IJNRD.ORG ISSN : 2456-4184
INTERNATIONAL JOURNAL OF NOVEL RESEARCH AND DEVELOPMENT (IJNRD) | IJNRD.ORG
An International Open Access, Peer-reviewed, Refereed Journal

OPTIMIZATION OF BRAKE DISC USING DIFFERENT MATERIALS AND DIFFERENT FLANGE THICKNESS

S. Dhanasekar[*1], J.Srinivas[2], A. Abi Yesuva[3], J. Suja[4]

[*1,2,]Assistant Professor, Department of Mechanical Engineering, Panimalar Engineering College, Chennai, Tamilnadu, India
[3]UG Student, Department of Mechanical Engineering, Panimalar Engineering College, Chennai, Tamilnadu, India
[4]Assistant Professor, Department of Mathematics, Panimalar Engineering College, Chennai, Tamilnadu, India
dhanasekarrlmech@gmail.com[*1],
jayaramansrinivas@gmail.com[2],
Joshuvahadzz@gmail.com[3],
Jesinsuja@gmail.com[4]

Abstract

Disc brakes are components of a vehicle that serve to slow or stop the rotation of the wheel. The main objective of the project is to study and evaluate the performance under severe braking conditions and there by assist in disc rotor design and analysis. ANSYS Workbench is a dedicated finite element package used for determining the temperature distribution, variation of stresses and deformation across the disc brake profile. In this present work, an attempt will be made to investigate the effect of stiffness, strength and variations in disc brake rotor design on the predicted stress and temperature distributions. By identifying the true design features, the extended service life and long term stability is assured. A transient thermal analysis will be carried out to investigate the temperature variation across the disc. Further structural analysis is also will be carried out by coupling thermal analysis. Also suggest a best combination of material and flange thickness used for disc brake rotor, which yields a low temperature variation across the rotor, less deformation, and minimum vonmises stress possible.

Keywords: Brake disc, flange, ANSYS workbench, Finite element analysis.

1. Introduction

1.1 Brake system

Brakes are the devices which used for slowing or stopping the vehicle quickly within a short period. The braking operation is reverse of the acceleration to the vehicle in acceleration the heat energy of extra fuel is converted in to kinetic energy of the vehicle. Whereas in braking operation the kinetic energy of the vehicle is converted into heat which is dissipated into the atmosphere .The amount of heat produced depends upon the friction between two contact surfaces.

1.2 Braking requirements

- The brakes must be strong enough to stop the vehicle with in a minimum Distance in an emergency.
- The driver must have proper control over the vehicle during braking and the vehicle must not skid.

- The brakes must have good characteristics i.e. their effectiveness should not decrease with constant prolonged application.
- The brakes should have good anti wear properties.

1.2.1 Functions of brake

- To stop or slow down the vehicle in shortest possible distance.
- To control the vehicle when descending a hill.

1.3.Classification of brakes (based on transformation of energy)

- ➢ Hydraulic brakes.
- ➢ Electric brakes.
- ➢ Mechanical brakes.
- ➢ Air Brake.
- ➢ Air Assisted Hydraulic Brake.

The mechanical brakes according to the direction of acting force may be sub divided into the following two groups: 1. Radial b rakes 2. Axial brakes

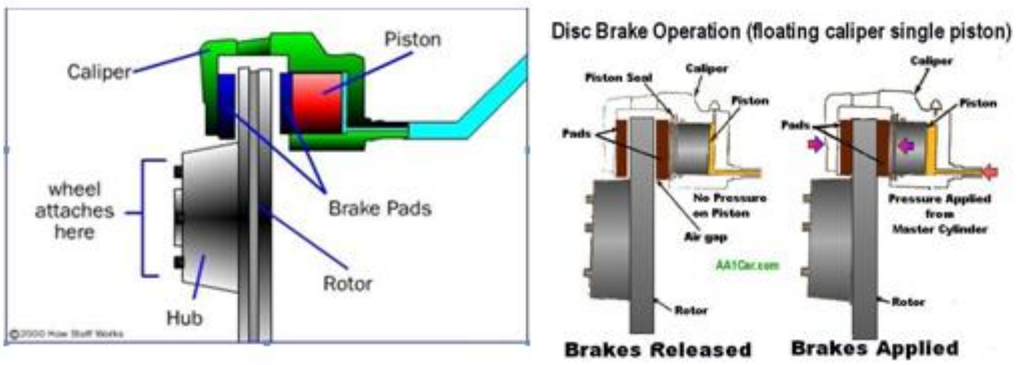


Fig.1 working principle of disc brake

1.3.1 Radial brakes

In these brakes the force acting on the brake drum is in radial direction. The radial brake may be subdivided into external brakes and internal brakes.

1.3.2 Axial brakes

In these brakes the force acting on the brake drum is only in the axial direction.

E.g. Disc brakes, Cone brakes.

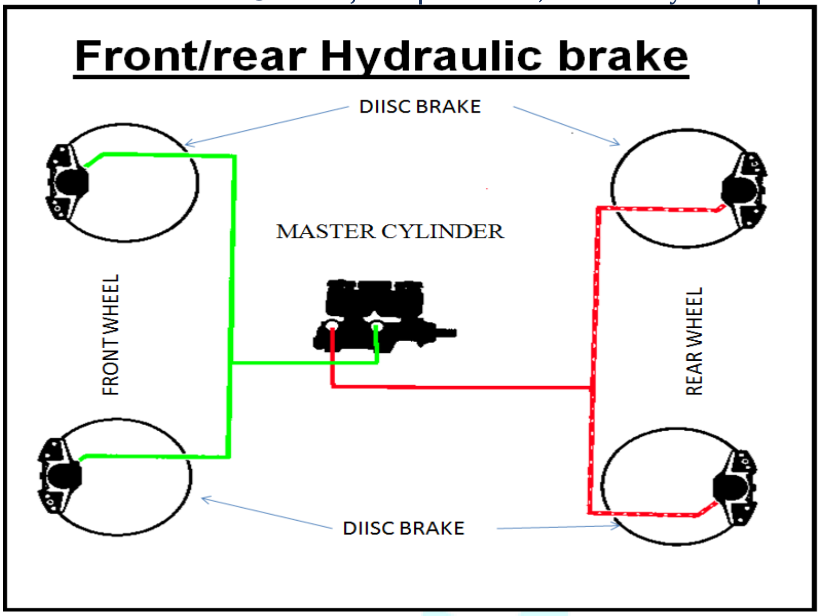


Fig.2 Front and Rear Wheel Hydraulic Brake

1.4 Working Principle of Disc Brake

A disc brake consists of a cast iron disc bolted to the wheel hub and a stationary housing called calliper. The calliper is connected to some stationary part of the vehicle, like the axle casing or the stub axle and is cast in two parts, each part containing a piston. In between each piston and the disc, there is a friction pad held in position by retaining pins, spring plates etc. passages are drilled in the calliper for the fluid to enter or leave each housing. These passages are also connected to another one for bleeding. Each cylinder contains rubber-sealing ring between the cylinder and piston. A schematic diagram is shown in the figure. The principle used is the applied force (pressure) acts on the brake pads, which comes into contact with the moving disc. At this point of time due to friction the relative motion is constrained.

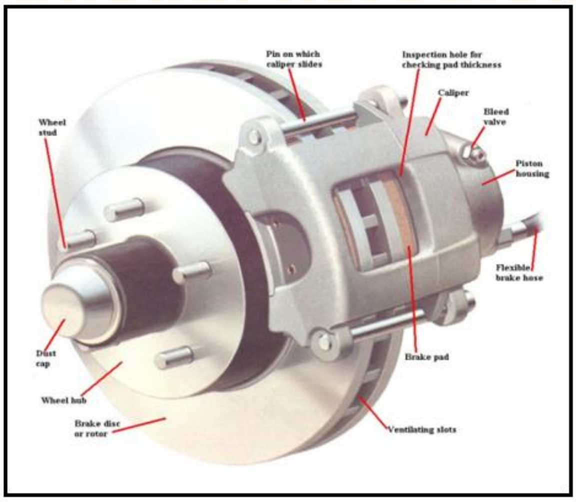


Fig. 3 Vents provided on Disc Brakes

Ventilated disc brakes have a set of vanes, between the two sides of the disc that pumps air through the disc to provide artificial cooling.

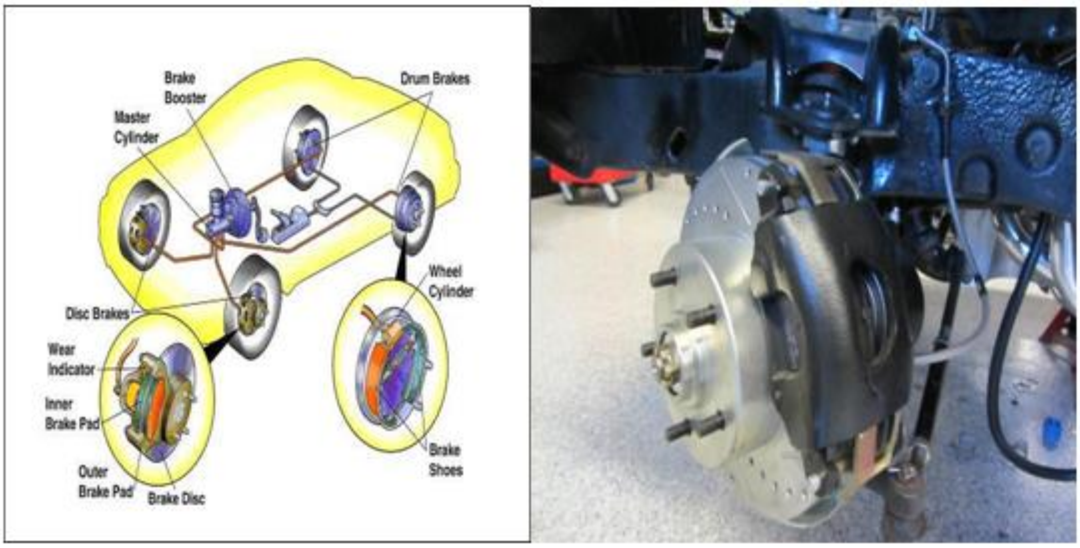


Fig.4 Location of Disc Brake in a car

Chary-Nathi [1] (2004)was carried out the Structural and Thermal analysis .In his project work an Axis-symmetric analysis of disc brake has been carried out using Plane 55 and Plane 42 through ANSYS 5.4 (F.E.A) software. Transient thermal analysis is carried out using the direct time integration technique for the application of braking force due to friction for time duration of 4,5 and 6 seconds. The maximum temperature obtained in the disc is at the contact surface and is observed to be 240.161°C. Static structural analysis is carried out by coupling the thermal solution to the structural analysis and the maximum Von Misses stress is observed to be 518 M Pa. Transient Thermal and Structural Analysis for three different combinations in each of the three different analyses are carried out separately and the results were compared. The different results obtained from the analysis, He concluded that disc brake with 10 mm flange width, 6.5 mm Wall Thickness and the material Cast Iron is the Best possible combination for the present application.

Chogdu Cho and Sooick [2] (2003) Thermal Analysis was carried out by Using carbon ceramic matrix disc brake material calculating normal force, shear force and piston force and also calculating the brake distance of disc brake. The standard disc brake two wheelers model using in Ansys, done the Thermal and Modal Analysis. During the Analysis calculated the deflection, total heat flux, Frequency and temperature of disc brake model. important to understand action force and friction force on the disc brake new material, which use disc brake works more efficiently, which can help to reduce the accident that may happen in each day. Haripa l- Shergil [3] was analyzed the Disc Brake. He used the different materials. The element analysis techniques were predicted the temperature distribution on the brake disc and identified the critical temperature of the brake rotor disc. All three modes of heat transfer (conduction, convection and radiation) have been analyzed. The results obtained from the analysis shows that different material on the same retardation of the car during panic braking shows different temperature distribution. The comparison was made between three different materials used for brake disc and the best material for making brake disc based on the rate of heat dissipation has been suggested.

Choy and Lee,[4] (2004)Finite element analysis of transient thermoelastic behaviors in disk brakes A transient analysis for thermoelastic contact problem of disk brakes with frictional heat generation is performed using the finite element method. To analyze the thermoelastic phenomenon occurring in disk brakes, the coupled heat conduction and elastic equations are solved with contact problems. The numerical simulation for the thermoelastic behaviour of disk brake is obtained in the repeated brake condition. The computational results are presented for the distributions of pressure and temperature on each frictionsurface between the contacting bodies. Eltoukhy and Asfour[5] (2008) present a paper on Braking Process in Automobiles: Investigation of the Thermoelastic Instability Phenomenon. In this chapter a case study regarding a transient analysis of the thermoelastic contactproblem for disk brakes with frictional heat generation, performed using the finite element analysis (FEA) method is described in details. The computational results are presented for the distribution of the temperature on the friction surface between the contacting bodies (the disk and the pad).

Zaid, et al. (2009) [6]The investigation of disc brake rotor by Finite element analysis. He has conducted a study on ventilated disc brake rotor of normal .The study is more likely concern of heat and temperature distribution on 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 response. ABAQUS/CAE has been used as finite elements software to perform the thermal analysis on transient response. In his analysis a better understanding on the thermal characteristic of disc brake rotor and assist the automotive industry in developing an optimum and effective disc brake rotor.

2. Materials and Methods

3.3 CATIA INTRODUCTION

CATIA is one of the world's leading CAD/CAM/CAE package. Being a solid modeling tool, it not only unites 3D parametric features with 2d tools, but also addresses every design- through- manufacturing process. Besides providing an insight into the design content, the package promotes collaboration between companies and provides them with edges over their competitors.

CATIA- Computer Aided Three Dimensional Interactive Application

CATIA, developed by Dassault Systems, France, is a completely re-engineered, next generation family of CAD /CAM/CAE Software solution.

CATIA serves the basic design task by providing different workbenches, some of the workbenches available in this package are

- Part Design workbench
- Assembly Design workbench

- Drafting workbench
- Wire frame and surface Design workbench
- Generative shape design workbench
- DMU kinematics
- Manufacturing
- Mold Design

PART DESIGN WORKBENCH

The Part workbench is a parametric and feature-based environment, in which you can create solid models. In the part Design workbench, you are provided with tool that convert sketches into other features are called the sketch-based features.

ASSEMBLY DESIGN WORKBENCH

The Assembly Design workbench is used to assemble the part by using assembly constrains. There are two types of assembly design,

Bottom-up

Top-down

In the bottom-up assembly, the parts are created in part workbench and assembled in assembly workbench.

In the top-down assembly, the parts are created in assembly workbench itself.

WIREFRAME AND SURFACE DESIGN WORKBENCH

The Wire frame and surface design workbench is also a parametric and feature based environment. The tools available in this workbench are similar to those in the part workbench, with the only difference that the tool in this environment are used to create basic and advance surfaces

Note: While creating surface we are considering that the surface having negligible thickness

3.4 DRAFTING WORKBENCH

The Drafting workbench is used for the documentation of the parts or the assemblies created in the form of drawing.

There are two types of drafting techniques:

Generative drafting

Interactive drafting

The generative drafting technique is used to automatically generate the drawing views of parts and assemblies.

In interactive drafting, you need to create the drawing by interactive with the sketcher to generate the views.

DMU KINEMATICS

This workbench deals with the relative motion of the parts. DMU Kinematics Simulator is an independent CAD product dedicated to simulating assembly motions. It addresses the design review environment of digital mock-ups (DMU) and can handle a wide range of products from consumer goods to very large automotive or aerospace projects as well as plants, ships and heavy machinery.

TERMS AND DEFINITIONS

Some of the important terms and definitions in CATIA are discussed below.

FEATURE-BASED MODELLING

Feature is defined as the smallest building block that can be modified individually. A model created in CATIA is a combination of a number of individual features and each feature is related to the others feature directly or indirectly, and therefore it can be modified at any time during the design process. This provides greater flexibility to the design.

PARAMETRIC MODELLING

The main function of this property is to derive the selected geometry to a new size or shape without considering its original size or shape. You can change or modify the shape and size of any feature at any stage of the design process.

ASSOCIATIVITY

As mentioned earlier, CATIA has different workbenches such as part Design workbench, Assembly Design workbench, and Drawing workbench. There exists bidirectional associativity between all these workbenches. This associativity ensures that if any modification is made in the model in any one of the workbenches of CATIA, it is automatically reflected in the other workbenches immediately.

CATPart

CATPart is a file extension associated with all the files that are created in sketcher, Part Design, and Wireframe and Surface Design workbenches of CATIA V5.

CATProduct

CATProduct is a file extension associated with all the files that are created in Assembly Design workbench of CATIA V5.

CATDrawing

CATDrawing is a file extension associated with all the files that are created in Drafting workbench of CATIA V5. The CAD model of the IC Engine with the piston was created in a commercial 3D CAD Software called CATIA. A Parametric model of the IC Engine was created from basic 2D curves and converting them into 3D solids using many options in CAD like Extrude and Revolve and Pattern. Stress concentration location was avoided by creating a blend in the model at all sharp edges to avoid singular stresses. The engine specifications were used to create the overall dimensions of the CAD model with the accurate bore dimensions. The model was so created in Parametric so that the model can be changed or modified according to the change in specification at a later stage, if anything required. A symmetric model of the IC Engine is used with symmetric Boundary Conditions since the model is completely symmetric about the planar centre as shown in below Fig. This project is mainly to model a brake disc according to the existing dimensions. In this project a brake disc is designed using the modelling software Pro/E and then the model is exported in the analysis software ANSYS by using STEP or Parasolid translation method.

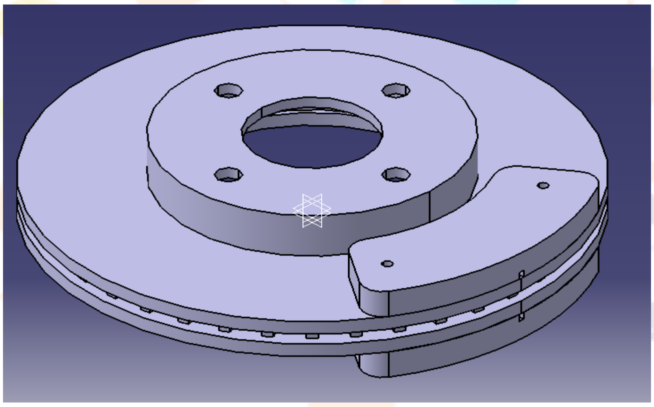


Fig. 5 CATIA Model with Flange thickness 7.5 mm

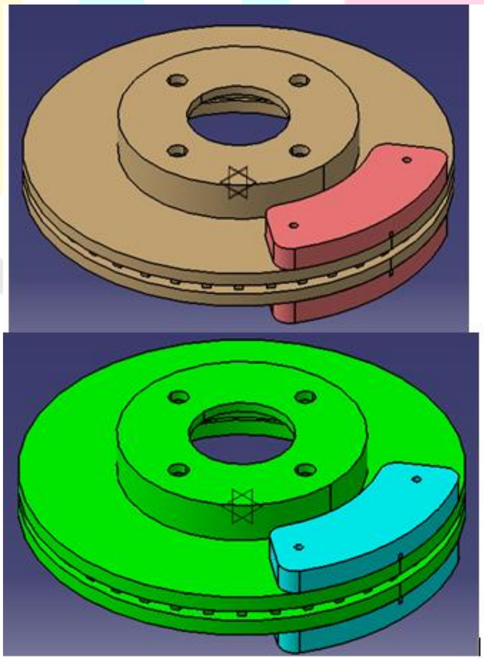


Fig.6 CATIA Model with Flange thickness of 8.5 mm and 10 mm

When the brakes are applied, hydraulically actuated pistons move the friction pads in to contact with the disc, applying equal and opposite forces on the later. On releasing the brakes the rubber-sealing ring acts as return spring and retracts the pistons and the friction pads away from the disc.

The main components of the disc brake are:

- The Brake pads
- The calliper, which contains the piston
- The Rotor, which is mounted to the hub

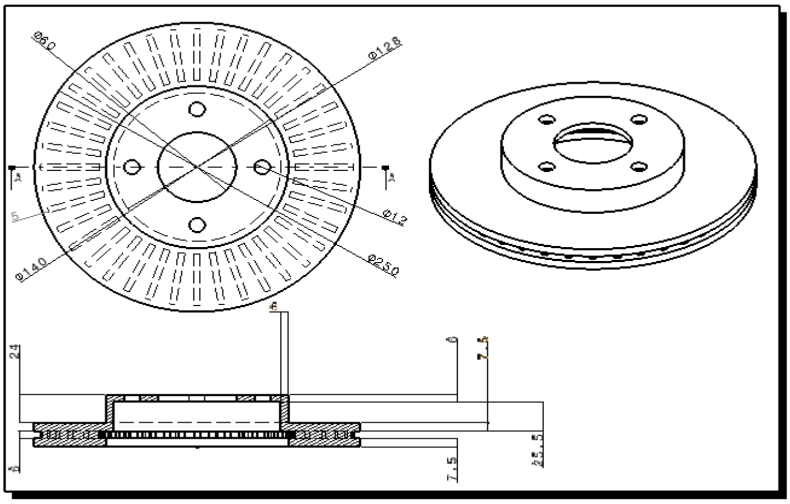


Fig. 7 Flange Drawing and Detailing

The finite element method provides a numerical solution to a complex problem. It may therefore be expected that the solution must converge to the exact formulation of the structure. Hence as the mesh is made finer the solution should converge to the correct result and this would be achieved if the following three conditions were satisfied by the assumed displacement function.

- The displacement function must be continuous within the element. Choosing polynomials for the displacement model can easily satisfy this condition.

- The displacement function must be capable of representing rigid body displacement of the element. This is when the nodes are given such displacement corresponding to a rigid body motion; the element should not experience and hence leads to zero nodal forces. The constant terms in the polynomials used for displacement models would usually ensure this condition. The displacement function must be capable of representing constant strain states within the element. The reason for the requirement can be understood if we imagine the condition when the body or structure is divided in to smaller and smaller elements. As these elements approach infinitedecimalsize the strain in each element also approach constant strain states. For one, two and three-dimensional elasticity problems the linear terms present in the polynomials satisfy the requirement.

3.8 Advantages of fem

The properties of each element are evaluated separately, so an obvious advantage is that we can incorporate different material properties for each element. Thus almost any degree of non-homogeneity can be included. FEM is based on the concept of description. Nevertheless as either the variations or residual approach, the technology recognizes the multidimensional continuous but also requires no separate interpolation process to extend the approximate solution to every point with the continuum. One of the important advantages of FEM is that it makes use of boundary conditions in the form of assembled equations. This is relatively an easy process and requires no special technology. Rather than requiring every trial solution to satisfy boundary conditions, one prescribes the conditions after obtaining the algebraic equations for individual's finite elements.

3.8.1. Limitations in fem

FEM reached high level of development as solution technology; however the method yields realistic results only if coefficient or material parameters that describe basic phenomena are available. The most tedious aspects of use of FEM are basic process of sub-dividing the continuum of generating error free input data for computer.

3. Results and discussion

FEA SOFTWARE – ANSYS

4.1 Introduction to ANSYS Program

Dr. John Swanson founded ANSYS. Inc in 1970 with a vision to commercialize the concept of computer simulated engineering, establishing himself as one of the pioneers of Finite Element Analysis (FEA). ANSYS inc. supports the ongoing development of innovative technology and delivers flexible, enterprise wide engineering systems that enable companies to solve the full range of analysis problem, maximizing their existing investments in software and hardware. ANSYS Inc. continues its role as a technical innovator. It also supports a process -centric approach to design and manufacturing, allowing the users to avoid expensive and time-consuming "built and break" cycles.

ANSYS analysis and simulation tools give customers ease -of-use, data compatibility, multi-platform support and coupled field multi-physics capabilities.

4.2 Evolution of ANSYS Program

ANSYS has evolved into multipurpose design analysis software program, recognized around the world for its many capabilities. Today the program is extremely powerful and easy to use. Each release hosts new and enhanced capabilities that make the program more flexible, more usable and faster. In this way ANSYS helps engineers meet the pressures and demands modern product development environment

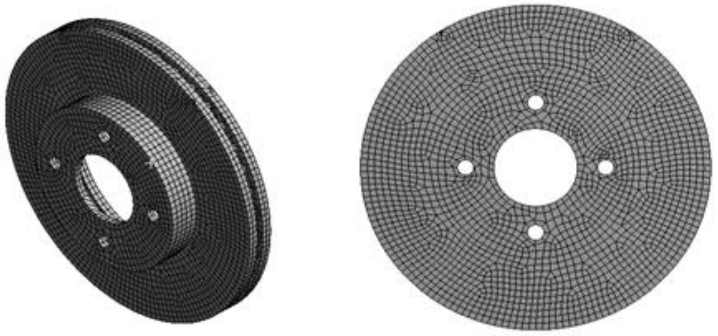
6.1 Introduction of Thermal Analysis

A Thermal analysis calculates the temperature distribution and related thermal quantities in a system or component. Typical thermal quantities are:

The temperature distributions

The amount of heat lost or gained

Thermal fluxes



Statistics	
Nodes	66997
Elements	19954
Mesh Metric	None

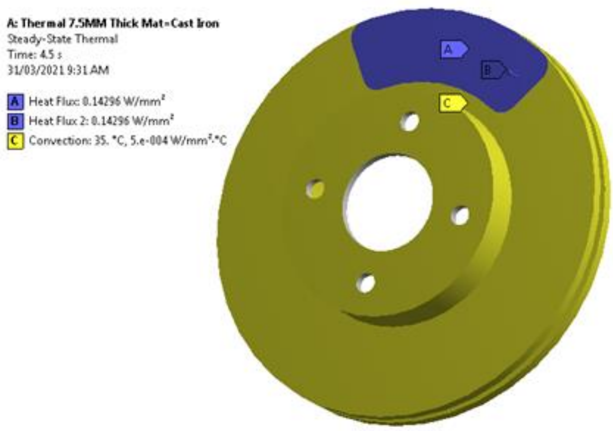


Fig. 8Heat convection& Convection applied Areas

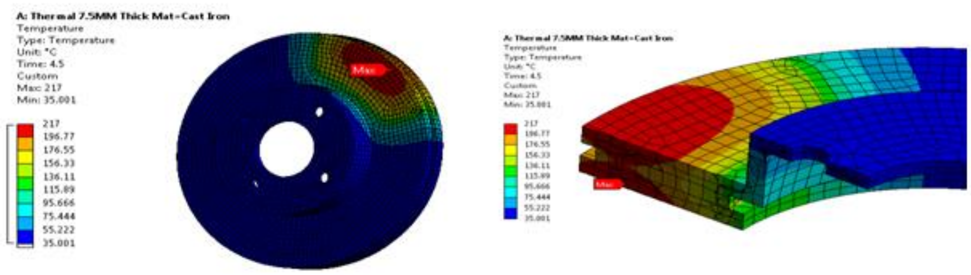
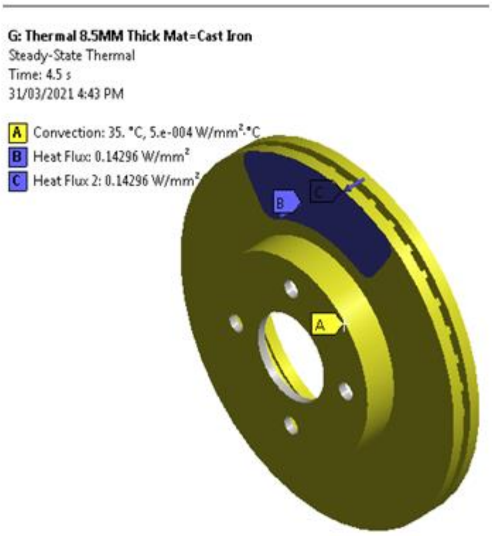
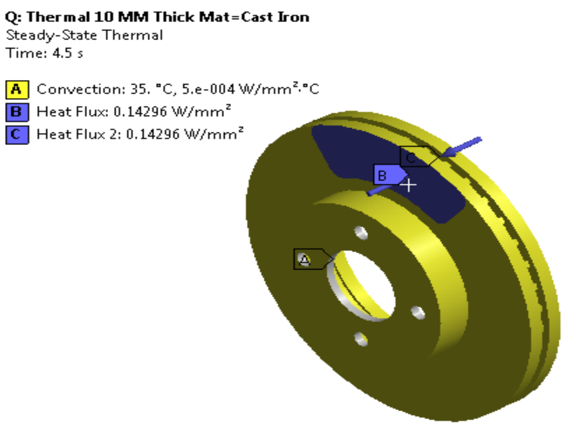
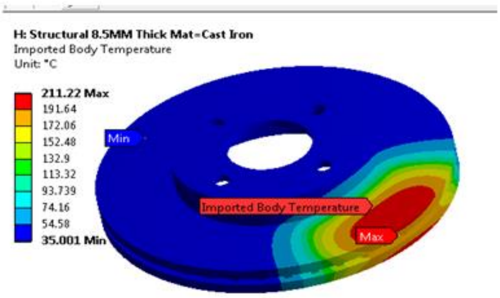
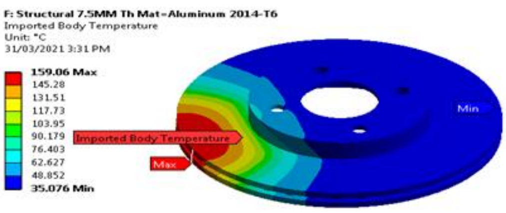


Fig. 9 Temperature distribution on ventilated type Disk brake on the front side

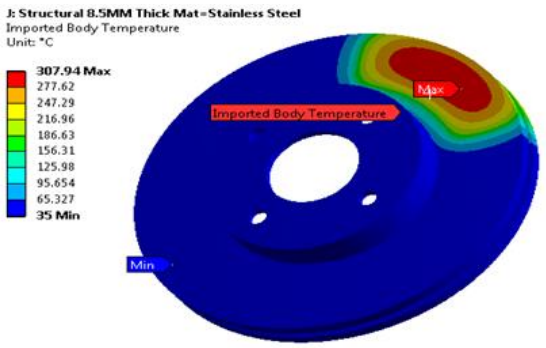

G: Thermal 8.5MM Thick Mat=Cast Iron
Steady-State Thermal
Time: 4.5 s
31/03/2021 4:43 PM
A Convection: 35. °C, 5.e-004 W/mm²·°C
B Heat Flux: 0.14296 W/mm²
C Heat Flux 2: 0.14296 W/mm²
A
B
C

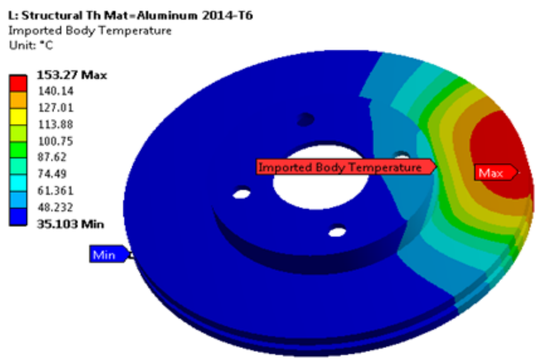

Q: Thermal 10 MM Thick Mat=Cast Iron
Steady-State Thermal
Time: 4.5 s
A Convection: 35. °C, 5.e-004 W/mm²·°C
B Heat Flux: 0.14296 W/mm²
C Heat Flux 2: 0.14296 W/mm²
A
B
C


H: Structural 8.5MM Thick Mat=Cast Iron
Imported Body Temperature
Unit: °C
211.22 Max
191.64
172.06
152.48
132.9
113.32
93.739
74.16
54.58
35.001 Min
Min
Imported Body Temperature
Max


F: Structural 7.5MM Th Mat=Aluminum 2014-T6
Imported Body Temperature
Unit: °C
31/03/2021 3:31 PM
159.06 Max
145.28
131.51
117.73
103.95
90.179
76.403
62.627
48.852
35.076 Min
Min
Imported Body Temperature
Max

4. Conclusion





5. Conclusion

The present study can provide a useful design tool and improve the brake performance of disk brake system. From the above Table we can say that all the values obtained from the analysis are less than their allowable values. Hence the brake disk design is safe based on the strength and rigidity criteria. Comparing the different results obtained from analysis. It is concluded that the Aluminum T6, Cast Iron materials of ventilated type disk brake with flange thickness of 10mm and 8.5 mm are best possible for the present application.Temperature and Stress, Displacement values for both 8.5 mm and 10 mm thickness are nearly equal. Comparing the different results obtained from the analysis, it is concluded that disc brake with **8.5 mm flange width**, and of material **Aluminum T6, Cast Iron** is the Best possible combination for the present application.

References

[1] International Journal of Innovative Research in Science, Engineering and Technology *(An ISO 3297: 2007 Certified Organization)*Vol. 2, Issue 12, December 2013.
[2] VollerG.P. , M Tirovic, R Morris and P Gibbens, "Analysisof automotive disc brake cooling characteristics", Proceedingsof the Institution of Mechanical Engineers, Part D: *Journalof Automobile Engineering* V **217**: pp657 (2003).
[3] Choi Ji-Hoon and In Lee "Finite element analysis oftransientthermoelasticbehaviours in disk brakes" ,Wear, V**257**, pp47–58 (2004)
[4] H.S. Qi and A.J. Day, "Investigation of disc/pad interfacetemperatures in friction braking", Wear, V **262**, Pages 505-513 (2007)

[5] Introduction to Finite Elements in Engineering, Tirupathi, R.Chandrupatla Ashok D.Belegundu.
[6] User Gu ide for ANSYS version 5.4.,11.0.
[7] Krishna Moorty ,CS,Finite Element Analysis--- Theory and Programming, Tata McGraw-Hill, New Delhi, 1987.
[8] Finite Element Analysis using ANSYS 11.0 --- Paleti Srinivas*Krishna Chaitanya Sambana * Rajesh Kumar Datti.
[9] International Journal of Emerging Technology and Advanced Engineering. Website www.ijetae.com(ISSN 2250-2459,ISO 9001:2008 Certified Journal, volume 3 Issue 9,Sebtember 2013)