Modelling and Topology Optimization of Car Rim to Reduce the Weight of the Wheel for Maximum Stiffness Condition Using Altair Inspire

Mr. T. Dharmendra*, Mr. S. Nazeem*, Mr. K. Naveen Sai*, Mr. J. Sudheer*, Mr N. Siva Shankar*, Mr. P Ramprasad, Mrs. K Aravinda****

***UG students, **Faculty, Department of Mechanical Engineering College, Pragati Engineering College (A)**

ABSTRACT: The wheel is a main mechanical term of the vehicular suspension system that supports the static and dynamic loads encountered during vehicle action. Since cars carry heavy loads of occupants as well as self-weight, the alloy wheel rim should be strong enough to withstand this load. Thus, their design should be done very cautiously. While designing such a main kind of automotive component taking care of protection and cost are very important concerns so that users can use it safely. Major five technical considerations while modelling any new alloy wheel rim are styling, aesthetic, mass, manufacturability, and capability Alloy wheel rims are automobile wheels that are made from an alloy of aluminium or magnesium metals or sometimes a mixture of both. In this study we have performed the mass optimization of wheel rim to reduce the weight of the wheel to perform efficiently in dynamic condition. We have performed the mass optimization for maximum stiffness condition to reduce the weight of the wheel. We have performed the structural analysis before and after the optimization of rim to make sure the stress and displacements results are same for safety conditions.

Key words: Modelling, Topology, Optimization, Rim, Altair, stiffness

I. INTRODUCTION

Archaeologies and historians of today see the introduction of the wheel as the real genesis of any old civilization. The wheel is the most significant discovery of old times. The wheel has developed from an oversized bearing to a fully integral part of any modern transportation vehicle. Modern motor vehicles are produced according to very strict rules to ensure the safety of passengers.

Materials to produce these wheels have become has sophisticated as a design and material can range from steel to nonferrous alloys like magnesium and aluminium. Automotive wheels have evolved over the decades from early spoke design of wood and steel. Today's modern vehicles are using the stamped metal configuration and modern cast and forged aluminium alloys rims. Since the 1970's several innovative methods of testing well aided with experimental stress measurement have been initiated.

Within the past 10 years, durability analysis (fatigue life predication) and reliability method for dealing with variations inherent in engineering structure have been applied to the automotive wheel Braking performance shows effect on the wheel rim parameters: size, weight, design and materials. The size of the wheel rim governs how much space there is between the rim and brake rotor. If the diameter of the wheel rim is higher there will be more scope for air flow around the brakes and therefore better cooling. The weight of the wheel rim is also an important issue. The handling. A more rigid wheel will reduce wheel flex. This is essentially important with low aspect ratio, high performance tires that can be generate high cornering forces

II. LITERATURE SURVEY

G. Ashok Kumar, M. Uma Mahesh, S. Madhu Sudan, T. choli raj [1] [Dec -2016 DESIGN AND ANALYSIS OF WHEEL RIM BY USINGCATIA &ANSYS The purpose of the car wheel rim is to provide a firm base on which to fit the tyre. Its dimensions, shape should be suitable to satisfactorily accommodate the particular tyre required for the vehicle. In this study a tyre of car wheel rim belonging to the disc wheel category is considered. Design in an important industrial activity which influences the quality of the product. The wheel rim is designed by using modelling software CATIAv5R18.In modelling the time spent in producing the complex 3-D models and the risk involved in design and manufacturing process can be easily minimized. So, the modelling of the wheel rim is made by using CATIA. Later this CATIA model is imported to ANSYS for analysis work. ANSYS software is the latest software used for simulating the different forces, pressure acting on the component and also for calculating and viewing the results. A solver mode in ANSYS software calculates the stresses, deflections, bending moments and their relations without manual interventions, reduces the time compared with the method of mathematical

 DOI : 10.35291/2454-9150.2022.0259

calculations by a human. ANSYS static analysis work is carried out by considered two different materials namely aluminium and forged steel and their relative performances have been observed respectively. In addition to this rim is subjected to Vibration analysis (Modal analysis), a part of Dynamic Analysis is carried out and its performance is observed. In this work by observing the results of both static and modal analysis obtained forged steel is suggested as best material.

T. S. Prasad, J. Reddy[2] July (2020) A Review on Modelling and Analysis of Car Wheel Rim using CATIA & ANSYS The essence of car wheel rim provides a firm base on which to fit the tire. Its dimensions, shape should be suitable to adequately accommodate the particular tire required for the vehicle. In this work a tire of car wheel rim belonging to the disc wheel category is considered. Design is an important industrial activity which influences the quality of the product. The wheel rim is modelled by using modelling software catiav5r17. By using this software, the time spent in producing the complex 3D models and the risk involved in the design and manufacturing process can be easily minimized. So, the modelling of the wheel rim is made by using CATIA. Later this CATIA modal is imported to ANSYS for analysis work. ANSYS is the latest software used for simulating the different forces, pressure acting on the component and also calculating and viewing the results. By using ANSYS software reduces the time compared with the method of mathematical calculations by a human. ANSYS static analysis work is carried out by considered two different materials namely aluminium and forged steel and their relative performances have been observed respectively. In addition to wheel rim is subjected to modal analysis, a part of dynamic analysis is carried out its performance is observed. In This paper by observing the results of both static and dynamic analysis obtained forged steel is suggested as best material.

E. Chandrashekhar, J. Rishi 29 August 2016. This present thesis gives a brief compilation of research related to analysis of deformation, equivalent stress, fatigue life and natural frequency for lighter alloy and composite automobile Wheel rim. The design of wheel rim for car application is completed paying extraordinary enthusiasm for the optimization of the weight of the wheel. The lightweight material is considered because it enhances the wheels response on braking because of a decrease in rotational weight of vehicle. By the lessening in upsprung weight, we can accomplish more precise steering and increase in fuel efficiency. By using the modelling software CATIAV5R18 the three-dimensional Wheel rim model is designed and saved in IGES format. Later for analysis purpose, model is imported to most recently utilized software ANSYS Workbench 15. The fundamental thing is to direct a stress analysis of automobile rim by considering the loads act on it, the loading conditions are applied and bolt holes are constrained in six DOF, to analyse the static and fatigue analysis of four different materials, namely Aluminium alloy, Magnesium alloy, Titanium alloy and Carbon-Epoxy composite material and their relative performances have been observed individually. In addition to this rim is subjected to modal analysis, from this vibration analysis the natural frequencies and mode shapes of Wheel rim are obtained. In this dissertation work by observing the results obtained from Static, Fatigue and Modal analysis, Carbon-Epoxy Composite material can be suggested as the best material for wheel Rim. The finite element analysis shows that the weight of the wheel can be optimized to around 7%. Index term – Wheel Rim, FEM, Weight optimization, Fatigue life, Modal analysis, Stress analysis.

MS Hebbal, Mukunda Dabair International Journal of Engineering Research & Technology (IJERT) 8 (7), 599-611, 2019. The purpose of the car wheel rim is to provide a firm base on which tire could be fitted. The spoke wheel was invented more recently, and allowed the construction of lighter and swifter vehicles. Alloy wheels are automobile wheels which are made from an alloy of aluminium or magnesium metals. The model is created in Catia and then it is imported in Ansys workbench through IGES file. The finite element idealization of this model was then produced using the tetrahedron solid element. The given car rim is made up of Al-6061 which is having precipitation aluminium alloy containing magnesium and silicon as its major alloying elements and this is subjected to different boundary conditions. In this work our aim is to carry out the different analysis such as static structural analysis, modal analysis and harmonic analysis. In static structural analysis we found out equivalent von mises stresses, deformation and maximum principal stresses and in the modal analysis we checked the modes shapes of the car rim and further we found out the harmonic's stresses, acceleration response, deformation response etc. Later we compared the induced stresses are less than the allowable stresses of the material.

CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Assault Systems. Written in the C++ programming language, CATIA is the cornerstone of the Assault Systems product lifecycle management software suite.

CATIA competes in the CAD/CAM/CAE market with Siemens NX, Pro/E, Autodesk Inventor, and Solid Edge as well as many others.

SUPPORTED OPERATING SYSTEMS AND PLATFORMS

CATIA V6 runs only on Microsoft Windows and Mac OS with limited products.

CATIA V5 runs on Microsoft Windows (both 32-bit and 64-bit), and as of Release 18Service Pack4 on Windows

 DOI : 10.35291/2454-9150.2022.0259

Vista 64.IBM AIX, Hewlett Packard HP-UX and Sun Microsystems Solaris are supported.

CATIA V4 is supported for those UNIXs and IBM MVS and VM/CMS mainframe platforms up to release 1.7.

CATIA V3 and earlier run on the mainframe platforms.

III. RESULTS AND DISCUSSION

Simulation of optimized model

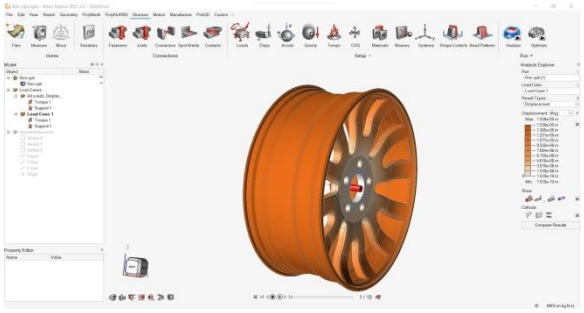


Fig: 1 Displacement of optimized model

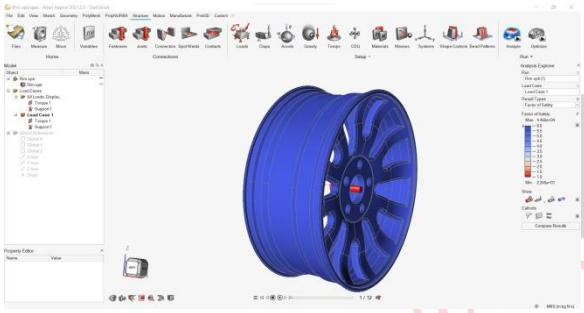


Fig: 2 Factor of safety of optimized model

Fig: 3 Maximum shear of optimized model

Fig: 4 Tension/compression of optimized model

Fig: 5 Von mises stresses of optimized model

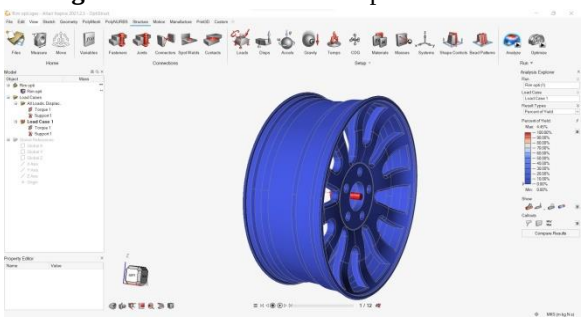


Fig: 6 Percent of yield of optimized model

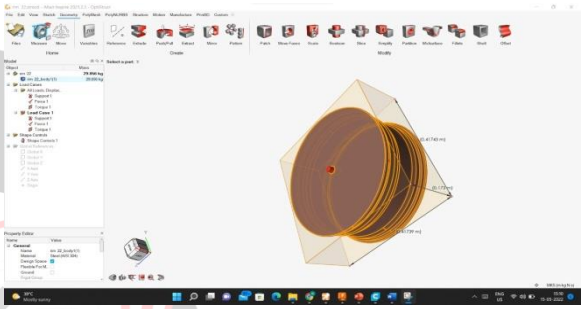


Fig: 7 Actual Model with Dimensions

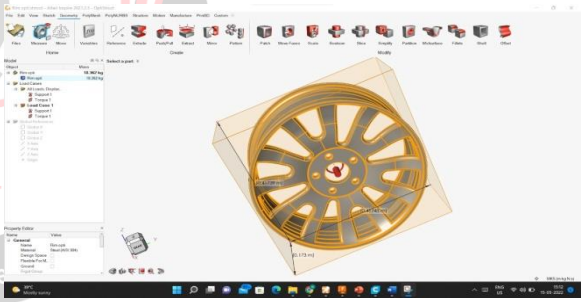


Fig: 8 Optimized Model with Dimensions

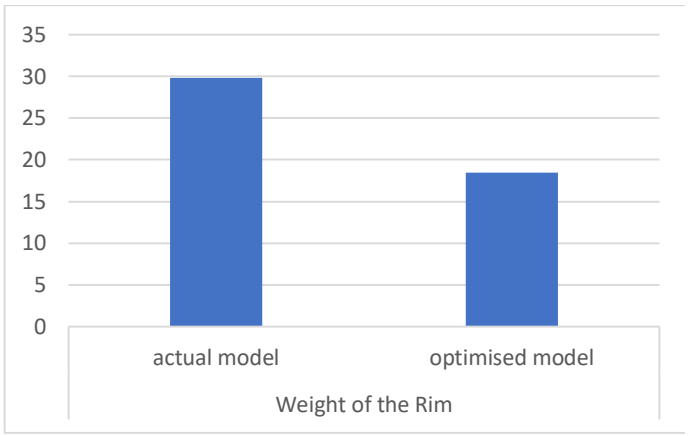
	Actual model	Optimized model
Weight of rim	29.856 kg	18.362 kg
Displacement of rim	5.856e^-05m	1.539e^-0.5m
Factor of safety of rim	6.54	4.46
Percent of yield	18.8%	4.45%
Maximum shear	2.343e^-07pa	5.521e^06pa
Tension or compression	4.035e^07pa	9.564e^06pa
Von mises	4.058e^07pa	9.564e^05pa

Table: 1 Results

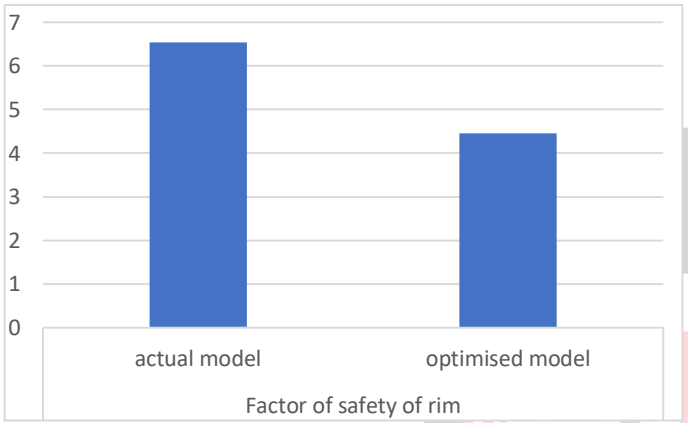
Graphs

Weight of the Rim	actual model	29.86	
	optimised model	18.45	

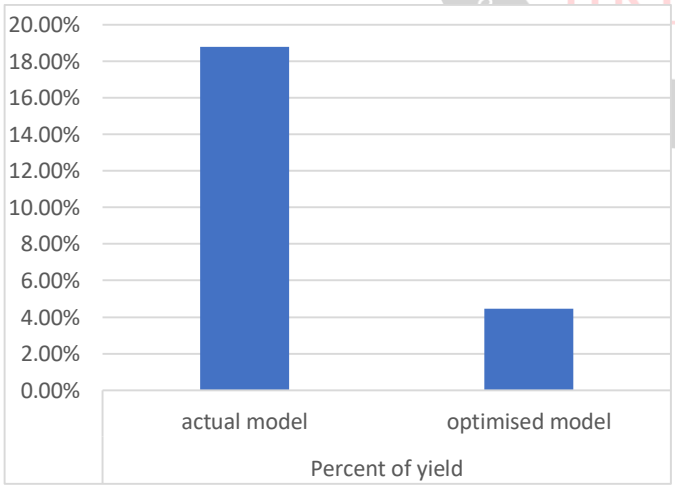
DOI : 10.35291/2454-9150.2022.0259



Factor of safety of rim	actual model	6.54	
	optimised model	4.46	



Percent of yield	actual model	18.80%	
	optimised model	4.45%	



IV. CONCLUSION

1. After performing the mass optimization for maximum stiffness contain the weight of the rim reduced to 18.63 kg.
2. After performing the structural analysis before and after the mass optimization the displacement of car rim reduced.
3. Factor of safety of the car rim for actual model and optimized model is in safe condition. The optimized model gives the factor of safety of 4.468
4. But the maximum shear stress, tension/compression and von mises stresses are increased when compare to the actual model. the stresses in the optimized increased due to different cross sections in the model causes more stress concentration.
5. When compare with percent of yield with actual model is reduced. The percent of yield in actual model is 18.8% and it is reduced to 4.45%.

By conducting this optimization, we have successfully reduced the weight of the car rim and validated with the actual model results. The optimized model gives better results and safe in condition.

V. REFERENCES

[1] G. Ashok Kumar, M. Uma Mahesh, S. Madhu Sudan, T. choli raj et al Dec -2016 DESIGN AND ANALYSIS OF WHEEL RIM BY USINGCATIA &ANSYS Volume: 03 Issue: 12 | Dec -2016 www.irjet.net e-ISSN: 2395 -0056.

[2] T. S. Prasad, J. Reddy et al. July (2020) A Review on Modeling and Analysis of Car Wheel Rim using CATIA & ANSYS doi:10.3390/su12218774 www.mdpi.com/journal/sustainability.

[3] E. Chandrashekhar, J. Rishi 29 August 2016. This present thesis gives a brief compilation of research related to analysis of deformation, equivalent stress, fatigue life and natural frequency for lighter alloy and composite automobile Wheel rim.

[4] MS Hebbal, Mukunda Dabair International Journal of Engineering Research & Technology (IJERT) 8 (7), 599-611, 2019. The purpose of the car wheel rim is to provide a firm base on which tire could be fitted. The spoke wheel was invented more recently, and allowed the construction of lighter and swifter vehicles. Alloy wheels are automobile wheels which are made from an alloy of aluminium or magnesium metals. The model is created in Catia and then it is imported in Ansys workbench through IGES file. The finite element idealization of this model was then produced using the tetrahedron solid element.

[5] J Stearns, TS Srivatsan, Xiaosheng Gao, Paul C Lam International Journal of Rotating Machinery 2006, 2006. This paper highlights the use of the finite element technique for analysing stress and displacement distributions in wheels of automotive vehicles when subject to the conjoint influence of inflation pressure and radial load.

[6] Alireza Afshari, Lars Ekberg, Luboš Forejt, Jinhan Mo, Siamak Rahimi, Jeffrey Siegel, Wenhao Chen, Pawel Wargocki, Sultan Zurami, Jianshun Zhang Sustainability 12 (21), 8774, 2020.

[7] Harish Panjagala, M Balakrishna, Shasikant Kushnoore, ELN Rohit Madhukar. MATEC Web of Conferences 172, 03006, 2018.

[8] K Srinivasa Rao, M Rajesh, G Sreedhara Babu Int. Research J. Engg., and Tech 4 (6), 2036-2042, 2017. The existing car model wheel rim is drawn in the design software Creo 2.0 and various loads and forces are theoretically calculated and applied on the model and analysed through Ansys (17.0) software.

 DOI : 10.35291/2454-9150.2022.0259