International Journal of Scientific & Engineering Research, Volume 5, Issue 4, April-2014 1135
ISSN 2229-5518

Finite Element Analysis of Disc Brake for Aluminium Alloys

Abhishek Kumar Tiwari, Akhilesh Kumar Tiwari, Pramod Yadav, Harigovind Singh Yadav, Shyam Bihari Lal

Abstract- The motive of undertaking this project of "Structural Analysis of Disc Brake" is to study and evaluate the performance under severe braking conditions and there by assist in disc rotor design and analysis. ANSYS package is a dedicated finite element package used for determining the variation of stresses and deformation across the disc brake profile.In this present work, an attempt has been made to investigate the effect of stiffness, strength and variations in disc brake rotor design on the predicted stress. By identifying the true design features, the extended service life and long term stability is assured. An attemot is also made to suggest a best combination of material and flange width used for disc brake rotor which has less deformation and minimum von-mises stress possible.

Index Terms—Pro/E, Ansys Workbench 14.0, structural Analysis, Disc brake, FEM

1 INTRODUCTION

A Brake is 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 ab sorb either kinetice energy of the moving member or the po tentioal energy given up by energy absorbed by brakes is dissipated in the form of heat. This heat is dissipated in to the surrounding atmosphere.

A disc brake comprises of a cast iron disc bolted to the wheel hub and a stationary housing called caliper. The caliper connected to some stationary part of the vehicle like the axle casing or the stub axke and is cast in two parts each part comprising a piston. In between each piston and the disc there is afriction pad carried in position by retaining pins spring plates etc. Passages are drilled in the caliper for the fluid to enter or leave each housing. These passages are also associated to another one for bleeding. Each cylinder holds rubber sealing ring between the cylinder and piston.

1.1 Principle:

The principal used is the applied the applied force (pressure) plays on the brake pads which comes into contact with the moving disc. At this point time due to friction the relative motion is constrained.

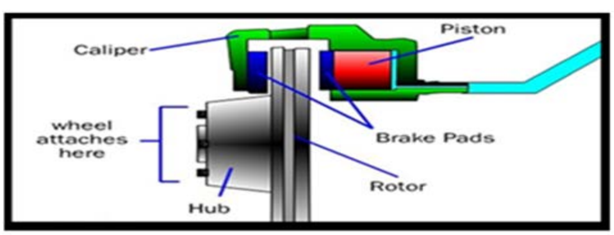

Fig.1 Disc brake working

- **Author name**- Abhishek Kumar Tiwari, Akhilesh Kumar Tiwari, Pramod Yadav ,Harigovind Singh Yadav Student of Mechanical engineering at B.I.T. Gorakhpur
- **Co-Author name-** Shyam Bihari Lal, Asst. Prof. Department of mechanical engineering, B.I.T. Gorakhpur, E-mail: shymefme@gmail.com

1.2 Working:

When the brakes are employed hydraulically actuated piston act the friction pads in to contact with the disc applying equal and opposite forces on the later. On freeing the brakes the rubber sealing ring acts returm spring and draws back the piston and the friction pads away from the disc.

The main parts of the disc brake are:

1: The brake pads.
2: The caliper which contains the piston.
3: The rotor which is mounted to the hub.

2. INTRODUCTION TO FINITE ELEMENT METHOD:

The finite element method is a mightily tool to find the numerical solution of wide range of engineering trouble. The method is general enough to deal any complex shape or geometry for any material under different boundry and loading circumstances. The generally of the finite element method fits the analysis requirement of todays complex engineering systems and design where closed from solution of govering equilibrium equation are usually not usable. In addition it is an efficient design tool by which designers can execute parametrice design studies by considering various design cases, (different shapes, materials, loads etc.)

The method origined in the aerospace industry as a tool to study sress in a complex airframe structres. It grows out of what was called the matrix analysis method used in aircraft design. The method has gained increased popularity among both or structure may be separated in to small elements of finite dimensios called finite elements the original body or the structure is then viewed as an assemblage of these elements.

3. General procedure of finite element method:

Step 1: Description of structure (domain).
Step 2: Selection of proper interpolation model:
Step 3: Ddervation of element stiffness matrices (characteristic

ISSN 2229-5518

matrices) and load vectors.

Step 4: Assemblage of element equation to obtain the equilibrium equation to obtain the equilibrium equations.

Step 5: Solution of system equation to find nodal values of displacment (field variable).

Step 6: Computation of element strain and stresses.

4. MESHING CRITERIA

Element type soliod10 node quadratic tetrahedral is shown in fig.2.

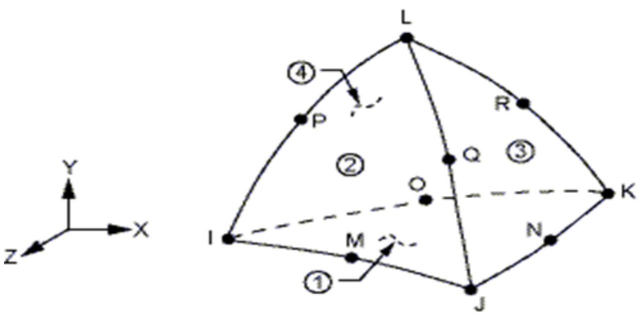

Fig.2: The 10 – node tetragonal elements (SOLID 187)

Pro/e and ansys workbench software are used for the FE analysis. The crankshaft kis modeled in 3D using pro/e software. IGES file is generated and the imported in ANSYS workbench software. The 10 node tetragonal elements (SOLID 187) were used as shown in figure finite element mesh was generated using tetragonal element with element lenth of .5 mm (2262 elements). The reason for Choosing thids element was to make the geometrical parts of a complicated mechanical compoment so enable us to gain more authentice results based on the high technique of fatigue life caculation.

SOLID187 element is a higher order 3-D 10-node element.
SOLID187 has a quadrice displacement behavior and is well suited to modeling irregular meshes (such as those produced from various CAD/ACM system) .the element is defined by 10 node having three degrees of freedom at each node. Translation in the nodal x,y, and z direction . The element has plasticity large deflection and large strain capabilities for simulating deformation of nerly incompressible elastoplastice materials and fully incompressible hyperlastice materials.

5. ANSYS

Ansys is practicable software for design analysis in mechanical engineering that's an introduction application fully integrated with pro- Engineer. This software uses the finite element method (FEM) to simulate the working condition of your design and predict their behavior. FEM require the solution of large systems of equation. Powered by fast solvers ansys makes it potentioal for designers to speedily check the integrity of their designs and search for the optimum own solution.

5.1 Procedure for ANSYS analysis:

A product development cycle typically admits the following steps.

- Build your model in the pro-Engineer system.
- Prototype the design.
- Test the prototype in the field.
- Valuate the results of the field's tests.
- Modify the design based on the field test results.

6. STRUCTURAL ANALYSIS

6.1 Introduction:

Structural analysis is the most usual application of the finite element method .The term structural (or structural) involves civil engineering .
Structures such as bridges and buildings bust also naval aeronautical and mechanical structures such as ship hulls aircraft bodies and machines housings as well as mechanical parts such as pistons machine parts and tool.

6.2 Types of structural analysis:

There are seven types of structural analysis usable in ANSYS. One can do the following type of structural analyses.

1. Static analysis.
2. Modal analysis.
3. Harmonice analysis.
4. Transient dynamice analysis.
5. Spectrum analysis.
6. Buckling analysis.
7. Explict dynamice analysis.

Table 1 properties of Aluminum alloy

Name	Aluminum Alloy
Model type	Linear Elastic Isotropic
Default Failure Criterion	Max von Misses Stress
Yield Strength	1.65e+0.008 N/m2
Tensile Strength	3.0e+0.008 N/m2
Elastic Modulus	7e+011 N/m2
Poisson's Ratio	0.33
Mass Density	2600 kg/m3
Shear Modulus	3.189e+010 N/m2

6.3 Procedure of static analysis

First of all we have prepared assembly in pro/ E for crankshaft and save as this part as IGES for Exporting in to ANSHYS workbench environement import IGES mode in ANSYS workbench simulation module.

7 Results and Discussion:

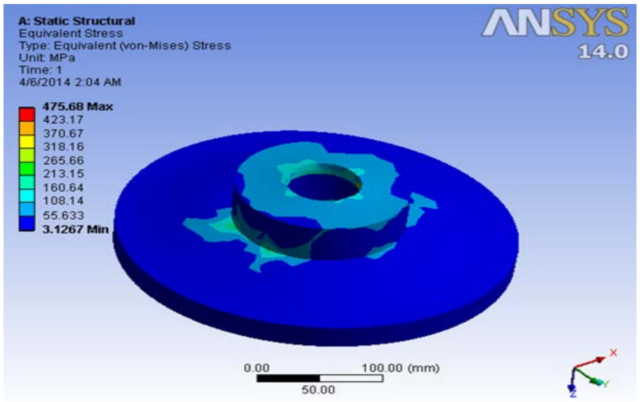

Fig.3 Equivalent stress

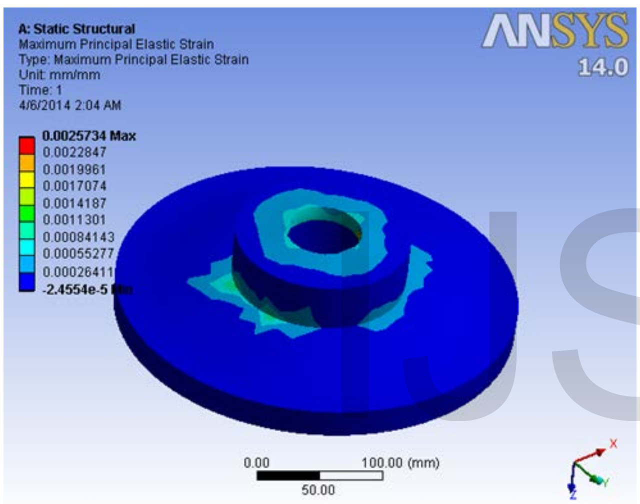

Fig.4 Maximum principal elastic strain

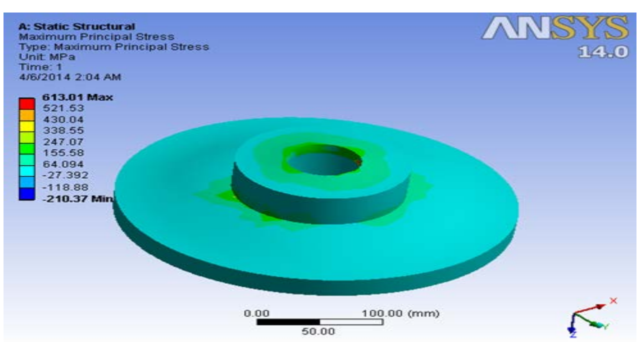

Fig.5 Maximum principal stress

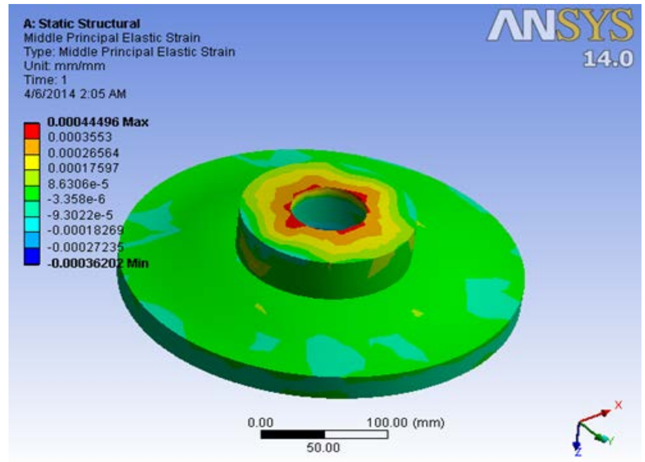

Fig.6 Middle principal elastic strain

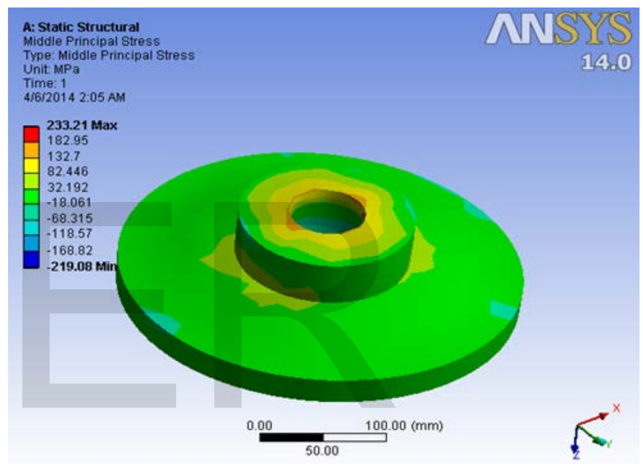

Fig. 7 Middle principal stress

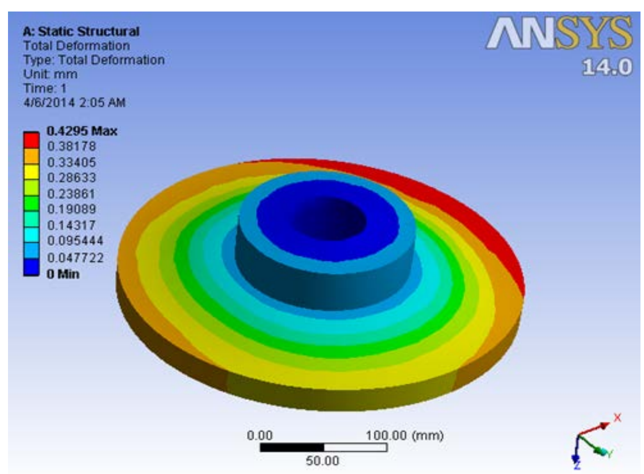

Fig. 8 Total deformation

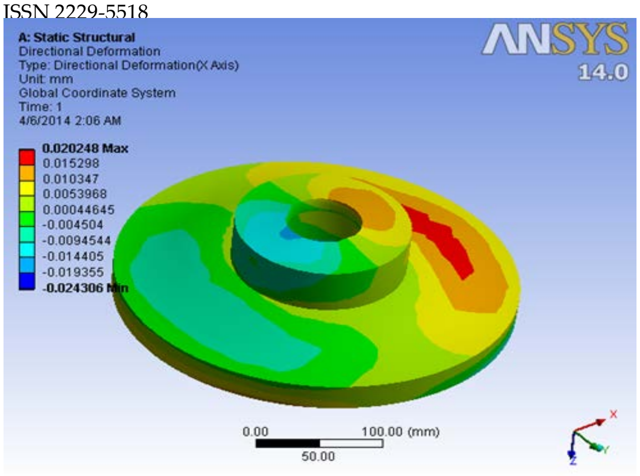

Fig. 9 Directional deformation

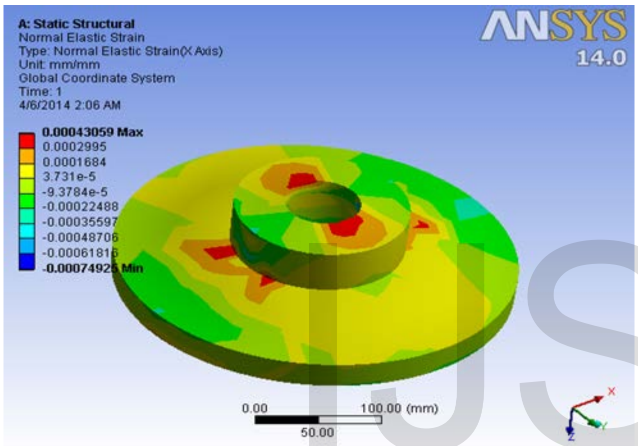

Fig. 10 Normal elastic strain

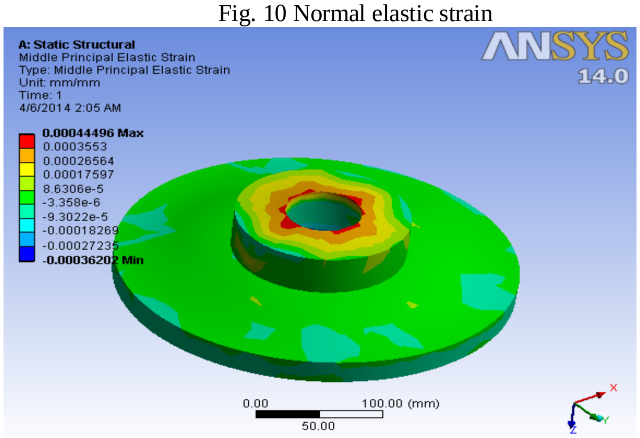

Fig. 11 Middle principal elastic strain

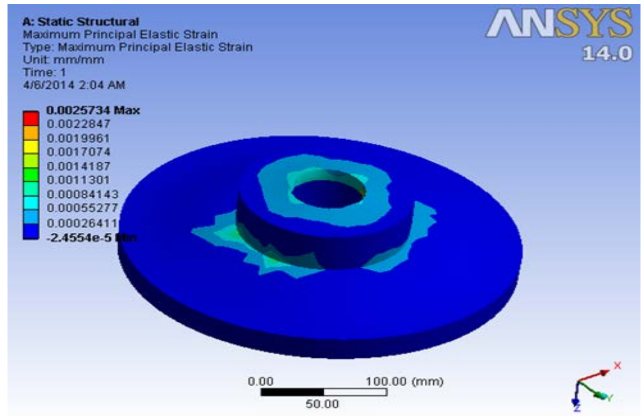

Fig.12 Maximum principal elastic strain

8. CONCLUSION

By noticing the Structural analysis results using Aluminum alloy stress assesses are within the permissible stress value. So using Aluminum Alloy is good for Disc Brake. By observing the frequency analysis, the vibrations are less for Aluminum Alloy than other materials since its natural frequency is less. And also weight of the Aluminum alloy reduces almost 3 times when compared with Alloy Steel and Cast Iron since its density is very minus. Thereby mechanical efficiency will be raised. By observing analysis results, Aluminum alloys are suitable material for Disc Brake.

9. **References**

[1] Finite Element Method, J.N.Reddy.

[2] Maitra, G.M, 2004, Hand Book of Gear Design, TataMcGrawHill, New Delhi.

[3] Liles, G. D. 1989. Analysis of disc brake squeals using finite element methods. SAE paper 891150, pp. 1138–1146.

[4] S.Mahalingam, R.E.D Bishop, 1974, "Dynamic loading of Gear tooth", Journal of sound and vibration, 36(2), pp179189 9. S.H.Choi,J.Glienicke, D.C.Han, K.Urlichs, April 1999, "Dynamic Gear Loads due to coupled lateral, Torsional and Axial Vibrations in a helicalGeared System" , Journal of vibration and acoustics, Vol 121.

[5] Zeping Wei., 2004"Stresses and Deformations in Involute spur gears by Finite Element method," M.S, Thesis, College of Graduate Studies and research, University of Saskatchewan.

[6] Mugeo Dong, Sok Won Kim, Nam-Ku, "Thermo physical Properties of Automotive Brake Disk Materials", Department of Physics, University of Ulsan pp-680 – 749.

[7] Guangqian g Wu, Lin He, Xianjie Meng, (2009), "Numerical Study on the Vibration Characteristics of Automobile Brake Disk and Pad", IEEE transactions, pp-1798-1802.