International Journal of Advanced Engineering & Innovative Technology (IJAEIT)
ISSN: 2348 7208
IMPACT FACTOR: 1.04

Design and Analysis of Disc Brake Rotor

Deovrut D. Jadhav, Shubham Ramchandra Mali, Mayuresh Mahendra Sapre,

Himalay Jayant Vidhate, Aniket Dilip Patil

ABSTRACT: Braking is a process which converts a vehicle's kinetic energy into mechanical energy which must be dissipated in the form of heat. During the braking phase, the frictional heat generated at the interface of the disc and pads can lead to high temperatures. The frictional heat generated on the rotor surface can influence excessive temperature rise which, in turn, leads to undesirable effects. In this project, solid type disc brake rotor of a vehicle, taken an investigation into the usage of various materials will be done so as to improve the braking efficiency and provide greater stability to the vehicle. Modelling of the disc brake rotor is done using CATIA V5R19, which facilitates collaborative engineering across various disciplines. The thermal and structural analysis of disc brake rotor is done using ANSYS 19.2, for determining the temperature distribution, variation of the stresses and deformation across the disc brake profile. A comparison is made between three different materials used for solid type disc brakes and the best material for making disc brake and type of disc brake have been suggested based on the magnitude of Von mises stresses, temperature distribution and deformation.

Keywords: Solid disk brake rotor, Fem analysis, Ansys, Catia/creo.

1. INTRODUCTION:

The brake disc (or rotor) is the rotating part of a wheel's disc brake assembly, against which the brake pads are applied. The material is typically gray iron, a form of cast iron. The design of the discs varies somewhat. Some are simply solid, but others are hollowed out with fins or vanes joining together the disc's two contact surfaces (usually included as part of a casting process). The weight and power of the vehicle determines the need for ventilated discs. The "ventilated" disc design helps to dissipate the generated heat and is commonly used on the more-heavily loaded front discs.

Discs for motorcycles, bicycles, and many cars often have holes or slots cut through the disc. This is done for better heat dissipation, to aid surface-water dispersal, to reduce noise, to reduce mass, or for marketing cosmetics.

****

Prof. Deovrut D Jadhav, Shubham R Mali, Mayuresh M Sapre, Himalay J Vidhate, Aniket D Patil ,

d3jadhav@gmail.com; Vishwaniketan's Institute of Management Entrepreneurship and Engineering Technology

(This paper is presented in National Conference ETAT-2019 held at ViMEET, Khalapur)

Slotted discs have shallow channels machined into the disc to aid in removing dust and gas. Slotting is the preferred method in most racing environments to remove gas and water and to deglaze brake pads. Some discs are both drilled and slotted. Slotted discs are generally not used on standard vehicles because they quickly wear down brake pads; however, this removal of material is beneficial to race vehicles since it keeps the pads soft and avoids vitrification of their surfaces. On the road, drilled or slotted discs still have a positive effect in wet conditions because the holes or slots prevent a film of water building up between the disc and the pads

2. LITERATURE REVIEW:

1. Bouchetara Mostefa, Belhocine Ali (2014) Presented paper on Thermo elastic Analysis of Disk Brakes Rotor [5]. In this Paper the main purpose of this study is to analyze the thermo-mechanical behaviour of the dry contact between the brake disk and pads during the braking phase. The simulation strategy is based on computer code ANSYS11. The modeling of transient temperature in the disk is actually used to identify the factor of geometric design of the disk to install the ventilation system in vehicles The thermal-structural analysis is then used with coupling to determine the

deformation and the Von-Mises stress established in the disk, the contact pressure distribution in pads. The results are satisfactory when compared to those of the specialized literature.

2. H. Zaid (2009) presented a paper on an investigation of disc brake rotor by Finite element analysis [8]. In this paper, the author has conducted a study on ventilated 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. In this study, finite element analysis approached has been conducted in order to identify the temperature distributions and behaviours of disc brake rotor in transient response. Modeling is done in CATIA & ABAQUS/CAE has been used as finite elements software to perform the thermal analysis on transient response. Material used is Grey cast iron, with maximum permissible temperature 550 C. For load analysis 10 cycles of breaking and 10 cycles without breaking (idle) operation is considered total of 350 seconds. Result provided during 1st, 5th and during 10th cycle. Thus, this sure study provide better understanding on the thermal characteristic of disc brake rotor and assist the automotive industry in developing optimum and effective disc brake rotor.

.

3. OBJETIVES:

- Design and heat calculation
- Thermal analysis to calculate temperature of disc
- Couple field thermo mechanical analysis of disc brake to find strength with temperature loading.
- Aluminium Alloy, Grey Cast Iron & Stainless steel will be considered for material study & best among all will be proposed.

4. METHODOLOGY:

The modelling of the disc brake is done by using pro-e/Catia-R19 and the analysis is performed by Ansys. The project consists of two types of analysis structural and thermal. Structural analysis is done to find the strength of the model and the thermal analysis is done to check the thermal resistance of the model. Here we will be modelling solid types of disc brakes for the study. Analysis on the both models by changing the materials, for this we take three different materials and done analysis on the respective model and the results were compared.

5. FE ANALYSIS OF CAR DISC:

Tool Used: Here Ansys 19.2 is used for analyzing natural frequencies and static deflection. CAD modeling is done in Catia software and .stp file were imported in ANSYs for analysis.

Thermal Analysis:

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

- The temperature distributions
- The amount of heat lost or gained
- Thermal gradients
- Thermal fluxes.

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components. In many cases, engineers follow a thermal analysis with a stress analysis to calculate thermal stresses (that is, stresses caused by thermal expansions or contractions).

The following thermal analysis topics are available:

- Types of Thermal Analysis
- Coupled-Field Analyses

Types of Thermal Analysis:

ANSYS supports two types of thermal analysis:

1. *A steady-state thermal analysis* determines the temperature distribution and other thermal quantities under steady-state loading conditions. A steady-state loading condition is a situation where heat storage effects varying over a period of time can be ignored.
2. *A transient thermal analysis* determines the temperature distribution and other thermal quantities under conditions that vary over a period of time.

Coupled-Field Analyses

A coupled-field analysis is a combination of analyses from different engineering disciplines (physics fields) that interact to solve a global engineering problem, hence, we often refer to a coupled-field analysis as a multiphysics analysis. When the input of one field analysis depends on the results from another analysis, the analyses are coupled.

Some analyses can have one-way coupling. For example, in a thermal stress problem, the temperature field introduces thermal strains in the structural field, but the structural strains generally do not affect the temperature distribution. Thus, there is no need to iterate between the two field solutions. More complicated cases involve two-way coupling. A piezoelectric analysis, for example, handles the interaction between the structural and electric fields: it solves for the voltage distribution due to applied displacements, or vice versa. In a fluid-structure interaction problem, the fluid pressure causes the structure to deform, which in turn causes the fluid solution to change. This problem requires iterations between the two physics fields for convergence.

The coupling between the fields can be accomplished by either direct or load transfer coupling. Coupling across fields can be complicated because different fields may be solving for different types of analyses during a simulation. For example, in an induction heating problem, a harmonic electromagnetic analysis calculates Joule heating, which is used in a transient thermal analysis to predict a time-dependent temperature solution. The induction heating problem is complicated further by the fact that the material properties in both physics simulations depend highly on temperature.

Some of the applications in which coupled-field analysis may be required are pressure vessels (thermal-stress analysis), fluid flow constrictions (fluid-structure analysis), induction heating (magnetic-thermal analysis), ultrasonic transducers (piezoelectric analysis), magnetic forming (magneto-structural analysis), and micro-electromechanical systems (MEMS).

Static Analysis:

A static structural analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time. A static structural load can be performed using the ANSYS solver. The types of loading that can be applied in a static analysis include:

- Externally applied forces and pressures
- Steady-state inertial forces (such as gravity or rotational velocity)
- Imposed (nonzero) displacements

Temperatures (for thermal strain)

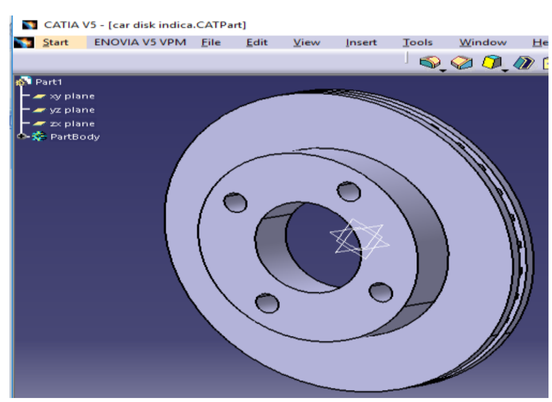
FE Analysis involves following major steps;

1) Pre-Processing
    - Geometry Modeling
    - Meshing
    - Material Properties
    - Contact Definition
    - Loading and boundary condition
2) Solution &
3) Post-Processing
    - Deformation (Static Analysis)
    - Stresses (Static Analysis)
    - Temperature (Thermal Analysis)

Geometry Details:

CAD Modeling of any project is one of the most time consuming process. One cannot shoot directly from the form sketches to Finite Element Model. CAD(Geometry) Modeling is the base of any project. Finite Element software will consider shapes, whatever is made in CAD model. CAD modelling of the disk brake is performed by using Catia.

Figure 1: CAD BRAKE DISC



Material Properties:

Table 1

Details of "Gray Cast Iron"	
Common Material Properties	
Density	7.2e-06 kg/mm³
Young's Modulus	1.1e+05 MPa
Thermal Conductivity	0.052 W/mm·°C
Specific Heat	4.47e+05 mJ/kg·°C
Tensile Yield Strength	0 MPa
Tensile Ultimate Strength	240 MPa

Table 2

Details of "Stainless Steel"	
Common Material Properties	
Density	7.75e-06 kg/mm³
Young's Modulus	1.93e+05 MPa
Thermal Conductivity	0.0151 W/mm·°C
Specific Heat	4.8e+05 mJ/kg·°C
Tensile Yield Strength	207 MPa
Tensile Ultimate Strength	586 MPa

Table 3

Details of "Aluminum Alloy"	
Common Material Properties	
Density	2.77e-06 kg/mm³
Young's Modulus	71000 MPa
Thermal Conductivity	table(T) = 0.14862 W/mm·°C
Specific Heat	8.75e+05 mJ/kg·°C
Tensile Yield Strength	280 MPa
Tensile Ultimate Strength	310 MPa

Meshing

Meshing involves division of the entire of model into small pieces called elements. This is done by meshing. It is convenient to select the free mesh because the has sharp curves, so that shape of the object will not alter. To mesh the plate the element type must be decided first. SOLID187 is used here for meshing plates.

SOLID187 Element Description:

SOLID187 element is a higher order 3-D, 10-node element. SOLID187 has a quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced from various CAD/CAM systems).

The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. The geometry, node locations, and the coordinate system for this element are shown in Figure: SOLID187 Geometry.In addition to the nodes, the element input data includes the orthotropic or anisotropic material properties. Orthotropic and anisotropic material directions correspond to the element coordinate directions

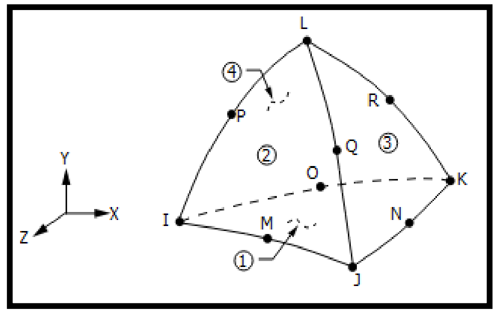


Figure 2: SOLID187 Geometry

SOLID187 Assumptions and Restrictions

- The element must not have a zero volume.
- Elements may be numbered either as shown in Figure: SOLID187 Geometry or may have node L below the I, J, K plane.
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for information about using midside nodes.
- When mixed formulation is used (KEYOPT(6) = 1 or 2), no midside nodes can be missed.
- If you use the mixed formulation (KEYOPT(6) = 1 or 2), the damped eigensolver is not supported. You must use the sparse solver (default).
- Stress stiffening is always included in geometrically nonlinear analyses (NLGEOM,ON). Pre-stress effects can be activated by the PSTRES

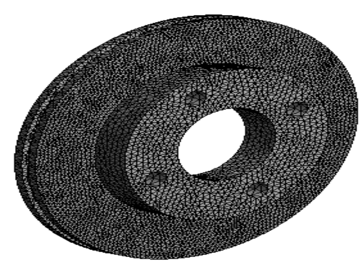
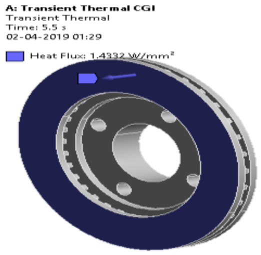
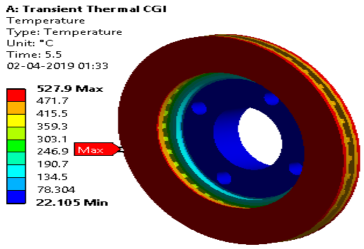
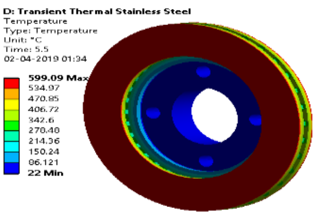


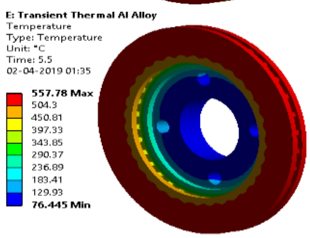
Fig 3: FE Modeling details; Nodes: 118572, Elements: 68555

Thermal Boundary Condition:

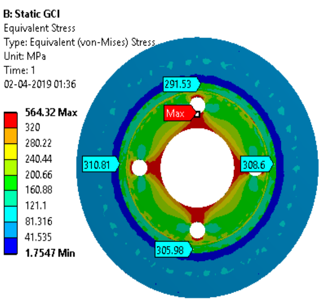


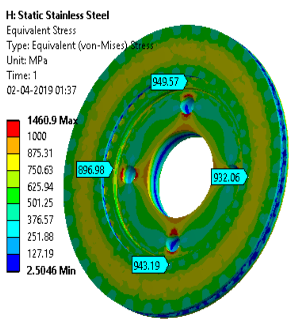


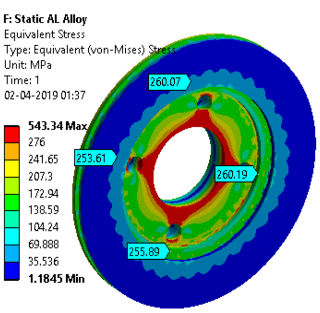




Static Analysis Results for temperature loading:

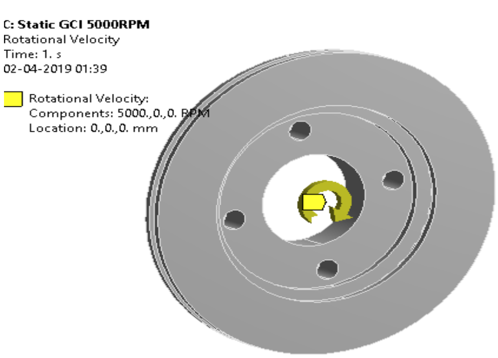




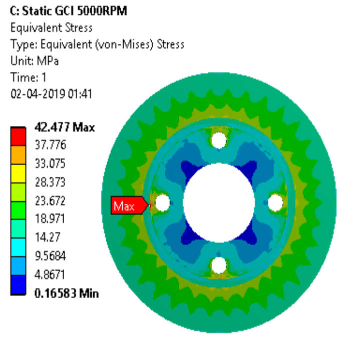


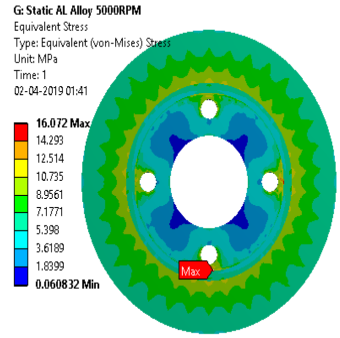
Static Loads and Boundary Conditions:

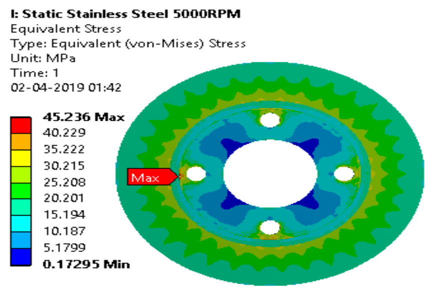
Rotational Velocity applied and stresses were plotted.



Static Analysis Results for Rotational Speed loading:

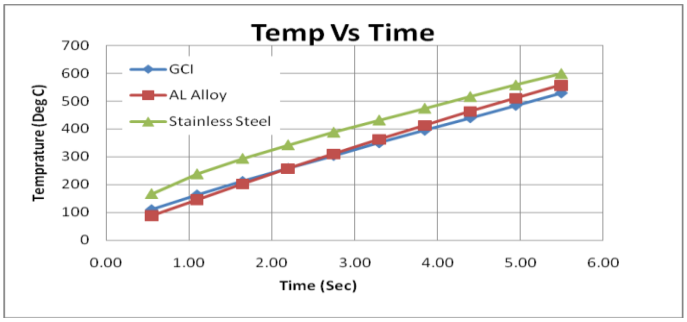


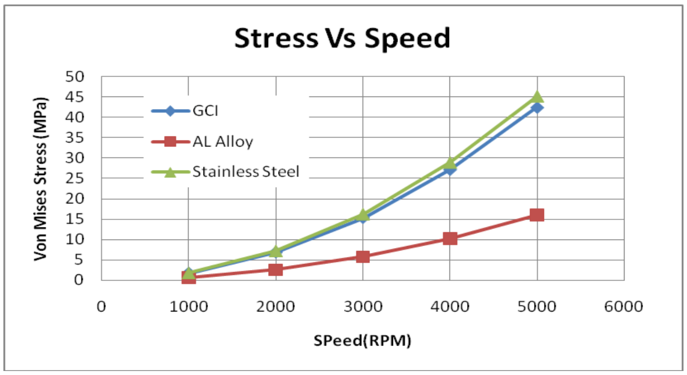


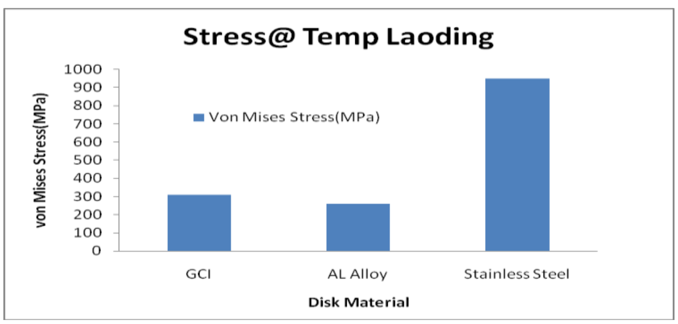


3. RESULTS: Thermal Analysis Results: Table 4

Braking on vehicle , Temp			
Time	GCI	AL Alloy	Stainless Steel
0.55	110	88	167
1.10	163	146	238
1.65	211	202	294
2.20	258	257	342
2.75	304	311	388
3.30	349	363	432
3.85	394	413	474
4.40	439	463	516
4.95	483	511	558
5.50	528	558	599







6. CONCLUSIONS:

This work has provided a comprehensive literature review of existing various research work carried out in terms of design, material selection methods, thermal analysis, structural analysis, FEA, Von misses stress, optimization and analysis of Brake Disc. An effort has been made to comprise all the important contributions to this area and highlighting the most pertinent literature available for investigating the brake disc.

Based on thermal and static analysis, the AL alloy materail is observed to be most safer and having higher FOS.

7. ACKNOWLEDGMENT:

The review presented in this work is by no means complete but gives a comprehensive representation of different thermal and structural analysis applied to brake disc. The author apologizes for unintentional exclusions of missing references and would appreciate receiving comments and pointers to other relevant literature for a future update.

8. REFERENCES

[1]. Introduction to Finite Elements in Engineering,Tirupathi, R.Chandrupatla Ashok D.Belegundu.

[2]. Finite Element Procedures, Klaus-Jurgen Bathe.

[3]. Finite Element Method, J.N.Reddy.

[4]. User Guide for ANSYS version 5.4.

[5]. Internet web site http://auto.howstuffworks.com/discbrake.htm.

[6]. Design Data Book,PSG college of technology.

[7]. Heat and Mass Transfer Data Book,C P Kothandaraman, S Subramanyan.