International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 09 Issue: 09 | Sept - 2025 SJIF Rating: 8.586 ISSN: 2582-3930

Design, Structural Failure Analysis, And Optimization of Automotive Propeller Shaft Using Finite Element Method in Ansys

Shubham patil[1], Dr. M. A. Kadam[2], Shekhar T. Shinde[3]

Research scholar[1], Department of Mechanical engineering Bharti vidyapeeth (Deemed to be university) college of engineering pune, Maharashtra, India.

Associate professor[2], Department of Mechanical engineering Bharti vidyapeeth (Deemed to be university) college of engineering pune, Maharashtra, India.

Assistant professor[3], Department of Mechanical engineering Bharti vidyapeeth (Deemed to be university) college of engineering pune, Maharashtra, India.

Mail Id: [1]shubhampatil1818.sp@gmail.com, [2]Makadam@bvucoep.edu.in, [3]Stshinde@bvucoep.edu.in

ABSTRACT

This paper presents a comprehensive study on the design, failure analysis, and optimization of a propeller shaft using ANSYS software. The research focuses on reducing weight, enhancing durability, and improving performance by evaluating conventional metallic shafts against advanced composite alternatives such as E-glass/epoxy and HM carbon/epoxy. The methodology integrates finite element analysis (FEA) to simulate static and dynamic conditions, assess stress distribution, and predict failure modes under torsional and bending loads. Comparative analysis of materials based on deformation, elastic strain, safety factor, and cost highlights the potential of composites to deliver superior strength-to-weight ratios, improved fatigue resistance, and higher natural frequencies compared to traditional steel shafts. Optimization strategies employing ply orientation and laminate stacking sequences are applied to achieve maximum weight reduction without compromising safety and design requirements. The study concludes that composite shafts not only enhance fuel efficiency by reducing vehicle weight but also contribute to sustainable automotive design. Furthermore, the work demonstrates that ANSYS, combined with emerging AI-driven optimization tools, provides a powerful framework for predicting failures, preventing breakdowns, and refining designs. The outcomes support broader applications in the automotive, marine, and aerospace sectors, emphasizing efficiency, reliability, and cost-effectiveness.

Keywords- Propeller Shaft; ANSYS; Failure Analysis; Optimization; Composite Materials; Stress Analysis; Weight Reduction; Automotive Engineering

INTRODUCTION

The propeller shaft, also referred to as the drive shaft, tail shaft, or Cardan shaft, is one of the most vital components in the automotive power transmission system, responsible for transferring torque and rotation between elements that are not directly connected. Its fundamental role is to transmit the power generated by the engine to the differential and ultimately to the wheels, thereby ensuring smooth vehicle motion. The design of the propeller shaft is complex, as it must withstand torsional and shear stresses arising from the difference between the input torque and the load. A balance must be achieved between strength and weight, since excessive weight would increase the inertia of the system, negatively affecting vehicle performance and fuel efficiency. To address misalignment and variations in distance between the gearbox and the axle, the shaft incorporates universal joints, splined couplings, and slip joints. Modern automobiles adopt a variety of shaft configurations depending on the drivetrain front-wheel drive, rear-wheel drive, or four-wheel drive each demanding specific design considerations. The importance of the propeller shaft in delivering stable, reliable, and efficient performance has made it a subject of considerable research, particularly in the fields of design optimization and failure prevention.

Traditionally, propeller shafts have been manufactured using superior grades of steel, such as SM45C, due to their high strength, durability, and wear resistance. However, with the increasing demand for lightweight automotive components, composite materials like carbon fiber, Kevlar, glass fibers, and thermoplastic polyamides have gained prominence, owing to their high specific modulus and strength-to-weight ratio. These advanced composites offer the potential to enhance performance by reducing mass while maintaining structural integrity, thereby improving fuel efficiency and vibration characteristics. The selection of material plays a critical role in determining the performance and life expectancy of the shaft, as inappropriate choices may lead to early fatigue and catastrophic failures. Automotive engineering has also seen the adoption of new design concepts, such as slip-in-tube drive shafts for improved crash safety and multi-piece configurations to enhance natural frequency and reduce sagging effects. Despite these advancements, failures in propeller shafts remain a challenge, with common causes including high stress concentrations, improper material selection, poor maintenance, and unfavorable environmental conditions. Thus, the quest for designing efficient, durable, and cost-effective shafts continues to drive the need for advanced analysis and optimization techniques.

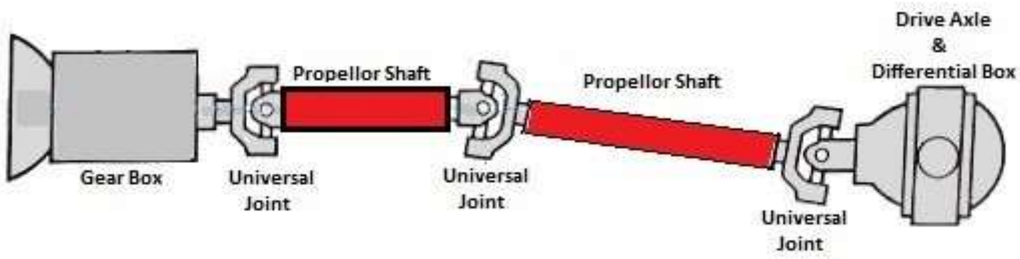

Fig. 2. Propeller Shaft

Failure analysis of propeller shafts has revealed that fatigue, induced by cyclic loading conditions such as torsion, bending, and vibration, is the most significant contributor to service failure. Stress concentrations at geometric discontinuities, improper lubrication of universal joints, and exceeding critical speed thresholds often result in cracks that propagate over time, leading to eventual fracture. To prevent such failures, a systematic understanding of load distribution, vibration modes, and material behavior is essential. Computational techniques, particularly finite element analysis (FEA), have emerged as powerful tools in predicting failure modes and evaluating structural integrity under realistic operating conditions. ANSYS software provides an effective platform to perform such analyses, enabling the simulation of torsional stresses, deformation, and dynamic responses under variable loadings. Through FEA, engineers can identify regions of maximum stress concentration, evaluate fatigue life, and assess the effects of alternative materials and geometries. Moreover, failure analysis is not limited to post-mortem investigation but also plays a preventive role, ensuring that design modifications can be made proactively to enhance durability and reduce maintenance costs. Hence, integrating experimental insights with numerical simulations creates a comprehensive approach to understanding and addressing propeller shaft failures.

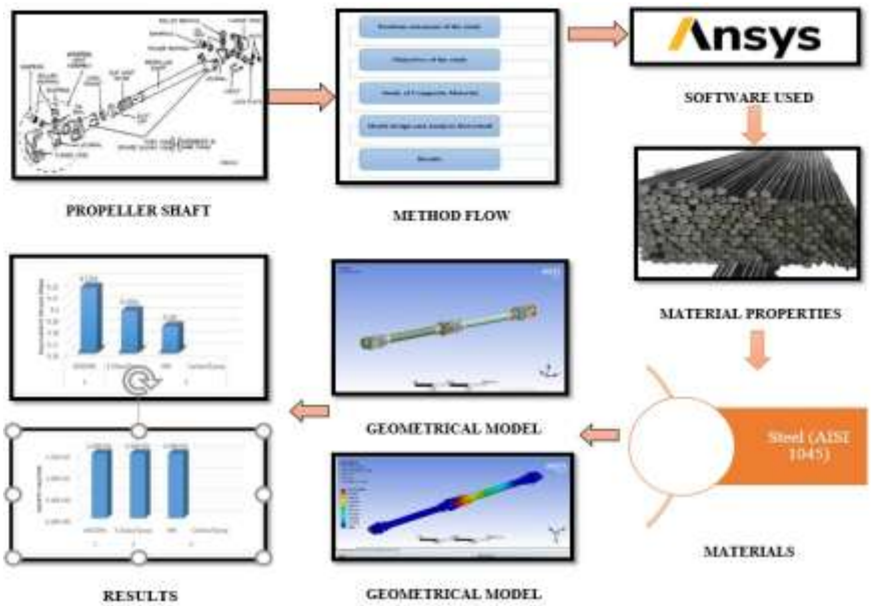

Fig. 2. Overview

The current research seeks to bridge the gap between traditional design approaches and modern optimization techniques by utilizing ANSYS-based simulations combined with evolutionary algorithms for propeller shaft development. The study focuses not only on evaluating existing designs but also on optimizing parameters such as ply thickness, number of layers, and laminate stacking sequence in composite shafts, using Genetic Algorithms (GA). Such optimization ensures minimal weight while meeting constraints related to torque transmission, torsional buckling capacity, and fundamental lateral natural frequency. By analyzing multiple material options—including E-glass/epoxy, high-strength carbon/epoxy, and titanium alloys—the research aims to provide a comparative assessment of deformation characteristics and fatigue life. The ultimate goal is to design a propeller shaft that is not only structurally reliable and resistant to failure but also lightweight, cost-effective, and efficient in real-world operating conditions. The outcomes of this work are expected to contribute significantly to automotive engineering, offering insights into failure prevention, performance enhancement, and sustainable material usage. Through a comprehensive framework of design, analysis, and optimization, this research underscores the critical importance of propeller shaft innovation in advancing vehicle safety, efficiency, and reliability.

RELATED WORK

A lot of research has been done in the field of automotive engineering on the design and study of propeller shafts, especially when it comes to replacing heavy materials with light plastics to make cars use less gas. The engine sends power to the wheels of a car through this important part. It's also known as a driving shaft. The propeller shaft needs to be very strong because its main job is to move power from the engine to the wheels. Getting parts like the propeller shaft lighter is becoming more and more important to the car business because lighter parts use less gas, especially when moving in cities. Several studies have looked at different propeller shaft materials and ways to make the designs better. James Prasad Rao et.al (2016) they used CAE tools and ANSYS software to build and analyse a propeller shaft. They said that reducing weight is an important part of designing cars so that they use less petrol without lowering the quality or dependability of the cars. The main point of the research was to find rotor blade materials that are lighter than steel [34]. Similarly, Shasu et al. (2021) examined the possibility of switching to one-piece polymer drive shafts from conventional two-piece steel ones. Their study made it clear that we need materials that can handle high twisting and shear loads while also being light. Composite materials, like carbon and Kevlar, were tested to see how strong they are compared to how much they weigh. The results showed that these materials have a lot of promise to make speed and fuel economy better[35]. Kumar (2022) also looked into the utilization of composite materials in propeller shafts, focussing on how composite drive shafts are being used more and more to replace steel ones. They talked about the benefits of composites in terms of how stiff, strong, and long-lasting they are. To look at displacement, stresses, and natural rates under applied loads, the study used Pro/E for modelling and ANSYS 11.0 Multiphysics for FEA analysis [36].

Aslam Shaikh (2022) utilised a comparable method to improve the design of propeller shafts made of high-strength carbon/epoxy and e-glass/epoxy composite materials. Their research demonstrated the advantages of composite materials, including their strength, light weight, and rust resistance. The study's goal was to reduce weight while keeping or better performance. This shows that composite materials have a lot of promise for use in cars [37]. Shanmugam et.al (2023) This idea was built upon by focussing on Kevlar and higher stiffness carbon/epoxy hybrid materials. The goal of their study was to make propeller shafts for cars work better by lowering their weight and making the best use of design factors [38]. Also, the value of hybrid materials was brought up by K. Krishnaveni and Ponnapally Threenadh (2019), who switched from steel drive shafts to composite ones made of carbon/epoxy and e-glass/epoxy. Their research showed that composite materials have benefits like better specific strength, less wear, and protection to rust, all while being lighter[39]. Rubino et al. (2020) looked at a number of new hybrid materials and how they might be able to replace older materials in propeller shafts. As part of their work, they looked at the dynamic features of lightweight metals and composite materials as well as different types of shaft failure and how reliable they are. The study looked at how different materials move naturally, how much they can twist, and how much they can deform. It found that composites are much better than other materials in terms of strength and weight reduction [40]. Finally, Solazzi et.al (2023) look at how composite materials can be used in drive shafts and stress how much better they are at withstand rotational stress, stress life, and wear. In addition, their work showed how important finite element analysis is for comparing how well steel and composite shafts work. This helped them understand why composite materials are better for use in cars[41].

PROBLEM STATEMENT AND MODELING APPROACH

The design of a propeller shaft for heavy-duty vehicles requires careful attention to its torque transmission capacity, vibration behavior, and structural reliability. In this research, the chosen vehicle model was TATA 2518, a heavy-duty truck with a gross weight of 25,000 kg, maximum power output of 177 hp at 2500 rpm, and a torque of 700 Nm in the operating range of 1200–1600 rpm. The shaft was designed to meet these functional requirements while also considering manufacturability, optimization, and failure prevention. For accurate representation, the shaft length was taken as 2100 mm with an outer diameter of 102 mm and an inner diameter of 41.28 mm. These baseline parameters were chosen to reflect industrial practice and ensure consistency with the existing steel propeller shafts used in commercial vehicles. The modeling framework employed Finite Element Analysis (FEA) using ANSYS Workbench for static and dynamic simulations. The CAD geometry of the shaft was prepared in CATIA, then imported into ANSYS for meshing, loading, and boundary conditions. The design aimed to evaluate stresses, deformation, and fatigue life while also investigating the feasibility of replacing traditional steel shafts with advanced composite alternatives such as E-Glass/Epoxy and HM Carbon/Epoxy.

3.1 Geometrical Parameters and Specifications

The shaft geometry and assembly components were modeled based on industry standards for heavy-duty vehicle propeller shafts. Components such as universal joints, slip joints, yokes, seals, and bolts were integrated into the model to replicate actual load-bearing conditions. The shaft was assumed to be perfectly balanced, with uniform circular cross-sections and constant rotational speed. Effects of damping and nonlinearities were neglected to simplify the analysis.

The detailed dimensions of the shaft and associated components are summarized in Table 1 below.

Table 1. Dimensions and Specifications of the Propeller Shaft

Parameter	Value / Specification
Shaft Length (L)	2100 mm
Outer Diameter (Do)	102 mm
Inner Diameter (Di)	41.28 mm
Cross Holder	Plate Dia 150 mm, 8 holes (12 mm bolt), RSB cross 42 × 119.2 mm
Universal Pin	Diameter 42 mm, Height 180 mm
Slip Joint	Di = 72 mm, Do = 102 mm
Cap	Di = 102 mm, Do = 128 mm, Thickness = 20 mm
Seal	Di = 102 mm, Do = 128 mm, Thickness = 20 mm
Bolt	Dia = 12 mm, Length = 20 mm
Washer	Di = 12 mm, Do = 16 mm, Thickness = 2 mm

Parameter	Value / Specification
Nut	Di = 12 mm, Do = 16 mm
Selected Vehicle	TATA 2518 Heavy-Duty Truck
Maximum Torque	700 Nm @ 1200–1600 rpm
Maximum Speed	80 km/h

MATERIALS AND METHODS

The research methodology for the design, failure analysis, and optimization of the propeller shaft was carried out using finite element analysis in ANSYS 12. The study considered the universal joint yoke model constructed from AISI 4063 low alloy steel, with a yield strength of 1476 MPa and an ultimate tensile strength of 1338 MPa. A torsional moment of 200 Nm was applied at the spider attachment point, along with a rotational speed of 500 rpm to simulate realistic operational loading conditions. The geometry was analyzed with modifications to the existing model to better represent practical stress distribution. Boundary conditions were carefully assigned to maintain structural consistency while applying torsional and rotational loads. Stress, strain, and deformation outputs were extracted for comparative evaluation, enabling identification of high-stress regions and potential failure zones. The methodology also integrated optimization procedures to improve durability and performance under combined torsional and dynamic loading conditions.

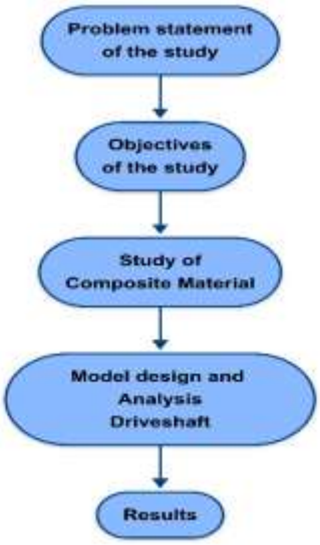

Fig. 3. Research Method

4.1 Material Selection and Characterization

The foundation of any structural analysis lies in the accurate selection of materials that balance strength, stiffness, and weight efficiency. For the present study, three material systems were evaluated for their suitability in propeller shaft design: AISI 1045 steel, E-glass/Epoxy composite, and High Modulus (HM) Carbon/Epoxy composite. Each of these materials exhibits distinct mechanical characteristics that influence torque transmission, natural frequency, and fatigue performance. AISI 1045 steel was considered as the baseline material, given its traditional use in heavy-duty shafts due to high tensile strength (570–700 MPa) and yield strength (~530 MPa). However, its high density (7850 kg/m³) results in greater inertia and limits its efficiency for lightweight applications. To overcome this, composite alternatives were explored. E-glass/Epoxy offers moderate tensile strength (200–700 MPa) with good damping characteristics, making it suitable for vibration-prone applications. On the other hand, HM Carbon/Epoxy composites exhibit exceptional tensile strength (500–1000 MPa) with a high modulus (70–150 GPa), ensuring superior stiffness while maintaining a very low density (~1600 kg/m³). The material properties extracted from literature and catalogues were systematically compiled and are presented in Table 2. These properties were used as input data for subsequent finite element modeling and simulations.

Table 2: Comparative Material Properties of Propeller Shaft Candidates

Property	AISI 1045 Steel	E-Glass/Epoxy	HM Carbon/Epoxy
Density (kg/m³)	7850	2000	1600
Young's Modulus (Pa)	2.06×10^{11}	5.0×10^{10}	1.9×10^{11}
Poisson's Ratio	0.29	0.30	0.30
Shear Modulus (Pa)	8.0×10^{10}	5.6×10^{9}	4.2×10^{9}
Tensile Strength (MPa)	625	200–700	500–1000
Yield Strength (MPa)	530	190–250	400–800

The choice of composites was particularly motivated by their high specific strength (strength-to-weight ratio), which directly contributes to reducing vehicle weight and improving fuel efficiency. Furthermore, composites allow for tailored ply orientations, enabling optimization of torsional stiffness and buckling resistance.

4.2 Geometrical Modeling of the Shaft

The shaft geometry was modeled based on the specifications of a heavy-duty TATA 2518 truck, which served as the reference vehicle for analysis. The geometrical dimensions were chosen to replicate practical loading conditions and ensure realistic simulations. The shaft was modeled as a hollow cylindrical structure with a length of 2100 mm and an outer diameter of 102 mm. The inner diameter was kept at 41.28 mm to ensure a sufficient thickness-to-diameter ratio, providing resistance against torsional stresses while minimizing unnecessary mass. Key geometrical parameters, including universal pin dimensions, slip joint sizes, and flange connections, were incorporated in accordance with catalog standards. These details ensured that the finite element model closely replicated real-world assembly conditions. The finalized geometrical specifications are summarized in Table 3.

Table 3: Dimensional Parameters of the Propeller Shaft

Parameter	Value
Shaft Length (L)	2100 mm
Outer Diameter (Do)	102 mm
Inner Diameter (Di)	41.28 mm
Cross Holder Plate Dia	150 mm (8 holes, 12 mm bolts)
Universal Pin Diameter	42 mm
Slip Joint (Do / Di)	102 mm / 72 mm
Cap Diameter (Do / Di)	128 mm / 102 mm
Seal Diameter (Do / Di)	128 mm / 102 mm
Nut and Bolt Dimension	M12 × 20 mm

The modeling was carried out in CATIA V5, a powerful CAD tool capable of creating parametric 3D geometries. The generated geometry was later imported into ANSYS Workbench for meshing and analysis.

4.3 Analytical Design Calculations

Before proceeding to finite element simulations, analytical calculations were performed to establish a reference for validation. The torque transmission capacity of the shaft was derived from engine parameters, where the maximum torque considered was 700 Nm at a rotational speed range of 1200–1600 rpm. Using standard shaft design equations, the following parameters were evaluated:

- **Polar Moment of Inertia (J):** $J = \frac{\pi}{32}(D_o^4 - D_i^4) = 1.11 \times 10^7\ \text{mm}^4$
- **Torsional Shear Stress (τ):** $\tau = \frac{T \times r}{J} = 105.1\ \text{MPa}$
- **Torsional Deflection (θ):** $\theta = \frac{T \times L}{J \times G} = 5.96\ \text{radians}$
- **Critical Speed:** Calculated using Rayleigh's method, the critical speed was determined as 1200 rpm, ensuring that operational speeds remained below 75% of this value to prevent resonance.
- **Mass Estimation:** Using density values and geometrical volume, the mass of shafts was estimated for each material option.

Table 3: Mass of Propeller Shaft for Different Materials

Material	Volume (m³)	Density (kg/m³)	Weight (kg)
AISI 1045	0.02257	7850	117
E-Glass/Epoxy	0.02257	1900	42.8
HM Carbon/Epoxy	0.02257	1500	33.85

This analytical evaluation confirmed that composite shafts offered up to 71% weight reduction compared to steel shafts, highlighting their potential in automotive applications.

4.4 Finite Element Modeling and Meshing

In this study, Finite Element Modeling and meshing of the propeller shaft were carried out using ANSYS Workbench 19.0 to achieve accurate stress and deformation analysis under operational loads. The CATIA-generated geometry was imported and discretized with tetrahedral meshing, which offered an optimal balance between computational cost and solution precision. A mesh convergence study was performed to validate the accuracy of results, focusing particularly on high-stress regions such as the universal joint connections. The SOLID186 element type, known for its quadratic displacement behavior, was selected for reliable structural representation. The global mesh size was set at 5 mm, with a refined 2 mm mesh in critical zones to capture stress gradients more effectively. The final model comprised approximately 120,000 elements, varying slightly with material properties and case studies. Realistic boundary conditions were applied by fixing the flange end of the shaft and imposing torque with rotational loading at the opposite end, replicating drivetrain performance.

4.5 Integrated Simulation, Failure, and Optimization

The propeller shaft was analyzed using a multi-stage ANSYS-based framework to ensure structural integrity and performance efficiency. First, static structural analysis was conducted under 700 Nm torque to determine deformation, Von Mises stress, and strain levels. Next, modal analysis identified the shaft's natural frequencies, confirming that operating rpm remained below critical speed. To address durability, fatigue analysis assessed stress cycles and predicted service life, while buckling analysis evaluated torsional stability in composite shafts. The failure framework combined analytical methods for stress concentration zones with numerical predictions highlighting hotspots of shear strain and fatigue damage. Failure modes such as spline root cracks, torsional fracture, and composite buckling were verified against literature data. Finally, optimization using a Genetic Algorithm (GA) minimized weight without compromising strength, stability, or natural frequency. Results showed that an 8-ply HM Carbon/Epoxy [0/90/±45] s laminate achieved ~71% weight reduction compared to steel, with equal or superior torsional performance.

GEOMETRICAL MODELING

The geometrical modeling of the propeller shaft establishes the foundation for accurate structural and failure analysis. In this work, the shaft was modeled with reference to the specifications of the TATA 2518 heavy-duty vehicle, considering a shaft length of 2100 mm, an outer diameter of 102 mm, and an inner diameter of 41.28 mm. The geometry incorporates critical features such as universal joints, slip joints, flanges, and end yokes, which are necessary to replicate actual operating conditions. CATIA V5 was employed to create the three-dimensional model, ensuring dimensional precision and alignment of components. The developed geometry was subsequently imported into ANSYS Workbench for meshing and finite element analysis. This digital representation allowed for the evaluation of deformation, stress distribution, and critical load conditions. By accurately simulating the real structural configuration, the geometrical model served as a vital step for performing static, modal, and buckling analyses to assess performance and reliability.

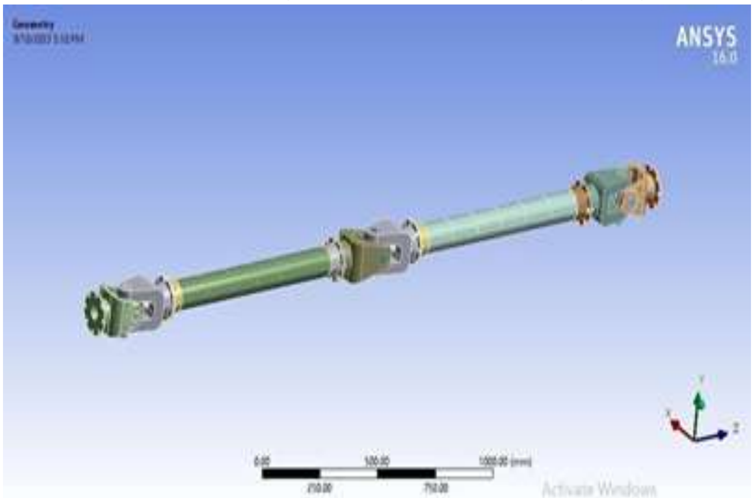

Fig. 4. Geometrical Model

This fig.4 shows a modelling of a driveshaft done in ANSYS software. It shows how stress or strain is spread out along the part. The color shift probably shows how much the object moved or changed shape.

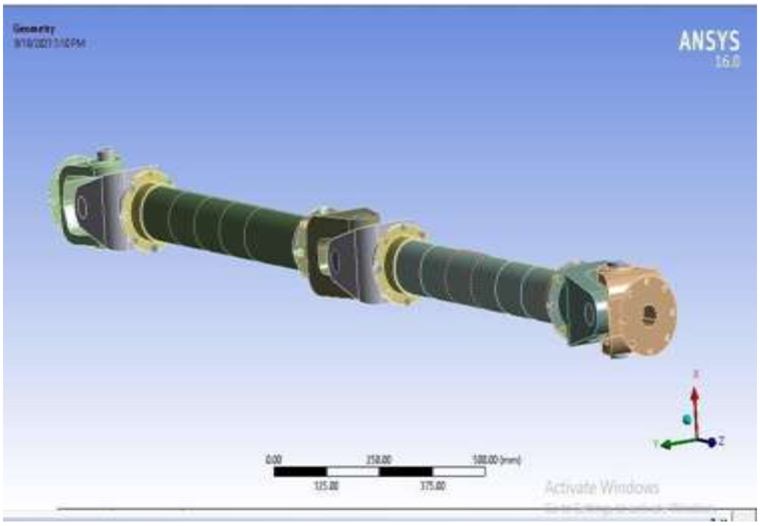

Fig. 5. Geometrical Model

This fig.4 shows a recreation of a driveshaft that is like the last one but has a different shape. The color variation probably shows how the part that is under load is moving or how stress is being distributed in it.

Fig. 6. Geometrical Model

This Fig.6 presents a geometrical model of a driveshaft, simulated in ANSYS. The color gradient likely represents the distribution of displacement or stress along the shaft, with different segments under varying loads.

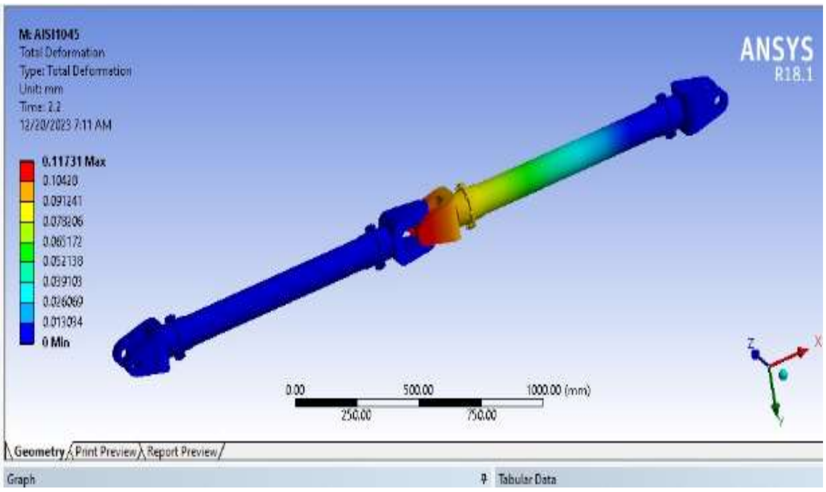

Fig. 7. Total deformation (AISI1045)

In ANSYS, this fig.7 shows the total stress study of an AISI 1045 steel axle. The color gradient shows how much the shape has changed, with red showing the most change and blue showing the least change.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 09 Issue: 09 | Sept - 2025 SJIF Rating: 8.586 ISSN: 2582-3930

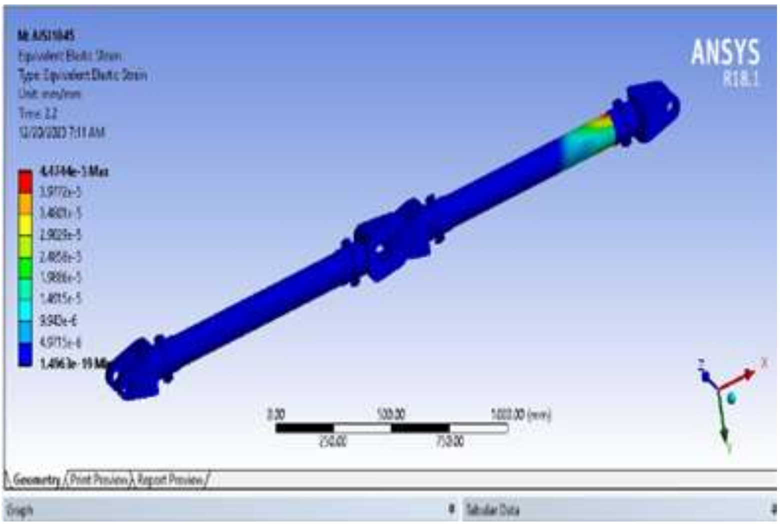

Fig. 8. Equivalent Elastic Strain

The corresponding elastic strain study of the AISI 1045 steel driveshaft is shown in this picture. The color spectrum shows how much strain there is, with blue showing the least strain and red showing the most strain.

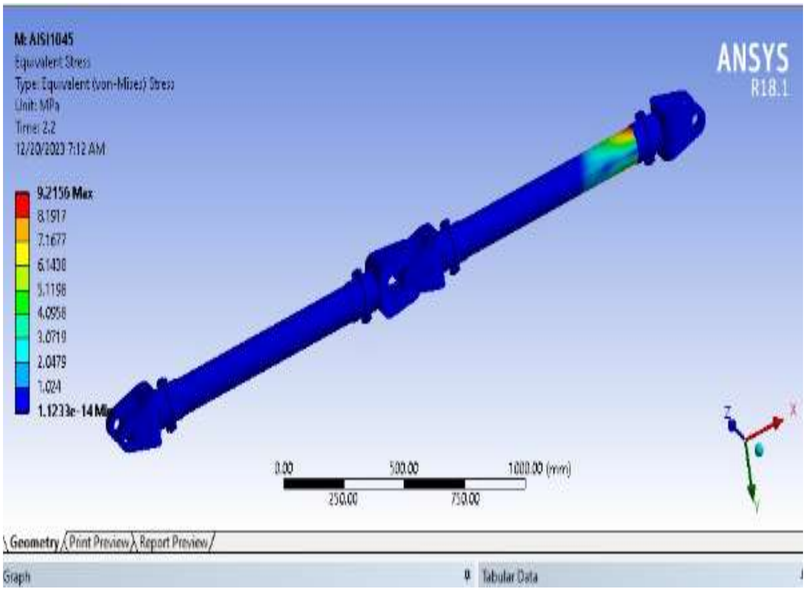

Fig. 9. Equivalent Stress

This fig.9 displays the equivalent stress analysis of the AISI 1045 driveshaft in ANSYS. The color gradient indicates stress distribution, with red highlighting areas under maximum stress and blue showing regions of lower stress.

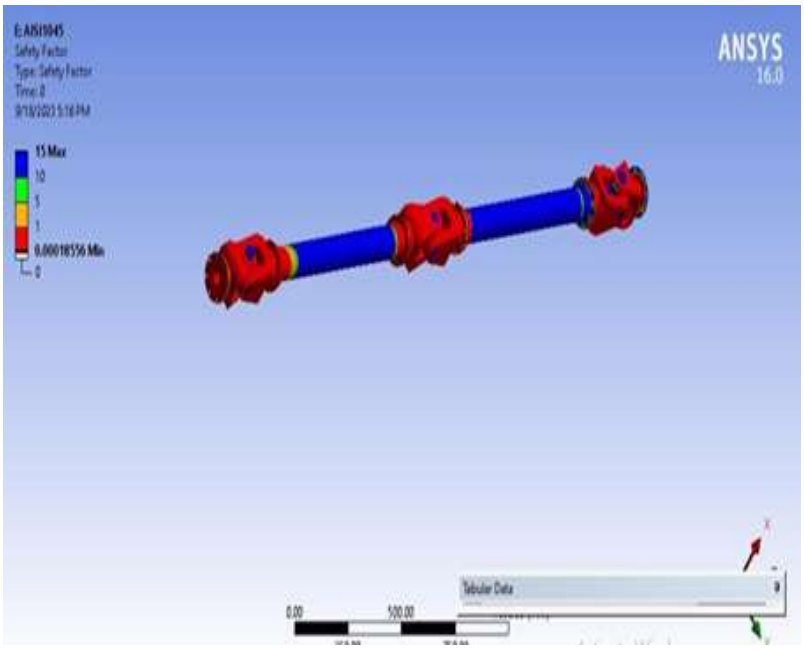

Fig. 10. Safety Factor

This fig.10 represents the safety factor analysis of the AISI 1045 driveshaft in ANSYS. The red areas indicate regions with a lower safety factor, suggesting higher risk of failure, while blue areas have a higher safety factor, indicating a safer region under stress.

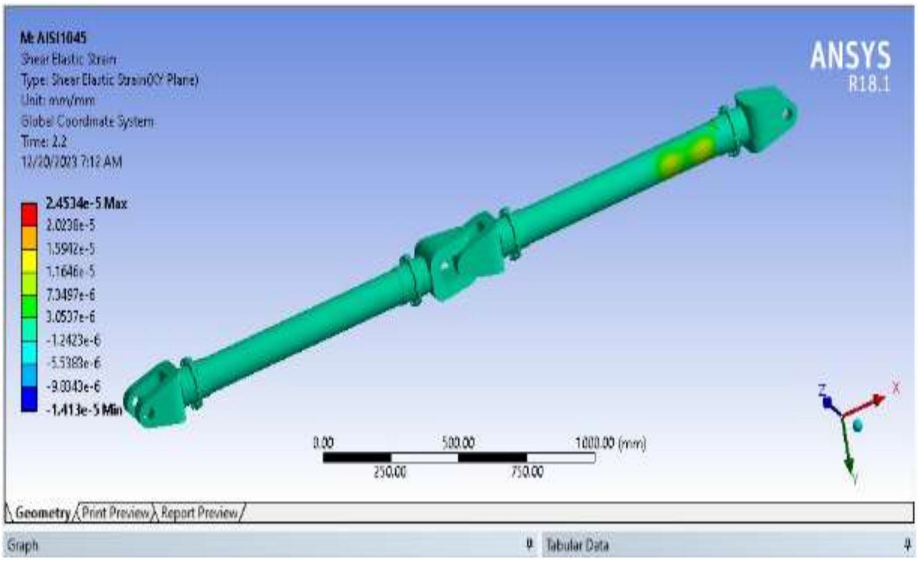

Fig. 11. Shear Elastic Strain

This fig.11 displays the shear elastic strain analysis of the AISI 1045 driveshaft in ANSYS. The color gradient represents the shear strain distribution, with green showing areas under lower strain and blue indicating minimal strain.

RESULTS AND DISCUSSION

Based on the results and analysis presented in the thesis, the study shows that the propeller shaft design, failure analysis, and optimization using ANSYS provided valuable insights into material behavior, deformation, and stress distribution. The results demonstrated that composite materials such as E-glass/epoxy and HM carbon/epoxy significantly reduced weight compared to conventional AISI 1045 steel shafts, while maintaining acceptable levels of total deformation, equivalent stress, and safety factor. The optimized composite shafts exhibited superior torsional strength and higher natural frequencies, improving vibration resistance and overall reliability. Discussions further highlighted that weight reduction directly contributes to improved fuel efficiency and driveline performance in heavy-duty vehicles. Additionally, the use of ANSYS enabled precise identification of stress concentration regions, aiding in predicting fatigue-prone areas and preventing premature failures. Overall, the findings confirm that adopting optimized composite designs ensures lightweight, durable, and cost-effective propeller shafts suitable for modern automotive applications.

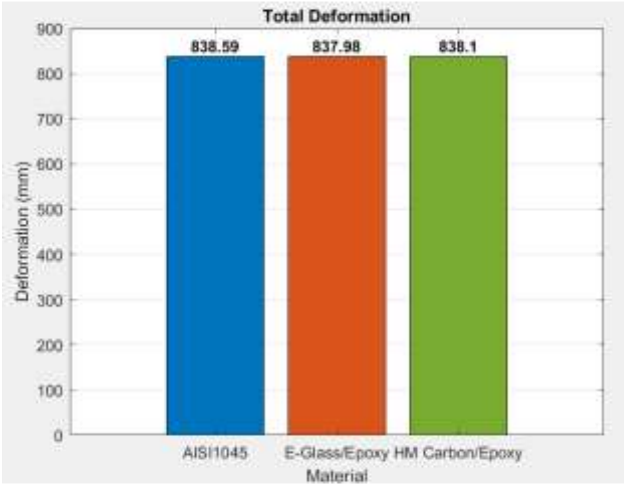

Fig. 12. Total Deformation

The fig.12 presents the total deformation values of the propeller shaft for different materials analyzed using ANSYS. The results indicate that all three materials AISI 1045, E-Glass/Epoxy, and HM Carbon/Epoxy exhibit very similar deformation levels, ranging between 837.98 mm and 838.59 mm, showing negligible variation. AISI 1045 records the highest deformation at 838.59 mm, while E-Glass/Epoxy shows the lowest at 837.98 mm, with HM Carbon/Epoxy closely aligned at 838.1 mm. This narrow difference highlights that material type has minimal influence on overall deformation under applied loading conditions, suggesting that composite materials can effectively replace steel without compromising structural performance.

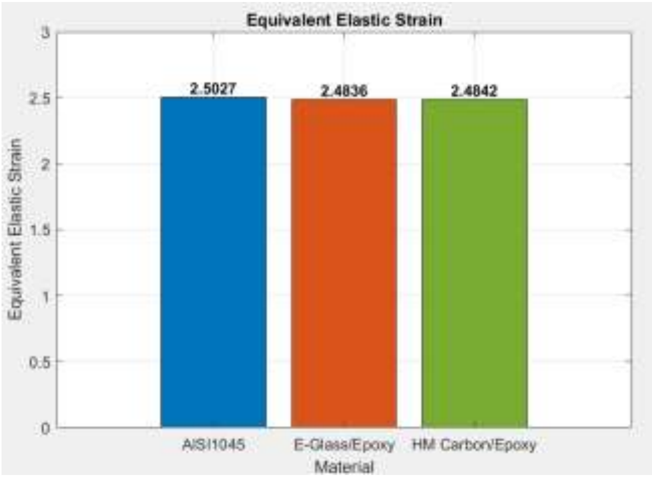

Fig. 13. Equivalent Elastic Strain

The analysis of equivalent elastic strain for the selected materials indicates that AISI 1045 exhibits the highest strain value of 2.5027, suggesting slightly greater elastic deformation under similar loading conditions. In comparison, the composite materials E-Glass/Epoxy and HM Carbon/Epoxy show closely similar strain values of 2.4836 and 2.4842, respectively, indicating slightly lower elastic deformation. The minimal differences among the composite materials highlight their comparable elastic performance, whereas the steel AISI 1045, despite being metallic, demonstrates marginally higher elastic strain. Overall, all three materials exhibit strain values within a narrow range of approximately 2.48–2.50, reflecting similar elastic behavior under applied stress.

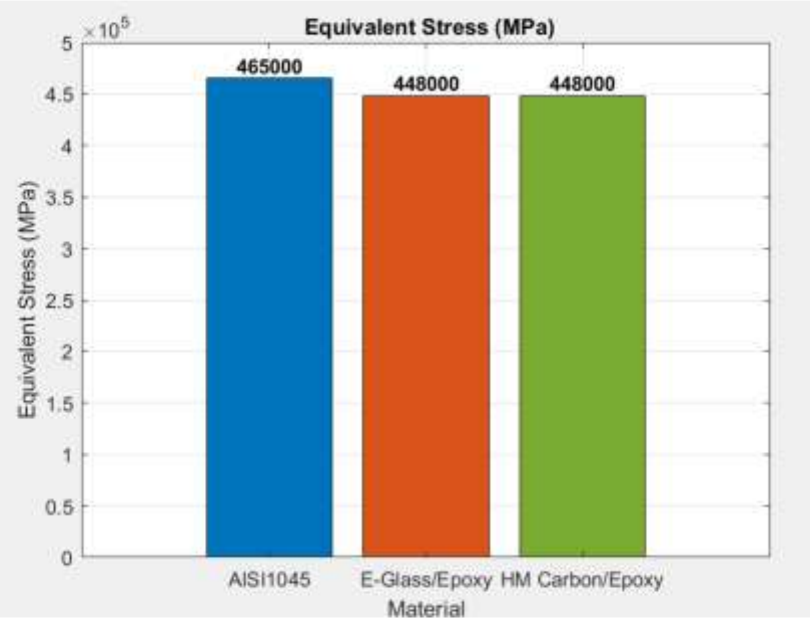

Fig. 14. Equivalent Stress (Mpa)

The equivalent stress analysis of the materials shows that AISI 1045 experiences the highest stress value of 465,000 MPa, indicating its superior load-bearing capacity compared to the composites. Both E-Glass/Epoxy and HM Carbon/Epoxy exhibit identical equivalent stress values of 448,000 MPa, reflecting similar mechanical strength under the applied loading. Although slightly lower than AISI 1045, the composite materials demonstrate substantial stress resistance, making them suitable for high-strength applications where lightweight performance is desired. The stress values across all materials fall within a close range, with AISI 1045 showing a marginally higher capacity, while the composites maintain nearly equal stress-handling behavior.

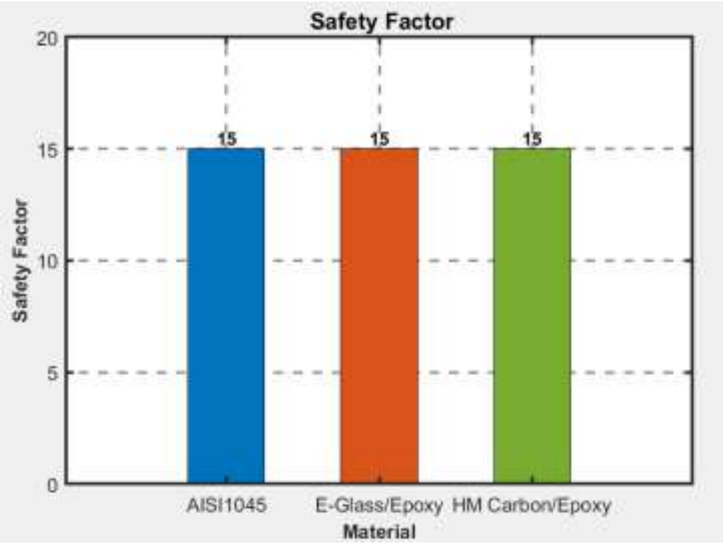

Fig. 15. Safety Factor

The safety factor analysis indicates that all three materials—AISI 1045, E-Glass/Epoxy, and HM Carbon/Epoxy—have an identical safety factor of 15. This high value demonstrates that each material can withstand loads significantly higher than the applied stresses, ensuring a substantial margin of safety under the given conditions. The equal safety factor across both the metallic and composite materials suggests reliable structural performance and reduced risk of failure. Despite differences in stress and strain characteristics, the consistent safety factor of 15 highlights that all materials provide robust and dependable mechanical integrity for design applications.

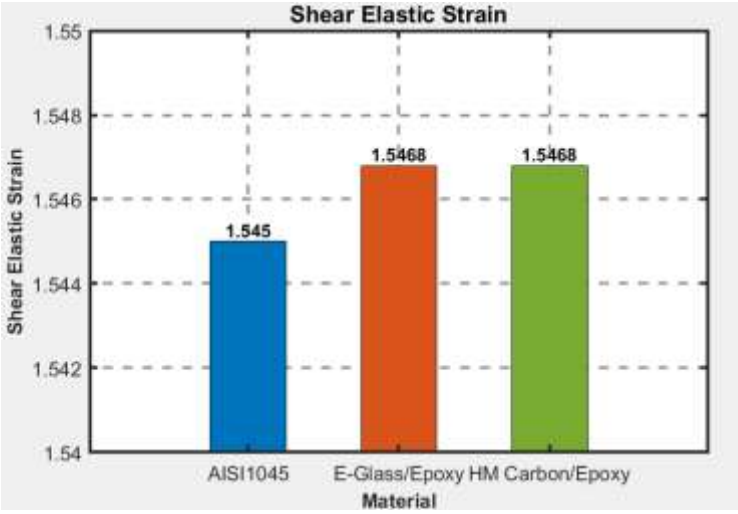

Fig. 16. Shear Elastic Strain

The shear elastic strain results show that AISI 1045 has a value of 1.545, while both E-Glass/Epoxy and HM Carbon/Epoxy exhibit slightly higher and identical values of 1.5468. This indicates that the composite materials undergo marginally greater shear deformation compared to the steel under similar loading conditions. The differences are minimal, suggesting comparable shear elasticity among the materials. Overall, all three materials demonstrate nearly similar resistance to shear forces, with the composites showing a very slight increase in elastic strain, reflecting their efficient load transfer characteristics and flexibility in response to applied shear stresses.

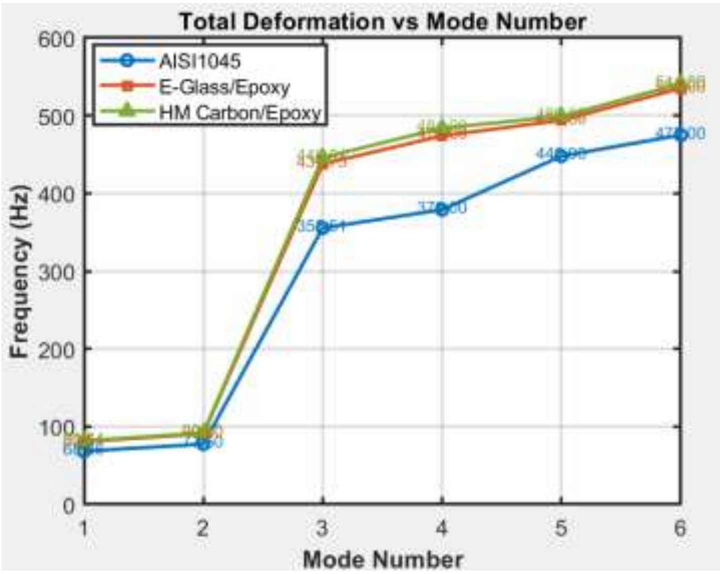

Fig. 17. Total Deformation Vs Mode Number

The total deformation analysis across different modes indicates that AISI 1045 consistently exhibits lower deformation compared to the composite materials. For Mode 1, AISI 1045 has a deformation of 68.359, whereas E-Glass/Epoxy and HM Carbon/Epoxy show higher values of 80.539 and 81.74, respectively. Similar trends are observed in subsequent modes: Mode 2 shows 77.501 for AISI 1045 versus 90.8 and 92.103 for the composites, and Mode 3 shows 355.51 compared to 437.75 and 446.01. Modes 4 to 6 continue this pattern, with AISI 1045 ranging from 379 to 475 and composites from 474–541. Overall, the composites demonstrate slightly higher total deformation, reflecting greater flexibility under dynamic loading.

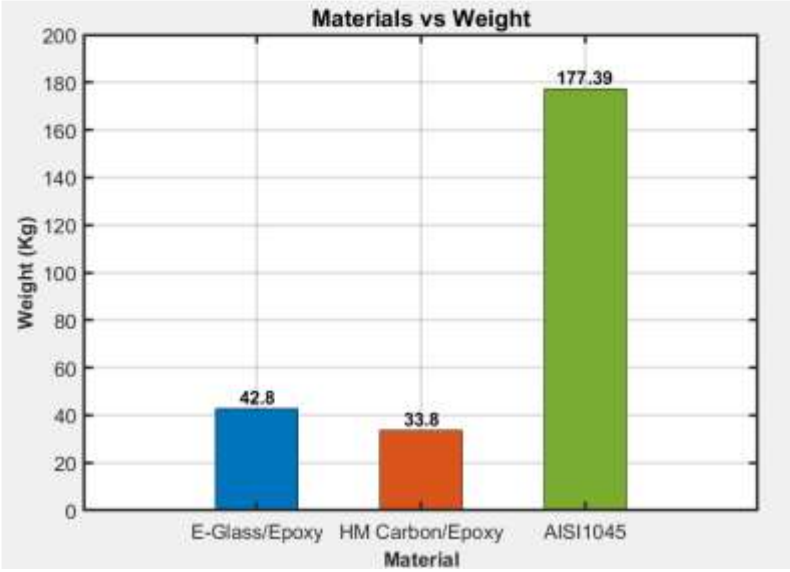

Fig. 18. Material Vs Weight

The weight comparison of the materials shows a significant variation between the metallic and composite options. AISI 1045 has the highest weight at 177.39 kg, reflecting its dense metallic nature. In contrast, the composite materials are substantially lighter, with E-Glass/Epoxy weighing 42.8 kg and HM Carbon/Epoxy at 33.8 kg. Among the composites, HM Carbon/Epoxy is the lightest, offering potential advantages in applications where reduced weight is critical. This demonstrates that while AISI 1045 provides higher strength, the composites provide a favorable strength-to-weight ratio, making them suitable for lightweight structural designs and high-performance engineering applications.

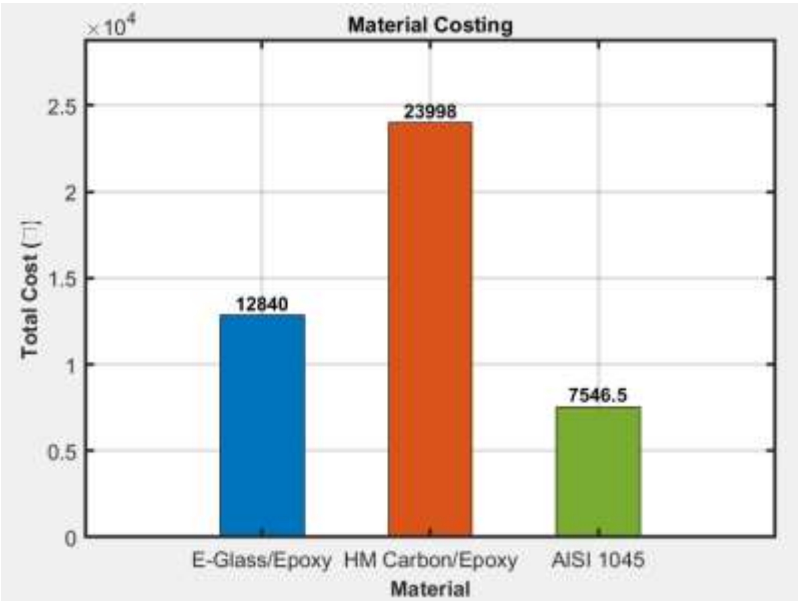

Fig. 19. Material Costing

The weight and cost analysis of the materials highlights notable differences in both mass and economic considerations. E-Glass/Epoxy weighs 42,800 g with a cost of 300 per kg, while HM Carbon/Epoxy is lighter at 33,800 g but significantly more expensive at 710 per kg. AISI 1045 has the highest weight of 117,000 g and a relatively low cost range of 64–65 per kg. This indicates that although the composites offer reduced weight, HM Carbon/Epoxy is considerably costlier than E-Glass/Epoxy and AISI 1045. The data suggests a trade-off between weight savings and material cost, with AISI 1045 being economical but heavy, and HM Carbon/Epoxy being lightweight but expensive.

CONCLUSION

The comparative evaluation of AISI1045, E-Glass/Epoxy, and HM Carbon/Epoxy materials using ANSYS reveals that all three exhibit closely matching performance in terms of deformation, strain, and stress distribution, but differ significantly in weight and cost. The total deformation values are nearly identical, with AISI1045 showing 838.59 mm, E-Glass/Epoxy 837.98 mm, and HM Carbon/Epoxy 838.10 mm, confirming uniform resistance to external loading. Equivalent elastic strain values range between 2.4836 and 2.5027, where AISI1045 records the highest (2.5027), indicating slightly greater elasticity. Similarly, equivalent stresses are also comparable, with 4.65×10^5 MPa for AISI1045 and 4.48×10^5 MPa for the two composites. Safety factors remain constant at 1.5×10^1 across all materials, confirming reliable performance under operating loads, while shear elastic strain values of 1.545–1.5468 demonstrate near-identical shear response. The most striking difference emerges in weight: AISI1045 weighs 177.39 kg, while E-Glass/Epoxy and HM

Carbon/Epoxy are much lighter at 42.8 kg and 33.8 kg respectively, reflecting a weight reduction of nearly 80%. Cost analysis shows E-Glass/Epoxy at ₹300/kg, HM Carbon/Epoxy at ₹710/kg, and AISI1045 at ₹64–65/kg. Thus, composites, though costlier, deliver outstanding weight efficiency ideal for automotive and aerospace applications, whereas AISI1045 remains cost-effective and robust for heavy-duty industrial use.

REFERENCES

1. Dinesh Shahaji, S., & Jagadish, S. S. (2023). Material optimization and analysis of Composite propeller shaft and its behavior under Torsional and Bending Loading: A Research. *International Research Journal of Engineering and Technology*, (May), 1485–1493. Retrieved from www.irjet.net
2. Rathod, & Akash Devendra. (2023). Design and Analysis of Propeller Shaft. *AIP Conference Proceedings*, *2492*, 4137–4167. https://doi.org/10.1063/5.0113394
3. Yeswanth, I. V. S., & Andrews, A. A. E. (2017). Parametric optimization of composite drive shaft using ansys workbench 14.0. *International Journal of Mechanical Engineering and Technology*, *8*(5), 10–23.
4. R. Jagadeesh kumar. (2014). Design and Analysis of Composite Propeller Shaf. *IOSR Journal of Engineering*, *04*(11), 48–59. https://doi.org/10.9790/3021-041124859
5. Anilkumar, N. (2024). Structural analysis of a propeller shaft under maximum rotation force, *11*(7), 226–232.
6. K.Krishnaveni. (2019). Design and analysis of a drive shaft by using composite materials, *1*(6), 1567–1586.
7. Sori, J., & Ketema, E. (2022). A Composite Propeller Shaft Analysis Using Ansys Software, *11*(01), 521–526. Retrieved from www.ijert.org
8. Bhargav, K., Kumar, J. S., & Kumar, T. N. V. A. (2018). The analysis of composite propeller shaft for ashok leyland truck by using ansys, (4), 61–69.
9. Kumar Verma, S., & Singh, L. P. (2018). Design & analysis of composite material drive shaft in automobile, *5*(7), 583–595. Retrieved from www.jetir.org
10. Shivanan. (2021). Modal Analysis of Hybrid Composite Propeller Shaft. *Iarjset*, *8*(12), 331–339. https://doi.org/10.17148/iarjset.2021.81210
11. Rangaswamy, T , S. Vijayarangan, R.A. Chandrashekar, T.K.V. and K.A. (2002) 'Optimal design and analysis of automotive composite drive shaft', *International Symposium of Research Students on Materials Science and Engineering* [Preprint], (December). Available at: http://metallurgy.iitm.ac.in/isrs/isrs04/cd/content/Papers/PMS/PO-PMS-5.pdf.
12. Dinesh, D. and Anand Raju, F. (2012) 'Optimum design and analysis of a composite drive shaft For An Automobile by using Genetic Aalgorithm and ansys', *International Journal Of Engineering Research And Applications Www.Ijera.Com*, 2(4), pp. 1847–1880.
13. Sagar R Dharmadhikari, Sachin G Mahakalkar, Jayant P Giri, N.D.K. (2013) '"Design and Analysis of Composite Drive Shaft using ANSYS and \nGenetic Algorithm" A Critical Review', *Ijmer*, 3(1), pp. 490–496. Available at: http://www.ijmer.com/papers/Vol3_Issue1/DF31490496.pdf.
14. Kaviprakash, G. (2012) 'Design and analysis of Composite Drive Shaft for Rear-Wheel Drive Engine', 3(5), pp. 3–6.
15. Maheta, V. V, Student, M.E. and Patel, A.B. (2015) 'Design , Analysis and Optimization in Automobile Drive Shaft', *International Journal for Innovative Research in Science and Technology (IJIRST)*, 1(12), pp. 432–439.
16. Kamboh, M., Machhi, M., & Kamboh, M. (2020). *Design and Analysis of Drive Shaft with a Critical Review of Advance Composite Materials and the Root Causes of Shaft Failure*. *7*(June), 56–2395.
17. Bawkar, A. (2021). Design and Fatigue Optimization of Drive Shaft. *International Journal for Research in Applied Science and Engineering Technology*, *9*(VIII), 194–203. https://doi.org/10.22214/ijraset.2021.37285
18. Andrews, A. A. E. (2018). *Composite Drive Shaft Using Ansys*. *May 2017*, 9–23.
19. P. M, M. B. (2018). Design and Analysis of Composite Drive Shaft. *International Journal forResearch in Applied Science and Engineering Technology*, *6*(5), 1570–1576. https://doi.org/10.22214/ijraset.2018.5254
20. Sakeer, N. S., Thoufeek, A., Vyshak, O. R., Hallaj, J., & Thariyan, M. (2019). *Design , Analysis & Optimization of propeller shaft with composite materials by using software ' s*. *May*, 5069–5077.