iJETRM
International Journal of Engineering Technology Research & Management

DESIGN AND ANALYSIS OF DISC BRAKE

R.Syam Sudhakar Rao*[1]
[*1]Assosicate Professor (Department of Mechanical Engineering, GNITC, Hyderabad, India)
S.SaiUpendra[2], P.Venu[3],R.SreeAravind[4]
[2,3,4] UG Scholar (Department of Mechanical Engineering, GNITC, Hyderabad, India)

ABSTRACT
Brake rotors (disc) are the key pieces of a vehicle that play an active role in safety and performance of the system. Materials which are used for the manufacturing of brakes may be different and they have different in temperature distribution. Number of times using the brake leads to heat generation during braking event. Hence, disk brake undergoes breakage due to high temperature distribution.
In this project, disc brake model is done by CERO software and the transient Analysis will be determined by using ANSYS. In further application Computational fluid thermodynamics (CFTD) will be applied to see the transfer of heat on application of brake while the brake is rotating. By using CFTD method, the temperature distribution based on rotation speed of rotary brake can be observed. With this reference, the alloy which is suitable based on the heat value rate , might be known.

Keywords:
Disc Brake, Thermal Analysis, Modal Analysis, Ansys

INTRODUCTION

The disc brake is 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 callipers. 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. This is connected to the wheel and/or the axle. To stop the wheel, 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. 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.
Disc-style brakes development and use began in England in the 1890s. The first calliper-type automobile disc brake was patented by Frederick William Lanchester in his Birmingham, UK factory in 1902 and used successfully on Lanchester cars. Compared to drum brakes, disc brakes offer better stopping performance, because the disc is more readily cooled. A disc brake consists of a cast iron disc bolted to the wheel hub and astationary housing called calliper.

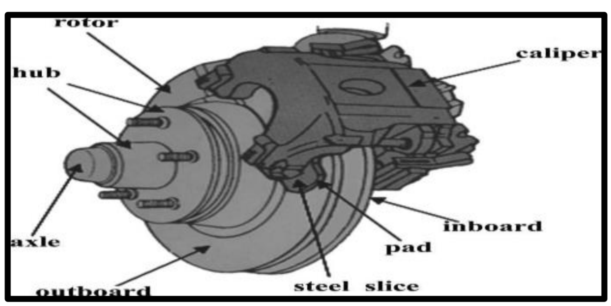

Fig. 1:Disc Brake

iJETRM
International Journal of Engineering Technology Research & Management

I. PROBLEM OCCURRED IN DISC BRAKE

Brake rotors (disc) are the key pieces of a vehicle that play an active role in safety and performance of the system.

Materials which are used for the manufacturing of brakes may be different and they have different in temperature distribution.

Number of times using the brake leads to heat generation during braking event. Hence, disk brake undergoes breakage due to high temperature distribution

Discs are made up mainly gray cast iron, so discs are damaged in one of three ways: scarring, cracking, warping or excessive rusting. Service shops will sometimes respond to any disc problem by changing out the discs entirely. This is done mainly where the cost of a new disc may actually be lower than the cost of workers to resurface the original disc.

II. SOLID WORKS

PRODUCER:

A.EXTRUDE

1.Click New (Standard toolbar) and open a new part
2. Click Extruded Boss/Base (Features toolbar) and select the Front plane.
3.Sketch a corner circle beginning at the origin.
4. Click Smart Dimension (Dimensions/Relations toolbar) and dimension the circle to 174.54mm .
5. Click Exit Sketch (Sketch toolbar) to exit the sketch.
6. Under Direction1:
7. Set End Condition to Blind.
8. Set Depth to 4.5.
9. Click to create the extrusion.

B.EXTRUDE CUT

1. Click New (Standard toolbar) and open a new part.
2. Click Extruded Boss/Base (Features toolbar) and select the Front plane.
3. Sketch a corner circle beginning at the origin.
4. Click Smart Dimension (Dimensions/Relations toolbar) and dimension the circle to 99.54 mm .
5. Click Exit Sketch (Sketch toolbar) to exit the sketch.

C. Gap Hole For Air Flow

1. On the Features toolbar, expand the Linear Pattern fly out toolbar and click Circular Pattern.
2. In the Property Manager, under Parameters:
3. Select the edge in the center of the part for Pattern Axis.
4. Select Equal spacing to pattern the number of instances uniformly around the axis within 360°. 25.
5. Set Number of Instances to 7.
6. Click in Features to Pattern.
7. In the fly out Feature Manager design tree in the graphics area, select the last features (Cut-Extrude2).

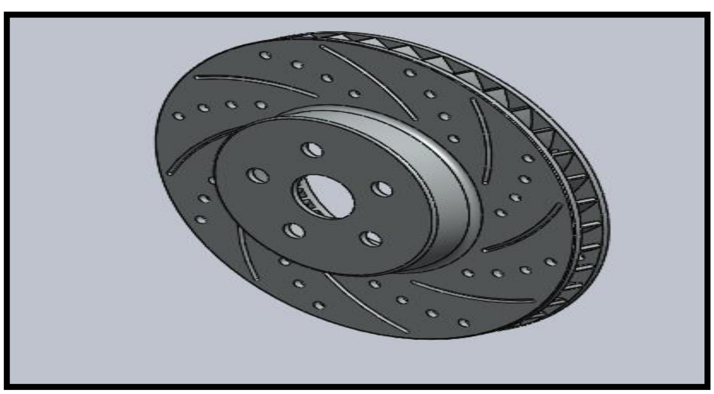

Fig2: Soild Model

iJETRM
International Journal of Engineering Technology Research & Management

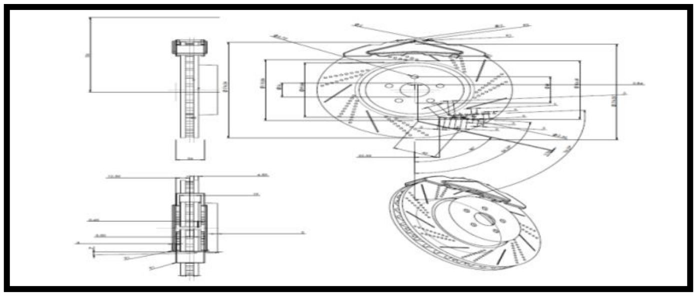

Fig3:Drafting

III. ANSYS

MODEL ANALYSIS

A. Generating The Mesh For The Model.

1. Double click on model cell will enter into mechanical window. Also, you will notice that in the Outline window, the Mesh node is displayed in the Tree Outline with a yellow thunderbolt attached to it.
2. Click on Mesh in the Tree Outline; the Details of “Mesh” window is displayed.

B. Setting The Boundary Conditions

After the mesh is generated, you need to set the boundary conditions under which the analysis is to be performed.

1. Right—click on Modal node in the Tree Outline and then choose Insert >Fixed Support from the shortcut menu displayed; Fixed Support with a question symbol is added under the Modal node in the Tree Outline. Also, the Details of “Fixed Support window is displayed.
2. In the Details of “Fixed Support” window, click on the Geometry cell to display the Apply and Cancel buttons, if not already displayed.

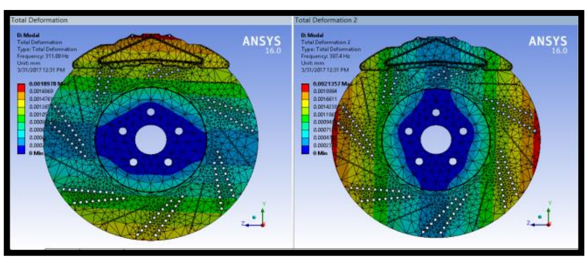

Fig 4 :Mesh for MA

Results	Minim...	Maximum	Units	Reported Frequency (Hz)
Total Deformation	0.	1.8978e-003	mm	311.08
Total Deformation 2	0.	2.1357e-003	mm	387.4
Total Deformation 3	0.	2.1392e-003	mm	800.
Total Deformation 4	0.	1.7159e-003	mm	1550.8
Total Deformation 5	0.	2.5111e-003	mm	1909.5
Total Deformation 6	0.	2.2572e-003	mm	1950.6

Fig 5 :Final Result

STATIC STRUCTURAL ANALYSIS

A. Generating The Mesh

1. After the model is created in the Design Modeler window, you need to generate the mesh for the model in the Mechanical window.
2. In the Project Schematic window, double-click on the Model cell in the static structural analysis system; the Mechanical window is displayed.
3. Select Mesh in the Tree Outline to display the Details of "Mesh" window.
4. In the Details of "Mesh" window, expand the Sizing node, if it is not already expanded. Also, notice that Default is displayed in the Element Size edit box.
5. The Element Size edit box is used to specify the size of an element. The element size specified in this edit box is

iJETRM
International Journal of Engineering Technology Research & Management

according to the size of the geometry. However, this edit box will not be visible when the
6. On: Proximity and
7. On: Proximity and Curvature .

8. Options are selected from the Use Advanced Size Function drop-down list. When Default is displayed in the Element Size edit box, it indicates that a default value, based on the size of the geometry, is already specified by the software.

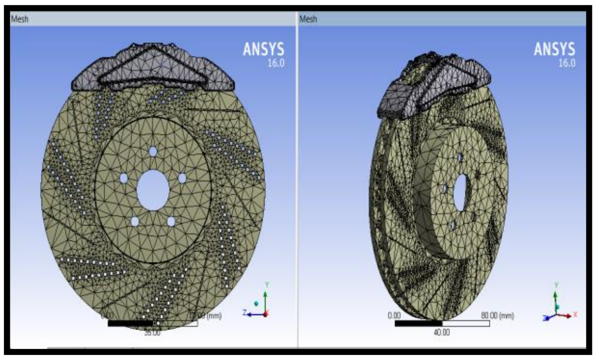

Fig 6 :Mesh for Structural analsys

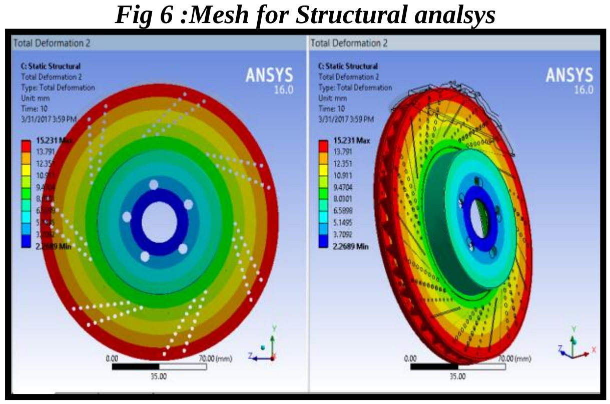

Fig 7 :Final Analsys

THERMAL ANALYSIS

1. Select the Steady-State Thermal node in the Tree Outline to display the Environment contextual toolbar.
2. From the Environment contextual toolbar, choose the Temperature tool; Temperature is added under the Steady-State Thermal node. Also, the Details of “Temperature” window is displayed.
3. In the Details of “Temperature” window, select the Geometry selection box to display the Apply and Cancel
4. Select the front face of the model
5. Choose the Apply button from the Geometry selection box in the Details of “Temperature” window; the face is selected for applying the Temperature boundary condition.
6. In the Details of “Temperature” window, specify 80 in the Magnitude edit box.
7. Select Temperature from the Steady-State Thermal node in the Tree Outline; the Graph and Tabular Data windows are displayed, as shown in Figures.

iJETRM
International Journal of Engineering Technology Research & Management

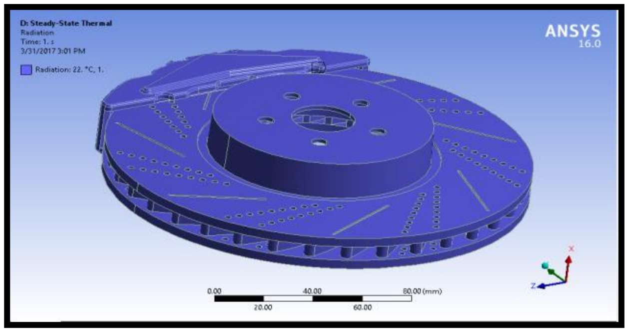

Fig 8 :Applying boundary condition

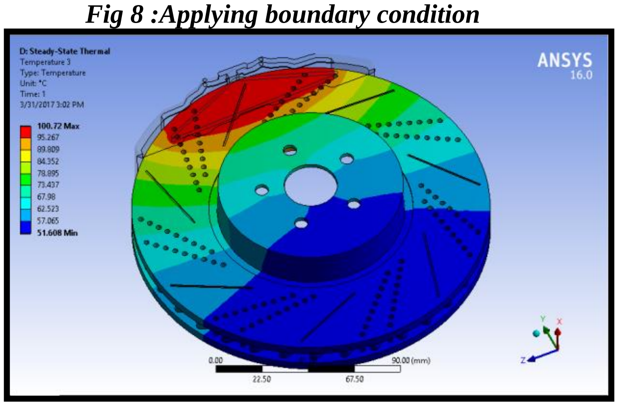

Fig 9:Thermal analsys

In the Details of “Total Heat Flux” window, select the Geometry selection box to display the Apply and Cancel buttons.

8. Choose the Body tool from the Select toolbar and then select the model,
9. Choose the Apply button from the Geometry selection box in the Details of “Total Heat Flux” window; 1 Body is displayed in the Geometry selection box.
10. Similarly, choose Directional Heat Flux from the Thermal fly out in the Solution contextual toolbar;
11. Choose the Solve tool from the Standard toolbar; the ANSYS Workbench Solution Status window is displayed. Notice that a tick mark is placed before Total Heat Flux and Directional Heat Flux in the Tree Outline,
12. Select Total Heat Flux from the Solution node in the Tree Outline; the corresponding contours displaying the distribution of heat flux are displayed in the Graphics screen.
13. Similarly, select Directional Heat Flux from the Solution node in the Tree Outline; the corresponding contours displaying the distribution of heat flux along the X axis are displayed in the Graphics screen,
14. The table given next displays the results obtained for the Total Heat Flux and
15. Directional Heat Flux.

16. It is obvious from the above results that the Total Heat Flux is maximum in the region where the model is marked red and minimum where it is marked blue. Each color contour depicts a value and can be seen in the Legend in the Graphics screen.
17. Exit the Mechanical window to display the Workbench window.

iJETRM
International Journal of Engineering Technology Research & Management

Results	Minimum	Maximum	Units	Time (s)
Temperature	51.608	100.72	°C	1.
Total Heat Flux	4.2174e-014	0.31535	W/mm²	1.
Temperature 2	96.661	100.08	°C	1.
Temperature 3	51.608	100.72	°C	1.
Total Heat Flux 2	3.0629e-005	1.9936e-002	W/mm²	1.
Total Heat Flux 3	8.1257e-005	0.31535	W/mm²	1.

Fig 10: Resultant Summary From Ansys Workbench

ACKNOWLEDGMENT

We would like to say sincere thanks to our guide **Mr. R. SYAM SUDHAKAR RAO,** Associate Professor, Department of Mechanical Engineering for Coordinating Projects for the suggestions and constant guidance in every stage of the project.

We also like to thank all our lecturers helping us in every possible way. On a more personal note, we thank our beloved parents and friends for their moral support during our project.

REFERENCES

[1] Dr. Ramesha, Santhosh Kumar and Bharath Shekar, "Temperature Distribution Analysis of Aluminum Composite and Cast Iron Brake Drum Using Ansys", 'International Journal of Emerging trends in Engineering and Development', 2012, Vol. 3, Issn 2249-6149, pp 281- 292.

[2] Ameer Shaik and Lakshmi Srinivas,"Structural and Thermal Analysis of Disc Brake Without Cross-drilled Rotor Of Race Car", 'International Journal of Advanced Engineering Research and Studies',2012 , Vol. I, Issn 2249-8974, pp 39-43

[3] Chogdu Cho and Sooick Ahn, "Thermo-Elastic Analysis for Chattering Phenomenon of Automotive Disk Brake", 'KSME International Journal', 2001,Vol-15, pp 569-579.

[4] Guru Murthy Nathi, K. Gowtham and Satish Reddy, "Coupled Structual / Thermal Analysis of Disc Brake", IJRET 2012, Vol.1, pp.539-553.

[5] V. M. Thilak , R. Krishnara Deepan & R.Palani ,"Transient Thermal and Structural Analysis of the Rotor Disc of Disc Brake ", International Journal of Scientific & Engineering Research Volume 2, Issue 8, August-2011 Issn 2229-551..

[6] Dr. Ramesha, Santhosh Kumar and Bharath Shekar, "Temperature Distribution Analysis of Aluminum Composite and Cast Iron Brake Drum Using Ansys", 'International Journal of Emerging trends in Engineering and Development', 2012, Vol. 3, Issn 2249-6149, pp 281- 292.

[7]Ameer Shaik and Lakshmi Srinivas,"Structural and Thermal Analysis of Disc Brake Without Cross-drilled Rotor Of Race Car", 'International Journal of Advanced Engineering Research and Studies', 2012 , Vol. I, Issn 2249-8974, pp 39-43