

Structural and Thermal Analysis of Vented and Normal Disc Brake Rotors

Yaswanth Moogilipuri

**M.Tech Student,
Machine Design,**

**Sarada Institute of Science Technology and
Management (Approved By Aicte, Approved By
Jntu Kakinada) Srikakulam.**

Mr. K Deepak, M.Tech

**Assistant Professor,
Mechanical Engineering,**

**Sarada Institute of Science Technology and
Management (Approved By Aicte, Affiliated To
Jntu Kakinada) Srikakulam.**

ABSTRACT:

In this paper, the disc brake is a device for slowing or stopping the rotation of a wheel. Repetitive braking of the vehicle leads to heat generation during each braking event. Transient Thermal and Structural Analysis of the Rotor Disc of Disk Brake is aimed at evaluating the performance of disc brake rotor of a car under severe braking conditions and there by assist in disc rotor design and analysis. Disc brake model and analysis is done using ANSYS workbench 14.5. The main purpose of this study is to analysis the thermo mechanical behavior of the dry contact of the brake disc during the braking phase. The coupled thermal and structural analysis is used to determine the deformation and the Von Mises stress established in the disc for the both solid and ventilated disc with two different materials to enhance performance of the rotor disc. A comparison between analytical and results obtained from FEM is done and all the values obtained from the analysis are less than their allowable values. Hence and rotor disc is suggested based on the performance, strength best suitable design, material and rigidity criteria.

1.0. INTRODUCTION:

A disc brake consists of a disc component bolted to the wheel hub and a stationary housing called caliper. The caliper is connected to some stationary part of the vehicle like the axle casing or the stub axle as 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 caliper for hydraulic fluid to enter or leave each housing. The 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 following figure.



Figure 1: Disc Brake full assemble view

1.1 WORKING PRINCIPLE:

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 brakes the rubber-sealing ring acts as a return spring and retract the pistons and the friction pads away from the disc (see fig).

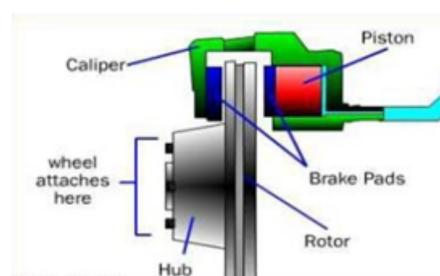


Figure 2: Working principle of Disc brakes

2.0. CALCULATIONS FOR INPUT PARAMETERS:

In the aspect of the car accident prevention, the braking performance of vehicles has been a critical issue. The rotor model heat flux is calculated for the car moving with a velocity 27.77 m/s (100kmph) and the following is the calculation procedure.

Data:

- 1) Mass of the vehicle = 2500 kg
- 2) Initial velocity (u) = 27.7 m/s (100 kmph)
- 3) Vehicle speed at the end of the braking application (v) = 0 m/s
- 4) Brake rotor diameter = 0.262 m
- 5) Axle weight distribution 30% on each side (γ)=0.3
- 6) Percentage of kinetic energy that disc absorbs (90%) k=0.9
- 7) Acceleration due to gravity g = 9.81 m/s²
- 8) coefficient of friction for dry pavement μ=0.7

a) Energy generated during braking:

$$K.E = k \frac{1}{2} \gamma \frac{m(u-v)^2}{2} = 129465.3 \text{ J}$$

b) To calculate stopping distance:

$$d = \frac{u^2}{2\mu g} = 56.18 \text{ m}$$

c) Braking Power:

Braking power during continued braking is obtained by differentiating energy with respect to time.

$$= 32366.325 \text{ W}$$

d) Calculate the Heat Flux (Q):

Heat Flux is defined as the amount of heat transferred per unit area per unit time, from or to a surface.

$$P_b = K.E / t = 32366.325 \text{ W}$$

d) Calculate the Heat Flux (Q):

Heat Flux is defined as the amount of heat transferred per unit area per unit time, from or to a surface.

$$Q = P_b / A = 1201084.422 \text{ W/m}^2$$

2.1 MATERIAL PROPERTIES:

The contact area between the pads and disc of brake components, heat is generated due to friction. For calculation of heat generation at the interface of these two sliding bodies, two methods are suggested on the basis of “law of conservation of energy which states that the kinetic energy of the vehicle during motion is equal to the dissipated heat after vehicle stop” [7]. The material properties and parameters adopted in the calculations are as shown in table [9].

Material Properties	Stainless Steel	Cast Iron
Thermal Conductivity (w/m K)	36	50
Density, ρ (kg/m ³)	7100	6600
Specific heat, c (J/Kg°C)	320	380
Thermal expansion, α (10 ⁻⁶ /K)	0.12	0.15
Elastic modulus, E (Gpa)	210	110
Coefficient of friction, μ	0.5	0.5
Heat transfer coefficient h (w/km ²)	150	120
Operating Conditions		
Angular velocity, (rad/s)	50	50
Braking Time, Sec	4	4
Hydraulic Pressure, P (Mpa)	1	1

Single stop temperature rise T max is the temperature rise due to single braking condition.

$$T_{max} = \frac{0.527 \times q \times \sqrt{t}}{\sqrt{(\rho \cdot c \cdot k)}} + T_{amb}$$

The relative brake temperature after the nth brake application can be calculated using relation,

$$T_{roa} - T_i = \frac{\left[1 - e^{(-nhAt_c) / (\rho cv)} \right] [\Delta T]}{1 - e^{(-hAt_c) / (\rho cv)}}$$

The compressive stresses ‘σ’ developed in the surface of a disc from sudden temperature increases is

$$\sigma = \frac{E}{1 - \nu} \times \alpha \times \Delta T$$

3.0.FINITE ELEMENT ANALYSIS:

3.1 INTRODUCTION TO FINITE ELEMENT METHOD:

The finite element method is a powerful tool to obtain the numerical solution of wide range of engineering problem.

The method is general enough to handle any complex shape or geometry, for any material under different boundary and loading conditions. The generality of the finite element method fits the analysis requirement of today's complex engineering systems and designs where closed form solutions of governing equilibrium equations are usually not available. In addition, it is an efficient design tool by which designers can perform parametric design studies by considering various design cases, (different shapes, materials, loads, etc.) and analyze them to choose the optimum design.

The method originated in the aerospace industry as a tool to study stress in a complex airframe structures. It grows out of what was called the matrix analysis method used in aircraft design. The method has gained increased popularity among both researchers and practitioners. The basic concept of finite element method is that a body or structure may be divided into small elements of finite dimensions called "finite elements". The original body or the structure is then considered, as an assemblage of these elements connected at a finite number of joints called nodes or nodal points.

3.1 GENERAL PROCEDURE FOR FINITE ELEMENT METHOD

The finite element method is a method of piecewise approximation in which the structure or body is divided into small elements of finite dimensions called finite elements and then the original body or the structure is considered as an assemblage of these elements connected at finite number of joints called nodal points or nodes. Since the actual variation of field variables like displacement, stress, temperature, pressure or velocity inside the continuum are not known, the variation of the field variable inside a finite element can be approximated by a simple function.

These approximation functions called interpolation models are defined in terms of the values of the field variables of the nodes. The nodal values of the field variable are obtained by solving the field equations, which are generally in the form of matrix equations. Once the nodal values are known, the approximating functions define the field variable throughout the assemblage of elements.

The solutions of general continuum problems by the finite element method always follow an orderly step-by-step process. The step-by-step procedure for static structural problem can be stated as follows: Step 1:- Description of Structure (Domain). The first step in the finite element method is to divide the structure of solution region into sub divisions or elements. Step 2:- Selection of proper interpolation model. Since the displacement (field variable) solution of a complex structure under any specified load conditions cannot be predicted exactly, we assume some suitable solution, within an element to approximate the unknown solution.

The assumed solution must be simple and it should satisfy certain convergence requirements. Step 3:- Derivation of element stiffness matrices (characteristic matrices) and load vectors. From the assumed displacement model the stiffness matrix $[K(e)]$ and the load vector $P(e)$ of element „e are to be derived by using either equilibrium conditions or a suitable Variation principle. Step 4:- Assemblage of element equations to obtain the equilibrium equations. Since the structure is composed of several finite elements, the individual element stiffness matrices and load vectors are to be assembled in a suitable manner and the overall equilibrium equation has to be formulated as

$$[K]\phi = P$$

Where $[K]$ is called assembled stiffness matrix, Φ is called the vector of nodal displacement and P is the vector or nodal force for the complete structure. Step 5:- Solution of system equation to find nodal values of displacement (field variable) The overall equilibrium equations have to be modified to account for the boundary conditions of the problem. After the incorporation of the boundary conditions, the equilibrium equations can be expressed as,

$$[K]\phi = P$$

For linear problems, the vector „ ϕ can be solved very easily. But for non-linear problems, the solution has to be obtained in a sequence of steps, each step involving the modification of the stiffness matrix $[K]$ and „ ϕ or the load vector P . Step 6:- Computation of element strains and stresses. From the known nodal displacements, if required, the element strains and stresses can be computed by using the necessary equations of solid or structural mechanics. In the above steps, the words indicated in brackets implement the general FEM step-by-step procedure.

3.2 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.

3.3 PROCEDURE FOR ANSYS ANALYSIS:

Static analysis is used to determine the displacements, stresses, strains and forces in structures or components due to loads that do not induce significant inertia and damping effects. Steady loading in response conditions are assumed. The kinds 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 (non-zero) displacements, temperatures (for thermal strain). A static analysis can be either linear or non linear. In our present work we consider linear static analysis.

The procedure for static analysis consists of these main steps:

1. Building the model.
2. Obtaining the solution.
3. Reviewing the results.

3.4 DIFFERENT CASES OF ANALYSIS:

In our present study the following different cases are considered:

Case I: Coupled Structural/Thermal Analysis is carried out for 4, 5, 6 seconds of braking on a cast iron solid disc.

Case II: Coupled Structural/Thermal Analysis is carried out for 4, 5, 6 seconds of braking on a steel solid disc.

Case III: Coupled Structural/Thermal Analysis is carried out for 4, 5, 6 seconds of braking on a cast iron ventilated disc.

Case IV: Coupled Structural/Thermal Analysis is carried out for 4, 5, 6 seconds of braking on steel ventilated disc.

3.5 PROCEDURE FOR CREATNG THE 3D MODELS:

The following are the commands are used in Unigraphics for creating solid and ventilated disk rotors. SKETCH: Creates a sketch in current application. Use the direct sketch tools to add the curves, dimensions, constraints etc.

EXTRUDE:

Creates a feature by extruding a section along a vector.

HOLE:

Add a holes to one or more solid bodies in a part or assembly with options for counter bored, counter sunk and threaded.

UNITE:

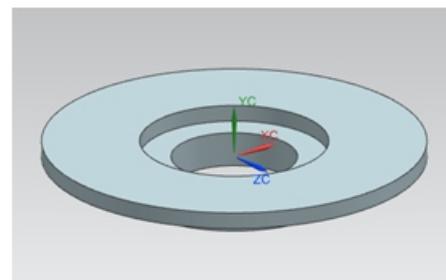
Combines the volume of two or more bodies into a single body.

SUBTRACT:

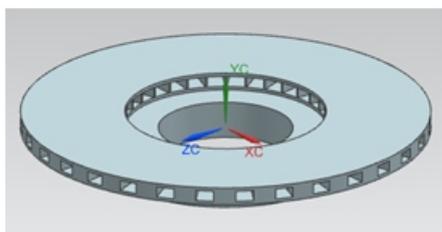
Subtracts the volume of a one solid body from another, leaving a void.

PATTERN FEATURE:

Copies features into many patterns or layouts with various options for pattern boundary, instance orientation, clocking and variance.



SOLID DISK CREATED IN UNIGRAPHICS



VENTILATED DISK CREATED IN UNIGRAPHICS

3.6 PROCEDURE FOR TRANSIENT THERMAL ANALYSIS

- 1) Start ANSYS workbench to display the workbench window.
- 2) In the workbench window, double click on the Transient Thermal analysis system in the analysis systems toolbox window, the Transient Thermal analysis system is added in the Project schematic window.
- 3) Import the solid disk Para solid file in to the geometry cell of the Transient Thermal analysis system.
- 4) Next double click on the Model cell of the Transient thermal analysis system, the Mechanical window is displayed.
- 5) Make sure Metric is selected from the Units menu in the menu bar.

MESHING DETAILS:

The goal of meshing in Workbench is to provide robust, easy to use meshing tools that will simplify the mesh generation process. The model using must be divided into a number of small pieces known as finite elements. Since the model is divided into a number of discrete parts, in simple terms, a mathematical net or “mesh” is required to carry out a finite element analysis. A finite element mesh model generated is shown in fig.3. The mesh results are as shown in table. The elements used for the mesh of the model are tetrahedral three-dimensional elements with 8 nodes.

Models	Material Type	Number of Elements	Number of nodes
Solid disk	Cast iron	8682	40935
	Stainless steel		
Ventilated disk	Cast iron	55426	112171
	Stainless steel		

THERMAL BOUNDARIES:

The boundary conditions are introduced into module ANSYS workbench, by choosing the mode of simulation and by defining the physical properties of materials and the initial conditions of simulations. In this work, a transient thermal analysis will be carried out to investigate the temperature variation across the both disc by applying heat flux value for repeated braking applications using ANSYS.

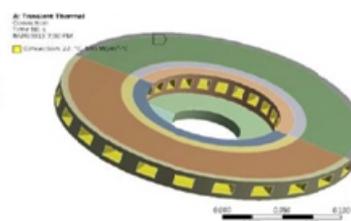
Heat flux q (for 4 seconds of braking) = 1201084.422W/m²

Heat flux q (for 5 seconds of braking) = 960867.373 W/m²

Heat flux q (for 6 seconds of braking) = 800722.81 W/m²

Convection film co-efficient (h) = 5.0 W/m² k

The temperature fixed at the hub bore grinds = 35°C



Thermal boundary conditions for ventilated disk rotor

Repeat the same procedure for transient thermal analysis on both stainless steel and cast iron solid disks and ventilated disk for the braking time 4, 5, and 6 seconds.

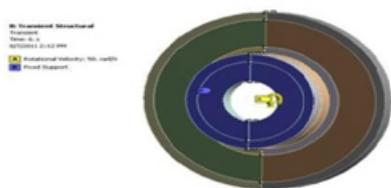
3.7 PROCEDURE FOR TRANSIENT STRUCTURAL ANALYSIS

- 1) Start ANSYS workbench to display the workbench window.
- 2) In the workbench window, double click on the Transient structural analysis system in the analysis systems toolbox window, the Transient structural analysis system is added in the Project schematic window.
- 3) Import the solid disk Para solid file in to the geometry cell of the Transient structural analysis system.
- 4) Next double click on the Model cell of the Transient structural analysis system, the Mechanical window is displayed.

5) Make sure Metric is selected from the Units menu in the menu bar.

STRUCTURAL BOUNDARY CONDITIONS:

The structural analysis is carried out by coupling thermal analysis. In addition convection heat transfer coefficient is applied at the surface of the ventilated disc for the analysis as shown in fig .



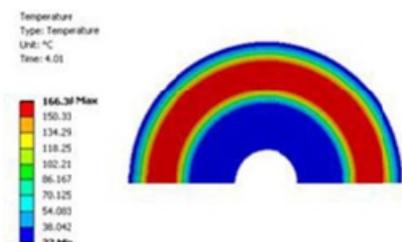
Structural boundary conditions

Repeat the same procedure for transient structural analysis on both stainless steel and cast iron solid disks and ventilated disk for the braking time 4, 5, and 6 seconds.

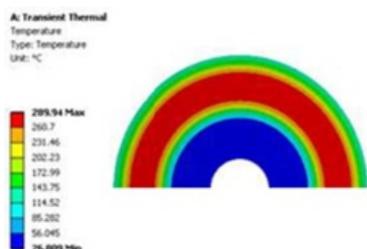
4.0.RESULTS AND DISCUSSIONS:

CASE-1 SOLID DISK:

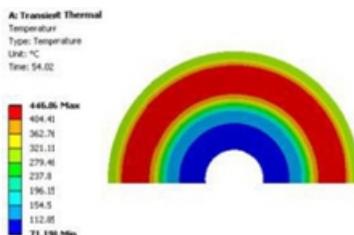
STAINLESS STEEL TEMPERATURE RESULTS AT BRAKING TIME 4 SECONDS



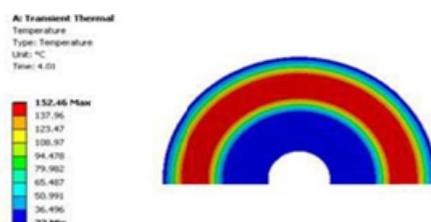
STAINLESS STEEL TEMPERATURE RESULTS AT BRAKING TIME 5 SECONDS



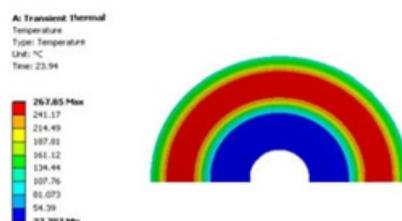
STAINLESS STEEL TEMPERATURE RESULTS AT BRAKING TIME 6 SECONDS



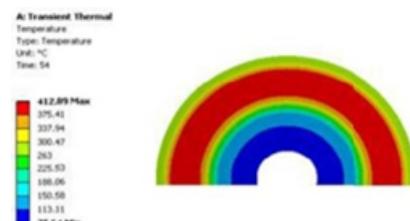
CAST IRON TEMPERATURE RESULTS AT BRAKING TIME 4 SECONDS



CAST IRON TEMPERATURE RESULTS AT BRAKING TIME 5 SECONDS

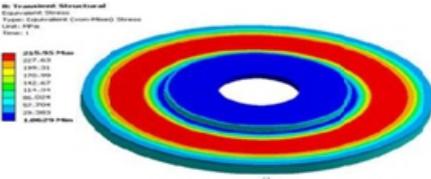
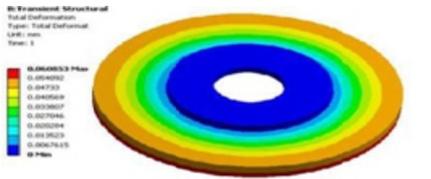


CAST IRON TEMPERATURE RESULTS AT BRAKING TIME 6 SECONDS

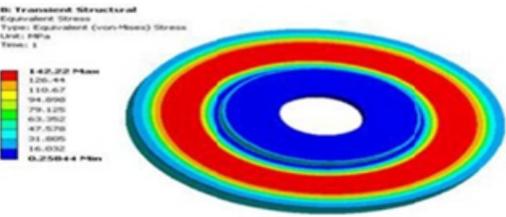
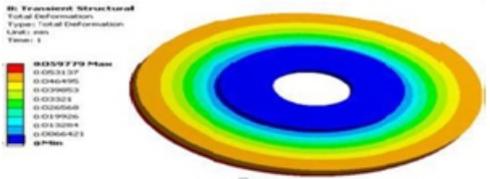


During each braking cycle, the temperature on the surface of the disk is raised. During 1st braking, the temperature rises from ambient temperature 22 to 166. Similarly for alternate braking applications, during 3rd braking and 6th braking it rises to 289 and 446 for solid stainless steel disc respectively. Similarly in the cast iron solid disc 1st, 3rd and 6th braking applications the temperature rise is 152, 267 and 412 respectively. The maximum temperature rise is indicated in red color and green color shows average temperature rise at the friction surface around the circumference of the disc as shown in figures.

STAINLESS STEEL DEFORMATION CONTOURS:



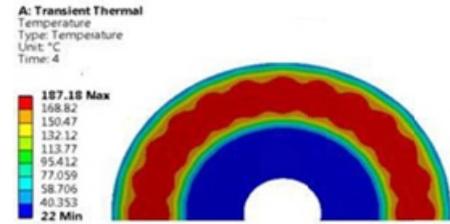
**Deformation and Von mises contours
 CAST IRON DEFORMATION CONTOURS:**



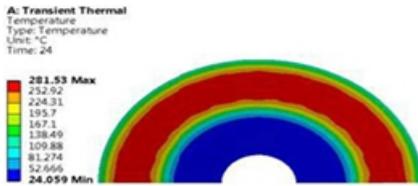
Deformation and Von mises contours

The distribution of the total distortion in solid stainless steel cast iron disc brake is shown in fig. The scale of values of the deformation varies from 0 μ m with 0.06mm which corresponds to the time of braking. After the 6th braking, the maximum deflection induced is 0.0608mm in SS disc and 0.059 in C I disc, which is less than the allowable deflection 0.5mm. During the total time simulation of braking for a full disc presents the distribution of the constraint equivalent of Von Mises Stresses to various moments of simulation as shown in fig12. The scale of values varies from 0 MPa to 255MPa in stainless steel disc and 142 MPa in cast iron disc, which is the maximum thermal stress induced at maximum temperature rise after 6th braking application.

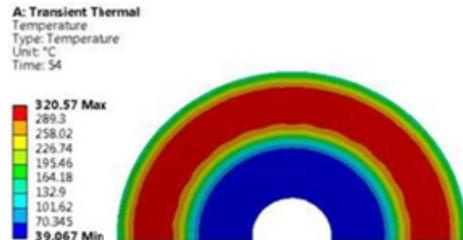
**CASE-2VENTILATED DISK:
 STAINLESS STEEL TEMPERATURE RESULTS AT
 BRAKING TIME 4 SECONDS**



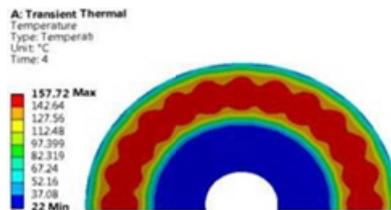
**STAINLESS STEEL TEMPERATURE RESULTS AT
 BRAKING TIME 5 SECONDS**



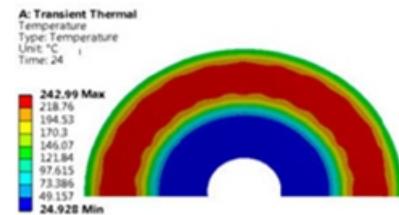
**STAINLESS STEEL TEMPERATURE RESULTS AT
 BRAKING TIME 6 SECONDS**



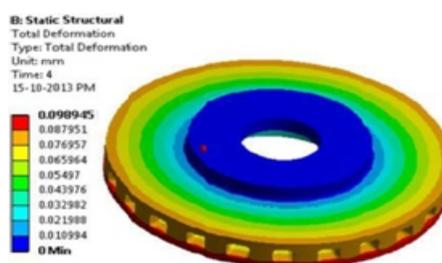
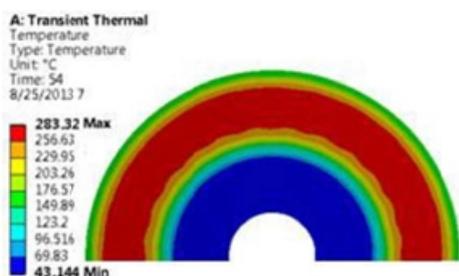
**CAST IRON TEMPERATURE RESULTS AT BRAKING
 TIME 4 SECONDS**



**CAST IRON TEMPERATURE RESULTS AT BRAKING
 TIME 5 SECONDS**

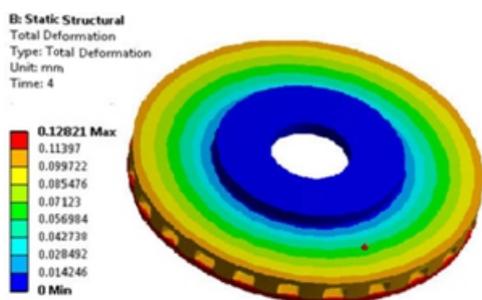


**CAST IRON TEMPERATURE RESULTS AT BRAKING
 TIME 6 SECONDS**

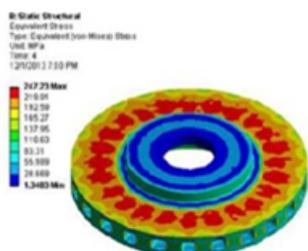


During each braking cycle, the temperature on surface of the disc is raises. During 1st braking, the temperature rises from ambient temperature 22 to 187. Similarly for alternate braking applications, during 3rd braking and 6th braking it rises to 281 and 320 for solid stainless steel disc respectively. Similarly in the cast iron solid disc 1st, 3rd and 6th braking applications the temperature rise is 157 , 242 and 283 respectively. The maximum temperature rise is indicated in red color and green color shows average temperature rise at the friction surface around the circumference of the disc as shown in figures.

STAINLESS STEEL DEFORMATION CONTOURS:



Deformation contours



Von -mises contours CAST IRON DEFORMATION CONTOURS:

Deformation and Von-mises contours

The distribution of the total distortion in stainless steel and cast iron ventilated disc brake is shown in fig. The scale of values of the deformation varies from 0 μm to 0.0128mm and which corresponds to the time of braking. After the 6th braking , the maximum deflection induced is 0.0128mm in SS ventilated disc and 0.098mm in C I ventilated disc, which is less than the allowable deflection 0.5mm. The distribution of the constraint equivalent of Von Mises Stresses . The scale of values varies from 0 MPa to 288MPa for in stainless steel disc and 110 MPa in cast iron disc. Which is the maximum thermal stress induced at maximum temperature rise 320 and 283 after 6th braking application respectively.

COMPARISON THE RESULTS OF VENTILATED AND SOLID DISKS:

S.NO	Material Flange width mm	Analytical Max. Temp °C	FEM Max. Temp °C	Deflection in mm	Analytical misses stress Mpa	FEM Von mises stress Mpa
SOLID DISC						
1	SS 24	464	446	0.0608	270	256
2	CI 24	425	413	0.059	160	142
VENTILATED DISC						
1	SS	340	321	0.105	186	247
2	CI	308	283	0.0908	99	110

Comparing the different results of temperature rise, deflection, and stress field obtained from analysis it shows that the ventilated cast iron disc has reduction in temperature, deflection and stresses. It is concluded that ventilated type cast iron disk brake is the best for the present application.

CONCLUSIONS:

Comparing the different results of temperature rise, deflection, and stress field obtained from analysis it shows that in the ventilated cast iron disc reduction in temperature, stresses and deformation by 31.47% and 22.5% 8% respectively than the solid disc. It is concluded that ventilated type disk brake is the best for the present application. 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.

REFERENCES:

1. Lee K (1999) Numerical prediction of brake fluid temperature rise during braking and heat soaking. SAE, International Congress and Exposition Detroit, Michigan, 1–4 March. URL: <http://delphi.com/pdf/techpapers/1999-01-0483.PDF>
2. Valvano T, Lee K (2000) An analytical method to predict thermal distortion of a brake rotor. SAE, World Congress, Detroit, Michigan, 6–9 March. URL: <http://www.sae.org/technical/papers/2000-01-0445>
3. Mackin TJ, Noe SC et al (2002) Thermal cracking in disc brakes. *Eng Failure Analysis* 9(1):63– URL:<http://www.ingentaconnect.com/content/els/13506307/2002/00000009/00000001/art00037>
4. Yevtushenko A, Chapovska R (1997) Effect of time-dependent speed on frictional heat generation and wear in transient axisymmetrical contact of sliding. *Arch ApplMech* 67:331–338. doi:10.1007/s004190050121
5. Ostermeyer GP (2001) On the dynamics of the friction coefficient. *J Wear* 254(9):852–858. doi:10.1016/S0043-1648(03)00235-7
6. Gao CH, Lin XZ (2002) Transient temperature field analysis of a brake in a non-axisymmetric three-dimensional model. *J Mat Proc Tech* 129:513–517. doi:10.1016/S0924-0136(02)00622-2
7. Dufre'noy P (2004) Two-/three-dimensional hybrid model of the thermomechanical behavior of disc brakes. *J Rail Rapid Transit Part F* 218:17–30. doi:10.1243/095440904322804402
8. Voldr'ich J (2006) Frictionally excited thermoelastic instability in disc brakes—Transient problem in the full contact regime. *Int J MechSci* 49(2):129–137. doi:10.1016/j.ijmecsci.2006.08.008
9. Naji M, Al-Nimr M, Masoud S (2000) Transient thermal behavior of a cylindrical brake system. *J Heat Mass Transf* 36:45–49