International Journal of Thermal Technologies
E-ISSN 2277 – 4114
Available at http://inpressco.com/category/ijtt/

Research Article

Investigation for Flow of Cooling Air through the Ventilated Disc Brake Rotor using CFD

Shelar Kapil†, Chopade Mahesh†* and Valavade A. P.‡

†University of Pune, MIT College of Engineering, Pune, India
‡Bharati Vidyapeeth Deemed University College of Engineering, Pune, India

Accepted 25 April 2015, Available online 01 June 2015, **Vol.5, No.2 (June 2015)**

Abstract

The ventilated disc brake rotor with vane passages formed between braking surfaces; acts as a centrifugal fan and facilitates the required air flow for cooling. To improve the performance of a ventilated disc brake rotor, selection of proper geometrical configuration is more important. Disc brake rotor with different geometrical configurations were analyzed for flow analysis. Numerical modelling can predict flow characteristics & can serve as an efficient design tool. Hence it has been used for making modifications in the existing rotor geometry vane passages and which will consequently contribute for better flow development. A CFD code has been employed for theoretical analysis which will determine the distribution of pressure and velocity vectors in the flow field. Numerical results for mass flow rate have been validated experimentally. The selection of disc brake rotor configuration for breaking application has been discussed based on the obtained results.

Keywords*: Ventilated Disc Brake Rotor, CFD, and Vane shape.*

1. Introduction

The main purpose of a braking system is to slow down a vehicle or to stop it completely within a reasonable amount of time. It is one of the most important safety features of a vehicle and therefore it must be reliable enough. Any moving vehicle possess kinetic energy by virtue of its motion. A braking system takes advantage of simple means of energy transfer in order to accomplish this objective, by converting the kinetic energy of the vehicle into some other form of energy. Most brakes use the principle of friction to convert this kinetic energy into heat energy. Inherent advantages of disc brakes over drum brakes have led to their universal use on passenger-car and light-truck front axles, many rear axles, and medium-weight trucks on both axles. Ventilated disc brake rotors are preferred over solid ones due to increased area of heat transfer and reduced mass. Increased rotor temperatures can lead to several significant problems like warping, cracking, hot judder, brake fading and overheating of brake fluid, seals and other components. The fluid temperature may rise to the point where the fluid vaporizes with the subsequent loss of braking.

Methodology

Considering the various failure modes explained in previous section, it is prime aim to increase brake efficiency and its reliability by varying geometrical configuration. The literature survey gives information towards several possibilities of improving conventional disc brake rotor configuration. Modelling of commonly used commercial brake rotor has been carried out using Pro-E Wildfire 4.0. The meshing tool of ANSYS viz. ICEM CFD is used for geometrical modelling and mesh generation for ventilated rotors. FLUENT is used to simulate the air flow through the ventilated rotors. The main aim of this research is to maximize the flow rate of air through the rotor. Several modifications were made in the flow passages of the rotor to achieve this aim. The domain of interest in this study is the flow passage of air, while the rest of the geometrical parameters of the rotor are kept constant. A segment of rotor has considered for analysis due its rotational symmetry. Periodic boundaries are applied to either side of the segment to represent the entire rotor. The rotors are considered as rotating discs in an infinite environment by a rotating frame of reference and the application of an open boundary condition to the domain. The atmospheric pressure of zero gauges is specified at the periphery of the domain. The surface temperature of rotor and ambient temperature of air are considered constant for entire simulation. The computational results are validated experimentally. The plots of different parameters are obtained for each modification and discussed in detail for their effect on flow development in rotor passages.

*Corresponding author: **Chopade Mahesh**

Objectives

The objectives are

1. Increase in mass flow rate of fluid through the rotor
2. Make the better use of available mass flow rate

Computational Modeling and Simulation

Different rotor geometries were considered for the CFD analysis.

a. SUV1 - The model has 36 vanes which are inclined at an angle of 36.87^0
b. SUV2 - 36 tapered radial vanes and axial inlet
c. Sedan - 40 straight radial vanes with a pattern of 1 long and 4 short vanes
d. Hatchback - pattern of 3 cylindrical pillar posts

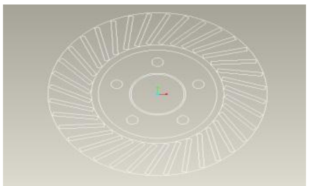
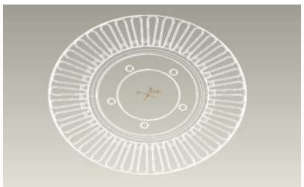

Figure 1(a) **Figure 1(b)**

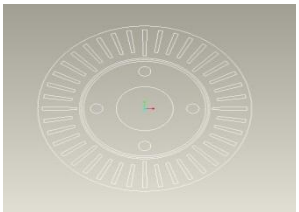
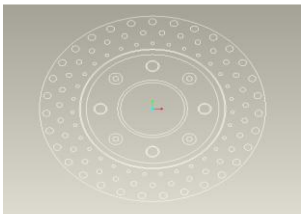

Figure 1(c) **Figure 1(d)**

The various analyses carried out are enlisted below

1. Grid Independent Study
2. Comparative analysis between tapered radial vanes, inclined radial vanes and circular pillar posts
3. Effect of long-short vane combinations

Modeling and Meshing

The 3D rotor geometries were imported in ICEM CFD in IGES format. The hexahedral mesh was used and boundary conditions given were pressure inlet/outlet, periodic wall boundary and constant wall temperature. As shown in the Fig. 1, colour coding was done for inlet (blue), outlet (yellow), periodic wall boundary (red) and wall (green).

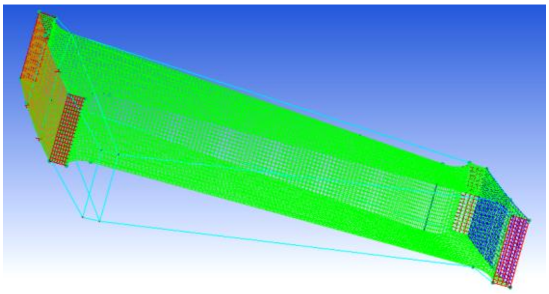

Figure 2 Hexahedral Mesh of Single vane Passage

Numerical Simulation

The following solution methods and conditions were chosen during the simulation in

Ansys Fluent

- Phase coupled SIMPLE algorithm
- Steady state, incompressible air flow
- Realizable k-epsilon viscous turbulence model
- Rotational frame of reference at a constant velocity of 60 rad/s
- Vane-wall interface with a constant temperature of 900 K
- Momentum-First Order Upwind Scheme
- Turbulence Kinetic Energy-First Order Upwind Scheme
- Turbulence Dissipation Rate-First Order Upwind Scheme
- Energy-First Order Upwind Scheme

The post-processing was carried by acquiring the plots of velocity vectors and pressure contours.

Variation in the Number of Vanes

It is important to optimize the number of vanes in order to achieve a balance between the mass flow rate, frictional losses, structural strength and the weight of the rotor.

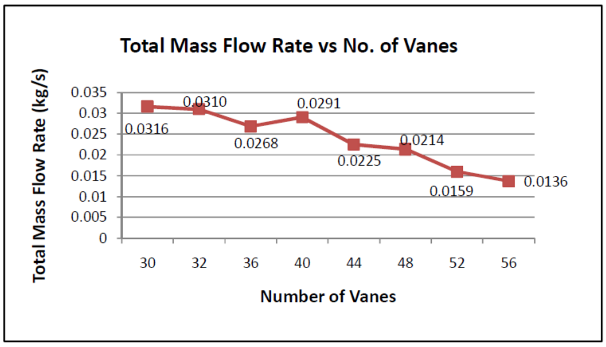

Figure 3 Variation in No. of Vanes vs. Mass Flow Rate

While varying the number of vanes, the wall thickness has been kept constant. The disc brake model used for analyses of variation in the number of vanes was an

SUV-1 model. Assuming rotational symmetry a single vane was used for analysis which greatly reduces computational time. There is a limit for increasing the number of vanes due to increased losses and weight considerations. The simulations were carried out for vane numbers ranging from 30 to 56 and the results for mass flow rate are plotted as shown in Fig. 3. The characteristics shown in the graphs indicate that the mass flow rate decreases gradually with increase in number of vanes, while the heat transfer rate has a peak at 40 vanes.

From the characteristics of both the graphs we can conclude that optimum characteristics are obtained for 40 vanes. The results for 40 vane number have been put forth with the help of velocity vector plots, pressure and temperature contours. The plot of the velocity vectors has shown in Fig.4. The contours of static pressure have shown in Fig. 5.

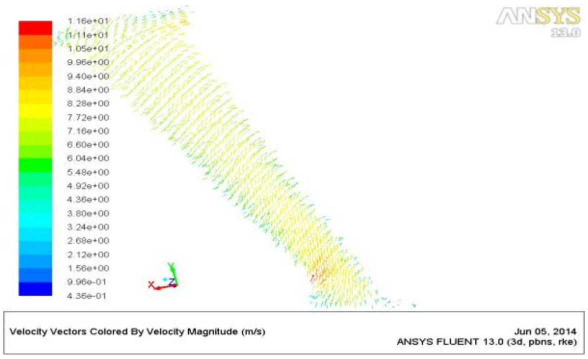

Figure 4 Velocity Contours

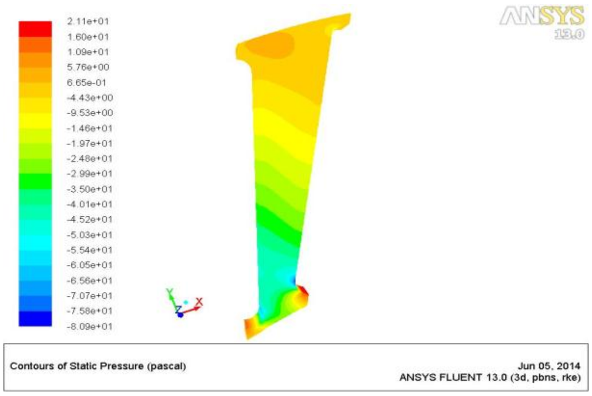

Figure 5 Pressure Contours

Thus the above analysis results indicate that the heat transfer and mass flow rates for the rotor which has 36 vanes is distinctly less than the rotor with 40 vanes. Comparison between the original rotor and the rotor with 40 vanes is given below:

Table 1 Numerical results for number of vanes

Rotor Type	Mass Flow Rate (kg/s)
SUV-1 Rotor (36 vanes)	0.0268
Optimized Rotor (40 vanes)	0.0297

For lower number of vanes, total vane area for heat transfer decreases and flow separation zone of the fluid increases due to increased distance between vanes. This is why the mass flow and heat transfer rate is less for 36 vanes as compared to 40 vanes. Apart from this there is also an increase in friction loss at the entry of the vane due to lower area at mouth of vane .

Comparative Analysis and Validation

This section deals with the comparative study between three different rotor designs with regards to mass flow rate. The results obtained numerically were validated experimentally. The different rotor patterns selected are as follows:

1. SUV-2
2. Sedan
3. Hatchback

These different types of rotor designs were selected so as to get a clearer understanding regarding the relations of different geometrical parameters with the heat transfer and mass flow rates.

A. Numerical Simulation

All the above mentioned rotors were analyzed numerically and results were obtained mass flow rates. All analyses were done for constant speed of 60 rad/s (572.88 rpm). The SUV-2 rotor was analyzed by taking a single vane for modeling and simulation. Considering rotational symmetry the results for the single vane were multiplied by the vane number to get overall results for heat transfer and mass flow rate. The meshed model of SUV-2 rotor is shown in Fig. 6 (a).

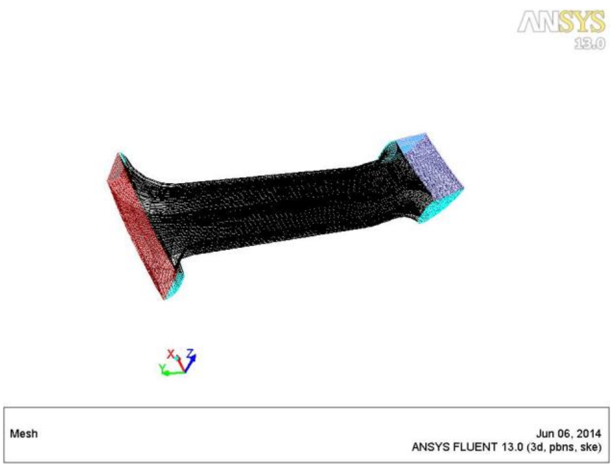

Figure 6 (a) Meshed Model of SUV-2

For the sedan VDBR the repeating pattern of 1 long and 4 short vane were used for analysis. The meshed rotor model is shown in Fig. 6 (b).

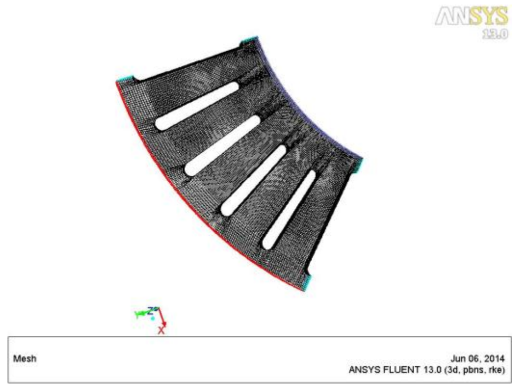

Figure 6 (b) Meshed Model of Sedan VDBR

The analysis for hatchback model was carried out using the repeating pattern of 3 cylindrical pillared vanes. The meshed geometry is shown in Fig. 6 (c).

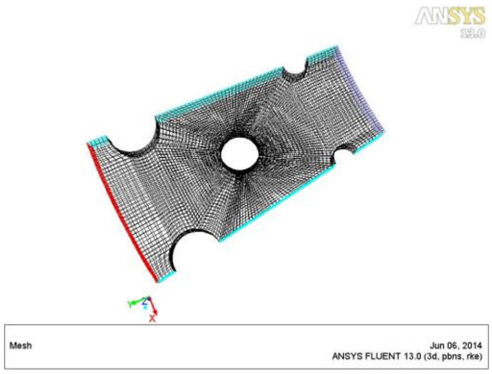

Figure 6 (c) Meshed Model of Hatchback Model

The results obtained from numerical analysis for heat transfer rate and mass flow rate are given below:

Table 2 Numerical Results

Rotor Type	Mass Flow Rate (kg/s)
SUV-2	0.01197
Sedan	0.008356
Hatchback	0.005614

The above results show that mass flow rate is highest for SUV-2 model which obvious due to its higher diameter. The mass flow rates for sedan are higher than hatchback.

Experimental Validation

For validating the numerical results it was necessary to measure the mass flow rates experimentally. This was done using a blower and a digital anemometer shown in Fig. 7 (a) and (b) respectively. The various components of the experimental apparatus are as follows:

- 1 HP single phase D.C. motor with shaft for mounting rotor
- Instrumentation panel consisting of RPM indicator and variable speed drive
- Volute blower and pipe assembly
- Digital anemometer for measuring flow speed

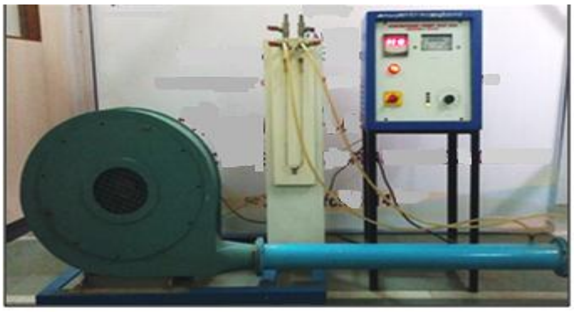

Figure 7 (a) Blower Apparatus

Figure7 (b) Digital Anemometer

The brake disc rotors were mounted on the shaft of the motor and rotated at different speeds by means of the variable speed drive. The speed range selected was 200 to 900 rpm. For mounting the discs onto the shaft two hubs were manufactured – one used for mounting the SUV-2 rotor while the other used for the hatchback and sedan. The hub used for the SUV-2 rotor is shown in Fig. 8 (c). The flow speed was measured by keeping the anemometer measuring head firmly against the pipe exit.

Figure 7 (c) Hub used for SUV-2 Rotor

The mass flow rates were calculated from flow velocity using the following formula:

$\dot{m} = \rho A v$

In the above equation,

$\dot{m}$ = Mass flow rate (kg/s)

ρ = Density of air = 1.15 kg/m3

A = Effective flow measuring area of anemometer = 2.7×10^{-3} m2

v = Flow velocity (m/s)

The mass flow rates measured have been tabulated for the rotor speed of 572.88 rpm below:

Table 3 Experimental Results

Rotor Type	Flow Velocity (m/s)	Mass Flow Rate (kg/s)
SUV-2	2.8936	0.0127
Sedan	2.8852	0.008958
Hatchback	1.8684	0.005801

The comparison of mass flow rate through the rotor obtained numerically and experimentally is shown below:

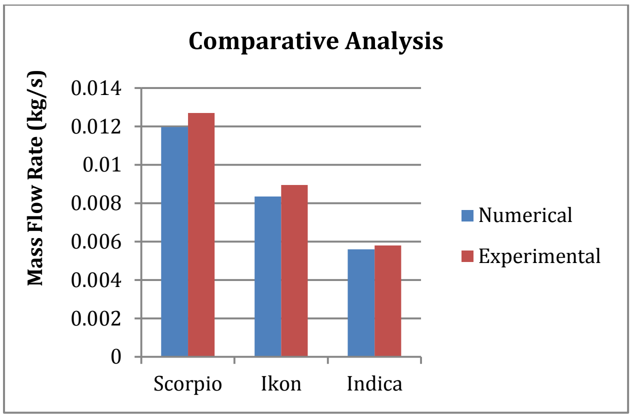

Figure 8 Comparative Analysis

Thus it is seen that the experimental results are in very good agreement with the numerical results for the mass flow rate calculations.

Effect of Long-Short Vane Combinations

The next parameter studied was effect of different combinations of alternate long and short vanes. The disc brake model used for analysis was the sedan model which has 40 straight radial vanes. It has recurring pattern of one long and four short vanes. CFD analysis was carried out to study the comparative analysis between combinations such as all short, all long and alternate long and short vanes. Geometrical construction of all three combinations is shown in Fig. 10 below:

(a) All short (b) All Long (c) Alternate Long-Short

Figure 9 Vane Combinations

CFD analysis of the above combinations was carried out to obtain mass flow rate which are tabulated below:

Table 4 Numerical Analysis

Vane combinations	Mass Flow Rate (kg/s)
All short	0.008428
All long	0.008500
Alternate long-short	0.008248

The combination of all long vanes gives more mass flow rate but as the vane geometry is closer to the inlet. The alternate long-short combination gives reduced mass flow rate due to reduced high pressure drop at inlet. This is because of the non-uniform inlet passage formed by alternate short vanes. Also, due to the angular inlet velocity, the air passing over the longer fin shifts to the next flow passage rather than flushing through the adjacent flow passage. This is illustrated by the pressure contours in Fig. 10.

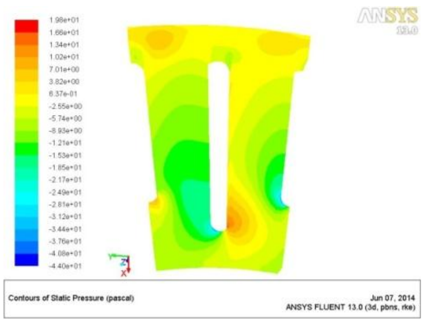

Figure 10 Pressure Contour (Alternate Long-Short)

Results and Discussion

The number of vanes is the parameter which decides the amount of area available for the air to flow through the vane passage of the disc brake rotor. However, there is a limit for increasing the number of vanes due to increased friction losses and weight considerations. The number of vanes were varied from 30 to 56 for the SUV-1 VDBR and the results for total mass flow rate from the rotor were obtained. In the optimized design with 40 numbers of vanes, the mass flow rate increased by 10. 82% while than that of with 36 numbers of vanes.

The comparative study between the numerical and experimental mass flow rates through the various rotors was done. The deviation of experimental values from numerical values of mass flow rate for SUV-2, sedan and hatchback was found to be 6.09%, 7.2% and 3.33% respectively. Thus, the experimental results were found to be in very good agreement with numerical results.

In the CFD analysis of combination patterns of all short, all long and alternate long-short vanes, the combination of all long vanes gave the highest mass flow rate. The proposed modification of using all short vanes in the optimized design (mass flow rate = 0.008428 kg/s) gives better results than the original design (mass flow rate = 0.008356 kg/s).

Conclusion

1. The study of different geometries of rotors like circular pillar post vanes (hatchback), axial inlet (SUV-2), inclined radial vanes (SUV-1) and straight radial vanes (sedan) was carried out. Thus, we covered varied designs of disc brake rotors belonging to different classes of automotive vehicles (Sedan, SUV and Hatchback). From our study, it can be concluded that reduction in recirculation zones will increase mass flow through the rotor.
2. The study of axial inlet rotor showed that axial inlet increases the mass flow. The analysis of inclined vanes showed that such rotors utilize the air flow more effectively than the radial ones. But the inclination of vanes being opposite for left and right side, the inventory and manufacturing cost is increased.
3. For SUV-1, the model with 40 vanes is proposed to be better than the original model with 36 vanes.
4. The analysis for straight radial vane VDBR combinations like all short, all long and alternate long-short vanes showed better mass flow with all long vanes, which is better than the original model with a pattern of alternate one long and four short vanes.

References

Parish, D. (2003) Automotive Ventilated Brake Rotor Aerodynamic Investigation. *MSc Thesis, School of Engineering, Cranfield University.*

Gautam Pulugundla (2008) CFD Design Analysis of Ventilated Disc Brakes. *MSc Research Thesis, School of Engineering, Cranfield University.*

Kapil Shelar, Mahesh Chopade (2013) Thermal Analysis of Automotive Brake Rotors. *International Conference on Design, Manufacturing and Mechatronics*.

S. Manohar Reddy, J. M. Mallikarjuna and V. Ganesan (2008) Flow and Heat Transfer Analysis of a Ventilated Disc Brake Rotor Using CFD. *SAE International ISSN 0148-7191.*

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

Kwangjin Lee (1999) Numerical Prediction of Brake Fluid Temperature Rise during Braking and Heat Soaking. *SAE 1999-01-0483. Page 8.*

J M Mallikarjuna S Manohar and V Ganeshan (2008) Flow And Heat Transfer Analysis of A Ventilated Disc Brake Rotor Using CFD. *SAE 2008, page 1, 9*.

A.D. McPhee (2008) Experimental Heat Transfer and Flow Analysis of a Vented Brake Rotor. *International Journal of Thermal Sciences 2008, page 4, 7.*

J M Mallikarjuna S Manohar and V Ganeshan (2006) Effect of Vane Shape on Flow and Heat Transfer Characteristics of a Brake Disc. *BSME-ASME International conference on Thermal Engineering, page 4, 5.*

David Phan *et. al.* (2003) Rotor Design and Analysis; A Technique using CFD and Heat Transfer Analysis. *SAE International, page 1, 4.*

Lisa wallis (2002) Air Flow and Heat Transfer In Ventilated Disc Brake Rotors With Diamond and Tear-Drop Pillars. *Numerical heat transfer, part A, 41: 643-655, page 9, 10.*

A. Nejat *et. al.* (2011) Heat Transfer Enhancement in Ventilated Brake Disk Using Double Aerofoil Vanes. *Journal of Thermal Science and Engineering Applications vol.3, Pages 8, 9.*