iJRASET
International Journal For Research in Applied Science and Engineering Technology

# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

**Volume:** 7 **Issue:** IX **Month of publication:** September 2019

**DOI:** http://doi.org/10.22214/ijraset.2019.9030

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
*ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177*
*Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com*

# Comparative Analysis of Disc Brake Rotor using CFD

Siddhesh D. Bhandari[1], Dr H. P. Khairnar[2]
[1,2]*Department of Mechanical Engineering, Veermata Jijabai Technological Institute, Mumbai –400 019, Maharashtra, India*

***Abstract:*** ***It is the main objective of engineers to reduce weight and maximize safety and performance of automotive components. Braking system is one of the most important systems of an automobile. Disc brake rotor design is an important aspect of brake system design. A significant research is being carried out in improving its performance and reliability. One of the major components of the braking system is the rotor. Research towards modifying the geometry and using different materials has been carried out to improve the braking performance. Material selection plays an important role in performance of the braking system.***

***Nowadays, analysis using CAE (Computer Aided Engineering) many brake designs can be studied and analysed at different operating conditions before actual production. Various aspects of braking performance such as temperature variation, heat dissipation rate, deformation and stress values can be obtained for different operating conditions for different materials and geometries. Through this research various aspects like heat dissipation in terms of heat flux, and heat transfer coefficient have been studied. The CFD (Computational Fluid Dynamics) analysis has been carried out using a pressure based solver in analysis software Ansys Fluent. The aim of this work is to study the effect of varying geometry of the rotor in different operating conditions. The study uses MRF (Multiple Reference Frame) formulation to simulate the air flow over a disc brake close to actual conditions.***

***Keywords: Rotor, CFD, CAE, MRF, Ansys Fluent.***

## I. INTRODUCTION

The main purpose of a brake system is to slow down the vehicle or to stop it completely within a reasonable amount of time. A brake system must therefore be reliable in order to provide the operator with a better control. Any moving vehicle contains kinetic energy by virtue of its motion. The faster the vehicle moves the higher is its momentum and kinetic energy. This energy is proportional to the square of the vehicle's speed. Most brakes use the principle of friction to convert this kinetic energy into heat energy. The brakes must therefore store and dissipate all this heat into the surroundings for subsequent braking stages to have a good braking efficiency.

Bryant studied the thermo elastic behaviour of a disc brake during heavy braking. His work was concerned with developing a design that would enable uniform heating of the disc brake, and more importantly uniform cooling of the disc brake. He modified the vent design of the disc to promote uniform cooling of the disc [1]. Prashant Patel et al studied the influence of vane geometry on the performance of the disc brake. A pillar type vane was compared with straight vane geometry for the transient thermal analysis. The proposed disc brake has lesser mass, and also had higher heat transfer coefficient, owing to better circulation air in straight vanes. It was proposed that a curved vane can further help improve the performance of disc brake [2]. G.Ranjith Kumar et al studied the effect of change in the vane design on the thermal performance of a cast iron disc brake. The proposed design with a different vane shape not only reduced the mass of the disc but also the von mises stress, in addition to that, a greater thermal flux was obtained. Curved vanes performed better than straight vanes in the steady state thermal analysis [3]. Jimit G. Vyas et al studied thermal behaviour of cross drilled and solid disc brake rotors under different braking intervals. From the results it was found that if the average braking cycle is short, solid disc brake is advantageous and if the average braking cycle is long, cross drilled disc brake rotor is more advantageous [4]. Voller et al has also done experimental analysis on a ventilated disc brake rotor and studied the contribution on modes of heat transfer. The author concluded that conduction was essentially independent of the speed of the vehicle and its significance is higher at lower speeds. It was also suggested that radiation is independent of speed whereas convection increases strongly with rotational speed [5]. Shelar et al studied the role of aerodynamics in the cooling of ventilated disc brakes. In the study, thermal convection was analysed, using velocity distribution and temperature contours. Also studied were, heat flux rate, air flow rate, and pressure distribution inside the rotors. It was found that with higher number of vanes the rate of heat transfer increased, with an increase in angular speed [6]. Charles studied the effect of number of vanes on the overall performance

©IJRASET: All Rights are Reserved

of the disc brake rotor. In the analysis, the results obtained showed that increasing vane number after a certain point does not result in increase in heat transfer coefficient or decrease in overall temperature after braking. The recirculation zone caused flow separation and conjointly it ended in energy loss in flow. This recirculation zone was least in case of 16 vanes, and with increase in vanes it increased causing an increase in energy loss and thus lesser performance [7]. More et al assessed the effect of rotational speed on the aero thermal performance. It was observed that the rotor speed has a 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. It has been observed that the total heat transfer rate through the passage increases with increase in the blade speed. This is mainly due to increase in mass flow rate through the passage [8].

In this study, we compare the influence of geometry on brake rotor performance using CFD software Ansys Fluent. The parameters of interest are heat transfer coefficient (*h*) and total heat flux. We also study the velocity distribution and corresponding temperature distribution in the disc.

## II. GEOMETRIC MODELLING

To carry out CFD or FEM analysis of any component, the solid model of the same is essential. The solid model for the same was created in a special CAD package Solidworks. Solidworks is a solid modeller and utilizes a parametric feature based approach to create models and assemblies. Parameters refer to constraints whose values determine the shape or geometry of the model or assembly.

Since the study dealt with heat transfer from the contact surfaces, the hub was not modelled. The addition of hub would have resulted in increase in computational time, and since the study is mainly focused on rotor geometry in terms of drilled and non drilled rotor and vane shape, the hub was not modelled. A total of six geometries were created. The first three geometries have straight vanes, and the other three geometries had tapered vanes. Two different vane shapes were used to study the effect of vane type in cooling of the disc. All the geometries were created in modelling software Solidworks. For the next step of mesh generation, all the models were saved in a 'Parasolid' file format. This allows us to mesh and analyse the geometry in analysis software Ansys.

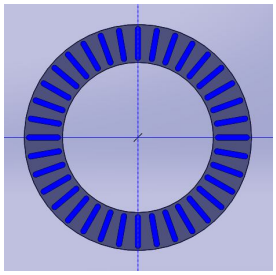


Fig.1 Straight vane disc cross section Solidworks model

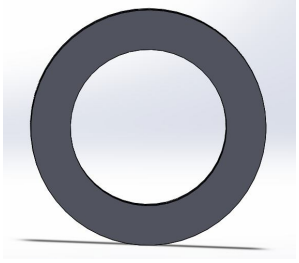


Fig.2 Straight vane disc Solidworks model

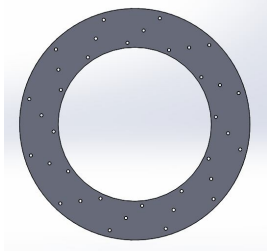


Fig.3 Straight vane three hole disc Solidworks model

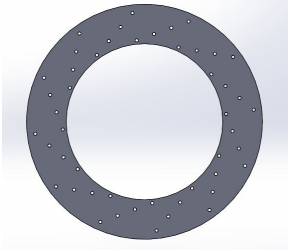


Fig.4 Straight vane four hole disc Solidworks model

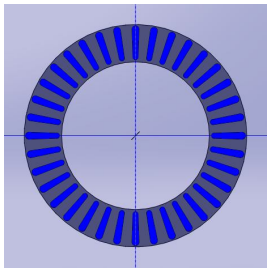


Fig 5. Tapered vane disc cross section Solidworks model

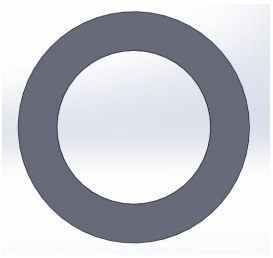


Fig 6. Tapered vane disc Solidworks model

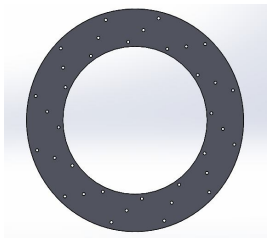


Fig 7. Tapered vane three hole disc Solidworks model

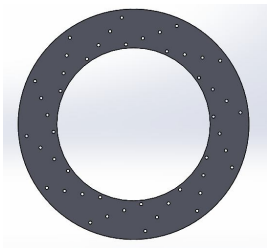


Fig 8. Tapered vane four hole disc Solidworks model

©IJRASET: All Rights are Reserved

## III. MESHING AND BOUNDARY CONDITIONS

The first step in this analysis was to determine the heat transfer coefficient and heat flux from the disc. To achieve this, the disc was analysed in CFD software Ansys Fluent. Prior to meshing we have to import the geometry in Fluent and create a domain.

### *A. Domain*

The bounding box of the object serves as the boundary of the simulation. All fluid objects must be in the domain. Domain size is very important in a CFD analysis. Too big a domain can increase computational effort and time without increasing accuracy of the solution. A smaller domain would reduce computational time, but would also provide us inaccurate solutions. The inlet, outlet, and outer boundaries of the stationary region are placed far enough from the rotor to prevent the full development of the upstream and downstream flow from affecting the results of the analysis. Proper selection of the flow domain upstream and downstream distance is very important to prevent recirculation of the flow that will cause convergence problems. A study of a quadcopter propeller [9] was used to determine the domain dimensions in this analysis. After making the domain, it is necessary to define various parameters like inlet, outlet, wall, disc surface, and rotating and stationary interface. Since this is a case of multiple reference frames, there were two domains, one rotating and other stationary. Also there has to be defined an interface between the two domains, that is rotating and stationary domain. Detailed description of all has been given in the following figures. A Boolean operation was carried out to create to different zones in the domain. Inlet and outlet walls of the stationary in the domain as shown in fig. 9, inlet wall is shown in blue and the outlet wall is shown in green. The wall at the inlet serves as inlet for air flow. The other walls of the stationary domain. are shown in fig. 10 grey define the domain boundaries within which the flow of air is to be considered. In fig. 11, the rotating domain and the disc are shown. The outer cylindrical casing is the rotating domain which is represented by green colour, encloses the blue coloured disc.

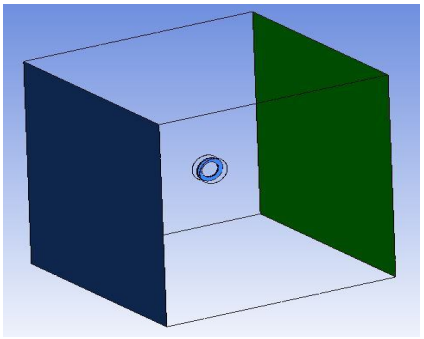


Fig.9 Inlet and Outlet walls of Stationary domain

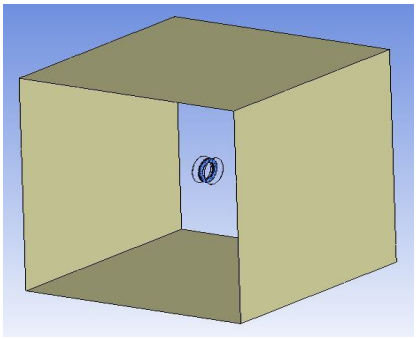


Fig. 10 Walls of Stationary domain

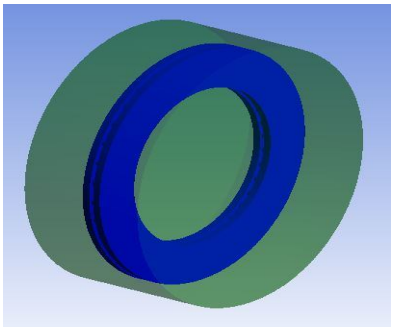


Fig. 11 Disc enclosed in Rotating domain

Table 1 Dimensions for stationary domain

| | |
|---|---|
| Width | 2329.6 mm |
| Height | 2329.6 mm |
| Length | 2168.4 mm |

Table 2 Dimensions for rotating domain

| | |
|---|---|
| Diameter | 281.6 mm |
| Thickness | 120.4 mm |

### *B. Meshing*

Mesh quality is an important aspect of the analysis. In the analysis software fluent improper meshes cannot be analysed. The mesh quality was checked before every analysis and was found to be suitable for the analysis. The orthogonal quality of the mesh obtained was within the required parameters. Mesh should be made as equilateral as possible and the difference in the shape of the mesh from its equilateral form is defined by its skewness. If a quadrilateral mesh is considered then the angle of an equilateral mesh is 90 degrees and for a triangular mesh is 60 degrees. The more the deviations to the equilateral mesh shape the lesser the accuracy of the solution. Universally 0.95 is the maximum allowable skewness ratio and the average skewness can be up to 0.33. The worst cells will have an orthogonal quality closer to 0, with the best cells closer to 1. The minimum orthogonal quality for all types of cells should be more than 0.01, with an average value that is significantly higher. The mesh statistics are as follows

©IJRASET: All Rights are Reserved

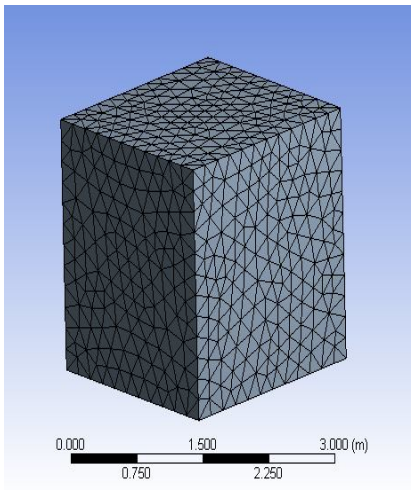



Fig 12. Stationary domain Meshing

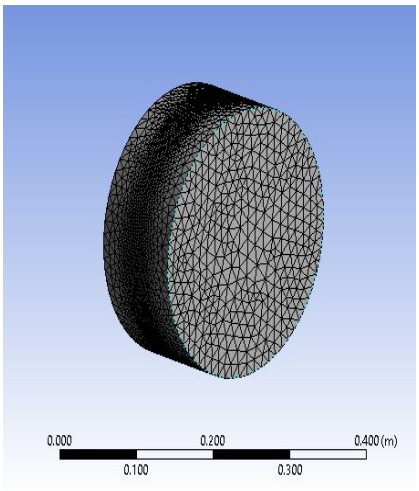



Fig 12. Rotating domain Meshing

Table 3 Meshing Statistics

| Geometry type | Elements | Nodes | Orthogonal quality | Skewness |
|---|---|---|---|---|
| Straight vanes | 1008629 | 198779 | 0.85697 | 0.23159 |
| Four holes | 1393954 | 271354 | 0.85846 | 0.22967 |
| Three holes | 1297725 | 253700 | 0.85819 | 0.23 |
| Tapered Vanes | 995405 | 195601 | 0.85674 | 0.23212 |
| Four holes | 1393954 | 271354 | 0.85846 | 0.22967 |
| Three holes | 1282246 | 250089 | 0.85837 | 0.22994 |

*C. Boundary Conditions*

A total of six geometries were considered for the analysis as explained before. In this analysis three vehicle speeds 80 kmph, 90 kmph and 100 kmph were used. In correspondence to these three speeds three temperatures for the disc were considered 450 K, 550 K and 650 K. The objective was to study how speed affects the change in heat transfer rate in different geometries. The inlet velocities were applied at the inlet wall of the stationary domain as shown in fig. 9. Atmospheric pressure was applied at the outlet wall. The remaining walls are considered in no slip condition and as stationary. The rotating domain was given the rotational speeds as mentioned in table 4.

Table 4 Boundary conditions

| Velocity of incoming air (kmph) | Temperature of disc (K) | RPM of rotor (rpm) |
|---|---|---|
| 80 | 450 | 922.6373 |
| 90 | 550 | 1037.9670 rpm |
| 100 | 650 | 1153.2966 rpm |

Having defined the boundary conditions and generated a suitable mesh, the models for solving the simulation have to be defined. The following models are applied to take convective heat transfer and turbulent flow field into consideration for a steady state simulation.

©IJRASET: All Rights are Reserved

Table 5 CFD parameters

| Parameters | Inputs |
|---|---|
| Solver type | Pressure based |
| Model used | K epsilon, standard |
| Reference values | Computed from inlet |
| Fluid used | Air (incompressible ideal gas) |
| Velocity inlet | 80,90,100 kmph |
| Pressure outlet | 1 atm |

## IV. RESULTS AND DISCUSSION

In this study, the air flow has direction been applied which would simulate the air flow direction moving over a car. As the car moves ahead, the air flow takes place along its length. Similar air flow conditions have been applied to create a realistic simulation. The direction of flow of air is from left to right (fig. 13). The velocity streamlines shown in fig. 13 represents the way in which the air flow takes place, it starts from the inlet wall, and ends at the outlet wall, through the rotating domain. When the streamlines pass through the rotating domain, it can be observed that, the streamlines changes its linear motion to rotational motion. Some of the streamlines at the edge of the rotating domain are simply deflected and continue with the linear motion similar to the flow of air over a cylinder.

The temperature distribution and cooling in the disc is dependent on the air velocity and also the air flow circulation in the disc. Thus the geometry that enables better circulation will have better heat transfer characteristics and will ensure a more uniform temperature distribution. On visual inspection of temperature distribution for various geometries it is clearly visible that the geometries with holes drilled on its surface have better circulation of air and thus enabling uniform cooling.

On visual inspection (Figures 15,16,17,21,22,23) it is can observed that, the temperature in the upper half of the disc is higher as compared to that in the lower half of the disc. This can be attributed to the fact that velocity of air in the upper half is less than that in the lower half. This can be explained by the following equation

$$v = r\omega$$

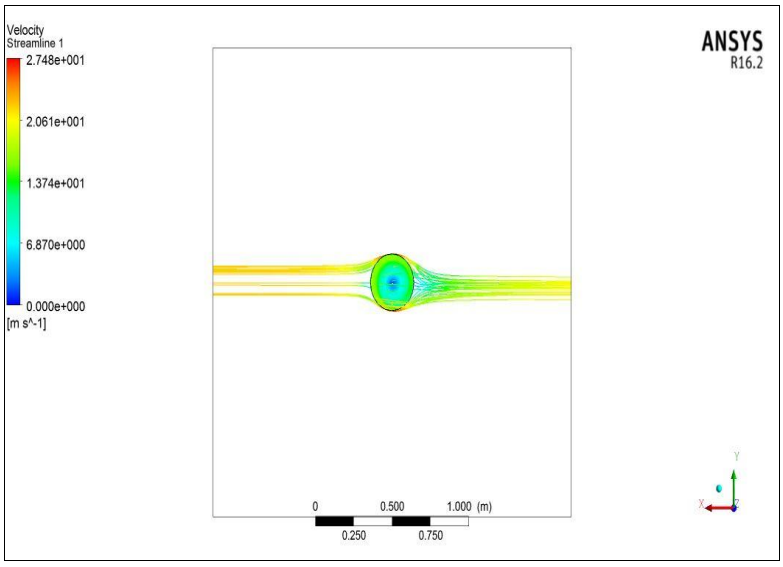



Fig 13. Air flow over disc

©IJRASET: All Rights are Reserved

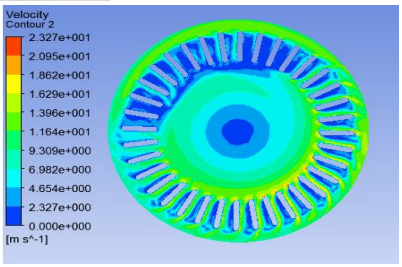



Fig 14. Velocity distribution at mid plane straight vane 450 K

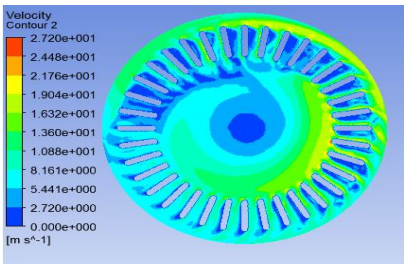



Fig 15. Velocity distribution at mid plane straight vane three holes 450 K

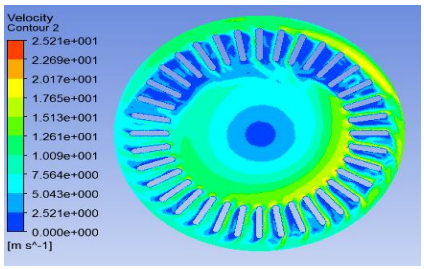



Fig 16. Velocity distribution at mid plane straight vane four holes 450 K

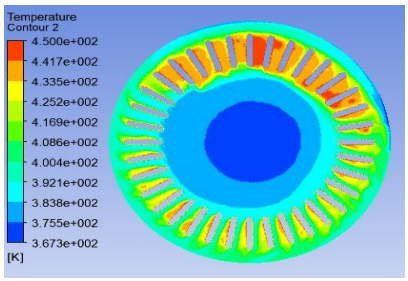



Fig 17. Temperature distribution at mid plane straight vane 450 K

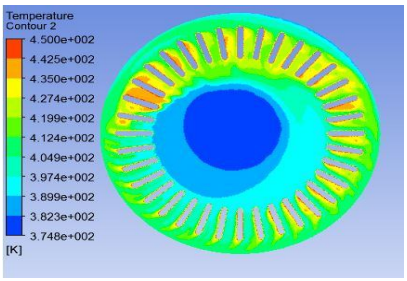



Fig 18. Temperature distribution at mid plane straight vane three holes 450 K

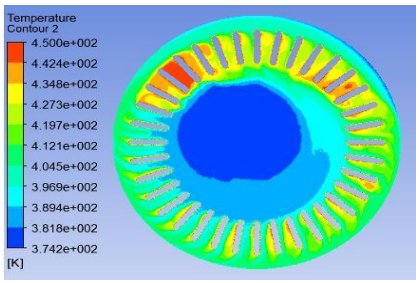



Fig 19. Temperature distribution at mid plane straight vane four holes 450 K

When we consider the upper half, the direction of linear velocity of air, and linear velocity of the rotating disc are in opposite direction, thus the net velocity of air is reduced. And in the lower half, the direction of linear velocity of air, and the linear velocity of the rotating disc add up because of being in the same direction, thus increasing the rate of flow of air, and also with that cooling. Thus more air is inducted in the bottom half and takes away heat rapidly.

The CFD results include the value of maximum heat transfer coefficient and maximum heat flux from the disc. It includes analyses carried out at three temperatures and three speeds. The effect of temperature of disc and speed of vehicle on the temperature distribution can thus be studied and analysed.

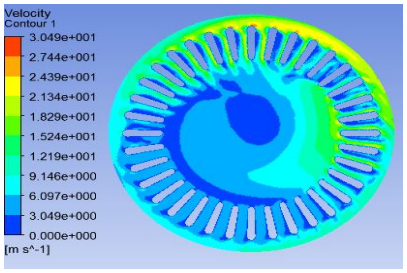



Fig 20. Velocity distribution at mid plane tapered vane 450 K

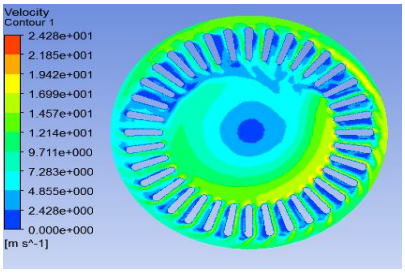



Fig 21. Velocity distribution at mid plane tapered vane three holes 450 K

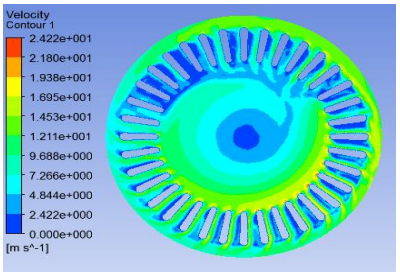



Fig 22. Velocity distribution at mid plane tapered vane four holes 450 K

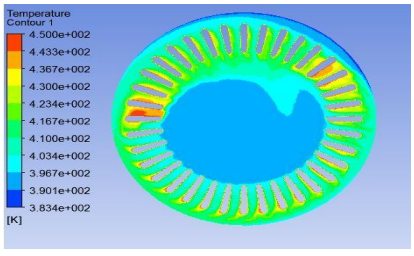



Fig 23. Temperature distribution at mid plane tapered vane 450 K

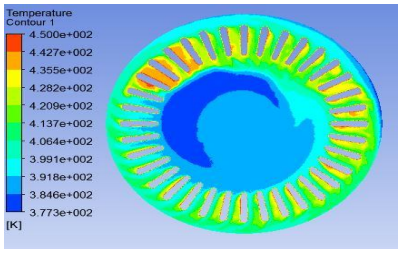



Fig 24. Temperature distribution at mid plane tapered vane three holes 450 K

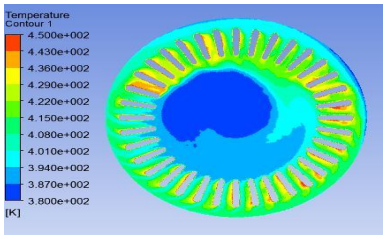



Fig 25. Temperature distribution at mid plane tapered vane four holes 450 K

©IJRASET: All Rights are Reserved

Another thing that can be established from fig.14 and fig. 17 is the relation between velocity contours and temperature contours. If we observe closely the regions in fig.14 where the velocity approaches zero, the temperature is maximum (fig. 17). And also the region where the velocity is maximum, temperature is minimum. The geometries in which the velocity contours are more uniform have corresponding temperature contours as seen in fig.15 and fig.18. This proves that drilling holes is benefitting the performance of the rotor.

### *A. Analysis 1*

The first analysis was carried out at the speed of 80 kmph and 450 K temperature as mentioned in table 4.The CFD results include the value of maximum heat transfer coefficient and maximum heat flux from the disc.

Table 6 CFD results for analysis 1

| Brake type | Total surface heat flux W/m2 | Maximum heat transfer coefficient |
|---|---|---|
| Straight vanes | 10203.1 | 63.0479 |
| Three holes | 10409.25 | 64.31813 |
| Four holes | 10762.45 | 66.50056 |
| Tapered vanes | 10556.52 | 65.28989 |
| Tapered three holes | 10024.06 | 61.93809 |
| Tapered four holes | 9335.021 | 57.68055 |

In the first case it was observed that the maximum heat transfer coefficient was obtained the rotor with straight vanes and four holes. It is quite clear that due to the presence of holes, the air flow and turbulence in the rotor has increased, thus higher value of *h* (Heat transfer coefficient W/m$^2$) is obtained. Also when straight vane rotor is compared to a rotor with tapered vane, it was observed that the value of *h* was higher along with the value of maximum heat flux. Thus by simply changing the shape of the vane, and without drilling holes, better heat dissipation was observed. In this analysis, the rotor with straight vanes and four holes had the best performance.

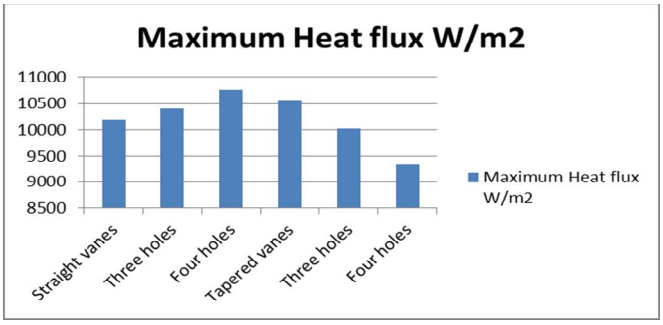



Fig. 26 Heat flux variation with respect to geometry for analysis 1

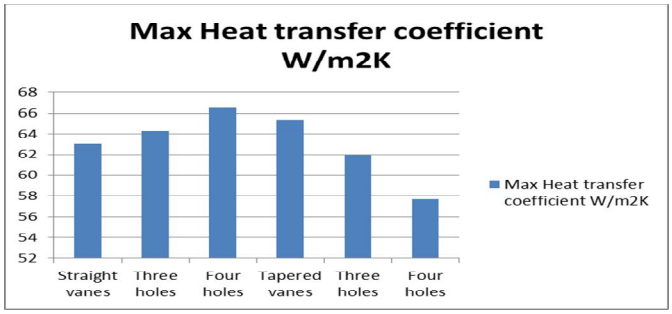



Fig. 27 Heat transfer coefficient variation with respect to geometry for analysis 1

### *B. Analysis 2*

The second analysis was carried out at the speed of 90 kmph and 550 K temperature as mentioned in table 4.

| Brake type | Total surface heat flux W/m2 | Maximum heat transfer coefficient |
|---|---|---|
| Straight vanes | 16338.33 | 62.5891 |
| Three holes | 18435.53 | 70.47062 |
| Four holes | 18700.21 | 71.41847 |
| Tapered vanes | 16648.41 | 63.58239 |
| Tapered three holes | 14348.42 | 59.79844 |
| Tapered four holes | 16992.1 | 64.89499 |

Table 7 CFD results for analysis 2

©IJRASET: All Rights are Reserved

In the second case it was again observed that the maximum heat transfer coefficient was obtained in the rotor with straight vanes and four holes. Thus the presence of holes is actually having an effect on the heat transfer characteristics. Moreover, we also see that when we go from three hole drilled rotor to four hole drilled rotor, the gain in performance is not significant. Thus having three holes can also be considered if the strength of the rotor is an important factor in choosing. Again when straight vane rotor is compared to a rotor with tapered vane, it was observed that the value of *h* was higher along with the value of maximum heat flux. Thus by simply changing the shape of the vane, and without drilling holes, better heat dissipation was observed. In this analysis also, the rotor with straight vanes and four holes had the best performance. However, unlike in the previous case, wherein the rotor with tapered vanes and four holes performed poorly as compared rotor with tapered vanes, here the previous had significantly higher value of maximum heat flux. Also the value of *h* was more. Another thing that is evident from the above two cases is that, in case of rotor with tapered vanes and three holes, its performance is the worst among all the rotor types. Also drilling holes in tapered vanes is not proving beneficial.

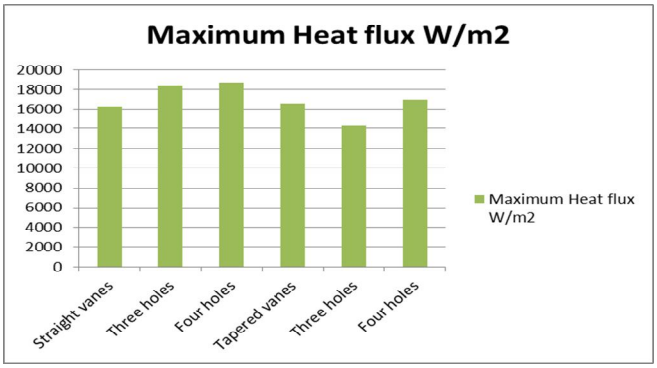



Fig. 28 Heat flux variation with respect to geometry for analysis 2

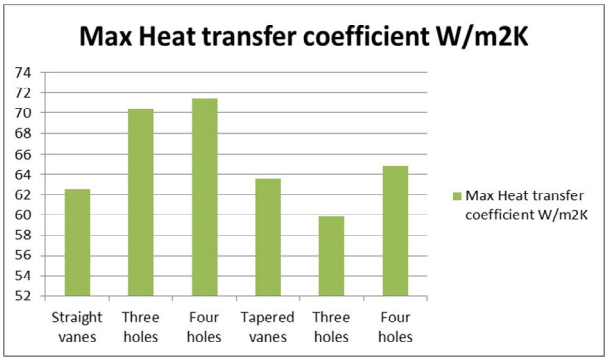



Fig. 29 Heat transfer coefficient variation with respect to geometry for analysis 2

### C. *Analysis 3*

The second analysis was carried out at the speed of 100 kmph and 650 K temperature as mentioned in table 4.

Table 8. CFD results for analysis 3

| Brake type | Total surface heat flux W/m2 | Maximum heat transfer coefficient |
|---|---|---|
| Straight vanes | 21989.5 | 60.77135 |
| Three holes | 25996.5 | 71.84528 |
| Four holes | 25359.43 | 70.08465 |
| Tapered vanes | 26236.81 | 72.50944 |
| Tapered three holes | 26809.02 | 74.09081 |
| Tapered four holes | 25369.92 | 70.1136 |

In the third case it was observed that the maximum heat transfer coefficient was obtained in the rotor with tapered vanes and three holes. The results obtained in the third case are completely different from that obtained in the first two cases. In fact, here, the best performing rotor is the rotor which performed the worst in the first two cases, that is, the rotor with tapered vanes and three holes. Another observation is that, the rotor with straight vanes and four holes has consistently performed well in all the three cases, and the rotor with straight vanes and three holes comes close too. Thus if strength is an important criterion in selection, the latter seems to be a better choice. Similarly as observed in the previous two cases, just changing the shape of rotors from straight vanes to tapered vanes is giving better heat dissipation. In all the three cases, the rotor type that has performed consistently is the rotor with straight vanes and four holes. Thus on the basis of the above three analysis, it is the best rotor design. The second best is the rotor with straight vanes and three holes. However, if drilling holes is not feasible, then simply changing the shape of the vanes is improving the performance.

©IJRASET: All Rights are Reserved

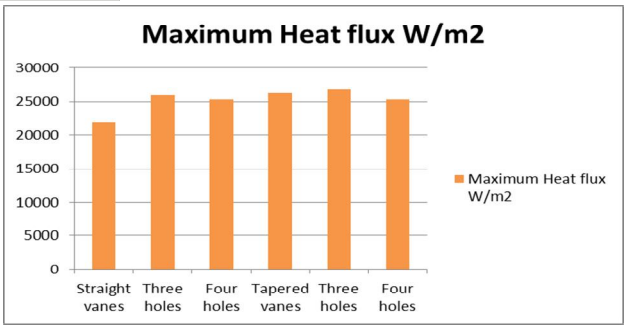



Fig. 30 Heat flux variation with respect to geometry for analysis 3

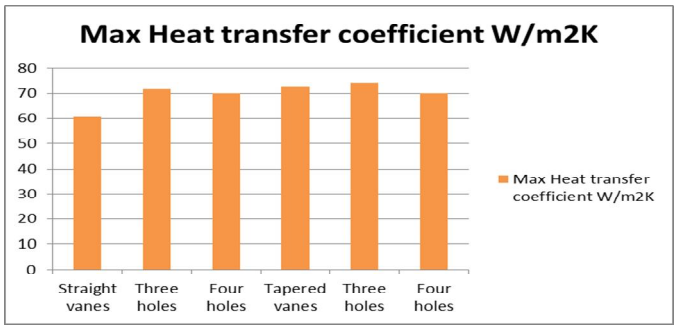



Fig. 31 Heat transfer coefficient variation with respect to geometry for analysis 3

## V. CONCLUSIONS

The main purpose of this study was to investigate the performance of disc brake rotors under different operating conditions for different geometries. A CFD analysis has been carried out to determine the same. From the results it is clear that drilling holes is indeed beneficial for the thermal performance of a disc brake. The geometry with straight vanes and four holes performed well in all the operating conditions. It can be used in place of non-drilled rotors.

## VI. FUTURE WORK

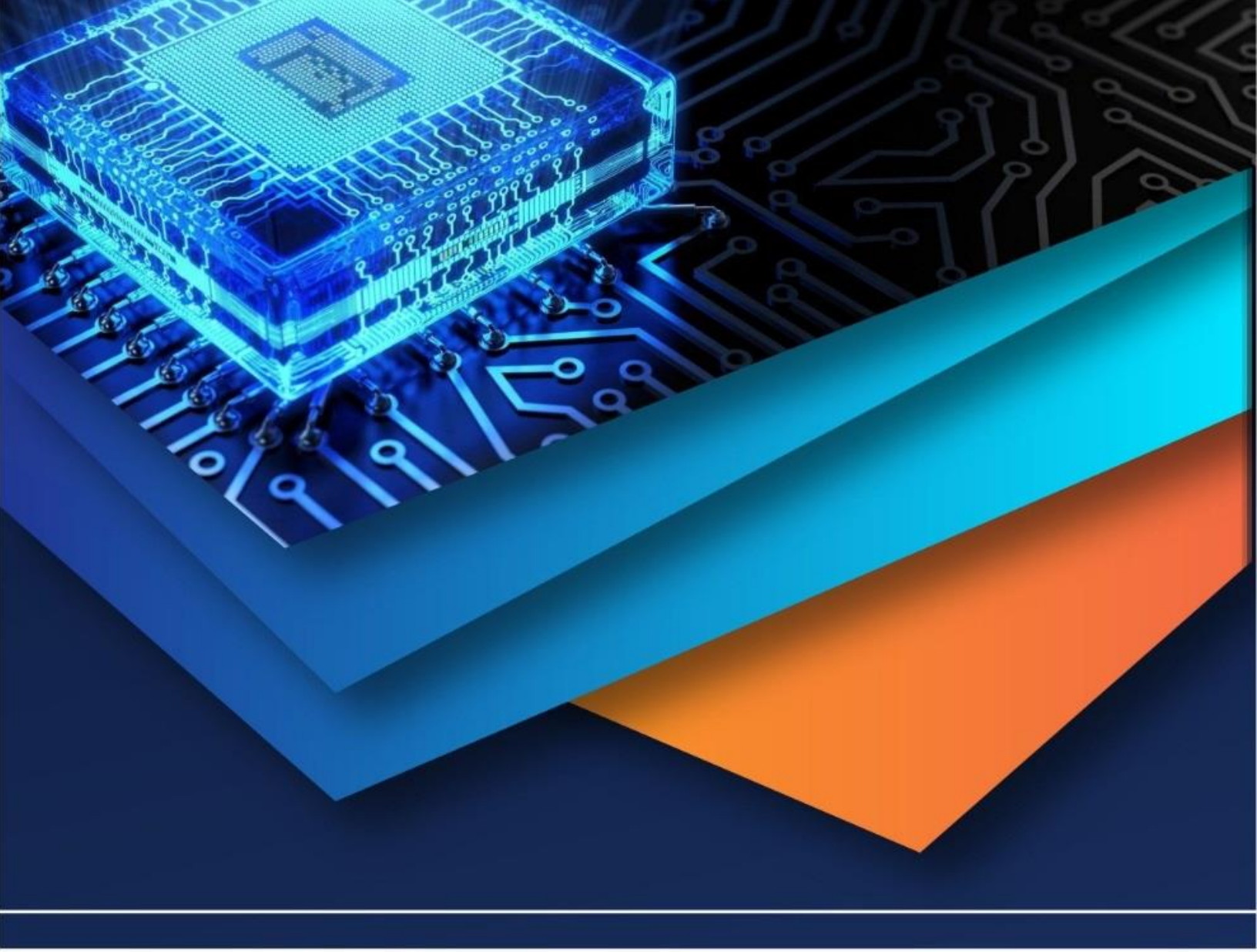
In this study, the hub was not modelled in the disc. Also brake calliper and the brake pads were omitted from the geometry. The inputs for future works are listed below

*A.* Modelling the hub, brake pad and calliper assembly to study their effect on air flow and heat transfer.
*B.* Analysing the disc at low speeds and very high speeds.
*C.* Modelling curved and pillar vanes and studying their effect on the overall performance of the brakes.

## REFERENCES

[1] David Bryant, John D Fieldhouse, Chris J Talbot, (2011), “Brake Judder – An Investigation of the Thermo-elastic and Thermo-plastic Effects during Braking”, International Journal of Vehicle Structures and Systems, Vol.3. No.1, pp. 58-73
[2] Mr. Prashant Patel,Prof. Prof. M.A.Mohite ,(2017), “Design optimization of passenger car front brake disc for improvement in thermal behavior, weight & Cost reduction.”, International Journal of Engineering Development and Research., Vol.5. Issue 2 , pp. 1079-1086
[3] G.Ranjith Kumar, S.Thriveni, M. Rajasekhar Reddy, Dr. G. Harinath Gowd (2014) “Design Analysis & Optimization of an Automotive Disc Brake”, International Journal of Advanced Engineering Research and Science, Vol.1. Issue.3, pp. 24-29
[4] Jimit G. Vyas, Prof. M. J. Zinzivadia, Prof. M. I. Kathadi (2017), “Design and Analysis of Solid and Cross-Drilled Disc Brake Rotors”, International Conference on Research and Innovations in Science, Engineering & Technology, Vol 1, pp. 294-301
[5] G P Voller, M Tirovic, R Morris, P Gibbens (2003), “Analysis of automotive disc brake cooling characteristics”, Journal of Automobile Engineering, Vol.217. Issue.8, pp. 657-666
[6] Kapil A Shelar, Shivprakash B Barve, Mahesh R. Chopade, Pramod E Chaudhari (2014) “Thermal Analysis of Automotive Brake Rotor”, International Journal of Current Engineering and Technology, Issue-3, pp. 129-132
[7] X. Charles, (2012), “Disc Brake and its Effective Air Cooling System Analysis by Computational Fluid Dynamics”, Middle-East Journal of Scientific Research, Vol.12. Issue 12, pp. 1602-1607
[8] Amol V. More, Prof.Sivakumar R. (2014) “CFD Analysis of Automotive Ventilated Disc Brake Rotor”, International Journal of Engineering Research and Applications, Vol.4. Issue.4, pp. 01-05
[9] Hairuniza Ahmed Kutty, Parvathy Rajendran (2017), “3D CFD Simulation and Experimental Validation of Small APC Slow Flyer Propeller Blade”, Aerospace, pp.01-11.

©IJRASET: All Rights are Reserved



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)