International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Email:editorijoer@gmail.com http://www.ijoer.in ISSN: 2321-7758

Vol.4., Issue.5., 2016
(Sept.-Oct.)

RESEARCH ARTICLE

ISSN: 2321-7758

MODELING AND FATIGUE ANALYSIS OF AUTOMOTIVE WHEEL RIM

MASTAN VALI SHAIK [1], D.L. KOMALA RAO[2]

[1]PG Student (M.Tech.) (CAD/CAM), [2]Assistant Professor
Department of Mechanical Engineering,
Chirala Engineering College, Chirala – 523155

MASTAN VALI SHAIK

ABSTRACT

A wheel rim is a highly stressed component in an automobile that is subjected to bending and torsional loads. Because of the long life and high stresses, as well as the need for weight reduction, material and manufacturing process selection is important in rim design. . Using the method proposed in this paper, the wheel life cycle was improved to over 1.0×10^5 and satisfied the design requirement. The finite element method is a powerful tool for the numerical procedure to obtain solutions to many of the problems encountered in engineering analysis. Structural, thermal and heat transfer, fluid dynamics, fatigue related problems, electric and magnetic fields, the concepts of finite element methods can be utilized to solve these engineering problems. For predicting the wheel fatigue life, the nominal stress method was integrated into the CAD / CAE technology to simulate the rotary fatigue test. In addition, an actual prototype of the test was done to verify the analysis. Various properties of alloys are investigated. Based on the results obtained steel alloy have more life and durability compared to aluminum.

Key words:- modeling, design, wheel rim ,manufacturing, automobile rim

1. INTRODUCTION

Several thousand years ago was the start of the history of wheel when the human race began to use the log to transport heavy objects. The original of the wheel were the round slices of a log and it was gradually re-inforced and used in this form for centuries on both carts and wagons.

This solid disc changed to a design having several spokes radially arranged to support the outer part of the wheel keeping it equidistant from the wheel centre.

A wooden wheel which used hard wood stakes as spokes was very popular as a wheel for many vehicles up to about 1920.

Truck wheels need to be durable and able to carry around weight. Lighter wheels can improve handling by reducing unsprung mass , allowing suspension to follow the terrain more closely and thus improve grip, however not all alloy wheels are lighter than their steel equivalents. Reduction in overall vehicle mass can also help to reduce fuel consumption .

Better heat conduction can help dissipate heat from the brakes , which improves braking performance in more demanding driving conditions and reduces the chance of brake failure due to overheating.

1.1. wheel rim description

A standard automotive steel wheel rim is made from a rectangular sheet metal. The metal plate is bent to produce a cylindrical sleeve with the two free edges of the sleeve welded together. At least one cylindrical flow spinning operation is carried out to obtain a given thickness profile of the sleeve — in particular comprising in the zone intended to constitute the outer seat an angle of inclination relative to the axial direction. The sleeve is then shaped to obtain the rims on each side with a radially inner cylindrical wall in the zone of the outer seat and with a radially outer frusto-conical wall inclined at an angle corresponding to the standard inclination of the rim seats. The rim is then calibrated. Wheel rim is the part of automotive where it heavily undergoes both static loads as well as fatigue loads as wheel rim travels different road profile. It develops heavy stresses in rim so we have to find the critical stress point and we have to find for how many number cycle that the wheel rim is going to fail.

1.2 Type of Wheel/Rim (Material)

Steel and light alloy are the main materials used in a wheel however some composite materials including glass-fiber are being used for special wheels.

(1) Wire Spoke Wheel: Wire spoke wheel is a structural where the outside edge part of the wheel (rim) and the axle mounting part are connected by numerous wires called spokes. Today's vehicles with their high horsepower have made this type of wheel construction obsolete. This type of wheel is still used on classic vehicles. Light alloy wheels have developed in recent years, a design to emphasize this spoke effect to satisfy users fashion requirements.

(2) Steel Disc Wheel : This is a rim which processes the steel-made rim and the wheel into one by welding, and it is used mainly for passenger vehicle especially original equipment tires.

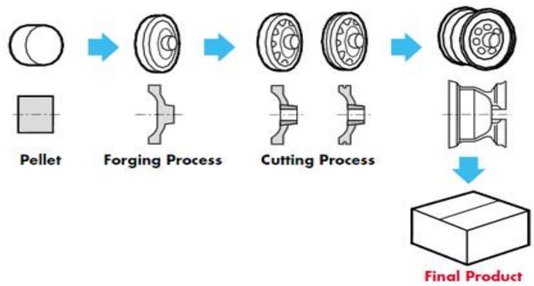
(3) Light Alloy Wheel: These wheels based on the use of light metals such as aluminium and magnesium have become popular in the market. These wheels rapidly become popular for the original equipment vehicle in Europe in 1960's and for the replacement tire in United States in 1970's.

1.3 Manufacturing Method of Wheel/Rim

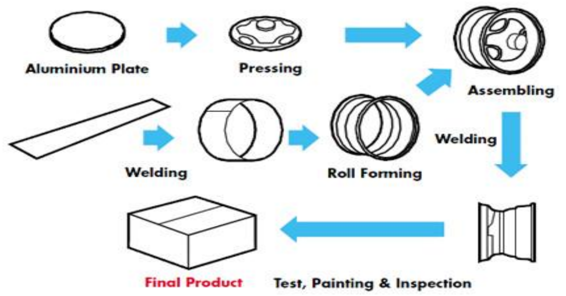
The steel disk wheel and the light alloy wheel are the most typical installation. The method of manufacturing the light alloy wheel, which has become popular in recent years, is explained here. The manufacturing method for the light alloy wheel is classified into two. They are cast metal or the forged manufacturing methods.

The aluminum alloy wheel is manufactured both ways, and the casting manufacturing method is used as for the magnesium alloy wheel. There are the following three methods of manufacturing the aluminum alloy wheel.

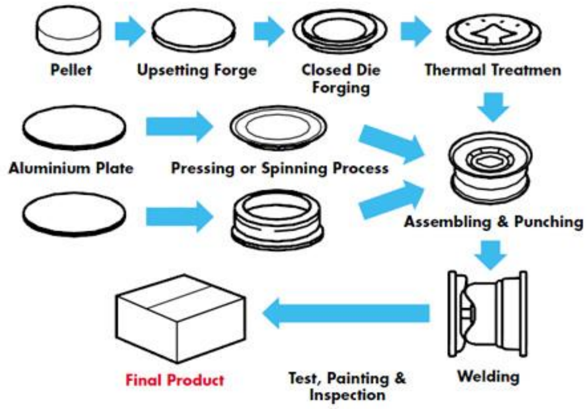
(1) Forging Method (for One Piece Rim)



(2) Forging Method (for Two Pieces Rim)



(3) Forging Method (for Three Pieces Rim)



International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Email:editorijoer@gmail.com http://www.ijoer.in ISSN: 2321-7758
Vol.4., Issue.5., 2016
(Sept.-Oct.)

1. 4 Test of Wheel

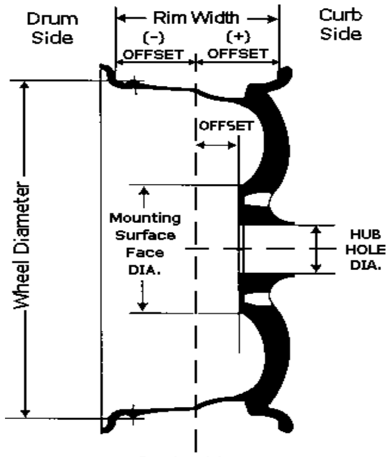
Wheels are part of a vehicle and as such subjected to a high load. The durability of the wheel is important for the safe operation of the vehicle. Therefore, it is necessary to examine a wheel for both strength and fatigue resistance

1.5 Maintaining rims

Very necessary but often overlooked, it is vitally important to inspect your motorcycle rims and clean them on a regular basis to help prevent spoke failure or corrosion weak points. You can definitely suffer flat tires if a few spokes fail on your motorcycle rims. This can happen under ordinary everyday conditions. The broken spoke pushes into the wheel and punctures the tube. So always keep your wheels clean and check them for signs of corrosion or other damage. It may only take one bad spoke to ruin your ride. The aluminum motorcycle rims are usually coated. Some chemicals used for bike maintenance of other systems (like brake fluid) can damage that coating. Once the bare aluminum on the motorcycle rim is exposed to air it can begin to corrode. Wheels can come under a lot of stress and even small areas of corrosion can become a point of failure.

2.MODELING OF WHEEL RIM

2.1 Wheel rim nomenclature



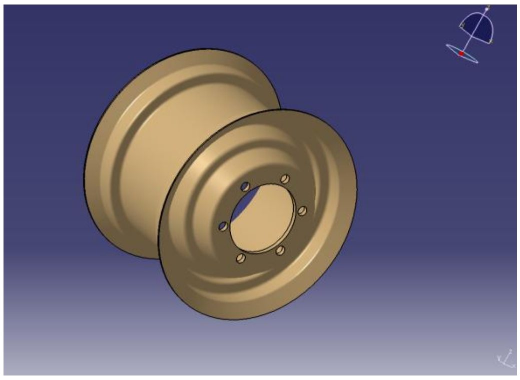
2.2 2D model of the wheel rim

Initially the 2D drawing of wheel rim is done by using CATIA according to dimensions specified in the Table 2.2.1

Table No.2.2.1

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

2.3 3D Model of the wheel rim



Isometric view

2.4 Generating the mesh using HYPERMESH Software

Hyper Mesh user-interface is easy to learn and supports many CAD geometry and finite element model files - increasing interoperability and efficiency. Advanced functionality within Hyper Mesh allows users to efficiently mesh high fidelity models. This functionality includes user defined quality criteria and controls, morphing technology to update existing meshes to new design proposals, and automatic mid-surface generation for complex designs with of varying wall thicknesses. Automated tetra-meshing and hexa-meshing minimizes meshing time while batch meshing enables large scale meshing of parts with no model clean up and minimal user input.

Benefits

- Reduce time and engineering analysis cost through high-performance finite element modeling and post-processing.
- The industry's broadest and most comprehensive CAD and CAE solver direct interface support.
- Reduce overhead costs of maintaining multiple pre- and post-processing tools, minimize "new user" learning curves, and increase staff efficiency with a powerful,

intuitive, consistent finite element analysis environment.

- Reduce redundancy and model development costs through the direct use of CAD geometry and legacy finite element models.
- Simplify the modeling process for complex geometry through high-speed, high-quality auto meshing, hexa-meshing and tetra meshing.
- Dramatically increase end-user modeling efficiency by eliminating the need to perform manual geometry clean up and meshing with Batch Meshed technology.

The process of generating a mesh of nodes and elements consists of three general steps.

1. Set the element attributes.
2. Set mesh controls (optional).
3. Meshing model.

The wheel rim solid model (.iges file format) is imported to HYPERMESH and the model is meshed with solid tetra element and saved in .hm file format thus finite element model is created

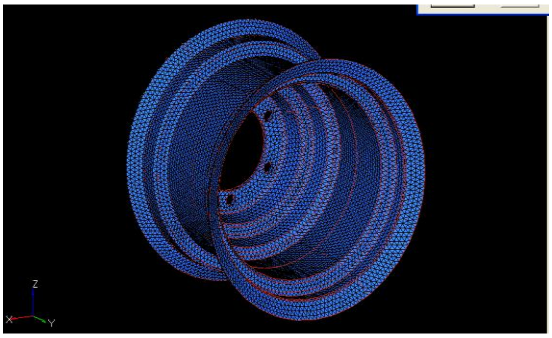


Fig2.4. Meshing finished model

3. FINITE ELEMENT ANALYSIS

3.1 Steps involved in FEM1

The different steps involved in the Finite element method are as follows:

Step1: Discretization of continuum: The first step in any FEM is to divide the given continuum in to smaller region called element. The type of elements has to be taken depending on type of analysis carried out like one dimensional, two dimensional, and three dimensional.

Step 2: Selection of displacement model: For the continuum discretized in to number of element, displacement variation over each of these element is unknown .Hence a displacement function is assumed for each of the element ,this function is called displacement model.

Step 3: Derivation of elemental stiffness matrix: The equilibrium equation for an element is determined by using the principal of minimum potential energy.

Step 4: Assembly of the element stiffness matrix: This step involves determining of global stiffness matrix. This is done by using the compatibility conditions at the nodes. The displacement of a particular node must be the same for every element connected to it. The externally applied loads must also be balanced by the forces on the elements at these nodes.

Step 5: Apply the boundary conditions : To obtain a unique solution of the problem, some displacement constraints (i.e. boundary conditions) and loading conditions must be prescribed at some of the nodes. This may be of the following forms

1) Elimination method
2) Penalty method
3) Multi constraint method

These boundary conditions are incorporated into the system of linear algebraic equations, which can then be solved to obtain a unique solution for the displacements at each node.

Step 6: To find unknown displacement, strain and stress : After solving the global equations, displacements at all the nodal points are determined. From the displacement values, the element strains can be obtained from the stress-strains relations. In FE formulation only the displacements are the independent variables, that is, forces, strains

3.2 Static Analysis

Static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).

Static analysis involves both linear and nonlinear analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

The FE analysis used for the major part of this work is static analysis which involves both linear and nonlinear structural analysis. Hence more prominence is imparted on Linear and nonlinear analysis in further sections.

(1) Linear Static Analysis: In linear analysis, the behavior of the structure is assumed to be completely reversible; that is, the body returns to its original undeformed state upon the removal of applied loads and solutions for various load cases can be superimposed.

The assumptions in linear analysis are:

1) Displacements are assumed to be linearly dependent on the applied load.
2) A linear relationship is assumed between stress and strain.
3) Changes in geometry due to displacement are assumed to be small and hence ignored.
4) Loading sequence is not important and the final state is not affected by the load history. The load is applied in one go with no iterations.

(2) Non Linear static analysis

In many engineering problems, the behavior of the structure may depend on the load history or may result in large deformations beyond the elastic limit. The assumptions/ features in nonlinear analysis are:

1) The load-displacement relationships are usually nonlinear.
2) In problems involving material non-linearity, the stress-strain relationship is a nonlinear function of stress, strain, and/or time.
3) Displacements may not be small, hence an updated reference state may be needed.
4) The behavior of the structure may depend on the load history, hence the load may have to be applied in small increments with iterations performed to ensure that equilibrium is satisfied at every load increment.

From the above assumptions, the finite element equilibrium equation for static analysis is:

$$[K]\,\{U\} = [F]$$

Where [*K*] is the linear elastic stiffness. When the above assumptions are not valid, one performs nonlinear analysis.

3.3 Description of element used in static analysis in ansys

SOLID45 Element Description

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.

The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. A reduced integration option with hourglass control is available. Fig(4) SOLID45 Geometry

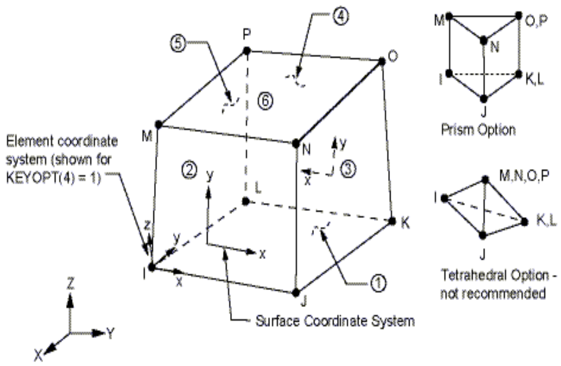


Fig3.3 SOLID45 Geometry

SOLID45 Assumptions and Restrictions

- Zero volume elements are not allowed.
- Elements may be numbered either as shown in Figure 45.1: "SOLID45 Geometry" or may have the planes IJKL and MNOP interchanged.
- The element may not be twisted such that the element has two separate volumes. This occurs most frequently when the elements are not numbered properly.
- All elements must have eight nodes.
- A prism-shaped element may be formed by defining duplicate K and L and duplicate O

and P node numbers (see Triangle, Prism and Tetrahedral Elements).

- A tetrahedron shape is also available. The extra shapes are automatically deleted for tetrahedron elements.

The procedure for a model analysis consists of four main steps:

1. Build the model.
2. Apply loads and obtain the solution.
3. Expand the modes.
4. Review the results.

Importing the Model: The finite element meshed model (.hm file format) of wheel rim is imported from Hyper Mesh Software to ANSYS Software.

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

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

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.

4. FATIGUE ANALYSIS

4.1 Fatigue Mechanisms: The fatigue failure mechanisms are divided into two classes: the primary mechanisms and the secondary mechanisms, according to the following definition:

- **Primary mechanisms:** mechanisms that are able by themselves to initiate and propagate fatigue cracks;
- **Secondary mechanisms:** mechanisms that are not able by themselves to promote fatigue fracture but may either initiate cracks or help on crack propagation of pre-existing cracks.
- A definition for the different fatigue mechanisms, either primary or secondary mechanisms, will be subsequently given. Some schemes of the mechanisms are shown on the damaged components section.

Primary mechanisms

- **Mechanical fatigue -** Mechanical fatigue is the widest definition and is traditionally related to components where external loads are applied for example on the connections/supports. In this definition cyclic stresses flow through the component and concentrate in critical points of the component due to loads/restraints that are applied in other points. If mechanical fatigue occurs at high temperature another mechanism, creep, is often active.
- **Thermal fatigue -** Thermal fatigue exists under two different situations: the first is in a singular component due to different temperatures (cyclic) in different areas of the same component; the second situation is, for a component with two dissimilar materials, for a certain temperature (cyclic) in both materials at the same time. In the first situation stresses arise due to the difference in temperature; in the second situation stresses arise through different dilatation coefficients of the same component (with at least two different materials). Due to high temperatures involved in the process and depending on the thermal cycle shape creep may also be active.

Secondary mechanisms

- **Wear-fatigue -** Wear fatigue exists when two bodies are not attached one to another but there is contact and a relative displacement between both components. There are the normal contact forces plus the tangential forces due to the sliding movement between both bodies.

- **Fretting fatigue -** Fretting fatigue is similar to wear fatigue because there is wear between the two bodies due to a relative displacement. The main difference is that the two bodies are commonly connected or attached one to the other for example with screws, and the relative displacement between both components is very small (traditionally between 1 to 100 ì m)
- **Abrasion fatigue -** Abrasion fatigue exists when two solid bodies are not in direct contact one to the other but a third body (for example dust) promotes the contact and load transmission between the initial two bodies. The third body (for example dust) may be involved in oil or water. Initially they cause pitting or spalling like on contact fatigue but in cases where a pre-existing crack exists they may promote crack propagation.
- **Corrosion fatigue -** corrosion fatigue exists when structural metals operate in deleterious environments. This detrimental environment accelerates fatigue crack growth. Even materials immune to SCC – Stress Corrosion Cracking are susceptible to CC – Corrosion Cracking (or corrosion fatigue cracking).

4.2 MSC. fatigue software: MSC Fatigue is a FE-based durability and damage tolerance solver that enables users with minimal knowledge of fatigue to perform comprehensive durability analysis. MSC Fatigue enables durability engineers to quickly and accurately predict how long products will last under any combination of time-dependent or frequency-dependent loading conditions. Benefits include reduced prototype testing, fewer product recalls, lower warranty costs, and increased confidence that your product designs will pass required test schedules Fatigue analysis itself usually refers to one of two methodologies: either the stress life or S-N method, commonly referred to as total life since it makes no distinction between initiating or growing a crack, or the local strain or strain-life (ε-N) method, commonly referred to as the crack initiation method which concerns itself only with the initiation of a crack.

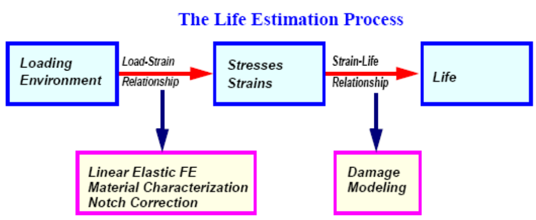
Benefits: Analysis using MSC.Fatigue significantly reduces costs associated with prototyping and testing by simulating fatigue life early in the design phase. Early simulation shortens time-to-market, improves product reliability, customer confidence and reduces costly recalls or other undesirable consequences of premature product failure.

Usage of MSC.Fatigue brings fatigue analysis up front in the design-to-manufacturing process and creates an MCAE environment for integrated durability management

4.3 Life Estimation Process

The life estimation process really centers around two major relationships.

1. The first relation is that of the loading environment to the stresses and strains in the component or model. This load-strain or load-stress relation is determined using finite element modeling and running linear elastic FE analysis. It is dependent on the characterization of the material properties and in some instances requires that a notch correction procedure take place. For the purposes of this discussion a notch correction is simply a way to compensate for plasticity from a linear FE analysis.
2. The second relation is that of the stresses or stains to the life of the component or model. This is accomplished by using damage modeling. Each fatigue life method has its own techniques to determine and



sum damage which shall be explained as you progress through the example problems.

The fatigue analysis is carried out in MSC fatigue tool .The von-misses stresses from ANSYS(.rst file format) is imported to the MSCfatigue and find the number of cycles to failures of crankshaft for forged steel and sintered aluminium.

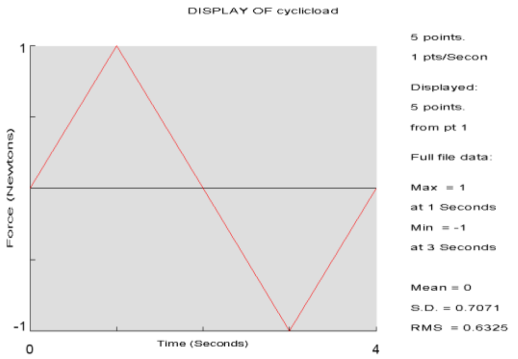
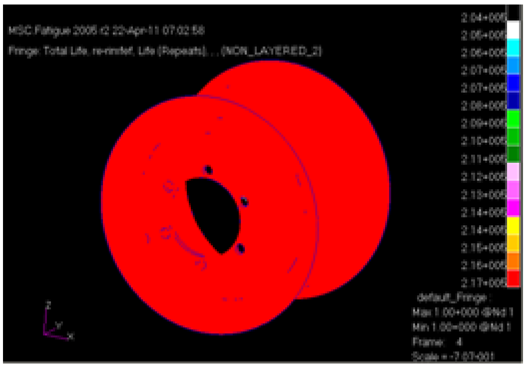


Fig 4.3 Type of Fatigue load inputting

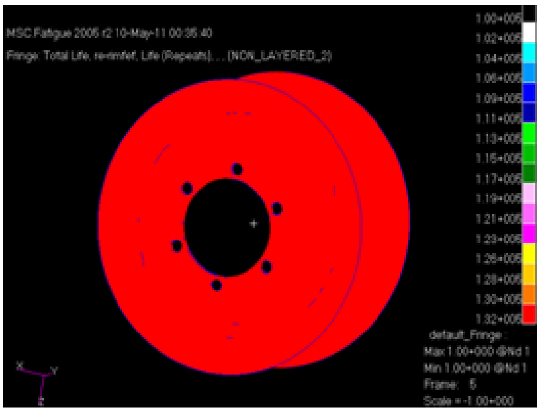
4.4 Fatigue plots and S-N curves

Steel alloy :



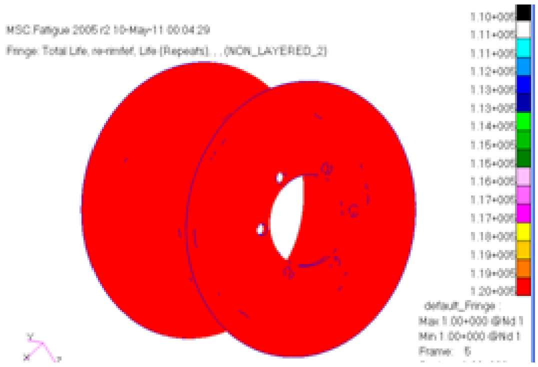
Fatigue strength=2.17*10^5 cycles

Aluminium alloy



Fatigue strength=1.32*10^5 cycles

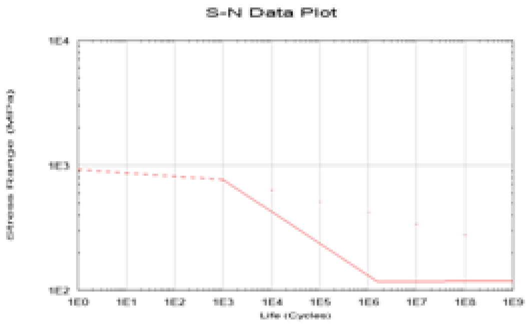
Magnesium alloy



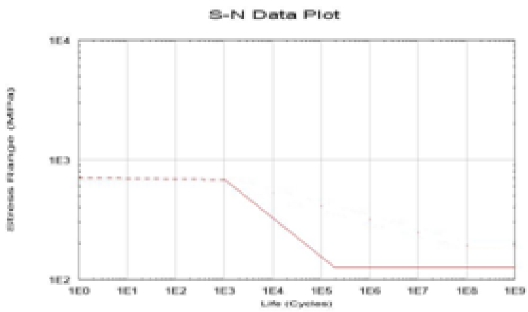
Fatigue strength=1.2*10^5 cycles

Forged steel

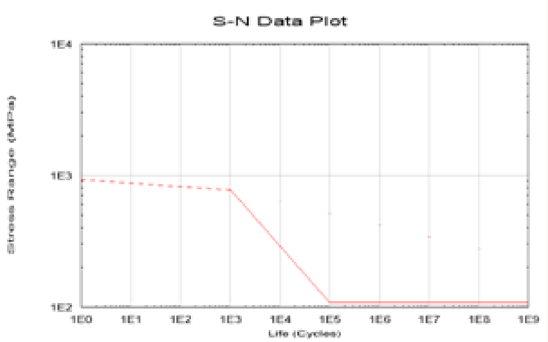
Fatigue strength=1.97*10^5 cycles



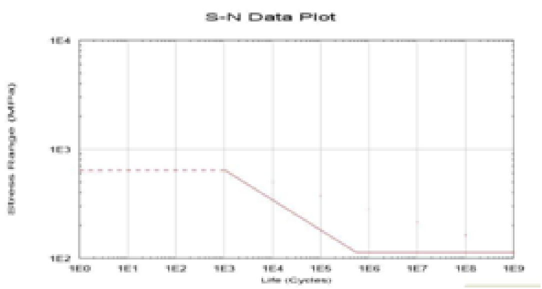
Steel alloy



Aluminium alloy



Magnesium alloy



Forged steel

5. RESULTS AND DISCUSSIONS

5.1 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

5.2 Results obtained from softwares:

Steel alloy:

Von misses stress (σ_v) =140.056 N/mm^2

Number of cycles to failure (N_f)=$2.17*10^5$Cycles

Aluminum alloy:

Von misses stress (σ_v) =48.326 N/mm^2

Number of cycles to failure (N_f) =$1.32*10^5$Cycles

Magnesium alloy:

Von misses stress (σ_v) =32.204 N/mm^2

Number of cycles to failure (N_f) =$1.2*10^5$Cycles

Forged steel:

Von misses stress (σ_v) =135.931 N/mm^2

Number of cycles to failure (N_f) =$1.97*10^5$Cycles

Table 5.2.1

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	$1.97*10^5$

6. CONCLUSION

1) The von misses stresses developed in steel alloy during static analysis is 140.056 N/mm^2 at load 21.3KN the stress is below yield stress of material for these stress range we have to find at what number of cycles the component is yielding or crack is going to initiates

2) During fatigue analysis of steel alloy the crack is initiating at N_f=$2.17*10^5$Cycles.

3) The von misses stresses developed in aluminum alloy during static analysis is 48.326 N/mm^2 at load 21.3KN the stress is below yield stress of material for these stress range we have to find at what number of cycles the component is yielding or crack is going to initiates

4) During fatigue analysis of aluminum alloy the crack is initiating at N_f=$1.32*10^5$Cycles.

5) The von misses stresses developed in Magnesium alloy during static analysis is 32.294 N/mm^2 at load 21.3KN the stress is below yield stress of material for these stress range we have to find at what number of cycles the component is yielding or crack is going to initiates.

6) During fatigue analysis of Magnesium alloy the crack is initiating at N_f=$1.2*10^5$Cycles.

7) The von misses stresses developed in Forged steel during static analysis is 135.931 N/mm^2 at load 21.3KN the stress is below yield stress of material for these stress range we have to find at what number of cycles the component is yielding or crack is going to initiates
8) During fatigue analysis of Forged steel the crack is initiating at N_f=1.97*10^{5}Cycles.
9) From results we can make out, in steel alloy the Number of cycles to failure (N_f)= 2.17*10^{5}Cycles is greater than Aluminium, Magnesium, Forged steel. Hence Steel alloy is more feasible to use than aluminum.

Hence steel alloy have more life and durability compared to aluminum

SCOPE FOR FUTURE WORK:

1) Further we can do optimization of material thickness to reduce the material consumption.
2) Further we can improve life of component by using advanced fatigue strain life approach.

7. REFERENCES

[1] K. Mahadevan and Balaveera Reddy, “Design Data Hand Book”.
[2] “Finite Element Analysis”, Chandra Pautla.
[3] “Strength Of Materials”, Ramambrutham.
[4] “Ansys User Manual”,
[5] “Metal Fatigue” , Ralfh Stefunson, Ali Fatemi & A.O. Cuph.
[6] “MSC Fatigue User Manual”,
[7] Metal_fatigue_in_engineering by Stefan.
[8] Fatigue Life Analysis of Aluminum Wheels by Simulation of Rotary Fatigue Test Liangmo Wang* - Yufa Chen - Chenzhi Wang - Qingzheng Wang School of Mechanical Engineering, Nanjing University of Science & Technology, China
[9] C.Bosi,G.L.Garagnani, Fatigue properties of a cast aluminium alloy for rims of car wheels.