Design and Analysis of Disc Wheel Using FEA

[*]K. RATNA KUMARI, [#]V. SRINIVASREDY

[*]M.Tech Student, [#]Assistant Professor, Department of Mechanical Engineering, QIS College of Engineering &Technology, Ongole, India.

ABSTRACT - The wheel is an integral part of any automobile. Design of wheel under various loads is to be studied for practical application to ensure safety and smooth running. Here, wheel is subjected to brake loads and mechanical loads due to weight of the vehicle. In cars, Disc brakes and Drum brakes are generally used on front wheels and rear wheels respectively. Due to application of brakes large amount of heat is generated which is distributed across the wheel section. The objective of the present work is to analyze the stresses induced in the given wheel design due to braking loads and structural loads. For this work, Linear thermal analysis with temperature dependent material properties and Linear static analysis has been performed using finite element software such as ANSYS. The heat generated by the braking is applied as loading and convective heat transfer has been applied as loading and convective heat transfer has been given as external boundary condition. The temperature plots at various regions are reported. The stress analysis is carried out by switching thermal analysis to structural analysis and Von mises stresses are plotted. Further, the analysis is also extended to static loading on the wheel. In this, 3D model is generated and this full model is meshed with a 3D solid element, the deflections and stresses are plotted. In static analysis we are considering the loads due to weight of the vehicle, side thrust and pressure due to application of brakes. Finally, the stability of the wheel is discussed with the plotted results.

Keywords – FEA, Disc Wheel, ANSYS, 3D Solid.

I. INTRODUCTION

The wheel is an assembly of hub, disc (body), rim and spokes (spokes are not necessary in many types).Wheel is the most common commodity in the market that will surely not go out of stock. Its purpose is not just to serve as contact point between the car and the road but also as a means to show off the car's performance. Besides supporting the vehicle's weight and protecting it from road shocks they also transmit power and are steered. The wheels are subjected to braking stresses and side thrusts during their motion. They are, therefore, required to be strong enough. Their construction has undergone drastic changes over the last few decades. From wooden spokes to metallic spokes, pressed steel to light alloys, welded rim to detachable rim, solid wheels to hollow hub type wheels are the novelties of such changes. Presently the trend is towards use of light-alloy wheels made of aluminium and magnesium alloys. These are light weight, almost corrosion free, casted wheels. They are aesthetically very good looking and are suitable for racing car applications too. The Objective of the present work is to perform the static analysis on certain wheel component (3D model) due to vehicle loads, braking pressures with two different materials as wheel material. After plotting the stresses and deflections, we finally suggest the suitable material among them as a car wheel material based on the stresses and deflections generated in the material. Thermal analysis is carried out by considering the heat loads which are caused due to generation of heat in the brake drum during the time of application of brakes which is followed by Coupled field analysis to get thermal stresses. We have taken the dimensions of certain wheel component; accordingly, the model is generated which was done using modeling package Pro-E. After generating the model in Pro-E, it is exported to IGES format. Then the IGES format of wheel is imported to Hyper mesh which is a meshing package. Using meshing techniques in Hyper mesh the meshed model created. After the completion of meshing the meshed geometry is exported to IGES format. For the analysis of IGES format of meshed wheel, it is imported to Ansys which is an analysis package. Static and Thermal analysis of wheel is done with the help of Ansys software.

II. FINITE ELEMENT METHOD

The basic idea in the Finite Element Method is to find the solution of complicated problem with relatively easy way. The Finite Element Method has been a powerful tool for the numerical solution of a wide range of engineering problems. Applications range from deformation and stress analysis of automotive, aircraft, building, defence, and missile and bridge structures to the field analysis of dynamics, stability, fracture mechanics, heat flux, fluid flow, magnetic flux, seepage and other flow problems. With

 DOI : 10.35291/2454-9150.2019.0415

the advances in computer technology and CAD systems, complex problems can be modeled with relative ease. Several alternate configurations can be tried out on a computer before the first prototype is built. The basics in engineering field are must to idealize the given structure for the required behavior. The proven knowledge in the computational aspects of the Finite Element Method is essential. In the Finite Element Method, the solution region is connected as built up of many small, interconnected sub regions called finite elements.

Although the name finite element method was given recently, the concept has been used several centuries back. For example, ancient mathematicians found the circumference of the circle by approximating polygon inscribed or circumscribed and can obtain a lower bound or an upper bound for the true circumference. Further, as the number of sides of the polygon is increased, the approximate values convert to the true value. These characteristics will hold true in finite element application. In recent times an approach similar to the finite element method involving the use of piece wise continuous functions defined over triangular regions, was first suggested by R. Courant in 1943 in the literature of applied mathematics.

III. FEA - SOFTWARE ANSYS

The rapid advances made in computer hardware and software led to significant developments in finite element analysis software. A number of general purpose finite element analysis software packages with a processor capability and facility for the user to have wide choice of several types of elements, analysis of different types of problems like static, dynamic, material and geometric non-linear, coupled situations, heat transfer, interaction problems etc, and pre and post processing features have been developed .the names of some of the popular packages are: ABAQUS, ADINA, ANSYS, ASKA, COSMOS. NISA, PAFEC, SAP, SESAM-80 etc.

The ANSYS program was introduced by Dr. John Swanson, Swanson analysis systems Incorporated. (SASI) in 1970.The ANSYS software has many finite element analysis capabilities ranging from a simple linear static analysis to complex nonlinear transient dynamic analysis.

3.1 ADVANTAGES OF ANSYS

ANSYS finite element analysis software enables engineers to perform the following tasks:

1. Build computer models or transfer CAD models of structures, products, components, or systems.
2. Apply operating loads or other design performance conditions.
3. Study physical responses, such as stress levels, temperature distributions, or electromagnetic fields.
4. Optimize a design early in the development process to reduce production costs.
5. Do prototype testing in environments where it otherwise would be undesirable or impossible (for example, biomedical applications).

The ANSYS program has a comprehensive graphical user interface (GUI) that gives users easy, interactive access to program functions and commands.

The component wheel is modeled using PROE as shown in the Fig. Using part drawing option, the wheel is generated. First the cross-section of the wheel is drawn, and then this cross-section is arranged at some distance to the axis. This cross section is rotated by 360 degrees to get the basic view of the wheel. This structure is then exported to IGES format for further work.

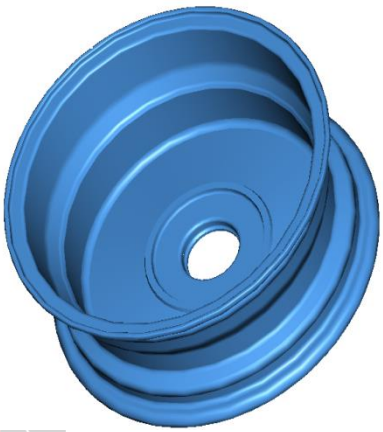


Fig.1: A Disc Wheel Modeled in PRO-E

Assigning Material Properties: Aluminium

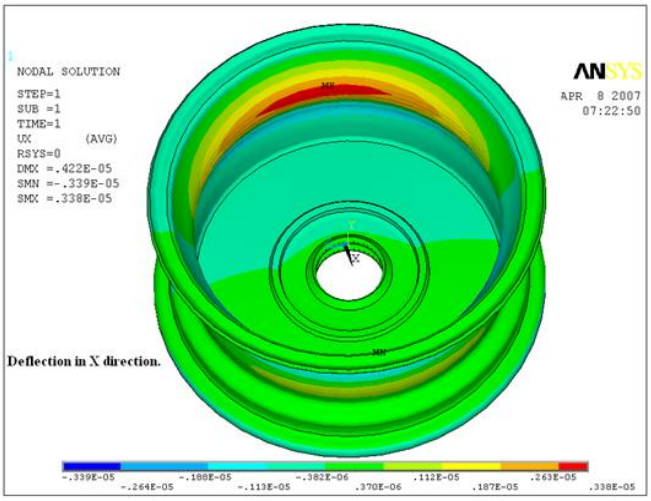
Density	:	2830Kg/m3
Modulus of Elasticity	:	71.7Gpa
Poissons ratio	:	0.33
Ultimate tensile strength	:	552Gpa
Yield tensile strength	:	490Gpa
Thermal Conductivity	:	153w/m0C
Specific heat	:	860 j /kg0C
2. Steel		
Density	:	7845Kg/m3
Modulus of Elasticity	:	200Gpa
Poissons ratio	:	0.29
Ultimate tensile strength	:	485Gpa
Yield tensile strength	:	450Gpa
Thermal Conductivity	:	50.7w/m0C
Specific heat	:	515 j /kg0C

IV. RESULTS AND DISCUSSIONS

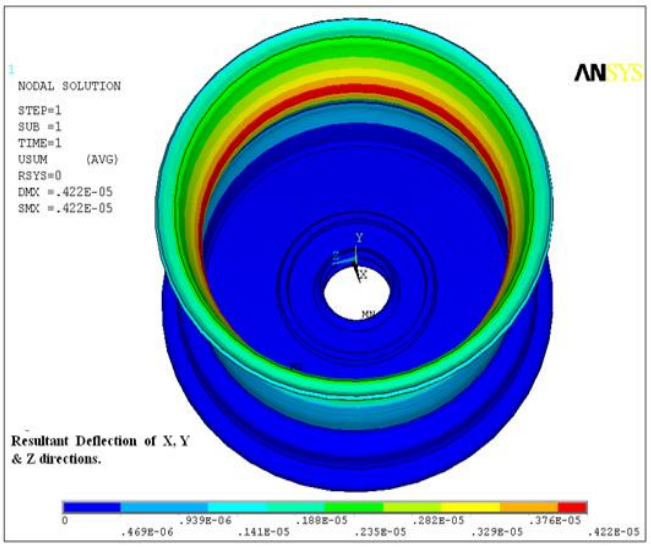
The following conclusions are drawn from the present work.

Static Analysis (Steel Alloy):

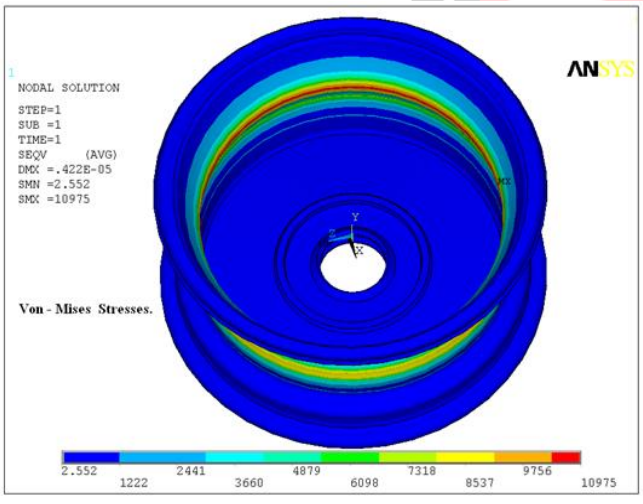
 DOI : 10.35291/2454-9150.2019.0415



Deflection in X direction.

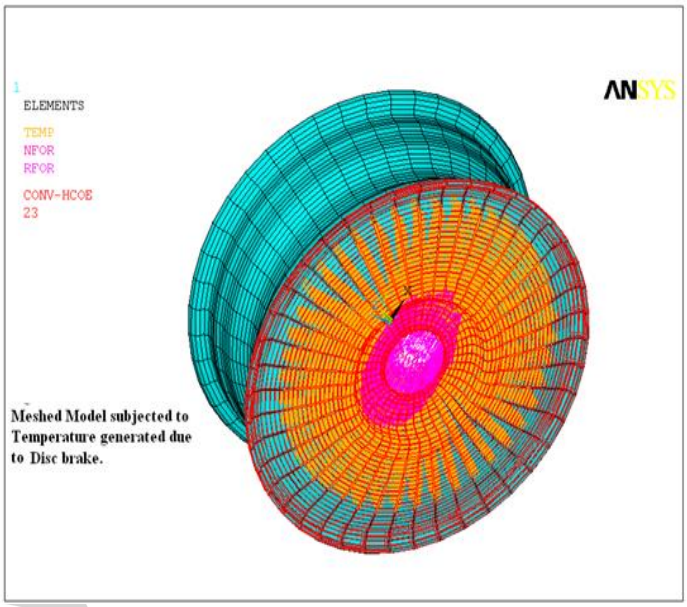


Resultant Deflection of X, Y & Z directions.



Von - Mises Stresses.

1. The Min. & Max. values of deflections in X direction are - .339E-05 m, .338E-05 m.
2. The Min. & Max. values of deflections in Y direction are - .400E-06 m, .271E-05 m.
3. The Min. & Max. values of deflections in Z direction are - .338E-05 m, .337E-05 m.
4. The resultant deflection is .422E-05 m.
5. The Min. & Max. values of stresses in X direction are - 6523N/m^2, 3188N/m^2 .
6. The Min. & Max. values of stresses in Y direction are - 7435N/m^2, 7305N/m^2 .
7. The Min. & Max. values of stresses in Z direction are 6537N/m^2 , 3197N/m^2 .
8. The Min. & Max. values of Von – mises stresses are 2.552N/m^2 , 10975N/m^2



Meshed Model subjected to Temperature generated due to Disc brake.

V. CONCLUSIONS

1. By using Aluminum and steel Materials Static and Thermal analysis was done to see stress and temperatures of Disc wheel.

2. by using aluminum and Steel Materials analyzed couple field analysis (structural and Thermal Analysis

3. by comparing Results of Steel and Aluminum. Finally, we can say Aluminum is better than Steel.

4. Maximum stress induced is within safe limit.

5. Maximum thermal stresses are setup when the temperature difference is maximum from outside to inside.

REFERENCES

[1] R.S.KHURMI and J .K.GUPTA, (2008). Machine Design, Division of S. Chand and Company Ltd, Ram Nagar New Delhi.

[2] Joe Y.G. Cha, Analysis of disc brake instability due to friction induced vibration, International of automotive technology, vol 9, No 2, PP 169-171.

[3] M. Bayat, The effect of ceramic in combination of functionally graded rotating disc, International journal of 110 Mahmood Hasan Dakhil, A. K. Rai, P. Ravinder Reddy & Ahmed Abdulhussein Jabbar computational method, vol 9, No 2,2012,22 pages.

[4] Utz von Wagner, (2011) Influence of dynamic brake pad properties on automotive disc brakeSqueal, PAMM, vol 11, Issue 1, Paper 345-346.

 DOI : 10.35291/2454-9150.2019.0415

[5] Haripal Singh and Harsh Deep Shergill, (2012) Thermal Analysis of Disc Brake Using Comsol International Journal on Emerging Technologies.

[6] Daniel Das. A, Christo Reegan Raj. V, Preethy. S and Ramya Bharani.G. (2013). Structural and Thermal Analysis of Disc Brake in Automobiles."International Journal of Latest Trends in Engineering and Technology.

[7] V. Chengal Reddy, M. Gunasekhar Reddy and Dr. G. Harinath Gowd (2013), Modeling and Analysis of FSAE Car Disc Brake Using FEM.International Journal of Emerging Technology and Advanced Engineering.

[8] Guru Murthy Nathi1, T N Charyulu ,K.Gowtham and P Satish Reddy,(2012), COUPLED STRUCTUAL THERMAL ANALYSIS OF DISC BRAKE ,GURU MURTHY NATHI, et al ISSN: 2319 – 1163 Volume: 1 Issue: 4

[9] K. NareshBabu and T.Siva Krishna, (2013) STRUCTURAL DESIGN AND THERMAL ANALYSIS OF CARDISC BRAKE ROTOR. Babu, et al, International Journal of Advanced Engineering Research and Studies.

[10] V. M. M. Thilak, R. Krishnaraj, Dr. M. Sakthivel, K. Kanthavel, Deepan Marudachalam and M.G, R.Palani, (2011)," Transient Thermal and Structural Analysis of the Rotor Disc of Disc Brake", International Journal of Scientific & Engineering Research Volume 2, Issue 8 .

[11] British Standards Institution (1990). BS EN 10 002-1:1990, 'Tensile Testing of Metallic Materials: Parti. Method of test at ambient temperature',

[12] Saiprasit Koctniyom (2000)," Thermal Stress Analysis of Automotive Disc Brakes"

[13] G. Cueva, A. Sinatora, W. L. Guesser and A .P .Tschiptschin, (2003). Wear resistance of cast irons used in brake disc, Wear 255 (1256-1260.

[14] Cobb, Harold M, (2010).The History of Stainless Steel.ASM International.p. 360. ISBN 1-61503-0107.

[15] WiczO. C. Zienkie and R. L. Taylor, (1999). The Finite Element Mothed Fifth Edition Volume 1: The Basis. Bulletin of the institute of mathematics, V35, No 4, pp.110, 114

 DOI : 10.35291/2454-9150.2019.0415