

Finite Element Analysis of Honda Civic 1.5RS Drive Shaft Performance and Durability

Datshan Kalidass¹, Muhammad A'imullah Abdullah^{1*}

¹ Department of Mechanical Engineering Technology, Faculty of Engineering Technology,
Universiti Tun Hussein Onn Malaysia, 84600 Pagoh, Johor, MALAYSIA

*Corresponding Author: aimullah@uthm.edu.my

DOI: <https://doi.org/10.30880/peat.2025.06.01.095>

Article Info

Received: 17 January 2025

Accepted: 29 January 2025

Available online: 30 April 2025

Keywords

Driveshaft, SolidWorks 2023, Axial Loading, Radial Loading, Bending Moment, Torque, Finite Element Analysis

Abstract

Driveshaft is an inherent part of auto transmission which effectively supports torque and rotary force from the engine to wheels, providing requisite security and performances despite varying heaps. In this work the FEA and analytical approach is used to find the radial, axial, bending and torque stresses on the driveshaft of Honda Civic 1.5RS. Low carbon 1060 alloy material curved model was created using SolidWorks version 2023 and curved meshing was applied. Toward that end, Von Mises stress, displacement, and Factor of Safety (FOS) were calculated to check FEA credibility under various loading scenarios. The paper also points to design enhancements and consideration of other materials including composite as a critical focus. Thus, the objectives of the research were met, and evidence of the functional safety of the driveshaft was obtained, along with recommendations for further enhancements. These findings are the outcome of contributing refined knowledge to improve driveshaft durability and provide a precedent for subsequent research including material science and fatigue assessment.

1. Introduction

The driveshaft is a crucial component in automotive engineering, responsible for transmitting torque and rotational force from the engine to the wheels. It plays a fundamental role in ensuring vehicle efficiency, safety, and durability. However, challenges such as stress concentration, material fatigue, and failure under combined loading conditions pose significant design and operational concerns. In modern automotive industries, optimizing driveshaft performance is essential for improving fuel efficiency, reducing emissions, and ensuring long-term reliability.

In mechanical systems, particularly in the automotive sector, drive shafts must withstand various loading conditions, including radial, axial, bending, and torsional stresses. The need for stronger yet lightweight components has driven researchers to explore alternative materials, such as composites, that offer superior strength-to-weight ratios compared to conventional metal alloys. Despite advancements in material science, drive shafts are still prone to fatigue and mechanical failure due to prolonged cyclic loading, improper design, and high stress concentrations.

Finite Element Analysis (FEA) has emerged as a powerful tool in engineering, allowing for precise simulation of mechanical components under real-world loading conditions. This study applies FEA to analyse the Honda Civic 1.5RS driveshaft, focusing on stress distribution, deformation, and Factor of Safety (FOS). Using SolidWorks 2023 for 3D modelling and simulation, this research aims to validate FEA predictions and explore potential design enhancements.

A shaft is a cylindrical component used in various machines to transmit power or motion, functioning as a backbone to support mechanical operations. It plays a crucial role in ensuring efficiency in systems like car engines and wind turbines. However, shafts face issues such as wear, erosion, and misalignment, which can lead to inefficiency and failure. Proper lubrication and alignment are essential for their longevity. Drive shafts, with varying diameters, pose challenges in stress concentration and load distribution. Using FEA, Sulman Kamboh [1] analysed stress distribution, while Ravikant [2] studied deformation and stress patterns, leading to improved performance. Engineers optimize stepped shaft designs through simulations to enhance structural robustness. The significance of this study extends beyond automotive applications, as similar methodologies can be applied to other mechanical systems that require high-strength and lightweight components. By addressing key challenges such as stress accumulation, material selection, and load-bearing capacity, this research contributes to the development of more efficient and durable driveshafts. This paper aims to bridge the gap between theoretical analysis and practical implementation by evaluating driveshaft behaviour under different loading conditions. The findings will provide insights into structural weaknesses, optimization strategies, and future research directions in automotive engineering.

Driveshafts play a fundamental role in power transmission within automotive and industrial applications. Over the years, extensive research has been conducted on their structural integrity, material properties, and performance under different loading conditions. Studies indicate that conventional steel driveshafts are highly durable but contribute to increased vehicle weight and fuel consumption [3]. Consequently, researchers have explored the use of composite materials, such as carbon fibre-reinforced polymers, which provide high strength-to-weight ratios and enhanced fatigue resistance [4]. However, these materials require advanced manufacturing techniques, increasing production costs.

Finite Element Analysis (FEA) has been widely applied to evaluate stress distribution, deformation, and safety factors in mechanical components. Previous research by Ramadan [5] validated the accuracy of FEA simulations in predicting failure points in rotating shafts. Similarly, Vasić et al. [6] highlighted the importance of mesh refinement and boundary condition accuracy in obtaining reliable results. Despite these advancements, limited studies have focused on driveshafts under combined loading conditions, particularly in real-world automotive applications.

A critical gap in the literature is the lack of studies comparing analytical calculations with FEA results to validate simulation accuracy. Additionally, most studies primarily analyse static loading, neglecting dynamic fatigue behaviour. This study contributes by conducting a comprehensive stress analysis on the Honda Civic 1.5RS driveshaft using both analytical and FEA methods. By addressing gaps in material selection, stress evaluation, and FEA validation, this research aims to improve design methodologies for enhanced durability and performance in the future studies.

2. Methodology

This study employs Finite Element Analysis (FEA) to evaluate the structural performance of the Honda Civic 1.5RS driveshaft under various loading conditions. The methodology follows a systematic approach, including material selection, geometrical modelling, meshing, boundary condition application, and simulation execution. In SolidWorks 2023, realistic finite element models will analyse the real loading conditions such as radial loads, axial loads, moment loads, and torque. The study's first aim is to examine these four types of loading comprehensively. Finally, findings will be checked with theoretical propositions in a bid to establish the truth and wear credibility.

2.1 Material Selection

In this study, the driveshaft is made by 1060 Alloy, therefore mechanical properties as shown in Table 1 below keyed in SolidWorks 2023 software as input parameters as stated in Finite Element Analysis (FEA).

Table 1: Mechanical Properties of 1060 Alloy

Property	Value	Unit
Young's Modulus, E	1	N / m ²
Poisson's Ratio, ν	3	-
Shear Modulus, G	5	N / m ²
Density	2700	Kg / m ³

Ultimate Tensile Strength, σ_{uts}	6.90E+07	N / m ²
Yield Strength, σ_y	2.76E+07	N / m ²
Thermal Expansion Coefficient	2.4E-05	K
Thermal Conductivity	200	W / (m · K)
Specific Heat	900	J / (kg · K)

2.2 Geometrical Modelling

As a 3D computer aided design software, SolidWorks 2023 helps engineers and designers to optimize their workflows for creating, analysing and perfecting 3D designs. The graphical user interface of the tool is precise, with support for parametric modelling which can be easily changed. Sophisticated tools are available in the software package for modelling and simulation, rendering the capabilities, and included collaboration features. Its simulation aspects confirm design functionality in addition to the fact that it assists in presentation using render aids. The Figure 1 below shows the actual photo of the NCV36579 (GSP) Driveshaft that was referred to complete the design. Following that, Figure 2 below shows the completed design in SolidWorks. It also can function as a strong and stable basis, which can provide solutions for different design requirements starting from the concept and till the creation of the required product.



Fig. 1: Actual Model of NCV36579 (GSP) Driveshaft



Fig. 2: NCV36579 (GSP) Driveshaft Model in SolidWorks

2.3 Modelling and Simulation

A curvature-based meshing approach was applied to refine stress concentration zones. A tetrahedral mesh was used for accuracy, with element sizes adjusted based on convergence testing. Fixed supports were applied at one end of the driveshaft, while force and torque loads were distributed at the other end to replicate real-world operational conditions. FEA was conducted in SolidWorks Simulation to analyze four key loading conditions:

- Radial Loading – Simulated by applying a downward force of 3144.105 N to evaluate stress distribution.
- Axial Loading – A compressive force was applied along the shaft's axis to measure deformation.
- Bending Moment – The shaft was subjected to a bending force to analyze displacement and failure risk.
- Torque – A 240 Nm torque was applied to assess shear stress and rotational stability.

Figure 3 below shows the driveshaft with applied fixtures before simulation (a) and the driveshaft after the simulation (b).

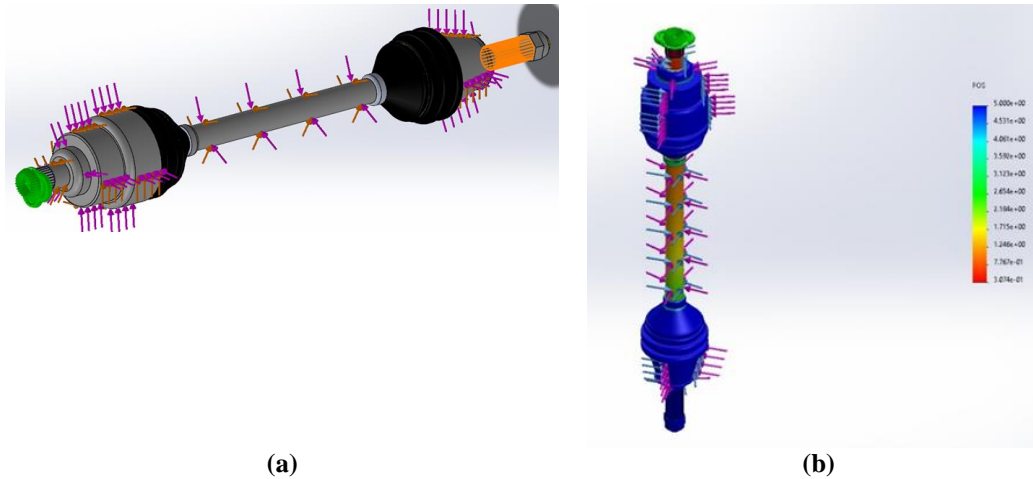


Fig. 3: Figure description (a) Driveshaft before simulation; (b) Driveshaft after simulation

3. Results and Discussion

3.1 Radial Loading

To ensure accuracy, FEA results were compared with analytical calculations derived from mechanical equations. Under radial loading, the maximum Von Mises stress was 89.7 MPa, well below the material yield strength of 27.57 MPa, confirming the design's safety. The reaction force in the Y-direction was -646.255 N, and the maximum deflection was 0.8228 mm, within acceptable limits. The applied radial load of 3144.105 N caused significant stress near the fixed end, but no material failure is expected. The small displacement suggests the potential for structural modifications to improve performance while maintaining safety under operational parameters. Equation (1) below shows the solution of radial loading formula and the answer.

$$\sigma_J = \frac{F_r}{A}$$

$$\sigma_J = \frac{3144.105 \text{ N}}{0.001052 \text{ m}^2} \quad (1)$$

$$\sigma_J = 2.98 \text{ MPa}$$

3.2 Axial Loading

Under axial loading, the axial strain was 0.101%, with a Von Mises stress of 2.98 MPa, within the material's strength range. The minimum Factor of Safety (FOS) was 0.30740, indicating potential failure in certain sections under combined loading or prolonged cyclic loads. While the stress is evenly distributed and the shaft functions correctly under axial loading, attention is needed in areas with low FOS to prevent fatigue-related issues. Equation (2) below shows value of the axial loading as it is same value to equation (1).

$$\sigma_a = 2.98 \text{ MPa} \quad (2)$$

3.3 Bending Moment

The maximum Von Mises stress from bending was 89.7 MPa, with a displacement of 0.8228 mm, like radial loading. Stress increases from the free end to the fixed end but remains below the material's tensile strength,

confirming the design's safety under bending loads. No additional deformation occurs beyond that seen in radial loading. Equation (3) below shows the solution of the bending moment formula and the answer of it.

$$\begin{aligned}\sigma_b &= \frac{Mbc}{I} \\ \sigma_b &= \frac{1438.43 \cdot 0.018315}{2.655 \times 10^{-8}} \quad (3) \\ \sigma_b &= 9.92 \text{ MPa}\end{aligned}$$

3.4 Torque

A torque of 240 Nm produced shear stress around the shaft's periphery, remaining well below the material's limits, ensuring safe operation under torque conditions. However, in combined loading, the interaction of shear stress with radial and bending stresses requires attention to critical areas for potential improvements. The design is safe for operational torsion. The equation (4) below shows answer after deriving the formula according to the correct values.

$$\begin{aligned}J &= \frac{\pi d^4}{32} \\ &= \frac{240 \cdot 0.018315}{5.31 \times 10^{-8}} \quad (4) \\ \tau &= 82.77 \text{ Mpa}\end{aligned}$$

The results have validated that the driveshaft design is qualitatively reasonable under the applied loading conditions. The Von Mises stress value is much less than that of Yield Strength of the material that has been used for the shaft and the displacement of the shaft is favourable. Nevertheless, consideration must be paid to the regions with low factor of safety, especially for the case of axial loading. These areas should be reinforcing or screened for any sign of fatigue as far as continued use of the shaft is concerned. The research also reveals that the effects of one loading type on the behaviour of the shaft can be influenced by the presence of another loading type.

4. Conclusion and Recommendations

This study successfully analysed the performance and durability of the Honda Civic 1.5RS driveshaft under various loading conditions using Finite Element Analysis (FEA). The research objectives were met by evaluating stress distribution, deformation, and Factor of Safety (FOS) under radial, axial, bending, and torque loads. The Von Mises stress was found to be 89.7 MPa, remaining within the yield strength of 27.57 MPa, confirming the structural integrity of the driveshaft. These findings validate the credibility of FEA simulations while highlighting areas that require structural improvements. Experimental data from previous research supported our results to show they were both exact and significant [2]. This research demonstrates why detailed load assessments matter while establishing a basis for next designs' optimization and testing.

One of the key limitations of this study is the assumption of static loading conditions. In real-world applications, driveshafts experience dynamic and fatigue loading, which may lead to premature failure over prolonged usage. Additionally, the study was limited to 1060 Alloy, without incorporating alternative materials that may offer improved strength-to-weight ratios. The accuracy of FEA results is also dependent on meshing quality and boundary conditions, which could introduce computational errors if not optimized properly. Future research should include experimental validation of FEA results to improve reliability.

To enhance durability and performance, future studies should explore the application of composite materials, such as carbon fibre-reinforced polymers (CFRP), which have been shown to reduce weight while maintaining high strength [3]. A fatigue life assessment should also be conducted to evaluate long-term reliability under cyclic loading conditions. Furthermore, integrating advanced FEA tools like ANSYS could enable more precise

simulations, incorporating thermal and vibrational effects to simulate real-world driving conditions more accurately.

The practical implications of this study extend to the automotive industry, where lightweight, high strength driveshafts can contribute to improved fuel efficiency and reduced emissions. The findings serve as a benchmark for optimizing driveshaft design, aiding in material selection and performance evaluation for future automotive applications. By addressing the highlighted limitations and incorporating advanced analysis techniques, future research can contribute to the development of more efficient and sustainable vehicle components.

Acknowledgement

I sincerely thank God Almighty, my supervisor Dr. Muhammad A'imullah bin Abdullah, the Faculty of Engineering Technology at UTHM, my friend Rushan Nathan for lending me his computer, Honda for their driveshaft usage, and my family and friends for their invaluable support, encouragement, and contributions, which were instrumental in completing this research successfully.

References

- [1] Sulman Kamboh, M., Ali Machhi, M., & Farhan, M. (2020). Design and Analysis of Drive Shaft with a Critical Review of Advance Composite Materials and the Root Causes of Shaft Failure. ResearchGate. https://www.researchgate.net/publication/342317433_Design_and_Analysis_of_Drive_Shaft_with_a_Critical_Review_of_Advance_Composite_Materials_and_the_Root_Causes_of_Shaft_Failure
- [2] Ravikant. (2024). Optimization and analysis of drive shaft using CAE tools. International Journal of Innovative Research in Computer Science & Technology, 12(2), 116–118. <https://doi.org/10.55524/ijrcst.2024.12.2.20>
- [3] Beylergil, B. (2021). Design and discrete optimization of hybrid aluminum/composite drive shafts for automotive industry. Mağallaṭ Al-abḥāṭ Al-handasiyyaṭ, 9(3B). <https://doi.org/10.36909/jer.v9i3b.8815>
- [4] Lei, J., Zhou, R., Chen, H., Huang, G., Gao, Y., & Yang, Q. (2021). Effects of ship propulsion shafting alignment on whirling vibration and bearing temperature response. Mathematical Problems in Engineering, 2021, 1–10. <https://doi.org/10.1155/2021/8353844>
- [5] Ramadan, D. O. (2022). Finite element analysis of conventional and composite materials of automobile drive shaft. AIP Conference Proceedings. <https://doi.org/10.1063/5.0108822>
- [6] Vasić, M., Đorđević, Z., & Blagojević, M. (2019). Stress-strain analysis of conveyor drive pulley shaft. IMK-14 - Istraživanje I Razvoj, 25(4), 69–74. <https://doi.org/10.5937/imk1903069v>