ISSN (ONLINE):2456-5717
International Journal of Advanced Research in Basic Engineering Sciences and Technology (IJARBEST)
Vol.4, Issue.4, April 2018

Design Modification And Optimization of Vehicle Rim using Finite Element Analysis – A Pot Hole Approach

[1]R.Masilamani, [2]N.V.Dhandapani, [3]R.Ratheesh Kumar, [4]Nived Devadas, [5]M.Elangovan
[1]Asst. Professor, [2]Professor, [2, 3, 4] UG Scholar
[1, 2,3,4,5] Department of Mechanical Engineering, Karpagam college of Engineering, Coimbatore, India

Abstract— **This work involves the design optimization of vehicle rim using Finite element analysis. In recent trend of design is moving towards maximum weight reduction of product without compromising its performance and strength, in the form of optimization. The 3-dimensional model of the alloy wheel was designed by using the reverse engineering technology. Existing alloy wheel dimensions are taken into CAD model and the part is completed with Solid works 2015. The CAD model is imported to ANSYS 15.0 version and finite element analysis steps were carried out like 1. Pre-processing 2.Solution and 3. Post-processing. In this work static analysis of wheel rim made with materials like steel alloy, aluminum alloy, forged steel and magnesium alloy is done. All the conditions were taken with respect to pot hole condition of the road Also validate the results experimental results. Alloy wheel are mostly made of aluminum alloy which can be sand or pressure die casted readily. This review work proposes a new alloy wheel design to optimize the weight with the help of finite element analysis.**

Keywords—optimization;reduction;Post-processing;alloy.

I. INTRODUCTION

Different vehicle rim materials have different effects on the strength, stability and life of the wheel rim. Similarly even changing the material properties of the vehicle rim wheel by a few inches can drastically affect the properties of the wheel rim. Materials used to produce these rims have a design and materials can range from steel to nonferrous alloys like magnesium and aluminum. Rims affect the braking performance of a vehicle as result of the following for parameters: size, weight, design or ventilation, materials. All wheels rims to be made up of two pressed components, the rim and the wheel disc, which are joined (welded) together.

II. LITERATURE REVIEW

Aluminum alloy wheels will not fail during service. The strength of the rim and fatigue life are critical. In order to reduce the cost, design for light weight and limited life is increasingly used for all vehicle components. In the actual product development rotary fatigues test is being used to detect the strength life of the wheel. Therefore, reliable design and a proper test procedure is required to guarantee the service strength under operational conditions and functioning of the wheel.

Design in an important industrial activity which in turn influences the quality of the product. The wheel rim is designed by using softwares such as SolidWorks v2012. In modeling the time spend in producing the complex 3D model and the risk involved is in design and manufacturing processes can be easily minimized. So, the modeling of the wheel

rim is made by using the software named SolidWorks. Later, this SolidWorks model is imported to ANSYS for analyzing the work. ANSYS software is one of the latest which is used for simulating the different forces, pressure acting on the component and also for calculating and viewing the entire results. A Solver mode in ANSYS software calculates the following such as stresses, deflections, bending moments and their relations without manual intervention, reduces the time compared with the method of mathematical calculation by a human.

To create simulations of various number of spokes on the alloy wheel designs that focus on reducing the mass of the currently designed one and for selecting better material. The new designs are reducing on the weight of spokes and modifying circular intersection at the intersection point of the spoke and the hub of the rim. The main objective of our project is to reduce the weight of the wheel and also to reduce the stress without exceeding the allowable stress on the specified material.

Aluminum is the metal with features such as excellent lightness, thermal conductivity, and corrosion resistance, characteristics of casting, low temperature, machining processes and recycling. This metals advantage is that it reduces the weight, having high accuracy and design choices of the wheel. The metal is being used for energy conservation because it is possible to recycle aluminum easily.

Solid works is the software which is used for creation and modification of the objects. In this software, the design and modeling feature is available. Design means it is the processes of creating a new object or modifying the existing one. Drafting means it is the representation or idea of the object. The function of the modeling is to create and convert 2D to 3D. By using this software, we can create the model of the wheel rim.

Static analysis shows the effect of steady loading conditions on a structure, while ignoring inertia and damping effects such as those caused by varying the time-varying loads. A static analysis is the one which includes steady inertia loads (such as gravitational and rotational velocities) and time varying loads that can be approximately as static equivalent loads.

In recent years, competition in the automobile market is increased with respect to fuel economy especially for the light commercial vehicles. Moreover there is a significant necessity about reduction of fuel consumption level in automobile companies. The weight of a vehicle is one of the most important factors which affects the fuel economy.

The weight minimization of wheel has more effective than the weight minimization of elsewhere in the vehicle due to the rotation movement of inertia effect during motion. Therefore, the wheel designs should be optimized by considering attributes of a light commercial vehicle such as NVH.

In this study the model correlation between CAE stimulations and related test is performed. For this purpose, mode shape and their natural frequencies are obtained from CAE simulations are compared with experimental model analysis results. After the correlation is provided, wheel design optimization proposals are given by considering the above stated.

During finite element modeling process ANSYS is used as a preprocessor. Finite element model of a wheel is composed using 34267 tetra finite elements. Moreover second order tetra finite elements are used in order to represent all possible mode shapes. Model analysis is going to be performed on a free-free boundary condition.

In order to verify the attained finite element model, experimental model analysis has been performed on free-free boundary condition. In this boundary condition, it has been attained by using elastic string. Testing has been performed using impact hammer and a three

axial accelerometer. LMS software has been used during experimental model analysis. LMS uses polymax method on determining the mode shape and natural frequencies.
In order to all mode shapes, 100 points has been determined and these points are defined to LMS program. Using these points, the 100 FRF has been obtained from structure. Free-free boundary condition and the determined 100 points can be calculated.

Optimization is done to reduce material consumption hence to reduce weight of the wheels. Hence the loading conditions were considered based on automobile weight applied over it. Each wheel in the automobile will carry the load by distributing among them. This load is considered to be along the radial direction and applied it in optimizing the model for mass.

Optimization of the wheel rim is done through Hypermesh – optimization solver. Optimization is carried out taking special reference to the minimum material requirement to sustain the stresses applied on the wheel during operation.

III. PROBLEM STATEMENT

The failure of vehicle rim wheel with respect to pot hole condition when it tends crack initiate near the hole which further gets propagated throughout the rim which leads to structural failure. In order to improve the structural life of rim, design optimization and material modification is necessary for which best performance to be produced by conducting design of experiments to find parametric design changes which gives higher performance. Modifications in parameters Fundamental design constraints like minimum thickness based on manufacturing, dimensional limitation (wheel size).Provide maximum area at stress concentration location (spokes) etc., minimize spokes to assist die casting, to finite element analysis integrated optimization.

Fig. 1(a)

Fig. 1(b)

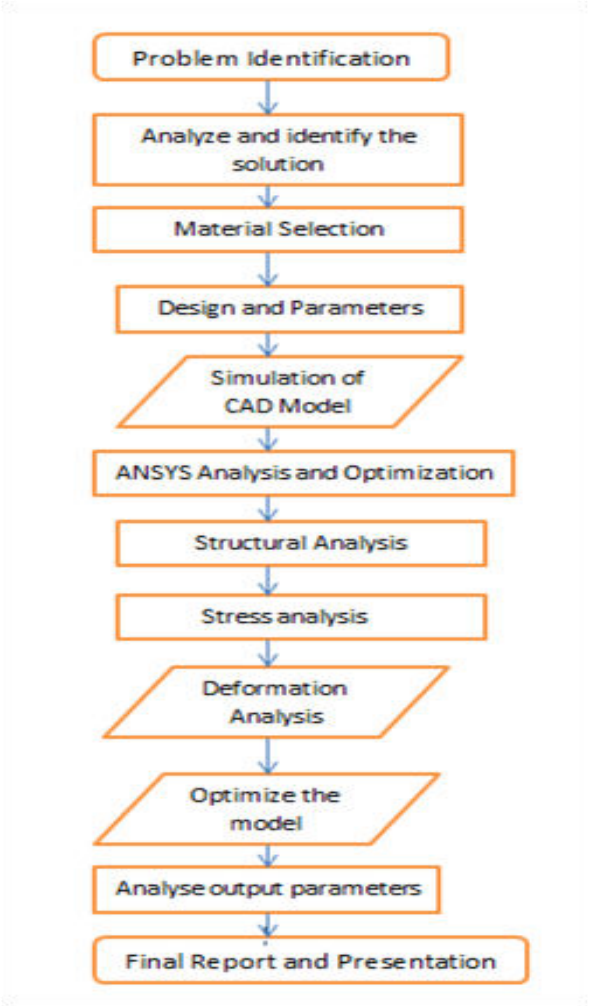

Fig. 2 Modelling

IV. MODELING

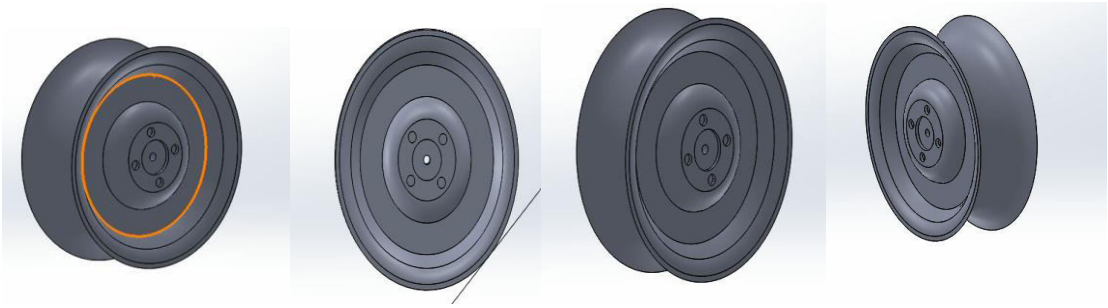

Fig. 3 Cad modeling is created using the specified dimensions

Expected result and conclusion

Design of a wheel rim which overcomes

- High torque,
- Cornering forces in wheel,
- Capable of dynamic road loads.

ACKNOWLEDGMENT

We, R.Ratheesh Kumar, Nived Devadas, M.Elangovan the members of the group working on this project and would like to take this opportunity to thank all those who have sincerely and whole heartedly lend their support and contributed in our project. We express our gratitude and thankfulness to our professors for their valuable timely guidance and wholehearted co-operation in preparing this project. We are also very much thankful to our project guide Mr.R.Masilamani, M.E., (PhD), Assistant Professor, Department of Mechanical Engineering, for his inspiring guidance with helpful suggestions for our project whenever we approached them.

REFERENCES

[1] B.Anusha Srikanta, P.Veera Raju, M.tech (PhD), (2014), 'Geometrical and Material Optimization of Alloy Wheel for Four Wheeler', International Journal of Research and Innovation.Vol.1. – Issue1.

[2] R.Masilamani,etal "A Review on Usage of Carbon Fiber Reinforced Plastics in Automobiles", International Journal of Pure and Applied Mathematics, Volume 117 No. 20 2017, 537-544

[3] C.Bosi, G.l.Garagnani, R.Tovo. (2004), 'Fatigue properties of a cast aluminium alloy for Rims of a Car Wheel', Metallurgical Science and Technology, Italy. Vol.2.Issu-2.

[4] Rick Borns, Don Whitacre (Alcoa Inc.) (2004), 'Optimizing Designs of Aluminum Suspension Components Using An Integrated Approach', SAE Paper 05M-2.

[5] Masilamani.R ,Saravanakuamr.J, Deepak.S Gowtham.K Mohammed Ansal.M, "A Review Of Vehicle Roof Structure Analysis Using CFRP" (ISBN: 978-93-84943-19-6), National Conference on "Recent Trends in Engineering, Technology and Management"

[6] P.Meghashyam, S.Girivardhan Naidu and N.Sayed Baba, (2013), 'Design and Analysis of Wheel Rim using CATIA & ANSYS', International Journal of Application or Innovation in Engineering & Management (IJAIEM).Vol.1- pp 14-20.

[7] Muhammet Cerit, (2010), 'Numerical Simulation of Dynamic Side Impact Test For an Aluminium Alloy Wheel', Scientific Research and Essays Vol. 5(18), pp. 2694-2701, 18 September, 2010.

[8] Rick Borns, Don Whitacre (Alcoa Inc.) (2004), 'Optimizing Designs of Aluminum Suspension Components Using An Integrated Approach', SAE Paper 05M-2.

[9] N.Satyanarayana & Ch.Sambaiah, (2012), 'Fatigue Analysis of Aluminum Alloy Wheel under Radial Load', International Journalof Mechanical and Industrial Engineering (IJMIE), ISSN No. 2231 – 6477,Vol-2, Issue-1, 2012.

[10] Masilamani.R, Krishnaraj.C, Sivabalaji.S "Enhancing the Passenger's Safety by Automatic Door Locking System and Self Locking of Starter Motor", International Journal of Innovative Research in Science, Engineering and Technology, ISSN 2319-8753, Volume 5, Issue 7, pp 13-17, 2016

[11] Sourav Das, (CAE Analyst), (2014), 'Design and Weight Optimization of Alloy Wheel', International Journal of Scientific and Research Publications, Volume 4, Issue 6, June 2014.

[12] Saran Theja, Shankar G, Vamsi Krishna, (2013), 'Design Analysis of Two Wheeler Lighter Weight Alloy Wheel', Indian Journal of Engineering, Volume 6, Number 15, November 2013.

[13] M. Yaman, B.Yegin,Ford Otosan, Turkey (2013), 'A Light Commercial Vehicle Wheel Design Optimization For Weight, NVH and Durability Considerations', 5th ANSA & UETA International Conference.

[14] S.Arunkumar., R.Masilamani, "Design and Analysis of Steering Mechanism Using Planetary Gear", International Journal of Applied Engineering Research, Research India Publications, ISSN 0973-4562, Volume 10, Number 19, pp 14520-14525, 2015

[15] BGN. Satya Prasad, M Anil Kumar – Hyundai (2013), 'Topology Optimization of Alloy Wheel', Simulate to Innovate, Altair Conference – 2013, India.

[16] Ravi Lidoriya, SanjayChaudhary and Anil Kumar Mohopatra, (2013), 'Design and Analysis of Aluminum Alloy Wheel using PEEK Material', International Journal of Mechanical Engineering and Research. ISSN No. 2249-0019, Volume 3, Number 5 (2013), pp. 503-516.

[17] Mi Guofa, Liu Xiangyu, Wang Kuangfei, Fu Hengzhi, (2008), 'Numerical Simulation of Low Pressure Die-Casting Aluminum Wheel', China-Foundry, Research & Development.Vol.6.No.1. PP 48-52 .