International Journal of Interdisciplinary Innovative Research & Development (IJIIRD)
ISSN: 2456-236X
Vol. 02 Issue 02 | 2018

Review of Design,Analysis of Four Wheeler Alloy Material Rim using FEA Method under Cornering Fatigue Test

Rahul B. Chopade[1], Prof Ajitabh Pateriya[2], *Dr.* A. D. *Shirbhate*[3]

[1] *PG Student, Mechanical Engineering Department,V.B.K.C.O.E.Malkapur,M.S. India*
[2] *Associate Professor, Mechanical Engineering Department, V.B.K.C.O.E.Malkapur,M.S. India*
[3] *Associate Professor, Mechanical Engineering Department, PRMITR, Badnera,M.S. India*

ABSTRACT

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.

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. A Multi-objective analysis concept is carried out to optimize the weight of the Rim. Also, to determine whether the moment is applied at mounting holes or at Hub also. Work is carried out in steps by step manner. We tried to minimize the number of Experiments and levels of Experiments.

Keyword : - FEA, CAD, DFCT,CFT

1. INTRODUCTION

In auto industries, wheels are considered as most critical components as it play a vital role in human safety. From past decades, wheel producers are using new materials and manufacturing technologies in order to improve the wheel's aesthetic appearance and design. Steel wheels are widely used for wheels due to their excellent properties, such as lightweight, good forge ability, high wear resistance and mechanical strength. Ensuring the reliability and safety of wheel is very important. [1]

Analysis of the rims consists of numerically analyzing the stress levels that rims experience during operating conditions. These stress levels will then serve as input parameters for a fatigue analysis of the rims to evaluate their respective fatigue life. Additionally, the load bearing capacity of the bolt pattern will be evaluated for conditions of severe loading. The finite element (FE) method is implemented for all rim analysis. The reliability of FEA approach is based on their previous experience in fatigue analysis studies .The magnitude of the static load and pressure contributes to increasing the stresses on the rim components. [2]

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;

i. It should be as light as possible so that unsprung weight is least
ii. It should be strong enough to perform the above functions.
iii. It should be balanced statically as well as dynamically.
iv. 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.[4]

1.1 Cornering fatigue test

The dynamic cornering fatigue test is a standard SAE test, which simulates cornering induced loads to the wheel. Fig 1 shows the test system in which the test wheel is mounted to the rotating table, the moment arm is fixed to the wheel outer mounting pad with the bolts and a constant force is applied at the tip of the moment arm by the loading actuator and bearing, thus imparting a constant rotating bending moment to the wheel. If the wheel passes the dynamic cornering fatigue test, it has a good chance of passing all other required durability tests.

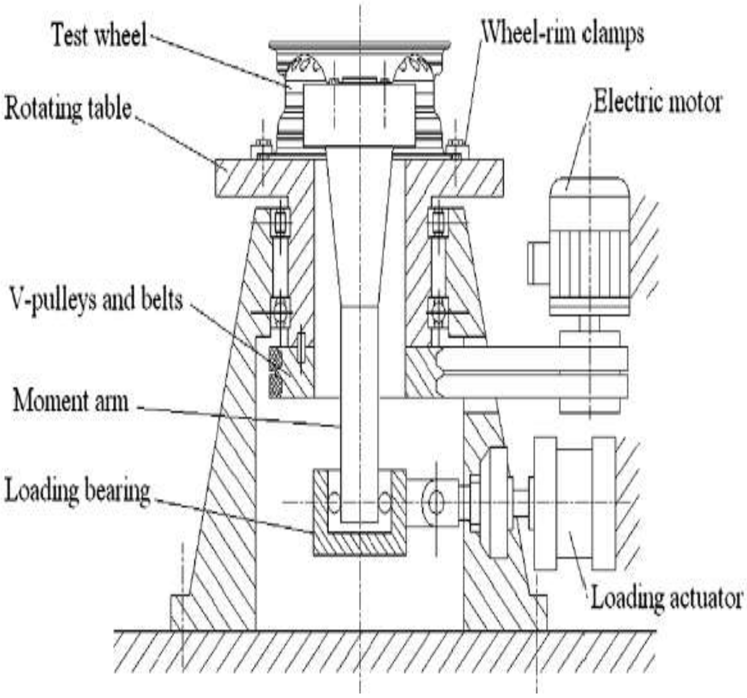

Fig 1 Sketch of the dynamic cornering fatigue test system

The Conering Fatigue Testing (CFT) machine allows the simulation of an endurance fatigue test on car/light truck/bus wheels subjecting them to cornering fatigue stress and holding the test conditions constant throughout the test duration. This is the type of stress a wheel experiences during turning.

The Cornering fatigue test machine performs testing on wheels, under rotating condition, with the bending moment applied at 90 degree to the test wheel.

The test machine is configured to test one wheel at a time. The machine contains load control and speed control servo loops. Applied load is controlled to +/- 1% of set point within stated load range. The servo loop is dynamic in that it will continuously compensate for load changes due to flexing of the wheel.

The deflection of the rod under load is also measured at two locations to provide deflection around 360°. The deflection signals the start of a crack at the disc wheel and therefore, is an important criterion for shut down of the test.

The servo loop can work in constant load mode or constant speed mode. In constant load mode the instantaneous bending moment measured is compared with the user supplied set point producing an error signal. This error signal id fed to the unbalance mass rotation drive which modifies the speed so as to bring this error signal within acceptable limits. The speed servo loop works in order to maintain a user supplied speed regardless of the resulting bending moment.

The function of the machine is to apply a rotating bending moment to the wheel; the resulting stresses on the wheel are very similar to the stresses created under cornering by a car or truck on the road. The Rotary fatigue test machine applies a force to the wheel central disc. The wheel failures on the machine will be to the center of the test wheel and or in the area of the wheel mounting holes.

The machine may be controlled in a manual or fully automatic mode (load, speed) by means of controls mounted on the front of the control panel. The following settings can be adjusted:

i. No. of load cycles
ii. Bending moment or speed set point
iii. Percent value of increase in deflection for shut down criteria
iv. Maximum speed for shutdown criteria
v. Maximum bending moment for shutdown criteria
vi. Information regarding the wheel under test

The following parameters are displayed on the front of the control panel:

i. Applied Load (force / moment)
ii. Speed (moment RPM)
iii. Deflection
iv. Cycles (Total revolutions of wheel)

Photograph of Dynamic Cornering Fatigue Test Setup

1.2 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.

i. Endurance test in direction of radius of rim

The tire on the test rim is rotated under high pressure condition on steel drum and the durability of the rim is examined. Sometimes, test is done giving camber angle and adding a side force.

ii. Test of disc

The rim flange is tested by applying a load from an arm mounted to the hub. A bending moment is applied while the rim rotates.

iii. Impact test

The case where the wheel collides with curb of the road or a large obstacle is assumed and the fall impact examination is done.

iv. Others

The test for welding between rim and disc and the nut seat tightening etc. are provided in the vehicle test standard. Moreover, nondestructive testing such as X ray and color check, etc. are adopted to the light alloy wheel to detect the defects in the casting process. Bead Unseating Test, provided in the tire safety standards, for a mounted tire and the rim is also applied. In addition tests are carried out in the field with the assembly mounted on a vehicle under various road surfaces.

1.4 Failure of a wheel rim

If you have been in an accident or purchased a bike with unknown history it is possible that your motorcycle wheel could be out of true. The wheel might seem to oscillate laterally (side to side) or appear to move up and down (out of round).

Motorcycle rims can be casually inspected by supporting the bike on the centre stand or other stand and spinning them while viewing side on or edgewise. A really bad wobble will be obvious even to someone like me!. You can secure a sharp pencil to the fork or swing arm to help measure smaller variations. If the wheel is badly out of true, especially if the cause is from an accident, you may want to let a professional motorcycle rims shop or dealer do the repair. Sometimes the cause is just from lazy spoke maintenance (shame on you)! The wheel can slowly drift out of true over time. This kind of thing can be repaired yourself if you are up to it.

If you have just had new tires installed and you feel or see a wobble it is more likely that the tire is the cause not bent rims. What can happen when mounting a new tire is the installer fails to get the new tire fully seated on the motorcycle rims. It may be close and because the tire has a tube in it there will be no leak to give it away. What you need to do is this.

i. Examine the sidewall of the tire where it meets the rim to see if there is any indication that the tire is not fully seated. This might show up as a slight variation in the measurement between a mould line on the tire and the rims. This is best done on a center stand if you have one.
ii. Have the installer correct any problem you find. Sometimes stock rims can be difficult to seat properly (or unseat for that matter).
iii. Sometimes what the tire installer will do to correct the problem is overinflate the tire to force the tire to seat. TAKE CARE! I am not suggesting you try this yourself, it can be very dangerous.
iv. Also make sure the tire is installed correctly, arrow pointing in the direction of travel.

2. REVIEWS OF PAPERS

XiaofeiWan (2016) et al The traditional fatigue test of wheel comprising the radial and cornering fatigue tests cannot simulate the real stress state of wheel well. Biaxial wheel fatigue test combining the set wotraditional tests has become an internationally recognized method that can reproduce the real 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 fEUWAES3.23 and SAEJ2562, and the simulation of biaxial wheel test and fatigue life estimation considering the effects of tire and wheel camber is performed by applying the whole load spectrum specified inES3.23 to the wheel. The radial and cornering fatigue tests are also simulated, and the results are compared with ones of the biaxial fatigue test. The research shows that the proposed method provides an efficient tool for predicting the fatigue life of the wheel in the biaxial fatigue test. [1]

In this paper, a method of the biaxial fatigue test of a wheel according to EUWAES3.23 is proposed based on the FE-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 whole biaxial loads sequence specified in ES3.23 to the wheel. The conclusions can be drawn as follows: (1) 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 significant influence on stress distribution of the wheel. The simulation considering the wheel camber angles may provide more practical results 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.

(3) 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 two traditional fatigue tests, the biaxial fatigue test has provided a more rigorous but more practical standard for the development of wheel.[1]

Antonio D'Andrea (2016) et al The finite element model (FEM) results of a pavement structure are used to evaluate how stress state at the layer interface varies during the passage of a wheel over the road surface and to qualify the reasonability of existing dynamic tests used to characterize interface shear behavior. FEM interface stress states, inside and outside the tire track, are compared with stress histories undergone by specimens tested with the most common devices, such as guillotine or inclined. The outcomes clearly highlight that none of the existing devices can mimic the typical stress histories of the different alignments, merely approximating more or less some of these. In fact, in certain conditions, the inclined test configuration is quite good for alignments close to the rim of the wheel. This is no longer true if the traffic wander is considered. The real stress state may only be reproduced with a device that can independently manage normal pressure and shear [2]

A FE Model has been used to evaluate how stress state at the layer interface varies during the passage of a wheel over the road surface, thus obtaining a ‘‘loading story”. The histories are very different according to whether the wheel passes directly over the point of observation, laterally to it or along other intermediate alignments. In the first case normal pressure and longitudinal shear stress are relevant, while in the second case the transversal 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, causes a 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 transversal 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 or have rsine 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.[2]

A Irastorza-Landa(2016) et al., Precursors of failure are dislocation mechanisms at the nanoscale and dislocation organization at the mesoscale responsible for long-range internal stresses and lattice rotation. Detailed information on the link between both scales is missing, computationally and experimentally. Here we present a method based on x-ray Laue diffraction scanning providing time and sub-micron spatially resolved evolution of geometrical necessary dislocations in volumes that are similar to what advanced computational models can achieve. The approach is used to follow dislocation patterning during accumulation of fatigue cycles using a newly developed miniaturized shear device. Performed on Cu during cyclic shear, it reveals early dislocation patterning influenced by pre-existing dislocation structures. The quantitative information on non-homogeneous structure formation and its evolution corresponds to the need for synergies with continuum dislocation plasticity simulations of fatigue or any other type of plastic deformation [3]

In conclusion, using micro-beam x-ray Laue diffraction we have provided the first quantitative information on the transition from uniform to non-uniform dislocation structures in a Cu crystal during cyclic shear. The proposed method is fast, allows imaging of areas far beyond typical pattern dimensions and provides information on rotational gradients and GND densities over several tens of microns, i.e. length scales similar to those addressable in dislocation density based computational models . It can be applied synergistically for validating predictions of dislocation patterning up to large deformation strains during continuous loading or to study the formation of persistent slip bands after many more fatigue. It therefore has a large potential to facilitate a breakthrough in our understanding of 2D dislocation patterning and our possibilities to predict failure by advancing computational models. The proposed technique is complementary to 3D x-ray microscopy using wire techniques and 3D-EBSD , both restricted to a snapshot in time of the microstructure. Together they have the potential to boost synergies between modeling and experiment [3]

Weiwei Song(2015)et al This paper details 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 micro structural 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.[4]

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. EDX and 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.[4]

Reza MasoudiNejad(2015)et al, says Accurate prediction of fatigue crack growth on railway wheels and the influence of residual stresses by finite element method (FEM) modeling can affect the maintenance planning. Therefore, investigation of rolling contact fatigue and its effect on rolling members life seem necessary. The objective of this paper is to provide a prediction of rolling contact fatigue crack growth in the rail wheel under the influence of stress field from mechanical loads and heat treatment process of a railway wheel. A 3D nonlinear stress analysis model has been applied to estimate stress fields of the railway mono-block wheel in heat treatment process. Finite element analysis model is presented applying the elastic–plastic finite element analysis for the rail wheel under variable thermal loads. The stress history is then used to calculate stress intensity factors (SIFs) and fatigue life of railway wheel. The effect of several parameters, vertical loads, initial crack length and friction coefficient between the wheel and rail, on the fatigue life in railway wheels is investigated using the suggested 3-D finite element model. Three-dimensional finite element analysis results obtained show good agreement with those achieved in field measurements. [5]

Three-dimensional finite element analysis for the simulation of fatigue crack growth in mono-block wheels is developed in this paper. The heat treatment of wheel and wheel/rail contact was simulated in order to estimate the stress field resulted from these statuses. The stress changes values in three directions of coordinate axis located at the wheel tread are highly important during the heat treatment. The circumferential residual stress in wheel was obtained almost equal to the yield stress of the material. According to the results of the finite element analysis for the mechanical residual stresses resulted from the service condition, the maximum stress of Von Mises equal to 604 MPa was obtained. The obtained results for the fatigue crack growth have been obtained based on applying the numerical method and using FRANC3D software. The parametric analysis of the fracture in this research has been done using the linear fracture mechanics. The critical crack was crack 1 and its critical length was calculated as 37 mm which the geometrical limits of the wheel have not let reach this length practically and before reaching the critical length, the wheel gets the fracture. In addition, the fatigue crack growth analysis in the railway shows the crack growth mode as the shear combined modes II and III without residual stress. That is, the mode I is compressive and will not have any effects on the crack growth. Whereas in presence of residual stresses, the stress intensity factor of mode I is in tension and have a significant effect on the crack growth. Also, the length of the initial crack has a great effect on the fatigue life so that decreasing the initial length, increases the fatigue life. Regarding this issue that the initial crack is generally resulted from the manufacturing processes and its growth, the fatigue life in railway wheels can be remarkably improved by being more careful and with more controlling in these two fields. Finally, it can be stated that the high rate of railway system expenses decreasing requires the optimization of holding and repairing related to the wheels.[5]

Gang Fang (2015),et al., investigated 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.[6]

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. [6]

Zhanbiao Li (2014) e al, 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 modelling 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 travelling speed and an 82_ wheel angle, and (2) application of a 26,900 kg static load followed by an 8.05 km/h travelling speed and an 82_ wheel angle. The results revealed that travelling 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.[7] 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 two-piece 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 manoeuvre conditions – static loading followed by rotation travelling and steering of the rims. The two simulated Conditions encompassed the rim and tire assemblies travelling at speeds of 8.05 km/h with 26,900 kg static load and 24.14 km/h with 23,100 kg static load, respectively. The performances of the rim components were assessed in terms of stresses, fatigue lives, and, for the two-piece rim, bolt connection forces. Based on the analyses, the following conclusions were drawn.

1. Compared to the conventional five-piece rim, the proposed two-piece rim decreased the maximum von Mises stresses by over 56.4% and 52.4%, the maximum 1st principal stresses by over 30.8% and 45.1%, respectively for the 8.05 km/h and 24.14 km/h conditions. Consequently, the fatigue lives were increased by over two orders of magnitude for the two-piece rim.

2. The bolt forces were assessed from the beam elements in the FE model of the two-piece rim. The shear forces (transverse direction) were found to be negligible. The maximum axial forces were 111,360 N and 195,680 N, for the 8.05 km/h and 24.14 km/h conditions, respectively.

E. R. Weishaupt(2014) et al.,says The right-rear wheel of a full-size pickup truck involved in a single-vehicle accident was alleged to have fractured, leading to loss of vehicle control. The wheel was found fractured and partially separated from the vehicle by a distance of approximately 200 feet from the final resting point of the vehicle. The wheel was manufactured using a common casting procedure utilizing aluminum alloy A356 in the T6 condition. In subsequent litigation, the presence of porosity in the wheel was alleged to have precipitated the failure. Radiographic inspection of the wheel displayed the presence of shrinkage porosity in a quantity less than the maximum allowed according to the part specification. The recovered portions of the right-rear wheel exhibited evidence of impact overload fracture, including significant deformation. Inspection of the remaining artifacts also revealed that the right-front wheel exhibited a similar fracture as the rear.[8]

The right-rear wheel met the manufacturer's specifications for porosity, which were based on a system of grading the casting by comparing radiographs of the wheel to a series of standard radiographs from ASTM E155. The subject right-rear, as well as the companion right-front wheel failed as the result of a one-time impact loading event. Evaluation of the accident scene revealed several objects in the vicinity that could have provided the type of loading required to damage the right-front and rear wheels in such similar manners. The simple observation of porosity on the fracture surface of the cast aluminum wheel is not indicative of a manufacturing defect, rather, it is an expected occurrence and many tools are available to the failure analyst to determine the effect of the porosity on the performance of the part. Using the tools discussed in this paper, the failure analyst can arrive at the correct determination that the wheel did not unexpectedly and catastrophically fail, terrain or other hazards damaged the front and rear wheels. [8]

Zhan-GuangZheng (2014) et al., proposed a computational methodology 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. [9]

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. The following conclusions are drawn based on all study results above:

1. The fatigue failure critical locations are estimated, and the nodal points on the spoke salient are identified as the critical locations, which agree well with the actual crack locations.

2. According to stress analysis of the key locations based on the critical plane theory, two principle stresses are not proportional and unstable principle planes are changing with loading direction. Principle planes variation changes a little, varying from _40_ to 30_, and the stress states of automotive steel wheel are in biaxial tensile and compression stresses during dynamic cornering fatigue test.

ZhanbiaoLi(2014) et al., studies a five-piece rim and a two-piece bolt-connected rim d 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 modelling 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 travelling speed and an 82_ wheel angle, and (2) application of a 26,900 kg static load followed by an 8.05 km/h travelling speed and an 82_ wheel angle. The results revealed that travelling 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.[10]

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 two-piece rim, sized 33–13.00/2.5, compatible with the tire, were modeled using previously validated approaches. Using the validated FE models, the rims werenumerically investigated under severe loading and manoeuvre conditions – static loading followed by rotation travelling and steering of the rims. The two simulatedconditions encompassed the rim and tire assemblies travelling at speeds of 8.05 km/h with 26,900 kg static load and 24.14 km/h with 23,100 kg static load, respectively. The performances of the rim components were assessed in terms of stresses, fatigue lives, and, for the two-piece rim, bolt connection forces. Based on the analyses, the following conclusions were drawn.[10]

i. Compared to the conventional five-piece rim, the proposed two-piece rim decreased the maximum von Mises stresses by over 56.4% and 52.4%, the maximum 1st principal stresses by over 30.8% and 45.1%, respectively Time (s)0.0 0.2 0.4 0.6 0.8 1.0for the 8.05 km/h and 24.14 km/h conditions. Consequently, the fatigue lives were increased by over two orders of magnitude for the two-piece rim.

ii. The bolt forces were assessed from the beam elements in the FE model of the two-piece rim. The shear forces (transverse direction) were found to be negligible. The maximum axial forces were 111,360 N and 195,680 N, for the 8.05 km/h and 24.14 km/h conditions, respectively.

iii. Although the magnitude of the static load contributes to increasing the stresses on the rim components and the bolt connecting forces, driving and steering speeds are the key factors in high stresses and high bolt connection forces. This is most notable during hard cornering of the wheels. [10]

Nagvendrakumarkanoje (2014)etal.,says during running due to frequent braking and suddenly jamming of brakes the Railway wheel skids over rails. This frequent skidding removes large amount of metal from the surface known as Wheel-flat defect. In this paper the wheel-flat and a subsurface crack in the beneath is studied using FEA. If the wheel-flat is not detected early the subsurface crack can originate in the beneath due to inclusion, may leads to fatal accidents. In this study wheel material is taken Elastic-plastic and J-Integral factor has been obtained. The wheel–rail vehicle is modelled as a mass–spring–damper system.[11]

With the use of kinematic hardening model and Elastic-Plastic material of wheel, there was more scope to study stresses around crack. The subsurface crack is prominent and can be originated any time during the life cycle. In this work the comparison of J-Integral value or Energy Release rate for different inclusion gives an idea that the crack originated due to presence of inclusion when their orientation is 20 deg to the tangent to wheel tread just below the wheel-flat surface are more prominent during the life cycle of wheel when especially wheel material is Elastic-Plastic. The presence of MnS inclusion is more like to show a consistent growth in J-Integral value as compare to Al2O3 and it can induce crack more easily in the tread of the wheel during life cycle and also helps the crack propagate the crack easily by inducing stresses. All the inclusion gives high J-Integral value for an inclined crack. The J-Integral value decreases as the crack depth increases.[11]

C. Senatore (2014) et al., presents an analysis of rigid wheels – dry sand interaction and compares experimental results with predictions from established terramechanics theory. A novel experimental setup, based on sensing elements placed on the wheel surface, allows inference of normal and tangential stress at the wheel-terrain interface. A particle image velocimetry (PIV) analysis is used to study the soil kinematics under the wheel. The analysis of stress profiles shows that stress patterns under lightweight vehicle wheels conform reasonably well to established terramechanics theory developed for heavy vehicles. For the wheel under investigation, the stress distribution had minor variation along wheel width for low slip conditions. The wheel model proposed by Wong and Reece was analyzed in light of the stress and soil kinematics measurements available. It was found that, by appropriately characterizing the model coefficients c1 and c2, and understanding the physical meaning of the shear modulus kx, it is possible to obtain torque, drawbar force, and sinkage predictions within 11% (full scale error) of experimental data.[12]

This paper presented an analysis of the interaction phenomena governing lightly loaded rigid wheel performance on dry sand. The stress analysis highlighted that stress distribution across wheel width varies significantly only for large slip. For large positive slip, stress is higher at the wheel edges, while for large negative slip the opposite is true. Comparison with the Wong and Reece model showed that this model is in theory able to characterize the mobility performance of lightweight vehicles. However, the empirical nature of method should not be forgotten. It was found that the coefficients c1 and c2 do not depend on vertical load, and therefore can be considered constant for a specific wheel-soil configuration. The analysis of soil kinematics under the wheel showed that the hypotheses behind the shear stress formulation are not entirely valid: this explains why the shear modulus kx, obtained from direct shear tests (or other shear tests), does not produce accurate results when used in the Wong and Reece model.[12]

Denghong Xiao propose a multi-objective topology optimization methodology for steel wheel, in which both the compliance and eigen frequencies are regarded as static and dynamic optimization objectives. Compromise programming method is employed to define the objectives of multi-objective and multi-stiffness topology optimizations, whereas mean-frequency formulation is adopted to settle eigen frequencies of free vibration optimization. To obtain a clear and useful topology optimization result, cyclical symmetry and manufacturing constraints are set, the influences of which on the outcomes are also discussed. With an appropriate value of the minimum member size, a rough topology optimization of the steel wheel is obtained. The optimization result is modified according to the actual structure and manufacturing process. Moreover, based on this result, eight different steel wheel modes are established to analyze the influence of the manufacturing process and draw beads on the wheel performance through finite element simulation. Simulation results are verified by conducting a stress test of a commercially available wheel. Compared with its initial design, the optimized wheel disc exhibited decreased mass at 0.15 Kg at percentage of 4.57%, manifesting the effectiveness of the proposed method.[13]

In this paper, a novel steel wheel design method based on multi-objective topology optimization is presented. The compromise programming method is applied to describe the static loaded multi-stiffness topology optimization, and the dynamicformulation is used to establish the subsequent optimization problem mainly concerning free vibration. Based on the structure of a wheel in market, the multi-objective optimization problem of a wheel disc is formulated, where the minimum compliance and the maximum fundamental eigenvalue act as the two objectives. Improved dynamic frequency characteristic and lightweight effect are achieved in the topology optimization design. The following conclusions are drawn based on the analysis of multi-objective topology optimization results:

i. Compromise programming method can be well implemented in multi-objective topology optimization considering the compliance and Eigen frequencies. The Eigen frequencies did not switch their orders in the optimization process because the mean frequency formulation is adopted to settle Eigen frequencies of free vibration optimization.
ii. Compared with the conventional topology optimization, the result of muti-objective topology optimization is much more comprehensive than that of the single load and one object.
iii. MMS is extremely important to eliminate numerical instabilities such as checkerboard and mesh dependency. When the MMS is determined by an appropriate value, the removed design material has a simple and clear element density distribution with small compliance value, but few iteration numbers.
iv. The wheel model established according to the results of topology optimization cannot be used directly. The wheel requires an original design range and edge boundaries to construct the structures which can be used in reality. [13]

Jung-Won Seo (2013) et al.,studies the contact fatigue damages on the rail surface, such as head check, squats are one of the growing problems. Fracture of rail can be prevented by removing the crack before it reaches the critical length. Therefore, the crack growth rate needs to be estimated precisely according to the conditions of the track and load. In this study, we have investigated the crack growth behavior on rail surface by using the twin-disc tests and the finite element analysis. We have verified the relationship between the crack growth rate and the variety of parameters as cracks grow from the initiation stage. [14]

Contact fatigue test using specimens and finite element analysis were conducted in order to evaluate the fatigue crack growth mechanism occurring on a rail according to lubricating condition and the following conclusion was obtained:

Under the condition of no lubrication and when slip ratio is 1%, cracks initiated in most specimens after 5 _ 105 cycle. However, after cracks grew up to a certain length, the crack did not propagate due to wear.

i. Under the condition of lubrication, the crack occurred on most contact surfaces. The cracks continuously grew due to the contact pressure. In addition, crack growth rate was accelerated by the effect of hydrostatic pressure. Finally, the shelling occurred where a part of contact surface fell off.
ii. The surface crack growth rate increased as the crack length increased. However, the crack growth rate decreased when the crack length exceeded a certain level. gate shape of Type 3 showed reduced shear rate by about 11%.[14]

AlexandruValentinRADULESCU(2012) et al.,says Wheels have vital importance for the safety of the vehicle and special care is needed in order to ensure their durability. The development of the vehicle industry has strongly influenced the design, material selection and manufacturing processes of wheels. The wheels loading manner is a complex one; further improvement and efficient wheel design will be possible only if the loading will be better understood. In this paper, the car rim is analyzed with finite element method, using the 400 loading test. 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 with those obtained by using an experimental stand. Finally, the Wöhler curve for the car rim is obtained.[15]

The theoretical model, realized with the finite element method, demonstrates the existence of two zones with high stresses, disposed in the central area of the disk. These stresses are responsible for the fatigue breaks of the rim. The experimental results confirm the existence of these zones, where the fissures appear. Following optimization of the car rim, we managed to reduce supplementary costs, eliminating the hub of stresses and increasing the reliability of the rim.[15]

2.1 Part Design using CATIA

We Created sketched features including, cuts, and slots made by either, extruding, revolving sweeping along a 2-D sketched trajectory, or blending between parallel sections, create “pick and place” features, such as holes, shafts, chamfer, rounds, shell, regular drafts, flanges ribs etc. We also sketched cosmetic features, reference datum planes, axes, points, curves, coordinate systems, and shapes for creating non solid reference datum, modify, delete, suppress, redefine, and reorder features. Created geometric tolerances and surface finished on models, assign defines, and units, material properties or user specified mass properties to a model..

2.2 Static and fatigue analysis

The present work deals with estimating the fatigue life of aluminum alloy wheel by conducting the tests under radial fatigue load and comparison of the same with that of finite element analysis. Fatigue life prediction using the stress approach is mostly based on local stress, because it is not possible to determine nominal stress for the individual critical areas. The necessary material data for fatigue life prediction with the stress concept is the well-known S–N curve. Therefore, S–N curves are required for each specimen which reflects the stress condition in the critical area of the component. In the fatigue life evaluation of aluminum wheel design, the commonly accepted procedure for passenger car wheel manufacturing is to pass two durability tests, namely the radial fatigue test and cornering fatigue test. Since alloy wheels are designed for variation in style and have more complex shapes than regular steel wheels, it is difficult to assess fatigue life by using analytical methods. In general, the newly designed wheel is tested in laboratory for its life through an accelerated fatigue test before the actual production starts. Based on these test results the wheel design is further modified for high strength and less weight, if required. 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 a alloy wheel.

2.3 Wheel meshing

When the wheel is meshed, in estimated data change gradient big spot, it needs to adopt more intensive grid to better reflect the changes of data. In the wheel hub, the danger zones are rim, junction with rim and rib, and the area around bolt hole. The stress concentration region corresponding grid distribution should be dense; but the rim the stress cannot consider nearly in the entire parsing process, the corresponding grid distribution should be sparse

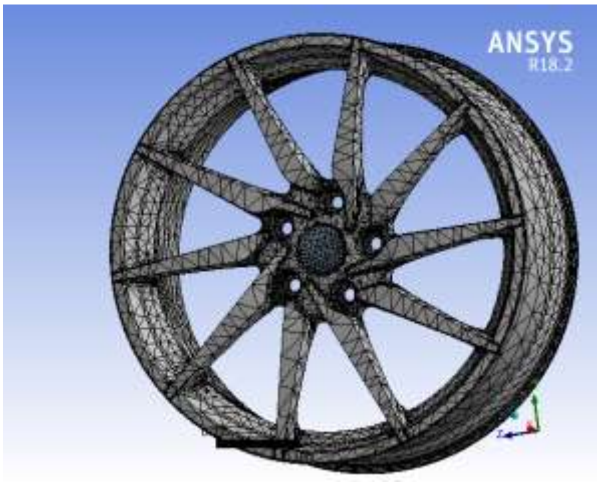

Fig 2 Meshing Desing1

Fig 3 Meshing Desing 2

3. CONCLUSIONS

The research papers studied focuses on Finite Element Analysis of Wheel Rim. Along with FEA the simulation of the dynamic cornering fatigue test, Radial fatigue test and biaxial fatigue tests of the automotive wheels. Also experimentation of the entire wheel rim test is also studied in those works

i. Investigated to examine stress levels and fatigue lives on critical regions of wheels which are made from different materials by simulating static analysis with cyclic loads is found to converge with experimental results.
i. A computational methodology is proposed for fatigue damage assessment of metallic automotive components and its application is presented with numerical simulations of wheel various fatigue tests.
ii. Use of 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 the durability performance of wheel evaluated by radial fatigue test and cornering fatigue tests.
iii. The study reveals the fatigue failure mechanism of the fractured steel wheel after the fatigue test.
iv. Designing for Alloy wheel used in four wheeler by collecting data from reverse engineering process from existing model and evaluating by analyzing the model by taking the constraints as ultimate stresses and variables as two different alloy materials and different loads and goals as maximum outer diameter of the wheel and fitting accessories
v. Study of fatigue lifetime prediction method of alloy wheels was proposed to ensure their durability at the initial design stage.
vi. The premature failure of a truck steel wheel prototype that occurs during the course of radial fatigue tests is studied using finite element analysis. Finite element-based stress analysis showed that the crack initiation regions on the wheel disc are subjected to stress concentration.

4. REFERENCES

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

[2]. D'Andrea, A., &Tozzo, C. (2016). Interface stress state in the most common shear tests. Construction and Building Materials, 107, 341-355.

[3]. Irastorza-Landa, A., Van Swygenhoven, H., Van Petegem, S., Grilli, N., Bollhalder, A., Brandstetter, S., &Grolimund, D. (2016). Following dislocation patterning during fatigue. ActaMaterialia, 112, 184-193.

[4]. Song, W., Woods, J. L., Davis, R. T., Offutt, J. K., Bellis, E. P., Handler, E. S., ... & Stone, T. W. (2015). Failure analysis and simulation evaluation of an AL 6061 alloy wheel hub. Journal of Failure Analysis and Prevention, 15(4), 521-533.

[5]. Nejad, R. M., Farhangdoost, K., &Shariati, M. (2015). Numerical study on fatigue crack growth in railway wheels under the influence of residual stresses. Engineering Failure Analysis, 52, 75-89.

[6]. Fang, G., Gao, W. R., & Zhang, X. G. (2015). Finite element simulation and experiment verification of rolling forming for the truck wheel rim. International Journal of Precision Engineering and Manufacturing, 16(7), 1509-1515.

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

[8]. Weishaupt, E. R., Stevenson, M. E., & Sprague, J. K. (2014). Overload Fracture of Cast Aluminum Wheel. Journal of Failure Analysis and Prevention, 14(6), 702-706.

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

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

[11]. Sharma, S. C., &Harsha, S. P. (2014). EPFM Analysis of Subsurface Crack Beneath a Wheel Flat Using Dynamic Condition. Procedia Materials Science, 6, 43-60.

[12]. Senatore, C., &Iagnemma, K. (2014). Analysis of stress distributions under lightweight wheeled vehicles. Journal of Terramechanics, 51, 1-17.

[13]. Xiao, D., Zhang, H., Liu, X., He, T., & Shan, Y. (2014). Novel steel wheel design based on multi-objective topology optimization. Journal of Mechanical Science and Technology, 28(3), 1007-1016.

[14]. Seo, J. W., Kwon, S. J., Lee, D. H., & Choi, H. Y. (2013). Analysis of contact fatigue crack growth using twin-disc tests and numerical evaluations. International Journal of Fatigue, 55, 54-63.

[15]. Radulescu, A. V., Cananau, S., &Radulescu, I. (2012). MECHANICAL Testing Methods Concerning The Stress Analysis For A Vehicle Wheel Rim. In The 3rd International Conference On Diagnosis And Prediction In Mechanical Engineering Systems Dipre.

[16]. Topac, M. M., Ercan, S., &Kuralay, N. S. (2012). Fatigue life prediction of a heavy vehicle steel wheel under radial loads by using finite element analysis. Engineering Failure Analysis, 20, 67-79.