wjert, 2021, Vol. 7, Issue 1, 161-206. *Original Article* ISSN 2454-695X

World Journal of Engineering Research and Technology

WJERT

www.wjert.org Impact Factor Value: 5.924

DESIGN AND ANALYSIS OF WHEEL HUB FOR WEIGHT OPTIMIZATION BY USING VERIOUS MATERIAL

***[1]Pravinkumar Tayade, [2]Dr. P. A. Makasare and [3]S. I. Bharadiya**

[1]Project Author, Siddhant College of Engineering.

[2]Project Guide, Siddhant College of Engineering.

[3]Project Co-Guide, Siddhant College of Engineering.

Article Received on 20/11/2020 **Article Revised on 10/12/2020** **Article Accepted on 30/12/2020**

***Corresponding Author**
Pravinkumar Tayade
Project Author, Siddhant College of Engineering.
pravin.tayade@yahoo.co.in

ABSTRACT

In this study, various methods are used with reference to available data from existing papers and reviewers comment for optimization of wheel hub with the help of different materials. This study helps the designers for selection of perfect material for wheel hub with proper weight reduction with best structural design. This weight optimization of wheel hub is done using FEA methodology. Model for the hub is prepared with actual dimensions and this model is checked with FEA analysis using different material. The best suit material is founded out for wheel hub with lighter in weight and minimum cost all the optimization process is conducted by use of modeling software Catia version 5 R19 and FEA tool Ansys workbench 16. In this wheel hub weight optimization process we can analysis the key areas of researches as wheel hub shape optimization, static load analysis on wheel and fatigue load analysis using FEA. Finally we will get the lighter weighted wheel hub with same strength for sustain the same loading in operating vehicle conditions.

1. INTRODUCTION

The wheel hub and steering knuckle are very critical components in assembly of wheel suspension, which allows the wheel to move up and down and rotate about the axis as well as they are capable to take vertical load of vehicle. Hub is also known as the knuckle. The wheel hub and steering knuckle also allows steering wheel to turn around while turning. The assembly of front tyre and hub rotates in same plane using suspension. The knuckle is attached between the tire and a spindle that can be move in a vertical plane in order to allow motion to the wheel by a suspension assembly. The force exerted on hub and knuckle assembly is bumping shocks from road, cornering load and breaking torque by the application of brakes. The force applied by the bump is in cyclic nature. While production of wheel for sport vehicle it is need to be made lighter in weight to attain high speed. For the sports vehicle the mass of wheel hub should be minimized; therefore while designing the sports vehicle the designers keep this as key aspect and design the vehicle wheel hub for minimum weight and maximum stress and force sustaining capability. The production and design of the wheel hub and knuckle is important parameters in optimizing the weight of the vehicle. The mass reduction without compensating the strength of the wheel hub and knuckle assembly is done by the designers to optimize the weight of the vehicle. Weight or mass of the vehicle can be reduced by using various aspects like proper material selections, design and analysis method and optimization method. Steering knuckle subjected to time varying load during service life, leading to fatigue loads. The hub and knuckle assembly also transfers the whole weight of the vehicle into wheels, which lead to stress on mountings. Wheel hub and knuckle can be analyzed without due consideration to bearing design. In automotive suspension, a steering knuckle is that part which contains the wheel hub or spindle, and connected to the suspension components, variously it is known as steering knuckle, spindle, knuckle or hub The wheel and tire assembly attach to the hub or spindle of the knuckle where the tire/wheel rotates while being held in a stable plane of motion by the knuckle/suspension assembly of double-wishbone suspension, the knuckle is attached to the upper control arm at the top and the lower control arm at the bottom. The wheel assembly is shown attached to the knuckle at its Centre point. Suspension systems in any vehicles uses different types of links, arms, and joint to late wheels move freely, front suspensions also have to allow the front wheel to turn. Steering knuckle/spindle assembly, which have two separate or one complete parts attached together in one of these links. Hub is the part attached to knuckle; the purpose of a wheel hub is to attach a wheel to a motor shaft.

Wheel hubs application is also used to attach lifting arms, release doors and pulleys to motor shafts, etc. Wheels are mounted on to hubs via the wheels face or it’s Centre. The wheel is attached through fasteners for easily removed for storage or servicing and hub is good strength. Hubs are typically attached to the motors by closely sliding over and locking in to engagement with their shafts transferring torque from the motor, through the hub and to the wheel. Here the manufacturing hub must be capable of rigidly supporting it’s contribute of the total weight of a vehicle without fail during its expected life span. If the hub geometry and material selection are too low in quality, then it will break assembly which cannot be repaired after the failure.

1.1 History

Hubs, also known as freewheeling hubs are fitted to some (mainly older) four-wheel drive vehicles, allowing the front wheels to rotate freely when disconnected (unlocked) from the front axle. This is done to reduce the mechanical resistance of the front-portion of the drive train when four-wheel drive is not in use. The hub, along with the wheel, is designed to engage (lock) onto the axle, to be powered by the drive train in four-wheel drive; or the hub can disengage (unlock) from the axle when four-wheel drive is not needed, thus allowing the front wheels to rotate freely within the hub. The hub is a component where the wheel is directly mounted to, and is outside the axle.

The benefits of unlocking hubs for normal road use are mainly found in increased fuel efficiency. When the front hubs are locked, even if no power is sent to the front axle (by means of a transfer case), the turning of the wheels will still spin the front axle, differential, and driveshaft, which puts extra load on the engine. Unlocking the hubs disconnects the wheels from the axle, which eliminates this extra load. Other benefits also include keeping the front-differential free from unnecessary wear, quieter operation, less vibration, and lower wear in other drive line components. However, many manufacturers list engaging the hubs (even in 2WD mode) for several miles a month to lubricate the front drive train as part of the vehicle's regular maintenance schedule.

Mechanically activated locking hubs are activated by hand by turning a switch on the end of the axle. The advantage to mechanical hubs is that they are often considered more robust, and less prone to issues due to maintenance negligence. The disadvantage of this is that the driver needs to get out of the vehicle to activate the hubs.

In some other vehicles, automatic locking hubs are used which, as the name implies, engage automatically when 4WD is desired. The main advantage is that the driver does not need to leave the vehicle to connect the wheels to the axle. The disadvantage with this system is that most designs require the vehicle to move some distance (usually a whole wheel turn, often going backwards) after engaging the hubs in order for the hubs to engage or disengage. This might not be possible if the vehicle gets completely stuck before the hubs have been engaged, meaning automatic hubs require more caution on the driver's part.

Locking hub mechanisms also generally extend further beyond the wheel than most axles, and exposed hub locks can be broken or damaged by off-road conditions, rendering 4×4 useless and leaving the vehicle stranded. Also, in some axle designs (such as those used on older Land Rovers), the top swivel bearing can become starved for lubrication (which is normally supplied by oil slung up by the rotating axle), unless the hubs are locked every few hundred miles. Also, since locking hubs generally do not require a key to operate, they can be maliciously locked or unlocked by persons other than the vehicle owner.

1.2. Theory

Brief Description to Hub and Upright: In automotive suspension, a steering upright is that part which contains the wheel hub or spindle, and attaches to the suspension components; variously it is known as steering knuckle, spindle, upright or hub.

Fig. 1.1: Wheel Hub.

The wheel and tire assembly attach to the hub or spindle of the knuckle where the tire/wheel rotates while being held in a stable plane of motion by the knuckle/suspension assembly. As shown fig. of double- wishbone suspension, the knuckle is attached to the upper control arm at the top and the lower control arm at the bottom.

The wheel assembly is shown attached to the knuckle at its center point. Suspension systems in any vehicles uses different types of links, arms, and joint to let the wheels move freely, front suspensions also have to allow the front wheel to turn. Steering knuckle/spindle assembly, which have two separate or one complete parts attached together in one of these links. Hub is the part attached to upright, the purpose of a wheel hub is to attach a wheel to a motor shaft. Hubs are also used to attach lifting arms, release doors and pulleys to motor shafts.

Wheels are typically attached to hubs via the wheels face or its center. The wheel is attached through fasteners to hub due to a good strength and can be easily removed for storage or servicing. Hubs are typically attached to the motors by closely sliding over and locking into engagement with their shafts transferring torque from the motor, through the hub and to the wheel. Hub must be capable of rigidly supporting its share of the total weight of a vehicle without failure during its expected life span. If the hub geometry and material selection are inadequate, it will break assembly which cannot be repaired.

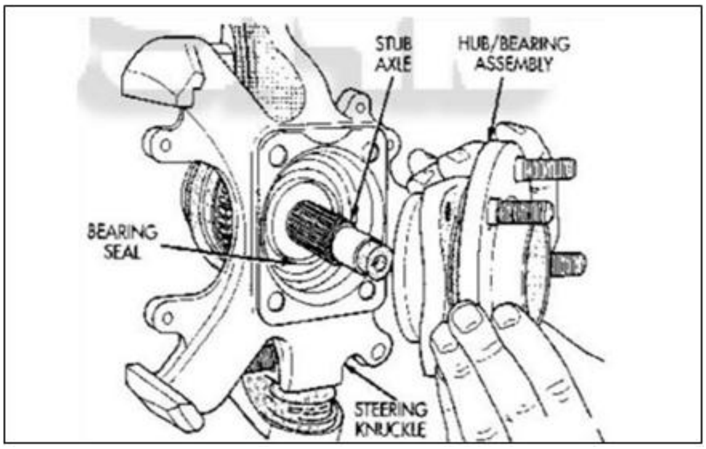


Fig. 1.2: Assembly of Wheel hub and parts.

1.3 Problem Statement

Our previous vehicle utilized wheel hub made of mild steel and cast iron. Mild steel provided strength to hub against bending during cornering and bump situations and resisting against torsion during braking. The total mass of the wheel hub is considerably high respective of total vehicles mass. Wheel hub contributes significant weight in total vehicle weight. Although the previous design has worked very well in terms of strength but it had following major issues which became important design points for the future design:

- The wheel hubs were very heavy
- The brake rotors once fixed could not be removed if in case it so damaged.
- The External diameter was large which restricted us to use bearing of larger dimension adding more weight.

So I am doing the project of design the wheel hub of for material optimization. That the project to find the optimal design for hubs. Earlier we did first the basics calculation for the hubs, but now we want to know more about the real work of it. In the first point, that we want to know when the current hubs will break, because that piece need support a number of cycles and they wouldn´t like that pieces break in the middle of the race. For that we want to know best suit and lighter material for wheel hub which assist the vehicle for great operation under heavy load and high speed. Known if parts endure or not endure this number of cycles, the parts are manufactured adapted to the required needs. With some calculus and with CATIAV5R19 and ANSYS19.2, it's possible to solve this problem. We can find out optimum material for wheel hub for less weight.

Images of failure

- **Crack near the hub**

Fig. 1.3: Failure of hub.

Fig. 1.4: Failure of hub.

1.4. OBJECTIVES

1. To prevent the failure of wheel hub due to crack initiated near the hub which further gets propagated throughout the hub which leads to fatigue failure.
2. To improve material optimization.
3. To improve design optimization is necessary for which best material has to be selected by conducting design of experiments.

1.5 Function of wheel hub

Function of The Wheel Hub, in most applications the center of a wheel, or other rotation components. The Hub mainly used to attach a bearing which allows the wheel to rotate around an axle. The bolt and attachment for the wheel is a part of the Wheel Hub. Wheel Hub's located at a driven axle of the car transfer torque from the driveline to the wheel. The Wheel Hub's on the rally cross car is equal due to cost reduction and the fact that the car is four-wheel-driven. The current assembled Wheel Hub with upright, which is a part of the suspension that attaches the Hub, the strut and the control arms and driveshaft included is presented in Figure.

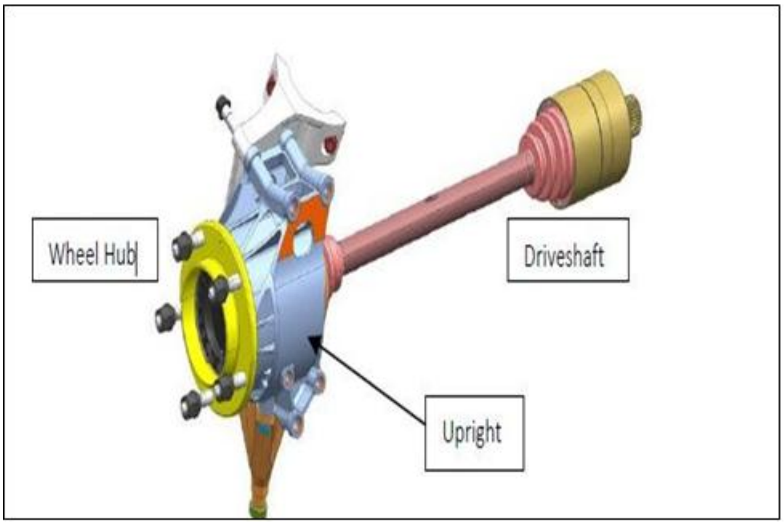


Fig. 1.5: Complete Wheel Hub, upright and driveshaft.

1.6 Scope

Day by day the competition is increasing with new innovations and ideas in automobile sectors. With these innovations, a new path is created in the product development. In this development there is a large scope in modifying the existing materials or replacing old products by new and advanced material products. Automotive organizations are paying their major interest in the weight reduction of components because the weight of the vehicle is the most important factor to be considered as it affects the fuel economy. This weight can be reduced by introducing new materials with better properties and manufacturing process with optimization of design. By this we can achieve more fuel efficiency in vehicles due to reduced weights. Minimizing the weight in the wheel is more effective than minimizing the weight in other components because of its rotational moment of inertia effect during its motion and also the tyre take the overall vehicle load and provides cushioning effect. This project is with the design of aluminum alloy wheel hub for the automobile application by paying special reference to optimization of the mass of wheel hub by using current opportunities and trends. By reducing the weight we can achieve the objective the reducing of un sprung mass, by which the inertia loads and overall weight are reduced with improvement of performance and fuel economy. There is large scope for reducing the mass of aluminum wheel hub by changing or replacing the materials with composites to increase the bearing of stresses and to decrease its mass and volume. From the finite element calculations it is found that the mass of the wheel hub can be reduced to 50% from the existing MS wheels hub. The analysis also shows that after the optimization the stresses generated from the wheel hub will be below the yield stress. This gave a new approach in the field of optimization of passenger car wheel hub

1.7 Design Methodology

Below is the methodology by which we are performing dissertation work in order to optimize the best material for the wheel hub. Methodology used in the analysis and design of the wheel hub.

1. Research Paper and study of past work done

For This Project We were focusing on finding research papers for prediction of research gap and the idea to find new concept with mind-set of project development regarding design and manufacturing. The research papers were gave us the domains and works which were already completed and provided lots of information regarding design and analysis of wheel Hub for weight optimization by using various material.

2. Collection of Data to design wheel hub

From research papers and resources we were collect the data for actual design to overcome the bugs and challenges. And we were come to know about various material used for wheel hub. The all collected data was used for getting proper path for development.

3. CATIA design for model building of Hub

For our project the next step to design of wheel hub with approximate calculations of different forces acting on wheel hub. For Catia modeling we have taken the help of some of the research papers and car catalogue we developed the 3D model of our project that is wheel hub design in CATIA v5 software

4. Preparation of test set up for testing

In this step we will do the testing of wheel hub with different forces acting on it while wheel hub, this will give us data regarding different parameter which help us to improve the design of wheel hub and for selection of better material.

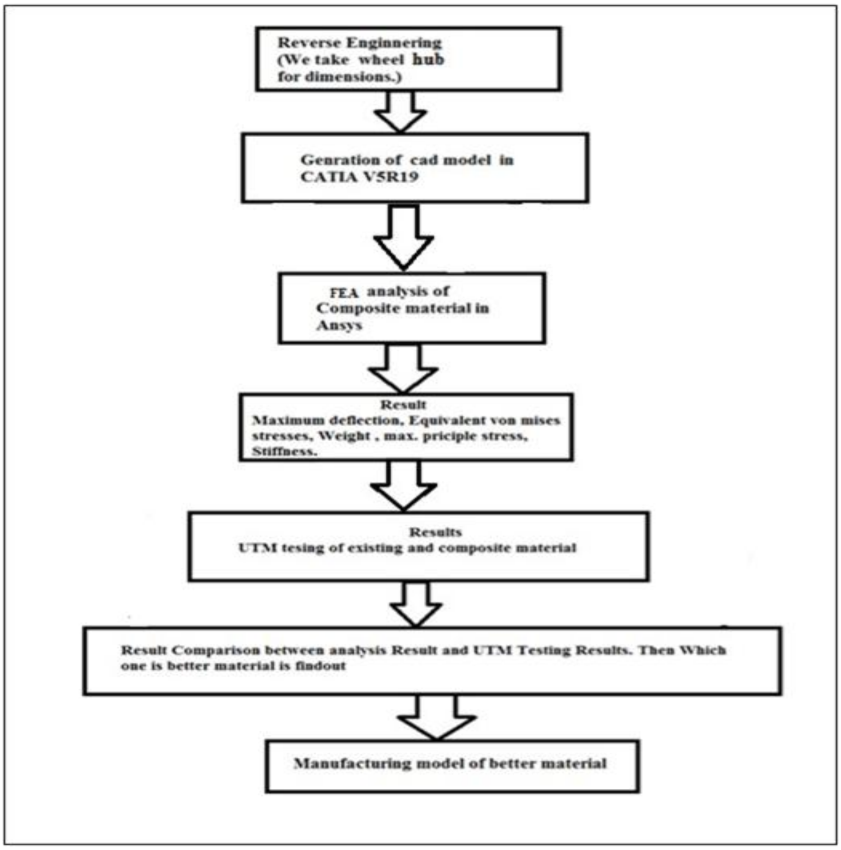


Fig. 1.6: Flowchart of Methodology.

1.8 Organization of Dissertation

- Chapter 1. Includes the Introduction, Problem statement, Objectives, Scope and methodology.
- Chapter 2. Covers Literature review, review of papers, Comment on reviewed papers, Gap Identifies.
- Chapter 3. Includes the Material selection parameters, Properties of material, Calculations regarding load on the hub.
- Chapter 4. FEA analysis of all material and lab testing of materials.
- Chapter5. Presents the Results obtained from FEA analysis and lab testing.
- Chapter 6. Discusses the Conclusions and Future scope

1.9 Plan of Execution

Months/ Activity	Jun	Jul	Aug	Sep	Oct	Nov	Dec	Jan	Feb	Mar	Apr	May
A												
B												
C												
D												
E												
F												
G												
H												
I												
J												

Fig. 1.7: Plan of work.

Activities

1. A= Topic finalization
2. B= Literature Review
2. C= Formulation of Problem
3. D= Parametric analysis
4. E=Development of CAD models of system
5. F= Purchasing of components
6. G= Manufacturing
7. H= Assembly and Testing
8. I= Results and Conclusion
9. J= Report Writing

2. LITERATURE SURVEY

Begin with a literature review, lot of papers and journals have been read up and a part of it has been considered in this project which will be useful for the analysis performed in this dissertation.

[1] Kaixian Yue,

This paper concluded that the three kinds of acceleration models for the accelerated life testing for fighter-bomber wheel hubs corresponding to three kinds of SCF are built, and t-test is used to estimate the relation among them. When the significance level is set as 0.02, the three acceleration models are equivalent, which suggests that the acceleration factor for fighter-bomber wheel hubs made by 2014Al is not sensitive to SCF. The fatigue life tests of

2014 aluminum alloy, which is extensively used to make fighter-bomber wheel hubs. The BLUE (best linear unbiased estimation) and MLE (maximum likelihood estimation) are adopted in the estimation of the acceleration models, and the acceleration models under the conditions of different SCF (stress concentration factor) are compared by statistical method to determine the sensitivity of SCF to the acceleration factor of the fighter-bomber wheel hubs made by 2014Al.

[2] M. Amura, L. Allegrucci, F. De Paolis, M. Bernabei.

This paper concluded that the forged 2014-T6 aluminium alloy main wheel of an AMX aircraft failed during pre-flight taxiing. Failure occurred on the wheel housing hub and its origin was located on the interior edge of the bearing cup housing. The wheel experienced high cycle’s fatigue. A plastic deformation was produced during the wheel mounting operations caused an abnormal stress concentration. It has been identified as cause of a progressive fracture mechanism. Consistent optical, metallographic and electron microscopy evidences were collected. Moreover, Finite Element Analysis (FEA) demonstrated that the observed defects produced the stress level needed to fatigue to initiate. Recommendations were issued in order to improve the Non- Destructive Techniques (NDT) used to monitor the wheel structural integrity. Laboratory tests showed that ultrasounds in this case would be far more efficient than the prescribed eddy currents.

Kazem Reza Kashyzadeh, Mohammad Jafar Ostad-Ahmad-Ghorabi, Alireza Arghavan

This paper concluded that the fatigue testing which is one of the most significant ones. Another issue is the high cost in practical ways, and to cope with this issue various ways must be assessed and analyzed, one of the best and the most efficient ways is modeling and testing in virtual software environments. In the present paper, predict fatigue life of suspension component and package of automotive suspension are the main purposes. First, using MATLAB software, road roughness according to the intercity roads for constant vehicle velocity (100Km/h) has been studied. After that frequency response of components has been analyzed, its critical points determined to calculate the fatigue life of the part, and the amount of critical stress obtained based on Von Misses, Tresca and Max Principle criterion for a quarter car model (passive suspension System in 206 Peugeot). Fatigue life of the vehicle components are calculated in terms of kilo- Meters in specialized fatigue software such as 116944, 92638.9, 46388.9 and 191388.9 Km respectively wheel hub, pitman arm, suspension arm and package of suspension. Finally, to prove the given results of

the finite element method compared with reported results by other researchers
and the results match very well with those.

[3] J. Janardhan, V. Ravi Kumar, R. Lalitha Narayana,

This paper concluded that the vehicle (car) may be towed without the engine but at the same time even that is also not possible without the wheels, the wheels along the tyre has to carry the vehicle load, provide cushioning effect and cope with the steering control. The main requirements of an automobile wheel are; it must be strong enough to perform the above functions. It should be balanced both statically as well as dynamically. It should be lightest possible so that the unsprung weight is least. The Wheel has to pass three types of tests before going into production, they are Cornering fatigue test, Radial fatigue test and Impact test. In this thesis radial fatigue analysis is done to find the number of cycles at which the wheel is going to fail. The 2D of the wheel was created in MDT, the drafting package and the same was exported to ANSYS, the finite element package using IGES translator where the 3D model of the wheel is created. The wheel is meshed using SOLID 45 element. A load of 2500N was applied on the hub area of the wheel and a pressure of 0.207N/mm2 is applied on the outer surface of the hub. The pitch circle holes are constrained in all degrees of freedom. The analysis is carried under these constraints and the results are taken to carryout for further analysis i.e. fatigue module to find the life of the wheel.

[4] Malapati.M, K V S Karthik,

This paper concluded that the hub is one of the main parts of automobile wheel that is mounted on the center to wheel. When the axle rotates along with it the wheel hub also rotates. While the vehicle is running, the wheel is affected by different loads and stresses at different temperatures. Generally hubs are made up of ductile iron and aluminum. As we know that the strength and weight of ductile iron is more when compare to aluminum. This research paper describe that Critical parameters such as stress, ultimate tensile stress, % elongation and hardness at different points on the both ductile iron hub and aluminum hub is evaluated by commercial analysis software ANSYS and expehubental way with a suitable material for wheel hub. The theoretical critical parameters such as stress, ultimate tensile stress, % elongation and hardness at different points were compared with expehubental critical parameters values of each material. Hence the deviation of critical parameters is below 10% so that the analytical values of critical parameters are expehubentally proved its correctness.

[5] Mehmet Firat, Recep Kozan, Murat Ozsoy, O. Hamdi Mete

This paper concluded that the 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 analyses. 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. 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.

[6] Fabio M. Santiciolli Riccardo Möller, Ivo Krause, Franco G. Dedini,

This paper concluded that the biaxial wheel fatigue test. This test is the state-of-art and the standard requirement for the validation of vehicle wheels. During this test, tire and wheel specimens run inside an inner drum while standardized vertical and horizontal loads are applied. Thus, the scenario of this test can be modeled in three levels: the multibody dynamics of the test facility, the wheel/tire/inner drum contact, and the analysis of the flexible wheel. In the pro- posed virtualization, the multibody dynamics of the test facility was implemented in MSC.ADAMS. The wheel/tire/inner drum contact was simulated by means of CDTire; as it works parallel to MSC.ADAMS, one single co-simulation could perform the tire dynamics and the test facility dynamics. Finally, the wheel strains were calculated by an ABAQUS simulation, which received the tire/wheel load data from the simulation in MSC.ADAMS and CDTire. A physical test facility and a physical wheel specimen were instrumented, allowing the comparison between acquired and simulated data. The use of this specialized software is a novelty in the virtualization of the scenario of this test; furthermore a high detailed simulation is required for the further development of such already well established test procedure.

[7] Gowtham, A S Ranganathan, S Satish, S John Alexis, S Siva kumar,

This paper concluded that the Wheel hub used for Student formula cars, the brake discs cannot be removed easily since the disc is mounted in between the knuckle and hub. In case of bend or any other damage to the disc, the replacement of the disc becomes difficult. Further using OEM hub and knuckle that are used for commercial vehicles will result in increase of un-sprung mass, which should be avoided in Student formula cars for improving the performance. In this design the above mentioned difficulties have been overcome by redesigning the hub in such a way that the brake disc could be removed easily by just removing the wheel and the caliper and also it will have reduced weight when compared to existing OEM hub. A CAD Model was developed based on the required fatigue life cycles. The forces acting on the hub were calculated and linear static structural analysis was performed on the wheel hub for three different materials using ANSYS Finite Element code V 16.2. The theoretical fatigue strength was compared with the stress obtained from the structural analysis for each material.

[8] Shi-Jian Luo, Ye-Tao Fu, Yu-Xiao Zhou,

This paper concluded that the two vehicle components, it is often difficult for manufacturers to confirm that their selected design is truly the most appropriate for and harmonious with a given car. For the purpose of providing guidelines for wheel hubs selection, this paper, therefore, presented a case study on 6 typical types of cars and 20 wheel hubs, examined their shape design styles and accordingly proposed a methodology for evaluating the perceptual matching quality between the two through a participatory expehubent, including (1) identification of Kansei attributes, (2) matching evaluation by ranking method, and (3) semantic differential (SD) evaluation of wheels and car bodies separately. Based on computing the correlations of SD evaluation value vectors between bodies and wheels, the matching quality was verified to have a strong relationship with the similarity of the components' design style. With a subsequent correspondence analysis of ranking data, the obtained projection value explicitly reflected the strength of the association between wheel hubs and corresponding car types. As a result, 6 positive and 3 negative representative design samples of wheel hubs were obtained for each type of car. Accordingly, design solutions were recommended, and the achievements can be readily employed by companies to propose appropriate designs that precisely meet market demands. Relevance to industry: This paper suggests that automobile product developers consider a matching relationship among the components within their cars. The matching quality of one component has been verified to be strongly related to the similarity between its shape design style and holistic car image. This paper's findings can be readily employed by automobile manufactures for predicting appropriate wheel hub designs according to the specific requirements of vehicle development. These well- founded design solutions will meet the market demands precisely.

MATERIAL SELECTION PARAMETERS

The selection of suitable engineering materials for the design and manufacture of automotive mechanical components remains one of the most difficult task and thus a problem in an automobile industry. Steel has been the dominant material used in the manufacturing automobile parts since 1920s. The choices of materials used by the automobile manufacturers are based on complex decisions variables and are determined by a number of factors and the increasing need to economize fuel. Various parameters are discussed below

3.1 Parameters for material selection

3.1.1 Energy Absorption capacity

Fundamentally, the key requirement of crash protection is to absorb and dissipate energy. In an impact, the crash structure must dissipate the energy of the impact whilst ensuring that the occupants of the vehicle are not subjected to excessive accelerations/forces, and that the survivable zone within the car remains intact (i.e. the crash structure does not ingress too far into the vehicle). This energy dissipation is achieved through plastic work done in deforming the material in the crash structure. Therefore, by comparing the capacity for energy dissipation of different materials, and considering their density, it is possible to assess which materials provide optimal energy dissipation per unit mass – specific energy absorption. In this study, which focuses on?

3.1.2 Weight

Impact attenuator is a basically vehicle component, therefore its weight should be minimum. Due to this Impact attenuator is must be made by light weight material.

3.1.3 Availability in Sheet Form

For manufacturing of the material in sheet form is necessary.

3.1.4 Cost

Cost of the material should be minimum. In recent days the carbon fiber is used for Impact attenuator, but the prize of carbon fiber is not affordable for Asian market.

3.1.5 Weld ability

The weld ability, also known as join ability, of a material refers to its ability to be welded. Many metals and thermoplastics can be welded, but some are easier to weld than others. A material's weld ability is used to determine the welding process and to compare the final weld quality to other materials. The bended sheets of material is joined by welding process. Therefore, material used for impact attenuator must have good weld ability.

3.1.6 Machinability

The term machinability refers to the ease with which a metal can be machined to an acceptable surface finish. Materials with good machinability require little power to cut, can be cut quickly, easily obtain a good finish, and do not wear the tooling much; such materials are said to be free machining

3.2 Material Cost Estimation

Material cost estimation gives the total amount required to collect the raw material which has to be processed or fabricated to desired size and functioning of the components. These materials are divided into two categories.

1. Material for fabrication

In this the material in obtained in raw condition and is manufactured or processed to finished size for proper functioning of the component.

2. Standard purchased parts

This includes the parts which was readily available in the market like Allen screws etc. A list is forecast by the estimation stating the quality, size and Standard parts, the weight of raw material and cost per kg. For the fabricated parts.

Cost of material per KG

Table 3.1: Material Cost Comparison.

SR. No	Material	Cost per kg
1	Cast iron	158/kg
2	Carbon epoxy fiber	1000/kg
3	Stainless steel	250/kg
4	Aluminum alloy	400/kg

3.3 Machining Cost Estimation

This cost estimation is an attempt to forecast the total expenses that may include manufacturing apart from material cost. Cost estimation of manufactured parts can be considered as judgment on and after careful consideration which includes labor, material and factory services required to produce the required part.

Procedure for Calculation of Material Cost:

The general procedure for calculation of material cost estimation is after designing a project,

1. A bill of material is prepared which is divided into two categories.
a. Fabricated components
b. Standard purchased components
2. The rates of all standard items are taken and added up.
3. Cost of raw material purchased taken and added up.

3.4 Material properties

Material property is an intensive, often a quantitative property, with a unit that is used as measure of the value to compare the benefits of one material versus another to aid materials selection.

Material property may be a constant or a function of one or more independent variables, such as temperature. They often vary to some degree, according to the direction in the material in which they are measured; a condition referred to as anisotropy.

3.4.1 Cast Iron

Material Property A48 Class 25 Cast Iron.

Table 3.2: CI Properties.

C.I	
Young's Modulus	1.1e+005 Mpa
Poisson's Ratio	0.28
Density	7200 kg/m^3
Ultimate tensile strength	240Mpa
Yield Strength	280Mpa

Chemical Composition of A48 Class 25Cast Iron:

C%: 3.20-3.50, Si%: 1.60-2.40, Mn%: 0.60-0.90, P% $\leq$ 0.20, S% $\leq$ 0.15, CE: 4.00-4.25.

3.4.2 Carbon-fiber

Carbon-fiber-reinforced polymer or often simply carbon fiber, is an extremely strong and light fiber-reinforced polymer which contains carbon fibers. The polymer is most often epoxy, but other polymers, such as polyester, vinyl ester or nylon, are sometimes used. The composite may contain other fibers, such as agamid e.g. Kevlar, Twaron, aluminium, or glass fibers, as well as carbon fiber. Carbon fiber is commonly used in the transportation industry; normally in cars, boats and trains. Although carbon fiber can be relatively expensive, it has many applications in aerospace and automotive fields, such as Formula One.

Carbon-fiber-reinforced polymer is used extensively in high-end automobile racing. The high cost of carbon fiber is mitigated by the material's unsurpassed strength-to-weight ratio, and low weight is essential for high-performance automobile racing. Racecar manufacturers have also developed methods to give carbon fiber pieces strength in a certain direction, making it strong in a load- bearing direction, but weak in directions where little or no load would be placed on the member. Conversely, manufacturers developed unidirectional carbon fiber weaves that apply strength in all directions. This type of carbon fiber assembly is most widely used in the safety cell monocoque chassis assembly of high-performance racecars. Many supercars over the past few decades have incorporated CFRP extensively in their manufacture, using it for their monocoque chassis as well as other components.

The price is around $10 a pound. Steel, on the other hand, costs less than a dollar per pound. Cost is the main hurdle carbon fiber will have to overcome before it can provide a viable energy solution. The second hurdle is waste disposal. When a typical car breaks down, its steel can be melted and used to construct another car (or building, or anything else made of steel). Carbon fiber can't be melted down, and it's not easy to recycle. When it is recycled, the recycled carbon fiber isn't as strong as it was before recycling. As it stands now, carbon fiber could use as impact attenuator material. It's lightweight, durable and safe. But it's also expensive and difficult to recycle.

3.4.3 Stainless steel

Stainless steel composition:

Table 3.3: Material Composition of SS.

Iron	C	Cr	Mn	Ni
80	0.15	16	0-5	0-5

The properties of stainless steel:

- Density = 7750 Kg/m3.
- Tensile yield strength = 207 MPa.

- Ultimate tensile strength = 586 MPa.
- Young's modulus = 193 GPa.
- Shear Modulus = 73 GPa.

3.4.4 Aluminum Alloys

Aluminium alloys are alloys in which aluminium (Al) is the predominant metal. The typical alloying elements are copper, magnesium, manganese, silicon and zinc. There are two principal classifications, namely casting alloys and wrought alloys, both of which are further subdivided into the categories heat-treatable and non-heat-treatable. Aluminium alloys are widely used in engineering structures and components where light weight or corrosion resistance is required.

3.5 Properties of project material

Table 3.4: Material Property Comparison.

	SS alloy	Mild steel	Carbon fiber	Alumium alloy 6061
Young's Modulus	1.1e+005 Mpa	70 e3Mpa	1.3e+6 Mpa	7 e+4 Mpa
Poisson's Ratio	0.28	0.2	0.2	0.33
Density	7700 kg/m^3	7850 Kg/m^3	1600 Kg/m^3	2710 kg/m^3
Yield Strength	250Mpa	248Mpa	238Mpa	276 Mpa

Material / Factor Considered	Aluminium	Magnesium Alloys	Steel C1008 (Sheet metal wheel)	Carbon Fiber	Forged steel
Weight	Light	Medium	Heavier than Mg & Al	Light	Heavier than Mg & Al
Appearance	Pleasant	Pleasant	Bright	Somewhat dull	----------
Cost of Hub	Costlier	Costlier	Cheap	Costlier	Expensive
Material Cost	Costlier	Costlier	Cheap	Costlier	Cheap
Deformation under load	Too much deformation	Less as compared to Al	Less as compared to Al & Mg	----------	Less deformation than other materials.
Corrosion Resistance	Excellent	Excellent	Poor but improved by adding alloys.	Excellent	----------
Cast ability	Casted easily	Casted easily	Difficult to cast	----------	----------
Maintenance/Repair of Hub	Easy to repair	Can't repaired	----------	----------	----------
Effect on Unsprung Weight	Reduce	Reduce	More than the Al & Mg	----------	More than Al, Mg & Steel C1008
Heat Dissipation	Better	Best	Good	----------	Good
Durability	As equal to forged rim	Durable	----------	----------	Durable
Mechanical properties	Poor than Steel	Poor than Steel	As good as forged steel.		Better than forged wheels
Number of Piece in a Hub	If casted single piece & Sheet metal multi piece	Generally casted hence single piece	Generally two piece.	----------	Single piece
Flammability	Not Flammable	Flammable	Not Flammable	Flammable	Not Flammable
Driving comfort	--------------	--------------	--------------	------------	--------------
Road Handling	--------------	--------------	--------------	------------	--------------
Ground contact	Regain Easily	Regain Easily	Good ground contact for light weight rims.	Better ground contact.	Poor Ground contact.

Fig. 3.1: Material Comparison with different aspects.

3.6 Why We Use S.S instead of C.I-

Iron is a hard grey metal, and heavier than any of the other elements found on Earth. During a process, impurities or slag is removed from iron, and it is turned into a steel alloy. This confirms that steel is an alloy, whereas iron is an element. Iron exists in natural forms, and scientists have found it in meteorite rocks as well. The main difference between the two elements is that steel is produced from iron ore and scrap metals, and is called an alloy of iron, with controlled carbon. Whereas, around 4% of carbon in iron makes it cast iron, and less than 2% of carbon makes it steel.

Cast iron is cheaper than steel, and it has a low melting point with an ability to mold into any form or shape because it does not shrink when it gets cold. Steel is made with a controlled amount of carbon, whereas cast iron can have any amount of carbon. Carbons and other metals like chromium are added to the iron to make alloys and different qualities or grades of steel, such as stainless steel.

Other difference between steel and cast iron is their properties. Properties of steel is that it is mild, harder to cast and has a relatively high viscosity. The properties of cast iron is that it is brittle, more damping and absorbs vibration and noises. In a molten form, cast iron is sufficient enough to making casts of any kind, from components of various machines to intricate shapes, such as cast iron or wrought iron furniture or gates. It is slightly destructive upon drilling, produces powder, and does not bend or dent because it is very hard, but it breaks easily – unlike steel. Steel produces chips if it is grinded, and it is malleable. The strength of both cast iron and steel is also controversial, as some think steel is stronger than cast iron and others think that iron and steel are same thing, but the truth is that cast iron has a more compressive strength, and steel is more tensile. If compared with cast iron, steel is superior in tension, and does not rust.

Cast iron and steel are used as construction materials, and are used to make structures for buildings. Steel is used to make beams, doors etc. Cast iron has been used to make pipelines and guttering in the past. It is still used for making manhole covers; cylinder blocks in the engines of cars, and for very heavy and expensive cooking utensils, besides its other uses as a construction material. Steel is preferred by the automobile industry to make steel parts and components, and it is used in various other industries to make tools, knives, frames, nails etc.

3.7 Summary

1. Steel is an alloy or iron, and cast iron is a hard grey metal.
2. Cast iron is cheaper than steel, and has a low melting point with the ability to mold easily.
3. Steel is mild and harder to cast, and loses viscosity.

Calculations

4.1 Loading on Wheel Hub

Wheel hub is under breaking torque, Bumping force and cornering load in various uprating condition of vehicle.

4.1.1 Braking Torque

When brake is applied breaking force will applied on the disc which resist the rotation of wheel, ultimately the hub. That torque is directly passes through the wheel hub as the hub and break disc are firmly bolted to each other. The Disc Brake block represents a brake arranged as a cylinder applying pressure to one or more pads that can contact the shaft rotor. Pressure from the cylinder causes the pads to exert friction torque on the shaft. The friction torque resists shaft rotation.

Disc Brake Model

This figure shows the side and front views of a disc brake.

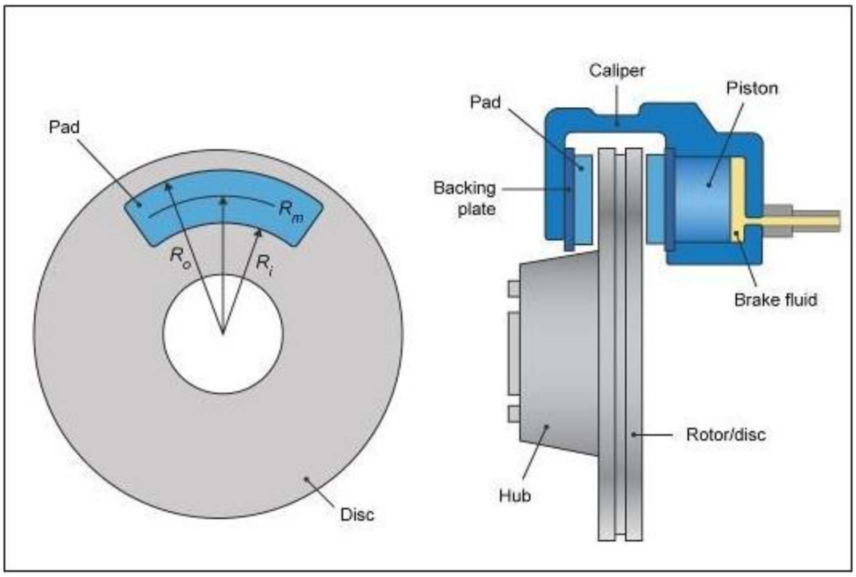


Fig. 4.1: Disc brake model.

A disc brake convert's brake cylinder pressure from the brake cylinder into force. The disc brake applies the force at the brake pad mean radius.

The equation that the block uses to calculate brake torque, depends on the wheel speed, Ω, such that when Ω≠0,

$$T = \frac{\mu_k P \pi {D_b}^2 R_m N}{4}.$$

However, when Ω=0, the torque applied by the brake is equal to the torque that is applied externally for wheel rotation. The maximum value of the torque that the brake can apply when Ω=0, is

$$T = \frac{\mu_s P \pi {D_b}^2 R_m N}{4}.$$

In any case,

$$Rm = \frac{Ro + Ri}{2}.$$

In the equations

- T is the brake torque.
- P is the applied brake pressure.
- Ω is the wheel speed.
- N is the number of brake pads in disc brake assembly.
- µs is the disc pad-rotor coefficient of static friction.
- µk is the disc pad-rotor coefficient of kinetic friction.
- Db is the brake actuator bore diameter.
- Rm is the mean radius of brake pad force application on brake rotor.
- Ro is the outer radius of brake pad.
- Ri is the inner radius of brake pad.

4.1.2 Cornering Force

When a car is in a steady state turn with constant speed on banking, the load is transferred from the inside to the outside pair of wheels. The cornering force is depend on the speed V, the radius of the bend R and the total weight of the vehicle. This cornering force is calculated by formula,

$F = m\ v^2/r$

Where,

- V is Vehicle speed.
- R is radius of turning.
- M is mass of vehicle.

4.1.3 Bump Force

Similarly, vehicle be capable to take the bumping force from road. In Off road terrains condition due to rough road surface or uneven road condition the bump force is acting. This bump is transfer to the wheel through tyre and hub. This bumping will applies in vertical direction.

Normally its magnitude is considered as 3g of weight of vehicle.When vehicle comes across bump, vertical force acts at the contact of road and tyre. This force is greater than the static force i.e. force due to self-weight, due dynamic nature of suspension system. This force is taken as three times of gravitational acceleration for our analysis. As front to rear weight distribution weight is considered as 50:50

From Newton's second law of motion: $F = m \times 3a$

Where,

- M is mass of Vehicle.
- A is Acceleration.

4.2 Load Calculations

Below are the forces acting on the hub while driving the vehicle.

4.2.1 The Vehicle at the instant of braking:

The braking force transferred to wheel hub is calculated as below, Brake pedal force:

1. The Brake applied on the pedal is assumed to be 300N (30.6 kgf)
2. Pedal ratio of every 4 wheeler is 6:1
3. fmax = force × pedal ratio

= 300 ×6

= 1800N

Where, fmax = force applied onto the master cylinder)

Hence, $P = \frac{fmax}{(\frac{\pi}{4}) \times d^2}$

(P=hydrostatic pressure, d=diameter of master cylinder's piston)

Fmax = P × $^{\pi}$ ×D2 [by Pascal's Law] 4

(Fmax = force acting on each Cylinder, D = diameter of the piston Cylinder in the caliper)

By Solving,

4. Fmax = fmax × (^{D})2 d

= (1800) × (0.03 / .019)2

= 4487.53 N

Torque acting on the disc:

T = Fmax × μ × Re × number of pistons per caliper

= 4487.5346 ×0.3 × 0.097 ×3

= 391.76N-m

μ = Coefficient of friction between brake pad and disc (0.3) Re = Radius of the disc.

4.2.2 The Vehicle at the instant of Bump:

When vehicle comes across bump, vertical force acts at the contact of road and tyre. This force is greater than the static force i.e. force due to self-weight, due dynamic nature of suspension system. This force is taken as three times of gravitational acceleration for our analysis. As front to rear weight distribution is 50:50.

Max velocity of Vehicle =156 kmph. Mass of the vehicle=1650kg

From Newton's second law of motion:

F = m × 3a Here,

= 1650 × 9.81× 3

= 65481.75 ^{kgm} (N)s

F= 48559.5N

Force Applied on each Wheel

= F 4

= 48559.54

F = 12139.8 N

4.2.3 The Vehicle at the instant of Cornering: When a car is in a steady state turn with constant speed on banking, the load is transferred from the inside to the outside pair of wheels. The cornering force which results from the speed V, the radius of the bend R and the total weight of the vehicle is:

m = 50 % 1650 kg

m = 1650/2 kg

r = turning radius = 10 m v = 35kmph

= 35 × 1000/3600

= 9.7 m/s Cornering Force: F= m v2/r

= (1650/2) × (9.7)2 10

Total force = 7762.42N

Braking torque = 391.76 N-m Bumping Force = 12139.8N Cornering Force = 7762.42 N

5.1 CATIA V5

5. CAD MODEL DESIGN

CATIA (Computer Aided Three-dimensional Interactive Application) is a multiplatform CAD/CAM/CAE commercial software suite developed by the French company Dassault System In our case, CATIA was used for a design a virtual "Wheel Hub". The pieces were design with the most similar real dimensions. When CATIA design was finished, the new files were moved files to ANSYS.

5.2 CATIA model Building of wheel hub

The baseline wheel hub model is prepared in CATIA using sketcher and part design. Workbench is used to analysis of the same. For the modeling we have taken the reference of wheel hub of Mahindra TUV. We have done reverse engineering of Mahindra TUV wheel hub. All dimensions are measured from actual model and prepared the model in CATIA.

The model prepared in CATIA is as shown in the below figure.

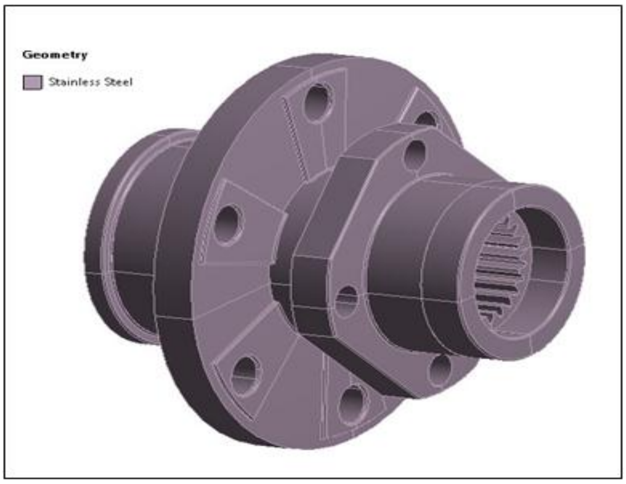


Fig. 5.1: Cad Model Wheel Hub.

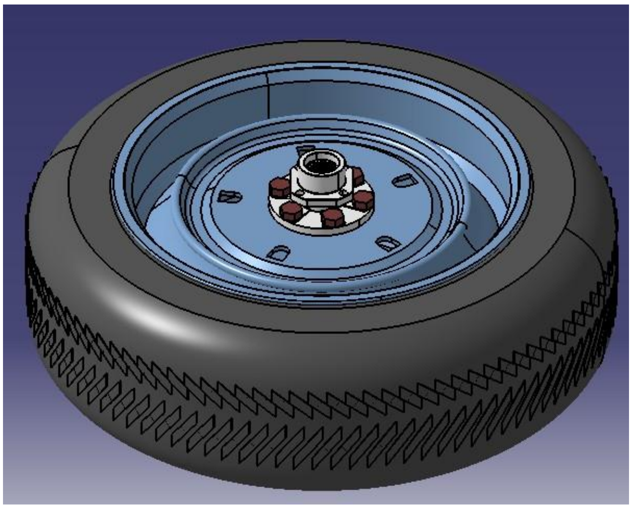


Fig. 5.2: Cad Model Wheel Assembly.

Similarly the model prepared is imported in the Ansys. Loading conditions for ansys analysis is taken as per the calculations done before. These loading conditions are most realistic one.

ANSYS Workbench

6. Finite Element Analysis

As solvers become faster and computers more powerful, the solution portion of finite element analysis shortens and a larger portion of overall simulation time is spent on pre-processing, including generating the mesh-the fundamental element-based representation of parts to be analyzed. Recognizing this trend, ANSYS, Inc. has addressed the need for faster and more reliable structural meshing with new technologies in ANSYS Workbench Simulation11.0 (also known as ANSYS Workbench Meshing 11.0). These new capabilities result in very robust meshing and save considerable amounts of time (especially for complex geometries) with features that automate many routine tasks while providing users high levels of control of their model. The ANSYS Workbench platform can generate meshes for structural, thermal, electromagnetic, explicit dynamics or computational fluid dynamics (CFD) analyses, but the meshing considerations vary for each. For example, lower order elements with a finer mesh density tend to be used in CFD analyses, whereas higher-order elements with a coarser mesh density may be preferred in structural analyses.

The design of products that need to survive impacts or short-duration high-pressure loadings can be greatly improved with the use of ANSYS explicit dynamics solutions. These specialized problems require advanced analysis tools to accurately predict the effect of design considerations on product response to sever readings. Understanding such complex phenomenon is especial important when it is too expensive or impossible to perform physical testing.

The ANSYS explicit dynamics product suite helps you gain insight into the physics of short-duration events for products that undergo highly nonlinear, transient dynamic events. These specialized, accurate and easy to use tools have been designed to maximize productivity.

Finite Element Method is a numerical procedure for solving continuum mechanics of problem with accuracy acceptable to engineers. Finite Element Method is a mathematical modeling tool involving discretization of a continuous domain? Using building-block entities called finite elements connected to each other by nodes for once and moment transfer. This process includes Finite Element Modeling and Finite Element Analysis. In displacement based FEM, stiffness of the entire structure (Part or assembly) is assembled from stiffness of individual elements. Loads and boundary conditions are applied at the nodes and the resulting sets of the simultaneous equations are solved using matrix methods and numerical techniques. In short, FEM is a numerical method to solve ordinary differential equations of equilibrium.

Starting with simple linear static stress and heat transfer analysis, Complex simulations involving highly non-linear, fluid flow and dynamic events can be successfully analyzed on a personal computer using a host of popular software including Cosmos, NASTRAN, ANSYS and NISA among others.

6.1 Steps In The Finite Element Calculation

6.1.1 Pre-Processing

Create and discrete the solution domain into finite elements. This involves dividing the domain into sub-domains, called 'elements', and selecting points, called nodes, on the inter-element boundaries or in the interior of the elements.

- Assume a function to represent the behaviour of the element. This function is approximate and continuous and is called the "shape function".
- Develop equations for an element.
- Assemble the elements to represent the complete problem.
- Apply boundary conditions, initial conditions, and the loading.

6.1.2 Solving

Solve a set of linear or nonlinear algebraic equations simultaneously to obtain nodal results, such as displacement values or temperature values depending on the type of problem.

6.1.3 Post-Processing

This stage involves processing the nodal data to get other information such as values of principal stresses, heat fluxes etc.

6.1.4 Building and Importing CAD model in ANSYS

The baseline Wheel hub model is prepared in CATIA using Sketcher and Part Design workbench. The model prepared in CATIA is as shown in the below figure

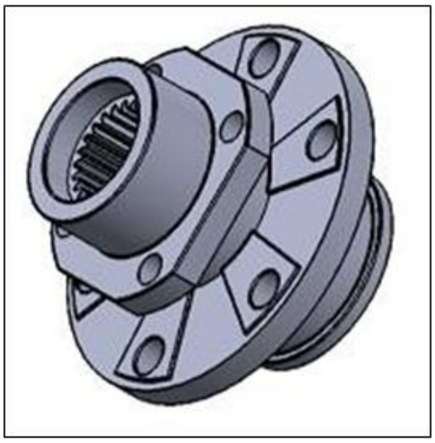


Fig. 6.1: Wheel hub CAD model in CATIA.

The CAT Part file is then exported in STEP format. The export of CAD model is done in CATIA. The same .STEP file is then imported in ANSYS workbench Design Modeller software. The STEP (Standard for the Exchange of Product model data) reader will both read and write model data to and from the STEP format. It is important to note that the STEP format does not store model data in the same way as the Design Modeller application. STEP format stores surface data, which upon import into the Design Modeller application is stitched together to form bodies. In some rare cases, the Design Modeller application model exported to STEP format may not produce the exact same geometry when imported again into the Design Modeller application. The STEP files are transferred as full solids in the ANSYS Mechanical application. Following figure shows the imported geometry.

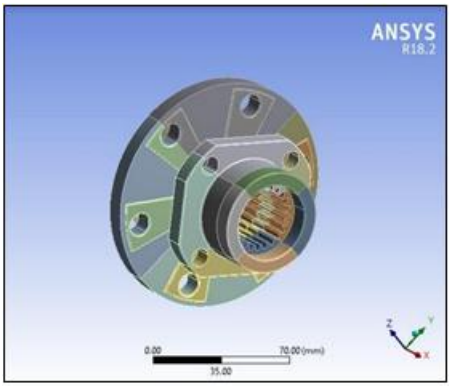


Fig. 6.2: CAD model imported in ANSYS.

6.1.5 Meshing

In the finite element analysis the basic concept is to analyze the structure, which is an Assemblage of discrete pieces called elements, which are connected, together at a finite Number of points called Nodes. Loading boundary conditions are then applied to these Elements and nodes. A network of these elements is known as Mesh. The maximum amount of time in a finite element analysis is spent on generating elements and nodal data. Preprocessor allows the user to generate nodes and elements automatically at the same time allowing control over size and number of elements. There are various types of elements that can be mapped or generated on various geometric entities.

The elements developed by various automatic element generation capabilities of preprocessor can be checked element characteristics that may need to be verified before the finite element analysis for connectivity, distortion-index etc.

Generally, automatic mesh generating capabilities of preprocessor are used rather than defining the nodes individually. If required nodes can be defined easily, by defining the allocations or by translating the existing nodes. Also on one can plot, delete, or search nodes.

The SOLID87 element is used to generate mesh of the disc. SOLID87 is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, temperature, at each node. The element is applicable to a 3-D, steady-state or transient thermal analysis. If the model containing this element is also to be analyzed structurally, the element should be replaced by the equivalent structural element (such as SOLID187). A 20- node thermal solid element, SOLID90, is also available. Mesh element of Solid 87 is shown in the below figure.

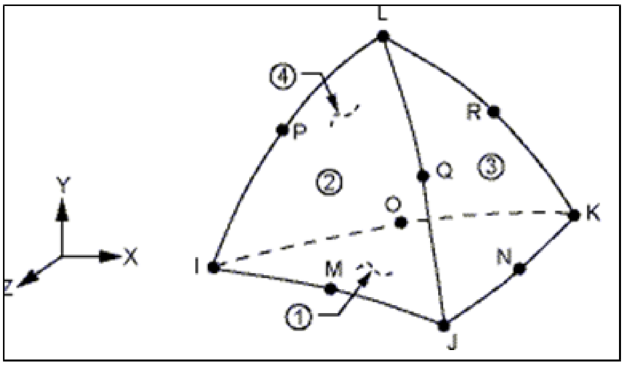


Fig. 6.3: Solid 87 elements.[3]

Meshed model of Wheel hub is as shown below in figure

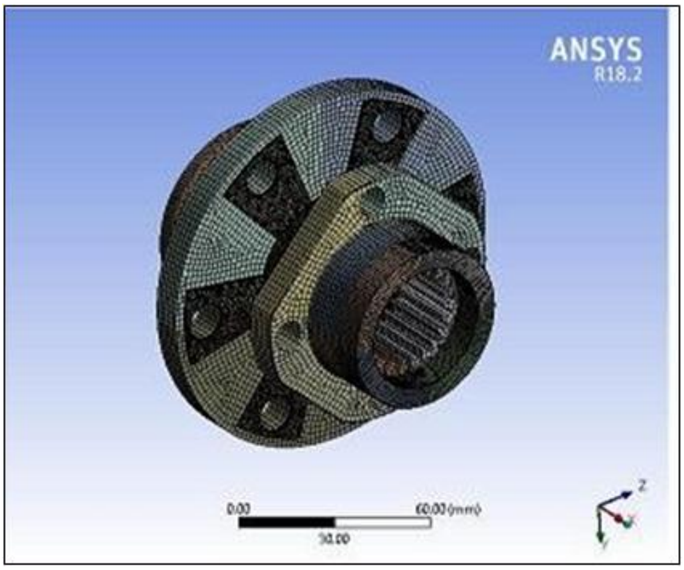


Fig. 6.4: Mesh model of wheel hub.

6.1.6 Loading and Boundary Conditions

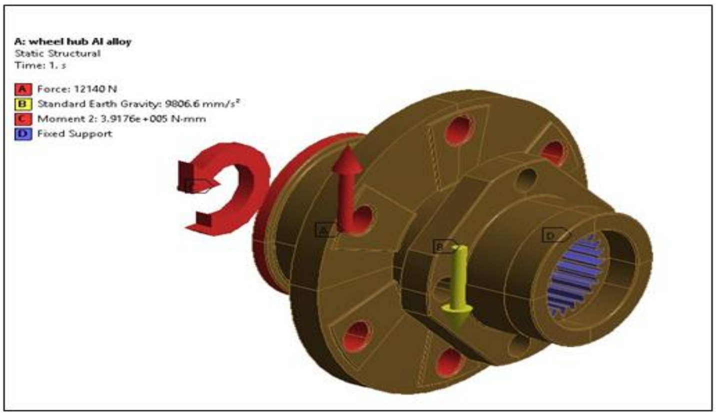


Fig. 6.5 boundary conditions of baseline wheel hub.

6.1.7 Solution

The solution phase deals with the solution of the problem according to the problem definitions. All the tedious work of formulating and assembling of matrices are done by the computer and finally displacements are stress values are given as output. Some of the capabilities of the ANSYS are linear static analysis, nonlinear static analysis, transient dynamic analysis, etc.

6.1.8 Post-Processing

It is a powerful user friendly post processing program using interactive color graphics. It has extensive plotting features for displaying the results obtained from the finite element analysis. One picture of the analysis results (i.e. the results in a visual form) can often reveal in seconds what would take an engineer hour to assess from a numerical output, say in tabular form. The engineer may also see the important aspects of the results that could be easily missed in a stack of numerical data.

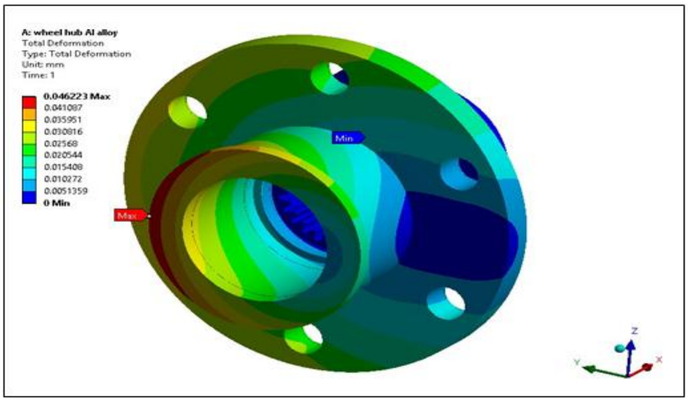


Fig. 6.6: Total Deformation in baseline.

Meshing of Wheel Hub

The basic idea of FEA is to make calculations at only limited (Finite) number of points and then interpolate the results for the entire domain (surface or volume). Any continuous object has infinite degrees of freedom and it's just not possible to solve the problem in this format. Finite Element Method reduces the degrees of freedom from infinite to finite with the help of discretization or meshing (nodes and elements). One of the purposes of meshing is to actually make the problem solvable using Finite Element. By meshing, you break up the domain into pieces, each piece representing an element. You need these elements to be able to apply Finite Element since Finite Element is all about having a basis local to an Element and stitching a bunch of local solutions together to build the global one. If you did not mesh and just assumed some basis that covered the whole domain that would be a Spectral Method. One other aspect of meshing is the accuracy of your solution.

We have imported the 3D Geometry in the ANSYS Design modeler to carry out geometry clean- up for mesh refinement in the critical areas of model.

Geometry is sliced so that the fine element size can be given to more critical areas. We have used second order SOLID186 element.

No of Nodes = 159192

No of Elements = 316008

Table 6.1 Mesh Quality check.

Quality Check	Acceptable Value	Achieved Value
Aspect Ratio	< 5.0	2.5231
Jacobian Ratio	> 0.5	2.7201
Skewness	< 0.7	0.2782
Element Quality	> 0.1	0.7773

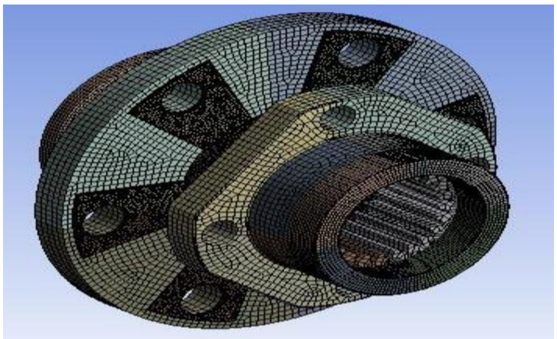


Fig. 6.7: Mesh model of wheel hub.

6.2 FEA Analysis of Wheel Hub

Based on calculation simulation is done with two combinations of load sets.

1. Combine effect of braking torque and bumping force.
2. Combine effect of braking torque and cornering force

Simulation is done for different material we have selected earlier. Maximum stress on the hub for each material is founded out by using FEA analysis. Below are the result set of maximum stress, deflection and factor of safety for each material is attached.

6.2.1 Analysis for Combine effect of braking torque and bumping force

Boundary conditions

1. Fixity of internal spline in all 6 degree of freedom.
2. Gravitational load in downward direction.
3. Braking torque of 391.76 N-m on hub collar.
4. Bumping Force of 12139.8N in vertical direction at wheel mounting holes of hub.

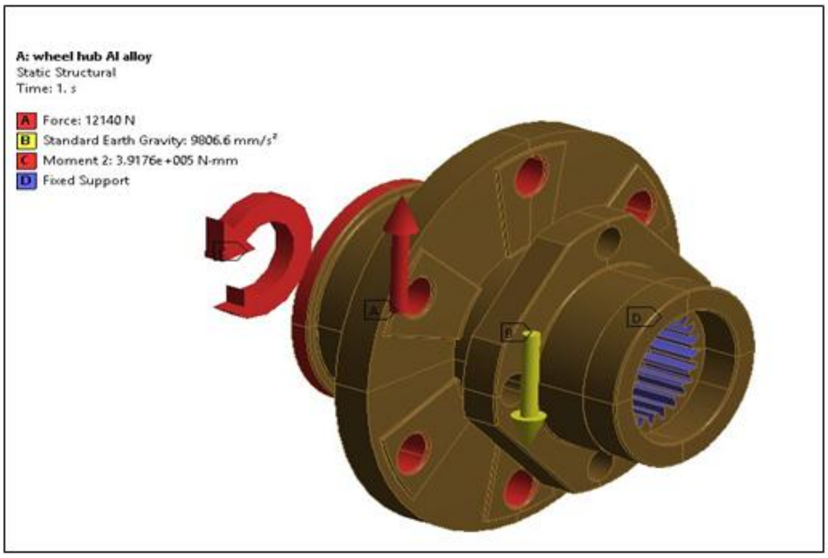


Fig.6.8: Boundary condition (Torque + Bump).

Results: Stress level and maximum deformation spectrum is traced for different material of hub as below.

- Aluminum alloy Stress level

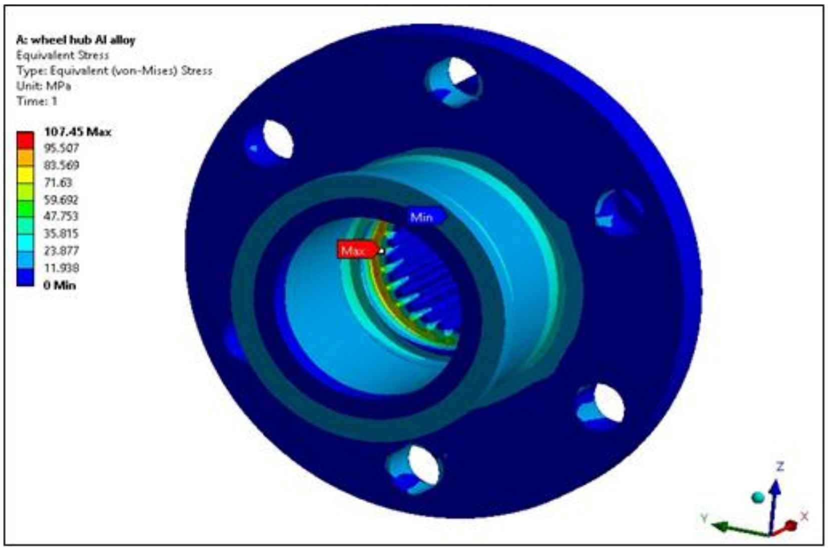


Fig. 6.9: Von-Misses Stress Contour plot for Al.

Deformation

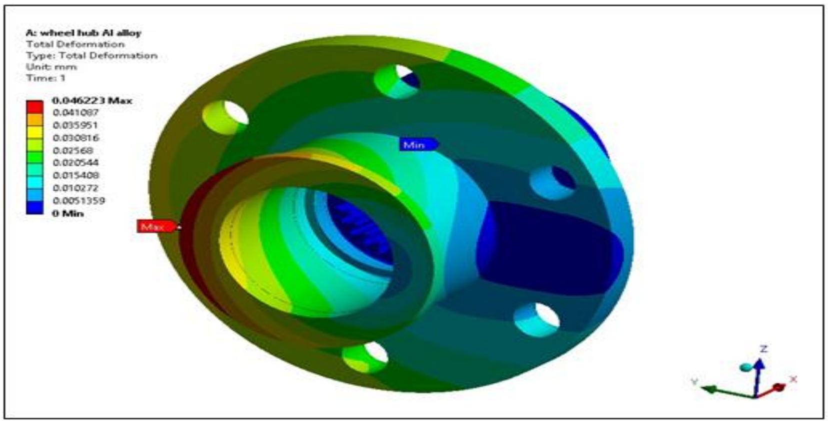


Fig. 6.3.3: Deformation Contour plot for Al.

Stainless steel Stress level

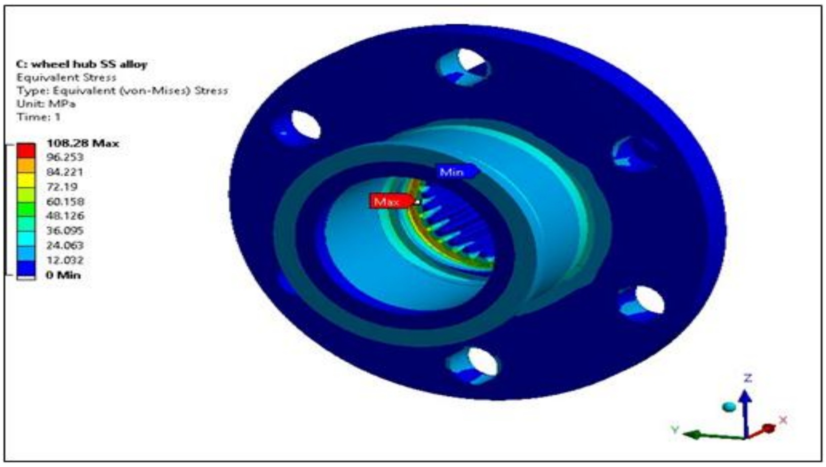


Fig. 6.10: Von-Misses Stress Contour plot for SS.

Deformation

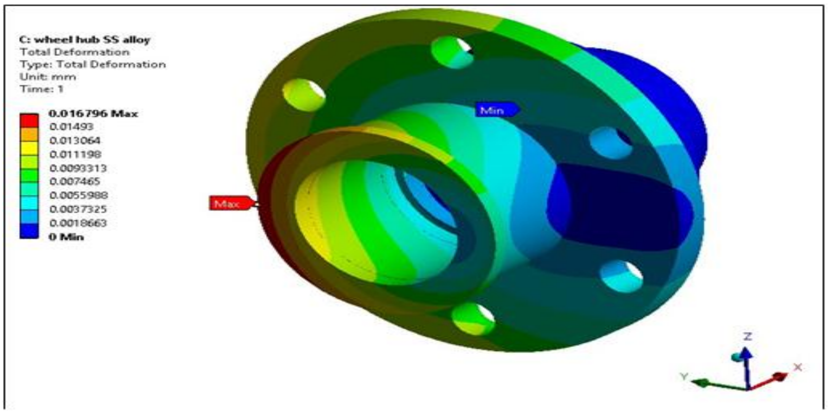


Fig. 6.11: Deformation Contour plot for SS.

MS

Stress level

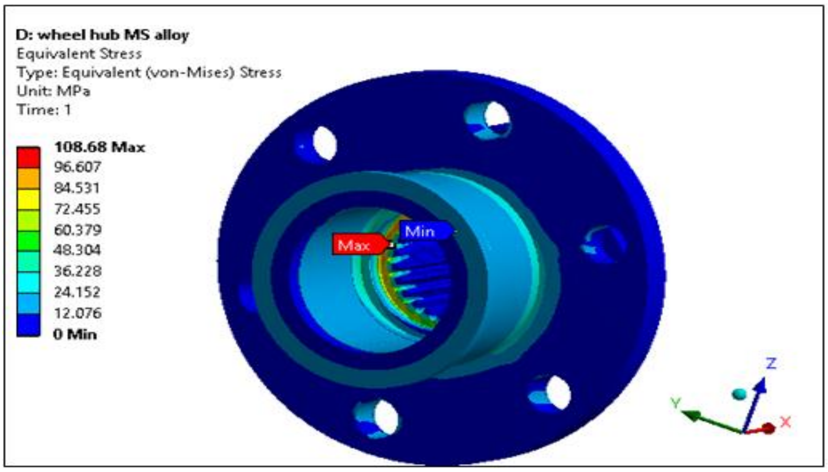


Fig. 6.12: Von-Misses Stress Contour plot for MS.

Deformation

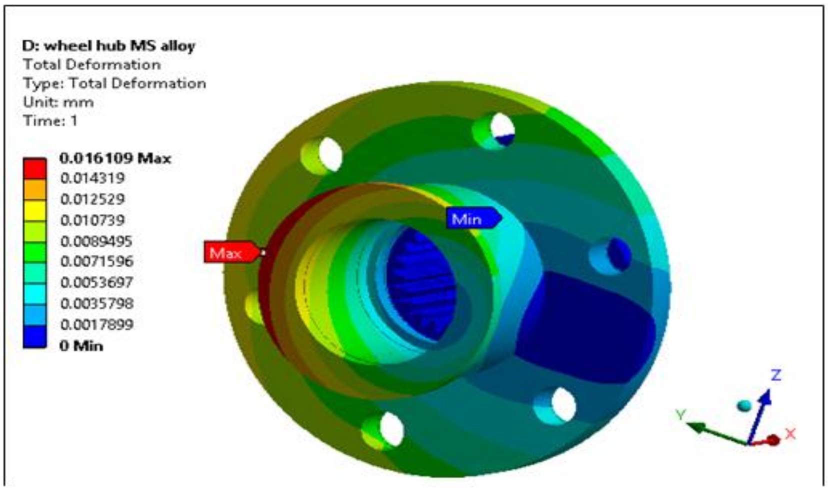


Fig.6.13: Deformation Contour plot for MS.

Carbon Fiber Stress level

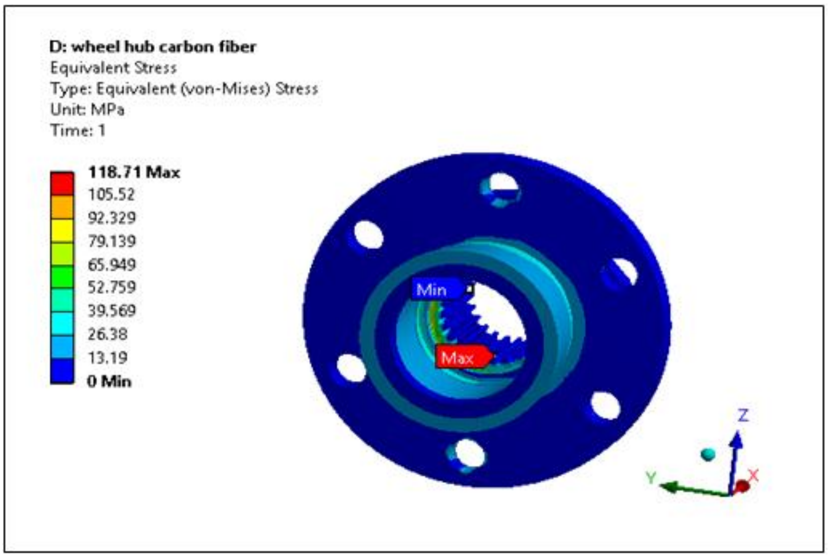


Fig. 6.14: Von-Misses Stress Contour plot for Carbon Fiber.

Deformation

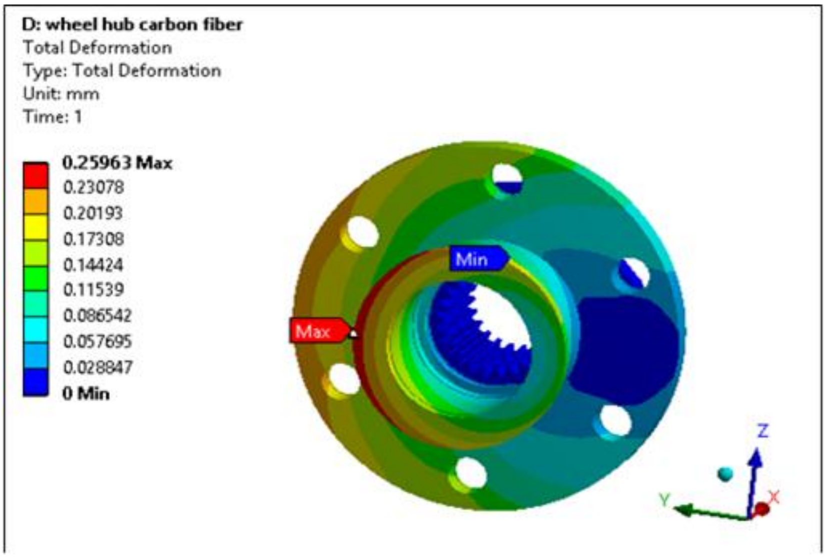


Fig. 6.15: Deformation Contour plot for Carbon Fiber.

Results for Combine effect of braking torque and cornering force

Boundary conditions

1. Fixity of internal spline in all 6 degree of freedom.
2. Gravitational load in downward direction.
3. Braking torque of 391.76 N-m on hub collar.
4. Cornering Force of 7768 N in couple form at top and bottom wheel mounting holes of hub.

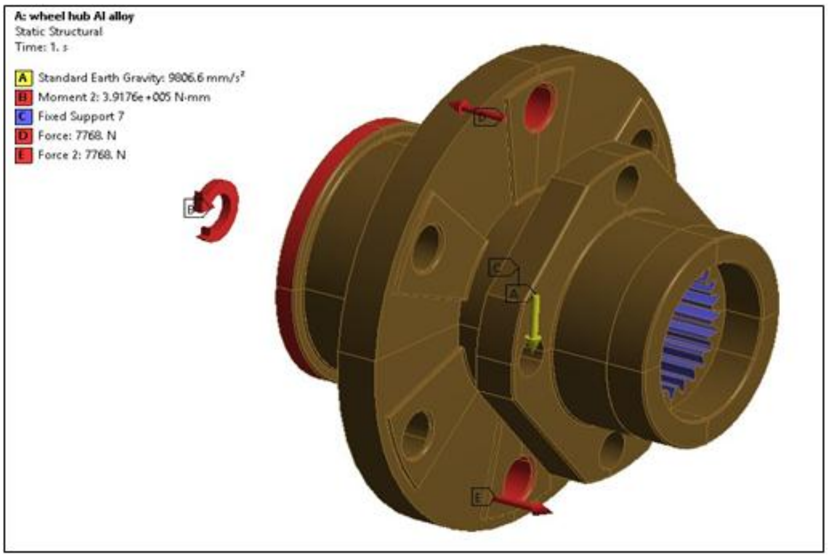


Fig.6.16 Boundary condition (Torque + Cornering Force).

RESULTS

Stress level and maximum deformation spectrum is traced for different material of hub as below.

- Aluminum alloy Stress level

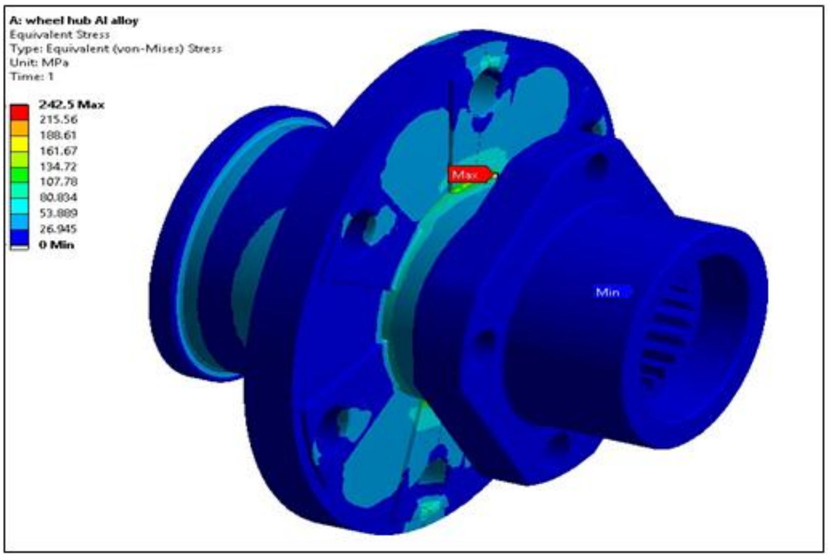


Fig. 6.17: Von-Misses Stress Contour plot for Al.

Deformation

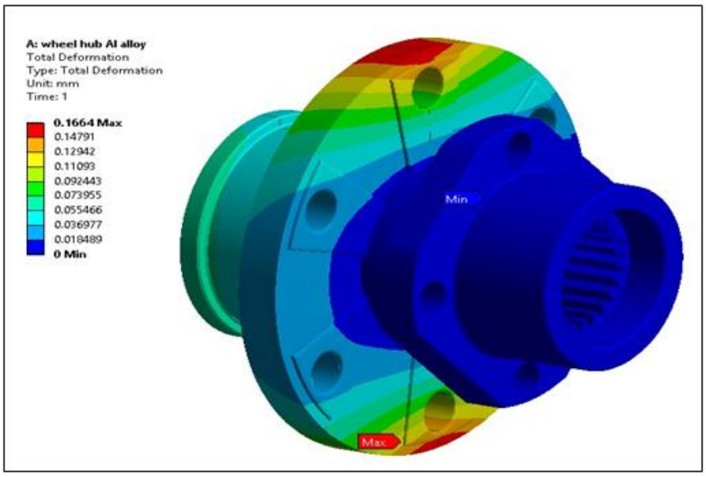


Fig.6.18 Deformation Contour plot for Al.

Stainless steel Stress level

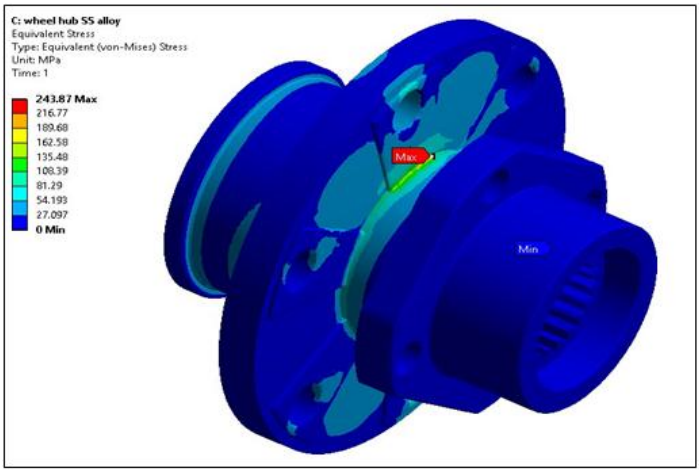


Fig.6.19: Von-Misses Stress Contour plot for SS.

Deformation

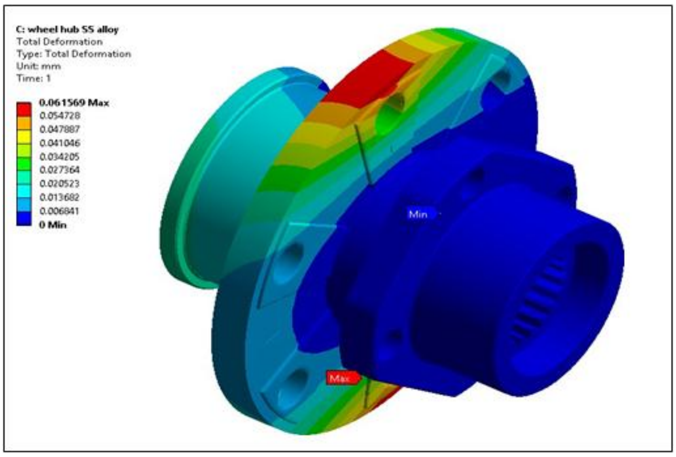


Fig.6.20 Deformation Contour plot for SS

MS Stress level

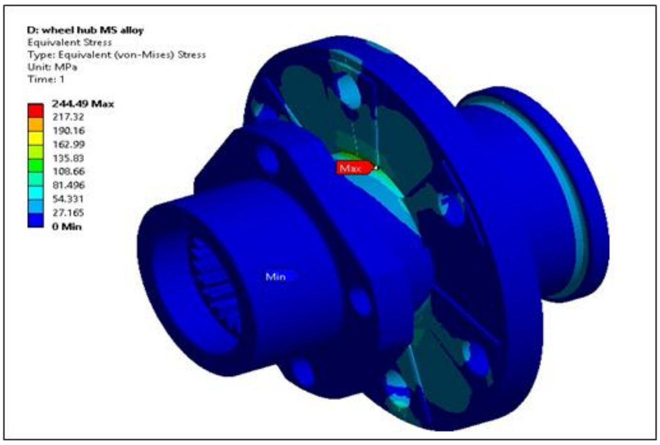


Fig. 6.21: Von-Misses Stress Contour plot for MS.

Deformation

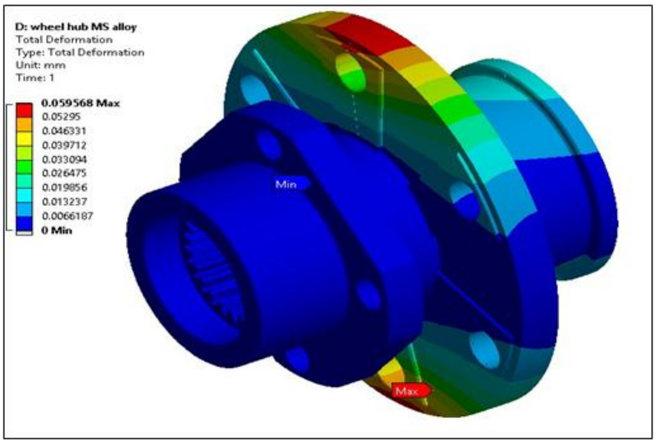


Fig. 6.22: Deformation Contour plot for MS.

- **Epoxy Carbon Stress level**

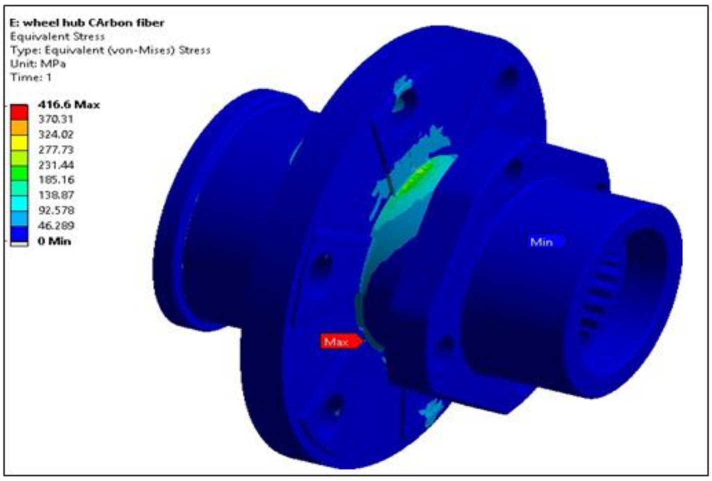


Fig. 6.23: Von-Misses Stress Contour plot for Carbon Fiber.

Deformation

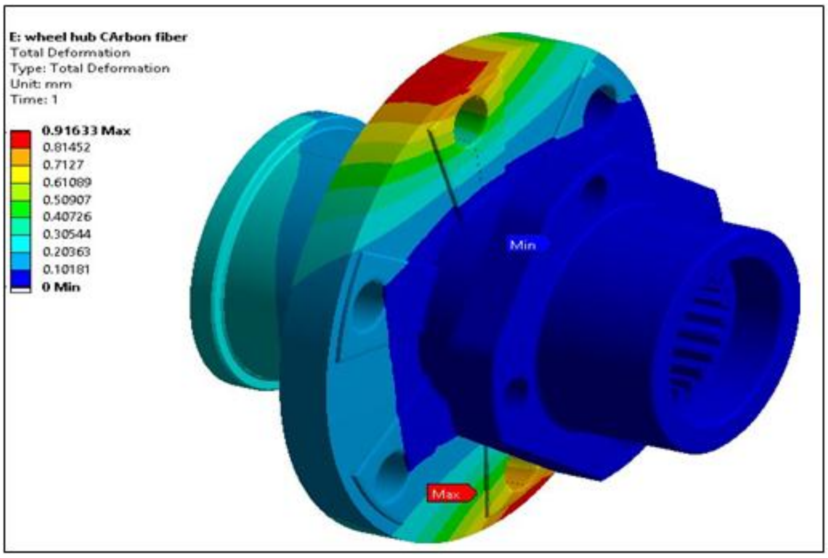


Fig.6.24: Deformation Contour plot for Carbon Fiber.

RESULT AND DISCUSSION

FEA results for various samples for 1st load case i.e. Results for Combine effect of braking torque and bumping force.

Table 7.1: Results (Load Case 1).

Sr. No	Description	Mass (Kg)	Deflection (mm)	Von-Mises stress (Mpa)	FOS	Yield strength of material (Mpa)
1	CF	0.347	0.2596	118.71	2.01	238
2	Mild steel	1.840	0.0161	108.68	2.28	248
3	SS 304	1.804	0.0167	108.28	2.31	250
4	Al alloy	0.587	0.0462	107.4	2.5	276

FEA results for various samples for 2nd load case i.e. Results for Combine effect of braking torque and cornering force.

Table 7 2: Results (Load Case 2).

Sr. No	Description	Mass (Kg)	Deflection (mm)	Von-Mises stress (Mpa)	FOS	Yield strength of material (Mpa)
1	CF	0.347	0.9160	416.60	0.57	238
2	Mild steel	1.840	0.0595	244.49	1.01	248
3	SS 304	1.804	0.0615	243.87	1.02	250
4	Al alloy	0.587	0.1664	245.50	1.12	276

By looking after above readings, it is cleared that for first case i.e. for Combine effect of braking torque and bumping force the factor of safety of all selected material is more than one. So all material are safe to take a combine load of braking torque and bump force. These materials are absolutely safe for first load case (Braking Torque + Bumping force) I second Load case i.e. for combine effect of braking torque and cornering force, factor of safety for carbon fiber is less than one means it is not safer material for hub. But else other material except carbon fiber have factor of safety is more than one. Means these are safe to use for manufacturing of wheel hub.

7. TESTING AND VALIDATON

Work piece is tested under compressive loading for MS and aluminum alloy. Deflection verses forces reading are traced. Behavior of work piece under continuous compression is tested on test rig and traced. Universal testing machine UTM is used for compressive testing. Similarly one analysis deck is prepared with same loading condition with realistic boundary condition as per the testing is done. Graph for defection force is traced for simulation also. Graph of deflection verses force for test is compared with simulation. Similarity in nature of graph for test and simulation indicates the validation of simulation results against test results.

8.1 Testing of MS

Test set up: Loading of MS hub On UTM machine

Fig. 8.1: Loading of specimen (MS Hub).

Result of testing: force verses deflection reading for MS hub on the test rig is measured and showed below. Graph for force verses deflection for MS is as below.

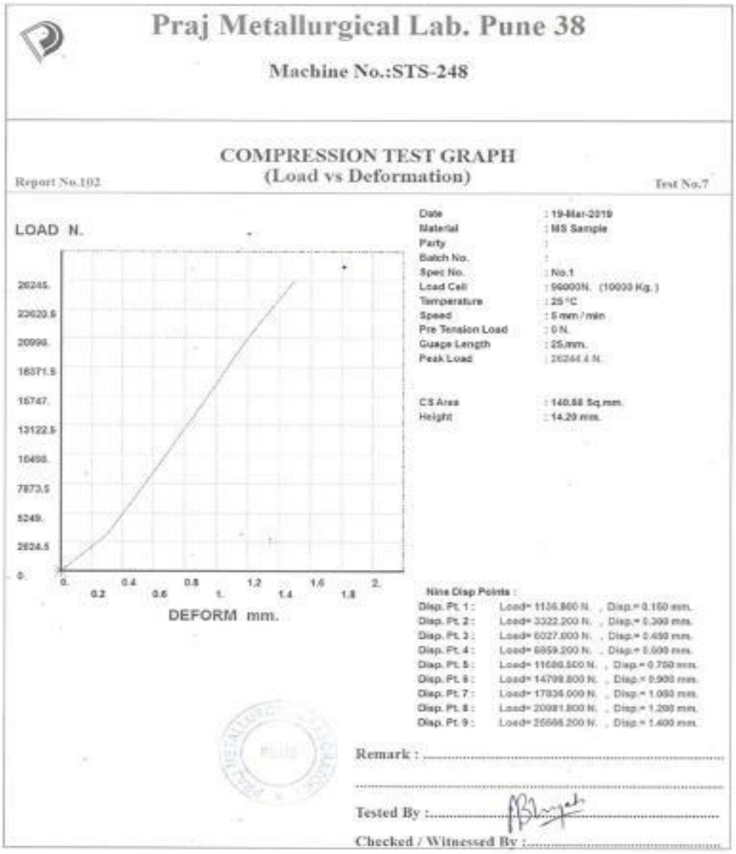


Fig. 8.2: Test Results (Load Vs Deflection) for MS hub.

Simulation Deck preparation as per test loading condition: simulation deck is prepared for wheel hub with loading similar to the test rig loading. The deflection for different loading is calculated and compared. The table for deflection by simulation and testing for similar load case is as shown below.

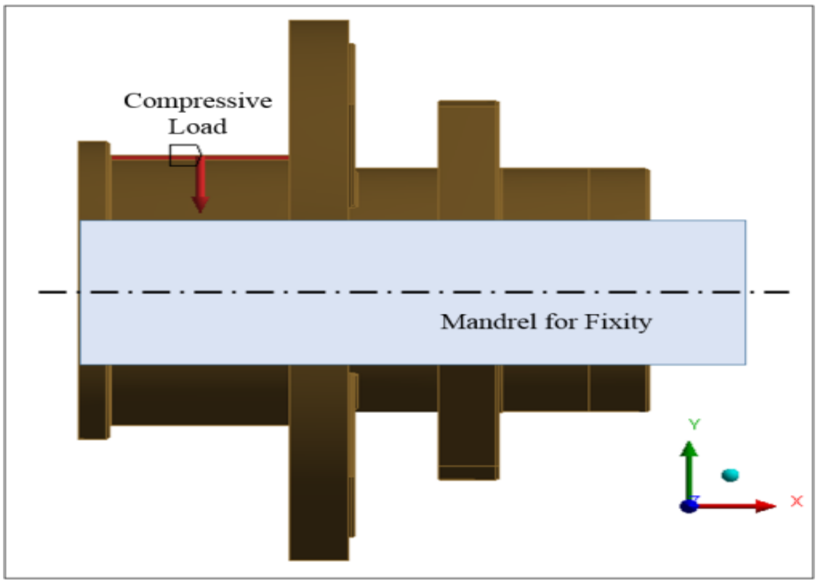
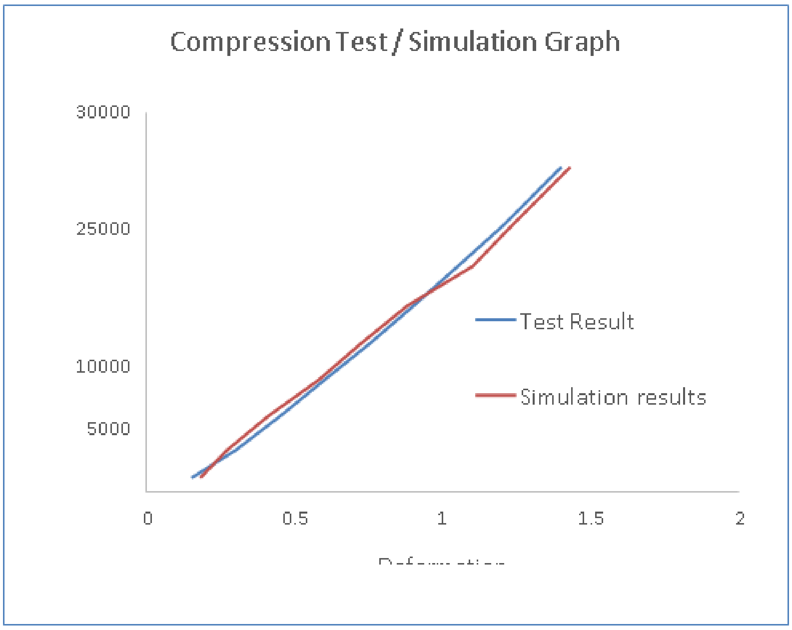


Fig 8.3: Simulation deck for MS hub testing.

Table 8.1: Results Comparison (Test and Simulation- MS hub).

Sr.No	Load (N)	Deflection (mm)	
		Test Result	Simulation results
1	1136.8	0.15	0.18
2	3322.2	0.3	0.27
3	6027	0.45	0.41
4	8859.2	0.6	0.58
5	11686.5	0.75	0.72
6	14709.8	0.9	0.88
7	17836	1.05	1.1
8	20981.8	1.2	1.23
9	25666.2	1.4	1.43



Graph 8.1: Deformation Vs Load (MS Hub – Test and Simulation).

Testing of Aluminum alloy

Test set up: Loading of Aluminum hub On UTM machine is done same as MS wheel hub mounting.

Result of testing: force verses deflection reading for aluminum hub on the test rig is measured and showed below. Graph for force verses deflection for aluminum is as below.

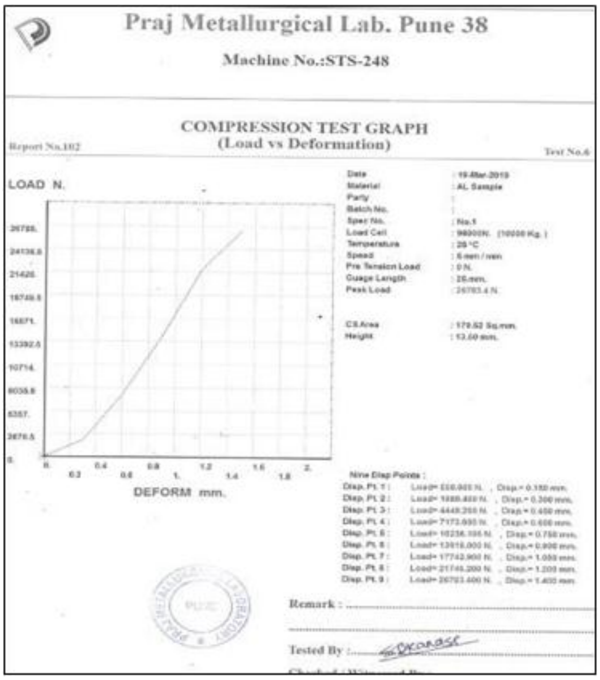
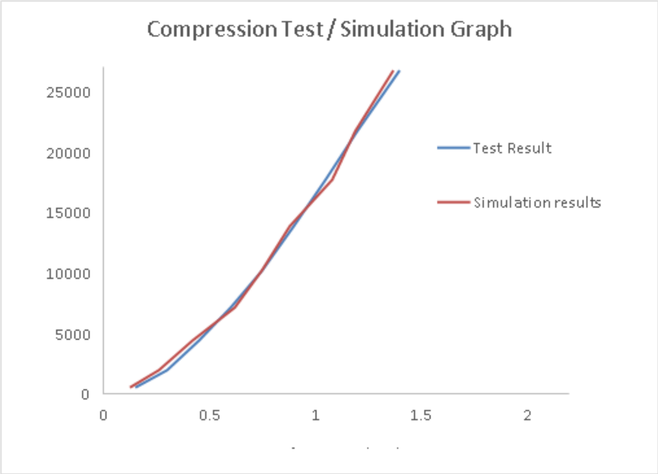


Fig. 8.5: Test Results (Load Vs Deflection) for Al Alloy hub.

Simulation Deck preparation as per test loading condition: simulation deck is prepared for wheel hub with loading similar to the test rig loading. The deflection for different loading is calculated and compared. The table for deflection by simulation and testing for similar load case is as shown below.

Table 8.2: Results Comparison (Test and Simulation- Al Alloy hub).

Sr.No	Load (N)	Deflection (mm)	
		Test Result	Simulation results
1	558.6	0.15	0.124
2	1989.4	0.3	0.26
3	4449.2	0.45	0.42
4	7173.6	0.6	0.62
5	10236.1	0.75	0.7478
6	13916	0.9	0.88
7	17742	1.05	1.08
8	21746	1.2	1.19
9	26783	1.4	1.37



Graph 8.3: Deformation Vs Load (Al Alloy Hub – Test and Simulation).

Hence it is proved that whatever simulation we have done are realistic as that of actual testing. We can refer the simulation results for conclusion purpose.

8. CONCLUSION

It is observed from the result, maximum stress are developed in the various type of material due to combination of load cases, Bump + Breaking torque and Cornering load + Breaking torque exactly matches with the field failure.

A comparison of maximum stress values and deformation is done for all tabulated material. By comparing Ansys result and FOS results and weight to strength ratio for all material, Aluminum alloy is better material. It can be used for modified wheel hub design and analysis as optimum material.

Table 9. 1: Results Table.

Sr. No	Description	Mass (Kg)	Load Case 1 Braking torque + bumping force.		Load Case 2 Braking torque + cornering force.	
			Von-Mises stress (Mpa)	FOS	Von-Mises stress (Mpa)	FOS
1	CF	0.347	118.71	2.01	416.60	0.57
2	Mild steel	1.840	108.68	2.28	244.49	1.01
3	SS 304	1.804	108.28	2.31	243.87	1.02
4	Al alloy	0.587	107.4	2.5	245.50	1.12

Table 9. 2: weight comparison.

	SS alloy	Mild steel	Carbon fiber	Aluminum alloy 6061
Density	7700 kg/m3	7850 Kg/m3	1600 Kg/m3	2710 kg/m3
Mass Of Hub	1.8049 Kg	1.8400 Kg	0.34701 Kg	0.5877 Kg

By comparing weight to strength ratio of above material, we can conclude that Aluminum alloy 6061 material is can be an option for the wheel hub. This material provides good weight reduction without compensating the strength of hub.

REFERANCES

1. Kaixian Yue, "Statistical Analysis of the Accelerated Life Testing for Fighter-Bomber Wheel Hubs", 1-4244-2509-9/09/$20.00 ©2009 IEEE.
2. M. Amura, L. Allegrucci, F. De Paolis, M. Bernabei, "Fatigue fracture of an AMX aircraft main wheel", Science Direct Engineering Failure Analysis, 2013; 27: 194–202.
3. Kazem Reza Kashyzadeh, Mohammad Jafar Ostad-Ahmad-Ghorabi, Alireza Arghavan, "Fatigue life prediction of package of suspension automotive under random vibration based on road roughness", Mediterranean Journal of Modeling and Simulation Med. J. Model. Simul, 2015; 04: 037-050.
4. J. Janardhan, V. Ravi Kumar, R. Lalitha Narayana , "Radial Fatigue Analysis of An Alloy Wheel" , Int. Journal of Engineering Research and Applications www.ijera.com ISSN : 2248-9622, December 2014; 4(12): 253-258.
5. Malapati.M, K V S Karthik, "Fatigue based Design and Validation of Automobile Wheel Hub", International Journal of Engineering Technology Science and Research IJETSR www.ijetsr.com ISSN 2394 – 3386, November 2017; 4(11).
6. Mehmet Firat, Recep Kozan, Murat Ozsoy, O. Hamdi Mete, "Numerical modeling and simulation of wheel radial fatigue tests", Science Direct Engineering Failure Analysis, 2009; 16: 1533–1541.
7. Fabio M. Santiciolli, Riccardo Möller, Ivo Krause, Franco G. Dedini, "Simulation of the scenario of the biaxial wheel fatigue test", Advances in Engineering Software, 2017; 1–11.
8. Gowtham, A S Ranganathan, S Satish, S John Alexis, S Siva kumar, "Fatigue based design and analysis of wheel hub for Student formula car by Simulation Approach", IOP Conf. Series: Materials Science and Engineering, 2016; 149: 012128. doi:10.1088/1757-899X/149/1/012128.
9. Shi-Jian Luo, Ye-Tao Fu, Yu-Xiao Zhou, "Perceptual matching of shape design style between wheel hub and car type", Science Direct International Journal of Industrial Ergonomics, 2012; 42: 90e102.
10. https://en.wikipedia.org/wiki/Wheel_hub_motor.