RESEARCH ARTICLE OPEN ACCESS

CFD Analysis of Automotive Ventilated Disc Brake Rotor

Amol V. More*, Prof.Sivakumar R. **
*(M.Tech CAD/CAM, VIT University, Chennai)
** (School of Mechanical and Building Sciences, VIT University, Chennai)

ABSTRACT
Disc brakes work on the principle of friction by converting kinetic energy into heat energy. The key objective of a disc brake rotor is to accumulate this heat energy and dissipate it immediately. The effect of rotational speed on the aero-thermal performance is assessed. The rotor speed is observed to have substantial effect on the rotor performance. The heat dissipation and thermal performance of ventilated brake discs intensely be influenced by the aerodynamic characteristics of the air flow through the rotor passages. In order to investigate the aero-thermal performance of the ventilated disc brake at several altered driving speeds of the vehicle, the simulations were carried out at 3 different rotational speeds of 44rad/s 88 rad/s and 120 rad/s. The semi-automatic geometric model is created using the package Solid Works and the mesh for the model is done using ICEM CFD and the Post processing of the results is done using FLUENT-14.5.The results are discussed and presented in detail.
Keywords – Brake rotor, convection, heat transfer, rotor speed, flow rate

I. INTRODUCTION

Braking system is one of the most significant safety components of an automobile. It is predominantly used to slow down vehicles from an initial speed to a given speed. In certain vehicles, the kinetic energy is able to be converted to electric energy and stored into batteries for future usage. Braking is accompanied by the generation of significant heat, some of which is dissipated to the ambient air by convection, some by conduction to the braking system components, and some stored in the components. When braking is of high intensity over short time or of low intensity but extending over a significant time the brake pad and rotor temperatures can increase substantially. Extremely high temperatures can lead to several significant problems. For example the coefficient of friction of the brake pad, being temperature dependent, may change distinctly and affect braking (usually called "fading"). Further, brake components may be exposed to large thermal stresses which produce permanent deformation (or "cracking") and consequently reduced braking performance. The fluid temperature might rise to the point where the fluid vaporizes with the successive loss of braking. As with almost any technology, disc brakes have both benefits and drawbacks. The disc brakes are lighter and dissipate heat directly from its surface to the atmosphere, hence are better than other kinds of brake systems in this concern. This reduces the effort when altering the direction of the moving vehicle. Due to this benefit the disc brake system is used widely in racing applications.

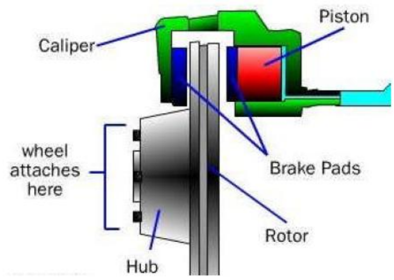

Figure I.Scheme of a disc brake

A common technique to improve the brake cooling is using a ventilated brake disc. Figure I show the scheme of a disc brake. It improves the convective cooling by means of air passages separating the braking surfaces. For years, ventilated brake rotors have been used for their weight savings and additional convective heat transfer from the air channels between the rotor hub cheeks (passages lacking in solid rotors). However, the amount of additional cooling due to this internal air flow is not well defined and depends on the individual brake rotor's geometry and the cooling air flow conditions around the brake assembly. Therefore flow analysis and heat dissipation have fascinated many researchers. Earlier work has addressed both aerodynamic [1, 2, 4, 5, 8] and heat transfer [3, 4, 6, 7] aspects of ventilated and solid discs. Most of the previous workers measured the disc exit flow features with pressure probes [2] and hot wire anemometer [8]. Anders Jerhamre [2] performed numerical simulation and compared their results with experimental data. Voller G.P. [3] conducted experiments for studying all modes of heat dissipation. Eisengraber [7] compared different

methods (thermocouples, pyrometers and thermo-scanner) to study the accuracy and suitability for testing the friction temperature of disc brakes.
Hence a detailed study is carried out in order to understand the effect of different rotor speeds on heat transfer rate and mass flow rate. The aero-thermal performance was studied.

II. METHODOLOGY

A. Geometry Considered

The solid model of the brake disc is shown in the Figure.II. The brake disc is having an outer diameter of 300 mm and inner diameter of 125 mm with 26 numbers of vanes.

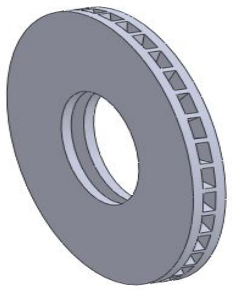

Figure II: 3D model of disc brake in solid works

The model was created in Solid-Works. The actual brake disc involves two rubbing surfaces parted by the blades. One of the rubbing surfaces is attached to the wheel with the help of mounting bolts. The disc brake rotor is rotationally periodic with blades and passages at equal angular spacing. The analysis done for half of the model which reduced the computational cost considerably.Disc rotor bas been modeled using a rotating frame of reference with periodic boundaries. The geometry and the flow pattern repeat themselves at a specified angle about the centerline.

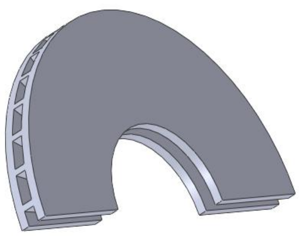

Figure III. Simplification of the original geometry

The flow exiting one periodic plane is similar to the flow entering the other periodic plane. The simplified model used for the analysis is shown in Figure.III.The model created is then exported to the ICEM CFD software in a parasolid file format.

B. Mesh generation

Whole domain of ventilated disc brake is meshed using the unstructured type of grid in ICEM CFD 2012. In unstructured grids, typically utilize triangles in 2D and tetrahedral in 3D. The benefit of unstructured grid methods is that they are much automated and, therefore, need little user time or effort. The critical parts are meshed with prism for better node connectivity. Figure IV shows the meshing of model.

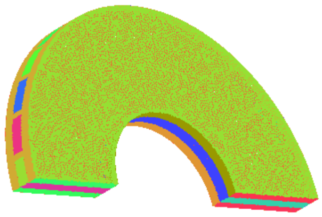

Figure IV. Meshing of fluid domain

TABLE 1.Mesh details

Domain	Mesh range	Type of mesh	Number of elements
Flow Region	Fine	Unstructured	41030
Inlet	Fine	Unstructured	4197
Outlet	Fine	Unstructured	9205
Periodic	Fine	Unstructured	9910
Wall Inlet	Fine	Unstructured	7041

The mesh was then imported into FLUENT V6. The mesh model was checked for quality and following quality was reported as shown in Table II.

TABLE II. Mesh quality

Entity	Value
Total number of elements	1132018
Maximum cell skewness	0.8339

C. Computational Approach

CFD is the method of solving the fundamental non-linear partial differential equations that govern the fluid flow, heat transfer and turbulence of flow. For the present work, CFD simulations are carried using the commercial CFD software package FLUENT-14.5. The disc brake being attached to the wheel also rotates at the speed of the wheel of the automobile and the flow inside the rotor is predominantly swirling and rotating. In problems involving rotating motion FLUENT adapts a method called rotating reference frame. The governing equations are solved by considering the walls to be stationary in the relative frame (default) and the fluid to be moving with acceleration of the reference frame. This acceleration of fluid is included as a source in the momentum equation.

In the CFD simulation, the following assumptions have been made:

1) Steady state air flow
2) Segregated solver and implicit formulation
3) Standard k-ω SST model
4) Standard wall functions

Second order upwind scheme is used for Momentum, Turbulence kinetic energy, Turbulence dissipation rate and energy. Steady-state conditions were assumed because the time dependent behavior was not significant. By using the second-order upwind scheme, higher accuracy was achieved. The flow in and around the disc brake rotor is incompressible as it is in a very low Mach number regime. The boundary conditions at the top and bottom of the domain is given as pressure inlet conditions. A pressure inlet boundary condition with 1atm pressure was applied at the domain inlet. For an incompressible flow FLUENT uses the Bernoulli's equation to find out the velocity from the pressure and the direction of velocity is as specified at the boundaries. The Navier-Stokes equation is then solved by FLUENT using SIMPLE algorithm.K-ω SST model works well for modelling a brake disc rotor turbulence as it resolves the viscous region effects very well compared to other two-equation models. This is because the K-ω SST model uses K-ω in the near wall region.

D. Boundary Conditions

The Grey cast iron is used for the solid geometry as from the perspective of stiffness, friction resistance and cost. The cast iron is commonly used in industry.

TABLE III. Properties of Grey cast iron

Material	Density(ρ)	Thermal Conductivity(K)	Specific Heat(Cp)
Grey Cast Iron	7200 Kg/m^3	45 W/mK	510 J/Kg K

The air is taken as a fluid and properties of air are copied from fluent database. Pressure based steady state solver is used for computing. Pressure inlet and pressure outlet conditions were given with specific hydraulic diameters. Vane-wall interface with constant temperature of 900 K with no slip boundary condition is considered. Heat flux is applied on the rotor walls where caliper and rotor are in contact. The three different speeds 44rad/sec, 88rad/sec and 120rad/sec were taken for the comparison.

III. RESULT AND DISCUSSION

It is known that for different rotational speeds the average velocity at the outlet changes. While performing the simulations the same grid has been used for all the rotational speeds. It has been seen that with increase in the rotational speed the mass flow rate through the passage increases as shown in Table IV.

TABLE IV.Exit velocity and mass flow rate at different speeds

Speed of rotation rad/s	Exit velocity m/s	Mass flow rate kg /s
44	148	0.1628
88	269	0.3125
120	494	0.4559

It can be seen from Figure.V that a linear proportionality is achieved for the variation of mass flow rate with rotor speed. The increase in the mass flow rate is mainly due to the increase in the exit velocity. As the speed of rotation increases the diffusion on the trailing edge of the suction side reduces. This is again due to higher energy in the flow which can overcome the adverse pressure gradient due to change in area from inlet to outlet. This is very useful as diffusion creates regions of low heat transfer thereby causing hot spot formation.

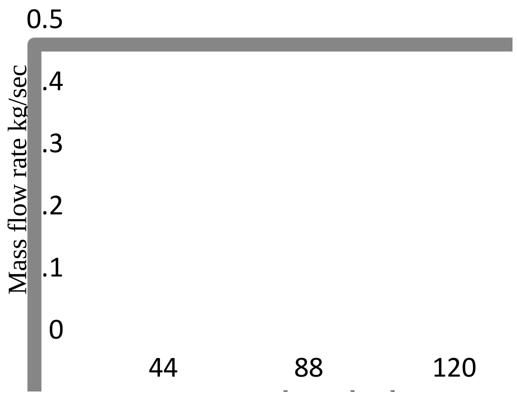

Figure V.Mass flow rate as a function of rotor speed

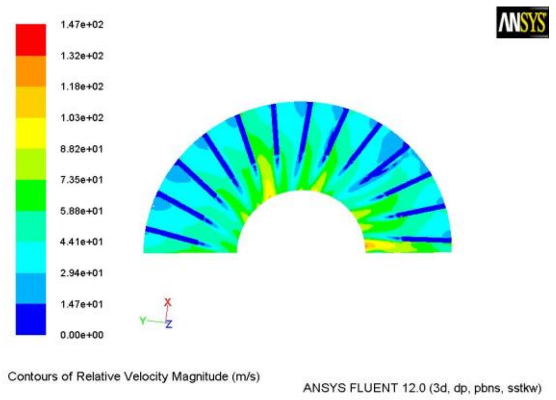

Figure VI.Relative velocity at midplane of rotor at 24rad/s

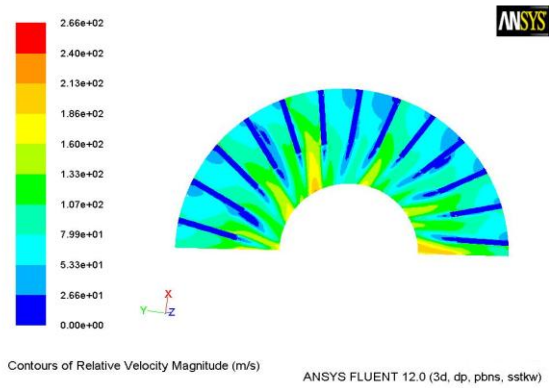

Figure VII.Relative velocity at midplane of rotor at 88rad/s

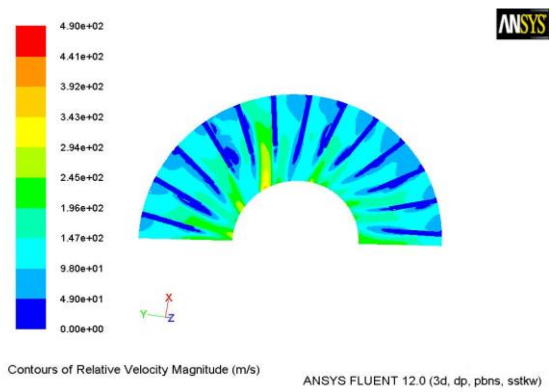

Figure VIII: Relative velocity at midplane of rotor at 120rad/s

It has also been observed that the total heat transfer rate through the passage also increases with increase in the blade speed. This is mainly due to increase in mass flow rate through the passage. But the variation of total heat transfer rate is not linear as shown in Figure IX.

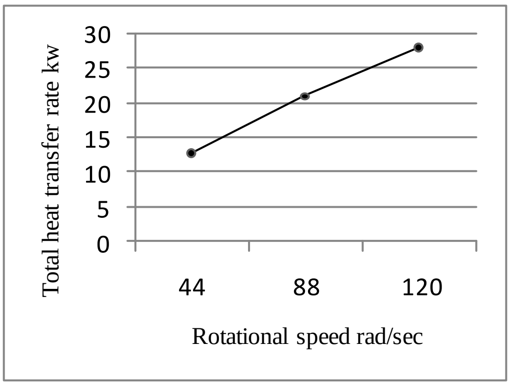

Figure IX.Total heat transfer rate as a function of rotor speed

It specifies that the change is not just due to increase in mass flow rate but also due to change in flow aerodynamics at higher rotational speeds. This can be better understood by observing the Nusselt number distribution on each of the internal surfaces at various speeds. The suction side heat transfer is affected by inlet separation due to flow direction change from tangential to radial, axial to radial and also due to the diffusion on the trailing edge as shown in Figure X. It can be observed that with increase in speed the diffusion on the trailing edge is reduced thereby improving the heat transfer rate. But the inlet separation increases with increase in the speed thereby affecting the rate of heat transfer from the suction side of the passage.

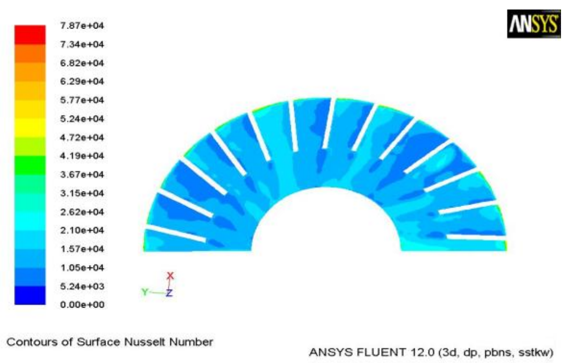

Figure X.Nusselt number distribution at 24rad/s

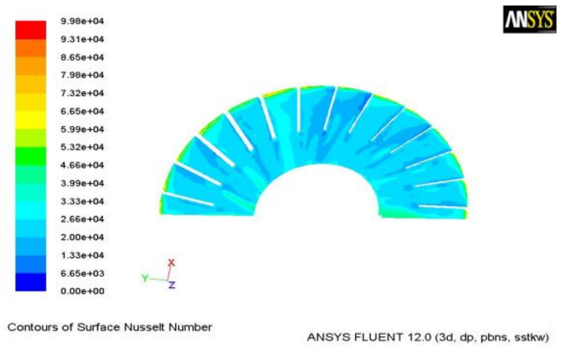

Figure XI. Nusselt number distribution at 88rad/s

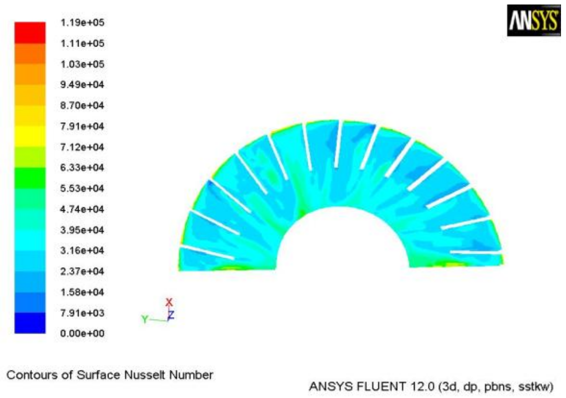

Figure XII.Nusselt number distribution at 120rad/s

The pressure surface heat transfer distribution uniformity improves with the increase in the rotational speed. This is due to highly accelerated flow towards the pressure side of the passage. Due to the marginal increase in the inlet separation with rotational speed the flow is accelerated more towards the pressure side at higher speeds thereby improving the heat transfer.

IV. CONCLUSION

The effect of the rotor speed on the aero-thermal performance of a ventilated disc brake was studied in detail.The increase in 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 aero-thermal performance of the rotor increases with rotational speeds due to increase in mass flow rate and rate of heat transfer.

V. FUTURE WORK

A few modelling assumptions were made for the existent work all through the CFD analysis of a disc brake rotor. These assumptions may affect the aero-thermal performance of the disc brake rotor. In order to understand the disc brake performance well, the following recommendations are made for future work in this area.

1. CFD analysis of the disc brake rotor with the inclusion of conduction through the metal. This will give a better understanding of the rotor hot spot formations.
2. CFD analysis with the inclusion of turbulators inside the rotor passage.
3. CFD analysis to determine the effect on the aero-thermal performance of the rotor by the inclusion of bolts and balancing clips.

REFERENCES

[1] David A. Johnson, Bryan A. Sperandei and Ross Gilbert, Analysis of the Flow through a Vented Automotive Brake Rotor, *ASME, J. of Fluids Engg, 125*, November 2003, 979-986.

[2] Anders Jerhamre and Christer Bergstrom, Numerical Study of Brake Disc Cooling Accounting for both Aerodynamic Drag Force and Cooling Efficiency, *SAE 2001-01-0948*,2001.

[3] Voller G. P., Tirovic M., Morris R and Gibbens P, Analysis of Automotive Disc Brake Cooling Characteristics, *Proc. Instn. Mech. Engrs. Part D: J. Automobile Engineering*, 2003,217, p.657-666.

[4] Lisa Wallis, Eddie Leonardi and Brian Milton, Air Flow and Heat Transfer in Ventilated Disc Brake Rotors with Diamond and Tear-Drop Pillars, *Numerical Heat Transfer, Part A,41*: 643-655, 2002.

[5] Michael D. Hudson and Roland L. Ruhl, Ventilated Brake Rotor Air Flow Investigation, *SAE 971033*, 1997.

[6] Thomas Valvano and Kwangjin Lee, An Analytical Method to Predict Thermal Distortion of a Brake Rotor, *SAE 2000-01-0445*, 2000.

[7] Reinhard Eisengraber, Jaroslaw Grochowicz and Matthias Schuster, Klaus Augsburg and Lars Kock, Comparison of Different Methods for the Determination of the Friction Temperature of Disc Brakes, *SAE 1999-01-0132*, 1999.

[8] Parish D., MacManus D. G., Aerodynamic Investigations of Ventilated Brake Discs, Proc.Instn. Mech. Engrs. *Part D: J. Automobile Engineering, 219, p.471-486,* 2005.

[9] Effect Of Vane Shape On Flow And Heat Transfer Characteristics Of A Brake Disc J M Mallikarjuna, S Manohar Reddy and Ganesan V, 22 December, 2006, Dhaka, Bangladesh.

[10] Cooling Analysis of a Passenger Car Disk Brake Thomas Schuetz *AUDI AG*, July 20, 2011.

[11] Experimental Investigation of the Mean and Turbulent Flow Characteristics at the Exit of Automotive Vented Brake Discs, G. Barigozzi, G. E. Cossali and A. Perdichizzi, October 6-9, 2002