International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 1, January - 2014

Analysis of Wheel Rim Using Finite Element Method

K. Venkateswara Rao and Dr. T. Dharmaraju
Department of Mechanical Engineering
Godavari Institute of Engineering and Technology, Rajamundry

Abstract

The three dimensional model of the wheel was designed using CATIA. Then the IGES format 3D model was imported into ANSYS. In the present work a detailed static analysis - displacement, maximum and minimum vonmises stresses and fatigue analysis of wheel rim under radial loads has been done. The application of finite element method for analyzing stress distribution and fatigue life of wheel rim was summarized.

1.Intorduction

The rim of a wheel is the outer circular design of the metal on which the inside edge of the tire is mounted on vehicles such as automobiles. Analysis of wheel rim made with materials like alluminium alloy, steel alloy, forged steel and magnesium alloy is done for fatigue strength.

The finite element method is a powerful tool or the numerical procedure to obtain solutions to many of the problems encountered in engineering analysis. In this method of analysis, a complex region defining a continuum is discretized into simple geometric shapes called finite elements.The domain over which the analysis is studied is divided into a number of finite elements. The material properties and the governing relationship are considered over these elements and expressed in terms of unknown values at element corner.

In the static analysis of wheel rim constraints will be applied on the circumference of the rim.

Fatigue analysis is done in MSC fatigue software, uses stress or strain results from finite element (FE) models. Usage of MSC fatigue brings fatigue analysis up front in the design-to-manufacturing process and creates an MCAE environment for integrated durability management.

2.Literature survey

Andrew D. Hartz (2002) formulated a finite element model of the classical bicycle wheel and compared published results with those revealed by ANSYS. Displacement ,strain and bending characteristics of wheel were etermined. The results indicated that ANSYS modeling can be a useful tool for analyzing simple structures such as the classical bicycle wheel. Liangmo Wang et.al(2009) proposed a new method for evaluating the fatigue life, The ABAQUS software was used to build the static load finite element model of aluminum wheels for rotary fatigue test.. The results indicated that the proposed method of integrating finite element analysis and nominal stress method was good and efficient to predict the fatigue life of aluminum wheels. Alexandru Valentin Raduelescu et.al (2012) analyzed the car rim with the finite element method using the 40⁰ loading test The finite element analysis was conducted in two stages- the analysis of the state of stresses in the central area of the rim for the intial and optimized versions. The static stresses are studied in order to find the zones with higher stress concentration and to suggest the better design solution. The results have been compared to those obtained using an experimental stand. Sunil N. Yadav and N. S. Hanamapure (2013) analysed the effect of camber angle on stress distribution and fatigue life of wheel rim of passenger car under radial load condition which arises due to off road field area and road unevenness. Finite element analysis (FEA) is carried out by simulating the test conditions to analyze stress distribution and fatigue life of the steel wheel rim of passenger carP. Meghashyam-et.al(2013) proposed that the modelling of the wheel rim is made byusing CATIA. Later this CATIA model is imported to ANSYS for analysis work. ANSYS software is the latest used for simulatingthe different forces, pressure acting on the component and also for calculating and viewing the results ANSYS static analysis work is carried out by considered two different materials namely aluminium and forged steel and their relative performances have been observed respectively. Inaddition to this rim is subjected to vibration analysis (modal analysis), a part of dynamic analysis is carried out its performance isobserved. In this paper by observing the results of both static and modal analysis obtained forged steel is suggested as best material

3. Analysis of wheel rim

Analysis detailed static analysis -displacement, maximum and minimum vonmises stresses and fatigue analysis of wheel rim under radial loads,. We have consider the stell,aluminum alloy and magnesium,forged steel for analysis the analysis following steps

Material properties
Model of the whell rim
Importing the model
Boundary conditions and Loading
Application of load

3.1 Material properties

- **Steel alloy:**
  Young's modulus (E) =$2.34*10^5$ N/mm^2
  Yield stress=240 N/mm^2
  Density =7800kg/m^3
- **Aluminum alloy:**
  Young's modulus (E) =72000 N/mm^2
  Yield stress=160 N/mm^2
  Density =2800kg/m^3
- **Magnesium alloy:**
  Young's modulus (E) =45000N/mm^2
  Yield stress=130 N/mm^2
  Density =1800kg/m^3
- **Forged steel:**
  Young's modulus (E) =210000N/mm^2
  Yield stress=220 N/mm^2
  Density =7600kg/m^3

3.2 Model of wheel rim

SOLID45 is used for the 3-D modeling of solid structures. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

Table No.1 wheel rim dimensions

Outer diameter	450 mm
Hub hole diameter	150 mm
Bolt hole diameter	20 mm
Rim width	254 mm

3D Model of the wheel rim

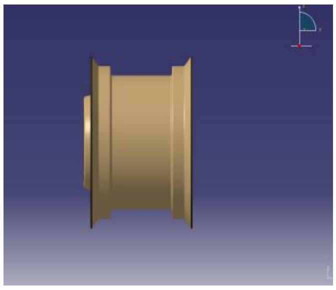


Fig 1 Model of the wheel rim

3.3 Importing the model

- The imported model is meshed by using Hyper mesh. the meshed model is as follows

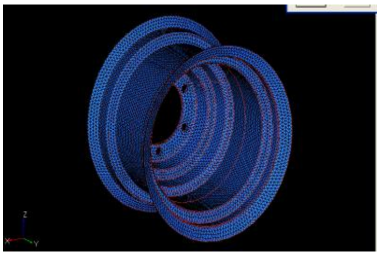


Fig 2: Meshing finished model

- The meshed model (.hm file format) of wheel rim is imported from Hyper Mesh Software to ANSYS Software.by (file>Import>IGES)
- Later this meshed model is defined with different materials namely steel, Aluminum and Magnesium alloy and forged steel subjected to static analysis

Centrifugal force, $F=mr\omega^2$ N
$\omega =2*(22/7)*N/60$ rad/s
Mass=24 kg
Speed=600 rpm
$\omega =62.8$ rps

By substituting, we get centrifugal force=21.3kN which acts at each node of the circumference of the rim.

3.3 Boundary conditions and Loading:

To get compressive and tensile stress, a load of 21.3kN is applied on the bolt holes of the wheel rim.

- **Displacements**
  a. Translation in x, y, z directions is zero.
  b. Rotation in x, y, z direction is zero.
- **Angular velocity** in x direction is zero,
  y direction is 62.8 rps
  z direction is zero.
- These conditions are applied on the six holes provided on the rim.

In the same way, Centrifugal force is also applied in the loading condition on the holes.

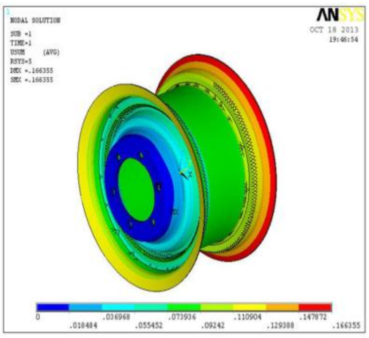
3.4 Application of load

After this meshed model is constrained at holes by all DOF where the bolts has to be placed. After constraining the meshed model, the model is subjected to a centrifugal force of 21.3kN

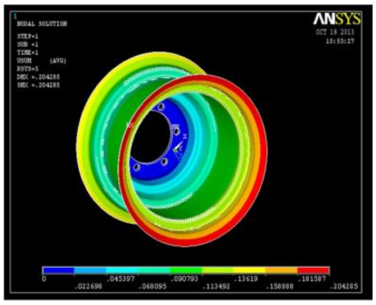
Later the results were obtained in the SOLVER module.Then analysis type is changed from static command to modal command and solution is done. Next solution results such as stress, displacement, von mises, ultimate strength etc. were calucated

4. RESULTS &DISCUSSION

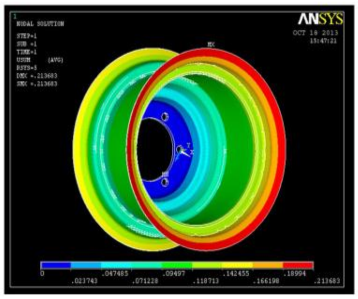
4.1 Displacement of Alloy wheel



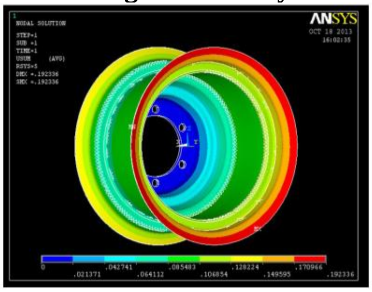
Steel alloy



Aluminium alloy

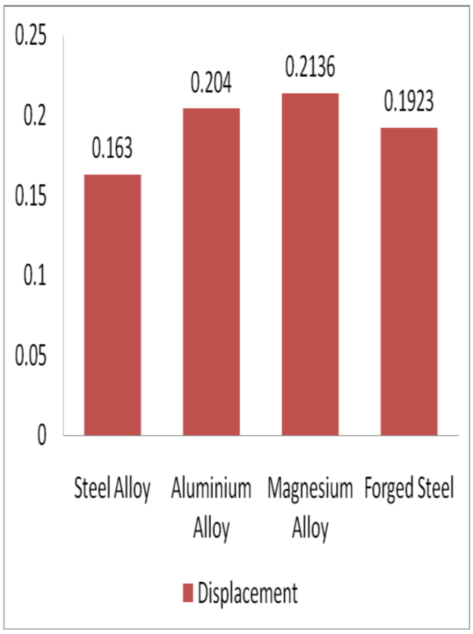


Magnesium alloy



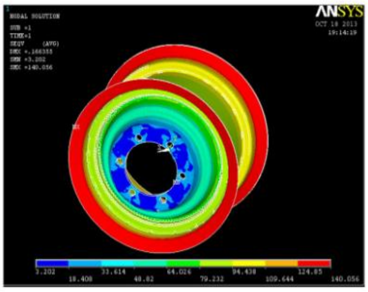
Forged steel

4.1.1 Displacement graph for alloy wheels

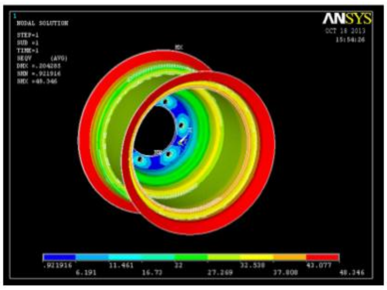


Displacement graph for alloy wheels

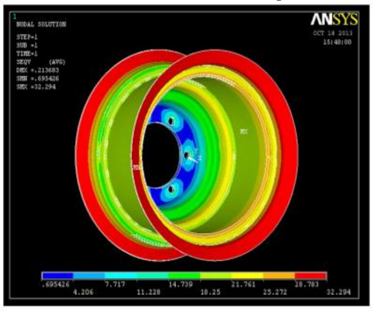
4.2 Stress plots for alloy wheels



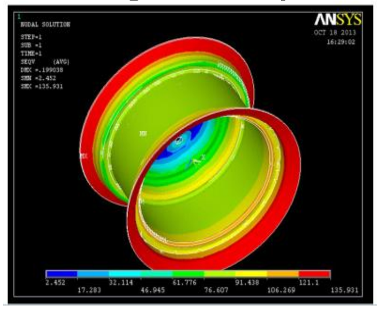
Steel alloy



Aluminium alloy

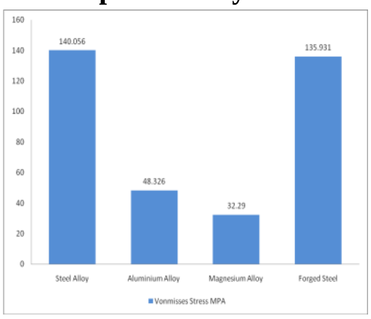


Magnesium alloy

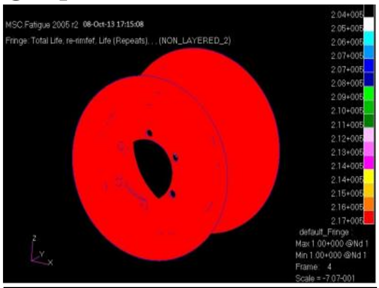


Forged steel

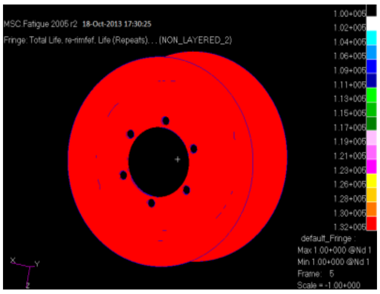
4.2.1 Stress Graph for Alloy wheels



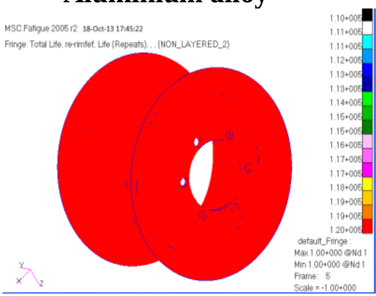
4.3 Fatigue plots and S-N curves



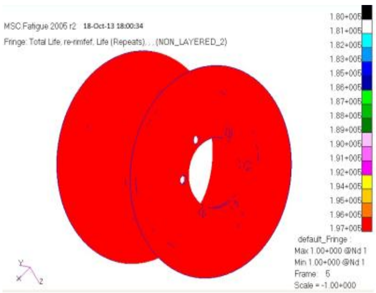
Steel alloy



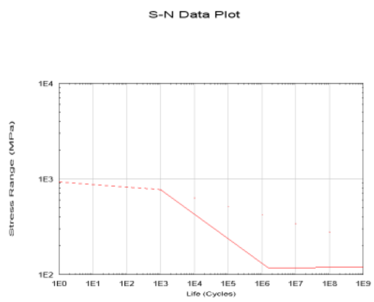
Aluminium alloy



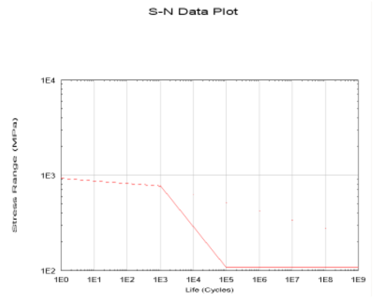
Magnesium alloy



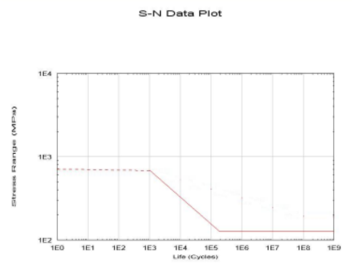
Forged steel



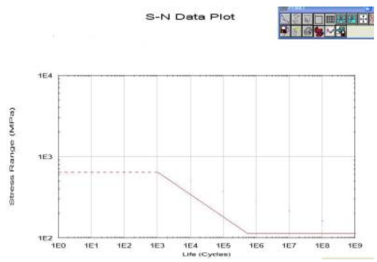
Steel alloy



Aluminium alloy

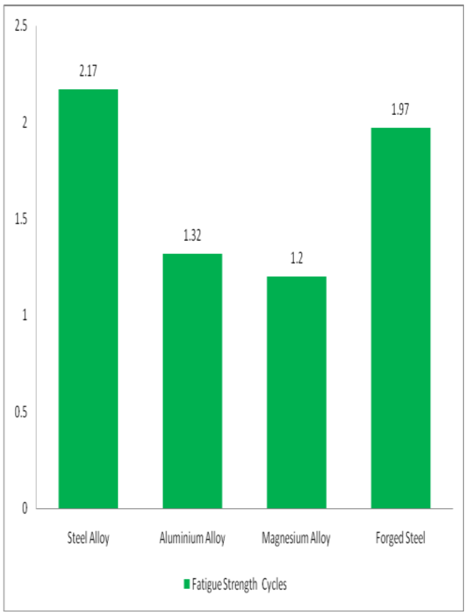


Magnesium alloy



Forged Steel

4.3.1 Fatigue graph for alloy wheels



Fatigue graph for alloy wheels

Material	Displacement (mm)	Vonmisses stress (Mpa)	Fatigue strength (cycles)
Steel alloy	0.1663	140.056	$2.17*10^5$
Aluminium alloy	0.204	48.326	$1.32*10^5$
Magnesium alloy	0.2136	32.29	$1.2*10^5$
Forged steel	0.1923	135.931	$1.97*10^5$

Out of the different materials used steel alloy was found to have greater vonmisses stress of 140.056Mpawhile magnesium alloy has the least vonmisses stress of 32.29 Mpa. Steel alloy has maximum number of cycles to failureN_f)=$2.17*10^5$Cycle While magnesium alloy has the least Number of cycles to failure (N_f) =$1.2*10^5$Cycles.

5.Conclusion

In steel alloy the number of cycles to failure is greater than Aluminium alloy, Magnesium alloy and Forged steel. Hence Steel alloy is more feasible to be used in wheel rim than other materials. Further optimization of material thickness to reduce the material consumption can be done and we can improve life of component by using advanced fatigue strain life approach.

6.References

[1] Andrew D. Hartz " Finite Element Analysis of the Classic Bicycle Wheel " , Raytheon Engineering and Production Support Indianapolis, Indiana ,July 18, 2002

[2] Liangmo Wang* - Yufa Chen - Chenzhi Wang - Qingzheng Wang et.al " the fatigue analysis of aluminium wheel rim" Strojniški vestnik - Journal of Mechanical Engineering 57(2011)1, 31-39-jme.2009.046

[3] Alexandru Valentin Raduelescu –Sorin Cananau -Irina Radulescu et.al "Mechanical testing methods concerning the stress analysis for a vehicle wheel rim "olume2, 33-39

[4] Sunil N. Yadav, et.al "Modeling and Analysis of Camber Angle on Fatigue Life of Wheel Rim of Passenger Car by Using Radial Fatigue Testing ",International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 2, Issue 5, September 2013

[5] P. Meghashyam- et.al " Design andAnalysis of Wheel Rim using CATIA & ANSYS " international journal of application or innovation in engineering management,Volume 2, Issue 8, August 2013