International Journal of Applied Engineering Research ISSN 0973-4562 Volume 19, Number 1 (2024) pp. 25-32

Performance Analysis on Thermal Reflection of Disc Brake Applying Soft Computing

Sourav Bhattacherjee[*1], Bidyut Jana [2], Alamgir Mondal [3] and Pinku Kora[4]

[*1] Faculty (CEng.-India) of Department of Mechanical Engineering, Sanaka Educational Trust Group of Institutions

[234] (UG Student), Department of Mechanical Engineering Sanaka Educational Trust Group of Institutions, Durgapur, West Bengal, India-713212.

Abstract

Braking is a process which converts the kinetic energy of the vehicle into mechanical energy which must be dissipated in the form of heat. The disc brake is a device for de-accelerating or stopping the rotation of a wheel. A brake disc (or rotor) usually made of cast iron or ceramic composites, is connected to the wheel or the axle. Friction material in the form of brake pads (mounted on a device called a brake calliper) is forced mechanically, hydraulically, pneumatically or electromagnetically against both sides of the disc to stop the wheel. The present research is basically deals with the modelling and analysis of solid and ventilated disc brake using Pro-E and ANSYS. Finite element (FE) models of the brake-disc are created using Pro-E and simulated using ANSYS which is based on the finite element method (FEM). In this research Coupled Analysis (Structural & Thermal analysis) is performed in order to find the strength of the disc brake.. Comparison has been done for displacement, stresses, nodal temperatures, etc. for the two materials to suggest the best material, applicable for FSAE (Formula Society of Automotive Engineers) car also. An analytical model is presented in this paper for the determination of the contact temperature distribution on the working surface of a brake. To consider the effects of the moving heat source (the pad) with relative sliding speed variation, a transient finite element technique is used to characterize the temperature fields of the solid rotor with appropriate thermal boundary conditions. In this paper three different cut patterns of brake disc are studied for heat transfer rate. Numerical results shows that the operating characteristics of the brake exert an essentially influence on the surface temperature distribution and the maximal contact temperature.

Keywords: Design, Analysis, Brake Disc, Soft computing,

INTRODUCTION

In today's growing automotive market the competition for better performance vehicle is growing enormously. The disc brake is a device used for slowing or stopping the rotation of the wheel. A brake is usually made of cast iron or ceramic composites include carbon, aluminum, Kevlar and silica which is connected to the wheel and axle, to stop the vehicle. A friction material produced in the form of brake pads is forced mechanically, hydraulically, pneumatically and electromagnetically against the both side of the disc. Based on the design configurations vehicle friction brakes are grouped into disc brakes and drum brakes. The disc brake acts as 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. 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. 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 heat generated between the brake pad & disc has to be dissipated by passing air over them. This heat transfer takes place by conduction, convection and somewhat by radiation. To achieve proper cooling of the disc and the pad by convection, study of the heat transport phenomenon between disc, pad and the air medium is necessary. Then it is important to analyze the thermal performance of the disc brake system to predict the increase in temperature during braking. Convective heat transfer model has been developed to analyze the cooling performance. Our project is about disc brakes modeling and analysis. In this papertwo different cut patterns ofbrake discs are analyzedfor heat transfer rate. Brake discs are provided with cuts to increase the area coming in contact with air and improve heat transfer from disc.

METHODOLOGY

The disc brake is a device used for slowing or stopping the rotation of the vehicle. Number of times using the brake for vehicle leads to heat generation during braking event, such that disc brake undergoes breakage due to high Temperature. Disc brake model is done by CATIA/PROE and analysis is done by using ANSYS workbench. The main purpose of this project is to study the Thermal analysis of the Materials for the Cast Iron, and HSS M2.A study has been conducted 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 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. Thus, this 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.

Formulation and development of governing equations:

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. 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.

Convergence requirement for fem:

The solutions of general continuum problems by the finite element method always follow an orderly step-by-step process as follows:

Step 1:- Description of Structure (Domain). The first step in the finite element method is to divide the structure of solution region in to 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

$[K]\varphi = P$, 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,

$[K]\varphi = P$. 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.

Procedure for 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 nonlinear. 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.

Modeling and Analysis

3.1 Build the Model

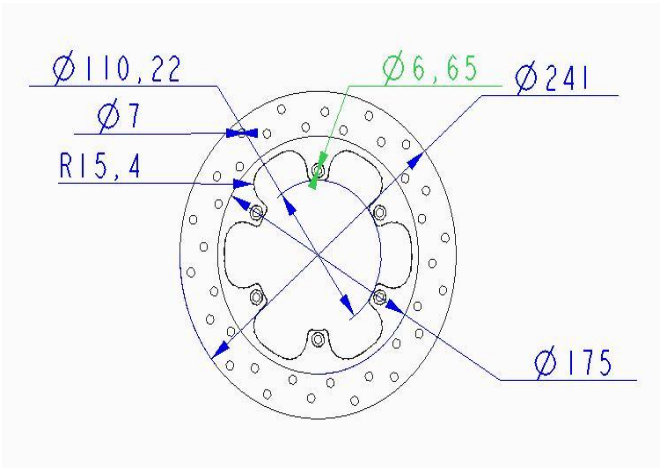

Figure : 3.1 Schematic Diagram of a Disc brake

In this step we specify the job name and analysis title use PREP7 to define the element types, element real constants, material properties and model geometry element types both linear and non-linear structural elements are allowed. The ANSYS element library contains over 80 different element types. A unique number and prefix identify each element type. E.g. BEAM 3, PLANE 55, SOLID 45 and PIPE 16

3.2 Material Properties

Young's modulus(EX) must be defined for a static analysis .If we plan to apply inertia loads(such as gravity) we define mass properties such as density(DENS).Similarly if we plan to apply thermal loads (temperatures) we define coefficient of thermal expansion(ALPX).

3.3 Obtain the Solution

In this step we define the analysis type and options, apply loads and initiate the finite element solution. This involves three phases:

a) Pre – processor phaseb) Solution phasec) Post-processor phase

3.3.1 Pre – Processor:

Preprocessor has been developed so that the same program is available on micro, mini, super-mini and mainframe computer system. This slows easy transfer of models one system to other. Preprocessor is an interactive model builder to prepare the FE (finite element) model and input data. The solution phase utilizes the input data developed by the preprocessor, and prepares the solution according to the problem definition. It creates input files to the temperature etc., on the screen in the form of contours.

3.3.1.1 Geometrical Definitions

There are four different geometric entities in preprocessor namely key points, lines, areas and volumes. These entities can be used to obtain the geometric representation of the structure. All the entities are independent of other and have unique identification labels.

3.3.1.2 Model Generations

Two different methods are used to generate a model:

a) Direct generation.b) Solid modeling

With solid modeling we can describe the geometric boundaries of the model, establish controls over the size and desired shape of the elements and then instruct ANSYS program to generate all the nodes and elements automatically. By contrast, with the direct generation method, we determine the location of every node and size, shape and connectivity of every element prior to defining these entities in the ANSYS model. Although, some automatic data generation is possible (by using commands such as FILL, NGEN, EGEN etc.) the direct generation method essentially a hands on numerical method that requires us to keep track of all the node numbers as we develop the finite element mesh. This detailed book keeping can become difficult for large models, giving scope for modeling errors. Solid modeling is usually more powerful and versatile than direct generation and is commonly preferred method of generating a model.

3.3.1.3 Mesh Generation

In the finite element analysis the basic concept is to analyze the structure, which is an assemblage of discrete pieces called elements, which are connected, together at a finite number of points called Nodes. Loading boundary conditions are then applied to these elements and nodes. A network of these elements is known as Mesh.

3.3.1.4 Finite Element Generation

The maximum amount of time in a finite element analysis is spent on generating elements and nodal data. Preprocessor allows the user to generate nodes and elements automatically at the same time allowing control over size and number of elements. There are various types of elements that can be mapped or generated on various geometric entities.

The elements developed by various automatic element generation capabilities of preprocessor can be checked element characteristics that may need to be verified before the finite element analysis for connectivity, distortion-index, etc. Generally, automatic mesh generating capabilities of preprocessor are used rather than defining the nodes individually. If required, nodes can be defined easily by defining the allocations or by translating the existing nodes. Also one can plot, delete, or search nodes.

3.3.1.5 Boundary Conditions and Loading

After completion of the finite element model it has to constrain and load has to be applied to the model. User can define constraints and loads in various ways. All constraints and loads are assigned set 1D. This helps the user to keep track of load cases.

3.3.1.6 Model Display

During the construction and verification stages of the model it may be necessary to view it from different angles. It is useful to rotate the model with respect to the global system and view it from different angles. Preprocessor offers this capability. By windowing feature preprocessor allows the user to enlarge a specific area of the model for clarity and details. Preprocessor also provides features like smoothness, scaling, regions, active set, etc. for efficient model viewing and editing.

3.3.1.7 Material Definitions

All elements are defined by nodes, which have only their location defined. In the case of plate and shell elements there is no indication of thickness. This thickness can be given as element property. Property tables for a particular property set 1-D have to be input. Different types of elements have different properties for e.g. Beams: Cross sectional area, moment of inertia etc. Shells: Thickness Springs: Stiffness Solids: None

The user also needs to define material properties of the elements. For linear static analysis, modules of elasticity and Poisson's ratio need to be provided. For heat transfer, coefficient of thermal expansion, densities etc. are required. They can be given to the elements by the material property set to 1-D.

3.3.2 Solution

The solution phase deals with the solution of the problem according to the problem definitions. All the tedious work of formulating and assembling of matrices are done by the computer and finally displacements and stress values are given as output. Some of the capabilities of the ANSYS are linear static analysis, non-linear static analysis, transient dynamic analysis, etc.

3.3.3. Post – Processor

It is a powerful user-friendly post-processing program using interactive color graphics. It has extensive plotting features for displaying the results obtained from the finite element analysis. One picture of the analysis results (i.e. the results in a visual form) can often reveal in seconds what would take an engineer hour to asses from a numerical output, say in tabular form. The engineer may also see the important aspects of the results that could be easily missed in a stack of numerical data. Employing state of art image enhancement techniques, facilities viewing of:

a) Contours of stresses, displacements, temperatures, etc. b) Deform geometric plots c) Animated deformed shapes d) Time-history plotse) Solid sectioningf) Hidden line plotg) Light source shaded ploth) Boundary line plot etc. The entire range of post processing options of different types of analysis can be accessed through the command/ menu mode there by giving the user added flexibility and convenience.

3.4 Discbrakecalculations

3.4.1 Assumptions

(1) The analysis is done taking the distribution of the braking torque between the front wheel and rear wheel is 32:68
(2) Brakes is applied on all the front wheel only.
(3) The analysis is based on pure thermal loading. The analysis does not determine the life of the disc brake.
(4) Only ambient air-cooling is taken in to account and no forced convection is taken.
(5) The kinetic energy of the vehicle is lost through the brake discs i.e. no heat loss between the tyres and the road surface and the deceleration is uniform.
(6) The disc brake model used is of homogenous material.
(7) The thermal conductivity of the material used for the analysis is uniform throughout.
(8) The specific heat of the material used is constant throughout and does not change with the temperature. 9. Heat flux on each front wheel is applied on one side of the disc only.

3.4.2 Calculation 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:

(i) Mass of the vehicle = 250 kg
(ii) Initial velocity (u) = 22.21 m/s (80 km/h)
(iii) Vehicle speed at the end of the braking application (v) = 0 m/s
(iv) Brake rotor diameter = 0.261 m
(v) (Static front axle load /Motor cycle total load) = (γ) = 0.3
(vi) Percentage of kinetic energy that disc absorbs (90%) k=0.9
(vii) Acceleration due to gravity g =9.81m/s^2
(viii) Coefficient of friction for dry pavement μ=0.45.

a) Energy generated during braking: K.E. =γk m (u-v)^2/ 2

b) To calculate deceleration time: v= u+at, Deceleration time = Braking time = 5s

c) Braking Power: Braking power during continued braking is obtained by differentiating energy with respect to time, P_b= (K.E. / t)

d) Calculate the Heat Flux (Q): Heat Flux is defined as the amount of heat transferred per unit area per unit time, Q= (P_b / A)

Table 3.1: Calculation for Input Parameters

Parameters	Formulae	Disc design		Stainless Steel	Cast iron
Kinetic energy	K.E.=γk m(u-v)^2/ 2	For all models		20958.022 J	20958.023 J
Deceleration Time	v = u + at	For all models		6 sec	6 sec
Braking Power	P_b= (K.E./ t)	For all models		4191.61 W	3494 W
Calculate the Heat Flux	Q= (P_b / A)	Model 1	=0.01473^2	142281.21	118567.66
		Model 2	=0.014145^2	148165.55	123471.22
		Model 3	=0.014939^2	140290.66	116908.87

3.4.3 Analytical Temperature Rise Calculations

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". The material properties and parameters adopted in the calculations are as shown in table.

Table 3.2: Material Properties for Stainless Steel and Cast Iron

Material Properties	Stainless Steel	Cast Iron
Thermal conductivity(w/m k)	36	50
Density , ρ (kg/m3)	7100	6600
Specific heat , c (J/Kg c)	320	380
Thermal expansion , α (10-6 /k)	0.12	0.16
Elastic modulus, E (GPa)	210	110
Coefficient of friction, μ	0.5	0.5
Film co-efficient h(w/km2)	240	280
Operation conditions		
Angular velocity,(rad /s)	50	50
Braking Time Sec	5	6
Hydraulic pressure, P (M pa)	1	1

3.5 Fem Models of Brake Disc with Meshing

Model 1:

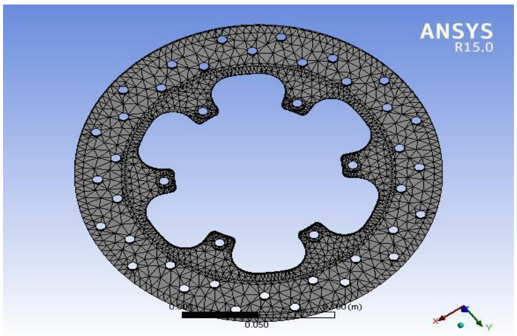

Figure 3.2: Meshing of Model 1

Sizing	
Relevance Center	Fine
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	2.5e-004 m
Statistics	
Nodes	53772
Elements	29244

Model 2:

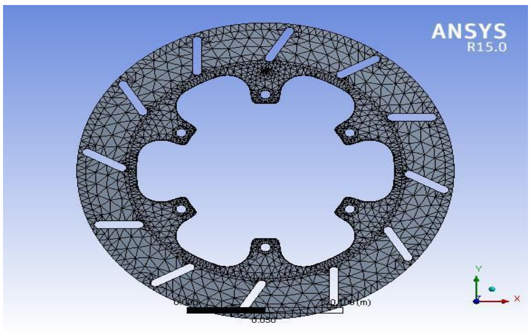

Figure 3.3: Meshing of Model 2

Sizing	
Relevance Center	Fine
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	8.5106e-005 m
Statistics	
Nodes	52633
Elements	28834

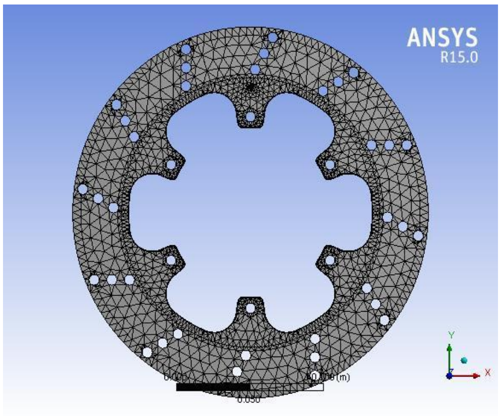

Figure 3.4: Meshing of Model 3

Sizing	
Relevance Center	Fine
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	8.5106e-005 m
Statistics	
Nodes	54667
Elements	30092
Mesh Metric	None

4. Results and Discussion

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 behavior of disk brake is obtained in the repeated brake condition. The computational results are presented for the distributions of pressure and temperature on each friction surface between the contacting bodies.

4.1. Stainless Steel

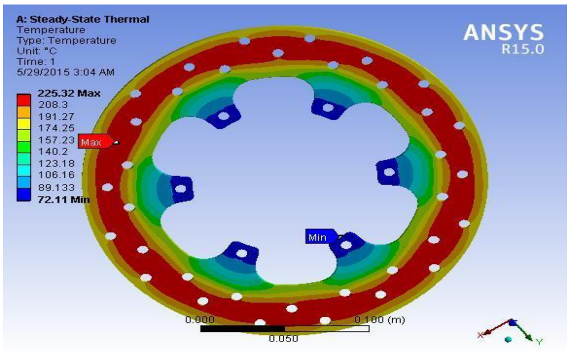

Figure 4.1.: Temperature distribution plot for SS Model No. 1

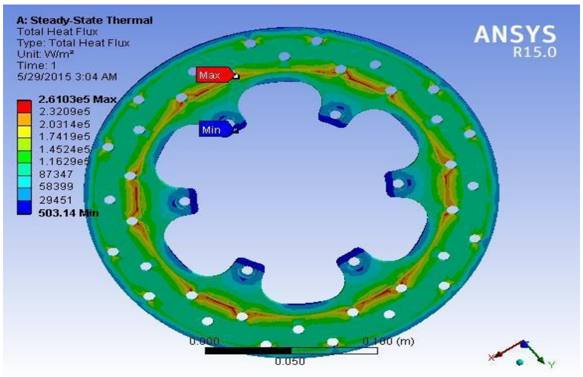

Figure 4.2: Heat flux plot SS Model No. 1

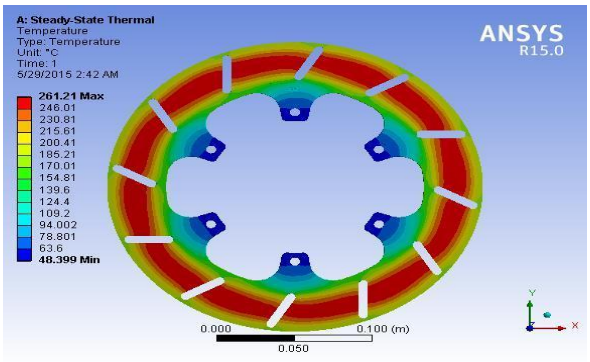

Figure 4.3: Temperature distribution plot for SS Model No. 2

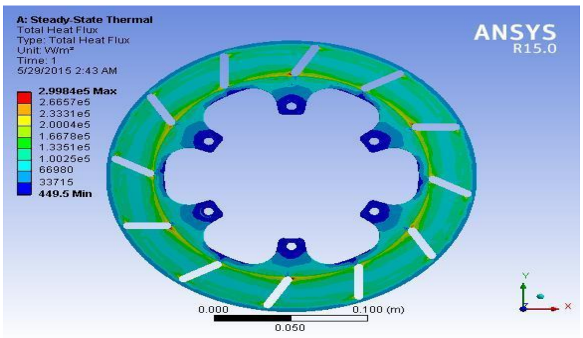

Figure 4.4: Heat flux plot SS Model No. 2

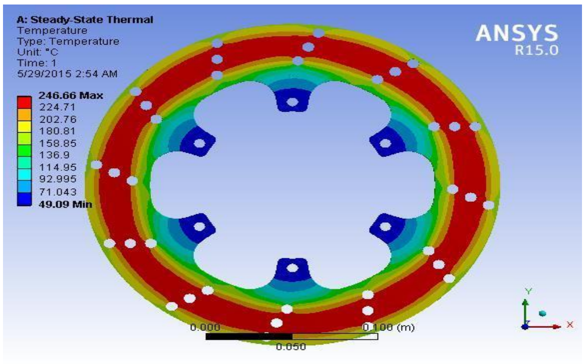

Figure 4.5: Temperature distribution plot for SS Model No. 3

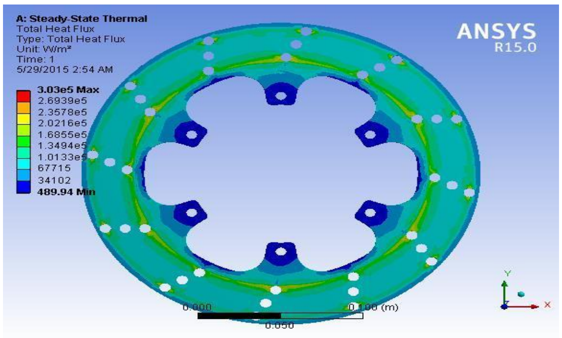

Figure 4.6: Heat flux plot SS Model No. 3 for SS Model No.3

4.2. Cast iron

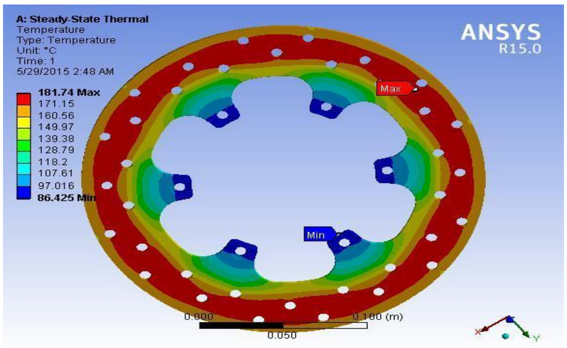

Figure 4.7.: Temperature distribution plot for CI Model No. 1

Model No. 1

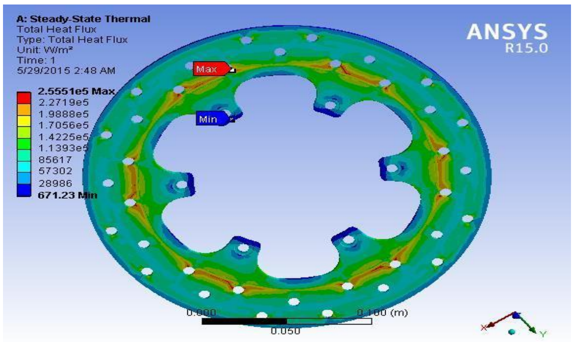

Figure 4.8: Heat flux plot CI Model No. 1

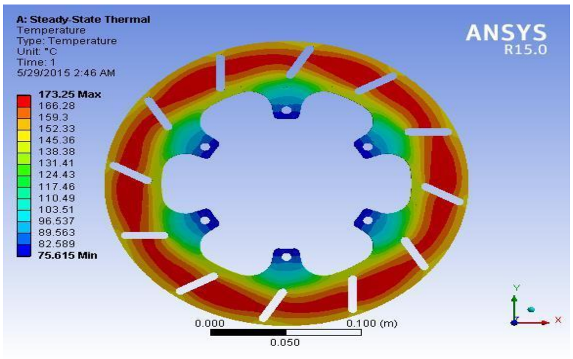

Figure 4.9.: Temperature distribution plot for CI Model No. 2

Model No. 2

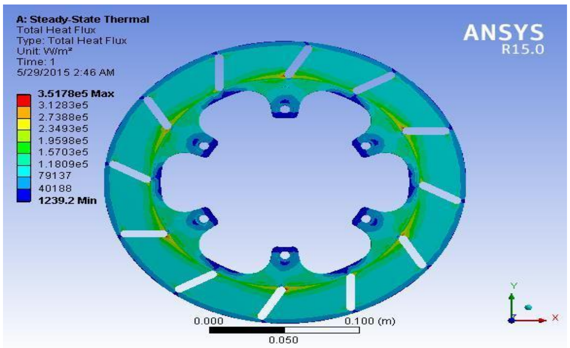

Figure 4.10: Heat flux plot CI Model No. 2

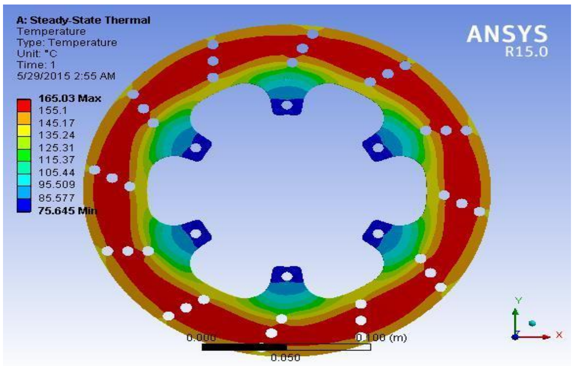

Figure 4.11: Temperature distribution plot for CI Model No. 3

Model No. 3

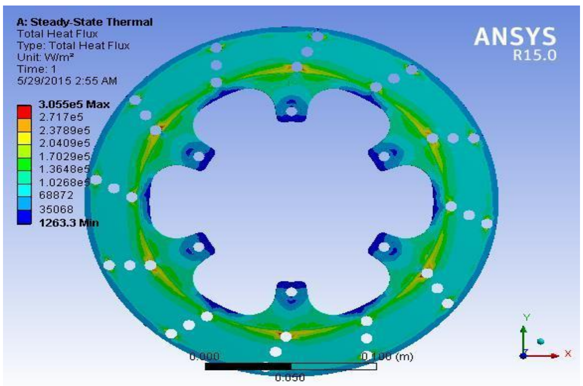

Figure 4.12: Heat flux plot CI Model No. 3

From the figures, given above, we can summarize the results in the following manner: -

Table 4.1 Maximum and minimum Temperature Distribution

Results	Temperature Distribution (°C)			
Material	Stainless Steel		Cast Iron	
	Min	Max	Min	Max
Model No.1	72.11	225.32	86.425	181.74
Model No.2	48.399	261.21	75.615	173.25
Model No.3	49.09	246.66	75.645	165.03

Table 4.2 Maximum and minimum Total Heat Flux

Results	**Total Heat Flux** (W/m^2)			
Material	Stainless Steel		Cast Iron	
	Min	Max	Min	Max
Model No.1	503.15	2.61×10^5	671.24	2.55×10^5
Model No.2	449.50	2.98×10^5	1249.2	3.51×10^5
Model No.3	489.95	3.05×10^5	1263.3	3.05×10^5

CONCLUSION

From our study of various design patterns for different materials we have observed that the maximum temperature rise for cast iron is considerably lesser as compared to stainless steel and thus on the basic of thermal analysis, cast iron is the best preferable material for manufacturing disc brake. However cast iron disc brake suffers a drawback of getting corroded when it comes in contact with moisture and hence it cannot be used in two wheeler and thus we prefer stainless steel.

Heat dissipation from disc brake also depends on the type of design pattern used. The different design patterns studied are:-

- **A.** Model No. 1- With more no. of circular holes
- **B.** Model No. 2- With kidney shaped holes
- **C.** Model No. 3- With less no. of circular holes

Heat transfer rate increases with number of cuts in the disc. This is because large area is exposed to air which makes more heat transfer through conduction and convection. But increase in number and size of cuts decreases the strength of disc. Among the above models best heat dissipation is observed for model 1 consisting large number of holes and made of stainless steel.

From comparison between the two materials for the Thermal values and material properties obtained from the Thermal analysis, low thermal gradient material is preferred. Hence best suitable design, low thermal gradient material Grey cast iron is preferred for the Disc Brakes for better performance. Further investigation can be carried out by experimenting material mechanical properties of more suitable materials with varying cross-section & optimal number of holes to predict the effective disc brake materials.

REFERENCES

[1] M. L. Aggarwal, Atul Sharma, "Deflection and stress Analysis of Brake Disc using Finite Element Method", Oct 19-20, 2012.

[2] A. Belhocine, C.-D. Cho, M. Nouby, Y. B.Y, A. R. Abu Bakar "Thermal analysis of both ventilated and full disc brake rotorswith frictional heat generation" Applied and Computational Mechanics 8 (2014) 5–24.

[3] C. Baron, T. Ingrassia, V. Nigrelli, V. Ricotta, Università degli Studi di Palermo, Dipartimento di Ingegneria Chimica, Gestionale, Informatica, Meccanica – 90128 Palermo, Italy "Thermal stress analysis of different full and ventilated disc brake", 34 (2015) 608-621; DOI: 10.3221/IGF-ESIS.34.67.

[4] SubhasisSarkar (Dr.) Pravin P. Rathod A. J. Modi "Research Paper on Modeling and Simulation of Disc Brake to Analyse Temperature Distribution using FEA" IJSRD - International Journal for Scientific Research & Development| Vol. 2, Issue 03, 2014.