© 2014 IJEDR | Volume 2, Issue 2 | ISSN: 2321-9939

Structural and Thermal Analysis of Brake Disc

Viraj Parab[1], Kunal Naik[2], Prof A. D. Dhale[3]

[1,2]Students of Mechanical Engineering, [3]Associate Professor
Shivajirao S. Jondhale College of Engineering, Dombivli (E), University of Mumbai.
[1]viraj_cr7@yahoo.co.in, [2]kpnaik85@gmail.com

***Abstract* - Disc(Rotor) brakes are exposed to large thermal stresses during routine braking and extraordinary thermal stresses during hard braking. The aim of the project is to design, model a disc. Modeling is done using catia. Structural and Thermal analysis is to be done on the disc brakes using three materials Stainless Steel and Cast iron & carbon carbon composite. Structural analysis is done on the disc brake to validate the strength of the disc brake and thermal analysis is done to analyze the thermal properties. Comparison can be done for deformation, stresses, temperature etc. form the three materials to check which material is best. Catia is a 3d modeling software widely used in the design process. ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements.**

***KeyWords* - rotor brake, catia, FEA, thermal analysis, static analysis.**

I. INTRODUCTION

Computer-aided design (CAD), also known as computer-aided design and drafting (CADD).CAD describes the process of drafting with a computer, and the output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions. CADD describes the purpose of streamlining design processes, drafting, documentation, and manufacturing processes.CAD is mainly used for detailed engineering of 3D models and/or 2D drawings of physical components, but it is also used throughout the engineering process from conceptual design and layout of products, through strength and dynamic analysis of assemblies to definition of manufacturing methods of components. It can also be used to design objects.

1.1 Introduction to Catia

Catia is the standard in 3D product design, featuring industry-leading productivity tools that promote best practices in design while ensuring compliance with your industry and company standards.

1.2 Introduction to FEA

FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition. FEA uses a complex system of points called nodes which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. In practice, a finite element analysis usually consists of three principal steps.

- **Preprocessing** - The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical "preprocessor" to assist in this rather tedious chore. Some of these preprocessors can overlay a mesh on a preexisting CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

- **Analysis** - The dataset prepared by the preprocessor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations [K][U]=[F]
where u and f are the displacements and externally applied forces at the nodal points. The formation of the K matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types. One of FEA's principal advantages is that many problem types can be addressed with the same code, merely by specifying the appropriate element types from the library.

- **Postprocessing -** In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. Typical postprocessor display overlays colored contours representing stress levels on the model, showing a full field picture similar to that of photo elastic or moiré experimental results.

1.3 Introduction to ANSYS

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated, or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

II. THE MODEL DESIGN AND BRAKING CONDITIONS

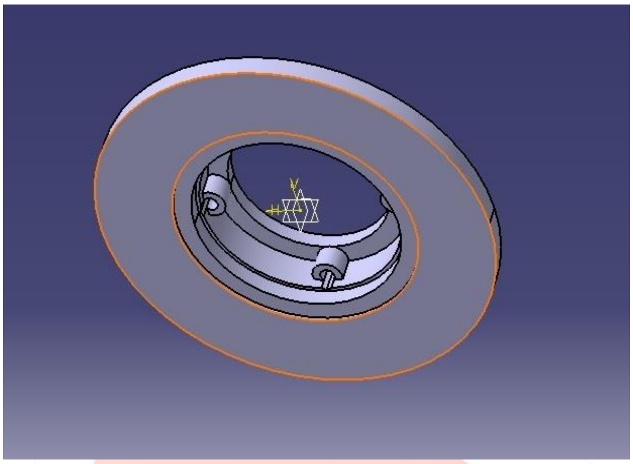

Fig.1 Isometric View of Rotor Disc

A three-dimensional solid with shape and dimensions is modeled in catia. The disc has a diameter of 0.220m and the thickness of the disc on the contact surfaces is 0.011m.

III. MODELING IN ANSYS

The first stage is to create the model which contains the fields to be studied in Ansys Workbench. In our case, we took only one quarter of the disc, then we defined the field of the air surrounding this disc. ANSYS workbench prepared various surfaces for the two fields in order to facilitate the mesh on which it will export the results to CFX using the command "Output to cfx". After obtaining the model on CFX Pre and specifying the boundary conditions, we must define the physical values on CFX to start calculation.

The disc is related to four adiabatic surfaces and two surfaces of symmetry in the fluid domain whose ambient air temperature is taken as equal to 280°C . An unsteady-state analysis is necessary.

(a) Physical Model

In this step, one declares all of the physical characteristics of the fluid and the solid. After the meshing, all the parameters of the different models are defined in order to start the analysis.

(b) Definition of the Domains

Initially, one validates and activates the elaborated models in the option "Thermal Energy" and performs the calculation of heat transfer in "Heat Transfer".

(c) Cast Iron Materials

We introduce into the library the physical properties of used materials. In this study, we selected the cast iron material with their thermal conductivities.

(d) Definition of the Boundary Conditions

The first step is to select the inlet and outlet faces of the heat flux. These options are found in the insertion menu “Boundary Conditions” in the CFX Pre. One has to select the options “Wall” and "Symmetry" because there is the possibility of adjusting a certain number of parameters in the boundary conditions, such as flux entering the disc.

(e) Application of the Interfaces Domains

The areas of interfaces are commonly used to create connection or linkage areas. Surfaces located between the interactions regions are reported as solid-fluid interface.

(f) Temporary Condition

Since the goal of this study is to determine the temperature field in a disc brake during the braking phase of a vehicle of average class, we take the following temporal conditions:

- Braking time = 20 [s]
- Increment time = 0.01 [s]
- Initial time = 0 [s]

Before starting the calculation and the analysis with ANSYS, it is ensured that the model does not contain any error.

(g) Launch of the Calculation

After verification of the model and boundary conditions, we run the calculation by opening the menu "File" and clicking on "Write solver file". The values of the coefficient of exchange will be taken at average values calculated by the minimal and maximum values obtained using ANSYS as it is indicated.

2.3 Meshing of the Disc

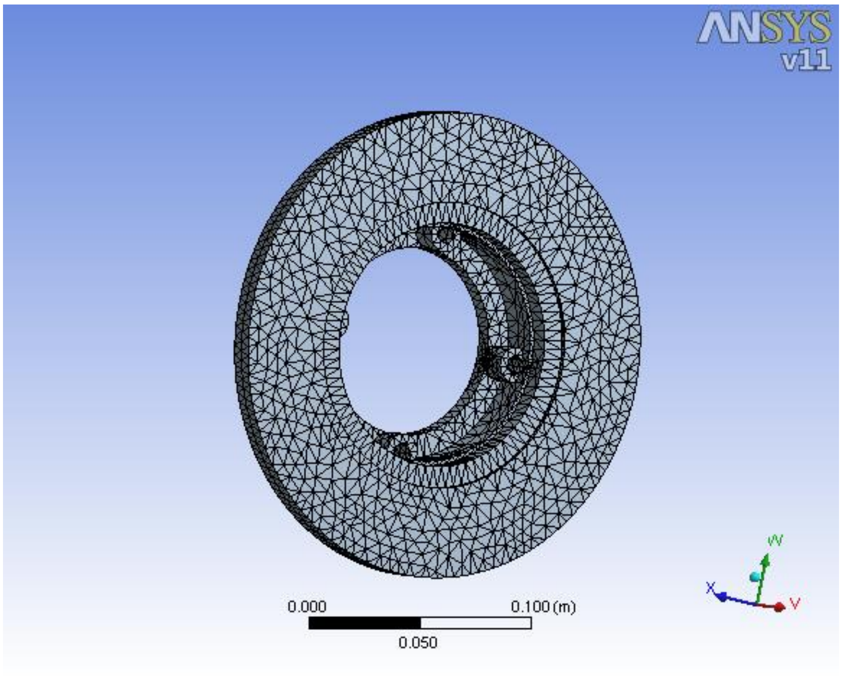

Fig.2 Meshed Model of Rotor Disc

The elements used for the meshing of the full and ventilated disc are tetrahedral three-dimensional elements with nodes (iso-parametric). In this simulation, the meshing was refined in the contact zone (disc-pad). This is important because in this zone, the temperature varies significantly. Indeed, in this strongly deformed zone, the Thermo mechanical gradients are very high. This is why an accurate account of the contact conditions involve the use of a refined mesh. Three meshes have been tested automatically using an option called convergence in ANSYS Workbench Multi physics.

IV. PROPERTIES

Table 1 Properties of Materials

Properties	Cast Iron	Stainless Steel	Carbon Carbon Composite
Density (kg/m^3)	7100	7750	1800
Young's modulus (GPa)	125	190	95
Poison's ratio	0.25	0.3	0.31
Thermal conductivity (w/m-k)	54.5	26	40
Specific heat (j/kg-k)	586	500	755
Coefficient of friction	0.2	0.22	0.3

V. RESULTS AND DISCUSSION

(a) Cast Iron

Following fig. shows results of total deformation, equivalent stress and temperature distribution of cast iron after analysis.

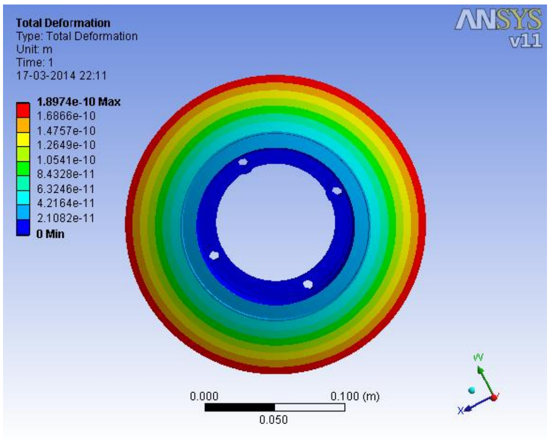

Fig.3 Total Deformation

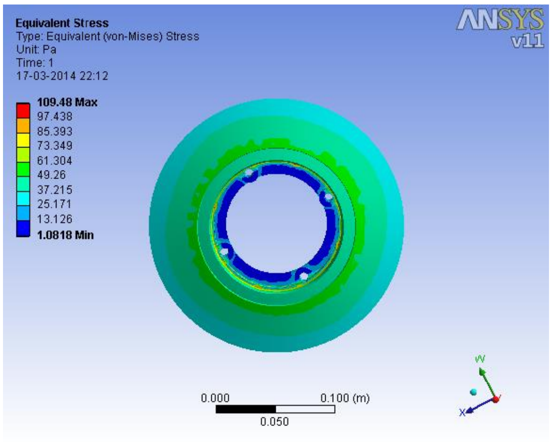

Fig.4 Equivalent Stress

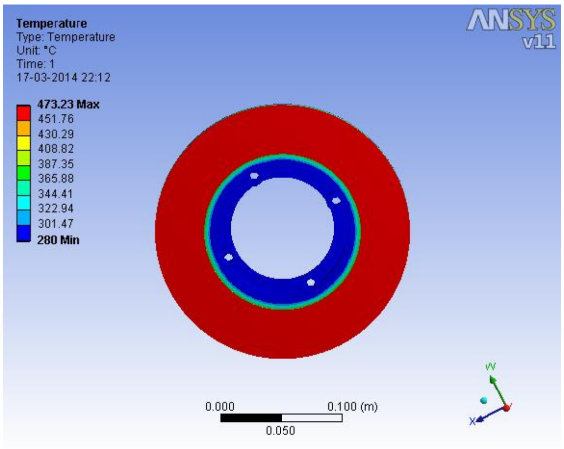

Fig.5 Temperature Distribution

(b) Stainless Steel

Following fig. shows results of total deformation, equivalent stress and temperature distribution of stainless steel after analysis.

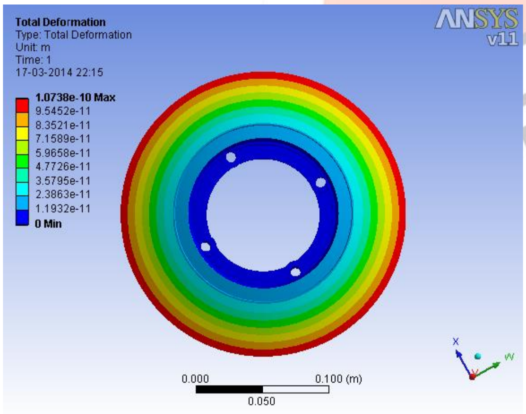

Fig.6 Total Deformation

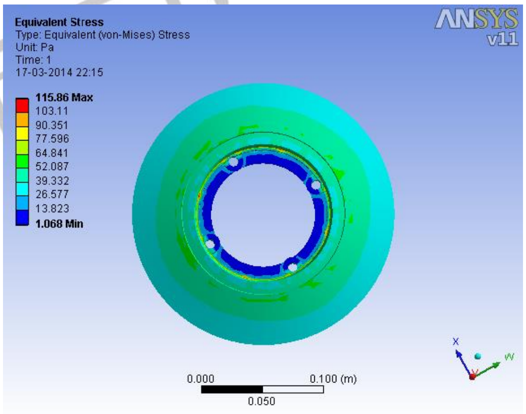

Fig.7 Equivalent Stress

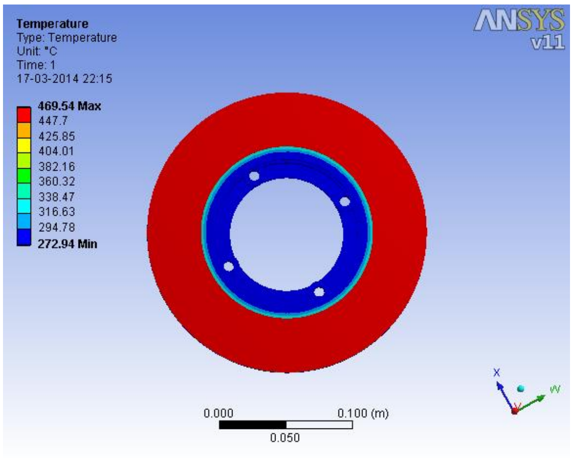

Fig.8 Temperature Distribution

(c)Carbon Carbon Composite

Following fig. shows results of total deformation, equivalent stress and temperature distribution of carbon carbon composite after analysis.

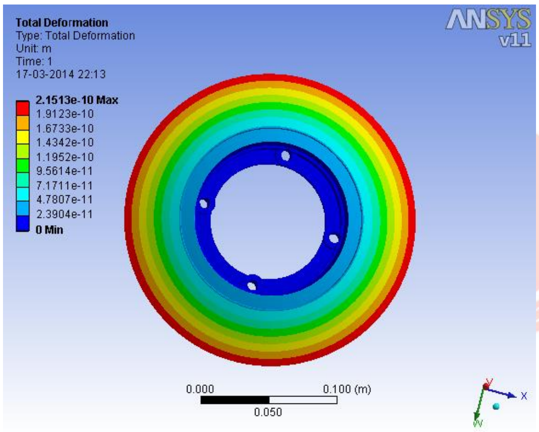

Fig.9 Total Deformation

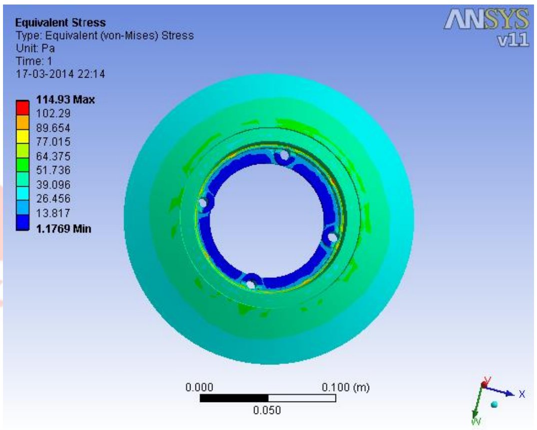

Fig.10 Equivalent Stress

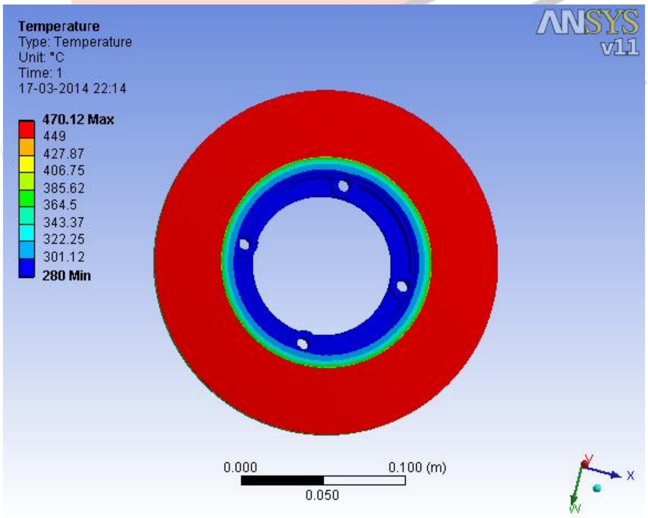

Fig.11 Temperature Distribution

Table no. 2 Results of Different Materials

Results	Cast iron		Stainless steel		Carbon carbon composite	
	max	Min	Max	min	max	min
Total deformation(M)	1.8974e-10	0	1.0738e-10	0	2.1513e-10	0
Equivalent stress(Pa)	109.48	1.0818	115.86	1.068	114.93	1.769
Temperature distribution(˙c)	473.23	280	469.54	272.92	470.12	280

VI. CONCLUSION

The transient thermo analysis of Disc brakes in brake applications has been per-formed. It is observed that the stainless steel can provide better brake performance than others from deformation point of view whereas cast iron provides better performance from stress point of view. The present study can provide a useful de-sign tool and improve the brake performance of Disc brake system. The values obtained from the analysis are less than their allowable values. Hence the brake Disc design is safe based on the strength and rigidity criteria.

REFERENCES

[1] D. Majcherczak, P. Dufrenoy and M. Nait -Abdelaziz, Thermal simulation of a dry sliding contact using a multiscale model −Application to the braking problem, Thermal stresses 2001, Osaka (Japan), pp. 437-440, Juin, 2001.

[2] F. Colin, A Floquet and D. Play; Thermal contact simulation in 2-D and 3-D mechanisms, ASME Journal of Tribology, 110 (1988) 247-252.

[3] W. S. Chung, S. P. Jung and T. W. Park, Numerical analysis method to estimate thermal deformation of a ventilated disc for automotives, Journal of Mechanical Science and Technology, 24 (11) (2010) 2189-2195.

[4] F. Talati and S. Jalalifar, Analysis of heat conduction in a disc brake system, Heat Mass Transfer, 45 (2009) 1047-1059.

[5] J. T. Kim and B. J. Baek, A numerical study of thermal performace in ventilated disk brake, Journal of the Korean Society of Tribologists & Lubrication Engineers, 17 (5) (2001) 358-364.

[6] Y. Choi, J. W. Choi, H. M. Kim and Y. W. Seo, Thermal dissipation performance of the ventilated brake disc having helical grooved vent, Journal of the Korean Society of Precision Engineering, 21 (3) (2004) 117-123.

[7] S. M. Kim, A study on thermal analysis in ventilated brake by FEM, Journal of the Korean Society of Machine Tool.

[8] International Journal of Scientific & Engineering Research Volume 2 , Issue 8, August-2011.