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

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

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

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

ABSTRACT

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. All experiments were considered at First Test, then proper Finite Element Analysis is done. Then Experimentation for the same test is done and compared. In this way a filter is applied to extensive Experimentation. For the safe combinations we carried DCFT with FEA as well as Experimentation.

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 Objective

- **i.** 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.
- **ii.** 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.
- **iii.** Stress distribution may not be that high on some other parts, hence, excess can be removed from these regions to prevent material extravagance.
- **iv.** 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,
- **v.** Prediction of stresses in to rim under dynamic conditions using FEA,
- **vi.** 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.
- **vii.** Study Failures in Wheel Rims
- **viii.** Estimate Life Cycle period of Rim

.

1.2 Cornering fatigue test experimentation

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.

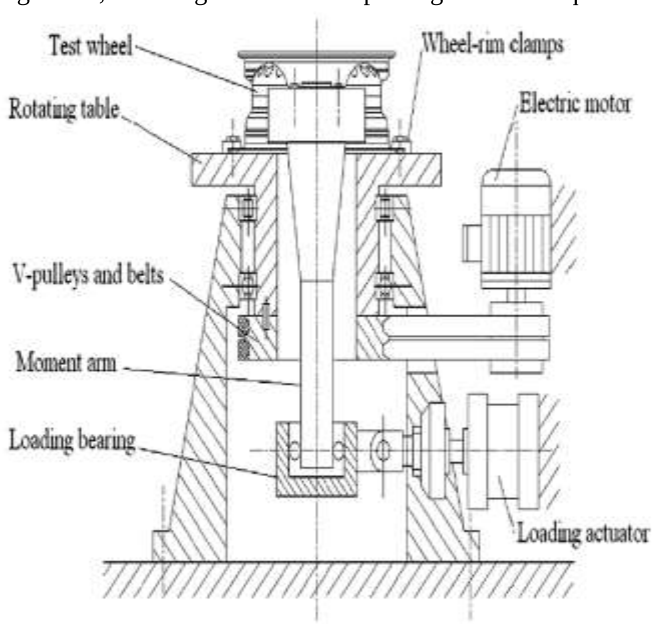

Fig 1 Sketch of the dynamic cornering fatigue test system

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)

2. FATIGUE ANALYSIS USING FEA

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. [7]

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 multiaxial 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. [7]

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.[7]

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

Fig 4 Fixed Support Design 1

Fig 5 Moment applied case 1 design 1

Combining all parameters Experiment Plan is designed as follows:

Table 1 Parameters

Sr.No.	Design No.	Material	Moment Applied	Condition
1	D1	M1	2095 Nm	C1
2	D2	M1	2095 Nm	C1
3	D1	M2	2095 Nm	C1
4	D2	M2	2095 Nm	C1
5	D1	M1	2095 Nm	C2
6	D2	M1	2095 Nm	C2
7	D1	M2	2095 Nm	C2
8	D2	M2	2095 Nm	C2

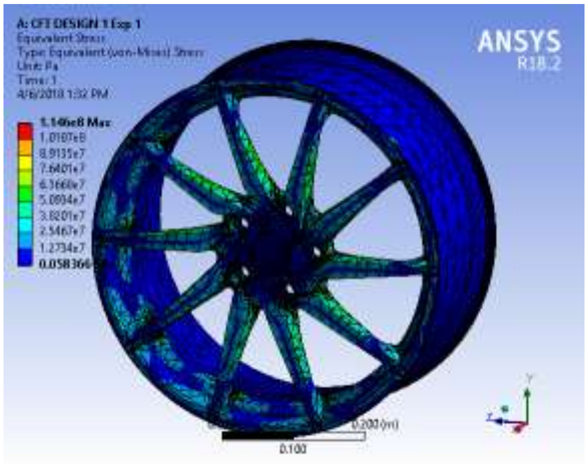

Fig 6. Equivalent stress experiment no 1

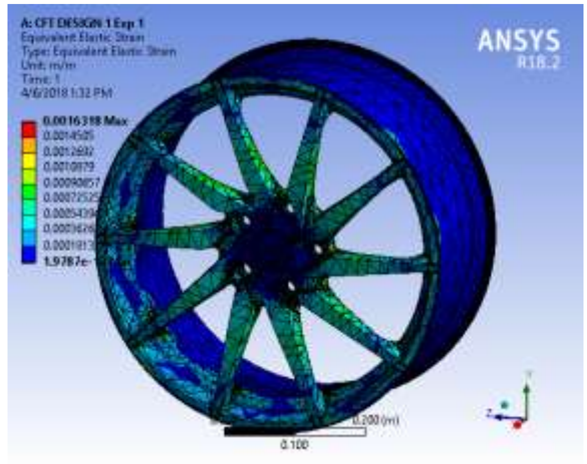

Fig 7. Equivalent strain experiment no 1

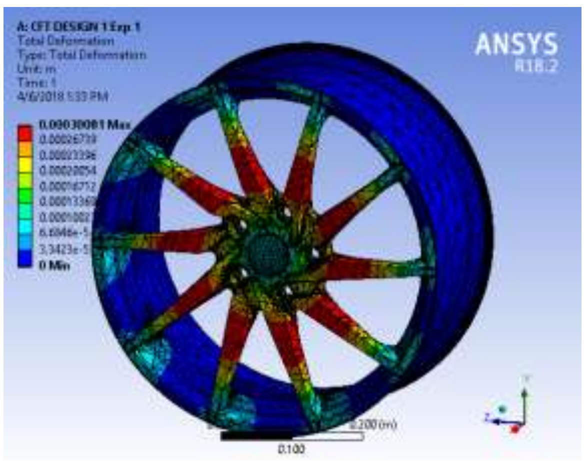

Fig 8Total deformation experiment no 1

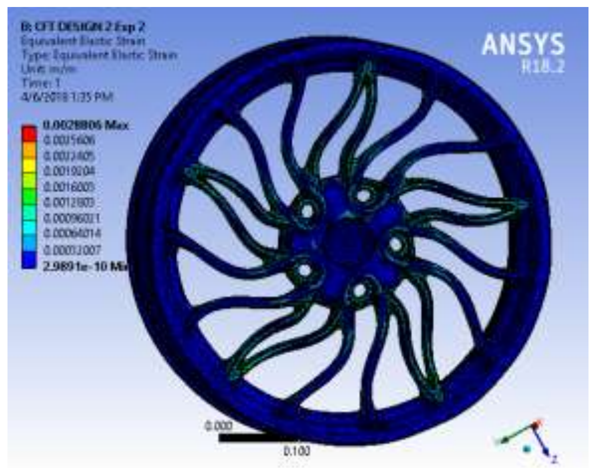

Fig 9 Equivalent stress experiment no 2

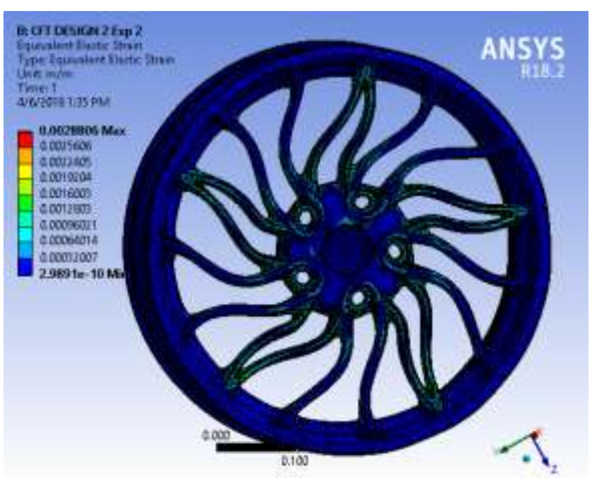

Fig 10 Equivalent strain experiment no 2

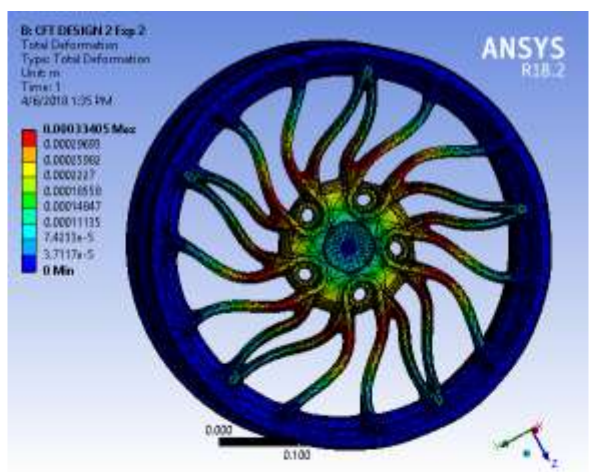

Fig 11. Total deformation experiment no 2

3. RESULT

Based on this type of FEM analysis, one can decide as to which parts are critical, then it can strengthen at those zones. On the other hand, stress distribution may not be that high on some other parts, hence, excess can be removed from these regions to prevent material extravagance. Furthermore, 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 considered following case of dynamic load or Moment applied on mounting holes and disc. During experimentation of strain gauges were used to know the strain developed at various positions. These strain gauge readings were compared with FEA results of both cases. The scheme with minimum error will be considered for study of stresses developed in rim and future failure analysis

Table 2 FEA Result

Expt.No.	Equivalent Stress(X 10^8 Pa)	Equivalent Strain	Total Deformation(m)	Remark
1	1.146	0.00163	0.0003000	SAFE
2	2.034	0.00288	0.000334	SAFE
3	1.143	0.00256	0.000474	SAFE
4	2.024	0.00452	0.000527	**FAIL**
5	1.167	0.00163	0.000301	SAFE
6	2.027	0.00286	0.000334	SAFE
7	1.165	0.00261	0.000475	SAFE
8	2.015	0.00450	0.000527	**FAIL**

From above analysis it is clear that Experiment no.4 have Maximum equivalent stress 2.024X10^8 also Experiment No.8 have Maximum equivalent stress 2.015 X10^8 which is greater than Tensile Strength of Magnesium 1.93X10^8 so these two combination Fails under given boundary conditions.

Table 3 Experimental Results for DCFT Experiment with FEA Results

Strain Gauge Position	Strain Values (X10^{-4})					FEA RESULTS	Difference
	Sample 1	Sample 2	Sample 3	Sample 4	Average		%
1	1.9	1.8	1.9	1.8	1.85	1.93	4.14
2	0.25	0.26	0.26	0.27	0.26	0.28	7.14
3	8.9	8.8	8.9	8.7	8.825	9.04	2.37

The results from the steel Wheel Rim dynamic cornering fatigue test of Case 2 showed that the baseline Wheel Rim failed the test and its crack initiation was around the hub bolt hole area that agreed with the simulation. The variation in FEA and actual Experimentation is below 10 %, which validates the CFT test. So it indicated that the moment is applied on hub also.

4. CONCLUSIONS

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. All experiments were considered at First Test, then proper Finite Element Analysis is done. Then Experimentation for the same test is done and compared. In this way a filter is applied to extensive Experimentation. For the safe combinations we carried DCFT with FEA as well as Experimentation. Here we got the final optimize result. Experimental results were compared to finite element results for validating the methods adopted. The experimental results and the modifications and identification of the proper methods for applying the moment on rim.

5. REFERENCES (Font-11, Bold)

[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.