International Journal of Current Engineering and Technology
ISSN 2277 - 4106

Research Article

# Thermal Analysis of Automotive Brake Rotor

Kapil A Shelar[Á*], Shivprakash B Barve[Á], Mahesh R. Chopade[Á] and Pramod E Chaudhari[Á]

[Á]Mechanical Engineering Department, MAEER's MIT College of Engineering, Kothrud, Pune

Accepted 12 March 2014, Available online 01 April 2014, **Special Issue-3, (April 2014)**

## *Abstract*

*The heat dissipation and thermal performance of ventilated brake discs strongly depends on the aerodynamic characteristics of the air flow through the rotor passages. In this paper, the thermal convection is analyzed using an analytical method, and the velocity distribution, temperature contours and Nusselt number are determined. Then numerical models for different rotors, pillar post rotors and vane rotors are generated and numerical simulations are conducted to determine the desired parameters. To analyze more realistic vane and pillar post rotor models, commercial CFD software packages, Fluent and Gambit, are used to simulate the heat flux rate, air flow rate, velocity distributions, temperature contours, and pressure distributions inside the rotors. Furthermore, sensitivity studies have been performed, to determine the effects of a different number of vanes or pillar posts, inner and outer radii and various angles of vanes.*

***Keywords:*** *Heat transfer, Brake rotor, Simulation, Geometrical optimization.*

## 1. Introduction

A braking system is one of the most important safety components of an automobile. It is mainly used to decelerate vehicles from an initial speed to a given speed. In some vehicles, the kinetic energy is able to be converted to electric energy and stored into batteries for future usage. These types of vehicles are known as electric or hybrid vehicles. However, these kinds of vehicles still need a backup system due to sometimes insufficient electric energy or failures which inevitably increase the cost of the vehicles. So friction based braking systems are still the common device to convert kinetic energy into thermal energy, through friction between the brake pads and the rotor faces.

When brakes are applied, the kinetic energy of the vehicle is converted into heat which is primarily dissipated by convection through the vanes of the rotor. Insufficient heat transfer leads to overheating of the rotor and eventually failure. This is more likely in extreme applications like emergency braking or excessive repetitive braking. Analytical modeling of the actual situation is a very complex and time consuming method. There are lots of assumptions involved with respect to the flow and heat transfer parameters. Experimental work is also highly time consuming and expensive. Experimental analysis on a test rig will give results which are very close to the actual situation, but it is not a feasible method in the development stage. The computational time is reduced by virtue of comparative study and hence simplifying the geometry. A suitable meshing strategy is very essential to a successful analysis and the same has been developed for the analysis in the pre-processor ICEM CFD. FLUENT is used as solver to obtain the solution for different cases by assigning suitable boundary conditions.

## 2. Analysis and assumptions

A disc brake rotor rotates with the wheel of the vehicle. It is essentially a rotating duct system on which a constant heat flux is applied. This heat is carried away by the flowing through the ductsThe ideal analytical solution for this would be obtained by using the conservation equations, namely conservation of mass (continuity equation), conservation of momentum (Navier-Stokes equation) and conservation of energy. A complete analytical solution of the Navier-Stokes equations is not possible. Hence, unable to employ this approach. An alternative for this is CFD technique. These techniques allow for some assumptions and give the results that we expect. The pressure distribution of the pad over the disc is not uniform. To estimate the flux, need to account for variables like pressure distribution, change in velocity etc. Hence the modeling of the problem using a constant heat flux condition is not feasible. The computational domain can be made smaller by analyzing only one flow passage due to axis-symmetric geometry. All the flow passages being similar, multiply the heat transfer rate from one flow passage by the number of passages to obtain the actual total heat transfer rate.

As this investigation is comparative, there are certain factors that are common to all rotors which can be

---

*Corresponding author: Kapil A Shelar

excluded from analysis to make geometry and meshing easier and reduce the computation time. These include factors like radiation, conduction etc. Natural convection occurs irrespective of any other parameters except temperature difference. Thus, it will contribute equally in all cases under consideration, as giving the same wall temperature.

## 3. Analysis using CFD

In order to improve the accuracy of simulation results, a commercial CFD solver, Fluent, was used to simulate the air flow inside different rotors and find the optimized shape of rotors. The 2-D mesh was built with ICEMCFD, using boundary conditions of pressure at the inlet, pressure at the outlet and the wall vane, as shown in figure. 1.

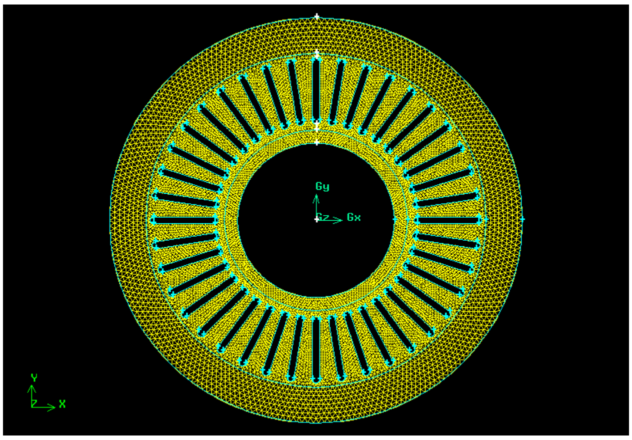

**Fig.1** 2-D mesh model of a 40-vane rotor

Numerical Simulations Using FLUENT

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

- Steady state air flow
- Segregated solver and implicit formulation
- Standard k-epsilon viscous turbulence model
- Standard wall functions
- Moving reference frame at a constant velocity of 44 rad/s
- Vane- wall interface with a constant temperature of 900 K
- Momentum-Second Order Upwind Scheme
- Turbulence Kinetic Energy-Second Order Upwind Scheme
- Turbulence Dissipation Rate-Second Order Upwind Scheme
- Energy-Second Order Upwind Scheme

The segregated solver traditionally has been used for incompressible and mildly compressible flows. The coupled approach, on the other hand, was originally designed for high-speed compressible flows . The air flow inside the vehicle is assumed incompressible, so the segregated solver and implicit formulation were used. The standard - model is a semi-empirical model based on model transport equations for the turbulence kinetic energy $(k)$ and dissipation rate $(\varepsilon)$ . It is used for fully turbulent flows, particularly in the inlet and outlet areas of the rotors. FLUENT can also model the air motion inside rotors by using the moving reference frame at constant speed. The non-equilibrium wall functions are recommended for use in complex flows involving separation, reattachment, and impingement where the mean flow and turbulence are subjected to strong pressure gradients and change rapidly . Standard wall functions were selected. By using the second-order upwind scheme, higher accuracy was achieved.

In the simulations, the heat transfer rate was found. The predicted velocity distribution is shown in figure 2. The predicted pressure distribution is shown in figure 2

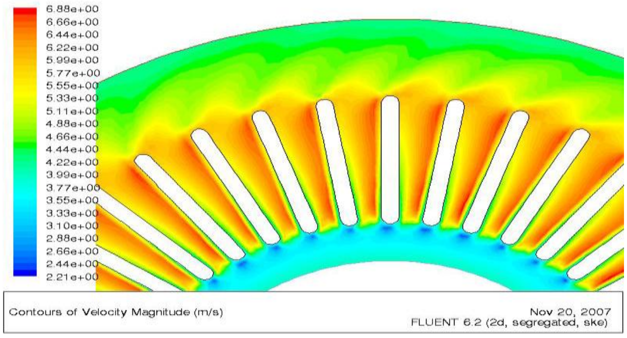

**Fig. 2**: Velocity contours of a 40-vane rotor

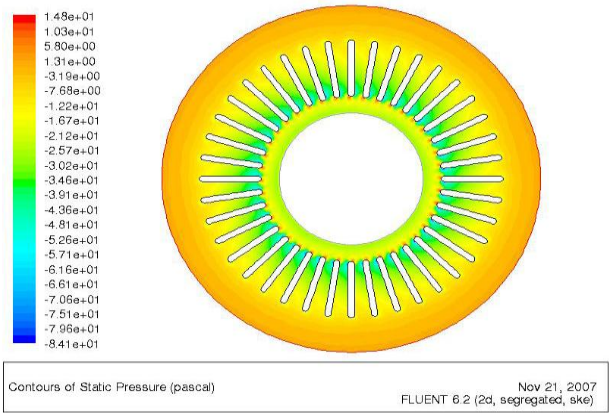

**Fig.3:** Static pressure contours of a 40-vane rotor

The velocity vector distribution and turbulence distribution are shown in figures.4 and 5, respectively.

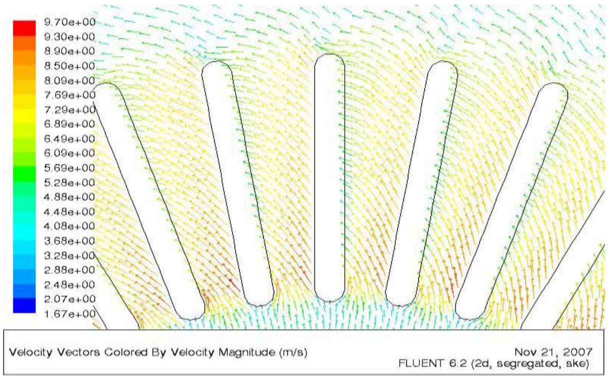

**Fig.4:** Velocity vector distribution of a 40-vane rotor

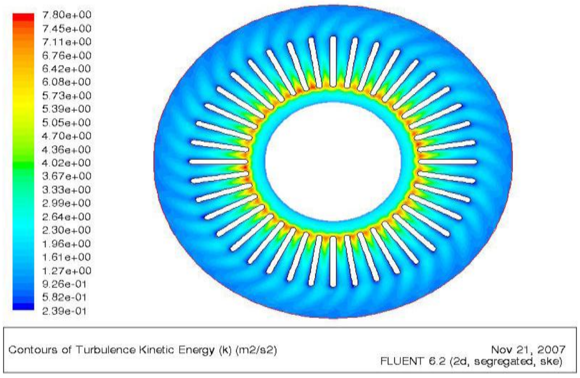

**Fig.5:** Turbulence distribution of a 40-vane rotor

To increase the thermal performance of rotors, both the air velocity and heat transfer rate should be increased. The models with different vane numbers, vane angles and radii of curvature will be investigated in the following sub-sections.

## 4. Results and discussion

### *1. Effects of vane numbers*

Five different rotors with 24, 32, 40, 50 and 60 vanes were chosen to evaluate the effects of vane numbers on the change of heat transfer rate. If more heat is transferred from the rotors, there will be better cooling performance.

Figure 6 shows the relationship between vane numbers and the heat transfer rate at angular velocities of 44, 88 and 120 rad/s, respectively.

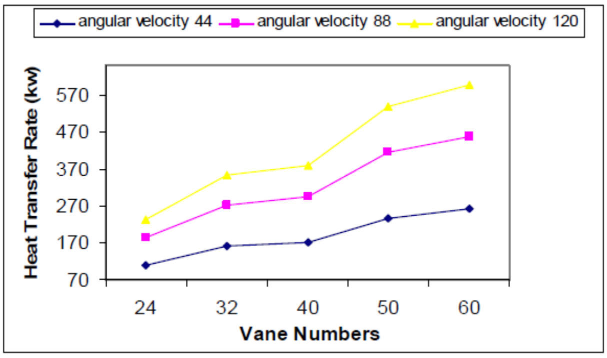

**Fig.6:** Vane numbers vs. heat transfer rate

As shown in figure. 6, with an increase of vane numbers, the heat transfer rate increases as well. This observation is consistent with the result derived from the previous analytical solution. The comparison of velocity distributions for five different models with 24, 32, 40, 50, 60 vanes is shown in figure 6, from which it is observed that the velocity distribution is uneven in each section of vane rotor. The rotors are all rotating at a constant velocity in the clockwise direction. The rotors with lower vane numbers have a larger non-uniformity of velocity distribution than those with more vane numbers. This occurs because the direction of air flow is not straight along the vanes, as shown in figure. 7. The air flow vectors in rotors with fewer vanes have larger angles along the vanes than those with more vanes.

### *2. Effects of vane angles*

To investigate the influence of vane angles on the heat transfer rate, two sets of rotor models with 32 and 40 vanes are developed. For each set of rotor models, the vane angles take the values of 100, 200, 300 and 400, respectively. With these rotor models, numerical simulations are conducted at angular velocities of 44, 88 and120 rad/s, respectively.

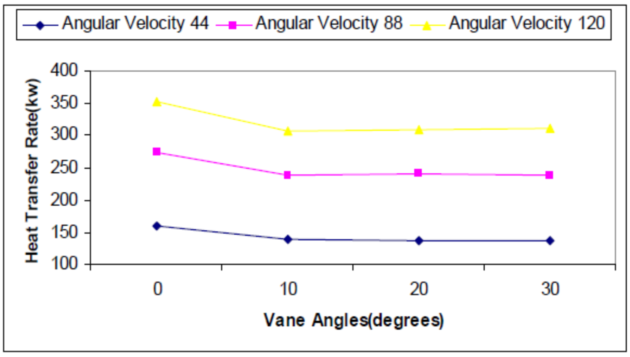

**Fig.7:** Vane angles vs. heat transfer (32 vanes)

## 5. Conclusion

Various braking rotor models with different geometric parameters, such as vane numbers, vane angles, radii of curvature and short-long ratios were studied numerically. The results demonstrated that CFD is an effective method for analyzing the heat transfer behaviour of vented disc brakes with different vane configurations. The simulation results indicate that an increase of vane numbers drastically improves the thermal performance by 63.5%, if the vane numbers increase from 32 to 60 at an angular velocity of 44 rad/s, by 67.9% at an angular velocity of 88 rad/s, and 69.2% at an angular velocity of 120 rad/s. The vane angles do not contribute to the improvement of thermal performance when the vane number is 32. With a vane number of 40, the heat transfer increases by 16% at an angular velocity of 44 rad/s, and 1.16% at an angular velocity of 120 rad/s. For the curved vanes, with 32 vanes, the thermal performance does not increase for curved vanes. But it does increase with 40 vanes by 27%, 30%, 31.6% at an angular velocity of 44, 88 and 120 rad/s, respectively, when the radius of curvature increases to 114 cm. From the simulation results, the short-long ratio does not substantially contribute to the improvement of cooling performance.

## References

Sun Li, the Disk Brake Design and Performance Analysis.

A.D. McPhee and D.A. Johnson, 47 (2008) 458-467. Experimental Heat Transfer and Flow Analysis of Vented Brake Rotor.

B.C. Song, K.H. Lee, International Journal of Automotive Technology, vol. 10, No. 3, pp 321-328(2009).Structural optimization of a Circumferential Friction Disk Brake with Consideration of Thermo elastic Instability.

H. Mazidi, S. Jalalifar, J. Chakhoo, Asian Journal of Applied Sciences, 4(2): 119-136, 2011. Mathematical Modelling of Heat Conduction in a Disk Brake System during Braking.

Adam Adamowicz, Piotr Grzes, Science Direct, Applied Thermal Engineering (2011) 1-9. Influence of Convective Cooling on a Disc Brake Temperature Distribution During Repetitive Braking.

Adam Adamowicz, Piotr Grzes, Science Direct, Applied Thermal Engineering 31(2011) 1003-1012. Analysis of Disc Brake Temperature Distribution during Single Braking Under Non-Axisymmetric Load.

Esmail M.A. Mokheimer, International Journal of Heat and Mass Transfer 45 (2002) 3631–3642. Performance of annular fins with different profiles subject to variable heat transfer coefficient.

Pulkit Agarwal, Mayur Shrikhande and P. Srinivasan, Proceedings of the World Congress on Engineering 2011 Vol III WCE 2011, July 6 - 8, 2011, London, U.K. Heat Transfer Simulation by CFD from Fins of an Air Cooled Motorcycle Engine under Varying Climatic Conditions.

Dewitt and Incropera, John Wiley and Sons .Fundamentals of Heat and Mass Transfer.

.Zhongzhe Chi, Thermal Performance Analysis and Geometrical Optimization of Automotive Brake Rotors.

.