Biomedical and Mechanical Engineering Journal (BIOMEJ)
e-ISSN: 2776-1983, p-ISSN: 2829-5242
Vol. 5, No.1, May 2025, pp 11-21

Finite Element Analysis of Static Failure in a Tempered and Hardened 1040 Steel Shaft

R Ruviana[1], A N Izzah[1,*], M R Firdaus[1], S T Darmawan[1], K F Nugroho[1], W Illaahiyah[1], and I Y Pratama[1]

[1] Department of Mechanical Engineering, Faculty of Engineering, University of Pembangunan Nasional Veteran Jawa Timur, 60294, Indonesia

*Corresponding e-mail: anisahni.ft@upnjatim.ac.id

Abstract. This study investigates the static load failure behavior of a shaft made from tempered and hardened AISI 1040 carbon steel using both analytical calculations and numerical simulations. The shaft, with a diameter of 30 mm and length of 350 mm, was subjected to a vertical force of 800 N and torsional moments of 190 Nm and 50 Nm at different ends. Mechanical properties of the material, including a yield strength of 659.10 MPa and ultimate strength of 892.70 MPa, were used to support stress analysis. Analytical methods based on classical mechanics were applied to calculate bending stress, shear stress, von Mises stress, and safety factors. These were then validated through finite element simulation using ANSYS Static Structural. Results showed that the maximum von Mises stress from simulation was 51.5 MPa, significantly below the material's yield strength, while the analytical calculation gave 71.7 MPa. The maximum shear stress was 26.2 MPa in the simulation versus 35.9 MPa analytically. Total deformation reached 0.546 mm in simulation, and 0.0848 mm from manual calculation. The equivalent elastic strain was also within elastic limits in both methods. The safety factor was 12.789 from simulation and 9.27 from manual calculation, indicating a highly safe design. Additionally, the fatigue life analysis revealed the shaft could withstand up to 100 million load cycles without failure. These findings confirm that the shaft remains structurally and functionally safe under the given static and cyclic load conditions.

Keyword: Static load failure; Finite Element Analysis; AISI 1040 carbon steel.

1. Introduction

Shafts are fundamental components in mechanical power transmission systems, serving as the medium to transfer torque and various forces between machine elements such as gears, pulleys, couplings, and belts. In real engineering applications, shafts are frequently subjected to a combination of loads, including torsion, bending, axial tension, and compression. These complex loading conditions result in intricate stress distributions across the shaft cross-section, which, if not properly considered, can lead to mechanical failure [1]. Shaft failure may not only cause damage to the component itself but can also lead to the breakdown of the entire system, posing safety risks and economic losses. The reliability of a shaft in operation greatly depends on material selection and the application of proper heat treatment processes. One commonly used material in shaft manufacturing is AISI 1040 carbon steel, which belongs to the medium-carbon steel group. This material is favored due to its balanced mechanical properties, particularly when enhanced by heat treatment processes such as hardening and tempering. The hardening process typically increases the tensile strength and hardness of the material by inducing a martensitic microstructure. However, this microstructure, while hard, is also brittle and susceptible to cracking. To address this, tempering is applied afterward to improve ductility and toughness by relieving residual stresses and adjusting the microstructure [2, 3]. While heat treatment processes offer significant advantages in improving mechanical performance, the alterations in microstructural characteristics also influence the material's failure behavior, especially under static loading conditions. Static load failure is often triggered by stress concentrations, geometric discontinuities, or microstructural flaws that may not be visible to the naked eye. Therefore, it is crucial to perform a comprehensive analysis combining theoretical approaches—based on mechanics of materials—and numerical simulations using Finite Element Method (FEM) software such as ANSYS. This integrated approach allows engineers to predict stress distribution patterns and locate potential failure points with greater accuracy [4, 5]. The primary objective of this study is to investigate static failure behavior in a shaft made of AISI 1040 carbon steel, which has undergone hardening and tempering heat treatment. The analysis incorporates both manual calculations based on classical mechanical theory and numerical simulations performed using ANSYS software. Through this dual-method approach, the study aims to validate theoretical predictions with numerical results and provide a deeper understanding of the mechanical response of heat-treated shafts under static loads. Given the widespread use of AISI 1040 steel shafts in industries such as automotive, heavy machinery, and manufacturing, ensuring their structural integrity under working conditions is essential. The insights obtained from this study are expected to contribute to the design and analysis of more reliable, safe, and efficient mechanical components, particularly in environments where failure could lead to critical operational consequences.

2. Literature Review

A comprehensive body of literature has addressed the various mechanisms that lead to shaft failure under mechanical loading. One of the most prevalent causes is the combined effect of bending and torsional stresses, which often surpass the material's elastic limit and initiate crack propagation or plastic deformation. According to [1] such combined stresses should be evaluated using equivalent stress criteria such as the von Mises criterion or the maximum shear stress theory. These criteria are crucial in assessing multiaxial stress states that occur in real-world shaft applications. Fatigue failure, particularly in components with stress concentrators such as notches or threads, has been a subject of extensive study. Schneider et al. conducted fatigue analysis on threaded joints using the local strain approach. Their findings revealed that regions of stress concentration serve as initiation points for fatigue cracks, which often propagate rapidly under cyclic loading. This underscores the importance of accurately identifying stress risers during shaft design [6]. The application of the Finite Element Method (FEM) has significantly advanced failure analysis in modern engineering. Beyond traditional analytical approaches, FEM enables detailed stress analysis in components with complex geometries and boundary conditions, which are often challenging to model accurately using closed-form solutions. Karayan et al. (2012) demonstrated through several mechanical failure case studies that FEM is capable of identifying localized stress concentrations, crack initiation zones, and deformation behavior that are not readily captured by theoretical calculations alone [7]. Similarly, Sivák et al. (2023) conducted a comparative study between FEM predictions and both experimental and analytical results, confirming the reliability of FEM in identifying critical stress zones in axially symmetric components with shape discontinuities [8]. These studies reaffirm the value of FEM in structural analysis, especially when failure risks are strongly influenced by local geometry and boundary conditions. Furthermore, a review published in *Procedia Structural Integrity* (2021) highlighted the role of FEM in simulating fatigue life and cyclic loading conditions, showing that the method is essential for accurate prediction of structural integrity under dynamic and repetitive stresses [9]. These findings collectively support the conclusion that FEM-based simulations are indispensable in modern mechanical design and failure prevention, particularly for components like shafts, where complex loading interactions and geometry-induced stress risers are common. In the specific context of rotating shafts, the integration of analytical and numerical approaches enhances the robustness of stress evaluations. For example, de Assis et al. (2023) investigated the influence of fillet geometry on stress concentration in shaft keyways under bending loads using FEM simulations [10]. They demonstrated that minor modifications to fillet shape can significantly reduce local stress peaks—findings that are difficult to predict using simplified analytical expressions alone Similarly, a study conducted by Engel et al. (2018) employed FEA to simulate shaft deflection and stress under three-point bending, showing excellent agreement with experimental measurements and reinforcing the reliability of FEM in capturing elastic behavior and stress distribution in shaft structures [11]. These cases collectively corroborate the importance of combining manual calculations with FEM analysis. While analytical methods offer initial estimations, FEM provides deeper insights into the effects of geometric discontinuities, boundary supports, and load locations—resulting in more accurate and dependable predictions of shaft performance under complex load conditions. In the context of shaft analysis using ANSYS or similar FEM tools, M. Majid et al. (2024) conducted a mesh convergence study on a 2 kW induction motor shaft and found that the precision of von Mises stress and deformation predictions strongly depends on mesh size. They observed that finer mesh resolutions (e.g. 1 mm vs 3 mm vs 2 mm element sizes) produce more stable and converged results, especially around bearing seats and rotor support areas [12] Moreover, Yaqin et al. (2021) addressed the critical role of safety factors in the design of rotating components, especially those subjected to dynamic and cyclic loads [13]. They advocated for the careful consideration of fluctuating stresses, which may lead to fatigue failure even when nominal stress values are within allowable limits. In a complementary study, Wibawa (2019) emphasized the necessity of evaluating deformation and elastic strain to ensure that the material operates within its elastic range [14] . This is particularly important in components that undergo repeated loading, where exceeding elastic limits may compromise structural integrity over time. Together, these studies form the foundation for understanding how shafts behave under real-world loading conditions and underscore the necessity of using both analytical and numerical approaches for accurate and safe mechanical design.

3. Materials and Methods

3.1. Material Properties

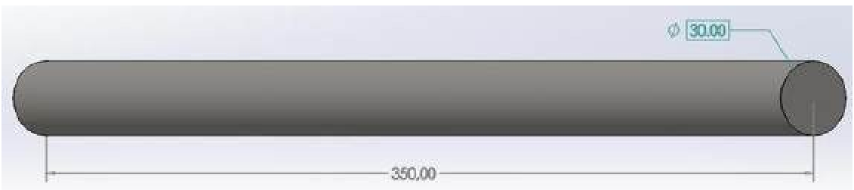

Figure 1. Workpiece Dimensions

The material utilized in this study is AISI 1040 medium-carbon steel, selected for its favorable balance of strength, hardness, and manufacturability after heat treatment. AISI 1040 is widely used in high-stress mechanical applications such as shafts, axles, and machine components due to its ability to undergo significant improvements in mechanical properties through hardening and tempering processes. The chemical composition of AISI 1040 steel primarily consists of carbon (C) ranging between 0.37–0.44% and manganese (Mn) between 0.60–0.90%, with the remainder being iron (Fe). The relatively high carbon content contributes to increased hardness and tensile strength post heat-treatment, while manganese improves toughness and hardenability (ASM International, 1990). Heat treatment was applied in the form of hardening followed by tempering. The hardening process involves heating the steel above its critical transformation temperature, followed by rapid quenching to form a martensitic structure. Subsequently, tempering was conducted at a controlled temperature to relieve residual stresses and improve ductility while retaining an elevated hardness level. This treatment makes the material suitable for components that experience significant static and cyclic loads. The mechanical properties of the heat-treated AISI 1040 used in this research, based on ASM Handbook and supporting technical data, are as follows:

Table 1. Mechanical Properties of the heat-treated AISI 1040

Yield Strength	: 659.10 MPa
Ultimate Tensile Strength	: 892.70 MPa
Young's Modulus	: 212.39 GPa
Density	: 7850 kg/m³
Poisson's Ratio	: 0.29

These values indicate that the material possesses a strong resistance to plastic deformation and offers sufficient stiffness for structural integrity under combined torsional and bending loads.

3.2. Geometry and Boundary

The shaft was modeled as a solid cylinder with a uniform diameter of 30 mm and a total length of 350 mm. This simplified geometry allows for accurate stress analysis while capturing the essential characteristics of a real shaft used in mechanical applications.
The loading conditions applied to the model represent typical real-world mechanical loads encountered by rotating and static shafts:

- A compressive axial force of 800 N is applied at the midpoint of the shaft.
- A moment of 190 Nm is applied at the fixed end (left side), simulating a resisting torque from a mechanical support or housing.
- A rotational torque of 50 Nm is applied at the free end (right side), representing an external drive or loading mechanism.

These boundary and loading conditions are intended to produce a combination of axial stress, bending stress, and torsional shear stress for a comprehensive analysis of shaft behavior under static loading.

3.3. Finite Element Simulation Setup

The simulation was conducted using the Static Structural module in ANSYS Workbench R2, which is widely used for analyzing linear static problems in mechanical structures. The analysis procedure included model import, meshing, boundary condition definition, material assignment, and post-processing.

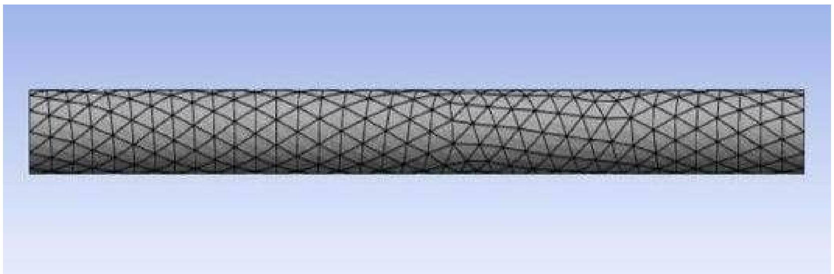

Figure 2. Meshing and Geometry Modelling

High-accuracy meshing was achieved using hexahedral (brick) elements, which are preferred for cylindrical geometries due to their superior accuracy in capturing stress gradients compared to tetrahedral elements. Mesh refinement was applied especially at areas of anticipated stress concentration, including the middle and ends of the shaft. A mesh convergence study was performed to ensure result accuracy and independence from mesh size, resulting in a total of approximately 120,000 elements.

- The left end of the shaft was fully constrained, preventing all degrees of freedom.

- The axial compressive force (800 N) was applied vertically at the center of the shaft to simulate static central loading.
- A moment of 190 Nm was applied at the fixed end to simulate bending and reaction torque.
- A rotational moment of 50 Nm was applied to the right end, simulating shaft drive loading. These combined loads are designed to mimic real operating conditions, which often involve multiple simultaneous mechanical stresses. The analysis used a direct sparse solver with displacement and force convergence criteria set to 1e-4. The shaft was evaluated using the von Mises stress criterion to identify areas with the highest risk of yielding. In addition, total deformation and elastic strain were observed to assess whether the material remained within the elastic region.

3.3. Calculations

3.3.1 Equivalent Stress (Von-Misses)

$$\sigma_y = \frac{M\,c}{I}$$

$$\tau = \frac{T\,c}{J}$$

$$I = \frac{\pi d^4}{64}$$

$$J = \frac{\pi d^4}{32}$$

$$\sigma_v = \sqrt{(\sigma_y)^2 \times 3(\tau_{xy})}$$

3.3.2 Maximum shear Stress

$$\tau_{max} = \sqrt{\left(\frac{\sigma_y}{2}\right)^2 + (\tau_{xy})^2}$$

3.3.3 Equivalent Elastic Strain

$$\varepsilon_{eq} = \frac{\sigma_v}{E}$$

3.3.4 Total Deformation

$$\delta_{max} = \frac{FL^3}{48EI}$$

3.3.5 Safety Factor

$$n = \frac{S_y}{\sigma_v}$$

Table 2. Mechanical Properties of the heat-treated AISI 1040

Symbols	Descriptions
σ_y	Maximum normal stress due to bending
M	Bending moment
c	Distance from the neutral axis to the outer surface usually ($c = \frac{d}{2}$)
I	Moment of inertia of the cross-section
J	Polar moment of inertia of the cross-section
τ_{xy}	Shear stress
T	Torque (torsional moment)
d	diameter

Biomedical and Mechanical Engineering Journal (BIOMEJ)
e-ISSN: 2776-1983, p-ISSN: 2829-5242
Vol. 5, No.1, May 2025, pp 11-21

σ'	Equivalent (von Mises) stress
ε_{eq}	Equivalent elastic strain
σ_v	Von Mises stress
E	Young's modulus
L	Length
S_y	Yield strength

4. Result and Discussions

4.1. Loading and Support Configuration

After the meshing process was completed, boundary conditions and loading scenarios were defined. A fixed support was applied at the left end of the shaft (Point A), a compressive force of 800 N was applied at the center of the shaft (Point C), and a torsional moment of 50 Nm was applied at the right end (Point B). The fixed support constrains all translational and rotational movements, simulating a shaft embedded in a bearing housing or machine frame.

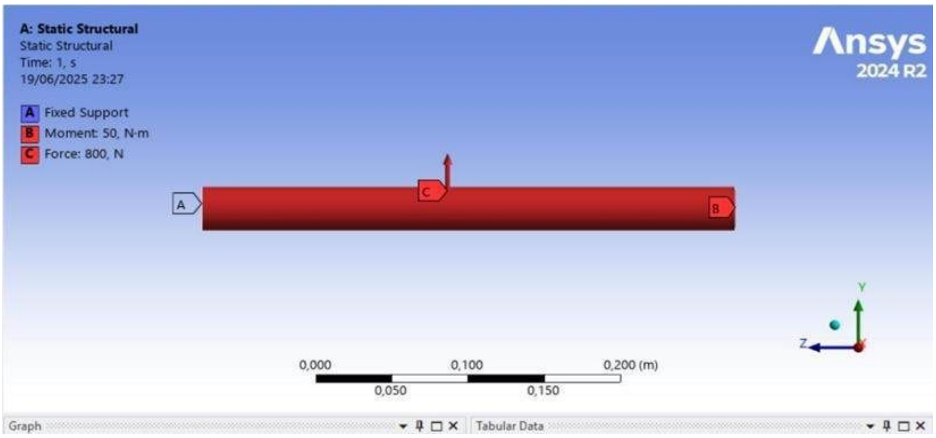

Figure 3. Loading and Support Configuration

The compressive force at the shaft's midpoint represents the load from components such as gears, pulleys, or other external forces acting perpendicularly to the shaft axis. Meanwhile, the torsional moment at the right end simulates torque loads from rotating systems such as motors or drive transmissions. This loading configuration closely approximates real-world mechanical applications, in which shafts are subjected to a combination of axial compressive forces and torsional moments simultaneously.

4.2. Equivalent (von-Mises) Stress

The von Mises simulation was employed to evaluate the magnitude of the combined stress (resulting from axial, compressive, bending, and torsional loads) acting at specific points within the material. Von Mises stress serves as a critical indicator for determining whether a material is likely to fail, particularly under complex loading conditions.

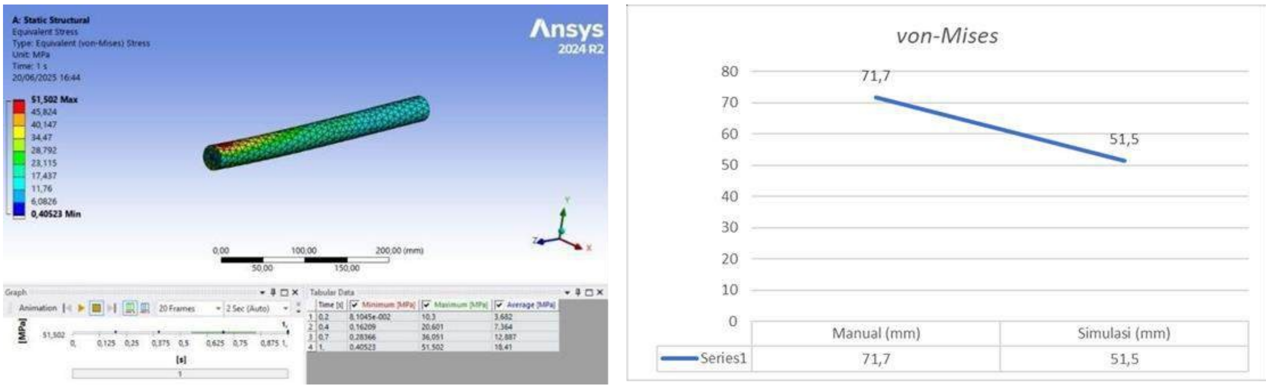

Figure 4. Equivalent (Von Mises) Stress Simulations and Stress Calculations

In this study, the von Mises simulation was conducted to ensure that the total stress experienced by the shaft remains below the yield strength, which represents the maximum limit before the material undergoes permanent deformation. The simulation results indicated that the maximum von Mises stress in the shaft reached 51.5 MPa. This stress was generated by the application of an 800 N axial load and a torsional moment at both shaft ends. When compared to the material's yield strength of 659.10 MPa, the resulting stress is significantly lower, indicating that the shaft remains safe and will not experience structural failure or permanent deformation under the given static loading. The manual calculation, in comparison, yielded a stress value of 71.7 MPa, which remains within a reasonable range. The discrepancy is considered minor and acceptable, primarily due to the simplifications used in manual calculations—typically involving basic combined loading formulas (bending and torsion)—while the ANSYS simulation incorporates more detailed numerical methods. It also considers local stress concentrations influenced by mesh refinement, load application points, and boundary conditions that more closely reflect real-world scenarios. Both the simulation and analytical results confirm that the shaft is capable of withstanding the applied load, as the calculated stress remains well below the yield strength limit.

4.3. Maximum Shear Stress

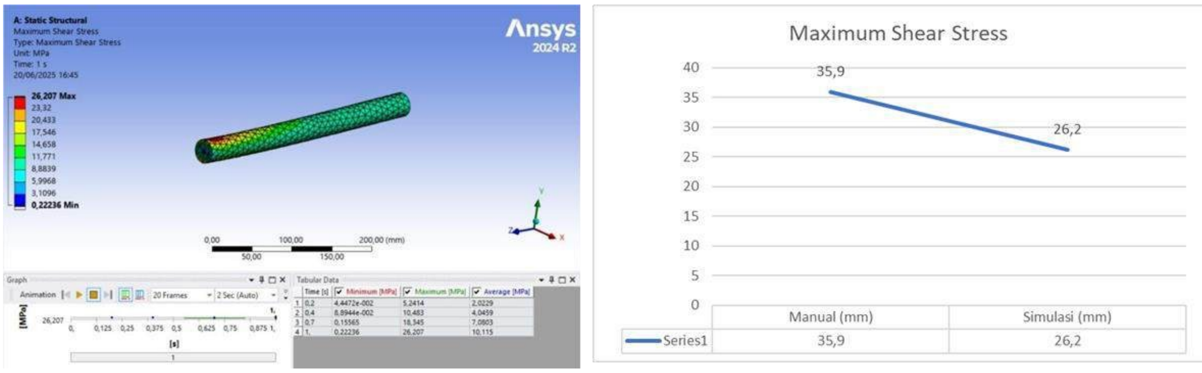

Figure 5. Maximum shear stress Simulations and Calculations

Maximum shear stress simulation provides an accurate representation of the shear stress distribution throughout the shaft. The ANSYS simulation results show that the maximum shear stress occurring in the shaft is 26.2 MPa, indicated by the red color on the stress contour. This value results from the combined loading of a transverse force of 800 N and torsional moments applied at both shaft ends (190 Nm on the left and 50 Nm on the right). This stress value is crucial for analysis as it highlights the potential for shear failure due to torsional loading. Compared to the manual calculation, which yields a maximum shear stress of 35.9 MPa, the difference is relatively small and within an acceptable range. This discrepancy is attributed to the fact that manual calculations often rely on idealized assumptions, such as uniform stress distribution across the cross-section, and typically do not account for stress concentrations or localized geometric effects. In contrast, the ANSYS simulation incorporates the actual shaft geometry, boundary conditions, mesh refinement, and the combined effects of force and moment interactions in a comprehensive manner. Overall, both the manual and simulation results confirm that the maximum shear stress remains well below the material's shear strength limit, indicating that the shaft is safe from shear failure. Thus, the ANSYS simulation serves as a strong validation of the manual analysis and provides a more detailed and realistic visualization of stress distribution within the component.

4.4. Total Deformation

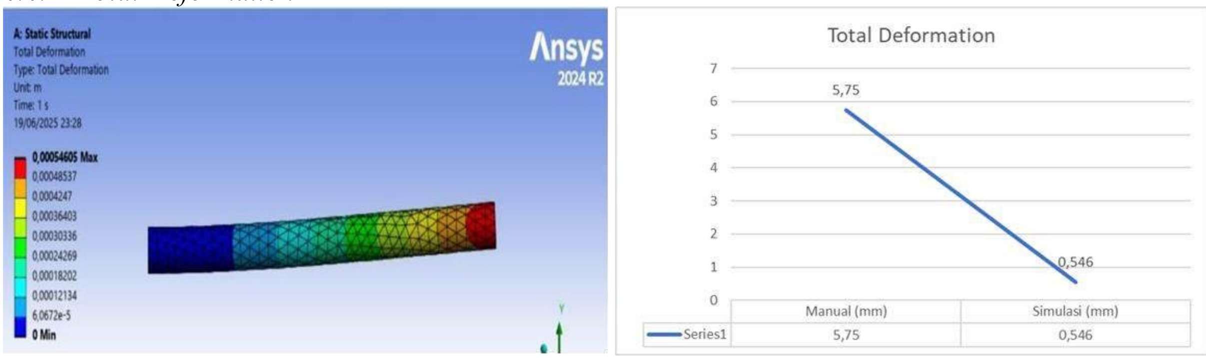

Figure 6. Total Deformation Simulations and Calculations

An important indicator for determining whether a material is capable of withstanding the applied load is deformation. Deformation occurs as a result of the material being subjected to external forces or loads. The smaller the deformation value, the stronger the material is considered to be (Wibawa, 2019). Simulation results indicate that the maximum deformation occurs at the right end of the shaft, with a value of 0.00054605 m or 0.54605 mm. The red color at the shaft tip in the simulation output represents the region with the highest deformation, resulting from the combination of an 800 N compressive force and a 50 Nm torsional moment applied at the end. In contrast, the area closest to the fixed support (left side) is shown in blue, indicating deformation close to zero. This value is considered relatively small for a steel shaft. Manual calculations show that the maximum bending stress is approximately 11.79 MPa, while the yield strength of tempered and hardened 1040 carbon steel reaches 659 MPa, meaning the stress remains far below the material's safe limit. Therefore, the shaft is still in a safe condition and has not experienced any damage. When compared to the manual calculation result of 0.0848 mm, the ANSYS simulation result of 0.546 mm shows only a small discrepancy. This difference arises because the manual calculation uses simplified elastic theory formulas, assuming a point load, a perfectly rigid shaft, and immovable supports. On the other hand, ANSYS employs the Finite Element Method (FEM), which divides the shaft into small elements and calculates the deformation of each element in detail. It also accounts for the interaction between bending, torsion, compression forces, and more realistic geometry and boundary conditions.

4.5. Equivalent Elastic Strain

The simulation results of equivalent elastic strain on the shaft illustrate the distribution of elastic strain due to the applied static loading. The color gradient in the model visualizes strain magnitude at each point, ranging from a minimum value of 4.46×10^{-6} m/m (represented by dark blue) to a maximum value of 2.43×10^{-4} m/m (bright red). The highest strain is observed on the left side of the shaft, near the support and reaction force area, which is consistent with the location of maximum moment and applied forces. Meanwhile, the strain gradually decreases toward the right end of the shaft, as indicated by the color transition from red to green and blue.

Biomedical and Mechanical Engineering Journal (BIOMEJ)
e-ISSN: 2776-1983, p-ISSN: 2829-5242
Vol. 5, No.1, May 2025, pp 11-21

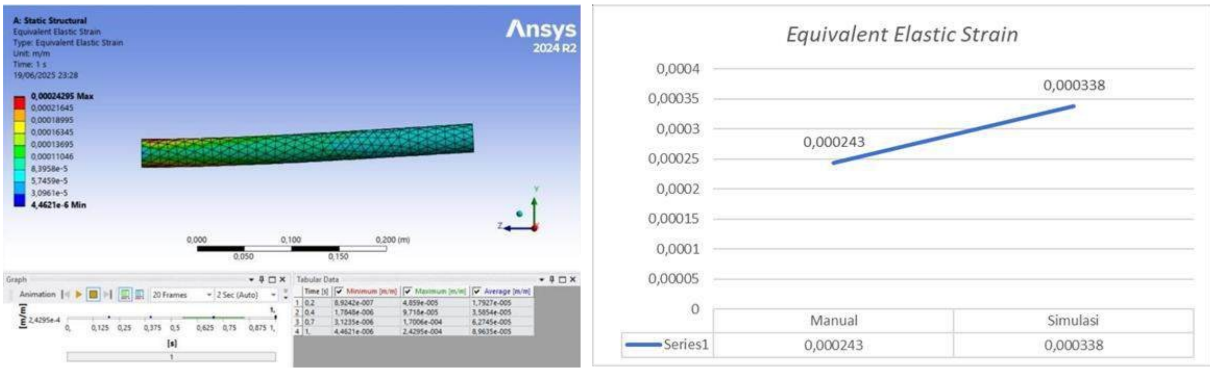

Figure 7. *Equivalent Elastic Strain* Simulations and Stress Calculations

The maximum equivalent elastic strain from the simulation, 2.43 × 10^{-4} m/m, compared to the manual calculation result of 3.38 × 10^{-4} m/m, shows a deviation of approximately 28.1%. This difference is considered reasonable, attributed to the simplifications in manual calculations and the more detailed loading and modeling conditions in the simulation, including mesh refinement and realistic boundary conditions. Equivalent elastic strain represents the combined multiaxial strain, calculated using the von Mises approach. It serves as a measure of the material's deformation within the elastic range (Sitepu, 2016). This parameter is critical for evaluating structural safety, identifying critical deformation zones, and serving as a reference in fatigue analysis and the potential for yielding or permanent deformation (Schneider et al., 2010). The simulation results provide a comprehensive depiction of the shaft's elastic deformation behavior and help ensure that the design remains within the material's elastic limit.

4.6. Safety Factor

Figure 8. *Safety factor of* Simulations and Calculations

The safety factor is a critical aspect in structural design and analysis, ensuring the reliability and safety of a given design. One of the most essential parameters in mechanical design and stress testing—particularly when a structure is subjected to external forces, whether tensile or compressive—is the safety factor itself (Yaqin, 2021). The safety factor value obtained from the simulation results confirms that the shaft design is safe under the given loading conditions. This is demonstrated by a safety factor greater than 1, which meets the general requirement for withstanding dynamic loading conditions (Pranoto, 2020). The ANSYS simulation shows that the shaft has a minimum safety factor of 12.789, indicating that the shaft is highly safe under the applied 800 N force and 50 Nm torque. This value is significantly above the commonly accepted safety threshold, which typically ranges between 1.5 to 3, thus eliminating the risk of structural failure. In comparison, the manual calculation yields a safety factor of 9.27, which is also well within the safe range. The discrepancy between the two results is acceptable, as manual methods generally rely on simplified formulas and specific assumptions, whereas simulation provides a more detailed and accurate representation based on actual shaft geometry and realistic load

Biomedical and Mechanical Engineering Journal (BIOMEJ)
e-ISSN: 2776-1983, p-ISSN: 2829-5242
Vol. 5, No.1, May 2025, pp 11-21

distribution. Therefore, it can be concluded that the shaft is structurally safe based on both analytical calculations and simulation results.

4.7. Life Cycle

The ANSYS simulation results, as shown in the figure, illustrate the fatigue life analysis of a cylindrical shaft under loading conditions. The uniform red coloration across the shaft surface indicates that the entire component has a very high fatigue life, reaching up to 1 × 10^8 cycles for both the minimum and maximum values. This result signifies that the shaft can withstand up to 100 million load cycles without experiencing permanent damage or crack initiation. A structure is generally considered to have a high service life if its fatigue life exceeds 1 × 10^3 cycles. In engineering analysis, fatigue life represents the number of loading cycles (such as rotation, vibration, or repeated pressure) that a material can endure before the initiation of microcracks or structural failure. The high value shown in this simulation confirms that the shaft design is highly resistant to fatigue failure, and the component remains structurally safe under repeated or cyclic loading conditions.

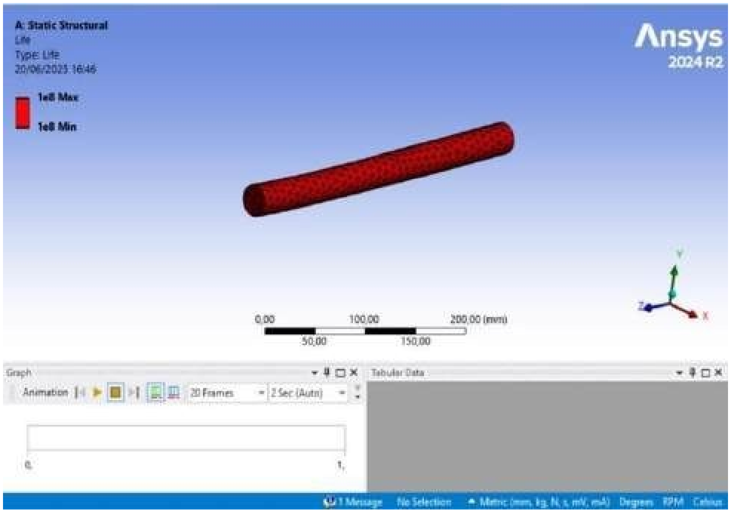

Figure 9. *Life Cycle*

5. Conclusion

Based on ANSYS R2 simulation and manual calculations using parameters such as von Mises stress, maximum shear stress, total deformation, equivalent elastic strain, and safety factor, the failure analysis of a tempered and hardened 1040 carbon steel shaft under static loading revealed that the shaft remains structurally safe under a force of 800 N and a moment of 50 Nm. The simulation results show a maximum von Mises stress of 51.5 MPa, significantly below the material's yield strength of 659.10 MPa, while the manual calculation yielded 71.7 MPa. The maximum shear stress was 26.2 MPa (simulation) and 35.9 MPa (manual), with acceptable deviation considering the idealized assumptions in manual analysis. The maximum total deformation from simulation was 0.546 mm, remaining within the elastic range, and consistent with the manual result of 0.0848 mm. The maximum equivalent elastic strain reached 2.43 × 10^{-4} m/m (simulation), lower than the 3.38 × 10^{-4} m/m from manual calculation, confirming that the shaft operates within the elastic region without permanent deformation. The safety factor obtained from simulation was 12.789, and 9.27 from manual calculation—both exceeding standard minimum thresholds, indicating a high safety margin. Furthermore, ANSYS life cycle analysis estimated the shaft's endurance up to 100 million loading cycles without failure. In conclusion, both numerical simulation and analytical results confirm that the shaft is structurally and functionally safe to withstand static and cyclic loading under its design conditions.

References

[1] R. G. Budynas, J. Keith Nisbett. 2015, *Shigley's Mechanical Engineering Design 10th Edition*.

[2] W. D. Callister, G. D. Rethwish. 2017, "Materials Science and Engineering An Introduction."

[3] G. E. Totten. 2006, *Steel Heat Treatment Metallurgy and Technologies*. CRC Press.

[4] R. C. Hibbeler. 2017, *Mechanics of Materials Tenth Edition in SI Units*. www.pearsonglobaleditions.com/hibbeler.

[5] Saeed. Moaveni. 2015, *Finite element analysis : theory and application with ANSYS*. Pearson Education.

[6] R. Schneider, U. Wuttke, C. Berger. 2010, "Fatigue analysis of threaded connections using the local strain approach." *Procedia Engineering*. vol. 2, no. 1, 2357–2366.

[7] A. Ivan, D. Ferdian, S. Harjanto, D. Marta, A. Ashari, H. Castane. 2012, *Finite Element Analysis Applications in Failure Analysis: Case Studies*.

[8] P. Sivák, I. Delyová, J. Bocko. 2023, "Comparison of Stress Concentration Factors Obtained by Different Methods." *Applied Sciences*. vol. 13, no. 24, 13328.

[9] A. Fajri, A. R. Prabowo, N. Muhayat, D. F. Smaradhana, A. Bahatmaka. 2021, "Fatigue Analysis of Engineering Structures: State of Development and Achievement." *Procedia Structural Integrity*. vol. 33, 19–26.

[10] P. B. N. De Assis, C. F. de F. Moura Júnior, I. O. Andrade, W. N. Da Silva, R. R. B. Medeiros. 2023, "Analysis of the influence of fillet geometry on stress concentration in keyway by the finite element method." *OBSERVATÓRIO DE LA ECONOMÍA LATINOAMERICANA*. vol. 21, no. 11, 22365–22375.

[11] B. Engel, S. S. H. Al-maeeni. 2018, "FE analysis and experimental determination of a shaft deflection under three-point loading." *Vibroengineering Procedia*. vol. 19, 199–204.

[12] L. Hotma, N. C. Majid, Marsalyna, J. Hendrawan, N. A. Kinasih, D. J. Febriansyah, B. Novariawan, M. R. Alfin, I. S. Putri, M. T. Afif, B. N. Fadjrin, E. D. Purnomo. 2024, "MESH CONVERGENCE TEST BY USING FINITE ELEMENT ANALYSIS (FEA) IN SHAFT LOADING OF 2 KW INDUCTION MOTOR." *Jurnal Rekayasa Mesin*. vol. 15, no. 3, 1271–1280.

[13] Yaqin, M., et al. (2021). "Consideration of Safety Factors in Rotating Components Under Dynamic Loads." *Journal of Mechanical Science and Technology*, 35(9), 4221–4229.

[14] Wibawa, R. (2019). "Elastic Deformation Assessment in Shafts Under Static and Cyclic Loads." *Jurnal Riset Mesin*, 6(2), 66–72.