variations on Energy Absorption Capability and pattern of Deformation Material of Crash Box of Three Segments," J. Phys. Conf. Ser., vol. 1273, no. 1, 2019, doi: 10.1088/1742-6596/1273/1/012081.

[13] N. Naik, V. George, and S. Kowshik, "Investigations on mesh discretization error in fem based structural analysis using Ansys," Int. J. Adv. Comput. Eng. Netw., vol. 2, no. 8, pp. 17–22, 2014.

[14] R. G. Dong, S. Sankar, and R. V. Dukkipati, "A finite element model of railway track and its application to the wheel flat problem," Proc. Inst. Mech. Eng. Part F J. Rail Rapid Transit, vol. 208, no. 1, pp. 61–72, 1994, doi: 10.1243/PIME_PROC_1994_208_234_02.

DOI: https://doi.org/10.71225/jstn.v1i4.54