Fatigue Analysis of Front Axle Using Finite Element Analysis

Shyam Sundar Tiwari[1], Rajesh Khodre[2]

[1]Research Scholar

[2]Professor

[1,2]Oriental Institute of Science and Technology

Abstract

This study article provides a complete investigation of fatigue conducted on the front axle of a heavy-duty truck under applied forces, utilising Ansys simulation software. The objective of this study is to evaluate the structural integrity, endurance, and potential points of fatigue failure of the front axle when subjected to stress circumstances representative of real-world scenarios. The approach employed in this study encompassed the development of a comprehensive three-dimensional (3D) model of the front axle. This model incorporated pertinent material properties and applied forces that are representative of typical trucking operations. The utilisation of finite element analysis (FEA) methodologies was employed in order to model the response of the axle to the applied forces and evaluate its fatigue characteristics. By means of meticulous examination, essential variables were assessed, including safety aspects, ultimate lifespan, and cumulative damage. The analysis of the safety factor demonstrated a significant margin of safety, as indicated by a maximum safety factor of 10, indicating a high level of reliability under the given loading conditions. Nevertheless, it is worth noting that certain localities exhibit lower safety factors, specifically the lowest value of 2.681. This observation draws attention to the possibility of fatigue vulnerability in these particular areas.

Keywords: Fatigue Analysis, Front Axle, Ansys Simulation, Finite Element Analysis (FEA),

1. Introduction

In the contemporary context of a highly competitive industrial environment, there exists a growing imperative for the implementation of production processes that are both more efficient and cost-effective the rationale behind this request arises from the objective of reducing production expenses, increasing efficiency, minimising time intervals, and concurrently improving product excellence [1]. In recent decades, there has been a growing emphasis on the optimisation of vehicle design and the extension of vehicle component lifespan, particularly in relation to the front axle beam (FAB). This shift in focus can be attributed to the prevailing global economic situation. The current off-highway vehicle industry necessitates components that include cost-effectiveness, lightweight characteristics, and durability to assure extended usage and fuel economy. The imperative to optimise material usage and employ advantageous surface treatments in order to enhance the durability of vehicle components is underscored by this requirement.

During the operation of a vehicle, the axle, which serves as the principal load-bearing component, undergoes cyclic stress changes as a result of variations in road conditions [2]. It is of utmost importance to ensure that the axle possesses the ability to endure fatigue failure during the entirety of its expected operational lifespan. The axle experiences a range of stresses in different orientations, primarily vertical bending resulting from drive torque, cornering, and braking forces. In real-world scenarios, these loads exhibit temporal variability [3]. The occurrence of vertical bending, a significant and recurring stress on an axle, especially in the context of heavy-duty commercial vehicles, mandates the utilisation of solid axles owing to their superior capacity to withstand huge loads. The presence of uneven road surfaces gives rise to dynamic stresses resulting from dynamic forces, which frequently contribute to the occurrence of axle fatigue failure [4].

2. Literature Review

Mr B Siva Ramakrishna et al. (2023) The objective is to calculate the natural frequency and static structural of wheel axle of the locomotive is modulating those frequencies and avoiding resonance by using the loads, thus the vibrations of wheel axle of the locomotive may reduce. The influence of wheel axle of the locomotive design resonance phenomenon is investigated by ANSYS software. In this project, the 3D model of wheel axle of the locomotive is modelled in CATIA V5 and imported into ANSYS 19.3 software to perform static and dynamic analysis to analyze strength and dynamic characteristics of wheel axle of the locomotive and optimize if required. Lufan Zhang et al. (2023) This paper collects information from a wide field of literature and summarizes the current status of fatigue-analysis research. It covers related theoretical knowledge, fatigue-life prediction methods, and fatigue design methods and their application scenarios, and it summarizes the challenges and research hotspots in the field. On the basis of this examination, future development directions of fatigue-life prediction methods are proposed. The conclusions will have a certain guiding role in the development of fatigue-analysis methods. Omkar Patil et al. (2022) The current project aims to design a good UPD for a heavy commercial vehicle that is effective enough to stop a passenger from falling under the rear of the truck and able to absorb a great deal of crash energy to attenuate accident severity. Also, an effective SUPD is designed to stop road users such as cyclists and pedestrians from falling under the wheels of a truck or trailer. Cuixia Zhang et al. (2022) The ANSYS Workbench is used to mesh the hub and import the mechanical and physical parameters. Considering full load, emergency turning and braking, and the treatment of constraints and loads, the stress concentration position is obtained, and the lightweight design is carried out. The free vibration analysis and fatigue life prediction of the hub after lightweight design are verified. After the lightweight design, the optimized hub mass is reduced by 7.52%, and its structural stability and strength meet the demands of various working conditions. The application of a lightweight hub reduces CO_2 emissions by 49875 kg during the life cycle of the vehicle. This study provides theoretical and methodological support for lightweight design of mechanical parts for low carbon. Durgeshwar Pratap Singh (2021) The function of the rear axle shaft of an automobile is to transmit power to the wheels. The rear axle shaft is subjected to cyclic loading, torsion loading, and the load due to the weight of the vehicle and passengers. The scope of the present paper is to find out the fatigue life due to cyclic loading and to find out the natural frequency of vibration of the rear axle shaft to predict the safe working condition for it. The tool used is ANSYS 14.0. AISI 4140 steel is considered as it is the most common material used in the manufacturing of rear axle shaft in auto rickshaws. Results show that the minimum and maximum life cycle of the shaft is 4544 and

106 cycles, respectively. The natural frequency of vibration varies from 92.18–515.19 Hz for the first six modes of vibration. These results will help in determining the suitable material for the manufacturing of the rear axle shaft. Wei He et al. (2021) This study presents a novel method for identifying the axle spacing and weights of vehicles. It only requires the flexural strain signal recorded from the weighing sensors, leading to both a reduction in the installation cost and broader applications of BWIM systems. The effectiveness and accuracy of the proposed method are demonstrated through numerical simulations. Laboratory experiments based on a scaled vehicle–bridge interaction (VBI) model were also conducted for verification. The results show that the proposed method has good accuracy for axle spacing and axle weight identification. Rishabh Chaudhary, Srishti Singh (2020) This raises the need for fatigue resistant materials for the components of aircraft. The axle of the aircraft supports the whole weight of the aircraft and hence is subjected to fatigue loading during its entire service life. Titanium alloys are commonly used material in aviation sector since decades. In this paper, the behaviour of two titanium alloys, namely TIMETAL 834 and Ti–6Al–4V has been compared under fatigue loading which are used for axle of the aircraft. The fatigue life of the axle is increased by 81,230 cycles for TIMETAL 834 which is 17.77% of life for component if Ti–6Al–4V is used. The 3D model of axle is built in Creo 3.0 and exported to ANSYS. The fatigue analysis is carried out using fatigue tool in ANSYS Workbench 2019 R2.

3. Research Methodology

Research methodology for conducting fatigue analysis of a front axle through Finite Element Analysis (FEA) using ANSYS software involves a systematic approach to evaluate the component's endurance under cyclic loading. Initially, the process begins with clearly defining the study's objectives and scope, outlining the operational conditions, and specifying the types of loads and anticipated outcomes. The core of the methodology lies in constructing a detailed 3D model of the front axle assembly within the ANSYS environment. This model must meticulously represent the axle's geometry, incorporating accurate material properties essential for predicting fatigue behavior. Material characterization is paramount, integrating fatigue-related properties like stress-life or strain-life data into the simulation. Mesh generation follows, where an appropriate mesh is created to capture critical areas susceptible to stress concentrations while maintaining computational efficiency. Boundary conditions and loads are then set up, mirroring real-world scenarios encompassing driving conditions, braking forces, and turning stresses, ensuring they are as close to reality as possible.

The subsequent step involves configuring ANSYS for fatigue analysis, selecting suitable fatigue life calculation methodologies tailored to the specific loading conditions and material properties. Executing the FEA simulations within ANSYS generates data on stress, strain, and predicted fatigue life for the front axle under various operational loads. Upon completion, meticulous analysis of the simulation results is conducted. This examination focuses on identifying areas prone to fatigue failure, stress concentrations, deformation patterns, and estimates of fatigue life. Comparisons are made against design criteria or industry standards to gauge the component's reliability under different loading conditions.

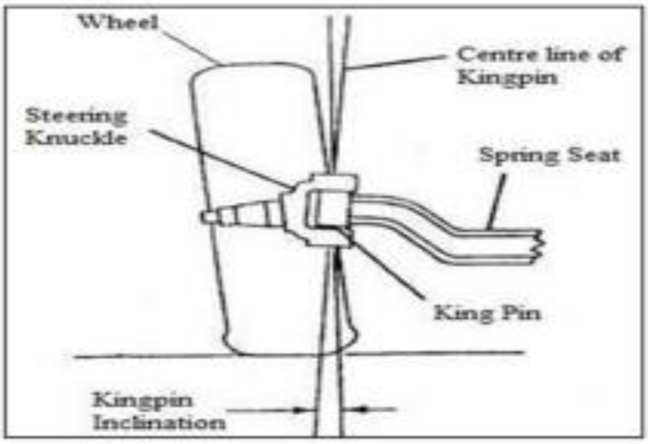

Figure 1. Components of front axle beam

3.1 Material Descriptions

The provided material data pertains to the structural steel properties crucial for designing a front axle intended for heavy-duty trucks. These constants serve as foundational elements in understanding how the chosen steel behaves under various conditions, aiding in the axle's design and performance predictions. Firstly, the Density of 7.85e-006 kg mm^-3 outlines the mass per unit volume of the steel. This metric is fundamental for calculating the overall weight and distribution of the axle, ensuring that the structural integrity and load-bearing capacity align with the truck's specifications. The Isotropic Secant Coefficient of Thermal Expansion at 1.2e-005 C^-1 is significant in predicting how the steel material will respond to temperature fluctuations. This coefficient denotes the rate at which the axle's dimensions will change concerning temperature variations, an essential factor in designing to accommodate thermal expansion or contraction without compromising functionality. The Specific Heat Constant Pressure of 4.34e+005 mJ kg^-1 C^-1 represents the amount of heat energy required to raise the steel's temperature by a certain degree at a constant pressure. This value aids in understanding the material's ability to absorb and retain heat, vital for assessing thermal performance and potential heat-related issues during operation.

Table 1. Mechanical Properties

Property	Value	Units
Density	7.85e-006	kg mm^-3
Isotropic Secant Coefficient of Thermal Expansion	1.2e-005	C^-1
Specific Heat Constant Pressure	4.34e+005	mJ kg^-1 C^-1
Isotropic Thermal Conductivity	6.05e-002	W mm^-1 C^-1
Isotropic Resistivity	1.7e-004	ohm mm
Yield Strength	998	Mpa
Ultimate Tensile Strength	998	Mpa
Poisson ratio	0.3	-

Modulus of Elasticity	1.6667e+005	Mpa
Young's Modulus MPa	2.e+005	Mpa

3.2 Finite Element Method

In the context of Finite Element Method (FEM), the phase involves preparing the geometry and model before analysis. One crucial step is creating a Finite Element (FE) model based on the physical structure using specialized software like UNIGRAPHICS NX-9, as mentioned. The goal is to accurately represent the beam axle assembly in a digital form suitable for analysis.

A. Pre-Processing

The Finite Element (FE) Solid Computer-Aided Design (CAD) model of the entire beam axle assembly has been generated utilizing UNIGRAPHICS NX-9 modeling software, as depicted in Figure 2.

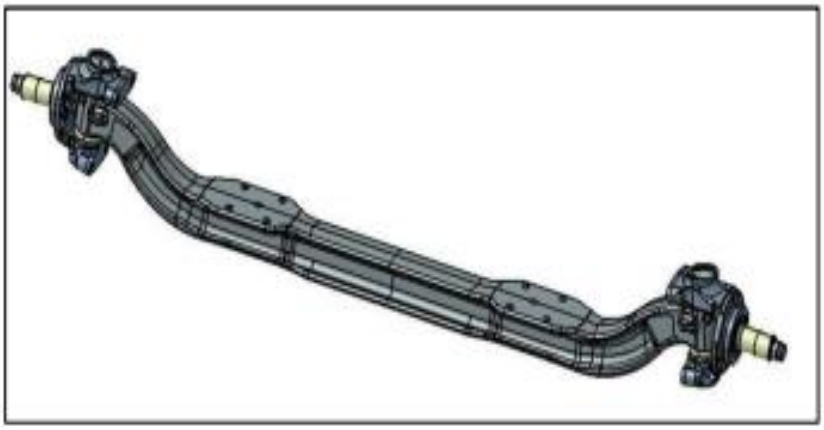

Figure 2. Axle Overview

B. Meshing

The entire assembly, comprising the front axle beam, steering knuckle, and kingpin, has been subjected to a meshing process using an 8mm grid size for the steering knuckle and kingpin, whereas the front axle has been meshed with a 5mm grid size. Tetrahedral elements have been utilised consistently throughout the entirety of the assembling process. The meshing sizing options have been configured to incorporate sophisticated sizing features, specifically closeness and curvature, with a centre of medium significance [13]. The Proximity Element Size parameter has been configured to a value of 5, but the maximum element size has been specified as 10 in the meshing option settings. The Front Axle Finite Element (FE) model consists of roughly 461,106 Elements and 703,886 Nodes.

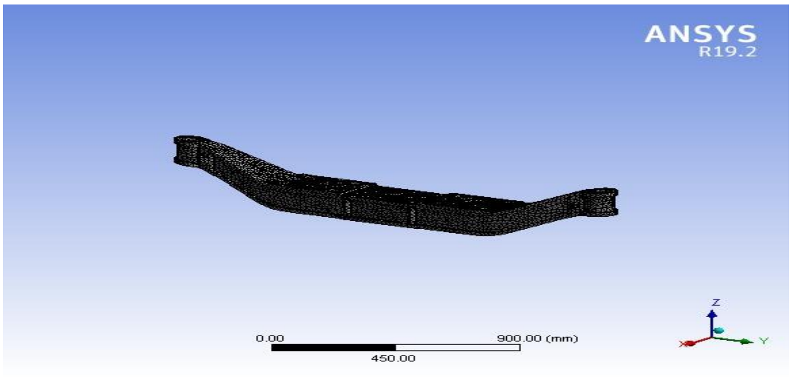

Figure 3. Meshing

C. Boundary Condition

Boundary condition validation is a critical step in Finite Element Analysis (FEA) to ensure that the model's behavior closely matches real-world performance.

1) Vertical Boundary Condition

Movement constraints have been imposed on the front axle assembly: the left support knuckle exhibits constraints in all three directions—X, Y, and Z—limiting its movement, whereas the right support knuckle is constrained only in the Y and Z directions, preventing motion along the X-axis [14]. This setup is crucial for controlling and defining the permissible degrees of freedom of the respective knuckles within the assembly, ensuring specific limitations on their movement and influencing the overall behavior and stability of the system.

2) Vertical + Braking Boundary Condition

The left spring pad has been subjected to movement constraints in all three directions—X, Y, and Z—limiting its freedom of movement. Conversely, the right spring pad has constraints applied only in the Y and Z directions, restricting motion along the X-axis. This configuration dictates the allowed degrees of freedom for each spring pad within the assembly, regulating their displacement and influencing the overall behavior and stability of the system [15].

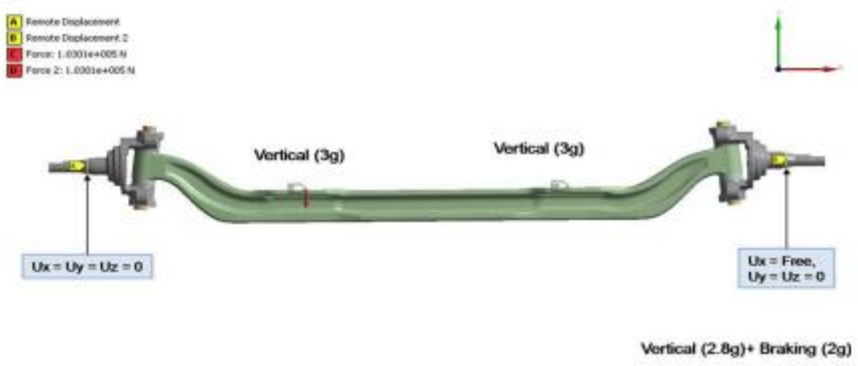

Figure 3. Vertical Braking Boundary Condition

3) Fix Support

In the field of engineering and structural analysis, a fixed support is a term used to describe a boundary condition or constraint that imposes limitations on specific degrees of freedom of a structure or component [16]. A fixed support is distinguished by its ability to restrict translation and rotation in all three axes, thus immobilising the structure in these directions.

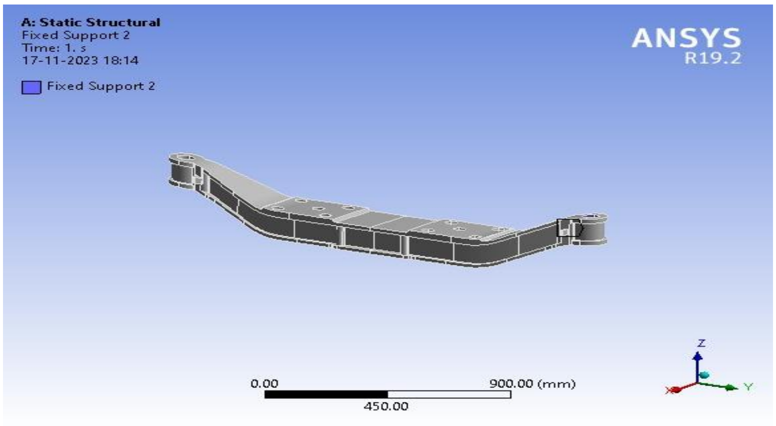

Figure 4. Fixed Support Boundary Condition

In the specific context of analysing a front axle through the utilisation of finite element analysis (FEA) or simulation tools such as Ansys, the application of a fixed support to the axle would entail the imposition of rigid fixation at specific points or nodes within the structure. This fixation effectively restricts any potential movement or rotation in directions that are perpendicular to the axle. In the analysis of a truck's front axle using Ansys or a comparable software, it is common practise to apply a fixed support at designated spots where the axle interfaces with the vehicle chassis or where it is mounted. This approach is employed to replicate the restrictions imposed by the mounting points in a real-world context [17]. When performing simulations or analyses on a front axle system, the exact representation of securing or constraining the axle in the actual vehicle assembly is achieved by using fixed supports.

4. Result and Discussion

The fatigue analysis of a truck's front axle using Ansys involves a meticulous process to ensure the axle's structural integrity under repeated loading conditions. Initially, a detailed 3D model of the front axle is created, incorporating the truck's actual design specifications and material properties. This model is then divided into smaller elements through meshing, enabling precise analysis.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 12 | December - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

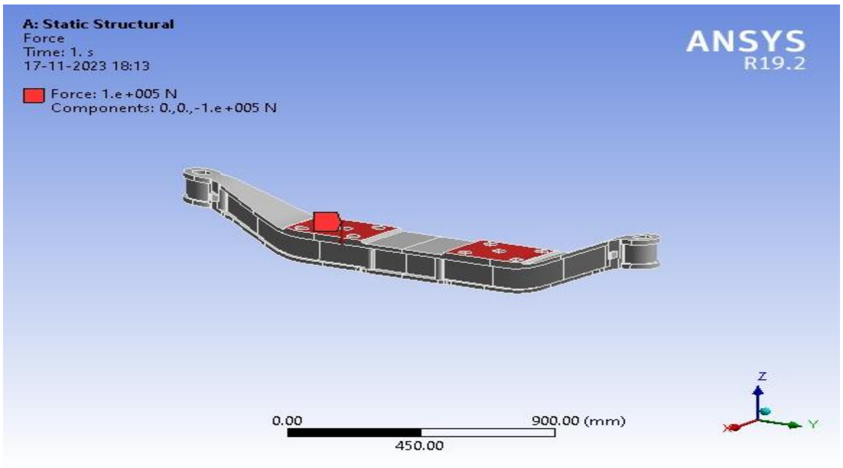

Figure 5. Applied Load

During the model setup, various applied forces, such as the specified force of 1e+005N, are incorporated to replicate real-world conditions. The forces being simulated in this study closely mimic the real loads that the axle undergoes during its operational use. Additionally, precise limitations are incorporated to accurately reproduce the mounting and support points. The pivotal stage encompasses the examination of fatigue, use methodologies such as the Stress-Life (S-N) or Strain-Life (E-N) approach. This stage involves the computation of stress or strain levels within the material and their subsequent comparison with fatigue parameters derived from material testing [19]. Through the assessment of stress or strain levels in relation to established fatigue curves, engineers are able to make predictions regarding likely locations of failure and estimate the number of load cycles that the axle can withstand before experiencing failure.

The interpretation of the fatigue analysis data is of utmost importance. The data is carefully examined by engineers in order to identify specific sections of the axle that are susceptible to fatigue failure. The present analysis serves to optimise the design by strengthening areas of weakness and transferring loads in order to improve durability and prolong the lifespan of the axle. The systematic method of conducting a full fatigue study in Ansys is crucial in guaranteeing the dependability and durability of the front axle, particularly when subjected to different operational situations. Ultimately, this analysis plays a significant role in enhancing the safety and performance of the truck.

4.1 Factor of Safety

The fatigue study of a truck's front axle, subjected to an applied force of 1e+005N as illustrated in the Ansys figure, involves the examination of the safety factor as a crucial factor in assessing the component's reliability when subjected to stress. The safety factor, ascertained through the analysis, provides insight into the level of safety between the applied load and the maximum stress that the axle can sustain prior to potential failure.

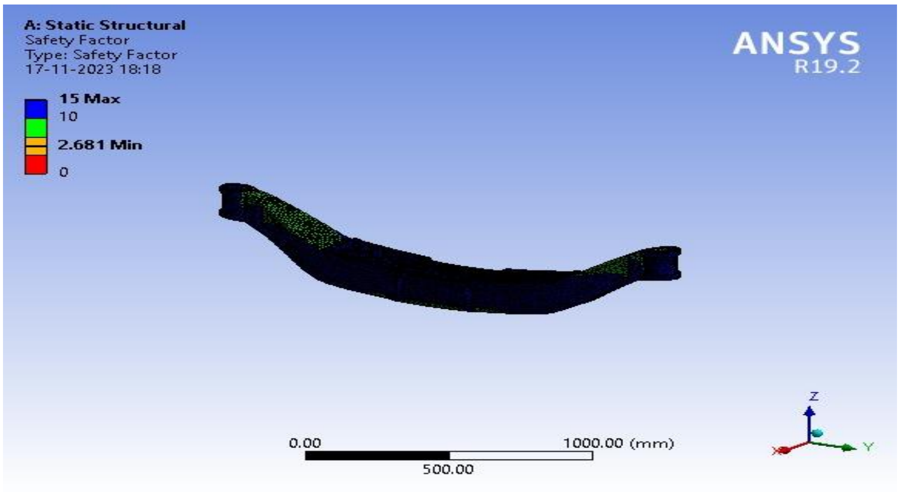

Figure 6. Safety of Factor

In the conducted analysis, the safety factor was determined to have a maximum value of 10 and a minimum value of 2.681. The assessment of the strength and endurance of the front axle relies heavily on the crucial safety factor values. A safety factor of 10 at its utmost indicates a significant disparity between the applied force and the maximum stress encountered by the axle. This observation implies a substantial degree of safety and resilience, suggesting that the axle possesses the capacity to withstand loads that exceed the applied force by a significant margin, without the occurrence of failure under optimal circumstances. In contrast, the minimum safety factor of 2.681, while lower than the maximum value, signifies a degree of safety wherein the imposed load is more than 2.5 times less than the maximum stress threshold that the axle can endure prior to experiencing failure.

4.2 Life

Under the specified parameters where an applied force of 1e+005 Newtons is considered, the fatigue analysis conducted on the front axle reveals a maximum estimated lifespan of 1e6 cycles. This analysis suggests that the axle has the potential to endure the stated load repeatedly for up to 1,000,000 cycles before the likelihood of fatigue-induced failure becomes a significant concern. The determination of a maximum lifespan of 1,000,000 cycles indicates the approximate threshold at which fatigue-related concerns regarding potential failure become prominent. This estimation is crucial in understanding the durability and endurance of the axle under the specified loading conditions. It offers valuable insights into the component's expected service life, highlighting the point at which the accumulated fatigue damage may reach a critical level, potentially leading to failure.

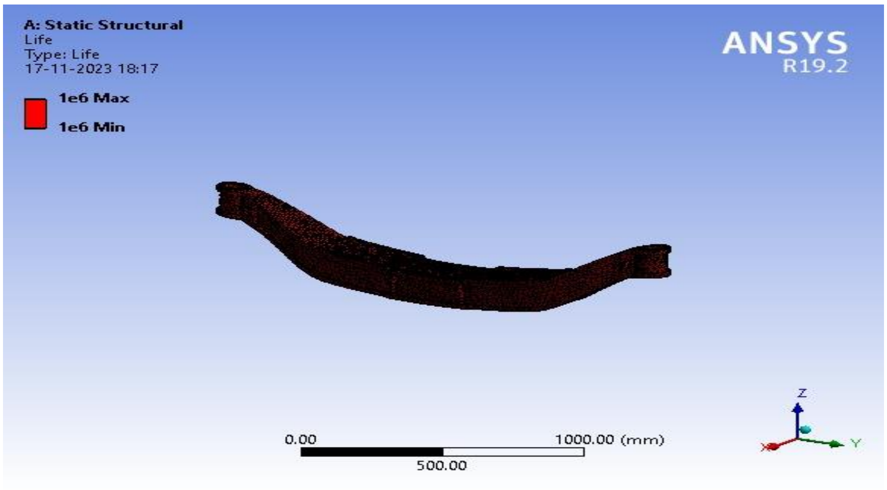

Figure 7. Life of axle

Fatigue analysis, the term "life" or "endurance" of a component pertains to the quantity of cycles it can endure when subjected to the imposed loading conditions prior to reaching a critical juncture where failure becomes a possibility. In this scenario, the maximum lifespan of 1e6 cycles serves as an approximation of the duration during which the axle can withstand the designated force prior to the possibility of failure due to fatigue. Hence, by integrating this data with the aforementioned safety factor values (with a maximum safety factor of 10 and a minimum safety factor of 2.681), a comprehensive comprehension of the front axle's durability can be achieved. This analysis reveals the safety margins and approximates the number of load cycles the axle can sustain before potential fatigue failure, considering the specific operating conditions.

4.3 Damage

Fatigue analysis, the term "damage" signifies the gradual accrual of fatigue-induced harm within a component subjected to cyclic loading over its operational lifespan. As a component endures cyclic stress, this continuous loading results in incremental damage accumulation. Upon surpassing a critical threshold, the likelihood of the component experiencing failure increases significantly. In the context of analyzing the front axle, the cumulative damage resulting from cyclic loading under specified conditions has been quantified, revealing a maximum damage value of 1000. This value of 1000 denotes the culmination of fatigue-induced damage experienced by the front axle throughout its exposure to cyclic loading induced by the applied forces. It serves as an indicator of the extent of accumulated fatigue damage within the axle, reflecting the severity of stress cycles encountered during its operational life. As this cumulative damage approaches or surpasses critical thresholds, the component's risk of failure escalates, signifying a point where the structural integrity of the axle might be compromised. The assessment of damage, often quantified using various fatigue damage accumulation models such as Miner's rule or other methodologies, provides crucial insights into the component's endurance and remaining service life.

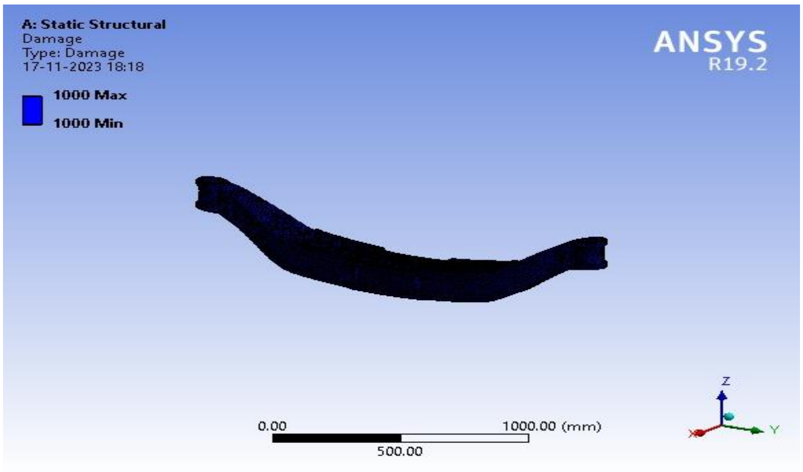

Figure 8. Damage of Axle

Engineers commonly employ damage accumulation models, such as Miner's rule or other established methodologies, to assess the cumulative impact of multiple loading cycles on the overall fatigue damage. When the cumulative damage surpasses or equals a critical threshold of 1 (representing 100% damage), it indicates that the component has reached its fatigue life and may be prone to failure.

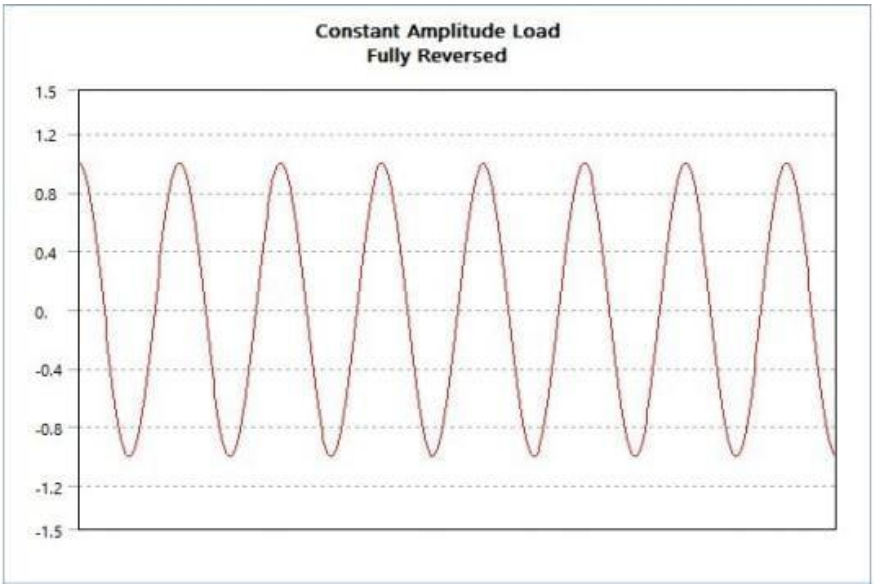

Figure 9. Fatigue analysis at constant amplitude load

Fatigue analysis, a constant amplitude load fully reversed from +1 to -1 represents a loading cycle that varies between maximum and minimum load values symmetrically around a mean or zero load. The load fluctuates cyclically from a maximum value of +1 to a minimum value of -1, creating a fully reversed loading condition.

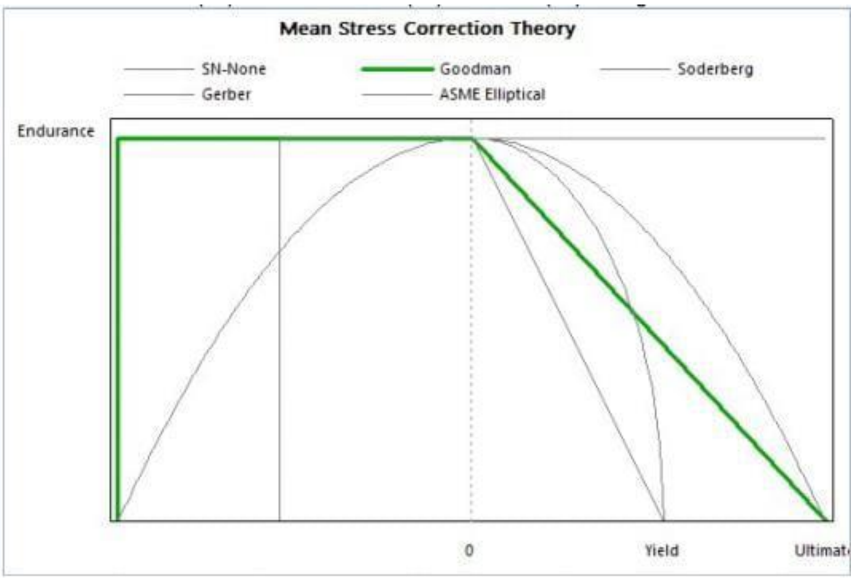

Figure 10. Stress vs Strain Curve

In the domain of fatigue analysis using Finite Element Analysis (FEA), the Mean Stress Correction theory holds significant importance in understanding how materials respond to cyclic loading conditions and predicting a component's fatigue life. This theory delves into the relationship between stress and strain, depicting the material behavior as it undergoes varying stress levels.Within the stress-strain curve, stress and strain exhibit a linear correlation until the material reaches its elastic limit. During this phase, the material behaves elastically, signifying that stress and strain are directly proportional within the elastic range. As stress increases, strain also increases proportionally, illustrating a linear behavior until the point of elastic limit.

5. Conclusion

The conducted fatigue analysis utilizing Ansys software has provided crucial insights into the structural behavior and durability of the front axle under the applied forces in a trucking scenario. Through meticulous modeling, precise simulations, and rigorous analysis, the study has elucidated critical aspects of the axle's performance. The safety factor analysis revealed a substantial margin of safety, with a maximum safety factor of 10, ensuring a considerable level of reliability and robustness of the axle under the specified loading conditions. However, attention is warranted in areas where the safety factor approached its minimum value of 2.681, indicating potential vulnerability to fatigue-related issues. The estimation of the maximum life of the axle, reaching 1e6 cycles, signifies its ability to endure the applied forces before reaching a critical fatigue failure point. This valuable insight into the expected lifespan of the axle underlines its endurance and reliability within operational limits. The assessment of maximum damage, reaching 1000, indicates the extent of accumulated fatigue damage experienced by the axle due to cyclic loading. This finding aids in understanding the component's remaining fatigue life and facilitates informed decision-making regarding maintenance, repairs, or potential design enhancements to mitigate risks associated with fatigue failure.

References

1. Ranjan, A., Mutra, R. R., Kirty, Y., Srinivas, J., Norhisham, M., & Reddy, D. M. (2023). Study and failure analysis of non-drive automotive rear axle of heavy commercial vehicle. *International Journal of Heavy Vehicle Systems*, *30*(1), 1-16.
2. Wen, C. K., Liu, Z. Y., Wu, G. W., Zhao, C. J., Chen, L. P., Yin, Y. X., & Meng, Z. J. (2023). Digital twin-driven fatigue life prediction framework of mechanical structures using a power density theory: Application to off-road vehicle front axle housing. *Measurement*, *220*, 113352.
3. Yavuz, I. (2023). FAILURE ANALYSIS OF A TRACTOR FRONT AXLE. *Materials and Technology*, *57*(2), 163-167.
4. Sivaraman, P., Ilakiya, P., Prabhu, M. K., Ajayan, A., & Nithyanandan, T. (2023). *Optimizing Front Axle Design for Heavy Commercial Vehicles: A Comprehensive Analysis of Structural and Mechanical Properties* (No. 2023-01-5076). SAE Technical Paper.
5. Berra Widén, E. (2023). Structural Analysis and Redesign of a L90 Front Axle Casing.
6. dos Reis, R. D. O., dos Santos, R. O. S., de Souza, V. B. F., Manenti, V. C., Lessa, F. F., Saveljevas, I. R., ... & Pfeifer, L. (2023). *Front Spring Development & Validation for Medium Duty Truck* (No. 2023-36-0368). SAE Technical Paper.
7. Kınıt, U., & Dikmen, F. (2023). Dynamic analysis for high-speed train hollow power axle. *Journal of Mechanical Science and Technology*, *37*(2), 583-594.
8. Zhang, L., Jiang, B., Zhang, P., Yan, H., Xu, X., Liu, R., ... & Ren, C. (2023). Methods for fatigue-life estimation: A review of the current status and future trends. *Nanotechnology and Precision Engineering (NPE)*, *6*(2).
9. Zhou, D., & Chang, J. (2022). Fatigue Analysis of a Light Truck Rear Axle Based on Virtual Iteration Method. *Shock and Vibration*, *2022*.
10. Stańco, M., & Kowalczyk, M. (2022). Analysis of experimental results regarding the selection of spring elements in the front suspension of a four-axle truck. *Materials*, *15*(4), 1539.
11. Shinde, J., Kharade, S., Mulani, A., Kokare, K., Tapase, C., & Terani, A. (2022). Design and analysis of front axle using Solidworks simulation.
12. Koh, Y., Ceylan, H., Kim, S., & Cho, I. H. (2022). Structural and fatigue analysis of jointed plain concrete pavement Top-down and bottom-up transverse cracking subjected to superloads. *Transportation Research Record*, *2676*(9), 76-93.
13. Verma, R. P., & Pant, S. (2021). Fatigue life prediction of front axle of truck at different crack directions using ANSYS. In *Advances in Engineering Design: Select Proceedings of FLAME 2020* (pp. 243-250). Springer Singapore.
14. Avikal, S., Bisht, A., Sharma, D., Hindwan, H., Yadav, S., Kumar, K. N., & Thakur, P. (2020). Design and fatigue analysis of front axle beam of a heavy duty truck using ansys. *Materials Today: Proceedings*, *26*, 3211-3215.
15. Maddewad, K., Jadhav, T., Bhosale, A., Yemle, S., & Jadhav, N. (2017). Optimization of front axle for heavy commercial vehicle by analytical and FEA method. *Int Res J Eng Technol (IRJET)*.
16. Dhande, K. V., & Ulhe, P. (2014). Design and analysis of front axle of heavy commercial vehicle. *Int J Sci, Technol Manag*, *3*.

17. Lu, S. K., Su, J. H., Liao, S. D., Su, J. Q., Wang, B., Yu, L., ... & Wen, S. H. (2012). Finite element analysis on Fatigue failure prediction of a rear axle housing of Vehicle based on Cosmos. *Applied Mechanics and Materials*, *121*, 843-847.

18. Zheng, S. M., Song, S. X., & Sun, Y. (2012). Fatigue life analysis and structural optimization of drive axle housing based on FE-SAFE. *Advanced Materials Research*, *472*, 419-426.

19. Tarighi, J., Mohtasebi, S. S., & Alimardani, R. (2011). Static and dynamic analysis of front axle housing of tractor using finite element methods. *Australian journal of Agricultural Engineering*, *2*(2), 45-49.