Structural Analysis of Automotive Disc Brakes

Y Madhu Maheswara Reddy
Assistant Professor, Mechanical Engineering Department
Methodist College of Engg. & Tech.
Hyderabad, India
mmr315@gmail.com

DR. P Usha Sri
Professor, Mechanical Engineering Department
Osmania University
Hyderabad, India

M Prasad
Assistant Professor, Mechanical Engineering
Methodist College of Engg. & Tech.
Hyderabad, India
prasadmatam@gmail.com

Abstract - **The braking system plays the crucial role in the safety of a vehicle and its design poses a critical challenge. The heat dissipation and thermal performance of ventilated brake discs strongly depends on the aerodynamic characteristics of the air flow through the rotor passages.**

In this thesis, the 3D CAD models for Solid disc and discs with different fin configurations namely, Straight Elliptical fins, Backward curved fins and Forward curved fins are designed. Each finned model disc with 12, 24 and 36 number of fins were developed using Catia V5R20. The 3D FE models have been developed for the above disc configurations using Hypermesh V11 and to know the structural strength of the disc brake, due to the introduction of fins commercial structural analysis ANSYS13.0 was used. It was observed that the introduction of fins is having a very little effect on the structural strength of the discs. Hence Forward curved disc with 36 numbers of fins is suggested for higher structural strength.

Key Words – Brake, Structural Analysis, Disc, Fins, Ansys.

I. INTRODUCTION

The disc brake or disk brake is a wheel brake which slows rotation of the wheel by the friction caused by pushing brake pads against a brake disc with a set of calipers. The brake disc (or *rotor* in American English) is usually made of cast iron, but may in some cases be made of composites such as reinforced carbon–carbon or ceramic matrix composites. This is connected to the wheel and/or the axle. To stop the wheel, friction material in the form of brake pads, mounted on a device called a brake caliper, is forced mechanically, hydraulically, pneumatically or electro magnetically against both sides of the disc. Friction causes the disc and attached wheel to slow or stop. Brakes convert motion to heat, and if the brakes get too hot, they become less effective, a phenomenon known as brake fade.

Disc-style brakes development and use began in England in the 1890s. The first caliper-type automobile disc brake was patented by Frederick William Lanchester in his Birmingham, UK factory in 1902 and used successfully on Lanchester cars. Compared to drum brakes, disc brakes offer better stopping performance, because the disc is more readily cooled. As a consequence discs are less prone to the "brake fade"; and disc brakes recover more quickly from immersion (wet brakes are less effective). Most drum brake designs have at least one leading shoe, which gives a servo-effect. By contrast, a disc brake has no self-servo effect and its braking force is always proportional to the pressure placed on the brake pad by the braking system via any brake servo, braking pedal or lever, this tends to give the driver better "feel" to avoid impending lockup. Drums are also prone to "bell mouthing", and trap worn lining material within the assembly, both causes of various braking problems. The aerodynamic characteristics of the mass flow were found to be reasonably independent of rotational speed, but highly dependent upon rotor geometry. Johnson et al. [2] used PIV (Particle Image Velocimetry) to measure air velocities through a high solidity radial flow fan utilized as a vented brake rotor. Choi and Lee [3] performed a transient thermo-elastic analysis of disc brakes in repeated braking applications, using a finite element method with frictional heat generation. Grieve et al.[4] performed parametric sensitivity studies to define suitable design-material combinations for a disc brake prototype.

For solid rotors, the highest temperature occurs on the surfaces of the rotors. To decrease the maximum temperature, the most effective way is to increase the thickness of the rotors. However, the increase is limited by the pistons. Materials also have effects on the rotor's temperature. From this research, steel is a better alloy to dissipate heat from the rotors. But in practical design problems, the thermal performance is not the only requirement. From the perspective of stiffness, friction resistance and cost, the cast iron material is common used in industry.

The powerful simulation packages HYPERMESH and FLUENT give more accurate solutions to fin and pillar post rotors. From the sensitivity studies, the fin rotors have better cooling performance.

The objective of current study is to estimate Von Mises Stress generated in solid discs. The combinations of various fin configurations namely Straight Elliptical, Backward Curved, Forward curved and the various numbers of fins (12, 24 and 36) are studied to predict the effect of these variables on the performance of disc brakes. The drawings for the Solid

Thermal Analysis on Disc Brakes

M Prasad
Assistant Professor, Mechanical Engineering Department.
Methodist Coll. of Engg. &Tech.
Hyderabad, India.
prasadmatam@gmail.com.

Y Madhu Maheswara Reddy
Asst Professor, Mechanical Engineering Department
Methodist Coll. of Engg. &Tech.
Hyderabad, India.
mmr315@gmail.com.

VS Ramesh Reddy
Asst. Professor, Mechanical Engineering Department.
Methodist Coll. of Engg. &Tech.
Hyderabad, India.
vsrreddy@gmail.com.

Abstract—**Braking system is one of the most essential functions of an automobile. Therefore, it is a must for all vehicles to have proper brake system. Due to critical system in the vehicle, many of researchers have conducted a study on brake system and its entire component. In this project, we has conducted a study on normal & drilled disc brake rotor of normal passenger vehicle with full load of capacity. The study is more likely concern of heat and temperature distribution on disc brake rotor .The motive of undertaking this project of "Thermal Analysis of Disc Brake" is to study and evaluate the performance under severe braking conditions and there by assist proper design of disc rotor out of thermal analysis. Transient state response has been conducted through the heat transfer analysis where to predict the worse case scenario and temperature behaviors of disc brake rotor. In this study, finite element analysis approached has been conducted in order to identify the temperature distributions and behaviors of disc brake rotor in transient responses. Ansys has been used as finite elements software to perform the thermal analysis on responses. Thus, results provide better understanding on the thermal characteristic of disc brake rotor and assist the automotive industry in developing optimum and effective disc brake rotor.**

Keywords: Disc Brake, Ansys, Temperature, Solid works

I.INTRODUCTION

Brakes are most important safety parts in the vehicles. Generally all of the vehicles have their own safety devices. Brakes function to slow and stop the rotation of the wheel. To stop the wheel, braking pads are forced mechanically against the rotor on both surfaces. They are compulsory for all of the modern vehicles and the safe operation of vehicles. In short, brakes transform the kinetic energy of the vehicle into heat energy, thus slowing its speed.

The braking system must be decelerate a vehicle in a controlled and repeatable fashion and when appropriate cause the vehicle to stop. The braking should permit the vehicle to maintain a constant speed when traveling downhill. The braking system must hold the vehicle stationary when on the flat or on a gradient.

Nowadays, there are lot of software has been developed in order to cater the modeling and the finite element analysis on the vehicle component such as (Automatic Dynamic of Mechanical Systems), SOLIDWORKS, ANSYS. There is an advantage of using that powerful computational analysis software where by using those would make it easier, less cost better accuracy and less computing time. Most of the software is used in the wide range of industries such as automotive, oil and gas, aerospace, marine, heavy duty engineering, construction, electro-mechanical and general mechanical industries. In this project, design package SOLIDWORKS and finite element package will be used to generate model and run analysis on the chosen component.

II.STATEMENT OF PROBLEM

Beside overall automotive parts, like engines, there are more crucial parts that engineers need to look into consideration. Suspension, brake, electrical, hydraulic and gear are all the crucial systems in the automotive areas. Each of the parts has their own functionality which brings life to the automation industries. Brakes are the crucial system in stopping the vehicle on all moving stages including braking during high speed, sharp cornering, traffic jam and downhill. All of those braking moments give a different value of temperature distribution and thermal stress. Good performance of disc brake rotor comes from good material with better mechanical and thermal properties. Good designs of disc brake rotor are varying across the range of the vehicles. There are different design and performance of disc brake rotor if compared between passenger, commercial and heavy duty vehicle. There are also other constraints such as cost, weight, manufacturing capability, robustness and reliability, packaging, maintenance and servicing.

This paper concerns of the temperature distribution and constraint of the disc brake rotor. Most of the vehicles today have disc brake rotors that are made of grey cast iron Grey cast iron is chosen for its relatively high thermal conductivity, high thermal diffusivity and low cost . In this paper, we will investigate on the thermal issues of normal two wheeler motor vehicle disc brake rotor, where the investigation is to determine the temperature behavior of the disc brake rotor due to severe braking of the disc brake rotor by using Finite Element Analysis (FEA).

Braking performance of a vehicle can be significantly affected by the temperature rise in the brake components. High temperature during braking will caused to Brake fade, premature wear, Brake fluid vaporization, Bearing failure, Thermal cracks, Thermally-excited vibration. Therefore, it is important to study and predict the temperature rise of a given

Load Bearing Simulation Studies of Various Honeycomb Structures for Use as Impact Barriers in Automobiles

VS Ramesh Reddy
Assistant Professor, Mechanical Engineering Department
MethodistCollegeof Engg.&Tech.
Hyderabad, India
Vsrreddy.mtech@gmail.com

M.prasad
AsstProfessor, Mechanical Engineering Department
Methodist College of Engg.&Tech.
Hyderabad, India
prasadmatam@gmail.com

Y Madhu Maheswara Reddy
Asst. Professor, Mechanical Engineering
MethodistCollege Of Engg.&Tech.
Hyderabad, India
mmr315@gmail.com

***Abstract*: The present work studied the efficiency of the square shaped honey comb structures under different minute variation of the cells aspect ratios, rib thickness for these different materials Alluminium1060 alloy, E-GLASS & S2-GLASS in with standing the loads that could arise in impact in impact of automobiles collisions. The width of the cells was studied in 3 different variations namely**

a) **Equal width in 'x' and 'z' direction**
b) **Width in 'x' direction >'z' direction**
c) **Width in 'z' direction >'x' direction**

The thickness of the ribs was studied under 2 different conditions

a) **Thickness of ribs in 'z' direction > that of in 'x' direction**
b) **Thickness of ribs in 'x' direction >that of in 'z' direction**

All the various conditions in the geometry of square cell honey comb structure are carried out under the condition of contact volume & weights–of–the–structure, thus making that impact resistance comparison relevant. The rib thickness various are adjusted subjected to this important constraints of constant weight of the material in all the impact barriers thus making the comparison of different designs meaningful as it is independent of weight or mass density for a given material.

1.INTRODUCTION

This project gives better shape for textile composite impact barriers by analyzing results using FEM based software COMSOL for impact analysis on

honey comb box type and triangular and hexagonal models, Solid Works software to model 3D models of honeycomb structures. This is going to help in finding out a alternative geometric shape which can be used as a replacement to the traditional hexagonal honeycomb structure and which can help in reducing the delimitation problem of honeycomb structure.

1. Selection of different geometric structures for better inner cores
2. Selection of different materials (composite fibers).
3. Use of solid Works to prepare 3D models.
4. Use of COSMOS to perform analysis.
5. Comparison of results of different geometric structures with traditional hexagonal honeycomb structure.
6. To provide a best suitable alternative for traditional hexagonal honeycomb structure.

2. MATERIALS

Composite Material For the specific carbon and glass fiber based composite materials often referred to loosely as 'composites 'Composites are formed by combining materials together to form an overall structure that is better than the individual components.

Composite materials (also called composition materials or shortened to composites) are materials made from two or more constituent materials with significantly different physical or chemical properties that when combined, produce a material with characteristics different from the individual components. The individual components remain separate and distinct within the finished structure. The new material may be preferred for many reasons: common examples include materials which are stronger, lighter or less expensive when compared to traditional materials. Typical engineered composite materials include:

3. HONEYCOMB STRUCTURES

Honeycomb structures are natural or man-made structures that have the geometry of a honeycomb to allow the minimization of the amount of used material to reach minimal weight and minimal material cost. The geometry of honeycomb structures can vary widely but the common feature of all such structures is an array of hollow cells formed between thin vertical walls. The cells are often columnar and hexagonal in shape. A honeycomb shaped structure provides a material with

Paper ID: ME201805

Economizing ofMachining Process by Heating the Work Piece in the Furnace to Increase ToolLife, Surface Finish and to Reduce the Power Consumption

First Author: Prabhakar K., Assistant Professor in Mechanical Engineering Dept.
Co- Authors:Dr. A. Rajasekhar, Professor and HOD, Mechanical Engg Dept.,
Dr.P.Shailesh, Professor in Mechanical Engineering Dept.,
Y. Madhu Maheswara Reddy Asst, Professor, Anil KumarK. Mechanical Engineering Dept.,Methodist College of Engineering andTechnology,Abids,KingKoti,Hyderabad, India

Abstract: This paper focuses on economizing of machining by increasing the temperature of the workpiece to above the room temperature and below the recrystallization temperature depending on thematerials to be machined and their compositions. The heating is achieved by using an electric furnacewhich is a safe practice compared to gas torch heating. The process parameters are calibrated by thelathe tool dynamometer to measure the cutting forces on X,Y and Z directions. Temperature measurementby Non contact type infrared thermometer .Surface roughness values are measured by –Surfaceroughness measuring equipment. thus the cost of machining is reduced and that/s how the Economizing inMachining is possible.

Key words: Furnace heating, Cutting Forces, Surface Roughness Tool life.

ISSN: 2395-5295 (Online), 2395-5287 (Print)
Journal: Grenze International Journal of Engineering and Technology (GIJET)

Paper ID: ME201807

Design and Thermal Analysis of Tractor Boot

Dr. P. Ravi Chander[1], Y. Madhu Maheswara Reddy[2], R. Manish[3], G. SaiRam[4]
[1-2] *Methodist College of Engg. &Tech./Mech. Engg. Department, Hyderabad, India*
Email: 1vira755@gmail.com, 2mm315@gmail.com
[3-4] *Student, Methodist College of Engg. &Tech./Mech. Engg. Department, Hyderabad, India*

Abstract— In today's modern world, Farmers use advanced farm equipment for the better yield and good quality products. In order to meet the increasing demandof Tractors in the Industry, the Production time has to be reduced satisfying the quality issues at the same time. The need for good quality products and 'Zero defect operations' are pivotal for any industry. One such defect to be eliminated is the damage caused to the rubber boot of tie rod in Dhruv tractors. Tie rod is a vital component in the steering system of any automobile. It helps in steering and handling of a vehicle. The problem arose is due to high temperature in the paint shop causing it to transmit the heat to the Polyurethane material. In order to cease the damage, various experiments were conducted which are as follows:(a) Masking the entire surface of rubber boot, (b) Covered the steel ring with an 'O' ring, (c) Used a thick rubber as a covering. The specimens were examined and observed that the rubber boot lacked thickness around the steel ring which is not able to sustain such high temperatures. Hence, the tie rod rubber boot is redesigned in NX-CAD with varying thickness and analysed the thermal stress and optimized the thickness of the rubber boot.

Index Terms—Farmers, Tractor, Tie rod, Steering, Rubber Boot and NX-CAD.

ISSN: 2395-5295 (Online), 2395-5287 (Print)
Journal: Grenze International Journal of Engineering and Technology (GIJET)

Paper ID: ME201809

Design Modelling andAnalysis of Light WeightFoldable cycle

Prasad .Matam[1], Y.Madhu M. Reddy[2]
[1,2] *Methodist college ofEngineering andTechnology/Mechanical Engineering Department, Hyderabad, India*
Email: prasadmatam@gmail.com, mmr315@gmail.com

Abstract— Bicycle is seen as a transportation course of action upgrading distinctive normal, monetary and social pointsof view. A combination of bicycle plan and setups for utility are used to pass onindividual resources, fundamentalsupplies, adolescents and essentially more. One explicit utility bicycle is the collapsing bicycle. Its arrangementempowers customers to easily transport the bicycle using less space when the bicycle is quot;crumbled" in to an insignificantsize. While using a collapsing bicycle with a bicycle travel, it empowers individuals ability to board travel vehicles. Theflexibility of a falling bicycle is furthermore appropriate for air travel and for when lacking halting and bicycle theft is animportant concern.In this paper, I am will do plan and examination of a foldable bike with reference to the hummingbirdcycle which is the lightest bicycle on the planet.In this undertaking principle expect to make structure of cycle which isnot exactly the hummingbird utilizing distinctive materials for the suspension and innovation to make it less weight.

Index Terms— Humming Bird Cycle , Solid Works,Ansys ,Different Materials

ISSN: 2395-5295 (Online), 2395-5287 (Print)
Journal: Grenze International Journal of Engineering and Technology (GIJET)

Design, Simulation and Analysis Of Automatic Multifastener Device

R Moses Krupavaram
Assistant Professor, Mechanical Engineering
MCET
Hyderabad, India
moseskrupavaram@gmail.com

Y M.Maheswara Reddy
Assistant Professor, Mechanical Engineering
MCET
Hyderabad, India
mmr315@gmail.com

M Amresh Reddy
Assistant Professor, Mechanical Engineering
MCET
Hyderabad, India
mudiam.amresh@gmail.com

Abstract— **This paper deals with the design, simulation and analysis of automatic multifastener device. This device is designed by considering M22 standard metric bolt for a four wheeler tire assembly system. Assembly technology is an essential component of modern industrial production, and threaded joints are among the most common and widely used types of fasteners. Current industry demands the tools that offer One hand operation, Extended service life, Operator Comfort, Ease of serviceability to minimize the downtime, High value verses low cast. These demands can be met by the use of automatic devices. Now days the use of these type of automatic devices has gained an important role in the automobile industries, Laboratories and also in all mechanical industries Main components used in this device are motor, base stand, nut fitting and removing plate and a keypad for on/ off switches and some mechanical parts are used for driving operation. This device is operated by a DC motor. Selection of DC. Here Spur gears are used for transmitting the power. Proposed design and mechanism has been done in Pro-e wild fire.Structrual analysis of the device is carried out in Ansys at different loads.**

Keywords- fastener; Device, bolt tightner, Gear assembly, DTDP Switch, Battery, structural analysis.

I. INTRODUCTION

Assembly technology is an essential component of modern industrial production, and threaded joints are among the most common and widely used types of fasteners. The broad spectrum of assembly equipment ranges from state of the art robotics to work stations using hand-held tools. The criteria of these assembly tasks are just as varied, depending upon the application needs, the production method, quantity and the accuracy requirements.

Conventional fastening requires tools like a wrench or an electric or pneumatic tool. Industrial assembly or joining technology increasingly requires meeting long-term safety-oriented, and function oriented solutions. Historically there have been disadvantages associated with the use of conventional tools such as inaccurate tightening, high tool wear and requires more time for fastening.

Current industry demands the tools that offer One hand operation, Extended service life, Operator Comfort, Ease of serviceability to minimize the downtime, High value verses low cost. These demands can be met by the use of automatic devices. Now days the use of automatic devices has gained an important role in the automobile industries, Laboratories and also in all mechanical industries Bolted assemblies are the most commonly used connecting systems in mechanics. And although they appear to be quite simple, bolted assemblies do pose several challenges at many levels: design department, assembly workshop, on-site, and maintenance. In an assembly that contains threaded fasteners, the nut or bolt needs to be physically tightened to a specific torque. A bolted assembly quite simply means the putting together of at least two parts using one or several bolts. The design and implementation of a bolted assembly requires a very strict methodological approach, for errors can lead to costly and often disastrous failures. Several studies have shown that incidents encountered on bolted assemblies are most often due to improper design of the assembly or poor implementation like tightening method, tooling, and inspection.

Today it is known that of all the various causes of failure like overloading, design flaws, manufacturing defects, and others the most frequent is improper assembly. Tightening problems, whether insufficient tightening, excessive tightening or heterogeneous tightening, alone account for over 30% of all bolted assembly failures. More specifically, 45% of all fatigue failures are estimated to be due to improper assembly. Therefore, the importance of the design of the bolted assembly and the means used to tighten it are of utmost importance.

V.K. Zamyatin manufactured a device ie-3115a and ie 3112 electrical manual impact (chum) nut tightner for the assembly and dismantling of threaded connections at industrial firms. They used IE-3115A tightener for the calibrating the tightening of bolts size M18-M30, strength class 3.6-6.6, and bolts size M12-M20, strength class 6.8- 14.9. It consists of an electric motor, a planetary reducing gear, an impact mechanism, a handle with switch, the current-supply cable, and they found that this deviceis used for the elimination of radio interferences. They used IE-3112 nut tightener for the calibration of tightening of high-strength bolt of different sizes (M22-M30) or medium-strength bolts (35 steel) with diameters up to M42 ram. This tightener is a reversible tool and is equipped with a protective switch-off device. The IE-

ISBN No.: 978-1-4799-4949-6 @ 2014 IEEE