



IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

“STATIC STRUCTURAL ANALYSIS OF CAR WHEEL USING FINITE ELEMENT METHOD - A REVIEW”

¹Namit Baraskar, ²Anil Elisala

1PG Scholar, Mechanical Engg. Department, B.M. Engg. College, Indore, M.P, India,

2 Associate Professor, Mechanical Engg. Department, B.M. Engg. College, Indore, M.P, India,

ABSTRACT

In this paper, the review about various papers which is based on analyzed with the finite element method and experimental works, using the different loading test. The static stresses are studied in order to find the zones with higher stress concentration and fatigue life prediction are studied in order to improve the life cycle of wheel and also to suggest the better design solution.. The purpose of the car wheel rim provider's a firm base on which to fit the tire. Its dimensions, shape should be suitable to adequately accommodate the particular tire required for the vehicle. In this study a tyre of car wheel rim belonging to the alloy wheel category is considered. The development of the vehicle industry has strongly influenced the design, the material selection and the manufacturing processes of the wheels. The wheels loading manner is a complex one; further improvement and efficient wheel design will be possible only if their loading will be better understood.

KEYWORDS: Alloy wheel, Rim , Static state, Fatigue, FEM.

I. INTRODUCTION

Wheel rim is an inseparable part of an automobile mounted on the axle hub of a vehicle. Its main functions are to rotate over the axle of a automobile so as to use power from engine to propel automobile, provide support for braking system over its body, dissipate heat generated in the body of wheel rim to surrounding environment, support whole body weight as well as withstand against impact load due to pot holes and road irregularities with constant load of rider body as well as constant tyre pressure.



Fig.1 Wheel Rim with Tyre

1.1 TYPES OF WHEEL (MATERIAL)

Steel and light alloy are the foremost materials used in a wheel rim however some composite materials together with glass-fibre are being used for special wheels.

A. Wire spoke Wheel

Wire spoke wheel is an essential where the exterior edge part of the wheel rim and the axle mounting part are linked by numerous wires called spokes. Today's automobiles with their high horse power have made this type of wheel manufacture obsolete. This type of wheel is still used on classic vehicles.

B. Steel Disc Wheel

This is a rim which practices the steel made rim and the wheel into one by joining (welding), and it is used mainly for passenger vehicles especially original equipment tires.

C. Light Alloy wheel

These wheels are based on the use of light metals, such as aluminium and magnesium has come to be popular in the market. This wheel rapidly become standard for original equipment vehicle in Europe in 1960's and for the replacement tire in United States in 1970's. The advantages of each light alloy wheel are explained as below.

D. Aluminium Alloy Wheel

Aluminium is a metal with features of excellent lightness, thermal conductivity, physical characteristics of casting, low heat, machine processing and reutilizing, etc. This metal main advantage is decreased weight, high precision and design choices of the wheel.

E. Magnesium alloy Wheel

Magnesium is about 30% lighter than aluminium and also admirable as for size stability and impact resistance. However its use is mainly restricted to racing, which needs the features of weightlessness and high strength. It is expensive when compared with aluminium

F. Titanium alloy wheel

Titanium is an admirable metal for corrosion resistance and strength about 2.5 times compared with aluminium, but it is inferior due to machine processing, designing and more cost. It is still in developed stage.

G. Composite material wheel

The composite material wheel is different from the light alloy wheel, and it is developed mainly for low weight. However this wheel has inadequate consistency against heat and for best strength

II. LITERATURE REVIEW OF PREVIOUS WORK

Liangmo Wang and Yufa Chen - Chenzhi Wang [1] have improved the quality of aluminum wheel rim using a new method for evaluating the fatigue life of aluminum wheel rim. The ABAQUS software was use in the project and it build the static load finite element model of aluminum wheel rims for simulating the rotary fatigue test. The equivalent stress amplitude of model was calculated based on the nominal stress method in the software by considering the effects of mean load, size, fatigue notch, surface finish and scatter factors in model. The fatigue life of aluminum wheel rims was determined by using the equivalent stress amplitude and aluminum alloy wheel in S-N curve. The results from the aluminum wheel in rotary fatigue bench test show in the baseline wheel rim failed the test and its crack generating was around the hub bolt hole area of wheel rim that agreed with the simulation of model. The wheel rim life cycle was improved by calculation and satisfied the design requirement. The results showed that the proposed method of integrating finite element analysis and nominal stress method was a good and efficient to predict the fatigue life of aluminum wheel rims.

T. Siva Prasad and T. Krishnaiah [3] are discuss to summary of vehicle wheel rim provides a base of rim structure on which tire is fitted. And dimensions of automotive and shape of wheel rim should be suitable for particular tire required for the vehicle. This project show vehicle wheel rim belongs to the tire. Design is most important for every industry which impacts the quality of the product. The wheel rim model is creating by using modeling software CATIA. Using CATIA software the time spent in creating 3-D models and the risk inset in the design and manufacturing process of wheel rim can be easily reduced. After creating complete model of wheel rim is import to ANSYS for analysis work. ANSYS is the software which is used for simulating and analysis the different forces, pressure load acting on the model and also calculating and see the results very easily. ANSYS static analysis work is carried on wheel rim model by considering two types different materials like aluminium and forged steel. and we find relative performances of wheel rim with help of ANSYS software. Wheel rim is acting to modal analysis; this is part of dynamic analysis. This paper observe the results static and dynamic analysis obtained forged steel is suggested as best material.

V.Karthi and N. Ramanan [4] are discuss to the design the motor cycle alloy wheel rim using the PRO-E design software and Analyzed with the help of ANSYS. ANSYS is a tool which is used for the evaluation of systems and structures of models. It is used to analyze complex structures of model very easily. There are three method used in the project which are preprocessing, analysis all loads and visualization. The author select material was an magnesium alloy, aluminium alloy, titanium alloy. The aluminum alloy is better to the steel metal wheel rims in durability and strength. These alloy have excellent wear resistance, anticorrosion properties and it have longer service life as tested by the stress frequency distribution. The analysis is done in software with the higher load can be applied on wheel rim. In every vehicle rear wheel and front wheel have the higher load that can be supported. The stress of the analysis is show result in the range of the yield strength of Aluminum alloy. The Displacement of alloy wheel is at the low value. This design is have in the safe condition.

Gaurav Machave and Pote Susheel Sambhaji [5] are describe the use of the study of pressure and load applied on wheel rim by experimental method author using Radial Fatigue Test (RFT) and finite element method used for analyzing stresses on wheel rim and displacement distributions in wheel rims of automobile when wheel to the conjoint influence pressure and

radial load on wheel. The commonly use thought in the design of the wheel rim. A potentially viable method for finite element modeling with the help of software subjected to loads is highlighted on wheel.

Saurabh M Paropate [6] is discuss that Alloy wheel rims which are made from an aluminum alloy or magnesium alloy Metals and also used mixture of both material. Alloy wheels differ from conventional steel wheel rims because it has lighter in weight, which improves the driving and handling of the automobile. Alloy wheels made up of mixture of different metal elements it reduce the unwanted weight of a vehicle like aluminum alloy wheels. The benefit of reduced unwanted weight of vehicle is more comfortable handling and reduction in fuel consumption by the vehicle. Alloy is an good conductor of heat, It improve heat dissipation from the Brakes and reducing the risk of brake failure condition. Motor cycle wheel rims are made of Aluminum Alloys. Author compared between aluminium alloy and other Alloy metals and composite materials. A parametric model is designed with help of software for Alloy wheel used in motorcycle from existing model. Wheels have static load and fatigue loads. This load develops heavy stresses in the wheel rim and thus it is necessary to find the critical stress point and shear stress in the wheel rims. The model has to be built by software and, loads are applied on wheel rim and solutions are obtained. It use model of motorcycle Bajaj pulsar 150 cc.

S Vikranth Deepak and C Naresh [7] are describe Alloy wheels are which are made from of aluminum alloy or magnesium alloy metals and also made from mixture of both alloy. Alloy wheels rim are generally different from steel wheel rims. It have lighter in weight, and it improves the handling of the car. Alloy wheels will reduce the unwanted weight of a vehicle compare standard steel wheels. The benefit of reduced weight is more accurate handling and reducing in fuel consumption by the vehicle. Alloy is an good conductor of heat, it improve heat dissipation from the braking condition, reducing the risk of brake failure under in driving conditions. At present vehicle wheel rims are made of Aluminum Alloys. In the project, author is compare Aluminum alloy and with other Alloy. In this project a model is designed with the help software. Design of wheel is analyzed and it taking ultimate stresses in two different alloy materials and different loads condition. The author used model of Ford Fiesta.

J. Janardhan and V. Ravi Kumar [8] are deals that importance of wheel in the automobile. The vehicle can be towed without the any engine but it is not possible without the wheels, the wheel rims and the tire have to suffer all vehicle loads, also provide the steering control. The main requirements of vehicle wheel are; it should be strong enough to all performance and functions. It must be balanced both condition statically and dynamically. It also should be light weight and reduce the unwanted weight is least. The Wheel rims have passed three types of load tests before going into production of wheel rim, they have acting fatigue test, Radial fatigue test and Impact load test. In the project, it had done radial fatigue analysis to find the number of cyclic load when the wheel is getting to fail. The 2D model of the wheel rim was created in MDT software and after that these model of wheel rim was exported to ANSYS, the finite element method used. IGES translator is using for where the 3D model of the wheel is created. The wheel rim is meshed using SOLID 45 element in ANSYS.

Ch. P. V. Ravi Kumar [9] are discuss that wheel rim load tests are required for designing of wheel rim and manufacturing of wheel rims is much requirements of tests. The impact loads performance of a wheel rim is the major problem. Numerical method implementation of impact load test is very necessary to shorten the design time, and improving performance of wheels and it reducing the cost. This project has decides the “Topology Optimization of Aluminium Alloy wheel rim” using

analysis. From the failed value of plastic strain for Aluminium Alloy Wheel rim is 4.0%, cracks will seen if the Plastic Strain value of alloy is greater than 4%. This analysis will show the plastic strain value is induced during impact load testing. Topology Optimization method has carried by increasing the thickness of the wheel rim at the plastic strain value show is below 4%. The main target of the project is to create a Finite element model Hexa and Penta elements using Hypermesh with, materials, loads and Boundary conditions are applied on model. Impact load analysis is carried by the using LS-Dyna software, it show the plastic strains of model during impact test. Topology Optimization is changing the thickness of the wheel rim of the Aluminium Alloy Wheel have the value of plastic strain is less than 4.0%.

III. CONCLUSION

Wide varieties of materials are available in the market which can be used for the wheel rim. Generally used wheel rim materials are Al alloy, Mg alloy, Steel C 1008, Forged Steel. Each material has some advantages over the other. If original equipment manufacturers require excellent aesthetic shape with very good heat dissipation without compromise with its associated costs then light weight material such as Al and Mg alloys can be used for wheel rims. But we will be used our proposed work composite materials wheel which will be analyzed in ANSYS design software.

REFERENCES

1. Guo M, Bhandarkar R, Lin B, Society of Automotive Engg., 2004Warrendale, Pennsylvania.
2. Liangmo Wang and Yufa Chen - Chenzhi Wang, "Fatigue life analysis of aluminium wheels by simulation of rotary fatigue test" *Journal of Mechanical Engineering*, 57(2011), 31-39.
3. C.L. Chang and S.H. Yang, "Finite element simulation of wheel impact test", *Journal of Achievement in Materials and Manufacturing Engineering*, vol 28(2), June 2008.
4. T. Siva Prasad and T. Krishnaiah, "A review on modeling and analysis of car wheel rim using CATIA and ANSYS" *International Journal of Innovation Science and Modern Engineering*, vol 2/Issue-6, ISSN: 2319-6386 (2014).
5. V.Karthi and N. Ramanan, "Design and analysis of alloy wheel rim" *International Journal of Innovation Research in Science Engineering and Technology*, vol 3/Issue-2, ISSN: 2319-8753 (2014).
6. Gaurav Machave and Pote Susheel Sambhaji, "Study of influence of pressure and load on wheel rim by radial fatigue test", *International Journal of Innovation Research in Science Engineering and Technology*, 4(2), ISSN 2277-9655 (2015).
7. Saurabh M Paropate, "Modelling and analysis of a motorcycle wheel rim" *International Journal of Mechanical Engineering and Robotics Research*, vol 2/no.3, ISSN: 2278-0149 (2013).
8. S Vikranth Deepak and C Naresh, "Modeling and analysis of alloy wheel for four wheeler vehicle" *International Journal of Mechanical Engineering and Robotics Research*, vol 1/no.3, ISSN: 2278-0149 (2012).
9. J. Janardhan and V. Ravi Kumar, "Radial fatigue analysis of an alloy wheel" *International Journal of Engineering Research and Applications*, vol 4, Issue-12 pp 253-258 (2014).
10. Ch. P. V. Ravi Kumar, "Topology optimization of aluminium alloy wheel", *International Journal of Modern Engineering Research*, vol. 3, Issue-3, ISSN: 2249-6645, pp 1548-1553.
11. S.Ganesh and Dr.P.Periyasamy, "Design and analysis of spiral wheel rim for four Wheeler", *The International Journal of Engineering and Science*, Volume 3, Issue 4, ISSN: 2319-1813 (2014).

12. M. Saran Theja and M. Vamsi Krishna, “Structural and fatigue analysis of two wheeler lighter weight alloy wheel”, *International Journal of Mechanical and civil Engineering*, vol. 8, Issue-2, PP 35-45 (2013)..