IJARSCT

ISSN (Online) 2581-9429

International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)

IJARSCT
Impact Factor: 6.252

Volume 2, Issue 2, February 2022

Weight Reduction of Steel Wheels by Replacing with Alloy Materials using Finite Element Analysis Technique

Mr. Kunal Sharad Tolambe, Mr. Navinkumar Ravindra Bele,
Mr. Abhishek Sunil Vyawhare, Mr. Akash Shivram Borade
Department of Mechanical Engineering
Bramha Valley College of Engineering & Research Institute, Nashik, Maharashtra, India
tolambekunal123@gmail.com, belenavinkumar7605@gmail.com,
abhishekvyawhare77@gmail.com, akashborade85@gmail.com

Abstract: *Wheel Rim must be robust sufficient to maintain load and additives with lesser weight and fee. weight loss is the main component in automobile industries, because if weight of car will increase the gas consumption as well as the fee required to run vehicle also increases. preserving this component in mind Wheel Rim ought to be layout with excessive energy through studying different factors like pressure stress values, deformation and so forth. Optimizing the Wheel Rim with the aid of varying parameters, no of spokes gift, geometry of spokes within the Wheel Rim for modelling and evaluation. This venture involves design, analysis and optimization of Wheel rim with constraints of equal strain and deflection of wheel rim underneath most load. Automotive businesses are paying their most important hobby inside the weight reduction of components to reduce gasoline fee. This weight may be decreased by means of introducing new substances and production techniques with optimization of layout. Minimizing the weight within the wheel is extra powerful than minimizing the burden in different components due to its rotational second of inertia effect during its movement and also the tyre take the general car load and offers cushioning effect. via lowering the weight, we are able to achieve the objective the lowering of unsprung mass, by way of which the inertia masses and average weight are decreased with development of overall performance and fuel economic system. on this paper an strive is made to decrease the burden of the wheel by way of changing the aluminum alloy with composites. From the finite detail calculations, it's miles found that the mass of the wheel rim may be reduced to 50% from the prevailing alloy wheels. The evaluation also suggests that once the optimization the stresses generated from the wheel rim can be beneath the yield stress. This gave a new approach within the field of optimization of passenger automobile wheel rim. in this work the modelling is performed through the usage of CATIA and analysis is made by way of using ANSYS..*

Keywords: Automobile Rim, Dynamic cornering Fatigue test, FEA software.

I. Introduction

Wheel makers are using new materials and manufacturing techniques to improve the aesthetic appearance and design of their wheels. Steel wheels are used for excellent properties such as B. Light weight, excellent forging ability, high wear resistance, mechanical strength. Commonly used for wheels. Ensuring the reliability and safety of the wheel is very important. [1]

The rim analysis consists of a numerical analysis of the stress level that the rim receives during use. The load capacity of the bolt circle is the rating under high load conditions. The finite element method (FE) is implemented in all rim analyzes. The reliability of the FEM approach is based on previous experience in fatigue analysis. The magnitude of static load and static pressure contributes to the increased stress on the rim component. [2]

II. Objective

The main goal of this task is to replace heavy steel wheels with alloy wheels. For this, we use the FEA method. Two designs and two materials are available. Aluminum alloy and magnesium alloy. The wheel must pass DCFT.

III. Methodology

Before starting the analysis, it is important to understand the current process. A literature review was conducted to understand the previous work done in the field stress analysis of the wheel rims. The following methods were used to validate the solution.

1. Experimental Method
2. Finite Element Method
3. Validation

According to standards Rim should pass following tests:

1. Dynamic Cornering Fatigue Test.

IV. Parameters

4.1 Number of Designs

1. **Design No.1-D01**
2. **Design No.2-D02**

4.2 Maximum Bending Moment Bending Moment Calculation-

The bending moment is calculated as follows:

$$B.M = F_R d + F_L R$$
$$= F_R d + \mu F_R R$$

Where F_R = Radial force acting on the wheel=1150 Kg

d=wheel offset=0.252

μ=Coefficient of friction between ground and tire=0.7

R=Maximum allowable Radius of statically loaded tire mounted on the wheel=0.250 m

B.M=488.75 Kgm

= 4795.64 Nm = **4800 Nm**

4.3 Materials

1. Aluminium Alloy-M1
2. Magnesium Alloy-M2

4.4 Conditions of Moments applied

1. Moment Applied on Hub Bolt Region- C1
2. Moment Applied on Complete Hub - C2

4.5 Experimentation Plan

Combining all parameters Experiment Plan is designed as follows:

For Each level of experiment 4 samples will be tested. Therefore total samples checked will be = No. of experiments X Samples = 8X4=32 samples to be tested and Moment applied will be 4800 Nm.

Sr. No.	Design No.	Material	Condition
1	D1	M1	C1
2	D2	M1	C1
3	D1	M2	C1

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Impact Factor: 6.252
Volume 2, Issue 2, February 2022

4	D2	M2	C1
5	D1	M1	C2
6	D2	M1	C2
7	D1	M2	C2
8	D2	M2	C2

Experimentation Plan

V. CORNERING FATIGUE TEST

For the purpose of validation the physical known as cornering fatigue test is carried out and discussed below.

5.1 Cornering Fatigue Test Experimentation

The dynamic cornering fatigue test is a standard SAE test that simulates the load of wheel cornering. Figure 5.1 shows a test system in which the test wheel is mounted on a turntable, the lever arm is bolted to the outside of the wheel, and a constant force is applied to the tip of the lever arm bearing via the load actuator. , Is given to give the wheel a constant rotational bending moment. If the wheel passes the dynamic cornering fatigue test, it is more likely to pass all other required durability tests.

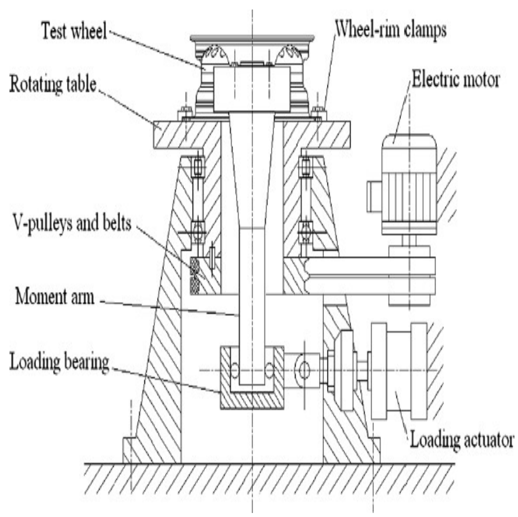

Figure 5.1: Sketch of the dynamic cornering fatigue test system

VI. FINITE ELEMENT ANALYSIS OF WHEEL RIM

Sample Result Plots: Experiment No.1

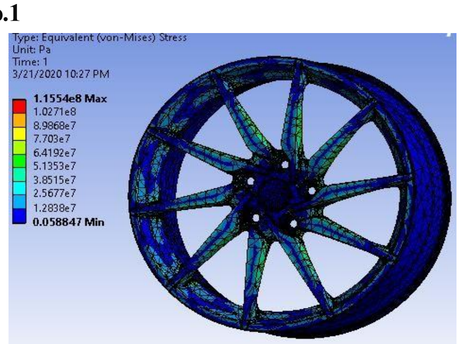

Figure 6.1: Equivalent Stress

DOI: 10.48175/IJARSCT-2753

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Impact Factor: 6.252
Volume 2, Issue 2, February 2022

Expt. No	Design No.	Material	Mass (Kg.)	Case	
1	D1	Al Alloy	9.84	C1	
2	D2	Al Alloy	9.30	C1	
3	D1	Mg Alloy	6.39	C1	OPTIMIZE LEVEL
5	D1	Al Alloy	9.84	C2	
6	D2	Al Alloy	9.30	C2	
7	D1	Mg Alloy	6.39	C2	OPTIMIZE LEVEL

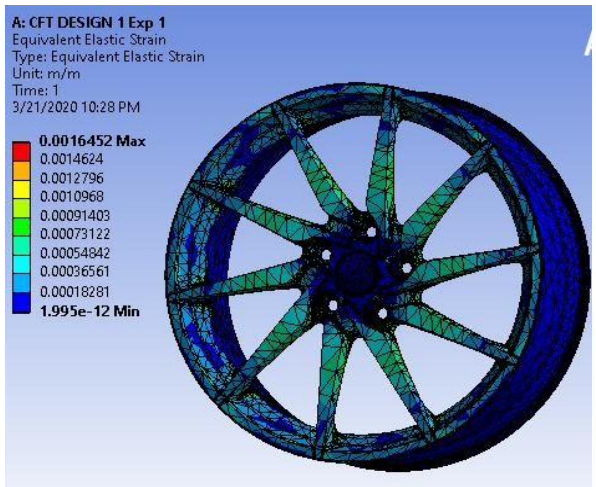

Figure 6.2: Equivalent Strain

Figure 6.3: Total Deformation

VII. RESULTS FROM FEA PLOTS

Expt	Eq. Stress (X 10^8 Pa)	Eq. Strain	Total Def. (m)	Remark
1	1.55	0.00164	0.000303	SAFE
2	2.06	0.00291	0.000338	SAFE
3	1.157	0.00260	0.000480	SAFE
4	2.049	0.00457	0.000533	**FAIL**
5	1.182	0.00168	0.000304	SAFE
6	2.049	0.00290	0.000300	SAFE
7	1.180	0.00265	0.000480	SAFE
8	2.040	0.00455	0.000534	**FAIL**

From above analysis it is clear that Experiment no.4 have Maximum equivalent stress 2.049 X10^8 also Experiment No.8 have Maximum equivalent stress 2.040 X10^8 which is greater than Tensile Strength of Magnesium 1.93X10^8 so these two combination Fails under given boundary conditions

VIII. CONCLUSION

The concept of multipurpose analysis is performed to optimize the weight of the rim. It also determines if torque is being applied to the mounting holes or to the hub. The work is done in stages. We found this:

1. Design 1 is suitable for the vehicle considered
2. Design 2 Have some issues related to strength and moment carrying capacity
3. Magnesium alloy is suitable with Design 1
4. It weighs only 6.39 Kgs.
5. Earlier rim weighs was 15.3 Kgs.
6. So reduction in mass is 15.3-6.39 =8.91 Kg.
7. Also we found that the moment is applied on bolting area as well as Hub area.

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. Acta Materialia, 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.