Ventilation Pattern Analysis Using Ansys CFD Fluent For Brake Disc

Govind Nanasaheb Patil[1], Prof.Mr.P.H.Jain[2]
[1]*M.E. (Mechanical Design) Student,* [2] *Project Guide*
Mechanical Engineering Department, T.P.C.T College of Engineering Osmanabad, Maharashtra, India

***Abstract*— Most of the disc brake failures concerning the rotor are mainly due to the overheating. Therefore, it is paramount to enhance the heat dissipation of the ventilated disc brake rotor so that it lasts longer and also functions efficiently. Over the past few years many researchers have come up with innovative designs to address this concern by the analysis of both flow and heat transfer characteristics inside a ventilated brake disc rotor. There are other factors such as weight, thickness of the rotor and sometimes even the number of blades in a rotor that keep changing with the requirement of the car manufacturer. So, there is a huge need within the disc brake manufacturing community for sensitivity analysis data so that heat dissipation and temperature uniformity can be maximized in spite of having some parameters fixed. The main aim of this CFD analysis is to study and predict the effect of various design parameters on the aero-thermal performance of a disc brake rotor.A commercial vehicle of 9.6T is considered for calculation.**

I. INTRODUCTION

Disc type brake development and its use began in England in the 1890s. Disc brakes were patented by Frederick William Lanchester in 1902 but the commercial use of these brakes started in the early 1950s [1].The brake disc is the rotating part of a wheel's disc brake assembly, against which the brake pads are applied. 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 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 have holes or slots cut through the disc for better heat dissipation, to aid surface-water dispersal, to reduce noise, to reduce mass.

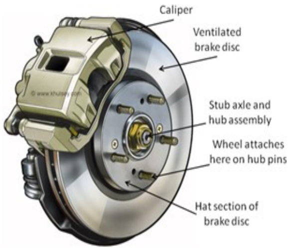


Fig. 1 Disc brake assembly

II. AIMS & OBJETIVES OF CFD ANALYSIS

The main aim of this CFD analysis is to study & predict the effect of various design parameters on the aero-thermal performance of a disc brake rotor. The various design parameters include:-

- Blade geometry.
- Speed of brake disc in RPM.
- Brake disc temperature.
- Number of blades in the rotor.
- Blade thickness.
- Ventilation pattern thickness.

These aims are accomplished by performing CFD analysis of the air flow through the rotor passage using ANSYS FLUENT and by studying the following aspects of the flow which affect the aero-thermal performance of the rotor which are as follows:-

- Mass flow rate through the rotor passage.
- Flow streamline of air flow through the ventilation passage.

The commercial CFD package, FLUENT, is used to perform analysis of the disc brake rotor passage flow. The semi-automatic geometric model is created using the package CATIA V5 and the mesh for the model is done using the software ANSYS ICEM CFD 14.5 and finally the post-processing of the results is thoroughly done using FLUENT and CFD POST 14.5 and is discussed and presented in detail. The analysis is done on a computational domain and reference taken from various journal papers for the periodic conditions to simulate two types of blade patterns.

Table 1: Vehicle parameters

Sr. No.	Parameters	Values	units
1	Gross vehicle weight	9.6	ton
2	Maximum speed	100	km/h
3	Wheel base	5.22	m
4	Tire specification	215/75R 17.5	
5	Tire dynamic radius	0.445	m
6	Centre of gravity height	1.2	m
7	Distance of CG from front axle	2.45	m
8	Distance of CG from rear axle	2.77	m

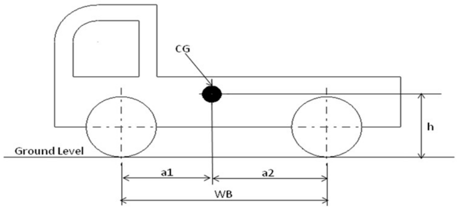


Fig. 2 Vehicle dimensions.

III. METHODOLOGY FOR CFD ANALYSIS IN ANSYS FLUENT

3.1.1. Model creation

The disc brake rotor is rotationally periodic with blades and passages at equal angular spacing. This reduces the computational cost significantly as the analysis can be done only for a single blade (in case of same pattern) and pair of blades (in case of different patterns). The model was created in CATIA V5. The actual brake disc consists of two rubbing surfaces separated by the blades. One of the rubbing surfaces (outboard surface) is mounted onto the wheel with the help of 6 mounting holes. Simulating the full geometry is not justified as it is requires a lot of computational effort. On the overall performance of a brake disc, the following simplifications and stages were performed to model the original brake disc as shown in the figure 5.1 and 5.2 for model A and model B respectively.

a. Simplifications

- The outboard side of the brake rotor is simulated as a flat plate because the wheel is mounted on outboard surface and the entry of air can be neglected.
- The bolts and the balancing clips are neglected.
- The grooves and balancing cut on the rubbing surfaces are also neglected.
- The leading and trailing edges match the outer and inner radii of the brake disc rubbing surfaces for the most part of the project although the lips are included later on to get a better analysis.
- Entry area of the disc is extended to stabilize the air flow.

b. Import and clean up the geometry

c. CAD model repair and CFX-Mesh generation

d. Meshing process in CFX-Mesh

- Define regions.
- Define mesh attributes
- Generate surface mesh.
- Generate volume mesh.
- Generate prism mesh near wall region.

Before generating the mesh, we should confirm that the geometry is free of any flaws that would inhibit optimal mesh creation.Figure 5.1 and 5.2 shows the CAD model repaired for ventilation pattern A and pattern B respectively. The CAD model repair methodology is implemented to create closed surface for mesh generation. All open surfaces which were arrived due to data conversion from .stp files were closed to avoid the improper meshing.

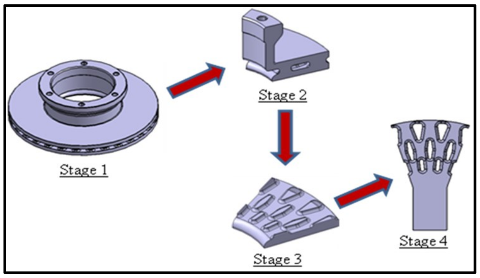


Fig.3 Stages for modelling of ventilation pattern A.

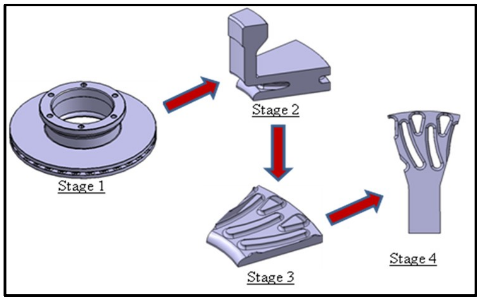


Fig.4 Stages for modeling of ventilation pattern B.

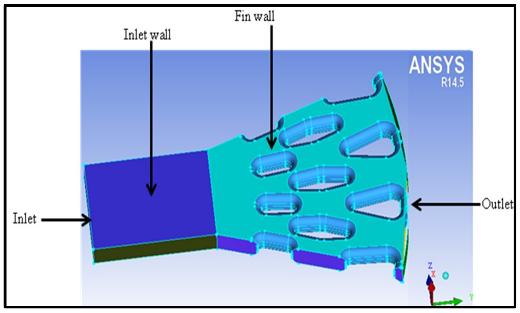


Fig5.1CAD model repair for ventilation pattern A.

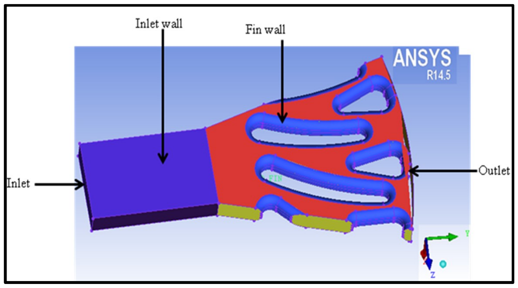


Fig5.2 CAD model repair for ventilation pattern B.

CFX-Mesh is a mesh generator aimed at producing high quality meshes for use in computational fluid dynamics (CFD) simulations. CFD requires meshes that can resolve boundary layer phenomenon and satisfy more stringent quality criteria than structural analysis.

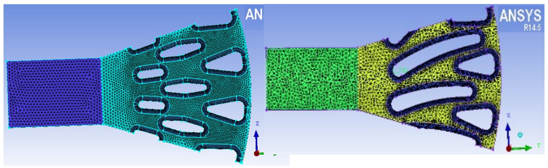


Fig. Surface mesh for ventilation pattern A&B.

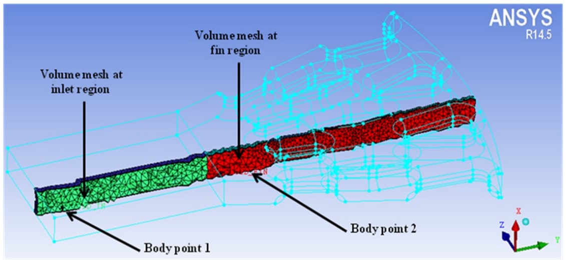


Fig.Volume mesh for ventilation pattern A.

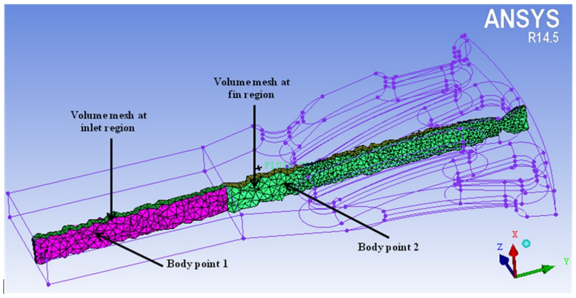


Fig. Volume mesh for ventilation pattern B.

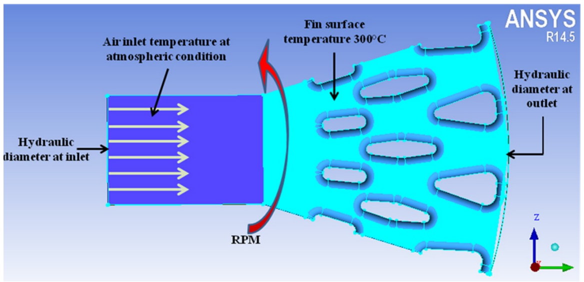
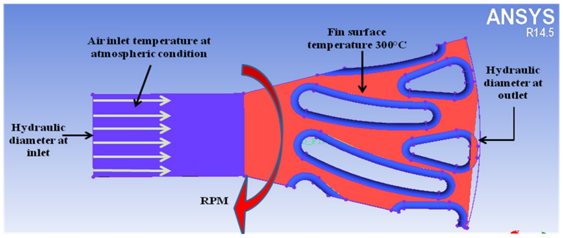


Fig. CFD boundary conditions for pattern A.



FigCFD boundary conditions pattern B.

IV. MASS FLOW RATE SIMULATION RESULT

The performance of the disc brake rotor is influenced by various parameters like heat transfer rate from the internal surfaces, mass flow rate through the passage and temperature uniformity on the surfaces. The increase in the mass flow rate through the passage increases the rate of heat dissipation. This is because of higher mass flow rates associated with higher velocities. The other important aspect is the temperature uniformity, the presence of separation zones or flow blockages within the disc can lead to local low heat transfer regions forming hot spotsTherefore, it is important for a brake disc to not just increase the rate of heat dissipation but also to avoid formation of hot spots. It is also desirable to maximize the efficiency of the cooling air by increasing the work done by it in heat dissipation during its transit through the ventilation passage.

The simulation was performed at various rotational speed varies from 463 rpm to 1643 rpm in order to get results of mass flow rate that would be required for running at higher speeds. Figure shows the attribute definition for ventilation pattern A and B respectively which is required for mass flow rate simulation. The mass flow rate through the datum rotor is calculated as function of individual passage mass flow rate (m). This is evaluated as an area integral at the exit of each passage by using the radial velocity component and the outlet area of the passage. The simulation was performed at various rpm in order to reduce the computational time that would be required for running at higher speeds. Initially an isothermal simulation was performed with a first order upwind scheme and with a convergence criterion of 1e-3. The solution obtained was used as an initial solution and the discretization scheme was made second order accurate and also the convergence criterion was reduced to 1e-4 in order to reduce the scaled residual numerical error in the computation is shown in the figure.

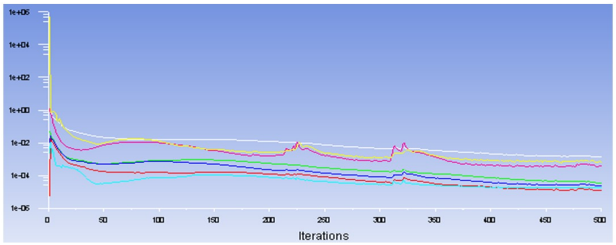


Fig. Convergence residual plot 657 rpm.

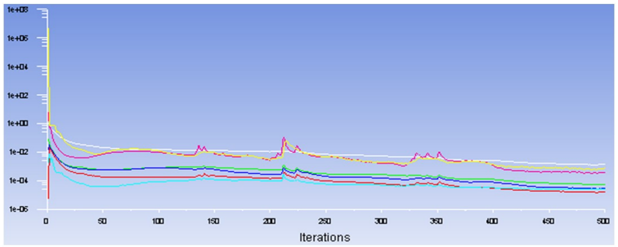


Fig Convergence residual plot for 986 rpm.

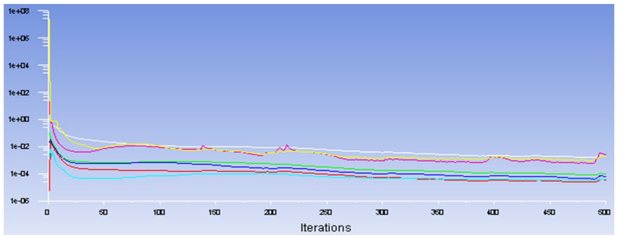


Fig Convergence residual plot for 1315 rpm.

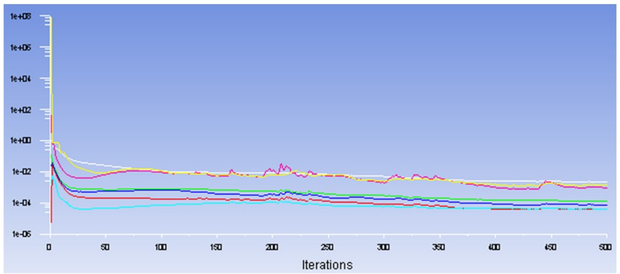
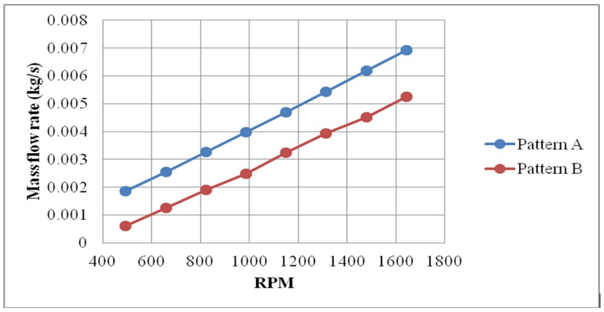


Fig Convergence residual plots for 1643 rpm.

V. I MASS FLOW RATE & STREAMLINE PLOT

The mass flow rate was analyzed through CFD for ventilation patter A and B is tabulated in table 5.1. The mass flow rate from the individual passage for A and B is compared and shown in figure.The hydraulic diameters for inlet and outlet for both the patterns are same. From the figure it shows that, ventilation pattern A has maximum mass flow rate as compared to pattern B. Figure 5.28 to 5.31 shows the streamline plots for ventilation pattern A and figure 5.32 to 5.35 shows the streamline plots for ventilation pattern B.

RPM	Pattern A mass flow rate (kg/s)	Pattern B mass flow rate (kg/s)
493	0.00186166	0.000615119
657	0.002560938	0.001248313
822	0.00326553	0.001914769
986	0.003987126	0.00249135
1150	0.004690663	0.003247256
1315	0.005437449	0.003942847
1479	0.006184432	0.004519457
1643	0.006931249	0.005249486



1. Streamline plots for pattern A

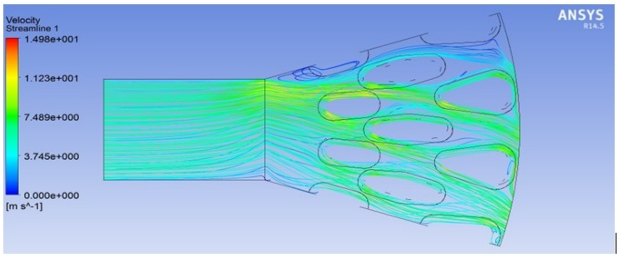


Figure 5.28 Air flow streamline plot for 657 rpm.

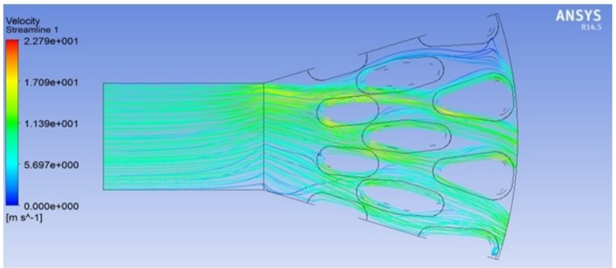


Figure 5.29Air flow streamline plot for 986 rpm.

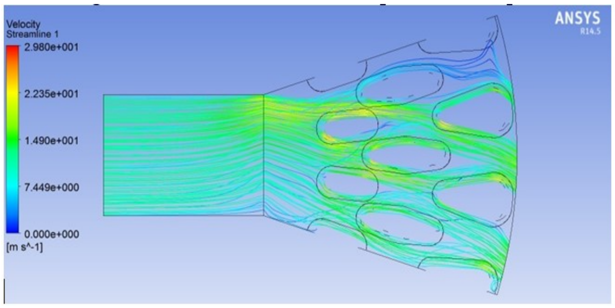


Figure 5.30Air flow streamline plot for 1315 rpm.

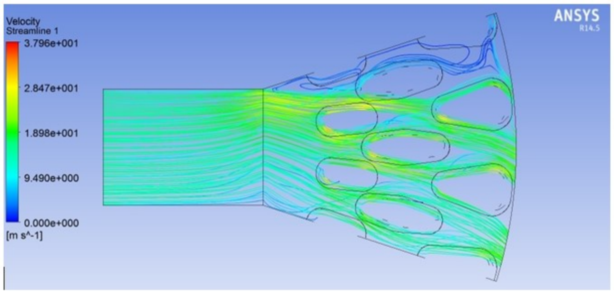


Figure 5.31Air flow streamline plot for 1643 rpm.

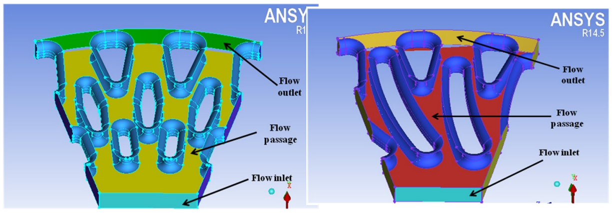


Fig Attribute definition for CFD of ventilation pattern A&B

V.II STREAMLINE PLOTS FOR PATTERN B

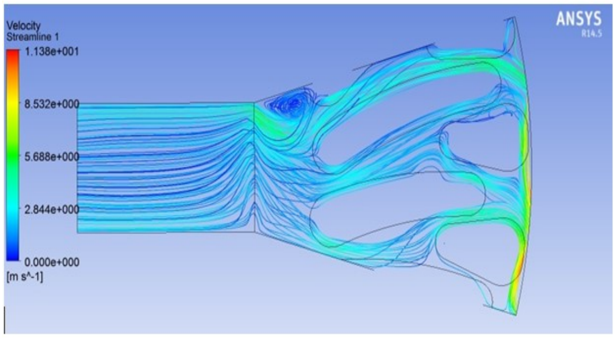


Figure 5.32Air flow streamline plot for 657 rpm.

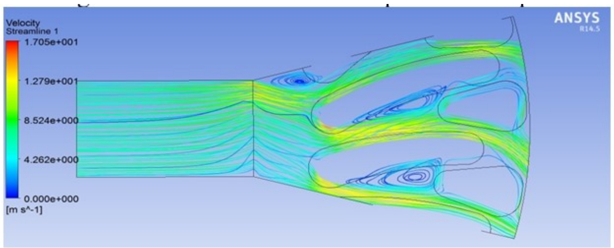


Figure 5.33 Air flow streamline plot for 986 rpm.

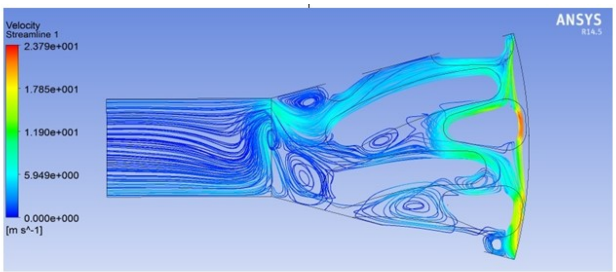


Figure 5.34Air flow streamline plot for 1315 rpm.

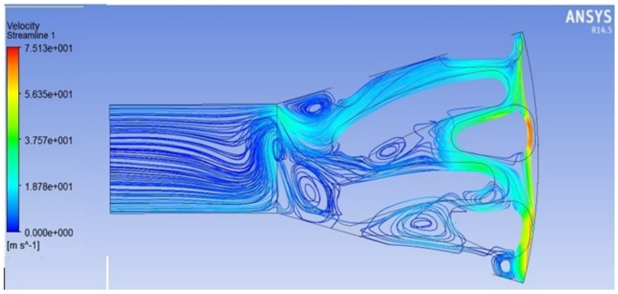


Figure 5.35Air flow streamline plot for 1643 rpm.

VI. CONCLUSION

The need for increased cooling of disc brakes led to the development of vented rotor discs, however the advantages of vented discs over solid discs is the subject of some conjecture. The primary advantage of vented rotors is increased heat dissipation from internal pumping of air; however, under slow speeds the pumping action of the vanes is minimal and only becomes pronounced as rotor speed increases. At higher speeds the airflow flowing around the disc as a result of the forward movement of the vehicle, tends to prevent effective pumping of air through the vanes. For this two type ventilation patterns was designed. An attempt has been made to analyze these ventilation patterns in computational fluid dynamics (CFD) analysis using ANSYS FLUENT. In CFD analysis, the mass flows from each ventilation patterns were analyzed and the values obtained are tabulated in table 5.1. Flow streamline plot is also mentioned in this report. From the CFD analysis, it is clear that the ventilation pattern A is best suitable for mass flow rate as compared with ventilation pattern B.

Finally, design optimization method is performed. This includes selection of candidate material and selection of ventilation pattern for better heat dissipation. As stated earlier, four different materials were analyzed for structural and thermal analysis using ANSYS. From structural and thermal analysis, it has been found that the material **FG260Cr** is optimum material for the brake disc as its total deformation is less as compared to other materials. As its allowable stress is less than ultimate tensile stress.

REFERENCES

[1]. Daniel A, Christo V, Preethy S. Ramya G. “Structural and Thermal Analysis of Brake Disc in Automobiles”, International Journal for Latest Trends in Engineering and Technology, volume 2, issue 3, pp 18-25, May 2013.

[2]. D. Murli, T. Ramkrishna and all, “Experimental and simulated studies on temperature distribution for various disc brakes” International Journal of Research in Mechanical Engineering and Technology, volume-3, issue-1, ISSN: 2249:5762, PP 34-40, April 2013.

[3]. M Rama et al, “Design, Structural and Thermal Analysis of Brake Disc”, International Journal for Futuristic Science Engineering and Technology, volume issue 3, pp 167-171, March 2013.

[4]. Atul Sharma, “Influence of Various Parameters for Reliable Analysis of Disc Brake Rotor”, Paper from Proceeding of National Conference, 3rd and 4th December 2012, pp 20-21, Published by “International Journal of Engineering Science.

[5]. S. Sraip, “Design Development of Lightweight Disc Brake for Regenerative Braking Finite Element Analysis”, International Journal of Applied Physics and Mathematics, volume 3 No. 1, pp 52 -58, January 2013.

[6]. Sachin Pujari, Dr.R.S.Bindu and V.P.Chakre, “A pre-requisite study on predicting heat flux and temperature for automotive brake disc”. Published in 3rd National Conference on Recent Developments in Mechanical Engineering, 2014