International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 05 | May 2021 www.irjet.net p-ISSN: 2395-0072

Heat Transfer Analysis of Automobile Disc Brake using Simulation Software

Sanil Kalekar[1], Devendra Prabhu[1], Tejal Raut[2], Ninad Pande[4]

[1]*Research Scholar, Theem College of Engineering, Boisar (India)*
[2]*Professor, Mechanical Engineering, Theem College of Engineering, Boisar (India)*
[3]*FEA & CFD Consultant,* (India)

---***---

Abstract - *Heat dissipation rate of automobile brake disc is studied in this paper. Large amount of heat is dissipated during automobile braking action. The braking energy is converted to heat due to clamping action of the brake pad with the disc. The amount of brake force required to stop the vehicle is calculated in this paper. Heat dissipation through conduction is studied using finite element analysis, also the braking system is optimized by ducting the brakes for forced convection due to inlet ramming air is studied. The effect of forced air, convection and radiation analysis is studied using computational fluid dynamics. Four different cases studies are analyzed and compared in this paper.*

Key Words: Finite element analysis, Computational fluid dynamics, Forced convection

1. INTRODUCTION

The braking system is one of the crucial system in an automobile experiencing high forces and wear and tear. The braking force is actuated by the master cylinder in correlation with the brake pedal. A high pressure a created in the brake lines which press the brake pad against the disc. Thus the braking action is achieved.

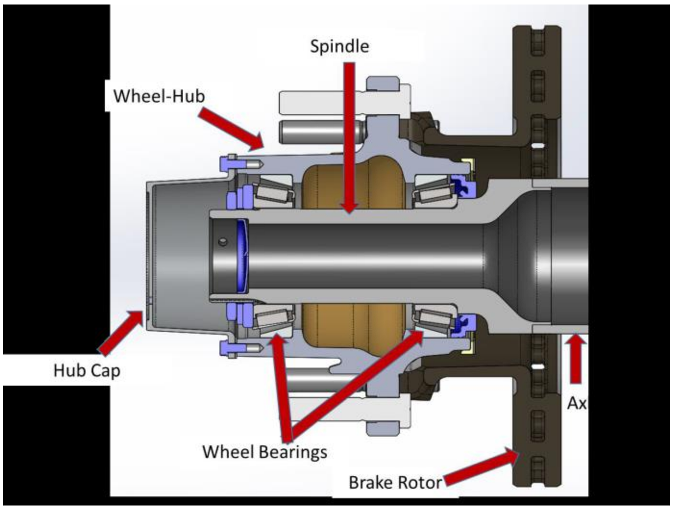

Fig -1: Shows the components and assembly of braking system [2]

The braking system consist of brake calliper, pads, master cylinder, brake lines, brake oil and brake disc. The brake disc is connected with the wheel. Thus reducing speed on the disc by braking action reduces speed of wheel which in turn stops the vehicle. This action produced tremendous amount of heat energy, which not taken care of introduces fatigue thermal failure in disc.

2 Literature survey

Daniel Das.A. et al, The aim of this paper was to investigate the temperature fields and also structural fields of the solid disc brake during short and emergency braking with four different materials. The distribution of the temperature depends on the various factors such as friction, surface roughness and speed. The effect of the angular velocity and the contact pressure induces the temperature rise of disc brake. The finite element simulation for two-dimensional model was preferred due to the heat flux ratio constantly distributed in circumferential direction. We will take down the value of temperature, and deformation for different pressure condition using analysis software with four materials. The Disc brakes are made up of cast iron. [1]

Mit Patel et al. Thermal FEA analysis of car disc if performed in this paper. Brake disc of a car is considered for analysis. Kinetic energy produced by the vehicle and corresponding braking force is calculated. Also, the heat generated by the disc is calculated using specific heat of the disc material. Thermal distribution of the disc is studied. [3]

O FP Lyons et. Al. An experiment is performed to study the heat transfer of disc brake using a jet. A duct is fabricated which ensured the supply of air to the disc. Various conditions such as no air-flow and angular flow are analyzed. A transient model is studied for investigating temperature profile with time. Effective cooling and reduced hot spots, thereby increasing the efficiency of the rotor. The results from the experiment show that the life cycle of the braking system is increased. Effect of forced convection reduces the maximum temperature and heat will be removed from the disc at a faster rate. [4]

3. Methodology

3.1 Kinetic energy and heat flux calculations

Vehicle for study taken as Renault Clio

Kinetic energy [K.E.] = ½ * m * v2

= ½ * 1670 * 33.332

= 927592.33 Joules

[Taking mass of vehicle as 1670 Kgs and speed as 120 km/hr = 33.33 m/s]

Braking torque of disc

$$\text{Braking force on each wheel X } \frac{\text{Radius of wheel(R)}}{\text{Radius of rotor (r)}}$$

$$= 316.6 \text{ X } \frac{228.6}{156.1}$$

$$= 463.64 \text{ N.m}$$

[Taking 18“ wheels. Only the relevant calculations required for analysis are shown]

Heat flux produced = Mass of disc X Specific heat X Change in temperature

$$\text{K.E.} = 9.2 \text{ X } 584 \text{ X } \delta T$$

$$\delta T = 172.64\ ^0C$$

Thus Temperature (max) = δT + T (ambient)

$$= 172.64 + 25$$

$$= 197.61\ ^0C$$

Assuming lowest air inlet velocity (v) = 5km/hr

$$= 1.4 \text{ m/s}$$

Taking r.p.m. of disc

$$v = \text{Radius of disc X } \omega$$

$$\text{r.p.m.} = \frac{1.4 \text{ X } 60}{0.3 \text{ X } 2 \text{ X } 3.14}$$

$$= 45 \text{ r.p.m.}$$

3.2 2D file from which the 3D geometry is made.

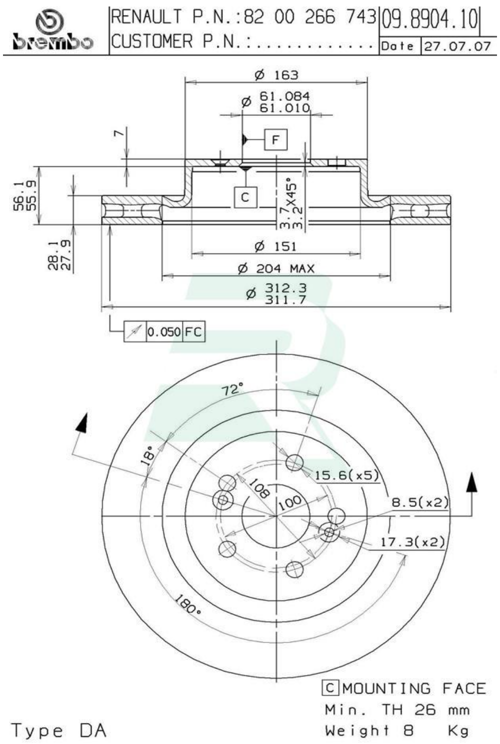

Fig -2: Shows the 2 D drawings of the component

3.3 Following are the details of 3D geometry

First a sketch is drawn as per the 2D drawings and then the sketch is revolved to obtain the desired dimension. The inner blocks are cut extruded and an arrangement of four holes to mount the disc is made. Commands such as extrude and revolve are used to complete the 3 D disc model.

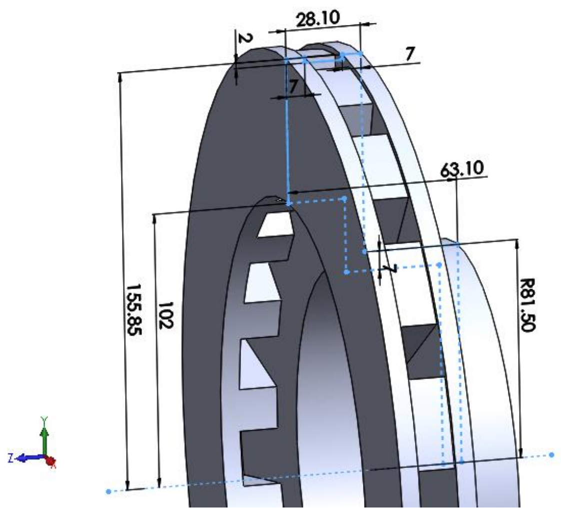

Fig -3: Shows the 3 D model of the disc for FEA analysis

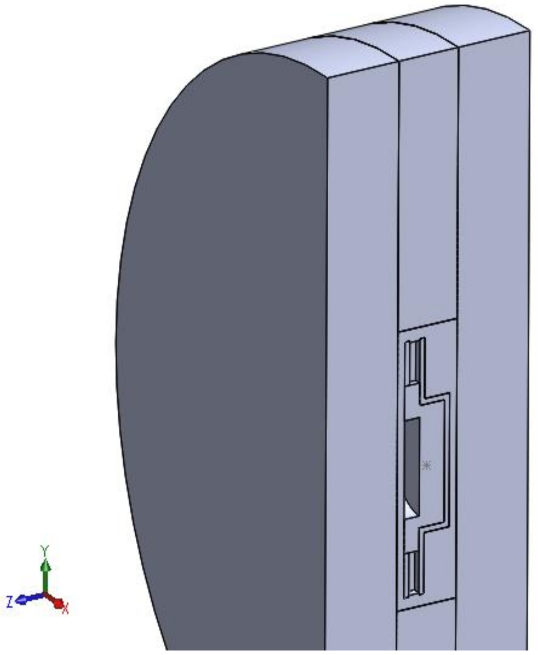

Fig -3: Shows the 3 D model of the disc for CFD analysis

We have taken same geometry for case 0 and case 1. The only difference is inlet velocity. Case 0 is assumed as ideal conditions without forced cooling and hence velocity taken as 0.1 m/s. Case 1 is assumed as forced cooling and thus velocity is taken as 1.4 m/s. Case 2 as more number of vanes thereby increasing the surface area. Case 3 has curved vanes to enhance the heat transferred to surrounding.

Disc cross section for case 1 & case 2

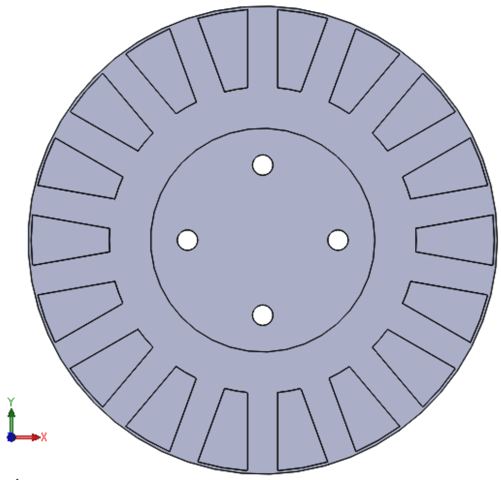

Fig -4: Shows the 3 D model of the disc for case 1 & 2 for CFD analysis

Disc cross section of case 3

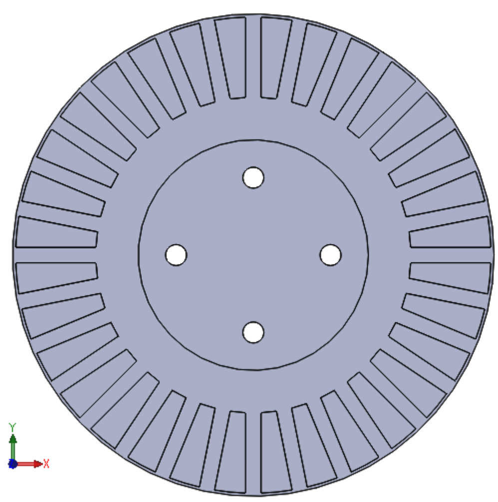

Fig -5: Shows the 3 D model of the disc for case3 for CFD analysis

Disc cross section of case 3

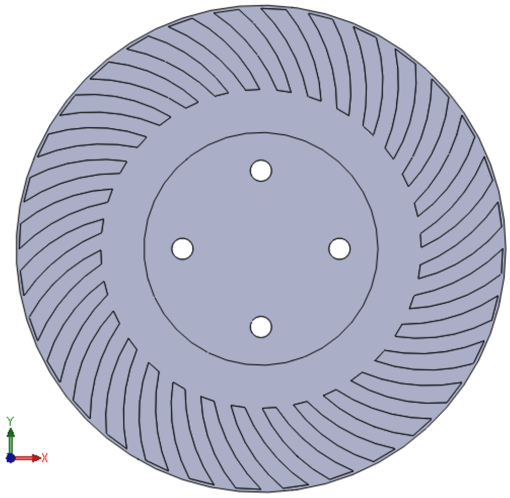

Fig -6: Shows the 3 D model of the disc for case 4 CFD analysis

Table -1: Shows different cases studies

Case 1	**18 straight perforation**	**0.1 m/s inlet air velocity**
Case2	**18 straight perforation**	**1.4 m/s inlet air velocity**
Case 3	**31 straight perforation**	**1.4 m/s inlet air velocity**
Case4	**31 curved perforation**	**1.4 m/s inlet air velocity**

3.4 Mesh parameters

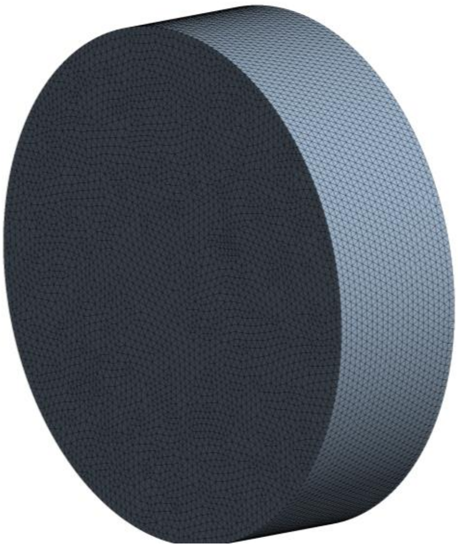

Fig -7: Shows the mesh model

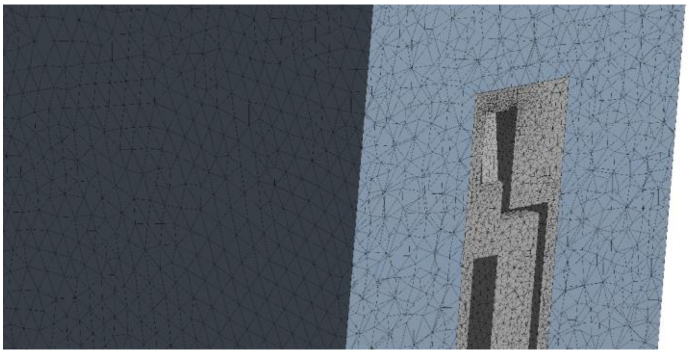

Fig -8: Shows the cross-section of mesh model

Proximity and curvature features are used for discretization of the domain. Proximity features covers the edges and cut features whereas curvature features take care of the curved edges. Curvature of 18 Degree is defined with minimum elements across the gap for proximity feature. The proximity and minimum element size is taken as 2 mm. The growth rate is taken as 1.1 for smooth transition of elements overall.

The mesh type used is 3D tetrahedral mesh. Tetrahedral elements are best suited for complex geometries. In this case study, there are lot of curved profiles and tetrahedral elements are best suited for such applications. This element has four vertices, six edges and four triangular faces. These elements are stiffer than hexahedral elements, but due its shape, it has more application in CFD problems. These elements have lower aspect ratio and are thus easily accepted by the CFD solvers.

3.5 Solver boundary conditions

FEA analysis

First thermal analysis is carried out to check the temperature distribution in the disc. Following images show the boundary conditions applied on the disc.Temperature is applied on the surface which is in contact with the disc pad. Heat will be generated once the pad comes in contact with the disc. Thus contact surface will get heated first and this heat will transfer to the remaining areas.

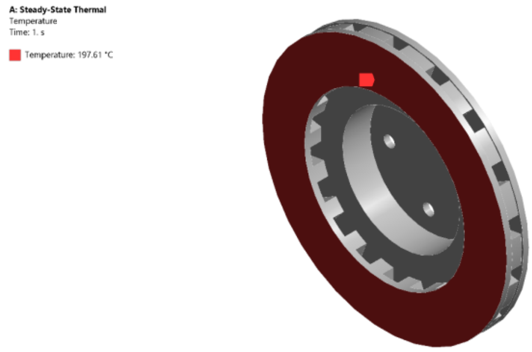

Fig -9: Shows the temperature applied at the face

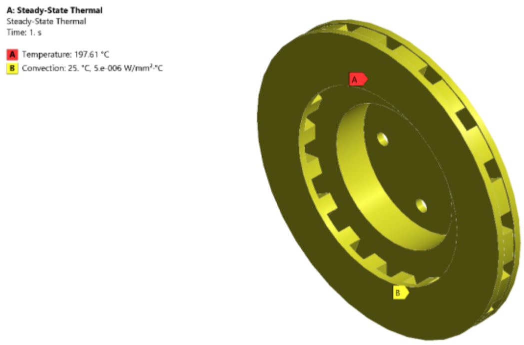

Fig -10: Shows free convection applied on all faces

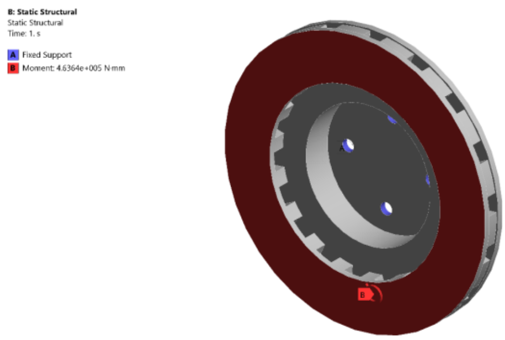

Fig -11: Shows the mechanical boundary condition

The above figure shows the boundary conditions use in mechanical analysis. The disc is fixed at the center. Moment of 463.64 N-m is applied on the disc surface which comes in contact with the pad. The stresses and deflection can be seen in the following images.

CFD analysis

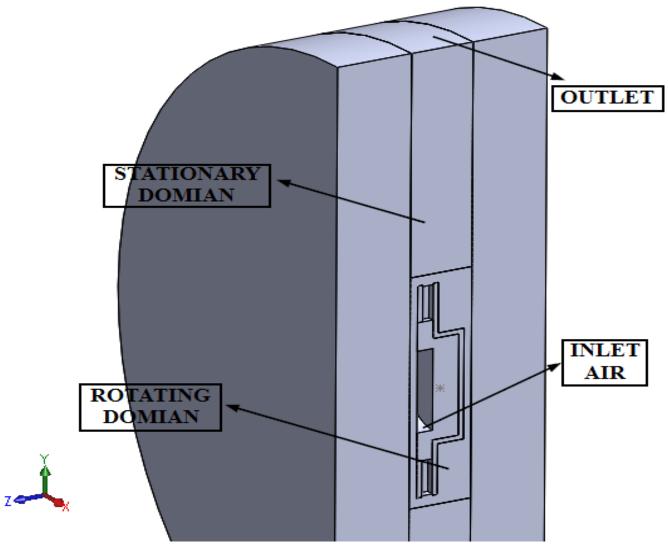

Fig -12: Shows the description of CFD model

Air is supplied from the inlet side at the center of the disc. This inlet air will pass over the vanes, carry heat and exit from the outlet. There are two domains, rotating domain and stationary domain. The rotating domain implies the rotating disc part and the stationary domain represents the stagnant air around it.

After conducting the FEA thermal analysis as per the temperature distribution within the disc, we majorly observe two temperature regions. One near to the disc the other one away from the disc.

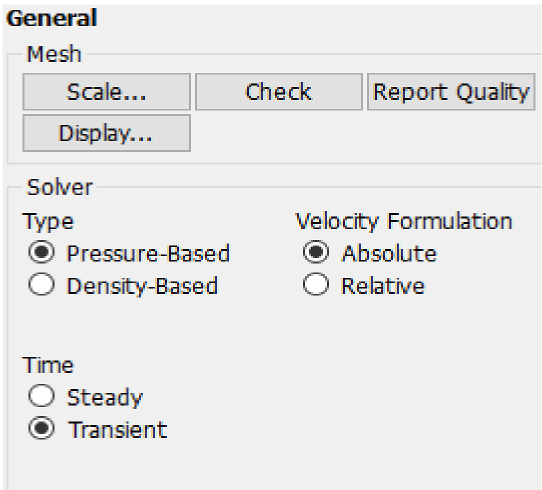

Fig -13: Shows the solver boundary conditions

The solver type determines the quantity which will be taken into consideration for calculating the flow parameters. Pressure based solver is used in applications where the fluid has low velocity and density based solver is used during higher Mach number flows. Thus as per our application we will be using pressure based solver. Velocity formulation is the method where velocity is formulated. Absolute method is used where the domain in stationary and relative method is used during rotating domain. It can be formulated with respect to first and last values (Absolute method) of after each and every point (relative method). Transient solver is used for time dependent analysis.

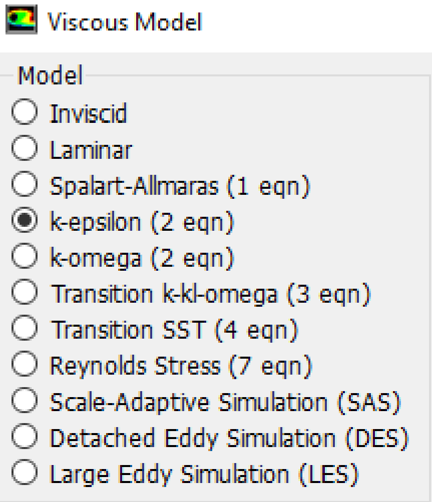

Fig -14: Shows the flow boundary conditions

The most commonly used turbulent module is K-epsilon. The flow will not follow a laminar pattern due to cross-section of the geometry. The flow of the medium depends upon velocity, pressure and change in cross-section. The k-epsilon model for turbulence is the most common to simulate the mean flow characteristics for turbulent flow conditions. This is a two equation model which gives a general description of turbulence by means of two transport equations. The 2 transported variables are turbulent kinetic energy k, which determine the energy in turbulence, and turbulent dissipation, which determines the rate of dissipation of the turbulent kinetic energy. For the wall function parameters, enhanced wall treatment functions is used which formulates the equations near wall using velocity formulations.

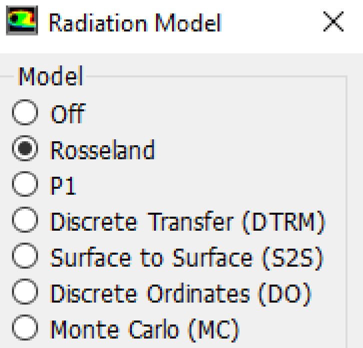

Fig -15: Shows the radiation boundary conditions

The Rosseland radiation model can be used to account for the radiation exchange in an enclosure of gray-diffuse surfaces. The energy exchange between two surfaces depends on their size, separation distance, and orientation. The energy flux leaving a given surface is composed of directly emitted and reflected energy. The reflected energy flux is dependent on the incident energy flux from the surroundings, which then can be expressed in terms of the energy flux leaving all other surfaces. The Rosseland model works on gray body concept.

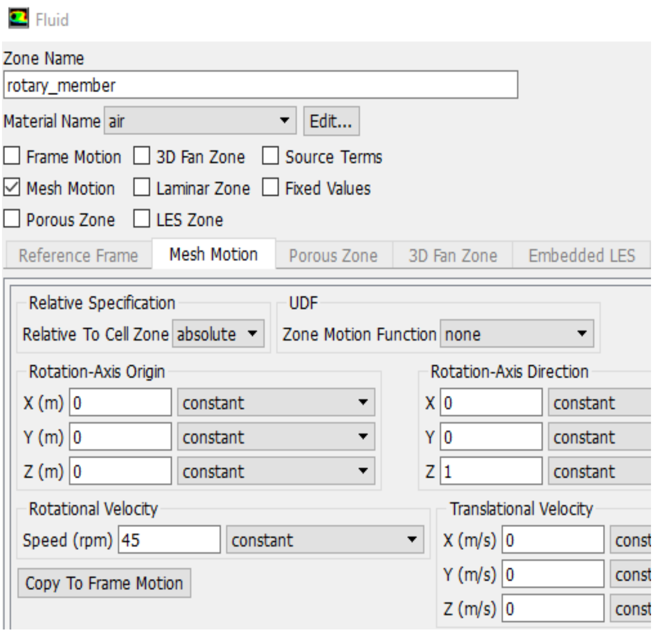

Fig -16: Shows the rotary boundary conditions

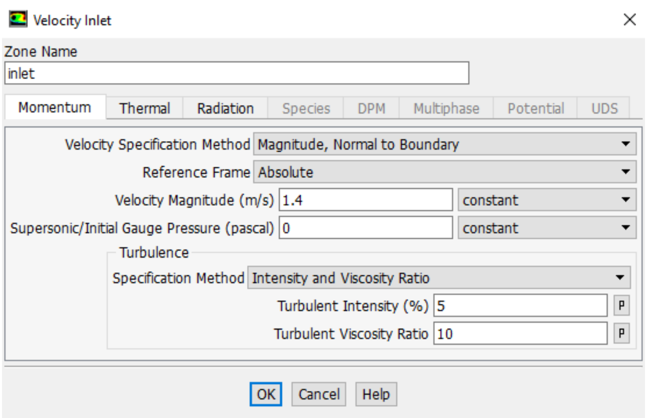

Fig -17: Shows the inlet boundary conditions

As per the calculations performed, the rotational speed of rotor was found to be 45 r.p.m. The disc is at the origin. A rotating reference frame method will be used to move the disc as per the r.p.m.

The inlet air velocity is taken as 1.4 m/s at ambient conditions

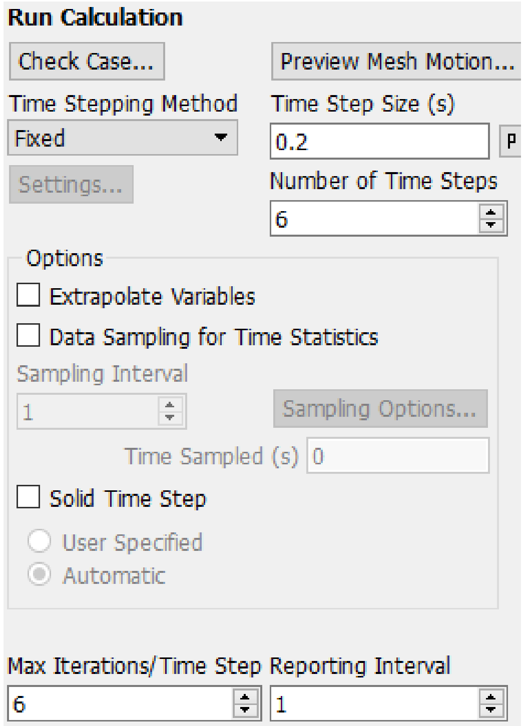

Fig -18: Shows the Time step boundary condition

As per said earlier, a transient analysis approach is considred. The time step for the analysis is considred as 0.2 sec & the number of time steps and Ilterations are 6 continued for 3.6 sec.

4. Analysis results

4.1 FEA results

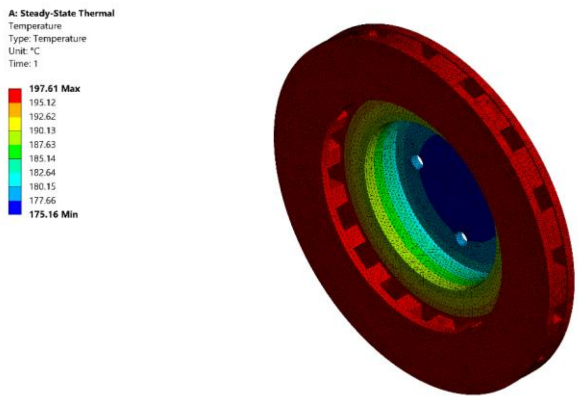

Fig -19: Shows the temperature distribution

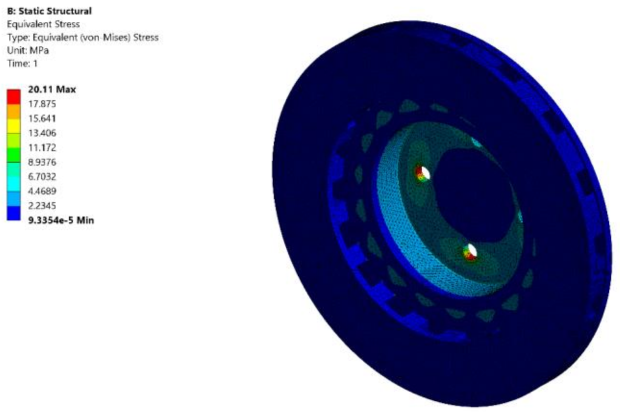

Fig -20: Shows the stress in disc

Table -2: Shows results of FEA analysis for different cases studies

FEA results	Max temperature in Degree Celsius	Stress in MPA
Case 1 & Case 2	197.61	20.11
Case2	193.61	20.4
Case 3	201.48	20.64

4.2 CFD results

Case 1

Velocity of air taken as 0.1 m/s

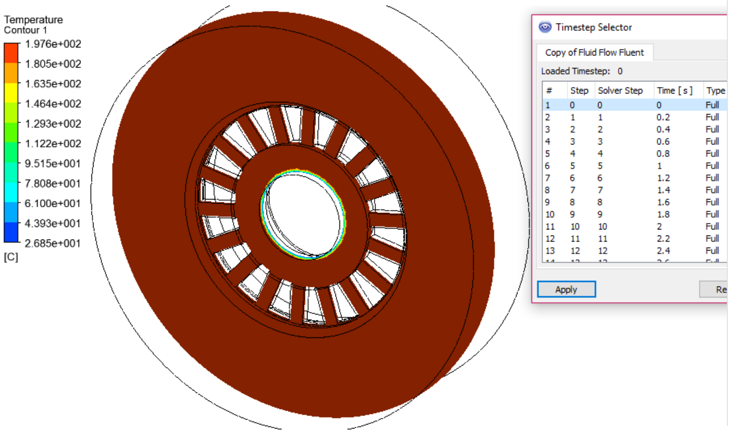

Fig -21: Shows temperature of disc, case 1 at 0 sec

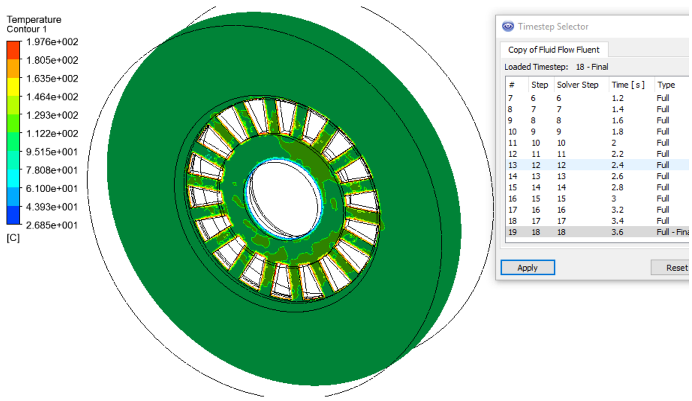

Fig -22: Shows temperature of disc, case 1 at 3.6 sec

Case 2

Velocity of air taken as 0.1 m/s

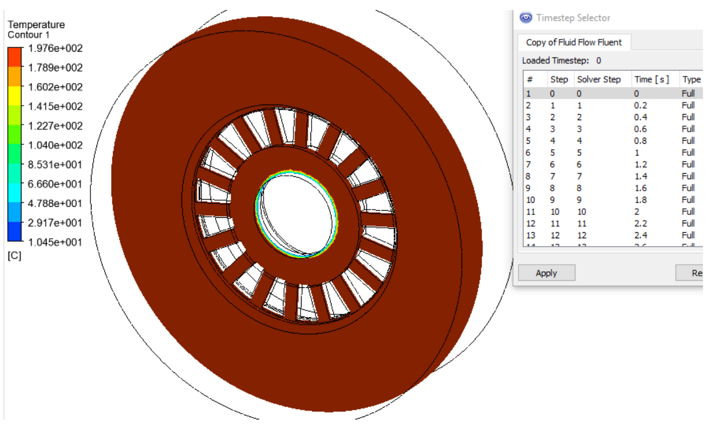

Fig -23: Shows temperature of disc, case 2 at 0 sec

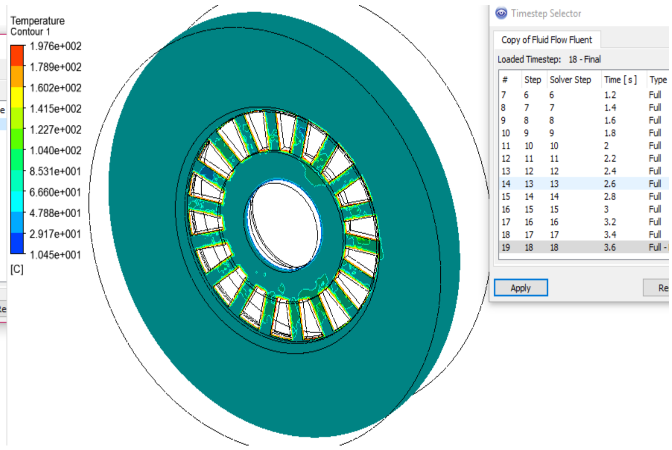

Fig -24: Shows temperature of disc, case 2 at 3.6 sec

Case 3

Velocity of air taken as 0.1 m/s

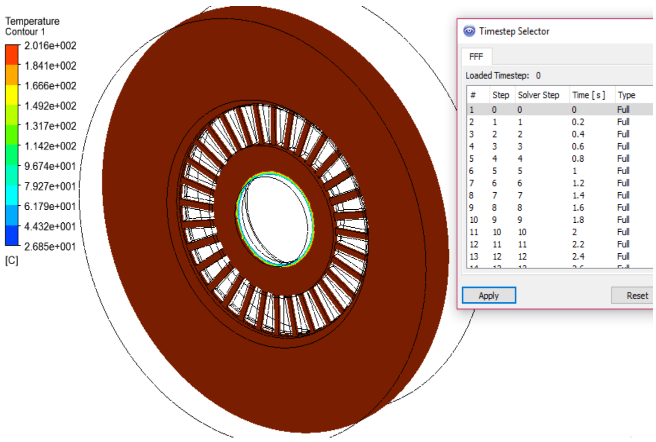

Fig -25: Shows temperature of disc, case 3 at 0 sec

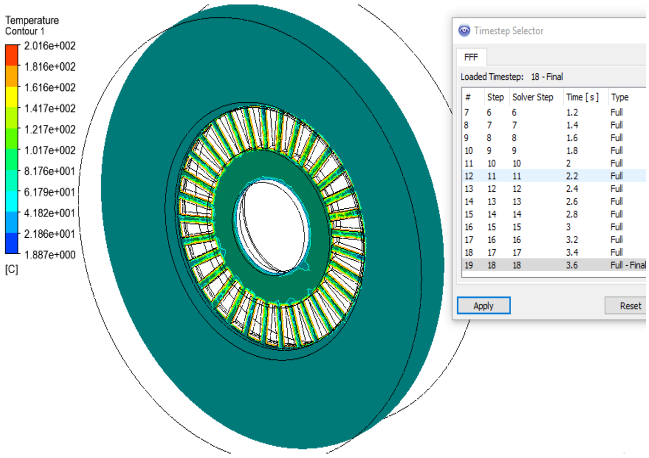

Fig -26: Shows temperature of disc, case 3 at 3.6 sec

Case 4

Velocity of air taken as 0.1 m/s

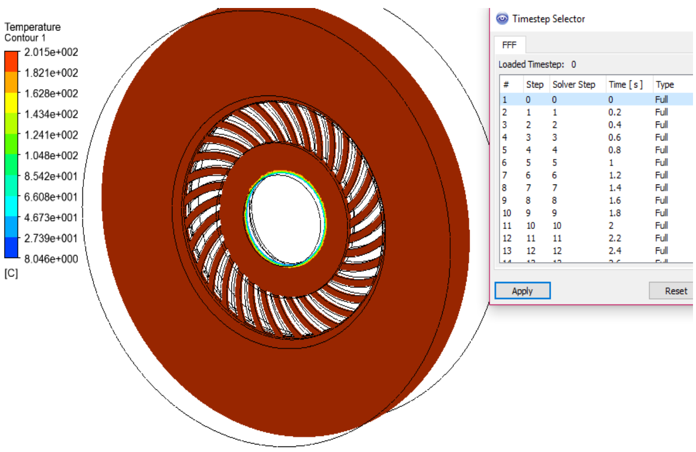

Fig -27: Shows temperature of disc, case 4 at 0 sec

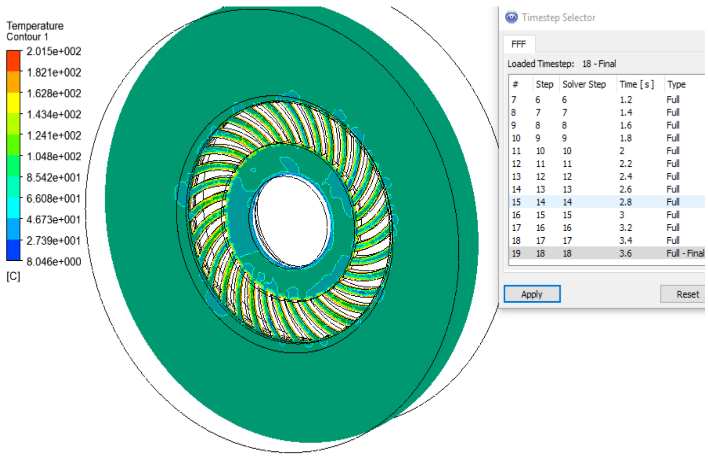

Fig -29: Shows temperature of disc, case 4 at 3.6 sec

5. CONCLUSIONS

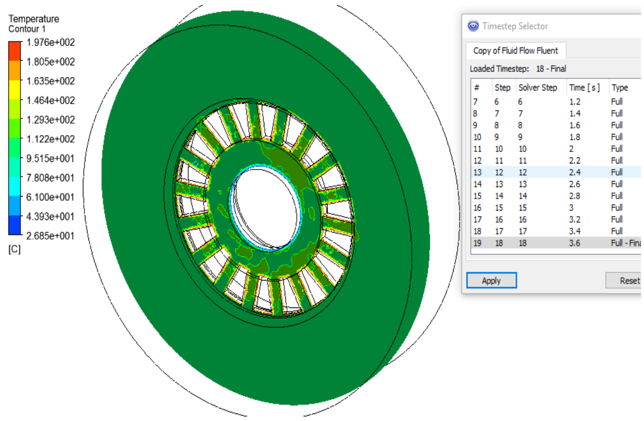

Fig -30: Shows the cross-section of mesh model

The above image is of case 0 where we have less/least air movement. We can see from the image that there are hot spots near the disc center. As there is less air velocity (0.1) at the inlet side, heat is not dissipated at the faster rate.

Less heat dissipation leads to hot spots. Higher temperatures at certain regions are termed as hot spots and these variations lead to higher difference in temperature which result in thermal fatigue. Thermal fatigue can lead to crack which result in failure of component. Following are comparative results as per cases.

These temperature and velocity probes are taken at the center of vane which is our main concentration area.

Table -3: Shows temperature plots of different case studies

Temperature				
cases	**0 sec**	**1 sec**	**2 sec**	**3.6 sec**
case 0	**465**	**435**	**410**	**390**
case 1	**465**	**424**	**379**	**358**

case 2	**462**	**420**	**364**	**325**
case 3	**470**	**424**	**362**	**326**

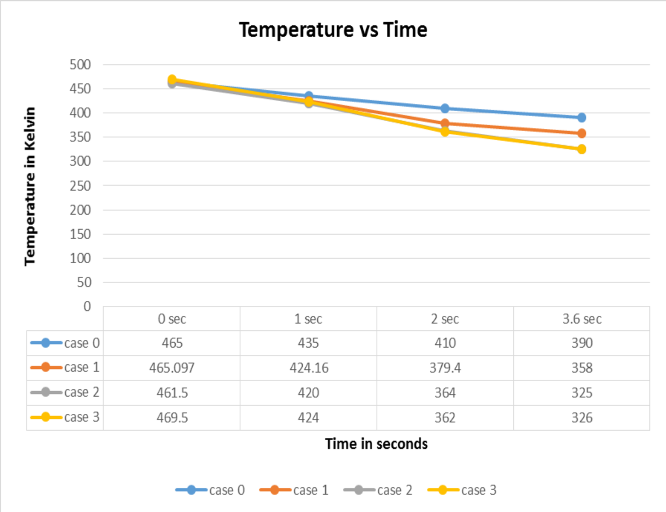

Chart -1: Shows Temperature versus time of all cases

Table -1: Sample Table format

Velocity				
cases	**0 sec**	**1 sec**	**2 sec**	**3.6 sec**
case 0	**0.01**	**0.3**	**0.3**	**0.2**
case 1	**0.15**	**0.5**	**0.44**	**0.49**
case 2	**0.16**	**0.4**	**0.35**	**0.44**
case 3	**0.18**	**0.6**	**0.58**	**0.6**

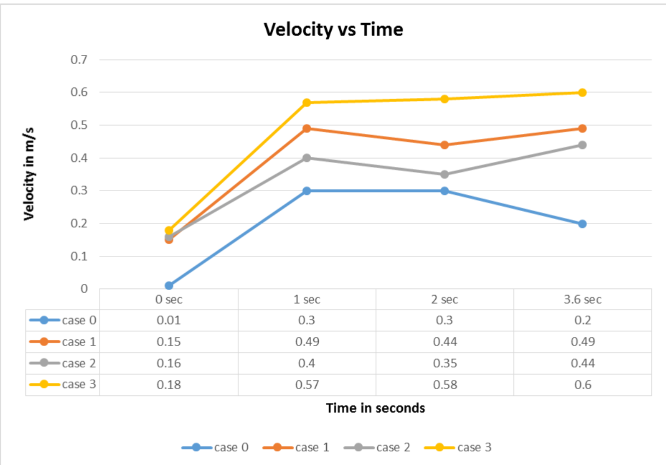

Chart -2: Shows Velocity versus time of all cases

From the above figure we observe that as we proceed from case 0 to case 3, we have better heat dissipation rates and velocity profile of the disc. Introducing air helps to increase the heat transfer from disc to the surrounding.

Higher velocity is observed at the last case. As more air entering the unit, forced convection is achieved. Also more air reduced the temperature drastically, thus reducing the hot spots.

REFERENCES

[1] Daniel Das. A, Christo Reegan Raj. V, Preethy. S, Ramya Bharani. G, "Structutal and Thermal Analysis of Disc Brake in Automobile", International Journal of Latest Trends in Engineering and Technology (IJLTET) Vol. 2, Issue 3 May 2013.

[2] Justin P. Fisher, 'A FEA/CFD study of heat transfer to Semi-Trailer Disc Brake wheel-End Components During A Braking Event' Grand Valley State University, 4-2019.

[3] Prof Mit Ptel, Mansi Raval, Jenish Patel, 'Design of Disc Brake's Rotor. IJEDR,2016, Volume 4, Issue 4, ISSN: 2321-9939

[4] O FP Lyons, D B Murray and A A Torrance, ' Ar jet cooling of brake discs, Department of Mechanical and Manufacturing Engineering, Trinity College, Dublin, Ireland, UK, 24 January 2008.

[5] Deepak S. Hugar, Prof. U. B. Kadabadi, 'Design and Thermal Analysis of Disc Brake for Minimizing Temperature', International Research Journal of Engineering and Technology (IRJET), Volume: 04 Issue: 07 July -2017, e-ISSN: 2395-0056, p-ISSN: 2395-0072.

[6] Subhasis Sarkar, Pravin P. Rathod, "Review paper on Thermal Analysis of Ventilated Disc Brake by varying Design parameter", International Journal of Engineering Research and Technology (IJERT) Vol. 2, Issue 12,December – 2013.

[7] D. G. Grieve, D. C. Barton, D. A. Crolla, J. T. Buckingham, Design of a lightweight Automotive Brake Disc Using Finite Element and Taguchi Techniques, Institution of Mechanical Engineers; 1998; 212, 4; ProQuest Science Journal.