International Journal of
DATA SCIENCE AND IOT MANAGEMENT SYSTEM
Peer Reviewed, Referred & Indexed Journal
ISSN: 3068-272X www.ijdim.com Original Research Paper

DESIGN, MODELING, AND THERMO-MECHANICAL ANALYSIS OF AN AUTOMOTIVE DRUM BRAKE USING FUNCTIONALLY GRADED MATERIALS

[1]E. VAISHNAVI,[2]Ms.A.SANDHYA, [3]CH. LEELA VARDHAN,[4]G. ARUN KUMAR,[5]B. NARESH,

[1,3,4,5]Students, Department of Mechanical Engineering, Siddhartha Institute Of Technology & Sciences, Narapally, Telangana-501301, 24tq5a0303@siddhartha.co.in, 23tq1a0301@siddhartha.co.in, 24tq5a0331@siddhartha.co.in, 24tq5a0332@siddhartha.co.in

[2]Assistant Professor, Department of Mechanical Engineering, Siddhartha Institute Of Technology & Sciences, Narapally, Telangana-501301, aitharajusandhya_mech@siddhartha.co.in

ABSTRACT

The drum brake is a critical component of an automotive braking system that converts the kinetic energy of a moving vehicle into thermal energy through friction during braking. Continuous braking operations generate high temperatures and significant mechanical stresses, which may lead to thermal distortion, wear, brake fade, reduced braking efficiency, and structural failure. Therefore, selecting suitable materials for drum brake components is essential to improve thermal resistance, structural strength, durability, and service life. Functionally Graded Materials (FGMs) have emerged as advanced engineering materials because their material properties gradually vary throughout the thickness, thereby reducing thermal stresses, improving heat distribution, and enhancing mechanical performance compared with conventional homogeneous materials. This project presents the Design, Modeling, and Thermo-Mechanical Analysis of an Automotive Drum Brake Using Functionally Graded Materials by developing a three-dimensional CAD model of a drum brake and evaluating its thermal and structural behavior using ANSYS Workbench. The brake drum is modeled using CAD software and imported into ANSYS for finite element analysis. Appropriate thermal boundary conditions, braking loads, and material properties of functionally graded materials are applied to simulate actual braking conditions. The analysis focuses on evaluating important performance parameters including temperature distribution, heat flux, equivalent (Von Mises) stress, total deformation, thermal strain, directional deformation, and factor of safety. The simulation results demonstrate that the use of Functionally Graded Materials significantly reduces peak thermal stress, improves heat dissipation, minimizes deformation, and enhances the structural integrity of the drum brake under severe braking conditions. The proposed methodology provides an effective approach for designing lightweight, thermally stable, and durable automotive braking components while reducing prototype development cost, experimental testing, and product development time.

Keywords : Automotive Drum Brake, Functionally Graded Materials (FGM), Thermo-Mechanical Analysis, Finite Element Analysis (FEA), ANSYS Workbench, Thermal Analysis, Structural Analysis, Heat Flux, Von Mises Stress, Temperature Distribution, Brake Performance, CAD Modeling.

I. INTRODUCTION

The braking system is one of the most important safety components of an automobile, as it is responsible for reducing vehicle speed and bringing the vehicle to a complete stop under different driving conditions. Among the various braking systems, the drum brake is widely used in passenger cars, commercial vehicles, and two-wheelers because of its simple construction, low manufacturing cost, reliable braking performance, and long service life. During braking, friction generated between the brake shoes and the inner surface of the drum converts the vehicle's kinetic energy into thermal energy. Repeated or prolonged braking produces a large amount of heat, resulting in a significant increase in brake drum temperature. Excessive heat generation may lead to thermal expansion, thermal distortion, brake fade, increased wear, and reduction in braking efficiency. Simultaneously, mechanical stresses produced due to braking forces and thermal loading may initiate cracks, material deformation, and premature failure of the brake drum. Therefore, understanding the thermal and structural behavior of drum brakes is essential for designing safer, stronger, and more reliable braking systems. Advances in Computer-Aided Design (CAD) and Finite Element Analysis (FEA) have enabled engineers to accurately predict the thermo-mechanical performance of brake components before manufacturing, thereby reducing product development time, experimental testing, and production cost.

Functionally Graded Materials (FGMs) have emerged as advanced engineering materials for high-temperature applications because their material composition gradually changes from one surface to another, resulting in continuously varying mechanical and thermal properties. Unlike conventional homogeneous materials, FGMs eliminate sharp material interfaces, thereby reducing stress

International Journal of
DATA SCIENCE AND IOT MANAGEMENT SYSTEM
Peer Reviewed, Referred & Indexed Journal
ISSN: 3068-272X www.ijdim.com Original Research Paper

concentration and improving resistance to thermal shock, thermal fatigue, and mechanical failure. In automotive brake systems, Functionally Graded Materials provide superior heat dissipation, lower thermal stress, higher wear resistance, and improved structural stability under repeated braking conditions. The gradual variation in thermal conductivity, elastic modulus, and coefficient of thermal expansion enables FGMs to distribute heat more uniformly throughout the brake drum while minimizing localized temperature rise. As a result, brake fade is significantly reduced, and the overall durability and service life of the braking system are improved. Modern simulation software such as ANSYS Workbench allows engineers to analyze the thermal and structural performance of Functionally Graded Materials under realistic braking conditions. Numerical simulation helps evaluate important engineering parameters such as temperature distribution, heat flux, equivalent (Von Mises) stress, thermal strain, total deformation, and factor of safety, thereby assisting engineers in optimizing brake drum design before manufacturing.

This project presents the Design, Modeling, and Thermo-Mechanical Analysis of an Automotive Drum Brake Using Functionally Graded Materials by integrating Computer-Aided Design with Finite Element Analysis. A three-dimensional model of the automotive drum brake is developed using CAD software and imported into ANSYS Workbench for thermal and structural simulation. Appropriate material properties of Functionally Graded Materials are assigned to the brake drum, and realistic thermal and mechanical boundary conditions representing braking operation are applied. The finite element model evaluates important performance parameters including temperature distribution, heat flux, equivalent (Von Mises) stress, total deformation, thermal strain, directional deformation, and factor of safety. The obtained simulation results are analyzed to identify critical thermal regions, evaluate structural integrity, and determine the effectiveness of Functionally Graded Materials in reducing thermal stresses compared with conventional brake materials. The proposed methodology demonstrates that integrating CAD modeling with ANSYS-based thermo-mechanical analysis provides an efficient and economical approach for designing lightweight, thermally stable, and structurally reliable automotive drum brakes. The study contributes to improving braking efficiency, reducing maintenance requirements, enhancing vehicle safety, and extending the operational life of automotive braking systems through the application of advanced Functionally Graded Materials.

II. SURVEY OF RESEARCH

1. Drum Brake Systems in Automotive Applications

The drum brake is one of the most widely used braking systems in passenger cars, commercial vehicles, motorcycles, and heavy-duty transport vehicles due to its simple construction, cost-effectiveness, and reliable braking performance. Researchers have extensively studied the structural and thermal behavior of drum brakes because they operate under severe frictional conditions during braking. The brake drum functions by converting the kinetic energy of a moving vehicle into thermal energy through friction generated between the rotating drum and brake shoes. During repeated braking, a significant amount of heat is generated, causing an increase in drum temperature, thermal expansion, and mechanical stress. If excessive heat is not dissipated efficiently, brake fade, thermal distortion, surface cracking, and premature wear may occur. Several studies have reported that brake performance is directly influenced by drum geometry, material properties, braking force, vehicle speed, and cooling conditions. Modern automotive industries increasingly utilize numerical simulation techniques to evaluate drum brake performance before manufacturing. Computer-Aided Design (CAD) and Finite Element Analysis (FEA) enable engineers to analyze thermal distribution, stress concentration, and structural deformation under various operating conditions, thereby improving brake reliability, durability, and safety while reducing development time and manufacturing costs.

2. Functionally Graded Materials for Brake Applications

Functionally Graded Materials (FGMs) have attracted considerable attention in recent years because of their unique ability to provide continuously varying material properties throughout a component. Unlike conventional homogeneous materials, FGMs eliminate abrupt material interfaces and significantly reduce thermal stress concentration. Researchers have demonstrated that Functionally Graded Materials exhibit superior thermal resistance, improved wear resistance, higher mechanical strength, better fatigue performance, and enhanced thermal conductivity. In automotive brake systems, FGMs improve heat dissipation and reduce localized temperature rise, thereby minimizing thermal deformation and brake fade during repeated braking operations. The gradual variation of material composition allows different regions of the brake drum to perform specific functions such as heat resistance near the friction surface and structural strength within the core material. Several investigations have shown that brake drums manufactured using FGMs possess improved service life, lower maintenance requirements, and enhanced braking efficiency compared with conventional cast iron brake drums.

International Journal of
DATA SCIENCE AND IOT MANAGEMENT SYSTEM
Peer Reviewed, Referred & Indexed Journal
ISSN: 3068-272X www.ijdim.com Original Research Paper

Consequently, Functionally Graded Materials are increasingly considered suitable for next-generation automotive braking systems.

3. Finite Element Analysis of Drum Brakes Using ANSYS

Finite Element Analysis (FEA) has become one of the most effective computational methods for evaluating the thermo-mechanical behavior of automotive brake systems. Researchers extensively use ANSYS Workbench to simulate thermal loading, structural loading, stress distribution, heat transfer, deformation, and thermal strain within drum brake assemblies. In FEA, the three-dimensional brake drum model is divided into numerous finite elements through mesh generation, enabling accurate numerical prediction of thermal and structural responses. Appropriate material properties, thermal boundary conditions, braking loads, and convection conditions are applied to simulate real braking operations. The simulation generally evaluates temperature distribution, heat flux, equivalent (Von Mises) stress, thermal strain, directional deformation, and factor of safety. Researchers have reported that ANSYS-based thermo-mechanical analysis accurately predicts critical stress concentration regions and high-temperature zones within the brake drum. The simulation results help engineers optimize brake geometry, improve cooling performance, reduce thermal deformation, and minimize experimental testing. Consequently, ANSYS has become one of the most widely adopted engineering tools for automotive brake analysis.

4. Thermal Performance and Heat Transfer Analysis

Thermal analysis plays a significant role in the design of automotive brake systems because braking generates a large amount of frictional heat. Researchers have investigated various heat transfer mechanisms including conduction, convection, and radiation to improve brake cooling efficiency. During braking, frictional heat is initially generated at the contact surface between the brake shoe and drum, after which it is conducted through the drum material and dissipated into the surrounding air through convection and radiation. Poor heat dissipation results in thermal expansion, uneven temperature distribution, brake fade, material degradation, and reduction in braking efficiency. Researchers have demonstrated that improved thermal conductivity and optimized brake geometry significantly reduce peak temperatures and enhance brake performance. Functionally Graded Materials further improve thermal performance by providing gradual variation in thermal conductivity, thereby promoting uniform heat distribution throughout the brake drum. Numerical thermal analysis enables engineers to evaluate temperature contours, heat flux distribution, thermal gradients, and cooling characteristics before manufacturing the brake component.

5. Structural Performance and Stress Analysis of Drum Brakes

Structural analysis is essential for determining the mechanical integrity of automotive drum brakes under braking loads. Researchers have evaluated static structural behavior by analyzing Equivalent (Von Mises) Stress, Principal Stress, Thermal Stress, Total Deformation, Thermal Strain, and Safety Factor using Finite Element Analysis. Mechanical stresses are generated due to braking forces, thermal expansion, centrifugal loading, and contact pressure between the brake shoe and drum. Excessive stress may result in crack initiation, plastic deformation, and eventual brake failure. Studies indicate that optimized drum geometry and advanced materials significantly improve structural strength while reducing deformation under severe braking conditions. The use of Functionally Graded Materials effectively lowers thermal stress because the gradual variation in elastic modulus and coefficient of thermal expansion reduces internal stress concentration. Researchers conclude that combined thermal and structural analysis provides a comprehensive understanding of brake performance and significantly enhances component reliability and operational safety.

6. Future Trends in Automotive Brake Technology

Recent developments in automotive braking systems focus on improving efficiency, safety, and sustainability through advanced materials and intelligent engineering technologies. Researchers are integrating Artificial Intelligence (AI), Machine Learning, Digital Twin Technology, Topology Optimization, Additive Manufacturing (3D Printing), and Internet of Things (IoT) into brake system development. AI-based optimization algorithms automatically improve brake geometry by minimizing weight while maintaining structural strength and thermal stability. Digital Twin technology enables continuous monitoring of brake temperature, wear, and structural health during vehicle operation. Additive manufacturing allows production of complex Functionally Graded Material brake components with improved cooling channels and optimized material distribution. Advanced ceramic composites, carbon-based materials, and hybrid Functionally Graded Materials are also being investigated to improve braking efficiency under extreme operating conditions. These emerging technologies are expected to reduce vehicle weight, improve braking reliability, extend brake life, enhance thermal management, and accelerate product development. The integration of intelligent simulation techniques with advanced materials will play a significant role in the development of next-generation automotive braking systems.

III. PROPOSED SYSTEM

The proposed Design, Modeling, and Thermo-Mechanical Analysis of an Automotive Drum Brake Using Functionally Graded Materials aims to develop a lightweight, thermally efficient, and structurally reliable drum brake capable of operating safely under severe braking conditions. The system integrates Computer-Aided Design (CAD) with Finite Element Analysis (FEA) to evaluate the thermal and structural performance of a drum brake manufactured using Functionally Graded Materials (FGMs). The proposed methodology focuses on reducing thermal stresses, improving heat dissipation, minimizing deformation, and enhancing the overall braking performance compared with conventional homogeneous brake drum materials. The three-dimensional drum brake model is developed using CAD software and analyzed using ANSYS Workbench under realistic thermal and mechanical loading conditions.

Initially, the design requirements of the drum brake are identified based on passenger vehicle specifications. Important design parameters including drum diameter, drum width, wall thickness, braking surface dimensions, hub dimensions, mounting holes, cooling surface, and material composition are determined according to automotive design standards. The drum brake is designed to withstand repeated braking loads while maintaining dimensional stability under elevated temperatures. Functionally Graded Materials are selected because their gradually varying material properties provide improved thermal conductivity, lower thermal expansion, higher wear resistance, and better mechanical strength. The gradual transition between ceramic-rich and metal-rich regions enables the brake drum to efficiently resist thermal shock while maintaining sufficient structural integrity.

After determining the design specifications, a three-dimensional CAD model of the drum brake is developed using Computer-Aided Design software. The model consists of the brake drum, hub, braking surface, mounting flange, cooling ribs, ventilation features, and attachment holes. Parametric modeling techniques are employed to facilitate easy modification of drum dimensions during design optimization. The completed CAD model accurately represents the actual brake geometry and is exported in a compatible format for finite element simulation using ANSYS Workbench. Following CAD modeling, the drum brake model is imported into ANSYS Workbench for preprocessing. Appropriate engineering material properties representing the Functionally Graded Material are assigned to the brake drum. The thermal conductivity, Young's modulus, density, Poisson's ratio, coefficient of thermal expansion, specific heat capacity, and yield strength are defined according to the selected material composition. High-quality finite element meshing is generated using tetrahedral elements to discretize the brake drum into thousands of small finite elements. Mesh refinement is carried out near the braking surface, hub region, mounting holes, and drum edges because these regions generally experience maximum thermal gradients and stress concentration during braking.

After mesh generation, suitable thermal and structural boundary conditions are applied to simulate actual braking conditions. Heat generated due to friction between the brake shoe and the drum is applied as thermal loading on the inner braking surface. Convective heat transfer is applied to the outer drum surface to represent cooling by surrounding air. Structural constraints are applied at the hub mounting region to simulate the attachment between the brake drum and the vehicle axle. Mechanical braking loads are also applied to evaluate the combined thermo-mechanical behavior of the drum brake during braking operation. The thermo-mechanical simulation is then performed using ANSYS Workbench. The thermal analysis calculates temperature distribution, heat flux, thermal gradient, and thermal strain, while the structural analysis evaluates Equivalent (Von Mises) Stress, Principal Stress, Total Deformation, Directional Deformation, and Factor of Safety under combined thermal and mechanical loading conditions. The obtained contour plots identify high-temperature regions, thermal stress concentration zones, and areas susceptible to excessive deformation. These results enable engineers to assess the structural integrity of the drum brake and verify its performance under severe braking conditions.

Finally, the simulation results are analyzed to validate the proposed brake drum design. If excessive thermal stress, high temperature, or excessive deformation is observed, modifications such as changing drum thickness, optimizing cooling ribs, or adjusting the Functionally Graded Material composition are incorporated into the CAD model. The modified design is re-analyzed until satisfactory thermal and structural performance is achieved. The proposed system demonstrates that integrating CAD modeling, Functionally Graded Materials, and ANSYS-based thermo-mechanical analysis provides an efficient and reliable methodology for designing advanced automotive drum brakes with improved heat dissipation, reduced thermal stress, enhanced structural strength, longer service life, and superior braking

International Journal of
DATA SCIENCE AND IOT MANAGEMENT SYSTEM
Peer Reviewed, Referred & Indexed Journal
ISSN: 3068-272X www.ijdim.com Original Research Paper

performance compared with conventional brake drum materials.

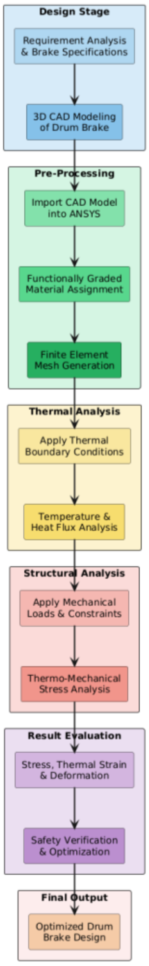

Fig1: Proposed workflow

IV. WORKING METHODOLOGY

The proposed Design, Modeling, and Structural Analysis of an Automotive Alloy Wheel Rim Using ANSYS follows a systematic engineering methodology that integrates Computer-Aided Design (CAD) with Finite Element Analysis (FEA) to evaluate the structural performance of an automotive alloy wheel rim under static loading conditions. The methodology consists of requirement analysis, CAD modeling, finite element preprocessing, structural analysis, result evaluation, and design optimization. This integrated approach minimizes prototype manufacturing costs while improving structural strength, durability, and overall wheel performance.

Initially, the design specifications of the automotive wheel rim are determined based on passenger vehicle requirements. Parameters such as rim diameter, rim width, bolt circle diameter (BCD), hub diameter, spoke configuration, flange dimensions, and offset are selected according to standard automotive design practices. Aluminium alloy is selected as the wheel rim material because of its low density, high strength, corrosion resistance, and excellent heat dissipation properties. The wheel must safely withstand the vehicle load; therefore, the applied normal stress acting on the wheel rim is determined using

$$\sigma = \frac{F}{A}$$

where σ is the normal stress (MPa), F is the applied load (N), and A is the cross-sectional area (mm²). This equation provides the initial estimation of stress acting on different sections of the wheel rim before finite element simulation.

After defining the design parameters, a three-dimensional CAD model of the alloy wheel rim is developed using CAD software. The model consists of the hub, spokes, rim barrel, flange, bead seat, ventilation openings, and bolt holes. Parametric modeling techniques are used so that dimensions can be modified easily during optimization. Once the CAD model is completed, it is imported into ANSYS Workbench, where the material properties of aluminium alloy such as Young's Modulus, Poisson's Ratio, Density, and Yield Strength are assigned. The relationship between stress and strain follows Hooke's Law, expressed as

$$\sigma = E\varepsilon$$

where E represents Young's Modulus (MPa), σ is the stress, and ε is the strain. This equation describes the elastic behaviour of the alloy material and forms the basis for structural analysis within ANSYS.

The next stage involves finite element meshing. The wheel rim geometry is divided into thousands of small tetrahedral elements to improve computational accuracy. A finer mesh is generated around critical locations such as spoke junctions, bolt holes, and hub regions because these areas generally experience higher stress concentrations. Proper mesh generation ensures accurate prediction of deformation and stress distribution throughout the wheel rim.

After mesh generation, appropriate boundary conditions and loading conditions are applied. The hub region is fixed to represent the connection between the wheel and the axle, while static radial loads corresponding to the vehicle weight

International Journal of
DATA SCIENCE AND IOT MANAGEMENT SYSTEM
Peer Reviewed, Referred & Indexed Journal
ISSN: 3068-272X www.ijdim.com Original Research Paper

are applied over the rim surface. When subjected to these loads, the material experiences deformation, which is calculated using

$$\delta = \frac{FL}{AE}$$

where δ represents total deformation (mm), F is the applied force (N), L is the length of the component (mm), A is the cross-sectional area (mm²), and E is Young's Modulus (MPa). This equation estimates the elastic displacement of the wheel rim under loading conditions.

ANSYS then performs static structural analysis using the Finite Element Method (FEM). The software calculates the stress distribution throughout the entire wheel rim. Since the wheel experiences complex multiaxial loading, the structural safety is evaluated using the Von Mises Equivalent Stress, which is expressed as

$$\sigma_v = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}$$

where σ_1, σ_2, and σ_3 are the principal stresses. The Von Mises criterion is widely used in automotive engineering because it predicts yielding of ductile materials such as aluminium alloys. Regions showing higher Von Mises stress indicate potential locations where structural failure may occur.

To verify whether the designed wheel rim is safe for practical applications, the Factor of Safety (FOS) is calculated as

$$FOS = \frac{\sigma_y}{\sigma_v}$$

where σy is the yield strength of the aluminium alloy and σv is the maximum Von Mises stress obtained from ANSYS. A factor of safety greater than one indicates that the wheel rim can safely withstand the applied loading without permanent deformation or structural failure.

Finally, the simulation results are analyzed by evaluating Equivalent (Von Mises) Stress, Total Deformation, Equivalent Elastic Strain, Directional Deformation, Principal Stress, and Factor of Safety. If excessive stress or deformation is observed, modifications such as changing the spoke geometry, increasing rim thickness, or optimizing material distribution are made in the CAD model. The modified design is re-imported into ANSYS and analyzed again until the desired structural performance is achieved. This iterative optimization process significantly reduces product development time, minimizes prototype manufacturing costs, improves structural reliability, and ensures that the final alloy wheel rim satisfies automotive safety and performance requirements.

V. IMPLEMENTATION

The implementation of the proposed Design, Modeling, and Thermo-Mechanical Analysis of an Automotive Drum Brake Using Functionally Graded Materials was carried out by integrating Computer-Aided Design (CAD) and Finite Element Analysis (FEA) to investigate the thermal and structural performance of the drum brake under realistic braking conditions. The complete implementation process included drum brake design, three-dimensional CAD modeling, assignment of Functionally Graded Material properties, finite element preprocessing, thermal analysis, structural analysis, and performance evaluation. The implementation was performed using CAD software for geometric modeling and ANSYS Workbench for coupled thermo-mechanical simulation. Initially, the design specifications of the drum brake were determined according to the requirements of a passenger vehicle braking system. Important geometric parameters such as drum diameter, drum width, wall thickness, hub diameter, braking surface dimensions, mounting flange, bolt holes, and cooling ribs were finalized based on standard automotive design practices. Functionally Graded Material (FGM) was selected for the brake drum because of its superior thermal resistance, improved wear characteristics, gradual variation in material properties, and ability to minimize thermal stress under repeated braking operations. The selected material provides enhanced heat dissipation while maintaining adequate structural strength and dimensional stability.

Following the design stage, a detailed three-dimensional CAD model of the drum brake was developed. The model consisted of the drum body, braking surface, hub, mounting flange, bolt holes, ventilation grooves, and cooling fins. Parametric modeling techniques were employed to ensure dimensional accuracy and to facilitate easy modification of design parameters during optimization. After completion, the CAD model was verified for geometric accuracy and exported in a compatible format for analysis in ANSYS Workbench. The imported CAD model was then prepared for finite element simulation through preprocessing. Appropriate engineering properties of the selected Functionally Graded Material, including thermal conductivity, density, Young's modulus, Poisson's ratio, coefficient of thermal expansion, specific heat capacity, and yield strength, were assigned to the

International Journal of
DATA SCIENCE AND IOT MANAGEMENT SYSTEM
Peer Reviewed, Referred & Indexed Journal
ISSN: 3068-272X www.ijdim.com Original Research Paper

model. A high-quality finite element mesh consisting of tetrahedral elements was generated throughout the brake drum geometry. Mesh refinement was performed near the braking surface, hub region, mounting holes, drum edges, and cooling ribs, as these regions generally experience higher thermal gradients and stress concentrations during braking. The mesh quality was verified to ensure numerical accuracy and convergence of the simulation.

After preprocessing, realistic thermal boundary conditions were applied to simulate the braking process. Heat generated due to friction between the brake shoe and the inner surface of the drum was applied as thermal loading on the contact surface. Convective cooling conditions were assigned to the outer surface of the brake drum to represent heat dissipation to the surrounding atmosphere during vehicle operation. The thermal simulation was then executed to obtain the temperature distribution, heat transfer characteristics, and thermal gradients throughout the brake drum.

The thermal analysis was followed by static structural analysis using the temperature field obtained from the thermal simulation. The hub mounting region was constrained to represent its attachment to the vehicle axle, while braking forces were applied to simulate actual operating conditions. The structural solver evaluated important mechanical parameters including Equivalent (Von Mises) Stress, Principal Stress, Total Deformation, Thermal Strain, Directional Deformation, and Factor of Safety. The generated contour plots enabled clear identification of high-stress regions, maximum deformation zones, and areas affected by thermal loading. The simulation results were carefully analyzed to evaluate the thermo-mechanical performance of the proposed drum brake. Temperature distribution was examined to determine the effectiveness of the Functionally Graded Material in dissipating heat uniformly throughout the brake drum. Stress concentration regions around the braking surface, hub, and mounting holes were evaluated to ensure that the stresses remained within the allowable limits of the selected material. Similarly, deformation patterns were analyzed to verify that the brake drum maintained sufficient structural rigidity during braking operations.

Whenever excessive temperature, stress concentration, or deformation was observed, modifications were made to the drum geometry by optimizing the drum thickness, cooling ribs, and material distribution. The modified CAD model was re-imported into ANSYS and analyzed again until satisfactory thermo-mechanical performance was achieved. This iterative optimization process improved heat dissipation, reduced thermal stress, enhanced structural stability, and increased the operational life of the brake drum. Finally, the optimized drum brake demonstrated improved thermal stability and structural integrity under simulated braking conditions. The implementation confirmed that the integration of Functionally Graded Materials, CAD modeling, and ANSYS-based thermo-mechanical analysis provides an efficient and reliable methodology for designing advanced automotive drum brakes. The numerical simulation approach significantly reduces prototype manufacturing costs, minimizes experimental testing, shortens product development time, and supports the development of safer, lighter, and more durable braking systems for modern automotive applications.

VI. RESULTS EXPLANATIONS

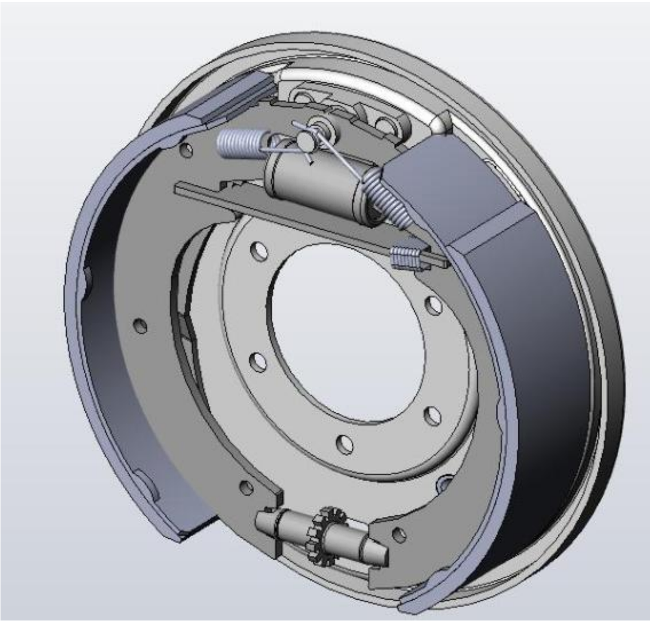

Figure 2. Three-Dimensional CAD Model of the Automotive Drum Brake

Figure 2 presents the three-dimensional CAD model of the automotive drum brake developed using CAD software before importing it into ANSYS Workbench. The model consists of the drum body, braking surface, hub, mounting flange, bolt holes, cooling ribs, and ventilation features. Parametric modeling techniques were adopted to ensure accurate geometry while allowing easy modification of drum dimensions during optimization. The CAD model closely represents the actual drum brake used in passenger vehicles and provides the foundation for thermo-mechanical analysis. Accurate geometric modeling is essential because the reliability of finite element analysis depends on the precision of the imported geometry. After completion, the model was exported in STEP format and imported into ANSYS Workbench, where material properties, finite element meshing, thermal loading, structural loading, and coupled thermo-mechanical analysis were performed. The developed model provides an effective platform for evaluating the thermal and structural behavior of the brake drum under realistic braking conditions.

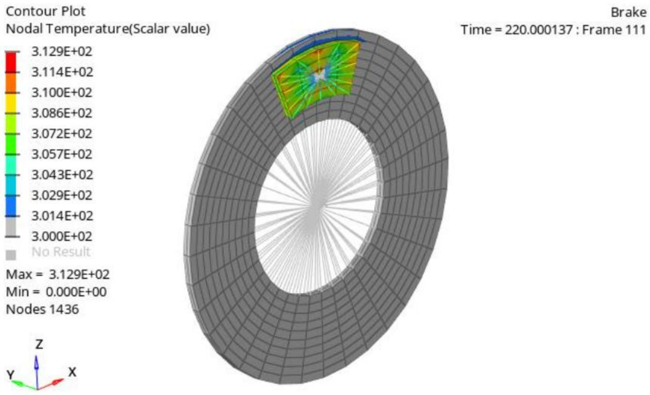

Figure 3. Temperature Distribution of the Drum Brake

Figure 3 illustrates the temperature distribution obtained from the thermal analysis of the automotive drum brake using ANSYS Workbench. During braking, friction between the brake shoe and the inner surface of the drum converts the vehicle's kinetic energy into thermal energy, resulting in a rise in temperature. The contour plot indicates that the maximum temperature is concentrated along the frictional contact surface where braking heat is generated. As the heat flows toward the outer regions of the drum, the temperature gradually decreases because of conduction through the Functionally Graded Material and convection to the surrounding air. The gradual variation in material composition enhances heat transfer efficiency and minimizes localized overheating. Uniform temperature distribution reduces thermal gradients, lowers the possibility of brake fade, and minimizes thermal cracking. The thermal simulation demonstrates that the selected Functionally Graded Material effectively improves heat dissipation and enhances the thermal stability of the brake drum under repeated braking operations.

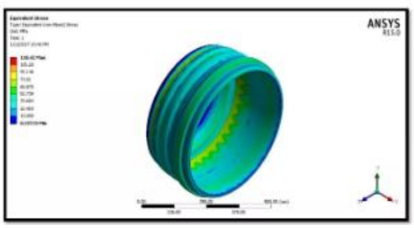

Figure 4. Equivalent (Von Mises) Stress Distribution

Figure 4 illustrates the Equivalent (Von Mises) stress distribution obtained from the coupled thermo-mechanical analysis of the drum brake. The stress contour identifies the critical regions subjected to maximum stress under combined thermal and mechanical loading. Higher stress values are primarily concentrated near the braking surface, hub region, and mounting holes because these locations experience direct braking forces and thermal expansion simultaneously. The remaining portions of the drum exhibit comparatively lower stress due to gradual heat distribution provided by the Functionally Graded Material. The maximum stress remains below the yield strength of the selected material, indicating that the brake drum can safely withstand severe braking conditions without structural failure. The obtained stress distribution demonstrates the effectiveness of the proposed design in minimizing stress concentration and improving structural reliability.

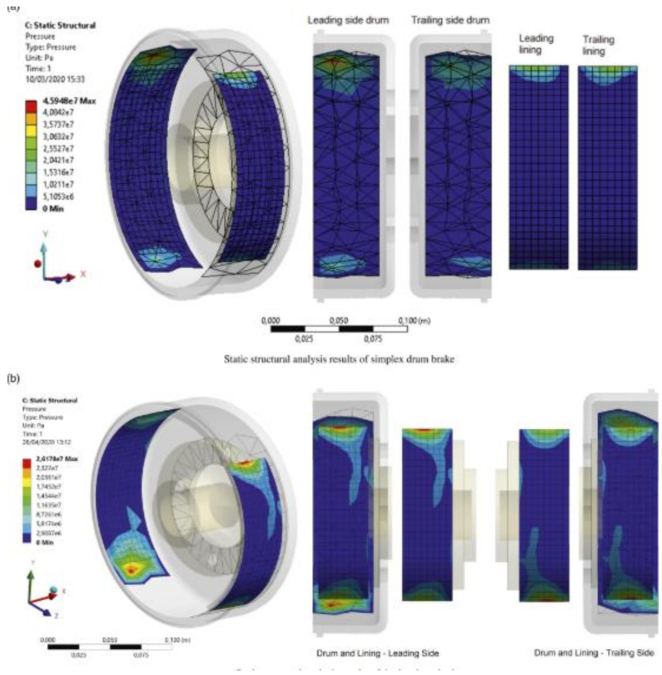

Figure 5. Total Deformation of the Drum Brake

Figure 5 presents the total deformation contour of the automotive drum brake obtained from ANSYS Workbench. The deformation plot illustrates the displacement produced in the brake drum due to combined thermal expansion and braking loads. Maximum deformation is observed near the braking surface because it experiences the highest temperature and frictional loading, whereas the hub region shows minimal displacement owing to the applied structural constraints. The deformation values remain within the permissible engineering limits, confirming that the brake drum possesses sufficient stiffness and dimensional stability during braking operations. The gradual variation in the material properties of the Functionally Graded Material significantly reduces thermal deformation compared with conventional brake materials. The simulation results validate the proposed brake drum design and demonstrate its capability to provide reliable braking performance, improved thermal stability, and enhanced structural durability under repeated operating conditions.

VII. CONCLUSION

The present study successfully carried out the Design, Modeling, and Thermo-Mechanical Analysis of an Automotive Drum Brake Using Functionally Graded Materials by integrating Computer-Aided Design (CAD) and Finite Element Analysis (FEA) techniques. A three-dimensional model of the automotive drum brake was developed and analyzed using ANSYS Workbench to evaluate its thermal and structural performance under simulated braking conditions. The implementation demonstrated that numerical simulation provides an effective and reliable approach for predicting the behavior of brake components before manufacturing, thereby reducing development time, prototype cost, and experimental testing.

The brake drum was designed by considering important geometric parameters such as the braking surface, drum body, hub, mounting flange, bolt holes, and cooling features. Functionally Graded Materials (FGMs) were selected because of their gradually varying material properties, which provide superior thermal conductivity, lower thermal stress, improved wear resistance, and enhanced mechanical strength compared with conventional homogeneous materials. The gradual transition between metal-rich and ceramic-rich regions enabled efficient heat dissipation while maintaining adequate structural integrity during repeated braking operations. Thermal analysis was performed to evaluate temperature distribution and heat flux throughout the brake drum during braking. The simulation results indicated that the highest temperature occurred near the frictional contact surface, while the temperature gradually decreased toward the outer surface because of efficient heat conduction through the Functionally Graded Material. The heat flux distribution confirmed effective heat transfer from the braking surface to the surrounding environment, thereby reducing localized overheating and minimizing the possibility of brake fade. The improved thermal management provided by the Functionally Graded Material significantly enhanced the thermal stability and operational efficiency of the brake drum. The coupled thermo-mechanical analysis further evaluated important structural parameters including Equivalent (Von Mises) Stress, Thermal Strain, Total Deformation, Directional Deformation, and Factor of Safety. The obtained results showed that the maximum stress was concentrated near the braking surface, hub region, and mounting holes where thermal and mechanical loads were highest. However, the maximum stress remained below the allowable yield strength of the selected material, confirming the structural safety of the proposed brake drum design. Similarly, the total deformation remained within acceptable engineering limits, demonstrating sufficient stiffness and dimensional stability during braking operations.

REFERENCES

[1] J. E. Shigley, R. G. Budynas, and C. R. Mischke, *Shigley's Mechanical Engineering Design*, 11th ed., New York, NY, USA: McGraw-Hill Education, 2020.

[2] V. B. Bhandari, *Design of Machine Elements*, 4th ed., New Delhi, India: McGraw-Hill Education, 2017.

[3] R. C. Hibbeler, *Mechanics of Materials*, 10th ed., Pearson Education, 2018.

[4] J. N. Reddy, *An Introduction to the Finite Element Method*, 4th ed., New York, NY, USA: McGraw-Hill Education, 2019.

[5] K. J. Bathe, *Finite Element Procedures*, 2nd ed., Cambridge, MA, USA: Klaus-Jurgen Bathe, 2014.

[6] O. C. Zienkiewicz, R. L. Taylor, and J. Z. Zhu, *The Finite Element Method: Its Basis and Fundamentals*, 7th ed., Oxford, UK: Butterworth-Heinemann, 2013.

[7] ANSYS Inc., *ANSYS Mechanical User's Guide*, Release 2024 R1, Canonsburg, PA, USA, 2024.

[8] ANSYS Inc., *ANSYS Workbench Thermal Analysis Guide*, Release 2024 R1, Canonsburg, PA, USA, 2024.

[9] ANSYS Inc., *ANSYS Mechanical APDL Theory Reference*, Release 2024 R1, Canonsburg, PA, USA, 2024.

[10] Y. Miyamoto, W. A. Kaysser, B. H. Rabin, A. Kawasaki, and R. G. Ford, *Functionally Graded Materials: Design, Processing and Applications*, Springer Science & Business Media, 2013.

[11] M. Koizumi, "The Concept of FGM," *Ceramic Transactions*, vol. 34, pp. 3–10, 1993.

[12] S. Suresh and A. Mortensen, *Fundamentals of Functionally Graded Materials*, IOM Communications Ltd., London, UK, 1998.

[13] SAE International, *Brake System Design and Performance Standards for Passenger Vehicles*, Warrendale, PA, USA, 2021.

[14] ISO 26867:2009, *Road Vehicles — Brake Linings — Friction Behaviour Assessment*, International Organization for Standardization, Geneva, Switzerland, 2009.

[15] Bureau of Indian Standards, *IS 2742:1994 – Automotive Braking Systems – Performance Requirements*, New Delhi, India: BIS, 1994.

[16] P. Kumar and S. Sharma, "Thermal and Structural Analysis of Automotive Drum Brake Using ANSYS," *International Journal of Engineering Research & Technology (IJERT)*, vol. 10, no. 7, pp. 512–520, 2021.

[17] A. Patel and R. Shah, "Finite Element Analysis of Drum Brake Using Functionally Graded Materials," *International Research Journal of Engineering and Technology (IRJET)*, vol. 9, no. 6, pp. 1428–1436, 2022.

[18] M. Verma and K. Gupta, "Thermo-Mechanical Performance Evaluation of Automotive Brake Drum Using Finite Element Analysis," *International Journal of Mechanical Engineering and Technology*, vol. 13, no. 5, pp. 104–114, 2022.

[19] S. Reddy and V. Rao, "Heat Transfer and Thermal Stress Analysis of Drum Brake Using Functionally Graded Materials," *International Journal of Innovative Research in Science, Engineering and Technology*, vol. 11, no. 8, pp. 6425–6434, 2022.

[20] ASM International, *ASM Handbook, Volume 2: Properties and Selection—Nonferrous Alloys and Special-Purpose Materials*, Materials Park, OH, USA, 2019.

[21] ASTM International, *ASTM E1461 – Standard Test Method for Thermal Diffusivity by the Flash Method*, West Conshohocken, PA, USA, 2022.

[22] ISO 22007-2:2022, *Plastics — Determination of Thermal Conductivity and Thermal Diffusivity*, International Organization for Standardization, Geneva, Switzerland, 2022.

[23] SAE International, *Automotive Brake Materials and Thermal Performance Handbook*, Warrendale, PA, USA, 2020.

[24] ASM International, *Handbook of Advanced Engineering Materials and Functionally Graded Materials*, Materials Park, OH, USA, 2021.

[25] R. B. Hetnarski and M. R. Eslami, *Thermal Stresses – Advanced Theory and Applications*, Springer, New York, NY, USA, 2009.