International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 04 Issue: 04 | April -2020 ISSN: 2582-3930

DESIGN AND FATIGUE ANALYSIS OF AUTOMOTIVE WHEEL RIM OF FOUR WHEELER

RINA GONDANE
PG STUDENT
DEPARTMENT OF MECHANICAL ENGINEERING
ABHA GAIKWAD PATIL COLLEGE OF ENGINEERING

Prof. DIPALI BHOYAR
Asst.professor
DEPARTMENT OF MECHANICAL ENGINEERING
ABHA GAIKWAD PATIL COLLEGE OF ENGINEERING

Abstract: - Reducing the weight of a racing vehicle can substantially improve its acceleration and general performance abilities. More specifically, reduction of the unsprung corner weight can provide noticeable performance gains in handling and responsiveness, leading to a quicker, more agile car due to a lower yawing moment of inertia. Unsprung weight reduction also improves the car's ability to maintain contact between the tires and the road surface for more consistent grip. The unsprung mass is mostly made up of the tires, wheels, and other components housed within the wheel package. The effect of this weight is especially significant in open-wheeled racecars because this mass is the furthest from the car's center of gravity. This is exactly the case for the Formula SAE (FSAE) race vehicles considered in this thesis.

Decreasing the weight of the wheel itself is a straightforward approach to reducing the unsprung corner weight as well as rotating mass. Even though there are various commercially available wheels for FSAE cars, the lightest aluminum options have plateaued in weight minimization. Also, maintaining high stiffness is important to minimize compliance and maintain favorable suspension dynamics, specifically camber. So, the idea of a lighter composite wheel is proposed. With the goal of developing a lightweight and stiff wheel, composite materials such as carbon fiber reinforced plastics are a good alternative to conventional metals due to their high stiffness to weight ratios. Through the use of finite element analysis software and physical testing, a laminated composite wheel was developed for the Jayhawk Motorsports FSAE racecars. The composite wheel is significantly lighter than the aluminum benchmark and maintains structural integrity as designed for the load cases compared herein. The details of its development are presented throughout the text of this thesis. When compared Aluminum alloy wheel has better results than Steel wheel.

Introduction

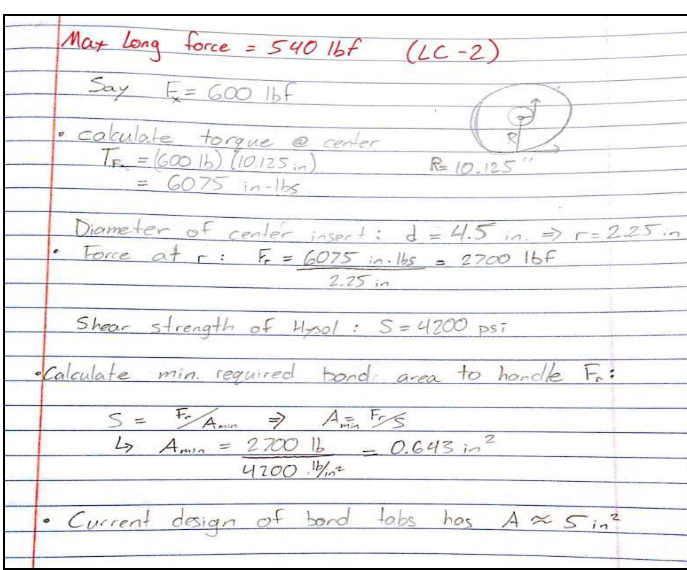
Automotive racing is an exciting and extremely competitive sport that is popular internationally with hundreds of different series for amateur and professional racers of all levels. The significance and presence of improved vehicle design, engineering and development has greatly increased over the years thanks to advancements in engineering and manufacturing technologies in racing. Each race vehicle's ability to compete and perform is dependent on many different design factors, but perhaps one of the most common efforts made by designers is to decrease the vehicle's weight wherever possible. Because race cars are made up of so many different components, weight reduction is possible in various ways, but one area that many agree to have significant effect on performance and handling is the reduction of unsprung corner mass.

Unique design and development opportunities are especially possible within racing series that allow for more freedom in overall vehicle design. A great example of this is the Formula SAE (FSAE) international collegiate competition series. In FSAE, engineering students design and build open-wheeled, single-seat race car prototypes per the rules and guidelines of the Formula SAE rulebook [1]. These students then compete with and race their vehicles at competitions all over the world with the purpose of displaying and proving their unique designs and automotive engineering abilities. This thesis investigates the development of a lightweight 13 inch composite wheel for FSAE racing, designed specifically for the Jayhawk Motorsports (JMS) race cars.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 04 Issue: 04 | April -2020 ISSN: 2582-3930

Figure 1.1: 2013 JMS race car

Damage of alloy wheel by using S-N curve. S-N curve is input for a A.356.2 material [4]. A detailed static and fatigue analysis of Aluminum alloy wheel under a radial loads has been done. Analysis of Aluminum alloy wheel A356.2 was carried out using FEA package. The 3 dimensional model of the wheel was designed using CATIA. Then the IGES format 3-D model was imported into ANSYS. Their thesis summarizes the application of finite element analysis technique for analyzing stress distribution and fatigue life of Aluminum alloy wheels subject to radial loads. Alloy wheels intended for use on passenger cars stipulate two types of fatigue tests, the dynamic cornering fatigue test and the dynamic radial fatigue test. As wheels undergo inconsistent, varying loads during their service life, fatigue behavior is a key consideration in the design and performance evaluation. But since alloy wheels are designed for styling and have more complex shapes than regular steel wheels, it is difficult to assess fatigue life by analytical methods. So, finite element analysis has been used to evaluate the performance of wheels over their life [5]. The entire wheel design of two wheeler was chosen and analyzed by applying different load and redesign the wheel again to minimize the deformation and material will be changed from Aluminum to PEEK (polyether ether ketone) the whole design is made by using NX 7.5. The whole design has been made as per original equipment manufacturer (OEM'S) requirement. Analysis has been done by Ansys 13.0 software to determine the various stresses, strain and fatigue life of the wheel [6]. At seeking a practical and comprehensive method for simulating the dynamic cornering fatigue test of the automotive wheels. The test of a steel passenger car wheel is simulated by combined use of the linear transient dynamic finite element analysis and the local strain approach. A rotating force of constant magnitude is applied to the moment arm tip to simulate the rotating bending effect on the wheel, with the wheel stationary. It is found that only a radial component of the rotating force is needed to obtain the sufficiently accurate radial normal strain histories of the elements located along the radial direction. The strain history of the element whose local stress–strain characteristic keeps linear and closest to the critical element is applied to predict the fatigue life of the critical element with Neuber's rule and local strain approach, which is quite close to the test results [7].



Max Long force = 540 lbf (LC-2)

Say F_x = 600 lbf

• calculate torque @ center

T_{F_x} = (600 lb)(10.125 in) = 6075 in-lbs R= 10.125"

Diameter of center insert: d = 4.5 in ⇒ r = 2.25 in

• Force at r: $F_r = \frac{6075 \text{ in-lbs}}{2.25 \text{ in}}$ = 2700 lbF

Shear strength of Hysol: S = 4200 psi

• Calculate min. required bond area to handle F_r:

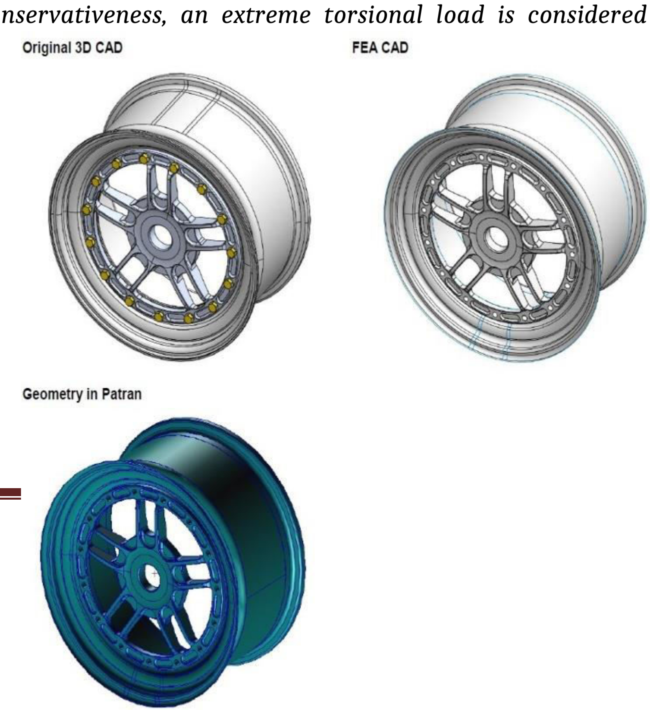
$S = F_r / A_{min} \Rightarrow A_{min} = F_r / S$

↳ $A_{min} = \frac{2700 \text{ lb}}{4200 \text{ lb/in}^2} = 0.643 \text{ in}^2$

• Current design of bond tabs has $A \approx 5 \text{ in}^2$

CALCULATION

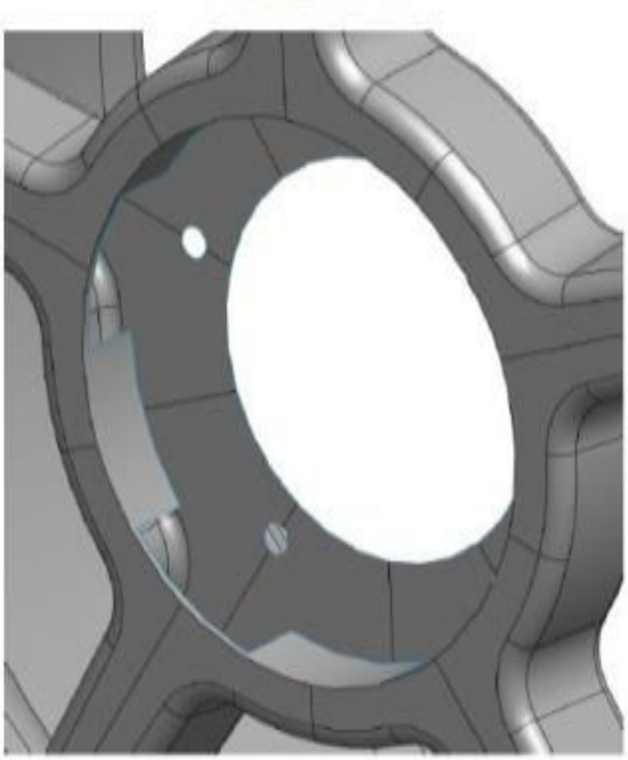
A simple calculation was done to determine sufficient surface area for bonding the aluminum insert into the CFRP rim. The hand calculation is shown below. For simplicity and conservativeness, an extreme torsional load is considered

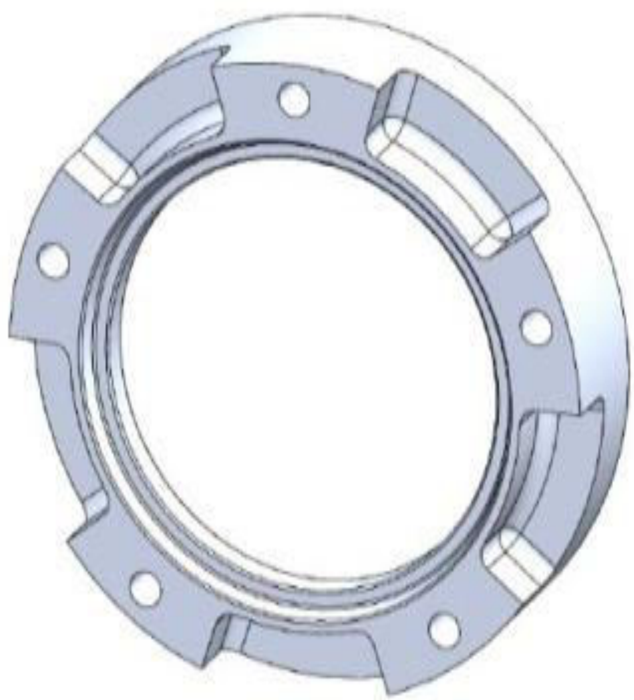


that would not be experienced in real life. The case used assumes that the center insert is fixed only by circumferential bond area and that there is now CFRP laminate between the insert and the hub. The required minimum bond area calculated is greatly exceeded in the final design.

Component	Weight (lb)	Material
Center	2.1570	6061-T6
Inside Hoop	3.8970	6061-T6
Outside Hoop	1.5840	6061-T6
Total	7.6380	

Material	E	ν	σ_{yield}	$\sigma_{ultimate}$
Al 6061-T6	10 Msi	0.330	40 ksi	45 ksi





PROBLEM DESCRIPTION DESCRIPTION

The steel wheel is a mechanical device, which provides rotary motion by means of a disk or circular frame revolving on an axis. The steel wheel can be used in heavy load and traction services. The size and contour of a specific wheel design is based on the load it must carry and the space limitations of the equipment on which it are used. The contour of the wheel is normally composed compose of five parts: the hub, plate, rim, tread and flange. Today, the steel

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 04 Issue: 04 | April -2020 ISSN: 2582-3930

wheel has been partially replaced by the Aluminum alloy wheel on vehicles, and this trend is expected to continue and become more popular in the near future. There are at least four good reasons why wheel industries now prefer to use Aluminum alloy to replace wrought steel: Aluminum alloy wheels are more loads worthy. Excellent brake system. Aluminum thermal conductivity is about three times higher than that of steel this physical property gives the wheel better brake reliability and longer life than the steel wheel; it is fuel- efficient. Suspension improvements. The suspension system of Aluminum alloy wheels is capable of responding much more quickly to changing surfaces and road conditions. This increased traction can improve vehicle acceleration, maneuverability and brake performance. It is particularly noticeable while driving at high speeds or on rough roads.

METHODOLOGY

The system of work is Geometric demonstrating of a wheel in CATIA. Static examination for steel wheel existing model under stacking conditions. So as to take care of the issue of the venture, a nitty gritty limited component investigation is proposed to decide the complete disfigurement and Comparable worry in static condition utilizing the examination programming ANSYS WORKBENCH. Modular examination for steel wheel existing model under stacking conditions. So as to take care of the issue of the task, a point by point limited component examination is proposed to decide the complete misshapening under frequencies at every mode utilizing the investigation programming ANSYS WORKBENCH. Weariness investigation for steel wheel existing model under stacking conditions So as to take care of the issue of the venture, a nitty gritty limited component examination is proposed to decide proportionate elective pressure, factor of wellbeing, exhaustion life utilizing the investigation programming ANSYS WORKBENCH. The over three investigation are rehashed for Aluminum compound wheel. In the wake of dissecting the two materials are analyzed.

MECHANICAL PROPERTIES

Modeling and Analysis

*The wheel body beneath consideration is generated within the CATIA CAD Modeling package. It is a powerful software used to create complex designs with tremendous precision. It has residences like Feature-based nature, Bidirectional associative property and parametric nature. Parametric capabilities are beneficial in reusing wheel version of automobile frame to create new version layout. The CATIA document is stored in *.IGES format (Initial Graphics Exchange Specification) is an alternate for product records in help of business automation. The trendy emphasis of IGES is to cast off the human presence from the "product facts". The valuable unit of information exchange in the IGES version is the utility, which includes numerous forms of entities. This technique keeps all of the significant associative and relationships between the application entities. Therefore IGES is to represent all product records, in a commonplace facts format, at some stage in a product's complete lifestyles cycle*

.FOR STEEL WHEEL

Modal analysis Results

In this modal analysis no loading conditions are considered