JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

OPTIMIZATION OF FOUR WHEELER WHEEL RIM FOR AL, MG AND STEEL ALLOY

[1] Khaire Punam Arjun, [2] Prof.Ghodake A.P, [3] Prof. Pawar H.U
[1]Student of Master of Mechanical Engineering, [2]Assistant Professor, [3]Head of Department
Department Of Mechanical Engineering
SND COE & RC, Yeola, Maharashtra, India

Abstract: Steel Disked wheels of car have to pass dynamic cornering fatigue test, the dynamic radial fatigue test, and the impact test .In the actual product development, the dynamic cornering fatigue test is used to detect the strength and fatigue life of the wheel. Therefore, a reliable design and test procedure is required to guarantee the service strength under operational conditions and full functioning of the wheel. Loads generated during the assembly may cause significant levels of stress in components. Under test conditions, these high levels of stress alter the mean stress level which in turn, alters the fatigue life and critical stress area of the components as well.

***Index Terms* - dynamic, impact test, fatigue, assembly**

I. INTRODUCTION

The wheel with tires takes full load, provides the cushioning effect to vehicle by absorbing vibration of the road surface unevenness and also assist in steering control. The alloy wheel has better aesthetic looks and easy of manufacturing than disc and wire wheel. The main requirements of an automobile wheel are:

- It should be as light as possible so that unstrung weight is least.
- It should be strong enough to perform the above functions.
- It should be balanced statically as well as dynamically.
- It should be possible to remove or mount the wheel easily.

It material should not deteriorate with weathering and age .In case, the material is suspected to corrosion, it must be given suitable protective treatment. Automotive wheels have much complicated profile shape and geometry. It must satisfy manifold design criteria for good style, less weight, good manufacturability and better performance. Along with fascinating wheel style, wheel design also needs to complete a lot of engineering objectives such as performance and durability requirements. Also to ensure driving comfort and road handling properties, the wheel must be as light as possible. Nowadays, reduction in wheel weight is a main point of concern in wheel industry. From wheel manufacturing point of view, wheel weight reduction is material cost reduction. So to reduce the manufacturing cost, wheel weight must be minimized, but without affecting mechanical performance. Wheel design and development is very time consuming process as it requires lot of testing and design iterations before going to production. To shorten development time and to reduce the number of iterations of test is an important issue. To achieve these objectives, computer aided engineering (CAE) is a useful tool and has been recently carried out to perform a wheel design. In this thesis Finite Element Analysis of Weld Test, Radial and Cornering Fatigue test using CAE software has been studied.

1.1 Problem Definition

1. Presently 24 spots are used to join disc and rim on spot weld machine having 3 stations. One cycle of 3 spots requires 6-7 seconds. There are 8 cycles of welding and 7 time indexing. One indexing requires 4-5 seconds. Loading and Unloading time is near about 3 seconds. So for production of one rim will require near about 94 seconds. Other operations need maximum 35 seconds. Now there is a bottle neck at this stage of production, in a shift with other operations 900 to 960 rims are produced but at welding machine 300 to 306 rims are produced. To cope up with this problem 2 Welding machines are installed still the maximum production of rims is 612 rims per shift. So this thesis aims at replacing the steel wheel by alloy wheels.

2. Automotive wheel, as a critical component in the vehicle, has to meet the strict requirements of driving safety. Traditionally, the new designed wheel is tested in the laboratory for its life through an accelerated fatigue test before the actual production starts. However, a physical prototype test time lasts at least 7 days and an average design period is 6 months or more depending on the requirement, so the time to test and inspect wheel during development is very consuming. At the same time, because steel wheel is designed for variation in style and has very complex shape, it is difficult to assess fatigue life by using analytical methods. In the last decade, many scholars and wheel manufacturers have been taking increasing attention to numerical analysis of wheel fatigue life.

3. Durability assessment of mechanical components early in the design phase plays a key role in the automotive industries. Traditionally, this has been performed mainly with prototype tests in the actual service conditions or by using simulative tests with digitally controlled servo-hydraulic equipment. However, the understanding of fatigue failure mechanisms under multi-axial loading conditions is still a practical need in order to propose design material modifications against active damage process. Therefore, the computer modeling and simulation of multi axial fatigue process is still a cost effective technique in reducing iteration cycles during product development processes.

II. OBJECTIVE

1. Optimization of Rim using different materials- AL Alloy, Mg alloy, Steel C1008
2. Develop probability model for prediction of fatigue failures of rim under RFT and Drop Test.
3. To eliminate welding of rim using alloy wheel.
4. Traditionally, wheel design and development is very time consuming, because it needs a number of tests and design iterations before going into production. In modern industry, how to shorten development time and to reduce the number of times of test are important issues. In order to achieve the above objectives, computer aided engineering (CAE) is a useful tool and has been recently carried out to perform a wheel design.
5. Stress distribution on rims varies from one region to another. Based on this type of FEM analysis, we have to decide as to which parts are critical, then, can strengthen those zones.
6. Stress distribution may not be that high on some other parts, hence, excess can be removed from these regions to prevent material extravagance.
7. If the residual stresses remain in the critical zones of the rim, it should be taken into consideration that these parts will be more enduring, hence safer. For FEA of Wheel Rim we have to study effect of Moment applied on mounting holes and disc,
8. Prediction of stresses in to rim under dynamic conditions using FEA.
9. Development of finite element analysis model of Wheel Rim to get a better understanding of the influences of stress condition on the mechanisms of the crack initiation and propagation in steel wheel.
10. Study Failures in Wheel Rims.
11. Estimate Life Cycle period of Rim.

III. SCOPE

1. Replacing steel Wheel.
2. Use of alloy wheel for four wheeler rim.
3. The optimization of four wheeler wheel in combination of RFT and drop test.

IV. METHODOLOGY

Before starting analysis it is important to understand current process. Literature review was carried out to understand past work carried out in field stress analysis of Wheel Rim

To validate the solution following methodology was adopted.

1. Finite Element Method
2. Experimental Method
3. Comparison of above two methods

According to standards Rim should pass following tests:

1. Radial fatigue Test
2. Impact test

V. LITERATURE REVIEW

Song et al. [1] this paper detail studies the failure analysis of a wheel hub from a student designed Formula SAE race car that fractured at the roots of the rim finger attachment region. The wheel hub was identified to be manufactured from a rolled Al 6061 alloy. The experimental characterization included fracture surface analysis and microstructural analysis using scanning electron microscopy, as well as compressive stress–strain testing and micro-hardness testing to determine its mechanical properties. Analysis of the fractured surfaces of the hub revealed beach marks and striations, suggesting a fatigue failure. A kinematic model was developed to determine wheel hub loadings as defined by the car driving history. Detailed loads calculated from a kinematic equilibrium model and material properties obtained from the experiment results were used in a finite element model to simulate the stress distribution and fatigue life of the wheel hub. The wheel simulation results were consistent with the failure mode determined from the fractography study. The failure of a wheel hub from a student designed Formula SAE race car that prematurely fractured at the roots of the rim finger attachment region was studied using experimental characterization, as well as FEA and fatigue life analysis. From spectroscopy, the wheel hub material was identified to be an Al 6061 alloy. SEM analysis revealed second-phase Mg2Si particles averaging 7.8 lm in size within the aluminum matrix. Fractography analysis revealed beach marks and striations on the fracture surfaces of the hub fingers. Considering the relatively small magnitudes of forces involved and the results from the experimental analysis, failure of the wheel hub was due to fatigue. A quasi-static kinematic model was developed to access forces on the wheel hub during the driving conditions defined by the usage history. FEA simulation results confirmed that the hub would fail at the roots of rim attachment fingers and fatigue life analysis predicted a service life of approximately 1000 miles significantly less than the 100 miles the car was driven prior to hub failure. The cyclical loading in the hard left turn loading condition and the wheel hub design allowing large stress concentrations at the roots of the rim fingers were contributing factors in the failure of the front right wheel hub of the car.

Zhanbiao et al. [2] studies a five-piece rim and a two-piece bolt-connected rim were investigated to examine stress levels and fatigue lives on critical regions. The finite element models of the rim/tire assemblies were developed and validated through tire engineering data and previously validated modeling approaches. The rim/tire assemblies were simulated under two conditions, (1) application of a 23,100 kg static load followed by a 24.14 km/h traveling speed and an 82 wheel angle, and (2) application of a 26,900 kg static load followed by an 8.05 km/h traveling speed and an 82 wheel angle. The results revealed that traveling and steering speeds were the key factors in causing high stresses and bolt tension forces. Compared to the five-piece rim, the two-piece rim decreased the maximum stresses by over 30% for both loading conditions; consequently the fatigue lives were increased by over two orders of magnitude. The maximum bolt forces for the two-piece rim were estimated to be 195,680 N and 111,360 N separately. In this study, the FE model of a tire sized 18.00–33 used for the container handler vehicle was developed and validated using engineering data obtained from the tire manufacturer’s engineering data. The conventional five-piece rim and proposed twopiece rim, sized 33–13.00/2.5, compatible with the tire, were modeled using previously validated approaches. Using the validated FE models, the rims were numerically investigated under severe loading and maneuver conditions – static loading followed by rotation traveling and steering of the rims. The two simulated conditions encompassed the rim and tire assemblies traveling at speeds of 8.05 km/h with 26,900 kg static load and24.14 km/h with 23,100 kg static load, respectively.

Fengxiang et al. [3] says that the resistance spot welding is considered one of main joint techniques that have been extensively used in the automotive industry with typically each vehicle containing several thousand spot welds. With rapidly increased requirements in lightweight, many automotive companies have widely been utilizing various advanced materials, such as aluminum alloy and advanced high strength steel (AHSS), to stamp into final products such as automotive body panels and other major structural frames. AHSSs, as one class of promising engineering materials used in car body structures, have drawn increasing attention recently, which allow reducing structural weight but enhancing the performance under operational and crashing conditions. Nevertheless, challenge remains in integration of AHSS into automotive structure through quality welds. Indeed, the weld ability of AHSS signifies a critical issue in vehicle production. To make better use of such advanced materials as dualphase steels for welding process, it is important to characterize and quantify their spot welding behaviors properly. In practical automobile applications, the nugget pullout is a preferable failure test approach because the load-carrying capacity is higher, and inter facial failure usually carries lower mechanical loading and absorbs less energy than the nugget pullout failure. Since the spot weld may generate an inherent crack along the weld nugget, it is of a critical importance to the study on failure modes. In this paper, the failure onsets of resistance spot welding in lap-shear specimens for advanced high strength steels (AHSS) (specifically, DP600 and DP980 base materials respectively) were studied based on analytical elastic and elasto-plastic finite element analysis. The analytical solutions to a lap-shear specimen with a spot weld nugget subjected to the uniformly distributed loading condition were derived. The elastoplastic finite element analysis (FEA) is also conducted to analyze the initial necking or thinning phenomenon. It is found that the angular locations of the maximum equivalent plastic strain or initial necking failure points are located at four angular intervals for the advanced high strength steel (AHSS) plate with a spot weld nugget. The derived stress distributions allow predicting failure behavior and evaluating damage evolution on many engineering structures jointed with spot welds.

Prabha et al. [4] say that wheels which are made from an alloy of aluminum or magnesium are Alloy wheels. They provide better heat conduction and improved cosmetic appearance over steel wheels and are typically lighter for the same strength. To flat steel discs and stamped metal configurations and modern cast and forged aluminum alloys rims of today’s modern vehicles are developed by automotive wheels over the decades from early spoken designs of wood and steel, carryovers from wagon and bicycle technology. For analyzing the stress and the displacement distribution in vehicle wheels subjected to conjoint influence of inflation pressure and the radial load this project work generalizes the application of Finite Element Analysis Techniques. The most commonly used considerations in Alloy wheels are illustrated. The analysis is carried out by using “ANSYS” and MSC NASTRAN finite element package, and the model is done by using “PRO/E”. The wheel is modeled by using ten nodded tetrahedron solid elements; for the analysis of linear elastic with isotropic conditions the constitutive material model is selected. With the radial load on the stress and displacement in tire rims, through experimental stress analysis and finite element analysis, we examine the effects of tire air pressure in conjunction. 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. The proposed safety factors will be useful for manufacturers/designers for reliable fatigue life prediction of similar structural components subjected to radial fatigue load. By using ANSYS we determine the total deformation and stresses developed in alloy wheel.

Satyanarayana et al. [5] studies the detail “Fatigue Analysis of Aluminum Alloy Wheel under Radial Load”. During the part of project a static and fatigue analysis of aluminum alloy wheel A356.2 was carried out using FEA package. The 3 dimensional model of the wheel was designed using CATIA. Then the 3-D model was imported into ANSYS using the IGES format. The finite element idealization of this modal was then produced using the 10 node tetrahedron solid element. The analysis was performed in a static condition. This is constrained in all degree of freedom at the PCD and hub portion. The pressure is applied on the rim. We find out the total deformation, alternative stress and shear stress by using FEA software. And also we find out the life, safety factor and damage of alloy wheel by using S-N curve. S-N curve is input for A.356.2 material. The total deformation of wheel maximum is 0.2833mm and minimum is 0.031478 at hub portion. The alloy wheel of shear stress maximum is 48.195 and minimum is -48.241 at hub. The equivalent stress is 163.97 and 0.038. The life of wheel maximum 1.7667e6 cycles and the minimum cycles of wheel is 1.6533e5 at a cross sectional area of 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. The proposed safety factors will be useful for manufacturers/designers for reliable fatigue life prediction of similar structural components subjected to radial fatigue load. By using ANSYS we determine the total deformation and stresses developed in alloy wheel.

Firat et al. [6] A computational methodology is proposed for fatigue damage assessment of metallic automotive components and its application is presented with numerical simulations of wheel radial fatigue tests. The technique is based on the local strain approach in conjunction with linear elastic FE stress analysis. The stress– strain response at a material point is computed with a cyclic plasticity model coupled with a notch stress–strain approximation scheme. Critical plane damage parameters are used in the characterization of fatigue damage under multi axial loading conditions. All computational modules are implemented into a software tool and used in the simulation of radial fatigue tests of a disk-type truck wheel. In numerical models, the wheel rotation is included with a non-proportional cyclic loading history, and dynamic effects due to wheel–tire interaction are neglected. The fatigue lives and potential crack locations are predicted using effective strain, Smith–Watson–Topper and Fatemi–Socie parameters using computed stress–strain histories. Three-different test conditions are simulated, and both number of test cycles and crack initiation sites are estimated. Comparisons with the actual tests proved the applicability of the proposed approach. In this paper, a computational methodology is proposed for fatigue life and failure prediction of automotive components and its application is presented with numerical simulations of radial fatigue tests of a disk-type truck wheel. Following a short review of theoretical models, the computer modeling of wheel radial fatigue tests were described in conjunction with linear elastic FE stress analysis. In simulation models, the wheel rotation is included with a non-proportional cyclic loading history, and wheel– tire interaction is neglected. The fatigue test cycles and failure locations for three test conditions were predicted using effective strain, Smith–Watson–Topper and Fatemi– Socie parameters using computed stress–strain histories.

D’Andrea et al. [7] studies the uses the finite element model (FEM) results of a structure to evaluate the stress state at the layer interface changes during the passage of a wheel over the road. In this study FEM interface stress states, inside and outside the tire track, are compared with the histories of samples tested with the most common devices. The histories are very different according to condition first, whether the wheel passes directly over the point of observation and second along other intermediate alignments. In the first case normal pressure and longitudinal shear stress are relevant, while in the second case the transverse shear stress is prevalent. The stress histories have been compared with those applied using currently available laboratory testing devices. Surely, the monotonic test modality is far from the actual field condition, unless an extreme load should be applied for a one-time slipping at the interface. For the interface fatigue accumulation phenomenon, to which this study is devoted, the dynamic tests are interesting and these judgments can be drawn: The stress conditions applied in alternate pure shear tests, such as for sinusoidal loading form in the direct shear test, can be only considered a precautionary simulation of the alignment under the wheel, as the longitudinal shear stress actually varies symmetrically from positive to negative. It is precautionary because the normal pressure applied halfway through loading time, as always happens in the field, and causes low damage potential followed by a higher fatigue life for the material. The guillotine test with one way impulses is only representative of alignments far from the wheel print edge, where the transverse shear stress reaches rather high values with a contextual very low vertical pressure. As it is impossible to apply the longitudinal shear stress, this modality can be less conservative than that in the field and, generally speaking, is of little relevance. The shear test devices that apply a constant compression generate stress histories very different from those expected in the field and not to be recommended because of the extra resistance due to the presence of the normal pressure throughout the test. The inclined devices with triangular shear impulses appear to simulate quite well the stress conditions of points located under the wheel edge zone. To conclude, none of the existing devices can mimic the typical stress histories of the different alignments, merely approximating more or less some of these. Currently, the best theoretical choice for a fatigue analysis may be to combine data from the inclined devices at different angles, such as the Italian SISTM, and alternate pure shear devices, such as the French DST. That is obviously not a practical way; then different test modalities and/or new devices are needed to better simulate on field interface stresses.

Zheng [8] A computational methodology is proposed to simulate wheel dynamic cornering fatigue test and estimate its’ multi-axial fatigue life. The technique is based on the critical plane theory and the finite element methods. The prediction of fatigue life is found to be in close agreement with the corresponding experiment. The stress states of wheel are basically biaxial tensile and compression normal stresses during the prototype test. The principal stresses are not proportional and the unstable principle plane is changing with loading direction, which indicates that the fatigue crack may occur first in the circumferential direction of steel wheel. A computational methodology is proposed for fatigue life and failure prediction of automotive steel wheel by the simulations of dynamic cornering fatigue test. Following with a short review of theoretical models, numerical simulation models were described in conjunction with bilinear elasto-plastic finite element stress analysis under wheel rotating loading. The fatigue life and crack initiation locations are calculated using effective strain, Brown–Miller damage criterion, rain flow counting method and Palmgren–Miner cumulative damage rule.

Xiaofei et al. [9] studies the traditional fatigue test of wheel comprising the radial and cornering fatigue tests cannot simulate there al stress state of wheel well. Biaxial wheel fatigue test combining the set traditional tests has become an internationally recognized method that can reproduce there loading condition of the wheel in service. Since the test is time and cost consuming, developing the simulation method on biaxial wheel fatigue test is urgently necessary. In this paper, a new method is proposed to evaluate the fatigue life of commercial vehicle wheel, in which the finite element model of biaxial wheel fatigue test rig is established based on the standard so FEAUS 3.23 and SAEJ 2562, and the simulation of biaxial wheel test and fatigue life estimation considering the effects of tire and wheel camber is performed by applying the hole load spectrum specified in ES3.23 to the wheel. The radial and cornering fatigue tests are also simulated, and the results are compared with one of the biaxial fatigue test. Their search shows that the proposed method provides an efficient tool for predicting the fatigue life of the wheel in the biaxial fatigue test. In this paper, a method of the biaxial fatigue test of a wheel according to EUSWAES 3.23 is proposed based on the integrated fatigue analysis. The finite element model of biaxial wheel fatigue test rig is established, and the simulation of biaxial wheel test and fatigue life estimation considering the effects of tire and wheel camber is performed by applying the hole biaxial load sequence specified in 3.23 to wheel. Biaxial wheel fatigue test is very different from the traditional radial and cornering fatigue tests. The wheel camber is generated by lateral force, which has a significance an influence on stress distribution of the wheel. The simulation considering the wheel camber angles may provide more practical result to evaluate the fatigue life of the wheel. (2)The dangerous positions are mostly located at the area of air ventilation hole and rim hump and the crack will firstly emerge at the area of air ventilation hole after a driving distance of 6980.6km. The durability performance of wheel evaluated by radial fatigue test and cornering fatigue test can satisfy each requirement well however, the wheel fails in biaxial fatigue test. Compared with the traditional fatigue tests, the biaxial fatigue test has provided a more rigorous but more practical standard for the development of wheel.

Fang et al. [10] the present research aims to investigate general laws of three-pass roll forming of steel Wheel Rim by finite element simulation. Firstly, finite element models of the rolling process were built on ABAQUS. To ensure the validity of models, some important settings as multistep construction, flexible boundary conditions of side rolls and nonlinear loading curves were considered, which provide the basis for high-accuracy numerical simulation of rim forming. Based on the results of simulation, each pass of the rim forming process was then analyzed. Especially, the investigations of wall thickness distribution and equivalent plastic strain on formed Wheel Rim are conducted, from which the role of three rolling passes and characteristics of rim forming can be summarized. Moreover, experiment results verified the reliability of finite element model. Subsequently, for analyzing the problems of welding-line cracking, model of flaring dies with various flaring angles were tried in simulations to discuss their influences on forming results of the Wheel Rim. FE models of the roll forming of the steel Wheel Rim are developed, and the three-pass roll forming of the Wheel Rim was simulated on ABAQUS/Explicit. Wall thickness and equivalent plastic strain of the formed rim are focused on. Simulation results are compared with the experiment measurement. In term of the wall thickness distribution, the simulation has good agreement with the experiment. The simulation error can be controlled below 7%.The work piece is rotated by the friction forces during the roll forming of the Wheel Rim, and it is easy to swing and shaking. It is hard to control the stability of the work piece both in production and FE model. The action of side rollers has to be considered, which contributes to the rotating work piece stability. In FE model, the side roller presses the work piece by the spring elements. Additionally, the flexible loading of the lower roller is necessary, and the velocity increases slowly to maintain the simulation stability. By the simulations, the roles of three passes of the rim forming are revealed. The first pass forms the basic axial outline of the rim from straight to curve. The second pass profiles the rim edge and the middle groove, and the basic dimension can be determined. The third pass finishes the corners of the rim profile. There are larger strains distributing on the rim edge, which is initially thinned and later thickened. Various flaring angles were tried to apply in the FE models for investigating its effects on the simulation results. The deceasing flaring angle obtained the large deformation, which leaded to the cracking at the weld zone of the Wheel Rim. The optimized flaring angle is obtained by the FE simulation of the flaring and three-pass roll forming.

VI. DETAIL STUDY OF WHEEL RIM

Automotive wheels have evolved over the decades from early spoke designs of wood and steel, carryovers from wagon and bicycle technology, to flat steel discs and finally to the stamped metal configurations and modern cast and forged aluminum alloys rims of today's modern vehicles. Historically, successful designs arrived after years of experience and extensive field testing. Since the 1970's several innovative methods of testing well aided with experimental stress measurements have been initiated. In recent years, the procedures have been improved by a variety of experimental and analytical methods for structural analysis (strain gauge and finite element methods). Within the past 10 years, durability analysis (fatigue life predication) and reliability methods for dealing with the variations inherent in engineering structure have been applied to the automotive wheel. Wheels are clearly safety related components and hence fatigue performance and the state of stress in the rim under various loading conditions are prime concerns. Further, wheels continue to receive a considerable amount of attention as part of industry efforts to reduce weight through material substitution and down gauging. Although 2 wheels are loaded in a complex manner and are highly stressed in the course of their rolling duty, light weight is one of the prime requirements, hence cast and forged aluminum alloys are essential in the design. Light aluminum alloy wheels enjoy a great popularity at present. For many consumers, a perceived exclusivity is a predominate factor, and these wheels are even considered as status symbols. The importance of wheel and tires in the automobile cannot be challenged. Without engine, car may tow, but without the wheels, this is not possible. The wheel with tires takes full load, and reduces friction, and provides cushioning effect to passenger by absorbing vibration due to road surface unevenness and assist in steering control. The alloy of conventional disc wheel in case of car and wire wheel as in case of motorbike has better aesthetic looks and easy of manufacturing.

The main requirements of an automobile wheel are:

1. It should be as light as possible so that unsprung weight is least.
2. It should be balanced statically as well as dynamically.
3. It should be possible to remove or mount the wheel easily.
4. It material should not deteriorate with weathering and age.

a) Material of Rim

The development of wheel is traced from a material viewpoint beginning with wood, the first documented wheel material and ending with new materials under development such as composites and titanium. While it is impossible to imagine what civilization would like without a wheel, many early civilizations has numerous other tools but did not possess wheels. Undocumented legend has it that Chinese philosopher was inspired while watching a flower rolled by wind over the grass. In the period from 1900 to 1935 there were many different types of wheel materials and methods of construction in use. These include wood spoke, cast and forged steel, disc steel, cast Aluminum and wire wheels of all the material used in early 1900's only one is not still in use today- wood. The predominant wheel material is now steel but the shape; size and method of manufacturing have drastically changed. By 1935 the passenger car wheel diameter has been reduced from 36"to 16" and rim width is increased from 3" to 6" with shrunk in rim diameter from 36" to 44"diameter to 20" to 24" diameter.

b) Steel

As the volume of passenger cars increased the only material and method of manufacture that could provide an economic wheel was the disc wheel formed from hot sheet rolled. The rim was made by roll forming a flash butt-welded hoop. Mechanically caped

SAE 1008 and 1010 grades were the typical rim materials. Mechanically caped steel provides higher usable metal yield from ingot and more uniform chemical through the thickness of the sheet which improved the butt weld ability. Rimmed steel in SAE grade 1012 and 1015 were used for the disc because on hot rolled sheet that was very low in alloy content. In the early 1950’s the tubeless tires were introduced and they added challenge for the wheel maker the rim had to be air tight. It was difficult to insure that air leakage would not occur around the rivet so other methods of attaching the rim to the disc were investigated and the resistance spot weld and the arc weld attachments were developed. The spot weld was initially favored because it was very similar in function to a rivet, and no material had to be added to add to the weld joint. The desire for lighter, more fuel efficient vehicle resulted in changing from rear wheel drive to front drive. This necessitated designing the wheel with much deeper disc to clear the front drive mechanism. The deeper disc increased the stresses so that heavier stock was required to provide adequate fatigue performance.

c) Aluminium Alloy Wheel

Aluminum is a metal with features of excellent lightness, thermal conductivity, corrosion resistance, characteristics of casting, low temperature, machine processing and recycling, etc. This metals main advantage is reduced weight, high accuracy and design choices of the wheel. This metal is useful for energy conservation because it is possible to re-cycle aluminum easily

Aluminum alloy:
Young’s modulus (E) =72000 N/mm2
Yield stress=160 N/mm2
Density =2600kg/m

d) Magnesium Alloy Wheel

Magnesium is about 30% lighter than aluminum, and also, excellent as for size stability and impact resistance. However, its use is mainly restricted to racing, which needs the features of lightness and high strength at the expense of corrosion resistance and design choice, etc. compared with aluminum. Recently, the technology for casting and forging is improved, and the corrosion resistance of magnesium is also improving. This material is receiving special attention due to the renewed interest in energy conservation.

Magnesium alloy:
Young’s modulus (E) =45000N/mm2
Yield stress=130 N/mm2
Density =1800kg/m3

e) Wheel Rim Description

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. For example, in a four wheeler the rim is a hoop attached to the outer ends of the spokes-arm of the wheel that holds the tire and tube. 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. To support the cylindrical rim structure, a disc is made by stamping a metal plate. It has to have appropriate holes for the center hub and lug nuts. The radial outer surface of the wheel disk has a cylindrical geometry to fit inside the rim. The rim and wheel disk are assembled by fitting together under the outer seat of the rim and the assembly welded together. 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.

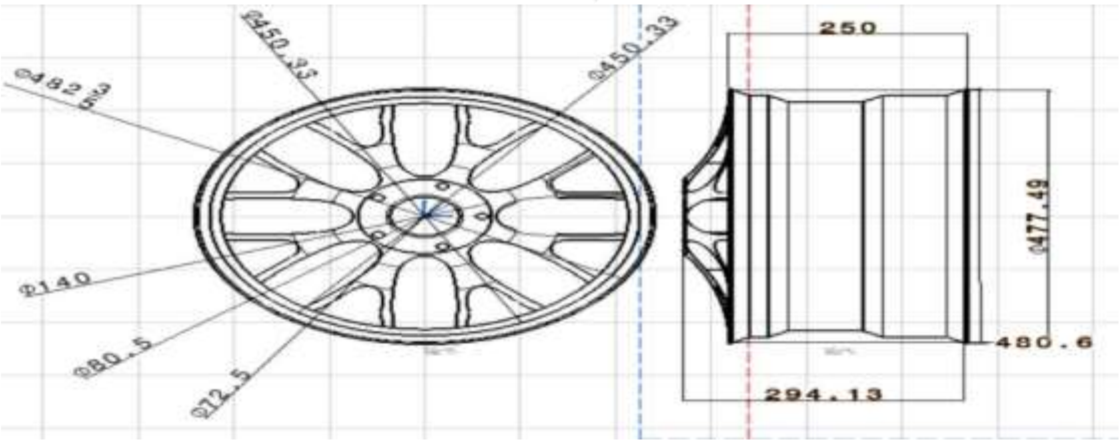

Fig: 2D Design of Wheel Rim

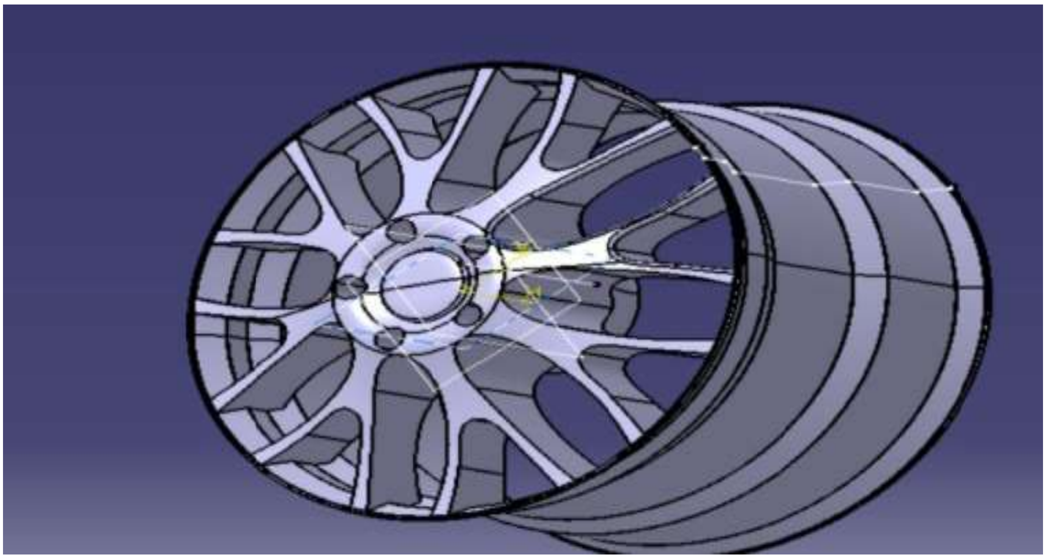

Fig.: 3D View of Wheel Rim

VII. CALCULATIONS OF RADIAL FATIGUE TEST

Table: Input Parameters Details

Parameters	Symbol	Value
Maximum Static Load Radius	R	0.2115
Coefficient of Friction	u	0.7
Inset or Offset	d	0.0165m
Minimum Load Design	Fmin	250kg
Maximum Load Design	Fmax	495kg
Test Acceleration Factor	s	2.25

Table : Machine constants

Machine Constants	Symbol	Value
Load	M	18.84kg
Radius at which load applied	r	0.292m
Length of Rod	L	0.8m
Test Acceleration Factor	K	2.25

Radial Fatigue Test load (L1) = $F_{max} \times K$

Radial Fatigue Test load (L2) = $F_{min} \times K$

Table: Load and Pressure Calculations

Parameters	Symbol	Value
Maximum RFT load	L1	10930 N
Minimum RFT load	L2	5520 N
Minimum Pressure	P1	241316.6 Pa
Maximum Pressure	P2	275790.4 Pa

VIII. FINITE ELEMENT ANALYSIS FOR RADIAL FATIGUE TEST AND DROP TEST

a) Stress Analysis using FEA

The finite element method have identified the failure locations in the wheel bolt holes,ventilation holes and welding seam. FEM decides critical and strengthen zones of wheel rim part.

Table: Radial Fatigue test Parameters

Sr. No.	Pressure	Load	Material
1	P1	L1	M1
2	P2	L1	M1
3	P1	L2	M1
4	P2	L2	M1
5	P1	L1	M2
6	P2	L1	M2
7	P1	L2	M2
8	P2	L2	M2
9	P1	L1	M3
10	P2	L1	M3
11	P1	L2	M3
12	P2	L2	M3

Where, P1= Minimum pressure
P2=Maximum pressure
L1=Minimum load
L2=maximum load

M1=Steel C1008
M2=Aluminium Alloy
M3=Magnesium Alloy

A. Equivalent stress for Steel C1008

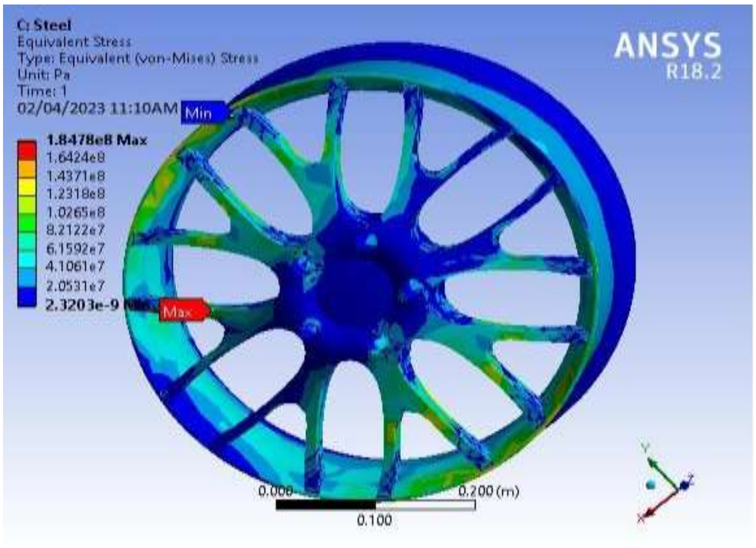

Fig.: Steel at Minimum Pressure (P1) and Minimum Load (L1)

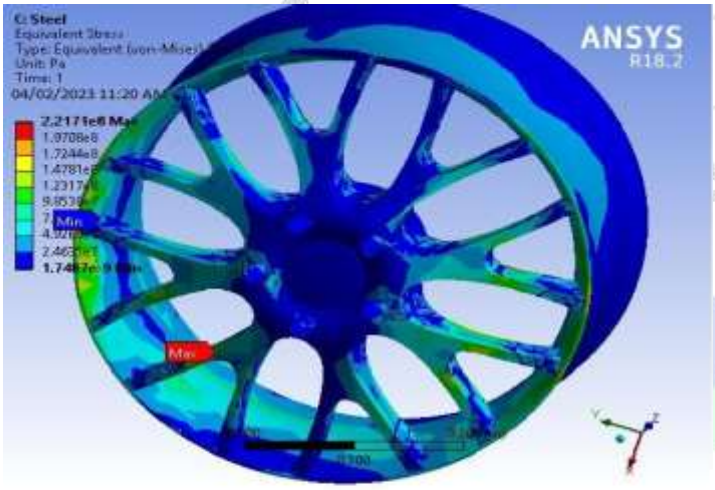

Fig.: Steel at Maximum Pressure (P2) and Minimum Load (L1)

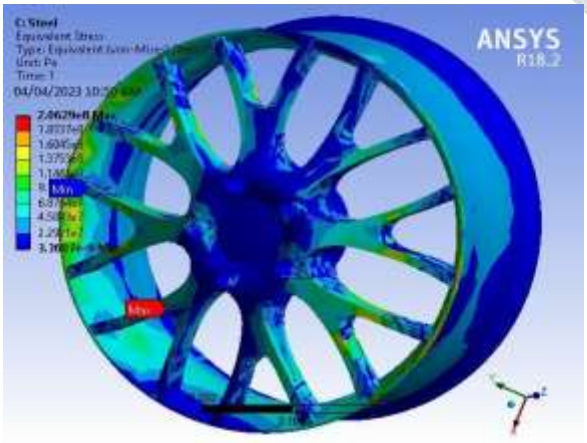

Fig.: Steel at Minimum Pressure (P1) and Maximum Load (L2)

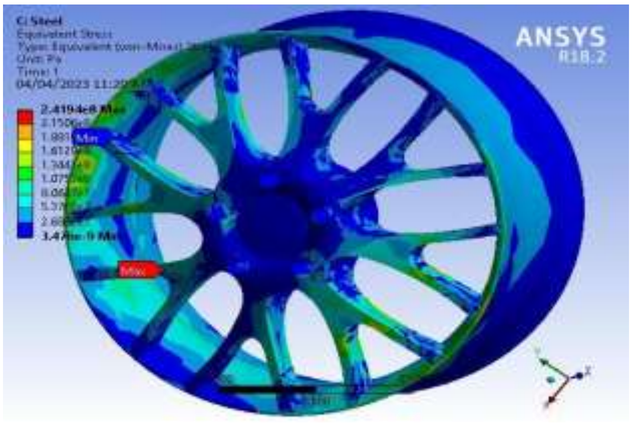

Fig: Steel at Maximum Pressure (P2) and Maximum Load (L2)

b) Equivalent stress for Aluminum alloy

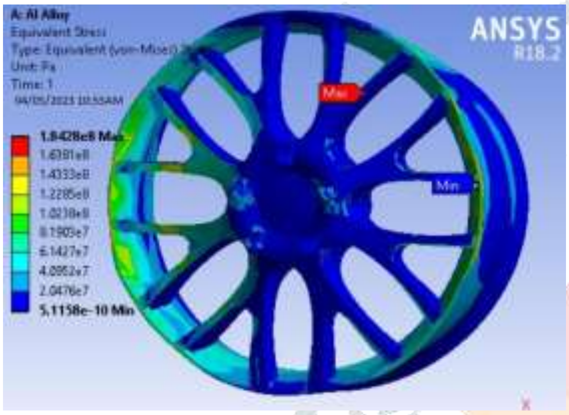

Fig.: Aluminum Alloy at Minimum Pressure (P1) and Minimum Load (L1)

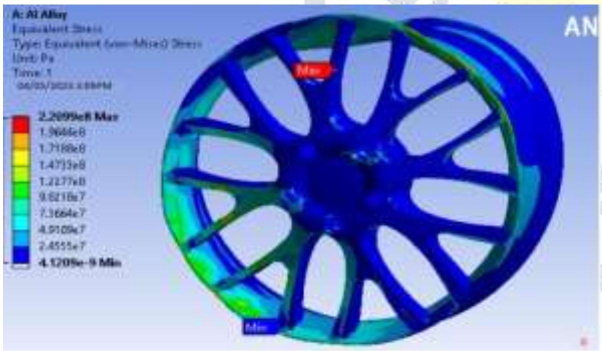

Fig.: Aluminum Alloy at Maximum Pressure (P2) and Minimum Load (L1)

c) Equivalent stress for Magnesium Alloy

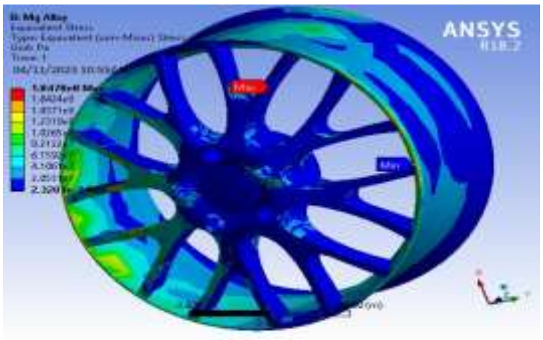

Fig.: Magnesium Alloy at Minimum Pressure (P1) and Minimum Load (L1)

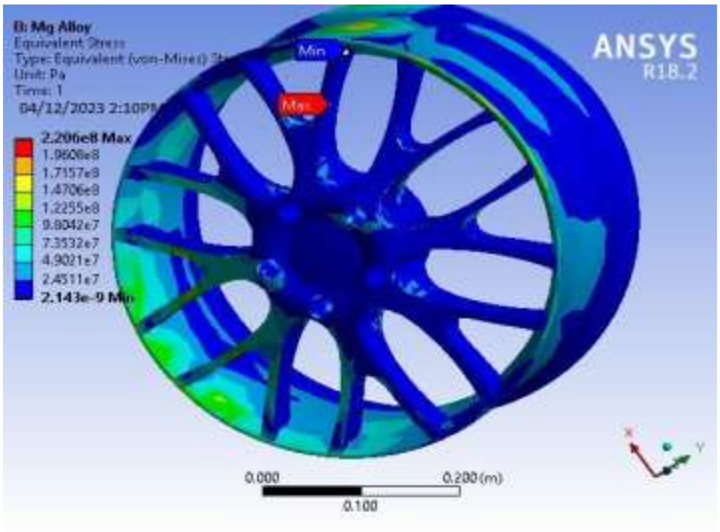

Fig Magnesium Alloy at Maximum Pressure (P2) and Minimum Load (L1)

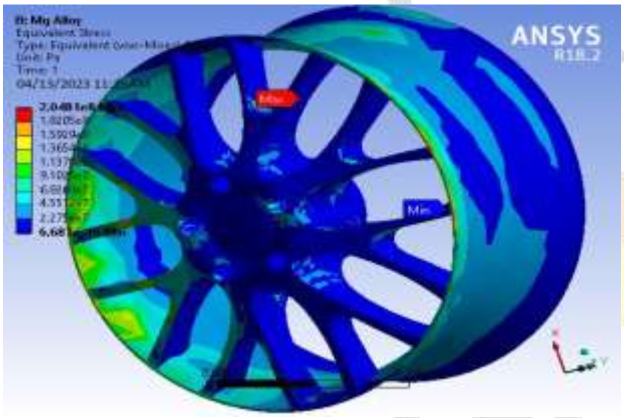

Fig: Magnesium Alloy at Minimum Pressure (P1) and Maximum Load (L2)

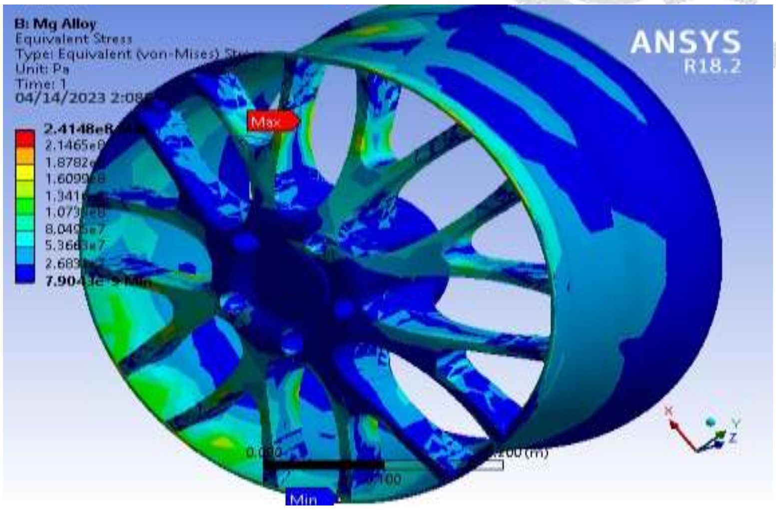

Fig.: Magnesium Alloy at Maximum Pressure (P2) and Maximum Load (L2)

Table : Equivalent stress results of RFT

Sr.No.	Pressure	Load	Material	Equivalent Stress ($X10^8$)	Result
1	P1	L1	M1	1.847	Safe
2	P1	L1	M2	1.842	Safe
3	P1	L1	M3	1.847	Safe
4	P1	L2	M1	2.217	Safe
5	P1	L2	M2	2.209	Safe
6	**P1**	**L2**	**M3**	**2.206**	**Unsafe**
7	P2	L1	M1	2.062	Safe
8	P2	L1	M2	2.052	Safe
9	**P2**	**L1**	**M3**	**2.048**	**Unsafe**
10	P2	L2	M1	2.419	Safe
11	P2	L2	M2	2.419	Safe
12	**P2**	**L2**	**M3**	**2.414**	**Unsafe**

Allowable stress for Steel=3.2×10Mpa
Allowable stress for Aluminium alloy=2.8×10Mpa
Allowable Stress for Magnesium alloy=1.93×10 Mpa

From above analysis it is clear that the Magnesium alloy rim fails for the given conditions. Therefore for optimization purpose we will consider the Aluminum Alloy and Steel.

Table: Optimize results

Sr. No.	Material	Mass(Kg)	Weight(N)	Result	Optimize result
1	Steel	67	657.27	Safe	
2	**Al Alloy**	**24**	**235.44**	**Safe**	**Optimize**
3	Mg Alloy	17	166.77	Failed	

From the above table it is clear that the Mass and weight of Steel and magnesium alloy is greater than aluminium alloy For Validation purpose we are using All Load and Pressure condition for Aluminum Alloy Material. So Experiments will be as follows:

Table: Optimize Results for validation

Exp.No.	Pressure	Load	Material	Equivalent Stress ($X10^8$)	Result
2	P1	L1	M2	1.842	**Safe**
5	P1	L2	M2	2.209	**Safe**
8	P2	L1	M2	2.052	**Safe**
11	P2	L2	M2	2.419	**Safe**

IX. RESULTS AND DISCUSSION

A. Radial Fatigue Test

Stain results of FEA and Experimentation are compared with each other.

Table : Comparison of Experimental and FEA Results of Radial Fatigue test

Exp. No.	Position	Experimental Results	FEA Strain Results	Difference (%)
2	1	0.00105	0.0011	2.14
	2	0.00130	0.0014	5.45
	3	0.00038	0.00040	6.02
5	1	0.00365	0.0039	6.05
	2	0.00168	0.0018	7.97
	3	0.00033	0.00034	2.99
8	1	0.00115	0.0012	7.26
	2	0.00145	0.0016	7.05
	3	0.00048	0.00049	2.20
11	1	0.00113	0.0012	4.50
	2	0.00138	0.0015	8.15
	3	0.00058	0.00061	6.31

From Experiment No. 2, 5, 8 and 11 shows difference between FEA results and Experimental results within acceptable range.

The Comparison between FEA Results and Experimental results of radial fatigue test are graphically shown in below:

In Figure point observed at position 1, 2 and 3 respectively having FEA values 0.0011, 0.0014, and 0.0004 compared to experimental values 0.00105, 0.0013 and 0.00038 slightly varied with respect to each other.

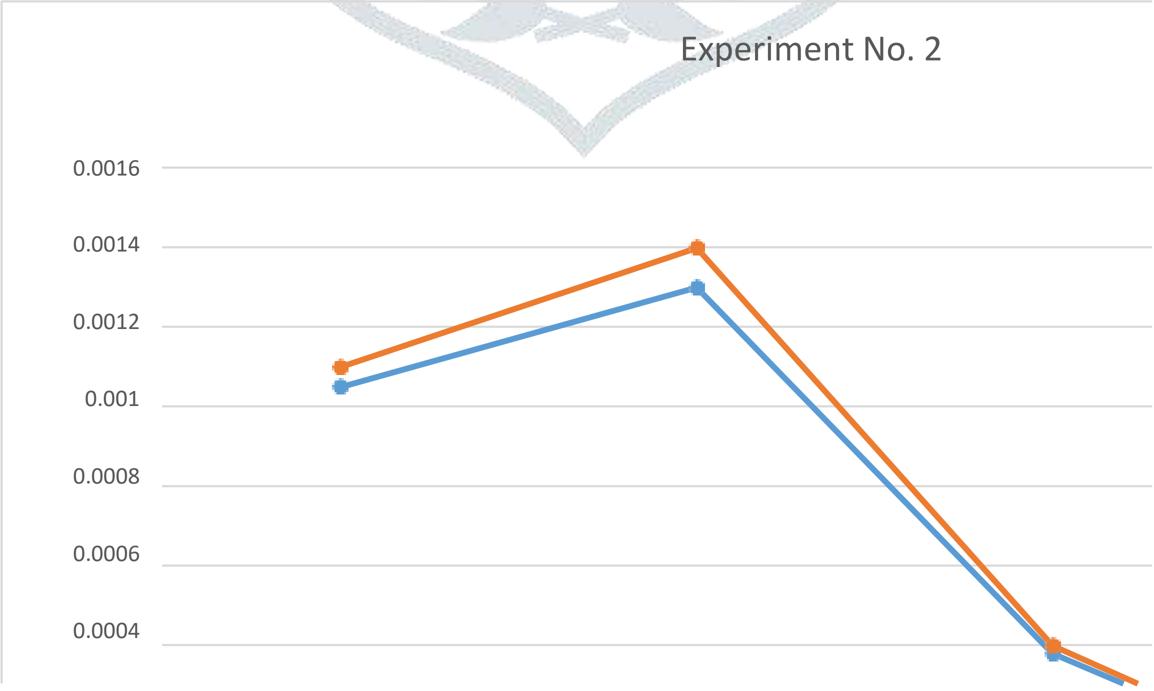

Fig. : Comparison of FEA and Experiment No. 2 Result

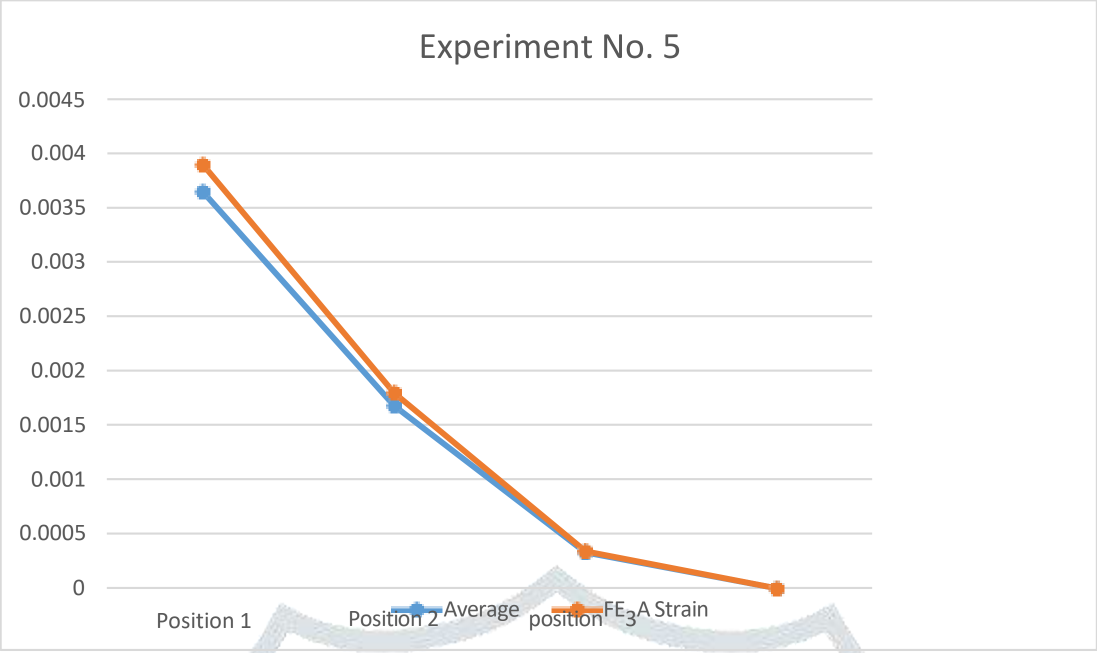

Fig. 7.2: Comparison of FEA and Experiment No. 5 Result

In Figure small deviation observed at position 1, 2 and 3 having FEA values 0.0039, 0.0018 and 0.00034 compared to experimental values 0.0036, 0.0016, and 0.00033 at respective position.

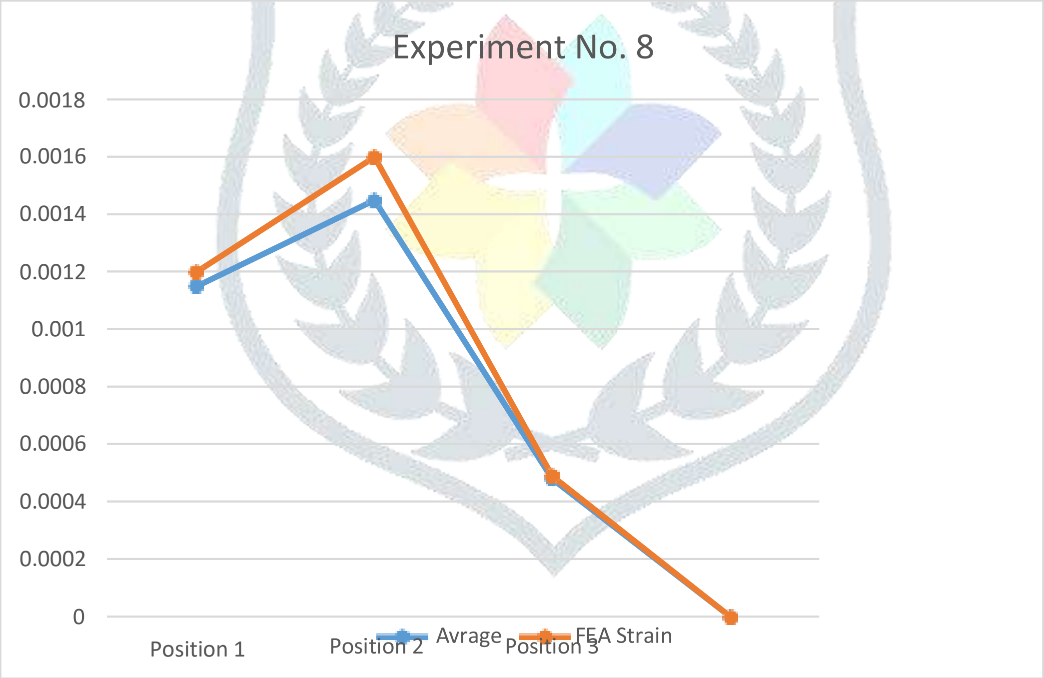

Fig. : Comparison of FEA and Experiment No. 8 Result

In Figure small difference observed at position 1, 2 and 3 having FEA values 0.0012, 0.0016 and 0.00049 compared to experimental values 0.00115, 0.00145, and 0.00048 at respective position.

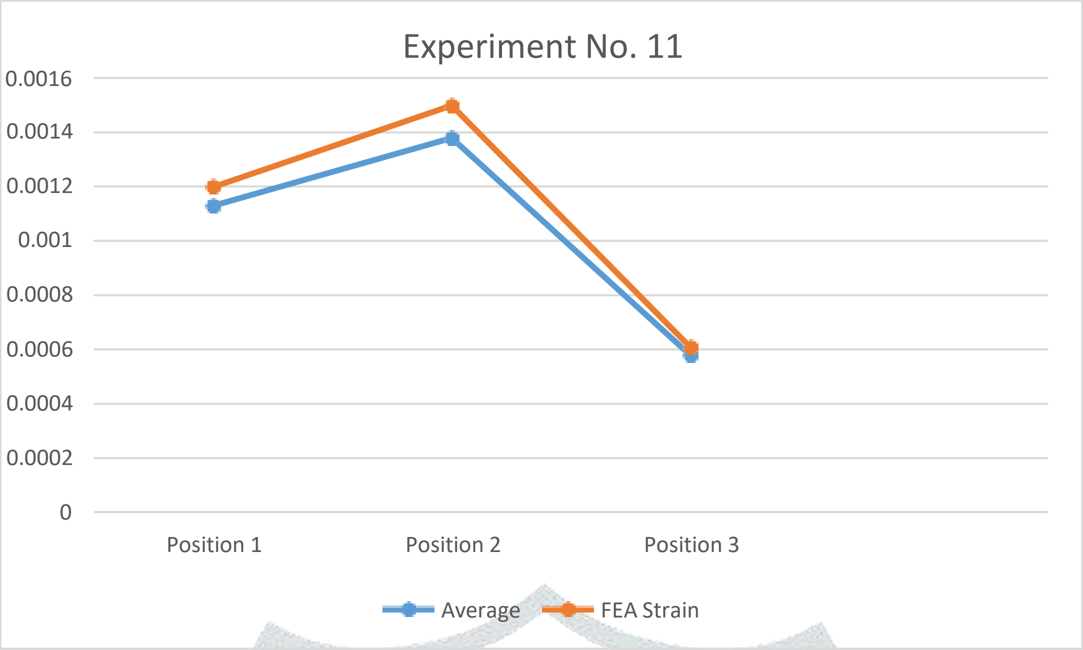

Fig. Comparison of FEA and Experiment No. 11 Result

In Figure small change observed at position 1, 2 and 3 having FEA values 0.0012, 0.0015 and 0.00061 compared to experimental values 0.00113, 0.00138, and 0.00058 at respective position.

Difference between FEA Strain results and Experimental Strain results of Radial fatigue test found to be within permissible range, this method for optimization fountto be valid.

B) Drop Test

Table : Comparison of Experimental and FEA Results of Drop Test

Position	**Experimental strain**	**FEA strain**	**Error %**
1	0.1795	0.191	6.02
2	0.1605	0.174	7.76

From above comparison, FEA results are near about same as that of Experimental values. Which Validates the Drop test and our Optimization work.

The Comparison between FEA Results and Experimental results of Drop test are graphically shown in below:

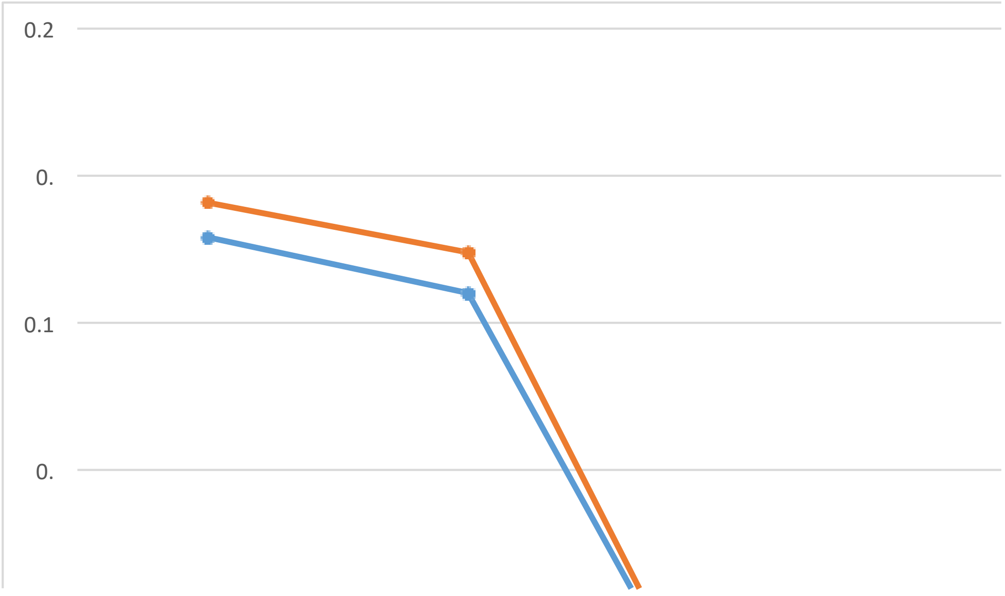

Fig Comparison of FEA and Experimental Results of Drop Test

X. CONCLUSION

1. FEA Carried out on Different materials like steel c1008, Al alloy, Mg alloy easily discriminates their affordability of use. Magnesium Alloy Fails for givenrange of loading. steel and Aluminium Alloy are safe for given loading.

2. Out of steel and Al alloy rim, Al alloy rim is 2.8 times lighter than that of steel. So Al alloy is best suited material for Wheel rim of Car.

3. RFT and Fea Gives nearly same results with deviation within permissible limit.

4. Al alloys Eliminates need of conventional welding Processes as wheel rim can be manufactured simply by casting.

XI. ACKNOWLEDGEMENT

With all respect and gratitude, I would like to thank all people who have helped me directly or indirectly for this dissertation. I express my heartily thankfulness towards my guide and PG-co-ordinator Mr. A. P. Ghodake to complete this dissertation work on “Optimization of Four Wheeler Wheel Rim for Al, Mg and Steel Alloy”. His ideas, directions gave me moral boost up to complete this dissertation. Thanks to Head of Mechanical Department Dr. H. U. Pawar, Principal Dr. D. M. Yadav and all the Teaching & Non-teaching staff members of Mechanical Engineering Department for providing necessary information and required resources timely. I intend to conclude this acknowledgement with the deep thanks and sincere to my friends and parents who have helped me to complete this dissertation.

XII. REFERENCES

1. Song, W., Woods, J. L., Davis, R. T., Offutt, J. K., Bellis, E. P., Handler, E. S., & Stone, T. W. “Failure Analysis and Simulation Evaluation of an Al 6061 Alloy Wheel Hub”, Journal of Failure Analysis and Prevention (2015), 15(4), pp.521-533
.

2. Li, Z., DiCecco, S., Altenhof, W., Thomas, M., Banting, R., & Hu, H. “Stress and fatigue life analysis of a five-piece rim and the proposed optimization with a two-piece rim”, Journal of Terramechanics, (2014), 52, pp.31-45.

3. Xu, F., Sun, G., Li, G., & Li, Q. “Failure analysis for resistance spot welding in lap-shear specimens”, International Journal of Mechanical Sciences, (2014), 78, pp.154-166.

4. Prabha M.V and Pendyala Veera Raju, “Design And Development Of Aluminium Alloy Wheels”, International Journal Of Advanced Science, Engineering And Technology, (2012),Vol 1, Issue 2, pp.55-60.

5. Satyanarayana, N., & Sambaiah, C. “Fatigue Analysis of Aluminum Alloy Wheel under Radial Load”, International Journal of Mechanical and Industrial Engineering (IJMIE), (2012), ISSN, (2231-6477), pp.1-6.

6. Zheng, Z. G., Sun, T., Xu, X. Y., Pan, S. Q., & Yuan, S. “Numerical simulation of steel wheel dynamic cornering fatigue test”, Engineering Failure Analysis, (2014), 39, pp.124-134.

7. Firat, M., Kozan, R., Ozsoy, M., & Mete, O. H. “Numerical modeling and simulation of wheel radial fatigue tests”, Engineering Failure Analysis, (2009), 16(5), pp.1533-1541.

8. D’Andrea, A., & Tozzo, C. “Interface stress state in the most common shear tests Construction and Building Materials”, (2016), 107, pp.341-355.

9. Irastorza-Landa, A.,Van Swygenhoven, H., Van Petegem, S., Grilli, N., Bollhalder, A.,Brandstetter, S., & Grolimund, D. “Following dislocation patterning during fatigue Acta Materialia”, (2016), 112, pp.184-193.

10. Wan, X., Shan, Y., Liu, X., Wang, H., & Wang, J. “Simulation of biaxial wheel test and fatigue life estimation considering the influence of tire and wheel camber”, Advances in Engineering Software, (2016), 92, pp.57-64