IJAER/March-April-2022/Volume-11/Issue-2 ISSN: 2278-9677

International Journal of Arts & Education Research

LITERATURE REVIEW ON INTEGRATED HUB ROTOR FOR ALL-TERRAIN VEHICLE

Shams Mukaddam

Mechanical Engineering, Trinity Academy of Engineering, Pune, India

Virendra Mudliyar

Mechanical Engineering, Trinity Academy of Engineering, Pune, India

Vijay Kolate

Assistant Professor, Mechanical Engineering, Trinity Academy of Engineering, Pune, India

Abstract

This paper summarizes the literature review of many known author across the world dealing with the problem of lightweight and effective construction of with integrated hub rotor assembly. Many studies including studies related to conventional hub rotor which means separate brake rotor, wheel hub end fixture assembly. The researchers have been studied deeply to understand various gaps that were void to fill in with new details such as thickness of the rotor different materials for a single integrated hub router manufacturing operations to be used by developing an integrated hub rotor. An integrated hub router as the name defines is a single assembly which consists of combining of up brake rotor wheel hub and their fixture assembly such as bearings nuts and bolts. All the analysis that were done using software non-destructive testing as well as destructive testing were deeply studied to get efficient details regarding the process.

I. INTRODUCTION

The literature review gives detailed study of various research paper all along is based absolutely on the format and assessment of the front hub for weight reduction and growth the strength of hub via the help of Nastran / Patran. The weight and dimension of the hub want to be as small as feasible because of the un sprung weight which further reduce the rotational mass.[1] Engineering thing with closing use of material and smooth manufacturability is a path in which preceding simulation via finite element technique is located to be very useful. Wheel hub of car is one of the primary and without a doubt vital component and needs first-rate material and format in low charge and avoids failure. The three number one elements of a wheel are the hub, the spokes and the rim. Sometimes those components can be one piece, every now and then or three. The hub is the center a part of the wheel and is the detail in which the wheel is established to the suspension via the wheel carrier (or knuckle).[2] The spokes radiate out from the hub and connect to the rim. The rim is the outer part of the wheel that holds the tire. A hub assembly carries the wheel bearing and hub to mount the wheel to vehicle. It is placed most of the brake rotor and axle.

The following research paper gives a brief study of various studies conventional methods of designing and manufacturing a Wheel Hub and brake rotor. Analyzed for all the possible loading conditions. These conditions were derived using a set of formulas which were used to derive various forces acting on the components.[3]

In custom-made brake rotors, brake fade and thermo-elastic deformation are common problems. During the braking phase, the vehicle's kinetic energy is transformed to mechanical energy. Because of the large frictional forces, more frictional heat is generated. The rotor temperature rises as a result of this heat. As a result of the combined influence of mechanical and electrical forces, the rotor deforms as the stresses and temperature increase. This issue is particularly acute in all-terrain vehicles (ATVs). In order to avoid this, because of this issue, a rotor pre-manufacturing examination is required. The mechanical forces operating on the rotor were investigated in this work.

Friction between the caliper pads and the rotor surface generates heat. Brake rotor diameters can be tweaked to fit your needs. With a reduction in weight, you can gain strength. CFD analysis of the rotor should also be performed to determine the exact value of the film coefficient of friction. Heat transfer by convection. In the ANSYS Workbench 14.0 software suite, the rotor under investigation was modelled, analyzed, and optimized. The dimensions of the rotor were chosen. Within the permitted range, rotor dimensions were chosen from 42 potential combinations of rotor diameter and thickness.

This sequential strategy was used to develop the rotors for the BAJA vehicle, and it proved to be successful. The rotor created in this study was constructed specifically for a BAJA racing ATV. The Society of Automotive Engineers hosts BAJA every year (SAE). When a product is custom produced, such as a brake rotor, it is necessary for a designer to test its reliability in high-temperature applications in order to choose the appropriate material and heat treatment procedures. The unsprung mass and inertia of spinning components have a crucial effect in race car vehicle dynamics. For any vehicle to accelerate better, it is always suggested to keep the mass of the vehicle as low as possible. As a result, it becomes the engineer's primary responsibility to find a balance between strength, dependability, and weight, allowing them to go deeper into the subject of optimization. Brake rotors or discs are responsible for providing a smooth braking surface for the pads to contact.

When force is given to the pedal, pressure is created at the calliper’s end, and the pads are pressed on the rotor surface. When the pads make contact with the spinning rotor, the friction slows the rotor and brings the wheel to a halt. The rotor absorbs and dissipates a large portion of the frictional heat generated. Solid rotors and vented rotors are the two most common types of rotors [4].

A single-seat, open-cockpit, open-wheel off-road vehicle with the engine behind the driver is known as an all-terrain vehicle (ATV). The critical components of designing and developing the front wheel hub of an ATV are discussed in this study. The design of the front wheel hub is discussed in this study, with the goal of making it light weight and strong. The material selection for the hub is discussed in this study, which includes two different types of material i.e., Al-7075-T6 and EN8, based on material strength to ratio.

With the use of numerous studies of the hub, this study aims to improve the design of the hub. With the help of ANSYS software's static structural module, the hub is examined in various loading scenarios to determine the suitable factor of safety. A front wheel hub is a component that connects the wheel to other suspension components via a stub axle and allows the wheel to spin freely on the bearing while being linked to the vehicle. For usage on automobiles and other vehicles, the brake rotor and wheel hub were combined into a single unit. Different hubs are used in traditional ATVs, and brake rotors are often made as two separate sections that are fastened together to enable for the repair of worn-out brake rotors. The disc has a short lifespan compared to the hub, which rarely needs to be replaced.

This model necessitates extra machining, which might lead to unbalance and misalignment during manufacture. The current method of producing the hub and rotor as two independent pieces necessitates the use of additional material for bolt flanges, resulting in a larger overall size. The reduction in the hub's rotational mass and the overall weight of the ATV result in an increase in acceleration. Furthermore, because to the motion of the ATV, such as braking, cornering, and six sense falls, the hub is subjected to continual pressures and impact stress. As a result, the hub should be constructed with a low weight and high strength in mind.[1]

A disc brake is a device that slows or stops a wheel's spin. When a car brakes repeatedly, heat is generated with each braking event. The purpose of the Transient Thermal and Structural Analysis of the Rotor Disc of Disk Brake is to evaluate the performance of a car's disc brake rotor during severe braking circumstances in order to aid disc rotor design and analysis. ANSYS Workbench 14.5 is used to model and analyse disc brakes.

The primary goal of this research is to examine the thermomechanical behaviour of the brake disc's dry contact during the braking phase. To improve rotor disc performance, the coupled thermal-structural analysis is utilised to determine the deformation and Von Mises stress established in the disc for both solid and vented discs using two distinct materials. All of the values obtained from the analysis are smaller than their permitted values, according to a comparison between analytical and FEM results. As a result, the most appropriate design, material, and rotor disc are recommended based on performance, strength, and stiffness parameters. The amount of time it takes to cool the disc is little. The area of contact between the disc and the pads is greater when the braking disc is in a solid body. A vented disc is commonly used in disc brake systems in car braking systems for enhanced cooling during braking, but the area of contact between disc and pads remains constant.

In this paper, a transient thermal analysis will be performed using ANSYS to explore the temperature variation across both discs by applying heat flow values for repeated braking applications. Coupling thermal analysis is used to perform additional structural analysis. In addition, the convection heat transfer coefficient is applied to the ventilated disc's surface. When comparing the varied temperature rise, deflection, and stress field data acquired from analysis, it is clear that the vented cast iron disc reduces temperature, stresses, and deformation by 31.47 percent, 22.5 percent, and 8 percent, respectively, when compared to the solid disc. The vented type disc brake is determined to be the best for the current application. All of the results of the analysis are less than the permitted limits. As a result of the strength and stiffness standards, the brake disc design is safe.[5]

The largest automotive assembly part of an automobile is wheel assembly, often known as hub assembly. The design of the Wheel assembly determines the dynamics and control behaviour of any vehicle. Design concerns and methods vary widely depending on the vehicle's purpose and intended use. This is mostly because the All-Terrain Vehicle is a source of concern. The vehicle's total unsprung mass was reduced as a result of this, and the vehicle's unsprung mass was reduced as well. The vehicle's dynamic performance was improved. On one side, the full wheel assembly is attached to the wheel rim with four mounting points, and on the other side, it is attached to the suspension control arms and tie rod, including both, with three mounting points.

The hub is the component of the wheel assembly to which the rim and disc are bolted and which is press-fit on the spindle through roller bearing. It is fixed from the bearing sheet during the analysis while two torques (driving and braking) and three forces (two bump and lateral) are applied to the relevant hard points. The rim's internal diameter is 10 inches, which limited the disc's size. The disc's maximum diameter was chosen after sufficient clearance was taken into account. During disc analysis, the vehicle's kinetic energy was measured in terms of heat flux applied to the contact patch of the disc and calliper pad. We chose AISI 1018 for brake disc manufacture because it has a good conductivity and appropriate strength.[6]

5. Thermal Analysis of disc Brakes Rotor: A comparative Report Shah E Alam1 , Yuvraj Vidhyadhar2 , Prashant Sharma3, Abhishek Jain4 1,2UG+PG Students, B.Tech+M.Tech (M&AE), Amity University Rajasthan, JAIPUR 3UG Student, B.Tech (M&AE), Amity University Rajasthan, JAIPUR 4Assistant Professor (M&AE), Amity University Rajasthan, JAIPUR

The authors present their findings from a thermal examination of a disc brake rotor used by two-wheelers in India in this paper. The purpose of this study is to figure out what the holes in the disc brake are for. Thermal analysis is carried out on two separate rotor types. The first is a plain rotor with no vents or holes, whereas the second is perforated (consists of holes). Researchers tried to figure out how much heat is lost from a rotor that is heated by disc brake friction when it is in operation. Researchers include convection and radiation when calculating heat loss. For both discs, the findings are compared.

The better-performing design is discovered based on the results acquired. The authors of this research examined two types of rotors, one with no holes and the other with holes, on the basis of heat dissipation. The purpose of this study is to see if the holes in the perforated rotor help with heat dissipation. Both rotors are the same size and material, and they were subjected to the same operating conditions before being examined with the identical boundary conditions. On the ANSYS workbench, a transient thermal analysis was performed to determine which of the two was the best. The outcomes are as predicted. It is thought that the perforated model will be more effective. The perforated disc proved to be more efficient, as the body's maximum temperature after 50 seconds was lower than the simple disc. This, however, is only one component of the in-depth analysis. The Thermal Stresses owing to Thermal Expansion will be examined next. Both discs can be subjected to structural analysis to assess the cumulative effect of temperature, i.e., thermal stresses and deformation (if any) caused by thermal expansion.[2]

6. Structural Analysis of Disc Brake Rotor K.Sowjanya#1 , S.Suresh*2 1M. Tech-(CAD-CAM), Siddharth Institute of Engineering and Technology, Puttur 2Associate Professor, Siddharth Institute of Engineering and Technology, Puttur

The focus of this research is on disc brake analysis. A brake is a device that applies artificial frictional resistance to a moving machine component in order to stop it from moving. Because disc brakes are typically made of cast iron, they were used to investigate the impact of strength differences on projected stress distributions. Materials for Aluminium Metal Matrix Composites are chosen and analysed. The results are compared to disc rotors that are already on the market. The Disc brake model was created with the Solid modelling software Pro/E. (Creo-Parametric 1.0). ANSYS Workbench is used to perform additional static analysis. The Deflection, Normal Stress, and Vonmises Stress are all determined through structural analysis. Grey cast iron is often used to create disc brake discs.

It was created at the same time that the usage of high-speed electronic digital computers grew, as did the emphasis on numerical methods for engineering analysis. A static analysis assesses the effects of constant loading conditions on a structure while neglecting inertia and damping effects induced by time-varying loads. Static analyses, on the other hand, can incorporate constant inertia loads (such as gravity and rotational velocity) as well as time-varying loads that can be represented as static equivalent loads (such as static equivalent wind and seismic loads). The greatest Von Mises stress was discovered after doing a static structural analysis by linking the Thermal solution to the structural analysis. Based on the Strength and Rigidity Criteria, the brake disc design is safe. When the various outcomes of the analysis are compared, it is determined that Cast Iron is the best potential combination for the current application.[7]

Wheel hubs must be designed based on stresses caused by operational loads operating on the wheel during consumer use. As a result, the service loading circumstances, as well as the generated stresses and hub fatigue qualities, must all be considered. The critical design and durability factors - operational loads, fatigue properties, which are dependent on material and manufacturing technique, and design - are reviewed in this paper, as well as the procedure for an optimal light-weight design.

The loading conditions, material behaviour, and manufacturing process are the most important design parameters. The stress level and stress state on the hub, as well as the service life, are influenced by operational loading and hub design. The resulting tension at any location on the hub is made up of residual stresses from manufacturing and component assembly, which are then overlaid on service stresses caused by wheel rotation. Vertical (Fz) and lateral (Fy) wheel loads create the most service pressures on the hub during driving in various load circumstances such as straight ahead on bumpy roads, extreme cornering, and off-road or shunt type loading. Aside from the load and stress spectrum, the permissible stresses determine the hub's fatigue life. Material qualities, residual stresses, pre-stress conditions, and the manufacturing process all influence the allowed stresses on each system point of the hub. Only if service-like deformations of all components can be assured and a load programme that corresponds to the mission in service can be replicated can laboratory testing provide verification of fatigue strength and adequate function in service.

As detailed in this study, operational loads can be defined by standardised spectra that are dependent on vehicle specifications and usage conditions. A test facility is required for the durability approval, where service-like deformations of the hub and neighbouring components can be replicated. The durability of the entire wheel-hub-subassembly can be verified using the given biaxial wheel/hub test facility. If there is a question of product liability, the implementation of standardised load programmes provides a common basis for proving acceptable fatigue strength in service and acts as a basic document related to the state-of-the-art approval. Validation of new designs, materials, and manufacturing processes can be done in a reliable, fast and cost-effective manner using the test procedure described.[8]

Nowadays, commercials are everywhere. Separate hub and rotor are used on all-terrain vehicles (ATVs). They have done design, evaluate, and manufacture INTEGATED HUB AND ROTOR (Brake Disc) for all ATVs in this research report. CATIA V5 and ANSYS WORKBENCH 16.0 are used to design and analyse components, respectively. Initially, we created an integrated hub and rotor for an ATV in the SAE BAJA 2017 competition, which proved to be quite reliable. We are constructing an integrated hub and rotor (IHR) in this study, taking into account various materials in order to reduce weight, overall size, and manufacturing costs. The rotor's radial outward flanges are removed, and the rotor is linked directly to the hub's cylindrical component.

The weight of the wheel assembly is reduced significantly due to the absence of flanges and joining elements like as nuts and bolts, which also helps to optimise the performance of the ATV with integrated hub and rotor. Because the functions of the hub and rotor differ, the hub requires high strength and ductility because it is subjected to severe loading during operation. Rotor, on the other hand, necessitates a high level of hardness, wear resistance, and compressive strength. Also, a higher thermal storage capacity is required to minimise distortion and cracking caused by thermal stresses, which is critical in the case of repeated high-speed stops.

To determine the temperature of the brake rotor during braking, a transient thermal analysis was done. For 2.5 seconds of braking, we estimated heat flux. Because the temperature attained after braking is minimal, thermal qualities that fluctuate over time can be ignored. Temperature distribution during braking is also studied using thermal analysis, as the degree of the wheel hub should not exceed a certain temperature. After considering the thermal behaviour and stress analysis of four materials, it has been determined that SG Cast Iron is the best choice for IHR, while En 24 and En 8 are the second and third choices, respectively.[9]

Study paper's main focus was on the design and experimental analysis of a brake system for an All-Terrain Vehicle (ATV) that competed in BAJA 2016. CATIA-V5 was used to design each component of the braking system, while ANSYS16 was used to analyse stress and temperature distribution. The braking material, heat generation, and heat dispersion of the disc were all key elements in the experimental investigation of the brake system. The materials were chosen based on yield strength, thermal conductivity, cost, and market availability.

To solve this problem, they devised a unique design known as the "SANDWICH ROTOR DESIGN." Normal disc cooling takes place by convection solely in the SS420, but because to the sandwich construction, aluminium acts as a heat sinker, and cooling now takes place by both conduction and convection. As a result, the rate of cooling rises. Using innovative brake techniques and analysing the best results for disc heat and stress distribution, rotor size, and power absorbed. Heat Flux, Kinetic Energy of the Vehicle, and Rotor Heat Absorption are all estimated to optimise the braking capability of the braking system.[10]

The wheel hub's weight was reduced as a result of this improved material without sacrificing functionality. This project is based on the design and analysis of a front hub using Nastran / Patran to reduce weight and boost hub strength. Because of the unsprung weight, the hub's weight and dimensions must be as little as possible. This reduces the rotational mass even further. Prior simulation using the finite element method has been shown to be very effective in engineering components with optimal material consumption and easy manufacturability. The wheel hub of an automobile is one of the most vital and important components, and it requires high-quality materials and design at a reasonable cost to avoid failure. The hub, spokes, and rim are the three essential components of a wheel. These components can be one piece, two pieces, or three pieces. The hub is the centre of the wheel, and it's where the wheel is connected to the suspension via the wheel carrier (or knuckle). The spokes extend from the hub and connect to the rim. The rim is the component of the wheel on which the tyre is mounted. To install the wheel to the vehicle, a hub assembly contains the wheel bearing and hub. It's found halfway between the braking rotor and the axle. The wheel hub is a critical component for street safety that must not fail under strain. Loading situation, manufacturing method, and material behaviour are the most important factors to consider while designing a wheel hub assembly. Due to the interactive nature of these characteristics, material fatigue behaviour will vary depending on wheel hub design and loading state.

The rim size, bolt pattern, and weight of the car all play a role in hub design. CATIA V5 R20, a 3D modelling software, was used to create a CAD model of the Wheel Hub. The rim size, bolt pattern, and weight of the car all play a role in hub design. The model is exposed to harsh conditions and a static analysis is performed in Nastran/Patran to determine the maximum stress produced in Hubs. At the rim mountings, the hub has a limitation. Braking Torque and Bump Force were determined analytically and applied to the Hub.

The Wheel Hub was created for an ATV with a mass of 440 kilogrammes and a top speed of 60 kilometres per hour. The Designed Hub provides stability while the wheels are rotating. The Hub's weight and dimensions are such that the rotational mass is reduced.[10]

The disc brake is one of the most important components in any automotive because it is used to reduce the vehicle's speed. The braking pads and discs are subjected to extreme wear and high temperatures during braking operations. The existing braking of vented disc brakes is modified in two stages, and analysis is carried out using simulation-based software Ansys to increase the life span and stress characteristics in this publication. The hole diameter of the disc and the contact area between the brake pad and the disc are modified in the modification, with the total weight of the braking system in mind.

The heat conductivity of the rotor can be enhanced by increasing the size of the hole on the rotor, according to various writers. On the other hand, while increasing the hole size, it is important to consider the mechanical properties of the rotor, as this can weaken the rotor and cause the braking system to fail. To retain acceptable structural strength, the hole diameter of the rotor was raised in this study, and the thickness of the disc plate was also increased. The improved designs were then subjected to study for maximum deformation, stress distribution, and life span. The results showed that, despite increasing the hole diameter, the maximum stress level in the redesigned designs was lowered to 435.92 MPa from 518.26 MPa for Disk 2. However, for the updated designs, there is a minor increase in distortion, although it is still within acceptable limits. It was discovered that the maximum life span has been reached.[11]

Grey cast iron is widely used to make rotor discs[1]. For many uses, the SAE maintains a specification for the manufacturing of grey iron[2]. It is principally made up of Iron 95 percent, Carbon 2 to 5 percent, Silicon 1 to 3 percent, and minor amounts of Sulphur, Manganese, and Phosphorus. Grey cast iron has a high specific heat capacity and thermal conductivity, making it ideal for rotor disc production. Young's Modulus of Elasticity, Torsional Modulus of Elasticity, Crushing Strength, Brinell hardness, and Endurance Limit are some of the other qualities.

Grey cast iron has the following material properties:

7100 kg/m3 density 125 GPa Young modulus

Specific heat: 586 J/Kg.K Poisson's ratio: 0.25

54 W/m thermal conductivity

We investigated five rotor disc materials for thermal analysis: grey cast iron, aluminium metal matrix composites, E glass fibre, ceramic, and hard rubber. The material of the rotor disc is homogeneous and isotropic. Axisymmetric is applied to the issue domain. During the analysis, the effects of inertia and body force are minimal. There is no stress in the rotor disc before the brake is applied. The brakes are applied to all four wheels. The rotor disc is of the solid kind, and it is not vented. The substance utilised for analysis has a constant thermal conductivity. Only ambient air cooling is taken into account. The rotor disc material's specific heat is constant and does not change with temperature.

The following are the key conclusions drawn from the work programme described in this study. For the duration of 4 seconds, a thermal analysis is performed for the application of braking force due to friction. The maximum temperature measured in a grey cast iron rotor disc is 167.56 degrees Celsius at the rubbing surface. Based on strength and stability standards, the grey cast iron rotor disc is safe. To find the best combination of rotor disc factors such as size, type, and materials, heat analysis for five distinct material combinations was performed, and the results were validated using digital logic and fracture analysis.[3]

A rallycross car's wheel hub is examined in order to lower the vehicle's unsprung mass. The problem statement focuses mostly on the suspension's unsprung mass, and more especially on the Wheel Hub. One of the goals of a car's suspension system is to dampen the vehicle's movement. Due to the extreme conditions that may result in lower traction and a loss of confidence for the driver, this is a crucial issue to consider when constructing a vehicle, especially a race car. To explain the relevance of having a low unsprung mass and how it impacts vehicle handling, a theoretical model is used. The parts are analysed using the Finite Element Method to verify the design and material selection. A process to evaluate the current Wheel Hub and make a decision to re-design the Wheel Hub is performed, and the parts are analysed using the Finite Element Method to verify the design and material selection.

The conclusion reached while working on this project is that there are numerous ways to improve the Wheel Hub. The design created in this research is a significant step forward in reducing the car's unsprung mass. The geometry of the suspension, and thus the car's behaviour in that regard, will be unaffected by the design. The benefits of switching to the redesigned Hub include increased automobile acceleration and improved tire-to-road traction. The constraints imposed on this project make it tough to make the necessary revisions to arrive at the best design. Even if the design improves the car by making little adjustments, modifying the upright and suspension geometry can result in a more inventive design with less unsprung mass and benefits in suspension geometry, heat flow, and other areas.[12]

The investigation and analysis of the Mahindra wheel hub using Finite Element Analysis is covered in this work (FEA). The modern automobile industry is moving toward vehicle speed acceleration by reducing weight while preserving strength. We looked at a number of weight-loss studies that had previously been conducted. Our goal is to use material optimization to lower the weight of the wheel hub. We compared other classes of materials, such as aluminium alloys, to the existing material. The best substitute for the existing material was found to be aluminium in the grade 7075 A6. Using FEA, we demonstrated that the new wheel hub is light in weight and has the same load carrying capacity as the old one while maintaining the hub's robustness. We also concentrated on wheel hub fatigue studies to determine the wheel hub's life.

The existing wheel hub is comprised of mild steel, which has a high density but low strength. In the case of mild steel, the maximum stress is likewise slightly higher. We discovered that the stress created is within the material's allowed limit after study and material optimization. Furthermore, the weight of an Al 7075-T6 wheel hub is 42% less than that of mild steel or stainless steel. In addition, the strength is around 1.5 times that of steel. As a consequence of analysing the results of various materials, Aluminium 7075-T6 is chosen as a feasible solution for creating a wheel hub. • However, due to its high market price, 7000 series aluminium is mostly used in the aerospace and motorsports industries. Reduced market prices for 7000 series aluminium could lead to improved utilisation in a variety of industries, resulting in energy savings. The design created in this research is a significant step forward in reducing the car's unsprung mass. The geometry of the suspension, and thus the car's behaviour in that regard, will be unaffected by the design. The analytical design (optimised current Wheel Hub) decreases mass by 41% and may be implemented with only minor changes and new elements. Although this design is not perfect, it will minimise the total mass of the Wheel Hub, which was one of the goals.[13]

Diverse methodologies are used in this study with reference to available data from prior articles and reviewers' comments for wheel hub optimization using various materials. This research aids designers in selecting the optimum material for wheel hubs while reducing weight and ensuring the greatest structural design. The FEA methodology is used to optimise the weight of the wheel hub. The hub model is created using actual dimensions, and it is then tested using FEA analysis with various materials. All optimization processes are carried out using modelling software Catia version 5 R19 and FEA tool Ansys workbench 16. The best suit material for wheel hub is discovered with the lowest weight and cost. We may analyse the major areas of research such as wheel hub shape optimization, static load analysis on wheels, and fatigue load analysis utilising FEA in this wheel hub weight optimization method.

As a result of the combination of load instances, maximum stress is created in various types of materials. Bump + Breaking torque and Cornering load + Breaking torque completely match the field failure. For each tabulated material, the maximum stress values and deformation are compared. Aluminium alloy is a better material when comparing Ansys and FOS results, as well as weight to strength ratios for all materials. It can be employed as an ideal material for improved wheel hub design and analysis. We can infer that Aluminium alloy 6061 material is a viable alternative for the wheel hub by evaluating the weight to strength ratio of the aforesaid materials. This material reduces the weight of the hub without sacrificing its strength.[14]

This thesis outlines the design of a hybrid HMMWV's wheel hub in order to incorporate an electric in-hub motor. The first chapter discusses the consumption of fossil fuels and their negative consequences. The limitations of currently available alternative fuels are also discussed. The concept of hybrid electric vehicles, its numerous types and combinations, and the various military hybrid vehicles are covered in Chapter 2. In the following chapters, vehicle performance characteristics such as velocity, acceleration, and gradeability are examined for various scenarios, and the calculation of gear ratios is discussed. The design restrictions are given, and the design was created based on the available space and the weight factor. The idea was also put through the test for vehicle crushing forces known as 20G and 8G. The material for the design was high-strength steel. The design's strength is checked by comparing the results to existing data. In all circumstances, Von Mises stresses are fewer than yield stresses. The nodal displacements are within a safe range. The components are modelled in Pro E, and the analysis is done in Algor.

The Gear Hub assembly is intended to fit within the available space and has withstood 20G and 8G stresses. The maximal Von-Mises stresses are always greater than the yield stresses. For all of the examples, nodal displacement is within acceptable limits. For various scenarios, vehicle performance analysis is also performed. Depending on the necessity (acceleration and hill climb) and space available, a customer can utilise any gear ratio between 4:1 and 6:1. If a different motor is available within the space constraints and provides more torque, it can be used. This design allows for one engine per wheel, lowering the weight of mechanical components like the transmission and differential. It reduces the ease with which each wheel's torque can be controlled, resulting in faster acceleration and greater agility, including pivot steering. Because the components in this assembly are not complex in shape, modified and newly designed components can be fabricated with ease. This concept is simple to implement into a traditional automobile assembly line. Additional components are minimal and can be inserted by employees on the production line. The number of usable parts is reduced, and disassembly is rapid and simple. In order to create a competent suspension and steering system for this assembly within the limited area, more research is required. In this instance, an A-arm suspension system is recommended.[15]

The goal of this study is to figure out what went wrong with an FSAE (Formula SAE) automobile front wheel hub that failed during testing. The hub of a Formula Student car must withstand forces throughout various dynamic conditions such as cornering, braking, and acceleration. Aluminium6061-T6 alloy was used to make the wheel hub, which was anodized to a depth of 25 microns. The defective component was subjected to a SEM examination in order to determine the root causes of the hub's failure.

Aluminium 6061 T6 alloy was used to make the wheel hub. Table 1 shows the chemical composition expressed as a percentage of total weight. The intermetallic phases in the aluminium metal matrix were dissolved by heating the alloy to 500 C for 9 to 10 hours. The solution zed microstructure was then frozen by quenching. After that, the material was artificially aged by heating it to 1800 C for 9 to 10 hours and then cooling it in air. The wheel assembly's hub and upright are composed of aluminium 6061-T6.

Anodized layer thickness: 25 microns Sulphuric acid is an electrolyte.

18-volt power supply

80 Amps of current

Time to complete: 45 minutes

The finite element wheel hub analysis was carried out to determine the stresses that could have caused the hub to fail. The stresses and limitations were discovered before to the investigation and then applied to the component. Findings from finite element analysis necessitate both stress distribution and deformation at various component locations. The stress value and distribution data were utilised to look into the causes of failure. In the ANSYS 16.0 environment, the entire component was meshed. The meshing of the tetrahedral elements was done with a 2 mm element size. The meshing was fine-tuned where there was a break in the shape of a step or a hole, for example. The total number of nodes was 75,700, with a total of 43,944 elements. The hub was subjected to an axial force of 1806.635 N and a torque of 223.3 N mm for analysis 160 MPa was discovered to be the maximum stress.

The failure causes are determined based on the findings of the SEM micrographs and the component's FEA analysis. The combined effect of strains due to bending, torsional shear, and axial stress is the principal cause of hub failure as a complete wheel hub assembly. The failure appears to have been caused by shear-induced cracking, based on the nature damaged surface. According to the FE study, the crack started in the weaker portion of the petal, which could be due to excessive stress localisation. The failure was caused by fatigue loads that developed on the wheel hub during braking and cornering, according to the FE and SEM analyses.[15]

According to this Research paper, The braking mechanism is one of an automobile's inevitability and safety-critical components. During the braking mechanism, a good rotor design and improved heat dissipation material provide higher performance. This research examines the stress and temperature distribution of a redesigned ventilated disc brake rotor with curved vents, holes, and slots. SolidWorks is used to create rotor finite-element models, which are then simulated using ANSYS. The structural and thermal properties of the rotor are compared to those of a reference disc brake rotor. In terms of stress generation, temperature distribution, and factor of safety, the proposed rotors surpassed the traditional ones. In addition, computation was performed to determine the Best suited material for one of the proposed designs. The end product gives a physical understanding of the structural and thermal properties of the geometrically redesigned rotor, which can be used in the automotive industry.

In this work, the properties of proposed disc brake rotors (MM1, MM2, and MM3) were compared to those of a reference rotor using steady–state thermal and static structure analysis. Hole, straight slots, vanes, and edge cuts have all been used in the creation of the proposed models. For mesh generation and numerical simulation, the ANSYS platform is employed. In structural analysis, the MM2 and MM1 are found to produce better results, with percentage (percent) improvements of 12.48 percent for equivalent stress and 22.47 percent for deformation, respectively. MM2 outperforms MM1 in terms of the safety factor. For the updated model MM2, various materials were used, and it was discovered that cast iron outperforms steel and CFRP laminate, while CFRP laminate generates less heat. After examining all data, it can be stated that the modified models outperform the standard disc brake rotor in terms of achieving high brake force while avoiding cracking and buckling during operation.[16]

II. RESULT AND DISCUSSIONS

Most of the studies involves separate manufacturing of wheel hub and brake rotor. Many researches showed use of Aluminum which is more expensive than EN8. Many models were not analyzed for the cornering force opposed by the steering system while taking a turn.

III. CONCLUSION

Various studies depicted use of wide range of materials thermal and structurally sound. All the manufacturing operation to attain high geometrical tolerance prototype to fulfill all the need of the various loading conditions such braking or clamping force , heat generated by clamping forces, torque on wheels, cornering force and bump force in various magnitudes and directions.

Hence widely the researchers opted for EN 8 for is structural and thermal properties which helped in attaining various goals that are put forward by the boundary conditions.

REFERENCES

[1] , Verma, S. Kumar, R. Bharj, and R. Kumar, “Design and development of the front wheel hub for All-Terrain Vehicle (ATV) / Himanshu Verma...[et al.],” *J. Mech. Eng.*, vol. 17, no. 1, pp. 49–62, 2020.

[2] A. J. Shah E Alam1, Yuvraj Vidhyadhar2, Prashant Sharma3, “Thermal Analysis of disc Brakes Rotor : A comparative Report,” *J. Inf. Sci. Comput. Technol. ISSN 2394-9066 SCITECH*, vol. 3, no. 2, pp. 196–200, 2015.

[3] Ishwar Gupta & Gaurav Saxena, “Structural Analysis of Rotor Disc of Disc Brake of BAJA SAE 2013 Car through Finite Element Analysis,” *Int. J. Automob. Eng. Res. Dev.*, vol. 4, no. 1, pp. 1–10, 2014, [Online]. Available: http://www.tjprc.org/view_archives.php?year=2014&jtype=2&id=56&details=archives.

[4] H. Nikam, P. Mishra, and S. Bharambe, “Design and Analysis of Brake Rotor with Parameter Optimization,” *Int. J. Automob. Eng. Res. Dev.*, vol. 4, no. 4, pp. 21–30, 2014, [Online]. Available: http://www.tjprc.org/view-archives.php?year=2014&jtype=2&id=23&details=archives.

[5] V. Dadi, R. K. Koteswara, and J. H. N. Rao, “Structural and Thermal Analysis of Rotor Disc of Disc Brake,” *Int. J. Sci. Eng. Adv. Technol.*, vol. 3, no. 9, pp. 477–482, 2015.

[6] N. Malik and P. Agarwal, “Design and analysis of wheel assembly of an ATV,” *Int. J. Mech. Prod. Eng. Res. Dev.*, vol. 7, no. 5, pp. 75–82, 2017, doi: 10.24247/ijmperdoct20178.

[7] K. Sowjanya and S. Suresh, “Structural Analysis of Disc Brake Rotor,” vol. 4, no. 7, pp. 2295–2298, 2013.

[8] G. Fischer and V. V. Grubisic, “Design criteria and durability approval of wheel hubs,” *SAE Tech. Pap.*, no. 724, 1998, doi: 10.4271/982840.

[9] M. A. Patil, B. Kale, A. Bhagade, C. Pimpalkhute, and A. Borkar, “Design and Analysis of a gearbox for an all terrain vehicle,” vol. 4, no. 11, pp. 690–701, 2018, doi: 10.9790/1813-0705011016.

[10] “Optimum Design and Experimental Analysis of Brake System for BAJA ATV Related papers Optimum Design and Experimental Analysis of Brake System for BAJA ATV.”

[11] S. Mithlesh, Z. A. Tantray, M. Bansal, K. V. K. P. Kumar, V. S. Kurakula, and M. Singh, “Improvement in performance of vented disc brake by geometrical modification of rotor,” *Mater. Today Proc.*, vol. 47, no. xxxx, pp. 6054–6059, 2021, doi: 10.1016/j.matpr.2021.05.006.

[12] E. Andersson, “OPTIMIZATION AND RE-DESIGN OF A WHEEL HUB TO REDUCE UNSPRUNG MASS OF A RALLYCROSS CAR Bachelor,” pp. 1–49, 2018, [Online]. Available: https://www.diva-portal.org/smash/get/diva2:1221239/FULLTEXT01.pdf.

[13] S. B. Kokate and G. R. Kulkarni, “Material Optimization of Wheel Hub using Finite Element Analysis,” *Int. Res. J. Eng. Technol.*, no. May, pp. 7252–7258, 2008, [Online]. Available: www.irjet.net.

[14] A. Of, W. Hub, F. O. R. Weight, O. By, and U. Verious, “World Journal of Engineering DESIGN AND

ANALYSIS OF WHEEL HUB FOR WEIGHT," vol. 7, no. 1, pp. 161–206, 2021.

[15] P. Hervás, R. & Millares, "No Title ال قوة عة," *CWL Publ. Enterp. Inc., Madison*, vol. 2004, p. 352, 2004, [Online]. Available: http://onlinelibrary.wiley.com/doi/10.1002/cbdv.200490137/abstract.

[16] M. H. Pranta, M. S. Rabbi, S. C. Banik, M. G. Hafez, and Y. M. Chu, "A computational study on structural and thermal behavior of modified disk brake rotors," *Alexandria Eng. J.*, 2021, doi: 10.1016/j.aej.2021.07.013.