Int. J. Mech. Eng. & Rob. Res. 2015

International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 4, No. 1, January 2015

Research Paper

NUMERICAL STIMULATION OF VENTILATED DISC COOLING EFFECT

Kiran C H[1]*

**Corresponding Author: Kiran C H,* ✉ *ckm.kiran1986@gmail.com*

Braking system is one of the important control systems of an automotive. For many years the disc brakes have been used in automobiles for safe retardation of the vehicles. During braking enormous amount of heat will be generated and for effective braking sufficient heat dissipation is essential. The thermal performance of disc brake depends upon the characteristics of airflow around the brake rotor and hence the aerodynamics is an important in the region of brake components. This project aims at maximizing the airflow distribution across the rotor for better heat dissipation and improving the cooling efficiency. A CFD analysis is carried out on the Skoda Octavia car braking system as a case study to make out the behaviour of airflow distribution around the disc brake components using FLUENT software. The result obtained from this analysis gives an insight idea of the airflow distribution around the brake rotor in order to minimize the temperature that affects the braking performance. Based on the results obtained, three new concepts were generated by incorporating air ducts guiding the air towards brake rotor to enhance the cooling effects. The results obtained for all the cases are analysed for effective cooling. From the results of temperature distribution it was observed that there is a considerable reduction in the max temperature generated during braking. 24 °C (611 °C to 587 ºC) decreased in maximum temperature for concept1 and 41 °C for concept2 by properly guiding air towards brake rotor and 110 °C decreased was observed in cocept3 is achieved.

Keywords: Ventilated disc brake, Heat dissipation, CFD, Air flow, Cooling effect

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

[1] Department of Mechanical Engineering, Azad Institute of Engineering and Technology, Azad Technical Campus, Near CRPF Camp., Lucknow 226002, India.

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.

Excessive thermal loading can result in surface cracking, judder and high wear of the rubbing surfaces. High temperatures can also lead to overheating of brake fluid, seals and other components

Braking system is one of the most essential mechanisms of a vehicle as shown in Figure 1. It is implicit that the braking system must be able to eliminate the kinetic and potential energy to facilitate a safe deceleration. Generally, the methodologies like regenerative-braking and friction-braking system are used in a vehicle. Due to limitations of Regenerative-braking system, friction based braking system is universally adopted for retardation of vehicles. Friction-brakes function by transforming the vehicles kinetic and potential energy into heat energyThe rate of heat generation in a friction-braking system is a function of the vehicles mass, velocity and rate of deceleration. When the brakes are applied a large amount of heat is generated in a brake system accordingly the surrounding brake components has to absorb the heat within a shorter period of time, but it is capable of storing only a limited amount of heat produced during braking. In addition, the efficient dissipation of heat is instrumental to the performance of the braking system.

If the temperatures go too high, problems in braking system crop up from wear and tear of components by screeching and rapid vibration to permanent dysfunction of braking

Figure 1: Shows the Layout of Braking System

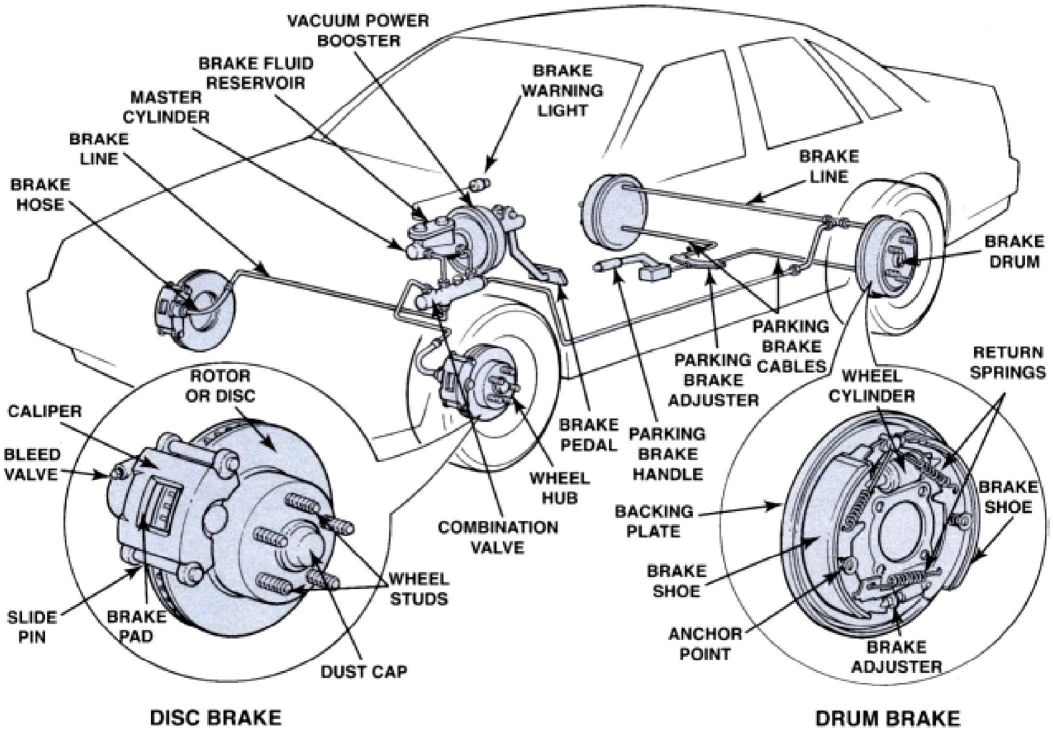
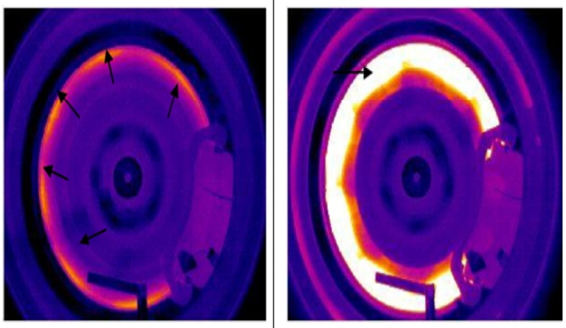
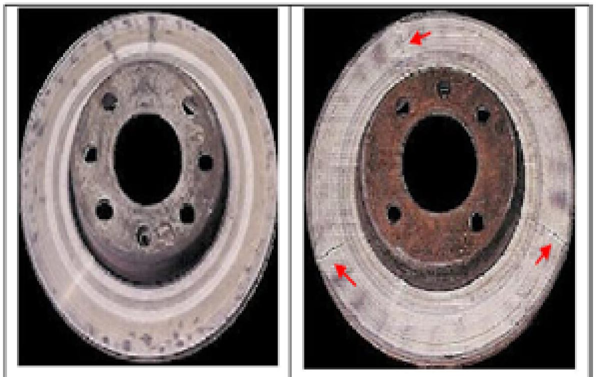


Figure 2: Shows Heat Generated Between Rotor and Pad



(a) Initial stage (b) Repeated braking

Figure 3: Shows Overheating of Brake Pad



(a) Hot spots (b) Cracks

systems. Figure 2 shows the heat generated at the initial stage of brake rotor and pads comes in contact. When a sudden brake is applied for a vehicle moving at a high velocity due to friction, more heat is generated during frequent braking is shown in Figure 3. Some of the factors that affect disc brake due to high temperature are brake fade; thermal judder and excessive component wear. Therefore, it is beyond doubt that the methods of cooling the components of a brake system are to be improved to minimize the risk of the problems and enable safer vehicular movement.

Though at first, the heat is absorbed by the surrounding components, later, continued braking radiates heat to the adjoining components through conduction and convection through atmosphere. Conduction is an effective method of heat dissipation but certain components get adversely affected. Sometimes when high temperatures are not controlled, they damage the tires of the vehicle. Therefore, convection to the atmosphere is the principal means to dissipate heat from the brake rotor. Normally, the forward movement of the vehicle directs the cooling air at the brake to transfer convection heat. So in order to achieve maximum cooling of the brakes, airflow must be regulated and directed to appropriate areas.

LITERATURE REVIEW

Much attention has been focused on improving the thermal performance of brake discs. Numerical simulations and Computational Fluid Dynamics (CFD) are commonly applied to brake disc thermal performance analyses. Many experimental studies have also been conducted to measure the air flow and temperature field inside the discs under braking operations. It has been demonstrated that CFD simulation results have achieved good agreement with those based on experimental studies. By study of literature, we came to know that, Numerical simulation investigation can be done by properly guiding the airflow from the front end of a car towards the wheel. Conventional method is considered as the best method to dissipate heat because it's dependent on speed which can be controlled. Improvements can be done by modifying the dimensional characteristics of ventilated vane disc rotor to enhance good airflow and heat dissipation characteristics. The component of disc brake assembly tries to transfer better heat to the surrounding air. The front end design approach influence the importance of bumper and grill design. This project aims at maximizing the airflow distribution across the rotor for better heat dissipation and improving the cooling efficiency.

ANALYTICAL MODEL

For fluid flow and heat transfer analysis across brake rotor, the governing equations, such as continuity equation, momentum (Navier Stokes) equation and energy equations are used in CFD for solving the solution

Continuity Equation: A continuity equation is a based on the principle of conservation of mass. For steady state it states that the mass of fluid entering a fixed control volume either leaves that volume or accumulates within it is constant. It is thus a "mass balance" requirement posed in mathematical form, and is a scalar equation.

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho.u)}{\partial x} + \frac{\partial(\rho.v)}{\partial y} + \frac{\partial(\rho.w)}{\partial z} = 0 \qquad ...(1)$$

Momentum (Navier Stokes) Equations: The momentum equation is a statement of Newton's second law and relates the sum of force acting on an element to its acceleration. Hence *F* = *ma* which forms the basis of the momentum equations. The three different momentum equations (*x*, *y* and *z*), altogether comprise the Navier Stokes equations that describe the flow of incompressible fluids.

$$\rho\frac{\partial u}{\partial t} + \rho u\frac{\partial u}{\partial x} + \rho v\frac{\partial u}{\partial y} + \rho w\frac{\partial u}{\partial z} = \rho g_x - \frac{\partial p}{\partial x} + \mu\frac{\partial^2 u}{\partial x^2} + \mu\frac{\partial^2 u}{\partial y^2} + \mu\frac{\partial^2 u}{\partial z^2}$$

$$\rho\frac{\partial v}{\partial t} + \rho u\frac{\partial v}{\partial x} + \rho v\frac{\partial v}{\partial y} + \rho w\frac{\partial v}{\partial z} = \rho g_y - \frac{\partial p}{\partial x} + \mu\frac{\partial^2 v}{\partial x^2} + \mu\frac{\partial^2 v}{\partial y^2} + \mu\frac{\partial^2 v}{\partial z^2}$$

$$\rho\frac{\partial w}{\partial t} + \rho u\frac{\partial w}{\partial x} + \rho v\frac{\partial w}{\partial y} + \rho w\frac{\partial w}{\partial z} = \rho g_z - \frac{\partial p}{\partial x} + \mu\frac{\partial^2 w}{\partial x^2} + \mu\frac{\partial^2 w}{\partial y^2} + \mu\frac{\partial^2 w}{\partial z^2} \qquad ..(2)$$

Energy equations: The continuity and momentum equations are adequate in situations, where the fluid is incompressible and the temperature differences are small. If the heat flux occurs (temperature not constant) an additional equation (energy) is enabled. An energy equation is a scalar equation. It has no particular direction associated with it. This equation demonstrates that, per unit volume, the change in energy of the fluid moving through a control volume is equal to the rate of heat transferred into the control volume plus the rate of work done by surface forces plus the rate of work done by gravity.

$$\frac{\partial}{\partial t}\left(\rho e + \frac{1}{2}\rho v^2\right) + \frac{\partial}{\partial x}\left(\rho u e + \frac{1}{2}\rho u v^2\right) + \frac{\partial}{\partial y}\left(\rho v e + \frac{1}{2}\rho v v^2\right) + \frac{\partial}{\partial z}\left(\rho w e + \frac{1}{2}\rho w v^2\right) =$$

$$k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial xz^2}\right) - \left(u\frac{\partial p}{\partial x} + v\frac{\partial p}{\partial y} + w\frac{\partial p}{\partial z}\right) +$$

$$\mu\left[u\frac{\partial^2 u}{\partial x^2} + \frac{\partial}{\partial x}\left(v\frac{\partial v}{\partial x} + w\frac{\partial w}{\partial x}\right) + v\frac{\partial^2 u}{\partial y^2} + \frac{\partial}{\partial y}\left(v\frac{\partial u}{\partial y} + w\frac{\partial w}{\partial y}\right) + w\frac{\partial^2 u}{\partial z^2} + \frac{\partial}{\partial z}\left(v\frac{\partial v}{\partial z} + u\frac{\partial u}{\partial z}\right)\right]$$

$$+2\mu\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial u}{\partial y}\frac{\partial v}{\partial x} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial v}{\partial z}\frac{\partial w}{\partial y} + \frac{\partial^2 w}{\partial z^2} + \frac{\partial w}{\partial x}\frac{\partial u}{\partial z}\right] + \rho u g_x + \rho v g_y + \rho w g_z \qquad ...(3)$$

Turbulence Model: A standard k- model is selected with reference to Rolf Krusemann and Gerald Schmidt (1995). It is a two-equation turbulence model derived from Reynolds Averaged Navier Stokes modeling. *K* is the turbulent kinetic energy, defined as the variance of the fluctuations in velocity and ε is the turbulence eddy dissipation rate at which the velocity fluctuations dissipate. This semi empirical model is robust and economic, most widely used in industrial and practical engineering turbulent flow problems, also suitable for parametric studies. The turbulence kinetic energy (*k*) and rate of dissipation (ε) is obtained from the following transport equations;

For turbulent kinetic energy:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_t}\right)\frac{\partial k}{\partial x_j}\right] + G_x - \rho\varepsilon + S_k \quad ...(4)$$

And for rate of dissipation:

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right) + \frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \quad ...(5)$$

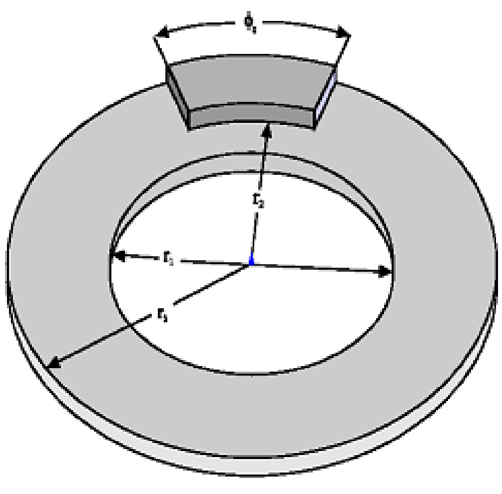
The turbulent viscosity, is computed by combining K and as follows,

$$\mu_t = \rho C\mu \frac{k^2}{\epsilon} \quad ...(6)$$

Brake Heat Transfer

Figure 4 shows the schematic shape of the disk and the pad in sliding contact is shown. As it is shown disk is like an annulus and pad is like a partial annulus. The brake system clamps the pads through the calliper assembly by brake fluid pressure in the cylinders. Rotary motion of the disk causes a sliding contact between the disk and the pad and generates heat.

Figure 4: Shows Schematic Shape of the Disk and the Pad in Sliding Contact



For calculation of heat generation due to friction, rate of dissipated heat via friction should be taken into account. This is all to do with the calculation of friction force and rate of work done by friction force. For calculation of friction force, the pressure distribution at the contact surface of the disk and the pad should be determined. Here, two types of pressure distribution are taken into account.

In the contact area of brake components; the pads and the disk; heat is generated due to friction. For calculation of heat generation at the interface of these two sliding bodies' two methods is suggested:

1. At the basis of law of conservation of energy the kinetic energy of the vehicle during motion is equal to the dissipated heat after vehicle stop.
2. By knowing the friction coefficient, pressure distribution at the contact area, geometric characteristics of the pad and the disk, relative sliding velocity and duration of braking action one can calculate the heat generated due to friction.

Brakes are essentially a mechanism to change the energy types. When a car is moving with speed, it has kinetic energy. Applying the brakes, the pads or shoes that press against the brake drum or rotor convert this energy into thermal energy. The cooling of the brakes dissipates the heat and the vehicle slows down. This is all to do with the first law of thermodynamics, sometimes known as the law of conservation of energy that states that energy cannot be created nor destroyed; it can only be converted from one form to another. In the case of brakes, it is converted from kinetic energy to thermal energy:

The different modes of heat dissipation are;

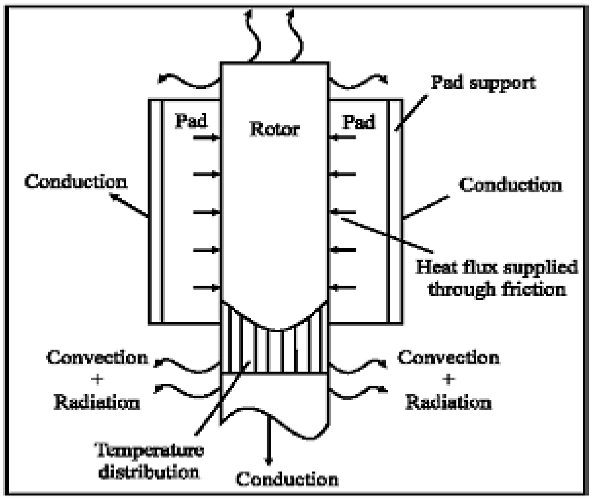
1. Conduction through the brake assembly and hub
2. Radiation to nearby components
3. Convection to the atmosphere

Conduction is an effective mode of heat transfer but it affects the adjoining components in terms of damaging the seals, bearing, etc.

Radiation heat transfer has the maximum effect of temperature that is to be controlled and it is estimated as negligible during normal braking conditions. So the convection is considered as primary means of heat dissipation from the brake rotor to the atmosphere. The Figure 5 shows the schematic form of heat transfer mechanism in a disc brake system.

As for conduction through surfaces, the heat flow can be expressed by Fourier's law of conduction as follows:

Figure 5: Shows the Heat Transfer Mechanism in Brake System



$$Q_{cond} = -kA\frac{dT}{dx} \quad \text{...(7)}$$

where, Q_{cond} is the heat transferred, k is the thermal conductivity, A is the area of contact and T is the temperature.

Similarly, the heat is transferred by convection, also known as Newton's law of cooling is governed by,

$$Q_{conv} = hA(T_s - T_\infty) \quad \text{...(8)}$$

where, Q_{cond} = Rate of heat transfer (W)

h = convection heat transfer coefficient (W/m^2K)

A = surface area of the rotor (m2)

T_s = surface temperature of the brake rotor (°C)

T_∞ = Ambient air temperature (°C)

GEOMETRIC MODELING AND CFD ANALYSIS

In this chapter the study of airflow distribution is carried out on the existing disc brake of Skoda Octavia passenger car as case study in order to reduce the temperature by guiding air towards brake rotor. The overall dimensions of car body and brake components are illustrated in Table 1.

Calculation for Input Parameters: The heat flux is calculated for the car moving with a velocity 22.22 m/s (80 kmph) and the following is the calculation procedure.

Data given:

Mass of the vehicle = 1330 kg (Table 2)

Initial velocity (u) = 22.22 m/s (80 kmph)

Final velocity (v) = 0 m/s

Tables 1: Shows the Specification of Skoda Car

All Dimension are in mm	
Overall length	4512
Overall width	1731
Overall height	1431
Wheel base	2512
Ground clearance	134
Front track	1513
Rear track	1494
Kreb weight	1330 Kg
Front brakes	Disc brakes
Rear brakes	Drum brakes
Front suspension	McPherson Sturt with wish bone arms
Rear suspension	Compound link crank axle with torsion stabilizer
Brake disc (size)	270 Dia x 22
Brake pads	115 x 50 x 20
Tires	196/65 R15
Wheel size	6J x 15"

Brake rotor diameter = 0.270 m

Assuming, full load condition (100% braking condition) Energy generated during braking = 44324.46J

Stopping distance (D) = 35.94 m

Heat generated/volume (q) = 1.5×10^7 W/m^3

Geometric Model

The vehicle used for the simulation is Skoda Octavia 1.9TDI, a four door passenger car. Knowing the flow is to be analysed the geometric modelling is constructed using CAD software tool (CATIA V5R16) as shown in Figures 6 and domain creation is shown in Figure 7.

Meshing (Pre-Processor)

To output the solution, the above tasks are carried out with the interaction between the user and the computer. This stage is done with the software Hyper Mesh, linked to the

Figure 6: Shows 3D Geometric Model of Car

Figure 7: Shows the Domain Creation

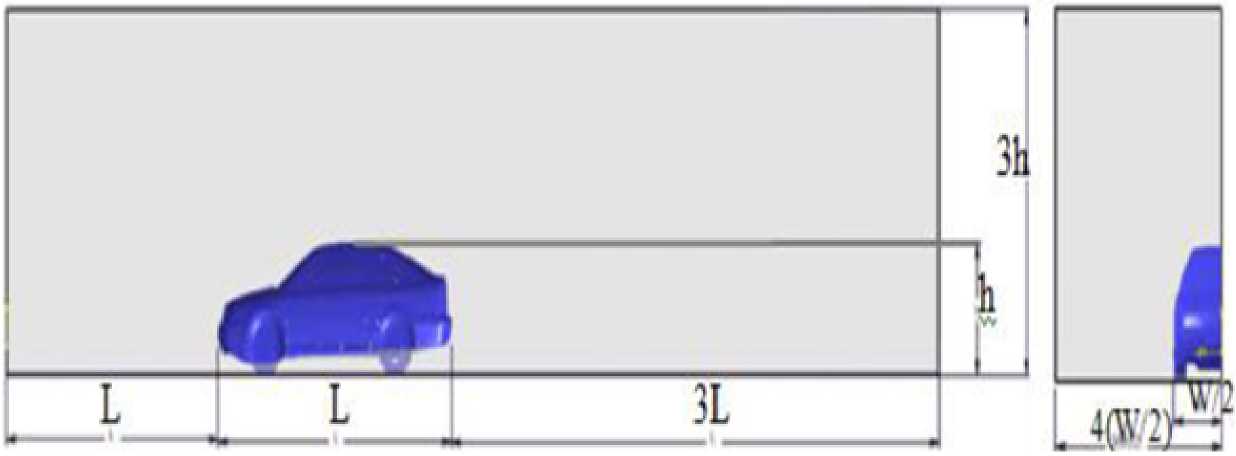


Figure 8: Shows the Fine Mesh Around the Wheel and Disc Brake

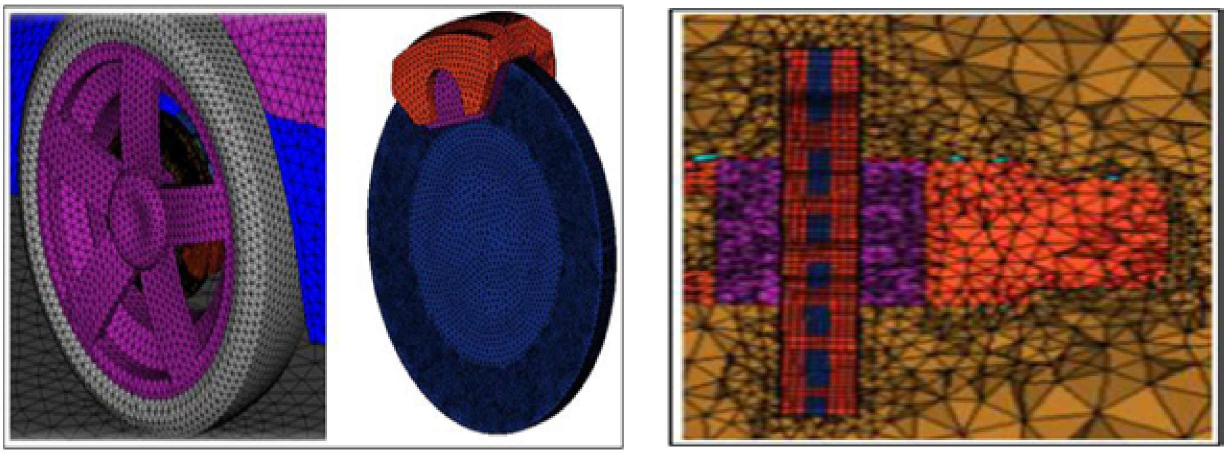
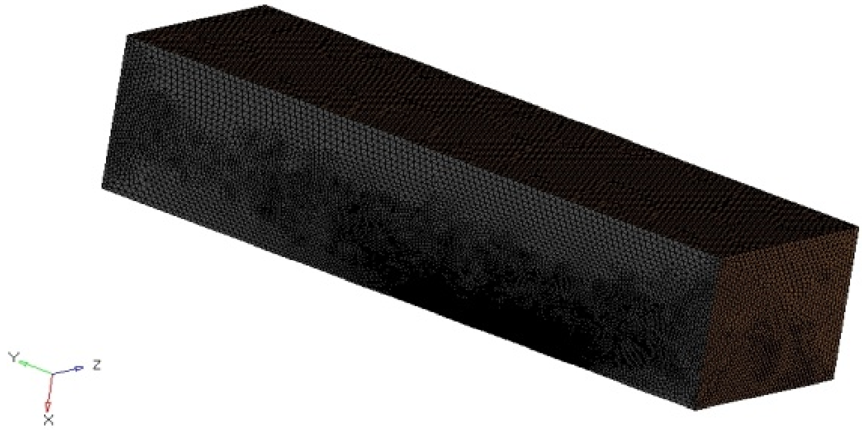


Figure 9: Shows the Fine Tetra Mesh Around the Surface of Body

Figure 10: Shows the Complete Meshing of Domain



FLUENT software are shown in Figures 8, 9 and 10.

CFD Analysis

CFD is primarily used as a design aid for predicting the performance characteristics of equipment involving fluid flow and heat transfer. It's capability to achieve fast and reliable convergence by solving the equations precisely.

Solver Setting and Importance

Turbulent intensity and viscosity was to specify the turbulence specification method and is defined as the ratio of root mean square of the velocity fluctuations to the mean flow velocity expressed by,

$$I = 0.16(R_e)^{-\frac{1}{8}} \quad ...(9)$$

Reynolds number

$$R_e = \frac{\rho \nu l}{\mu} \quad ...(10)$$

where 'ρ' is the density of fluid (Kg/m^3)

'ν' is the velocity of fluid (m/s)

'l' is the length of the obstacle (m)

'μ' is the co-efficient of viscosity

The Reynolds number and Mach number is calculated and mentioned in the following Table 2.

Table 2: Illustrates the Solver Parameters

For Velocity 22.22 m/s	
Mach number	0.1
Reynolds number	686497
Turbulent intensity	2.6

Table 3: Shows the Overall Dimension of Domain Creation

Domain length	22500 mm
Domain width	6800 mm
Domain height	4500 mm

Table 4: Shows the Input Parameter for Solution

Solver	Segregated
Formulation	Implicit
Time	Steady state
Velocity formulation	Absolute
Pressure discretization	Standard
Momentum discretization	First order upwind
Turbulent kinetic energy	First order upwind
Specific dissipation rate (omega)	First order upwind
Pressure velocity coupling	SIMPLE

Table 5: Shows the Material Properties of Brake Rotor and Pad

Component	Brake Rotor	Brake Pad
Material	Grey cast iron	Asbestos
Density Kg/m^3	7100	3500
Co-efficient of heat (C_p) j/Kg-°K	45	800
Thermal conductivity (K) W/m-°K	46	4

The material properties for disc rotor and pad such as thermal conductivity, specific heat and density are illustrated in the following Table 5.

Solution for Existing Model

Figure 13 shows the contours of static temperature distribution for the baseline model across the brake rotor. It is observed that the maximum temperature 610.64º.

Figure 11: Shows the Boundary Condition

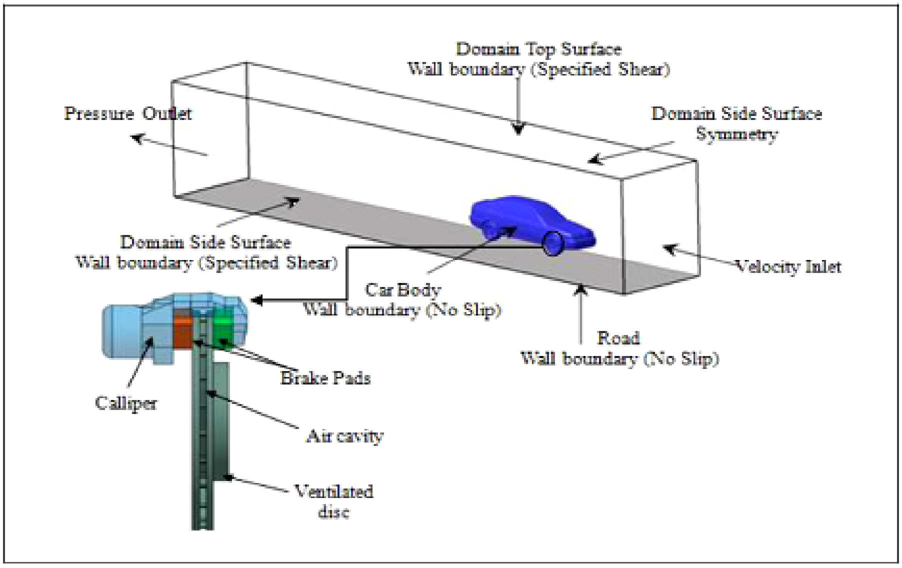


Figure 12: Shows Velocity Vectors for Baseline Model at a Velocity 22.22 m/s

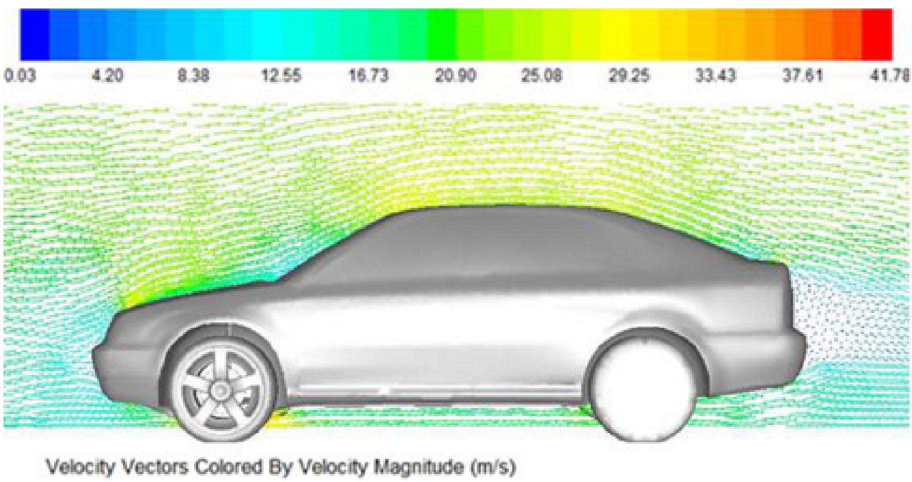


Figure 13: Shows Airflow Pattern at the Lateral Section and Temperature Contours of Ventilated Brake Rotor

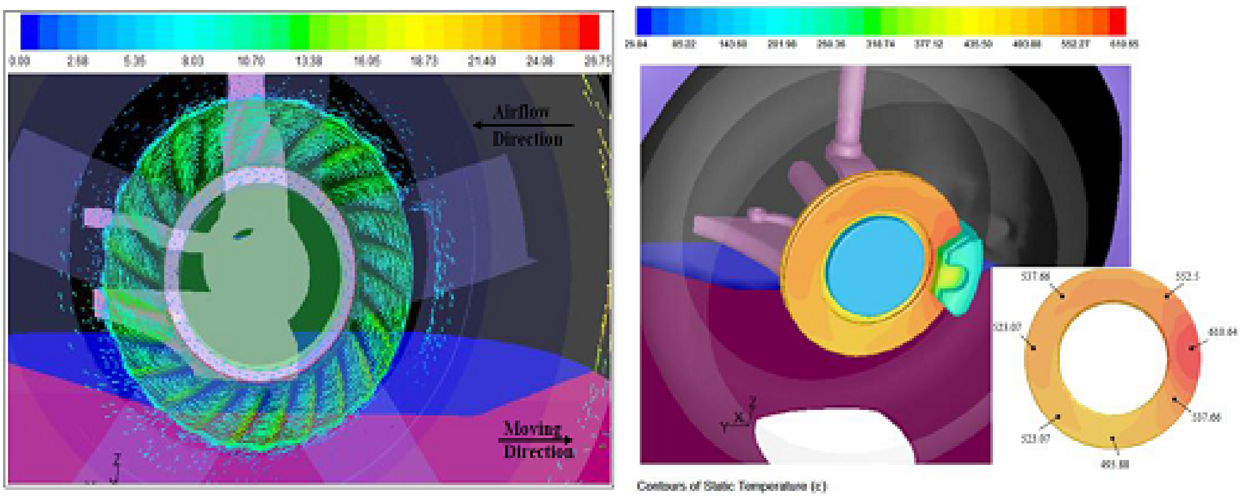


Figure 14: Shows Meshing of Concept1

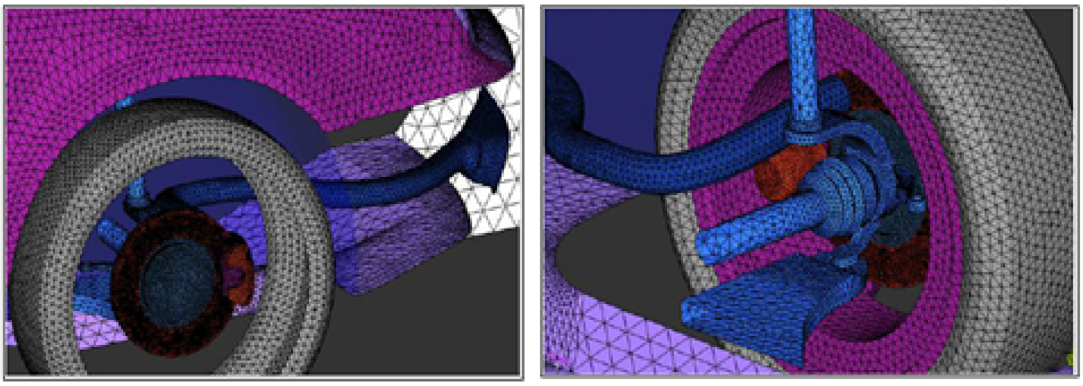
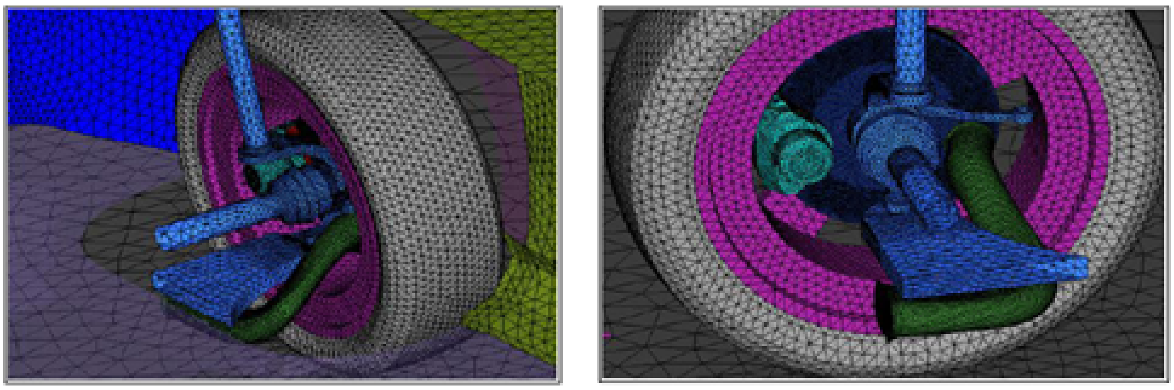


Figure 15: Shows Meshing for Concept2



Approach of Design Improvement

Due to friction the higher temperature in the brake rotor thermal stress are generated that affects higher thermal stresses are induced at high temperature that affects the performance during braking. The affects may leads to crack, brake judder, spots, etc., and in order to minimize the high temperature, the three different concepts are generated inconsideration of brake cooling performance. The concepts are shown in Figures 14 and 15 and concept3 is as same geometric of concept2 but change of material of brake rotor to ALMMC.

RESULTS AND DISCUSSION

Figure 16 shows the contours of static temperature distribution for the baseline model across the brake rotor. The maximum temperature observed is 610.64 ºC in between the pad and the rotor.

Figure 17 shows the contours of static temperature distribution for the modified model across the brake rotor. The maximum temperature observed is 586.21 ºC in between the pad and the rotor.

Figure 16: Shows Contours of Temperature Distribution Across Baseline Model

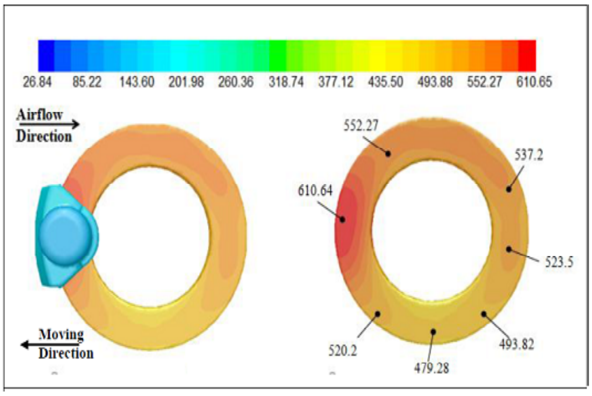


Figure 17: Shows Contours of Temperature Distribution Across Concept1

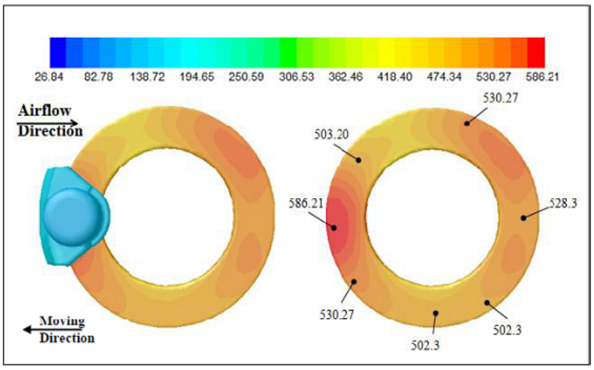


Figure 18: Shows Contours of Temperature Distribution Across Concept2

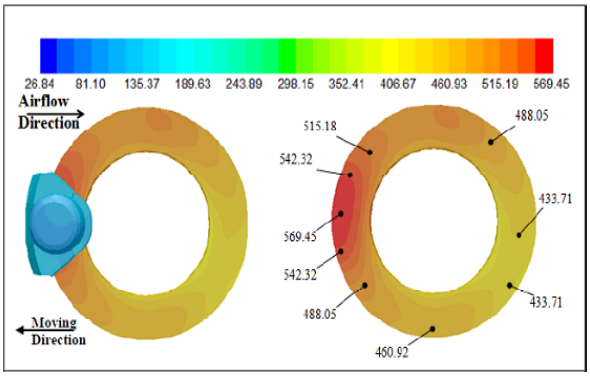
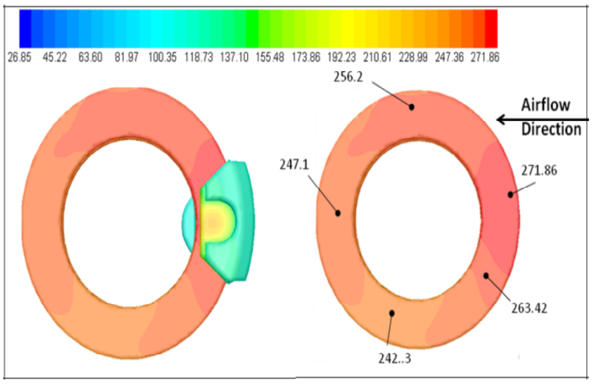


Figure 19: Shows Contours of Temperature Distribution Across Concept3



The Figure 18 shows the contours of static temperature distribution for the modified model across the brake rotor. The maximum temperature observed is 569.45 ºC in between the pad and the rotor.

Figure 19 shows the contours of static temperature distribution for the concept3 model across the brake rotor. The maximum temperature observed is 271.86 ºC in between the pad and the rotor.

CONCLUSION

A detailed study of the flow across the disc brake rotor and the suggestion for the new design concepts has been carried out. Following are the conclusions based on the analysis results.

The results achieved through CFD with acceptable accuracy for the baseline and the modified concepts help in understanding and visualizing the airflow across the brake rotor. The brake rotor serves as energy dissipater, so to achieve this better additional air is guided to provide an adequate cooling. The modified design concepts are found to be effective in terms of temperature reduction with increased airflow. Even a material change in the brake rotor yields better results than the modified concepts. Using together the Al metal material matrix composites rotor and the modified concept2 may yield better cooling. Considering the costs of composites material, the concept2 is an appropriate choice for more airflow than compared to concept. Half load result looks more practically than full load results.

From the results of temperature distribution it was observed that there is a considerable reduction in the max temperature generated during braking. 24 °C (611 °C to 587 ºC) decreased in maximum temperature for concept1 (air duct attached from bumper to disc rotor) and 41 °C for concept2 (wish bone type) by properly guiding air towards brake rotor.

SCOPE FOR FUTURE WORK

In addition to steady state analysis transient analysis can be carried out to study the heat transfer across the disc brake. Further the study of the heat transfer around the disc brake rotor may be done considering the radiation. Further investigation through CFD analysis can be carried out considering the rear disc brakto study the airflow distribution.

REFERENCES

1. Christer Bergstrom and Anders Jerhamre (2001), "Numerical Study of Brake Disc Cooling Accounting for Both Aerodynamic Drag Force and Cooling Efficiency", SAE Technical Paper.
2. Del Amno Nieto O, Gonzalez Hernandez M I and Canibano Alvarez E (2006), "Development of a Multidisciplinary Methodology for Brake Disc Design", X International Congress on Project Engineering, September 13-15, Valencia.
3. Galindo-Lopez C H and Tirovic M (2008), "Understanding and Improving the Convective Cooling of Brake Discs with Radial Vanes", Department of Automotive, Mechanical and Structures Engineering, Cranfield University, Cranfield, UK.
4. Gupta R B (2001), *A Text Book of Automobile Engineering*, 6th Edition, Published by Satya Prakashan, New Delhi.

5. Jack Williams (1985), "An Automotive Front-End Design Approach for Improved Aerodynamic Design", SAE Technical Paper.

6. Michael D Hudson and Roland L Ruhl (1997), "Ventilated Brake Rotor Airflow Investigation, University of Illinois at Urbana Champaign", SAE Paper, pp. 10-33.

7. Rolf Krusemann and Gerald Schmidt (1995), "Analysis and Optimization of Disc Brake Cooling via Computational Fluid Dynamics", SAE Paper, pp. 07-91.

8. Talati F and Jalalifar S (2008), "Investigation of Heat Transfer Phenomena in a Ventilated Disc Brake Rotor with Straight Radial Rounded Vanes, Asian Network for Scientific Information", *Journal of Applied Sciences*, pp. 3583-3592.

9. Warren Chan (2007), "Analysis of Heat Dissipation in Mechanical Braking Systems", Department of Mechanical and Aerospace Engineering, University of California, San Diego 2007-07.