Review on Fatigue Analysis of Aluminum Alloy Wheel under Radial Load for Passenger Car

[1] V.Dharani kumar, [2] S.Mahalingam, [3]A.Santhosh kumar
[1] PG Scholar, [2] Asst. Professor, [3]PG Scholar
[1]Department of Mechanical and Production Engineering
[1]Sathyabama University, Chennai, India.

***Abstract* – In automobile industries aluminum is used to manufacturing different parts of motor vehicles. The major advantage was found in aluminum is less weight with high strength. This aluminum replaces steel wheels due to its physical and chemical property. The aluminum alloy A356 wheel will subject to different loads under running conditions. Fatigue is most important factor to be considered for aluminum alloy wheel. In this paper, fatigue life of aluminum alloy wheel A356 has been reviewed about the radial loading and test conditions. The aluminum alloy wheel model was designed by using Solid works 14 software. Then 3D model was converted into IGES file then imported to ANSYS, we found the result of static and fatigue analysis of alloy wheel. Also we find out the life of wheel, factor of safety by using S-N curve.**

***Index Terms* - Aluminum Alloy Wheel A356, Solid Works 2014, ANSYS 14.5**

I. INTRODUCTION

The wheel is major component in vehicles which bears weight as well as maintain the contact between the body and road condition. While designing a wheel assembly the two major factor had to be considered such as safety and engineering criteria. The major engineering factors are impact load, vibration and weight of vehicle. Under different load and stress conditions, the wheel will be subjected to fatigue changes. Aluminum alloy wheels will give more strength during service period. The aluminum alloy wheels are made by using various techniques such as sand casting; gravity die casting, centrifugal, squeeze and low pressure die casting. Low pressure die casting method is best for producing aluminum alloy wheel for significantly reducing cavities, porosity, machining time and uneven shrinkage. Fatigue is important point of the wheel. The rotary fatigue test is used to detect the strength and fatigue life of the wheel. In this test we found the wheel rim strength and dynamic life of wheel.

II. LITERATURE SURVEY

Emmanuel et al (2014) investigated Radial fatigue test of automobile rim by using ansys from the investigation it was found that the maximum stress concentration occurred at the spokes and ventilation hole of the rim under loading condition [1]. Torgal et al (2012) investigated the Stress Analysis of wheel rim was found the fatigue crack initiation occurs at the most stress concentrated regions of the wheel air ventilation holes [2]. Liangmo Wang et al (2010) Fatigue Life Analysis of Aluminum Wheels by Simulation of Rotary Fatigue Test was found the results showed that the maximum stress area was located in the hub bolt hole area [3]. Satyanarayana et al (2012) Fatigue analysis of aluminum alloy wheel under radial loads from the investigation it was found that the fatigue life of alloy wheel using S-N curve [4]. Muhammet cerit et al (2010) Numerical simulation of dynamic side impact test for an aluminum alloy wheel from that investigation was found that the maximum stress takes place in the lug region of the wheel [5]. Sourav Das et al (2014) Design and weight optimization of aluminum alloy wheel form that investigation was found the optimization and evaluation the life of the aluminum wheel [6]. M. Saran Theja et al (2013) Structural and fatigue analysis of two wheeler lighter weight alloy wheel from the investigation it was to reduce the weight of the alloy wheel has been achieved [7]. B.anusha srikanta et al (2012) Geometrical and material optimization of alloy wheel for four wheeler [8]. P.Praveen et al (2014) Geometrical optimization and evaluation of alloy wheel four wheeler from that investigate the alloy wheels properties and to comparing other materials then we found the optimization of wheel [9]. P.Meghashyam et al (2013) Design and analysis of wheel rim using catia and ANSYS from that investigation we found the aluminum wheel rim is subjected to more stress compared to forged steel [10]. S Vikranth Deepa et al (2012) Modelling and analysis of alloy wheel for four wheeler vehicle from that we found the procedure for modeling the wheels and analysis with use of FEA [11]. Trapti Sharma et al (2013) Failure analysis of wheel rim from the investigation was found the obtained scrap is more as compared to earlier process so we earn more profit with perfect wheel [12].

III. EXPERIMENTAL INVESTIGATION

The experimental investigation includes discussion about physical properties of aluminum alloy, Radial fatigue test, and fatigue analysis.

Aluminum Alloy Wheel

Steel and light alloy are the main materials used in a wheel, and some of the composite materials are included such as glass-fiber are used for special wheels. Aluminum wheels have better performance than steel wheels. Aluminum is a metal with features

of light weight, low heat conductivity, no need for coting due to low corrosion resistance, low temperature, and recycling. The benefit of aluminum alloy to reduce weight is more precise steering as well as a nominal reduction in fuel consumption. Aluminum Alloy is an excellent heat conductor, then minimizing the brake failure under running conditions. There is an improvement in the steering response and road holding, especially when cornering. A physical property of aluminum wheel is shown in Table 1. The wheel formation and machining process is simple comparing to steel wheel. Aluminum wheel is single piece to forming the wheel; the forming procedure is Billet, Disk type forging, bore piercing, Spinning and heat treatment and machining.

Radial Fatigue Test

The test machine shall be equipped with a means of imparting a constant radial load as the wheel rotates. In the radial fatigue test is used to found the strength of the wheel and find out where maximum stress occurs under running condition. The systematic setup of the RFT test is shown in figure 1. The testing wheel axle is placed parallel to the drum axle is shown. The testing wheel tyre is directly contact to the rotating drum, the loads are applied through the drum the loads are depends on the wheel size.

The recommended minimum external diameter of the drum is 1.2 m for wheels with rolling radius of less than 0.5 m and 2.4 m for larger wheels. The test wheel (single application) and tyre fixture shall provide loading normal to the external surface of the drum and in line radially with the center of the test wheel and the drum. After testing the wheel is moves the NDT process then indicated by propagation of a crack existing prior to test, new visible cracks penetrating through a section, or the inability of the wheel to sustain load.

In this paper reviewed the concept of radial fatigue testing for estimating the fatigue life of the aluminum alloy wheel to conducting the manual testing and comparing to same with that of finite element analysis. Fatigue life prediction using the stress approach is mostly based on local stress, because it is not possible to determine nominal stress for the individual critical areas [4]. The finite element analysis method, using ANSYS software, is an effective approach to predicting the failure mode of an automotive components of wheel during the wheel design stage [1].

Table 1: The composition of aluminum alloy A356 [3]

Elements	**Si**	**Mg**	**Ti**	**Fe**	**Cu**	**Zn**	**Al**
Weight [%]	6.5~7.5	0.25~0.45	0.08~0.20	≤0.2	≤0.1	≤0.1	Others

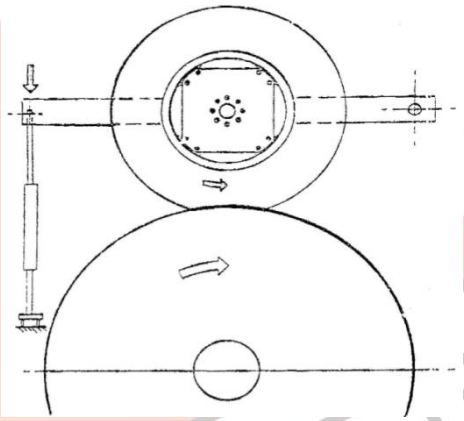


Figure1. Dynamic radial fatigue test

Fatigue Analysis

Under cyclic loading conditions fatigue is the major consideration for the materials. Crack propagation is the major cause for fatigue failure of static and cyclic loading materials. To overcome such failure radial test and FEA which provide the better way in design concepts. Experiment has shown that large percentage of structural failure are attributed to fatigue and as a result, it's an area which has been and will continue to be the focus of both fundamental and applied research.

Fatigue Analysis Using Fea Package

The computed stress–strain response is used to find out the fatigue crack initiation life using effective strain range parameters. The finite element is a mathematical method for solving numerical equations. It has the ability to solve complex problems and it will be represented in differential equation form. FEA has a history of being used to solve complex and cost critical problems. In another definition it is a time consuming process as well as solving approximate solution. Classical methods alone usually cannot provide adequate information to determine the safe working limits of a major civil engineering construction or an Automobile or a Nuclear reactor failed catastrophically the economic and social costs would be unacceptably high. In recent years, FEA has been used to solve structural engineering problems, fluid flow problems and dynamic problems. In our case to found the result in static structural analysis.

Based on the previous static analysis results, equivalent symmetric cycle stress amplitude of maximum stress amplitude node can be calculated. Considering the equivalent stress in the component S-N curve, the fatigue life is obtained [3]. To validate the prediction of fatigue life, wheel rotary fatigue test was conducted. The test results showed that the prediction of fatigue life using FEA was consistent with the result of bench test [3].

In this paper reviewed the wheel under radial fatigue test to finding the fatigue of the test wheel. In this test very useful for finding the strength of the aluminum wheel. The maximum bending moment was calculated by the following equation then applied

load to the wheel rim by using the RFT machine, the testing procedures are followed by the testing standard. If the manual testing process to finding the maximum stress of the wheel complete by 5 to 6 hours. In this study we using the finite element analysis by using ANSYS software to carry out radial fatigue test easy to finding the maximum stress of the wheel, in the process complete by one hour it is a time consuming process. If we using FEA easy to finding the deformation of the wheel as well as principal stress and equivalent stress of the wheel to found in fraction time.

Modeling and Analysis

The wheel design has done by using Solid works 2014 software. In this design created by using the sketch features are including extrude, extrude cut, revolving, holes, chamfer, roundness, shell, drafting, etc. the dimensions of the wheel is shown in table 2

Table 2 Wheel Specification [4]

Wheel Specifications	
Rim diameter	431.8mm
Rim width	152.4mm
Offset	45mm
PCD	100mm
Hub diameter	135mm

The present work was estimating the fatigue life of aluminum alloy wheel A356 by conducting manual machine testing under cycle loads and comparing the same test by using FEA. Fatigue life prediction using the stress approach is mostly based on local stress, because it is not possible to determine nominal stress for the individual critical areas.

In another way to find out the fatigue life of wheel by using the S-N curve. It is the relationship between the stress and the number of cycle. The maximum stress was found by ansys and the cycle is recommended. In this method loads are calculated by depends on the wheel and tyre size. The radial load Fr shall be determined from the formula:

$$Fr = Fv \cdot k \text{ [N]} \qquad (1)$$

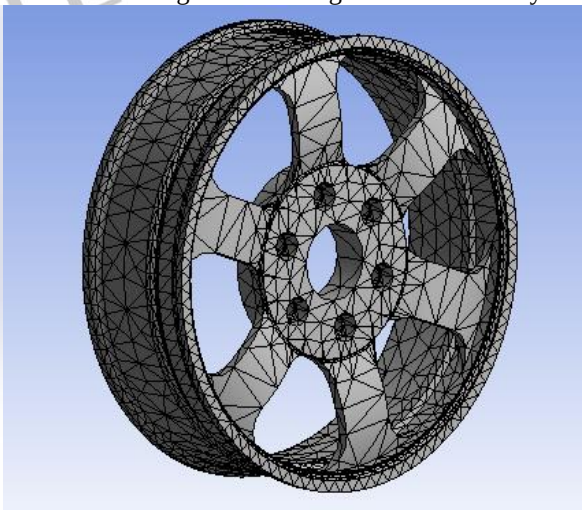
Where Fv is the applicable maximum rotating load in "N" and k is the constant accelerating load factor is 1.43. If the bending moment of the wheel by using equation 1. In the fatigue life evaluation is important factor to design the aluminum alloy wheel. If we design the new wheel first to develop the prototype, then that wheel should be passing the certain test. The commonly we following three critical test, there are radial fatigue test, cornering fatigue test and impact test. In this paper should contain the radial fatigue test for finding the wheel strength. If test wheel should pass the required test then to moving the manufacturing process. The alloy wheel production process is simple as well as comfortable compare then steel wheel. Since alloy wheels are designed for variation in style and have more complex shapes than regular steel wheels, it is difficult to assess fatigue life by using analytical methods.

The wheel three dimension diagrams was created by using solid works 2014 and the file was saved in IGES format into ANSYS. The 3D modal that was shown in figure 2. In this paper investigated the procedure for doing stress analysis and fatigue analysis using software. The maximum stress diagram is shown in figure 4 and maximum deformation of the test wheel under the bending moment is shown in figure 5. During the static analysis of the automobile wheel, it was discovered that the maximum stress concentration occurred at the spokes and mounting hole of the rim under the required loads Fr [1]. In this paper the wheels are designed by using the solid works software then the all critical dimension, testing methods and procedures all are followed by standard books and data book.

In the wheel dimensions are the wheel rim diameter is 431.8 mm, rim width is 152.4 mm, rim offset is 45 mm, length of the wheel is 86 mm is shown in table 2. The critical dimension is rim offset dimension, because the mounting position is major point of the wheel assembly, the loads are calculated by the rim offset position. In every wheels are different loads depends on the mounting area and the offset dimension.

Figure 2 Three Dimensional Aluminum alloy wheel

Figure 3 Meshing of aluminum alloy wheel



The meshing type is 10 node tetrahedral structural solid elements. The wheel was meshed by element edge length is 4mm. The total number of nodes and element is 213219 and 112743 respectively. The meshing diagram is shown in figure 3. The meshing was performed by using ANSYS workbench R14.5. figure 4 shows the analysis diagram to finding the maximum stress of the wheel.

Figure 4 maximum stresses (von-misses stress).

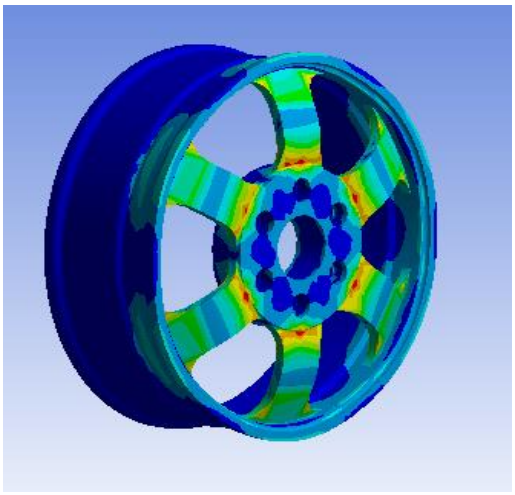
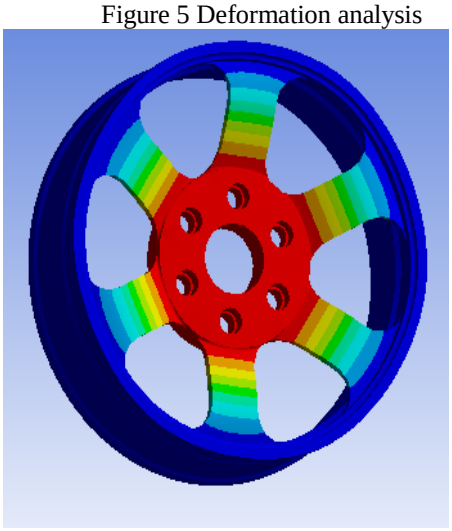


Figure 5 Deformation analysis



IV. RESULT AND CONCULTION

We found the result of Alternating stress Vs Cycles is shown in Table 2 based on the properties of the material. In Figure 5 shown the constant amplitude fully reversed load. Life and load of the Aluminum alloy wheel is shown in figure6.

Table 2: Alternating Stress Vs Cycles[4]

Cycles	Alternating Stress MPa
24076	234.12
34527	220
51601	204
1.1886e+005	175.59
1.8509e+005	160
2.9425e+005	146.32
5.1319e+005	130

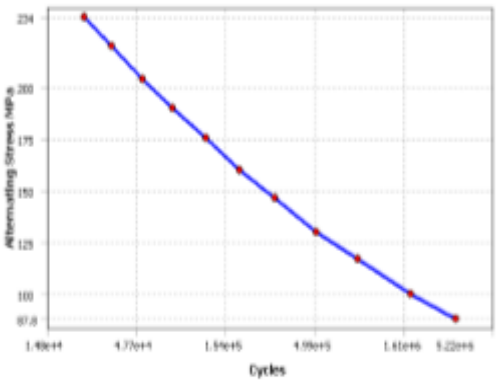


Figure 4 Alternating Stress and Cycle [4]

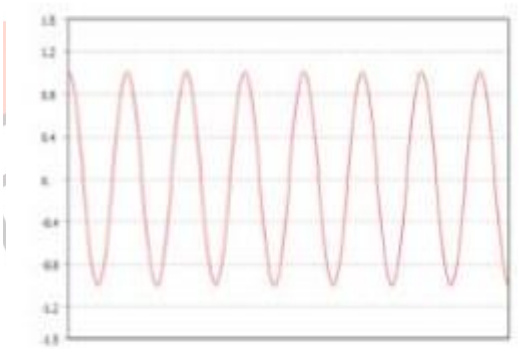


Figure 5 Constant amplitude fully reversed [4]

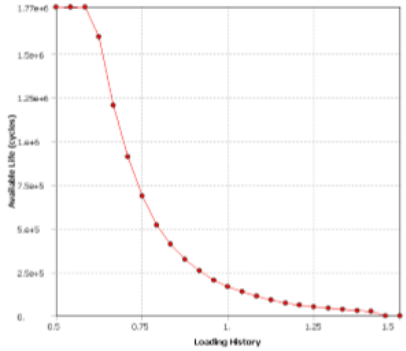


Figure 6 Life and load [4]

After completion of meshing we apply the pressure 2.9653 Mpa at rim, then we get the total deformation of wheel maximum is 0.3833mm and minimum is 0.131478 at hub portion. The alloy wheel of shear stress maximum is 49.195 and minimum is -49.241 at hub.

The equivalent stress is 165.97 and 0.058. The life of wheel maximum 1.7667e6 cycles and the minimum cycles of wheel is 1.6533e5 at a cross sectional area of the wheel. The wheel safety maximum at a hub portion because the load is maximum acting at a rim. Minimum load is acting at a hub. The damage of wheel high at a cross sectional area of wheel spokes. Finite element analysis is carried out by simulating the test conditions to analyze stress distribution and fatigue life, safety and damage of alloy wheel. The S–N curve approach for predicting the fatigue life of alloy wheels by simulating static analysis with cyclic loads is found to converge with experimental results. Safety factors for fatigue life and radial load are suggested by conducting extensive parametric studies.

REFERENCE

[1] Emmanuel M. Adigio and Ebughni O. Nangi (2014) Computer Aided Design and Simulation of Radial Fatigue Test of Automobile Rim Using ANSYS, Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, PP 68-73

[2] Torgal, S. and Mishra, S. (2012), Stress Analysis of wheel rim, International Journal of Mechanical Engineering and Research, Vol. 1, pp34-37.

[3] Liangmo Wang* - Yufa Chen - Chenzhi Wang - Qingzheng Wang (2010) Fatigue Life Analysis of Aluminum Wheels by Simulation of Rotary Fatigue Test, Strojniški vestnik - Journal of Mechanical Engineering 57(20110),31-39.

[4] N. Satyanarayana & Ch.Sambaiah (2012) Fatigue Analysis of Aluminum Alloy Wheel Under Radial Load .

[5] Muhammet cerit (2010) Numerical simulation of dynamic side impact test for an aluminum alloy wheel, scientific research and essays vol. 5(18), pp.2694-2701.

[6] Sourav Das (2014) Design and weight optimization of aluminum alloy wheel, international journal of scientific and research publications, volume 4.

[7] M. Saran Theja, M.Vamsi Krishna (2013) Structural and Fatigue Analysis of Two Wheeler Lighter Weight Alloy Wheel, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 8, Issue 2 , PP 35-45.

[8] B.anusha srikanta P.veera raju (2012) Geometrical And Material Optimization Of Alloy Wheel For Four Wheeler, International journal of research and innovation (ijri) (2012) 1401-1402, vol.1, issue.1

[9] P.Praveen, D.Gopichand (2014) Geometrical Optimization And Evaluation Of Alloy Wheel Four Wheeler, International Journal of Research and Innovation Volume No: I Issue No. : III

[10] P.Meghashyam, S. Girivardhan (2013) Design and analysis of wheel rim using catia and ANSYS volume 2, Issue 8.

[11] S Vikranth Deepak, C Naresh and Syed Altaf Hussain (2012) Modelling and analysis of alloy wheel for four wheeler vehicle ISSN 2278-0149, Vol.1,No.3

[12] Trapti Sharma, Mugdha Shrivastava, Pratesh Jayaswal (2013) Failure analysis of wheel rim. International Journal of Automobile Engineering Research & Development (IJAuERD) ISSN 2277-4785 Vol. 3, Issue 1.