Analysis of Disk Brake Rotor of a Two Wheeler

Muthanna K P[1], Akhil Mathew[2], Arunodhay P K[3], Jishnu V A[4], Vyshag Baburaj[5]

Department of Mechanical Engineering

[1]Assistant Professor,Coorg Institute of Technology,Ponnampet, Kodagu, Karnataka, India

[2, 3, 4, 5] UG Scholar,Coorg Institute of Technology,Ponnampet, Kodagu, Karnataka, India

Abstract*-The thermal performance and heat dissipation of brake disc is based on aerodynamic characteristics of flow though brake disc holes. In this project, the cooling performance of two wheeler brake disc is analyzed. To analyze the effect on the cooling performance CFD software package, Fluent is used for simulation of air flow rate, velocity distributions, temperature contours. Modeling of disc brake rotor is done using solid edge v18 software. Mesh for the model is done using ICEM-CFD and the post processing of the result in fluent 14.5. A Computational Fluid Dynamic (CFD) analysis for the heat generation problem of the disk brakes with using an ANSYS workbench. The result are discussed and presented in detail.*

Keywords-Truss, Tendon, Prestressing ,Staad pro.

I. INTRODUCTION

A brake is a device by means of which artificial frictional resistance is applied to moving machine member, in order to stop the motion of a machine. In the process of performing this function, the brakes absorb either kinetic energy of the moving member or the potential energy given up by objects being lowered by hoists, elevators etc. The energy engrossed by brakes is loss in the form of heat. This heat is loss in the nearby atmosphere to stop the vehicle, so the brake system should have following necessities: The brakes must be strong enough to stop the vehicle with in a lowest distance in an urgent situation. The driver must have proper control over the vehicle during braking and vehicle must not skid. The brakes must have well anti fade characteristics i.e. their effectiveness should not decrease with constant prolonged application. The brakes should have good anti wear properties.

II. CLASSIFICATION OF BRAKES

The mechanical brakes with respect to the direction of acting force possibly will be divided into the subsequent two groups: Radial Brake Axial Brake Radial brakes, In these brakes the force acting on the brakes drum is in radial direction. The radial brakes may be subdivided into external brakes and internal brakes. Axial Brakes: In these brakes the force acting on the brake drum is only in the axial direction. i.e. Disk brakes, Cone brakes.

III. DISK BRAKE

A disk brake consists of a cast iron disk bolted to the wheel hub and a stationary housing called caliper. The caliper is linked to several stationary piece of the vehicle such as the axle casing or the stub axle as is cast in two parts with every part containing a piston. In between every piston and the disk there is a friction pad held in location by retaining pins, spring plates etc. passages are drilled in the caliper for the fluid to come in or get out of every housing. As well the passages connected to another one for bleeding. Each cylinder contains rubber-sealing ring between the cylinder and piston. A schematic diagram is shown in the figure 1.1.

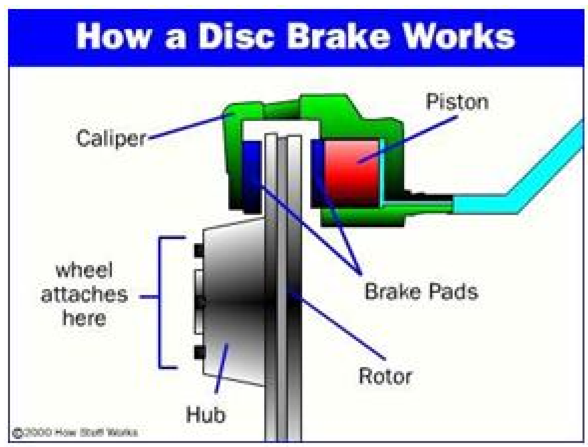


Fig. 1 Disk Brake

Problems in disk brake

In the trail of brake operation, the frictional heat is lost frequently into the pads as well as a disk, plus an infrequent irregular temperature distribution on the apparatus might induce harsh thermos-elastic distortion of the disk. The thermal distortion of a normally flat surface into a highly deformed state, called thermo elastic transition. It now and then happens in a sequence of stable continuously linked states operating situation changes. At other times, however, the stable evolution behavior of the sliding system crosses a threshold whereupon a sudden change of contact conditions occurs as the result of instability. This then calls a feedback loop that comprises the localized rise of frictional heating, the resultant localized stuffed, a localized pressure increases as the product of bulging, along with further rise of frictional heating as the result of the pressure increase. When this method leads to an accelerated transform of contact pressure distribution, the unpredicted hot unevenness of thermal distortion may raise unstably under several conditions, resulting in local hot spots and declining thermal cracks on the disk. This is known as thermo elastic instability (TEI). The thermo elastic instability phenomenon occurs more easily as the rotating speed of the disk increases. This area where the contact load is intense reaches very high temperatures, which leads to weakening in braking performance. Moreover, in the course of their presence on the disk, the passage of thermally distorted hot spots moving under the brake pads causes low-frequency brake vibration.

Objectives

The aim at this project is to analyze the fluid flow in disc brake rotor for two wheelers. Project objectives are as below:

- To understand the working principles, components, standards and theories through a literature study.
- To understand the working principle of FEA Software (ANSYS 14.5 workbench).
- To understand the fundamental of heat transfer through of disc brake rotor.
- Analysis of fluid flow in disc brake rotor.

Computational Fluid Dynamics:

Computational Fluid Dynamics or simply CFD is concerned with obtaining numerical solution to fluid flow problems by using computers. The equations prevailing the fluid flow trouble are the continuity, the Naiver-Stokes and the energy equations. These equations outline a system of coupled non-linear partial differential equations (PDEs). Because of the non-linear terms in these PDEs, analytical methods can yield very few solutions. In general, closed structure analytical solutions are promising simply if these PDEs can be prepared linear, either due to non-linear terms naturally drop out or due to nonlinear conditions are little as compared to extra conditions so that they can be ignored. If the non-linearity in the leading PDEs cannot be ignored, which is the condition for nearly all engineering flows, then numerical techniques are required to attain solutions.

CFD is the skill of changing the differential equation that lead to the Fluid Flow, with a given set of different algebraic equations, which in turn can be used to solve with the support of a digital computer to obtain the required estimated result.

The computational technologies enable us to study the dynamics of things that flow. With the use of CFD, it is easy to build a computational model which represents a structure or device that we wish to learn. We apply the fluid flow physics and chemistry to this virtual prototype and the software will output a prediction of the fluid dynamics and related physical phenomena. Thus, CFD is a complicated computationally based design plus analysis method. CFD software provides us with the authority to replicate flows of gases and liquids, heat as well as mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction as well as acoustics through computer modeling. With the use of CFD software, we are able to construct a 'virtual prototype' of the system or device that you desire to examine plus next relate real-world physics and chemistry to the model and the software will supply us among images plus data, that expect the performance of the given design.

CFD is predicting what will happen in the future, quantitatively, as quickly as fluids flow, regularly by way of the difficulties of:

- Simultaneous Flow of Heat.
- Mass Transfer (e.g., perspiration, dissolution).
- Phase Change (e.g., melting, freezing, boiling).
- Chemical Reaction (e.g., combustion, rusting).
- Mechanical Movement (e.g., of pistons, fans, rudders).
- Stresses and Displacement of immersed or surrounding solids.

CFD Analysis Process

The common procedure for conducting a CFD analysis is given below so as to offer a suggestion to get to know the variety of aspects of a CFD simulation. The process includes:

- Formulate the Flow Problem.
- Model the Geometry and Flow Domain.
- Establish the Boundary and Initial Conditions.
- Generate the Grid.
- Establish the Simulation Strategy.
- Establish the Input Parameters and Files.
- Perform the Simulation.
- Monitor the Simulation for Completion.
- Post-process the Simulation to get the Results.
- Make Comparisons of the Results.

Repeat the Process to Examine Sensitivities.

Formulation of the Flow Problem

Primary step of the analysis is to formulate the flow problem by finding answers for the following questions:

- Main objective of the analysis?
- Easiest way to make those objectives?
- What are the geometry should be included?
- What are the free stream and/or operating conditions?
- What dimensionality of the spatial model is required?
- Steady or unsteady modeling is appropriate?
- Nature of the viscous flow?

Modeling the Geometry and Flow Domain

Flow of the body is to be analyzed requires modeling. This specifies modeling the geometry by a CAD software .Approximations of the geometry as well as simplifications necessary to permit the analysis with sensible effort. Decisions are made as to the extent of the finite flow domain in the flow is to be formulated. Portions of boundary of flow domain comes in coincide with the surfaces of the body geometry. Other surfaces are free boundaries over which flow enters. The geometry and flow domain are modeled in such a way that to provide input for the grid development. Thus, the modeling often takes into account the structure and topology of the grid development.

Establishing the Boundary and Initial Condition

Since a limited flow domain is specified, physical conditions are necessary on the boundaries of the flow domain. The simulation usually starts starting from an initial solution and uses an iterative technique to arrive at a final flow field solution.

Grid Generation

The flow domain is discretized into a grid. The grid generation involve defining the structure plus topology and then generating a grid on that topology. Currently all cases involve multi-block, structured grids; however, the grid blocks may be abutting, contiguous, non-contiguous and overlapping. The grid should show some negligible grid quality as defined by measures of orthogonality, relative grid spacing, grid skewness, etc... Supplementary the maximum spacing should be consistent with the preferred resolution of significant features. The resolution of boundary layers requires the grid to be clustered in the direction normal to the surface with the spacing of the first grid point of the wall to be well within the laminar sub level of the boundary layer.

Establish the Simulation Strategy

The strategy for performing the simulation involves determining such things as the use of space-marching or time-marching, the choice of turbulence or chemistry model and the choice of algorithms.

Establish the Input Parameters and Files

A CFD codes generally requires that an input data file be created listing the values of the input parameters consisted with the desired strategy. Moreover the grid file that have the grid and boundary condition information is usually necessary. The files for the grid and initial flow solution need to be generated.

Performing the Simulation

The simulation is performed with various possible options for interactive or batch processing and distributed processing.

Monitor the Simulation for Completion

The Post Processing mode is actually a runtime visualizer which is instrumental in visualizing the development of the flow till the iterations are completed. On entering this mode the Flow domain will be displayed in the display area of the main window.

Post-Process the Simulations to get the Results

Post-Processing involves extracting the desired flow properties (pressure, mass flow etc.) from the computed flow field. On entering this mode the flow domain will be displayed in the display area of the main window

IV. MODELLING AND CFD ANALYSIS

Modelling of Disc BrakeRotor by Solid Edge V18 Software

Solid edge is a parametric, integrated 3D CAD/CAM/CAE solution created by Parametric Technology Corporation (PTC). It was the successful, parametric, feature-based, associative solid modelling software. The application works on Microsoft Window, Linux and UNIX platforms and provides features like solid modelling, assembly modelling and drafting, finite element analysis and NC and tooling functionality for mechanical engineer. Generally for modelling, packages like Solid Edge, Pro-Engineer, IDEAS will be used. In this analysis the model is created in the solid edge itself to eliminate the data losses that will occur if standard data exchange formats like IGES, STEPS are used. To create the model the details about the key point locations are taken as the inputs.

The aim of this project is to describe a contrast between the rotor disc of a standard motorcycle "BAJAJ PULSAR" as well as a non-standard rotor disc to study the relationship value between heat dissipation, rotor disc parameter etc.

Disc Brake Modeling Flow Chart

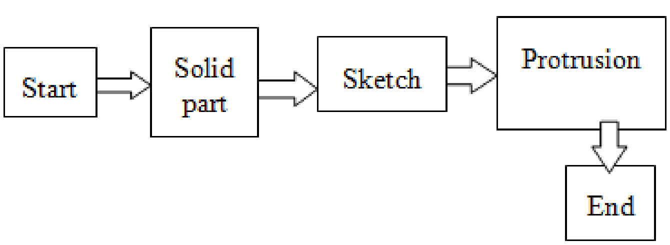


Fig. Disc brake modelling flow chart

To Construct Disc Brake Rotor

Sketching

1) Use all dimensions of "BAJAJ PULSAR" rotor disc.
2) To construct 2D sketch of disc brake rotor.
3) Select the same sketch which was previously selected i.e. front plane.
4) Using LINE COMMAND, POINT COMMAND, CURVE COMMAND, ARC COMMAND and FILLET COMMANDS, draw the disc rotor sketch as per dimension

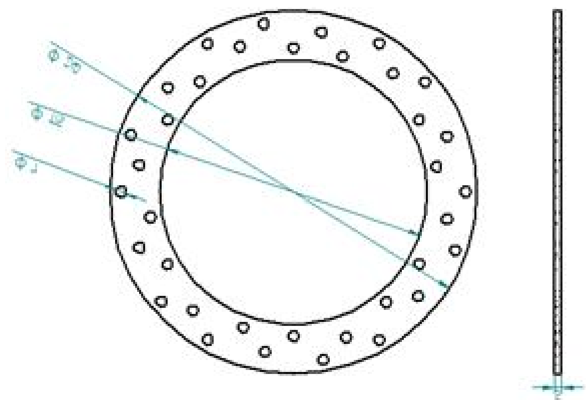


Fig. 2D view of disc brake rotor

Protrusion

1) Click on RETURN it will return to 3D mode part.
2) Select a protrusion from feature tool bar.
3) And give the protrusion of width 5 mm.
4) Select protrusion -Symmetric Extent, Extends the feature equally in opposite directions. And finish it.

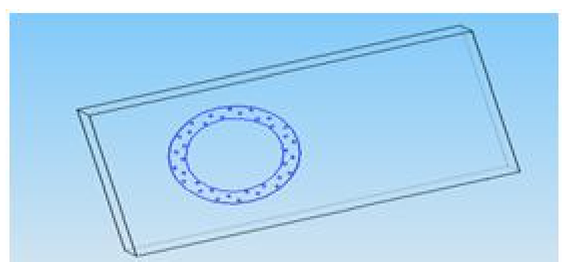


Fig. 3D view of disc brake with tunnel

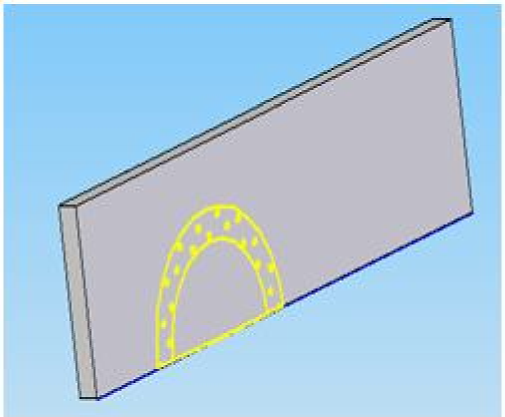


Fig. 3D view of disc brake rotor

To Construct TunnelSketching

1) Select any one of the reference plane say front plane.
2) Using LINE COMMAND draw the tunnel sketch as per dimension.
3) Click on RETURN it will return to 3D mode part.

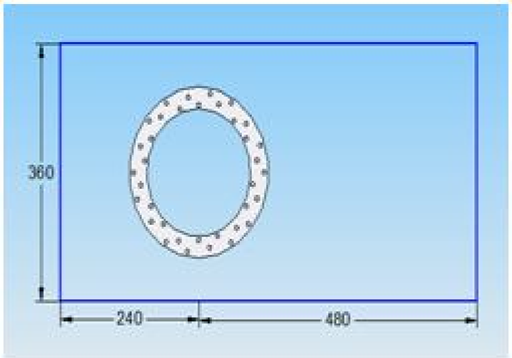


Fig. 2D view of disc brake with tunnel

Protrusion

1) Select a PROTRUSION from feature tool bar.
2) And give the PROTUSION of width 40mm.
3) Select Protrusion-Symmetric Extent, Extends the feature equally in opposite directions. And Finish it.
4) Then cut out the protrusion rotor using cut out command and finish it.

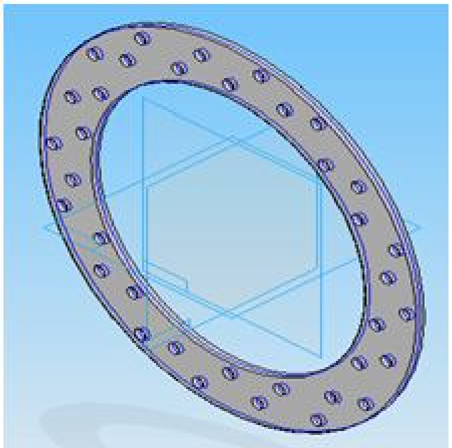


Fig. Half symmetry Meshing

Meshing is the breaking of physical/solution domain that can be a 2-D or 3-D domain into simpler sub domains or elements i.e. triangles, quadrilaterals for 2-D and tetrahedral, hexahedral for 3-D. Meshing make the solution easier and more accurate. The denser the meshing is more accurate the result will be but at the same time it will be more complex to solve the problem. Here we have used Quad/Tri mesh element and Pave type (creates unstructured grid of mesh element) to have more accurate flow near the rotor surface.

To have more fine meshing first of all edges of the geometry are meshed. The edges of the rotor geometry have more mesh counts.

Preparing for an Optimal Mesh

To save computational power it is always of high importance to avoid wasting cells by using too fine mesh where it is not needed. Since flow around the rotor will be affected differently by shapes and openings, a mesh with varying cell-size is desirable. In regions with details or shapes that strongly affect the flow, a fine mesh is needed to provide good results, whereas in regions where the flow is relatively unsawed, a coarser mesh can be used. By assigning parts and regions of different criticality from this perspective to different groups the mesh-size can be controlled in the solving phase.

Creating the Volume Mesh

To create a CFD-compatible mesh of high quality, several actions were performed in ICEM. The volume mesh, generated in ICEM-CFD, might contain some defects such as holes, overlapping edges and non-manifold surfaces. This can lead to problems during the volume mesh generation in the solving phase. The “Surface Wrapper” is a tool in ICEM which solves this problem by creating a continuous, “air tight", surface. This function searches for all surfaces that will be in contact with the outside air and creates a new surface from this data. Gaps and holes of bigger size than a reference value set by the user will be patched together.

Before generating the volume mesh, settings for controlling the cell size needed to be set. As previously mentioned, in order to avoid using an unnecessary large amount of cells, it is desirable to use a finer mesh in areas were the flow will be strongly affected and a coarser mesh in other regions. The growth of the volume mesh can be controlled in different ways. This is achieved either by setting a general growth rate, meaning that the speed of cell growth is predefined or by using refinement boxes. A block that encloses the object is then created and a maximum cell size inside the block can be set.

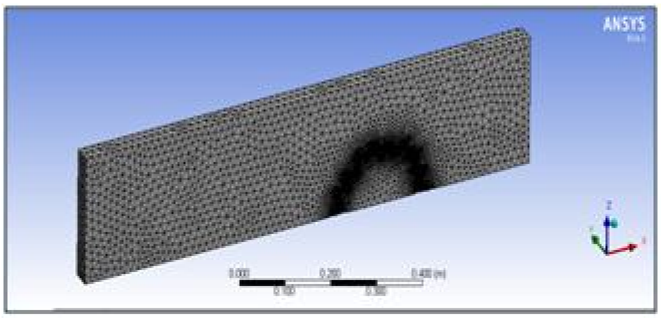


Fig .Meshing of 157032 cells boundary using ANSYS 14.5 workbench

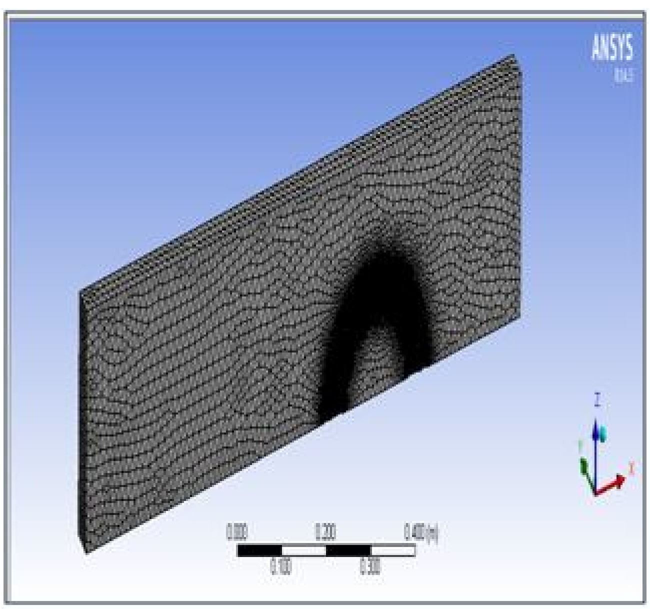


Fig. Meshing of 801834 cells boundary using ANSYS 14.5 workbench

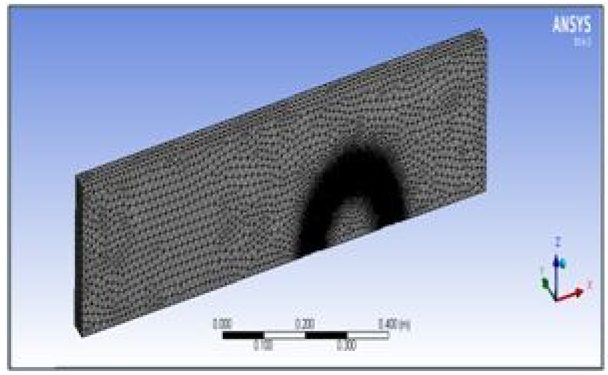


Fig. Meshing of 1337994 cells boundary using ANSYS 14.5 workbench

Boundary Conditions

Velocity boundary inlet conditions

Velocity boundary inlet conditions is generally used to outline the flow speed, together with relevant scalar properties of the flow, at flow inlets. During this case, the entire pressure isn't mounted, however can rise (in response to the computed static pressure) to a value that suit the velocity distribution. This condition is equally applicable to incompressible and compressible flows.

In special instances, a speed body of water could also be utilized in ANSYS FLUENT to outline the flow speed at flow exits. (The scalar inputs aren't utilized in such cases.) In such cases we have a tendency to should make sure that overall continuity is maintained within the domain.

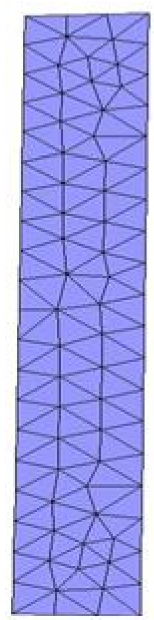


Fig. Inlet

Outlet Boundary Conditions

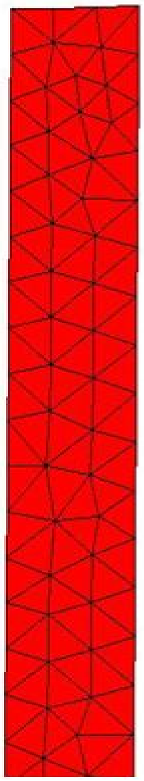
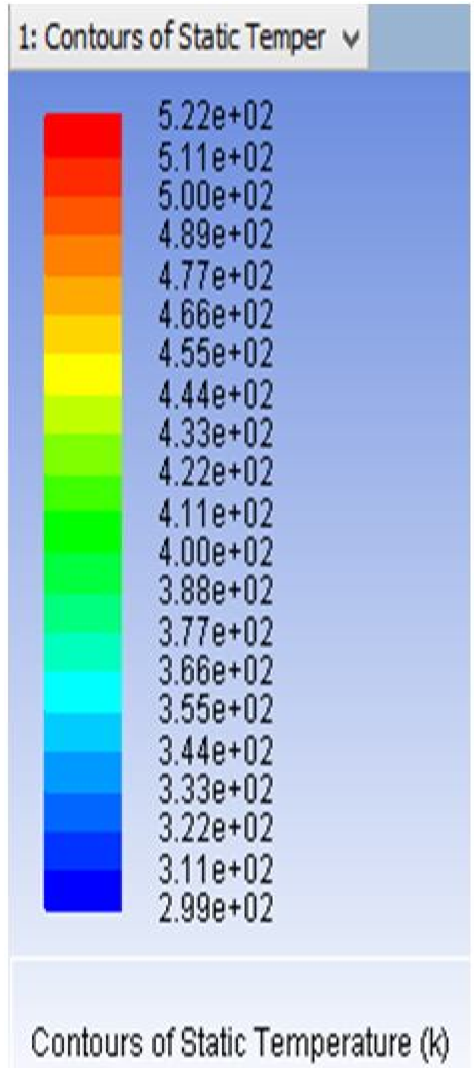


Fig. Outlet

Static pressure at outlet boundaries is needed for pressure outlet boundary conditions. The worth of the required static pressure is employed solely whereas the flow is subsonic. Ought to the flow become regionally supersonic, the required pressure can now not be used; pressure are cipher from the flow within the interior. All different flow quantities are cipher from the inside.

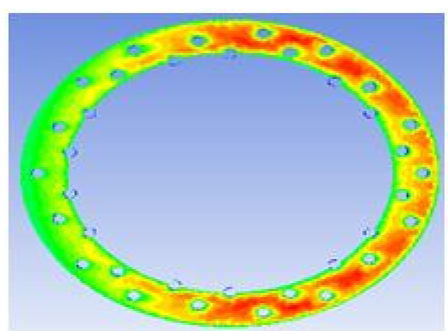
A set of “backflow” conditions is additionally such as ought to the flow reverse direction at the pressure outlet boundary throughout the answer method. Convergence difficulties are decreased if we tend to specify realistic values for the flowing quantities



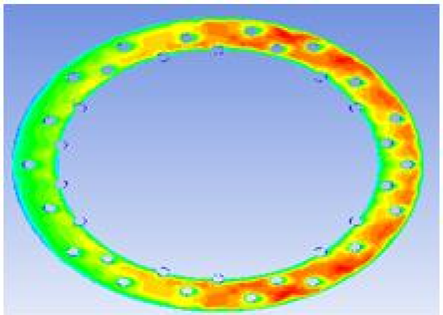
V. RESULT AND DISCUSSION

Temperature Distribution on Heated Wall, Heat Flux q= 15000W/m^2

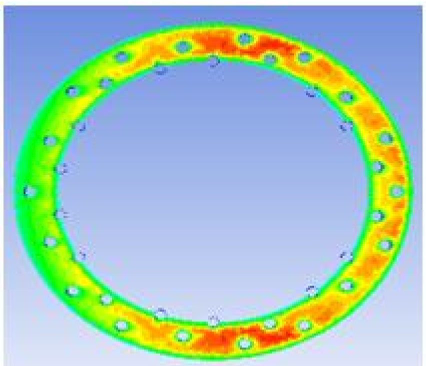
Velocity Magnitude 17 m/sec



(a) For 157032 cells

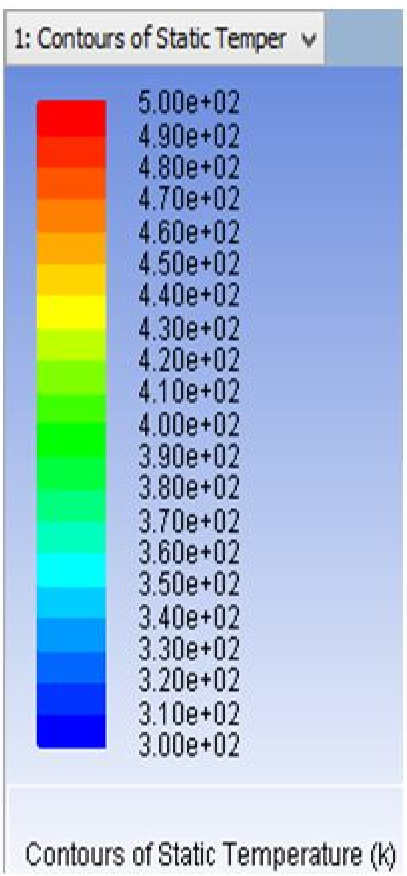


(b) For 801834 cells



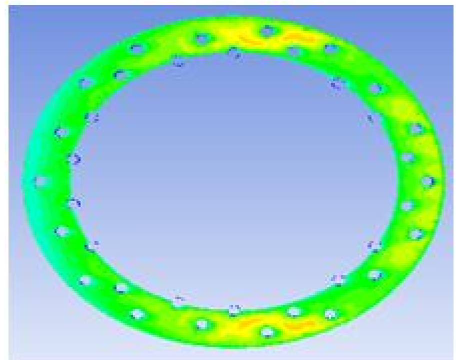
(c) For 1337994 cells

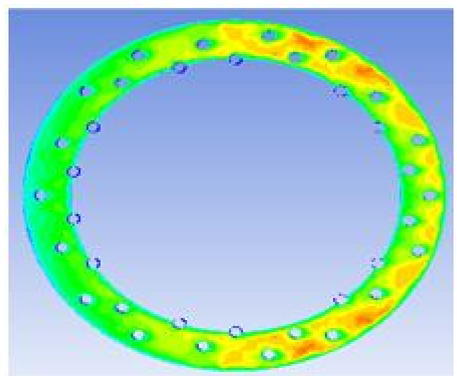
Fig. Temperature distribution through heated wall for different size of cells and velocity magnitude is 17m/s



The temperature distribution for velocity magnitudes 17 m/s can be seen in above figure. When the disc brake encounter the air, velocity at the beginning at the disc brake hits and velocity becomes zero due to stagnation indicated by green colour. The air passes at the holes to cool the disc brake. Heat generated on the disc rotor indicated by red colour.

Velocity Magnitude 25m/s





For 1337994 cells

Fig.Temperature distribution through heated wall for different size of cells and velocity magnitude is 25m/s

The temperature distribution for velocity magnitudes 25 m/s can be seen in above figure. When the disc brake encounter the air, velocity at the beginning at the disc brake hits and velocity becomes zero due to stagnation indicated by green colour. Cooling rate of heated wall decreases from inlet to outlet. As the increase in the flow of air the temperature on heated wall decreases.

Velocity Flow on Disc Brake Rotor of 801834 cells for Heat Flux is q=15000W/m^2

Velocity Magnitude 17m/s

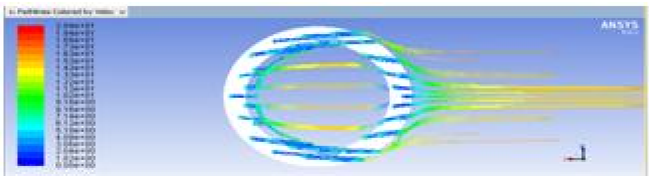


Fig. Velocity flow lines on disc rotor, of velocity 17m/s

Velocity Magnitude 25m/s

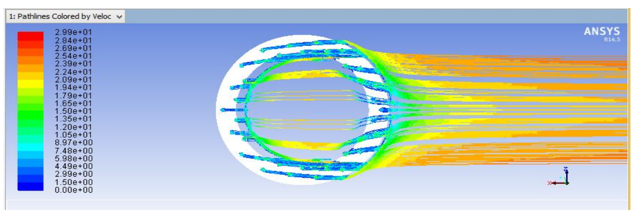


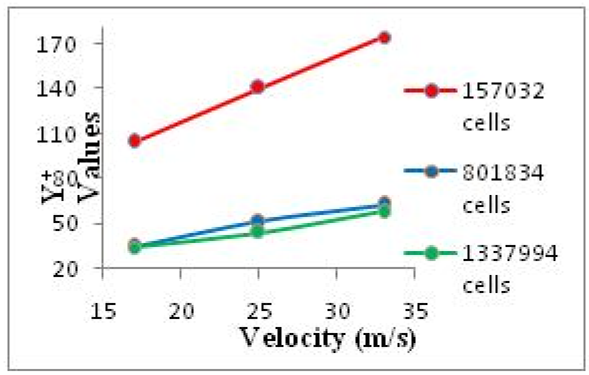
Fig. Velocity flow lines on disc rotor, of velocity 25m/s

The velocity contours for velocity magnitudes 17, 25 and 33 m/s can be seen in above figure. The following features can be observed.

- When the disc brake encounter the air, velocity of air that region becomes zero due to stagnation indicated by blue contour. (Point in a flow field where the local velocity of the fluid is zero.) The surrounding region has an increase in velocity due to deflection of flow from the front region of the disc brake.
- Flow over the roof shows that velocity is increased due to boundary layer separation occurring at that point.
- The velocity at rear end of the disc brake is reduced due to flow separation taking place resulting in the formation of wake region (pressure recovery region).

Grid independence test

No. of cells	Velo city, m/s	q in w/m^2 given	T_{max}, in ° C	Q in w	Yplus
157032	17	15000	248.87	130.293	104.23
	25	15000	198.19	130.293	140.26
	33	15000	170.93	130.293	174.48
801834	17	15000	223.41	130.232	34.65
	25	15000	177.76	130.232	51.18
	33	15000	150.8	130.232	62.53
1337994	17	15000	221.23	130.231	34.11
	25	15000	183.53	130.231	44
	33	15000	155.45	130.231	57.85



VI. CONCLUSION

- We are getting a yplus value of 34.11 for 17m/s velocity and for 1337994 cells. As the yplus value falls less than 100 we can conclude that the result we are getting is comparatively better.
- As the (mesh size decreases) number of cells increases the temperature is reducing, this shows the convergence of results.
- By flow analysis/path line we can observe the turbulence and swirl flow of air in the holes of the disc brake rotor.
- As the velocity increases the temperature induced in the disc brake is reducing that shows the better heat dissipation at higher velocities. We are getting 223.41°C for 17 m/s and 801834 cells, and 177.76°C for 33 m/s and 801834 cells.
- At the inlet the heat dissipation is high as the velocity will be high, compared to the outside region.
- For heat flux of 15000 w/m^2 and 801834 cells, the temperature is 223.41°C.
- For heat flux of 20000 w/m^2 and 801834 cells, the temperature is 288.22°C.

- As heat flux increases, induced temperature is also more.

REFERENCES

[1] Borchate Sourabh Shivaji, Prof. N. S. Hanamapure, Swapnil S. Kulkarni, "Design, Analysis and Performance Optimization of Disc Brake", International Journal of Advanced Engineering Research and Studies, E-ISSN 2249–8974, pp. 25-27, April-June 2014.
[2] Bouchetara Mostefa, Belhocine Ali, "Thermoelastic Analysis of Disk Brakes Rotor", American Journal of Mechanical Engineering, Vol. 2, No. 4, pp. 103-113, 2014.
[3] M. Sasikumar, "Design and Thermo-Structural Analysis of Disc Brake", International Journal in Physical and Applied Sciences, Volume 1, Issue 3, ISSN: 2394-5710, pp. 11-23, December 2014.
[4] N. Balasubramanyam, Prof. Smt. G. Prasanthi, "Design and Analysis of Disc Brake Rotor for a Two Wheeler" International Journal of Mechanical and Industrial Technology (IJMIT), Volume 1, Issue 1, pp. 7-12. October 2013-March 2014.
[5] Viraj Vijaykumar Shinde, Chetan Dhondiram Sagar, Baskar P. "Thermal and Structural Analysis of Disc Brake for Different Cut Patterns", International Journal of Engineering Trends and Technology (IJETT), Volume 11, No. 2, May 2014.