iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 Issue: X Month of publication: October 2018

DOI:

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue X, Oct 2018- Available at www.ijraset.com

Stress Analysis and Optimization of Front Wheel Hub

Bhushan Mohite
Mechanical Engineering Department, College of Engineering Pune (COEP), Pune University

Abstract: This paper describes weight reduction of front wheel hub of COEP SUPRA SAE Formula car. The objectives of this paper are to develop structural modeling, finite element analysis, optimization and final design validation of the wheel hub. Topology optimization technique is used for performing optimization of wheel hub. It helps in the reduction of unwanted material and thereby mass of the component decreases respectively. The structure of wheel hub was modeled utilized CATIA V5 software and analysis was performed using ANSYS Workbench software. The loading conditions were analyzed and Static analysis was carried out for finding the stress and deformation results. Optimization was performed with minimize mass as objective function. Based on the obtained FEA result, an optimized and enhanced design for high strength – low weight was obtained. The optimized wheel hub is 34.9 % lighter compared to initial design. Through this project, we have arrived at an optimized design of a front wheel hub, that is both strong and light weight with reduced material for best performance.
Keywords: CAD Model, Front wheel hub, Finite Element Analysis, ANSYS, CATIA, Topology Optimization.

I. INTRODUCTION

A. Necessity

Typical in high performance automotive and aerospace applications there is demand for reduced vehicle mass while maintaining adequate performance and safety.

SUPRA Formula SAE competition is no exception. The nature of the autocross style course favors vehicles with good acceleration capabilities and both the fuel economy and acceleration events add to the desire for a lightweight vehicle design. To increase the acceleration potential of a vehicle, or race car in this case, physics tells us that the traction force needs to be increased and/or the mass decreased.

Overall, as long as the reduced mass does not pose any threats to the integrity of the design—be it aircraft or racecar—there are rarely any drawbacks associated with decreased vehicle mass .This paper has significance to the automotive industry. This paper is an example of optimization of design through FEA. The design procedure used here can be standardized for automated design process.

B. Wheel Hub

Wheel Hub is an automotive part used in most cars, passenger vehicles and light and heavy trucks. A wheel hub is a mounting assembly for the wheel of vehicle; it houses the wheel bearing as well as supports the lugs. It is located between the brake drums or discs and the drive axle. The wheel is mounted to the bolts of the wheel Hub. The function of wheel hub is to keep the wheel spinning freely on the bearing while keeping it attached to the vehicle. The wheel hub is the only part that actually holds the wheels to the car.

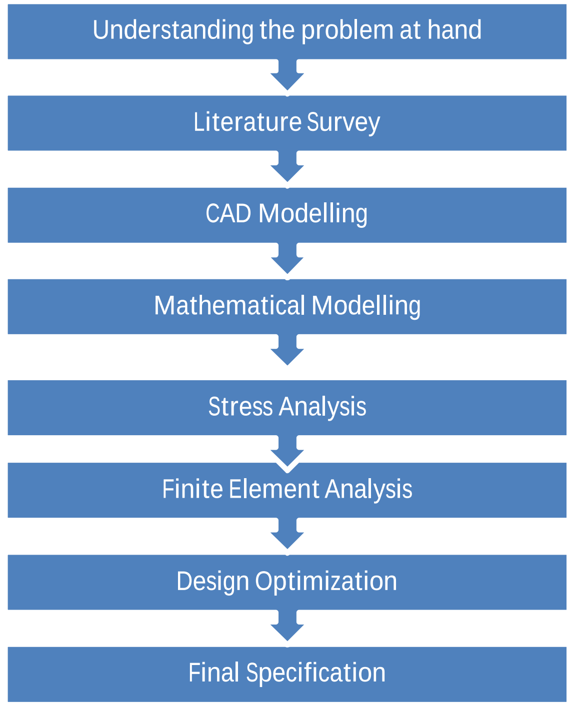
C. Objectives

The objectives of this design paper are summarized as follows:

1) To analyze the design and improve the design of Front wheel Hub of COEP SUPRA SAE Formula (FSAE) vehicle for high strength and light weight.
2) To understand the loading conditions, boundary conditions and the environment the component is subjected to.
3) Perform Finite element analysis to find out stress concentrations, there by finding areas to improve design.
4) Give final design with dimensions and material specifications.

II. DESIGN FLOW METHODLOGY

The approach to our design problem is elaborated through the flow-chart given below :



III. LITERATURE SURVEY

An extensive literature review was performed on different optimization techniques used for redesign of mechanical components. Idris Karen, Ferruh Ozturk presented a paper in which, a failed clutch fork was completely re-designed using topology and shape optimization approach. Stress life fatigue analysis was conducted to correlate the crack location between the failed component and the simulation model. The mass reduction obtained was 24%, maximum stress reduction of 9% was achieved and rigidity was improved up to 37% in comparison to the original clutch fork [1]

Gerhard Fischer, Vatroslov V. Grubisic [2] says that the design of wheel hub must be based on stress generated under customer usage through operational loads acting on wheels. Wheel hubs are highly steered safety components which must not fail under the applied loading conditions. The main parameters for design of wheel hub assembly are loading conditions, manufacturing process and material behavior. The influence of these parameters are interactive so material fatigue behavior will be changed depending upon the wheel hub design and loading conditions.

Grubisic, V., Fischer, G., and Heinritz, M. performed a design Optimization of Forged Wheel Hub. The authors show on the example of a forged front wheel hub how weight savings of up to 30% may be achieved on a standard hub forging. The weight savings are achieved through a design methodology called “Service Strength Analysis,” whereby both structural analysis and testing are done under simulated service loadings. The wheel hub is a highly stressed safety component on which the function of other components such as axles, wheels and brakes depend. Service Strength Analysis allows lightweight design to be carried out without sacrifices of reliability; it also allows an effective evaluation of the influence of new low cost manufacturing techniques on the service strength of parts. [3] S. Dhar conducted a Fracture analysis of wheel hub fabricated from pressure die Aluminum [4]. The author says that a catastrophic failure of wheel hub occurred during service. The nature of crack was a corner crack. An analytical investigation was carried out using tool of linear elastic fracture mechanics to establish the cause of failure. The non – linear behavior is due to the presence of material in homogeneities and discontinuities. An analytical estimation was carried out in order to calculate the minimum no. of cycles carried by wheel hub in service. The initiation of crack growth is complex because the heterogeneity and morphology of fracture surface. Fractographic and metallographic studies are carried out to assist the understanding of corner cracking problem. Shang, Shixian (Robert) [5] say that numerical investigation of wheel dynamics impact and cornering fatigue performance is essential to shorten design time, enhance mechanism performance and lower development costs. The desertion focused on two objectives:

1) Finite element models of a dynamic impact test on wheel and tire assembly were developed which considered the material in homogeneity of wheels. Comparison of numerical predictions with experimental measurements of wheel impact indicated that 20% reduction of initial striker kinetic energy provides an effective method for simplifying numerical modeling.
2) Numerical prediction of wheel cornering fatigue testing was considered. It proceeded in two methods, first was static stress analysis with bending direction applied to the hub. Second was dynamic stress analysis with application of a rotating bending moment applied to hub.

These papers represent a case study for the analysis and optimization of automobile component. These authors also used finite element software to indicate the maximum stress area, and some of them compared the results of numerical and experimental value.

IV. FORCE ANALYSIS OF FRONT WHEEL HUB

The main objective is to determine the magnitude and direction of the loads that act on the hub, which was used in the Finite element Analysis. The front wheel Hub experiences different forces due to the motion of the vehicle.The movements of interest are acceleration and braking and turning or cornering. The vehicle and its components are studied to determine what forces will be produced by each of these sources. Thomas D. Gillespie explains in his paper the effect of acceleration, braking, road load, steady state cornering, steering ride, rollover on vehicle performance. [6] In order to demonstrate how wheel loads can be calculated, a number of operational and simplifying assumptions are made. Preliminary analysis is done assuming steady state operating conditions. The assumptions are smooth road way, constant speed cornering, constant longitudinal acceleration, constant grade. All calculations presented are based on the main assumption that the chassis of the car under consideration is rigid. Forces acting on wheel hub are calculated as follows.

A. *Forces Acting on Wheel Hub*

1) The Vehicle at the instant of Braking: The braking force (Fb) transferred to wheel hub is calculated as below

$$Fb = \frac{d \times h \times m}{base} = 2080\ N$$

Where d= deceleration in terms of g's = 0.9 m/s^2 ; h= height of C.G = 475mm ; m= mass of vehicle = 300 kg

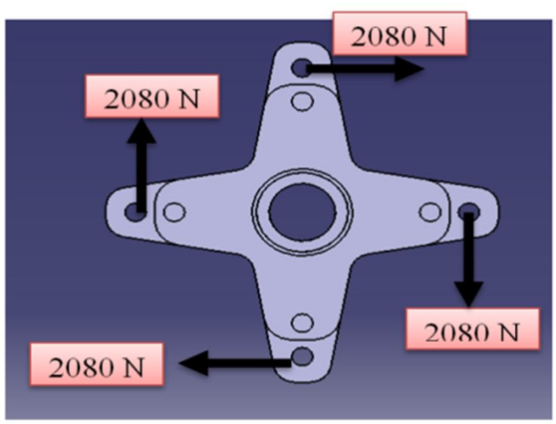


Fig.1 Braking forces

2) The Vehicle at the instant of Bump: When vehicle comes across bump, vertical force(Fw) acts at the contact of road and tyre. This force is greater than the static force(Fst) 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 40:60.

$$Fst = \frac{m \times 0.6}{2}$$

The vertical load acting at contact of road and tyre, when vehicle comes across bump, is taken as three times of gravitational acceleration. This force is transmitted to hub centre and given by:

$$Fw = 3 \times g \times Fst = 1912.9\ N$$

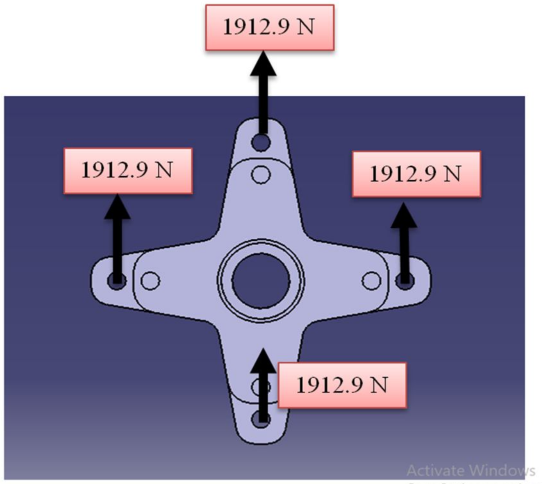


Fig.2 Forces due to bump

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:

$$Fc = \frac{m \times v2}{R} = 3111.5 \text{ N}$$

m = 40% of 300 kg ; v = 50 kmph ; R = 10 m

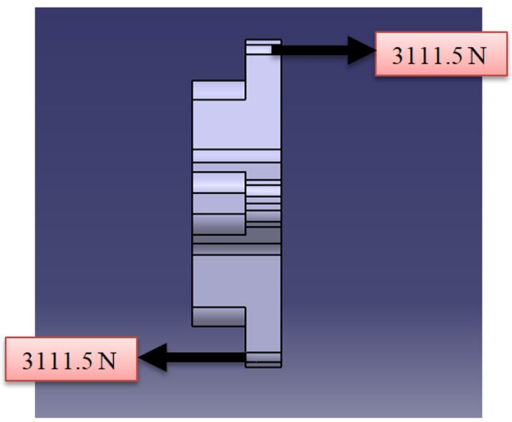


Fig.3 Forces due to cornering

a) Summary of forces

Braking force = 2080 N

Forces due to bump=1912.9 N

Forces due to cornering =3111.5 N

V. FINITE ELEMENT MODELLING AND ANALYSIS

A. Finite element modelling

Finite element modelling of any solid component consists of geometry generation, applying material properties, meshing the component, defining the boundary constraints, and applying the proper load type. These steps will lead to the stresses and displacements in the component. The forces calculated before are used for FEA analysis in ANSYS. Finally, results from FE analysis, which consist of stress, displacement history at different locations, were used as the input to the optimization process.

1) *Generation of Geometry of hub:* The dimensions of the front wheel hub were measured using a Vernier calliper and measuring tape with accuracy of 0.02 mm and 1 mm respectively. Having accurate dimensions of hub a solid model was generated using CATIA V5 R19 modelling software [7]

The solid model generated for the original front wheel hub is shown in figures below.

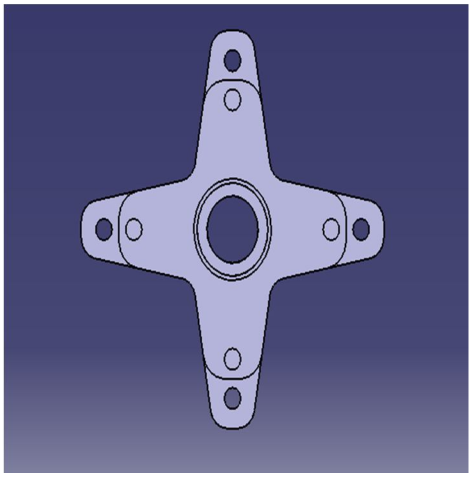


Fig.5 Front view of original wheel hub

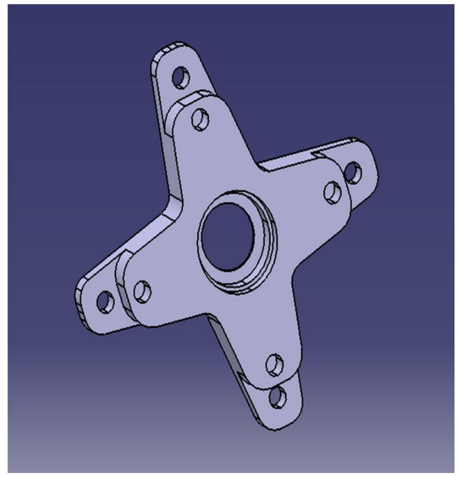


Fig.4 Isometric view of original wheel hub

Material properties (aluminium alloy) used in model are tabulated below

TABLE I. Material Properties (Aluminium Alloy 6061)

Symbol	Material of wheel hub	Aluminium alloy 6061
E	Young's modulus of Elasticity	7×10^4MPa
S_{yt}	Yield Strength	186 MPa
S_{ut}	Ultimate tensile Strength	225 MPa
µ	Poisson ratio	0.33
P	Density	2710 kg/mm^3

2) Meshing : Meshing of the entire component was done within the ANSYS environment. Tetrahedral free meshing was done, as the component had smooth curves, and cylindrical topology. The meshing was automatically refined at the points where there was discontinuity in the form of step, hole etc. The total no of nodes were 75700 and elements were 43944.

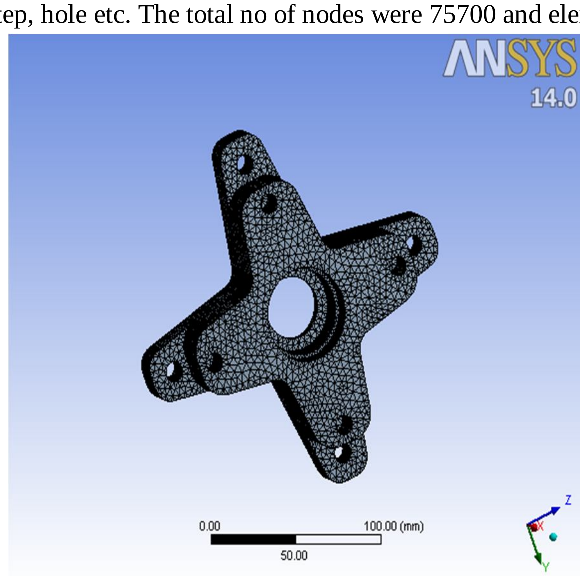


Fig.6 Meshing of Old Wheel Hub

3) Constraints and Loads : The bigger baseplate is bolted on to the wheel rim and his hence constrained in 3 Degrees Of Freedom. Loads are applied on the other base plate. The load was calculated as shown above in the calculations section. All the loads were applied on to the model and analysis was done

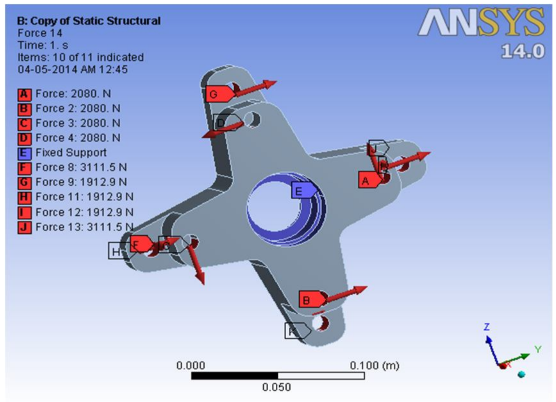


Fig 7 Loads acting on hub

B. *Finite element analysis and Results*

Finite element analysis was done in order to find the actual stresses that are experienced by the component under different loading. ANSYS 14.0 was used for the finite element analysis. The component was modelled in the ANSYS 14.0 modelling environment, with the same dimensions as the real one. A 3D linear static second order finite element analysis was solved using ANSYS [8] solver.

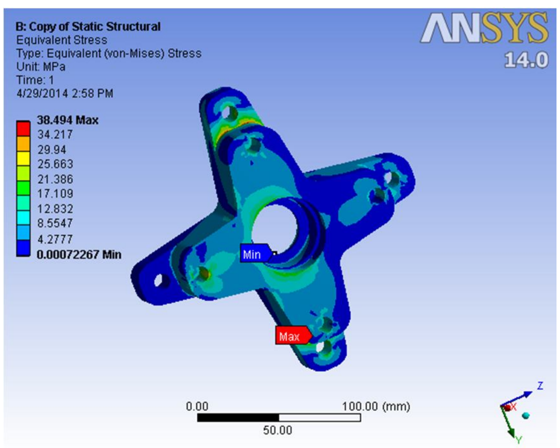


Fig.8 Stress distribution on original wheel hub

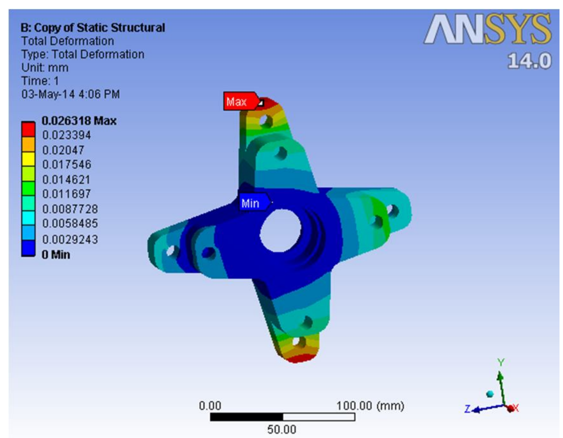


Fig.9 Deformation distribution on original wheel hub

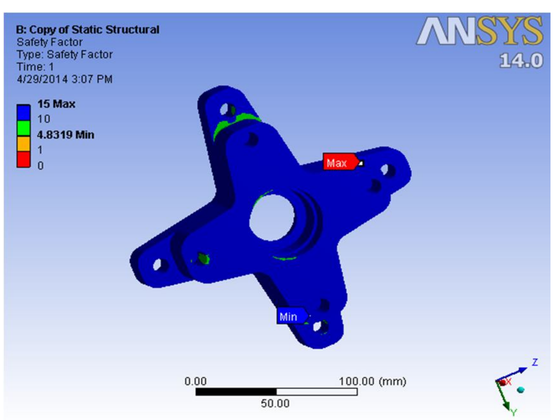


Fig.10 Factor of safety for stress

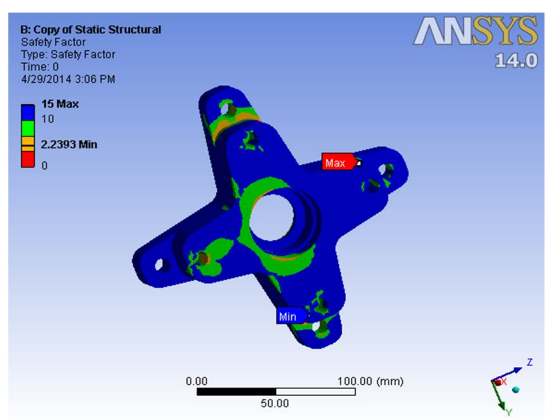


Fig.11 Factor of safety for fatigue on original wheel hub

The FEA analysis results of original wheel hub are :

Table II. Fea analysis results of original wheel hub

Parameter	Maximum	Minimum
Stress	38.494 MPa	0.00072267 MPa
Deformation	0.009336 mm	0
Factor of safety for stress	4.8319	0
Factor of safety for deformation	2.2393	0
Mass	1.1806 kg	-

The results show that there are areas of negligible near zero stress. Hence there is possibility for material reduction (that is weight reduction) from such areas.

VI. TOPOLOGY OPTIMIZATION

Optimization [9] is a process that finds a best or optimal solution for a problem from a number of alternative solutions in a search space. An optimisation problem is defined as finding values of variables that maximise or minimise the objective function while satisfying the constraints.

The main objective of this analysis was to optimize the weight of the wheel hub, which not only reduces the final production cost of the component, but also results in a lighter weight wheel hub which will increase the fuel efficiency of the vehicle.

Structural optimization is one application of optimization. Here the purpose is to find the optimal material distribution according to some given demands of a structure. Some common functions to minimize are the mass, displacement or the compliance (strain energy). Structural optimization can be separated in three different areas: size optimization, shape optimization and topology optimization.

The optimization approach considered in this study was topology optimization.Topology optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance. Using topology optimization, engineers can find the best concept design that meets the design requirements. The design of the component is optimized by the topological optimization in ANSYS. Topology optimization is used at the concept level of the design process to arrive at a conceptual design proposal that is then fine-tuned for performance and manufacturability. This replaces time consuming and costly design iterations and hence reduces design development time and overall cost while improving design performance.

The optimization problem solving process adopted is as follows:

A. *Objective function*

Objective function is defined as the parameters that are attempted to be optimized. In this study the weight of the component was the main objective. Optimization attempt was to reduce the weight and maintaining the stiffness within permissible limits.

B. *Constraints*

Since the current wheel hub used in the vehicle has proper dynamic performance, optimization was carried out in such a way that the equivalent stress at any location of the optimized model did not exceed the permissible equivalent stress at that location. So the bounded constraint in this component is maximum allowable stress of the material. The equality constraint is geometry limitations or fixed dimensions. Manufacturing feasibility and cost is another constraint.

C. *Design Variables*

Parameters that could be changed during the optimization process are design variables.

The design variables are upper and lower limit for size and geometry. In this study the design variables were volume and deformation of the wheel hub.

D. *Optimization Analysis Result*

In this section detail of optimization stage is discussed presenting the influence of optimization potential on maximum stress range, maximum displacement, fatigue strength and the weight difference with the original wheel hub.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue X, Oct 2018- Available at www.ijraset.com

The optimization on wheel hub was performed in ANSYS 14.The resulting model after optimization is shown in different views. We can see that the material is removed from area between bolt holes and bearing hole because the stress acting and the deformation at those areas was minimum and it was safe to remove material from those areas. Due to this topology optimization, the weight of wheel hub is minimized.

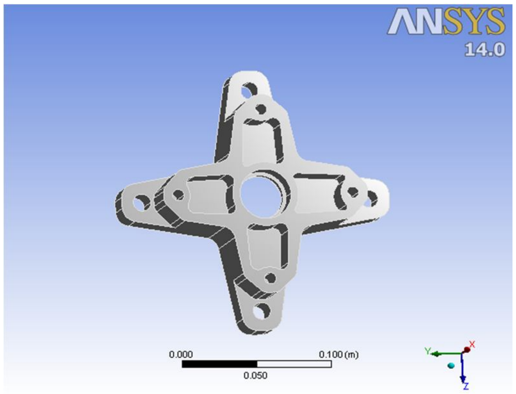


Fig.13 Isometric view of optimized wheel hub

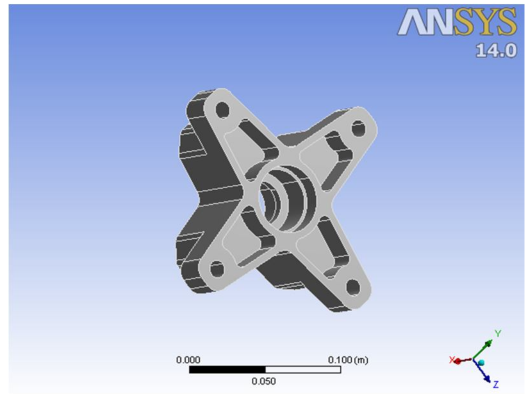


Fig.14 Isometric view of optimized wheel hub (backside)

VII. DESIGN VALIDATION AND OBSERVATIONS

The new optimized model of front wheel hub is tested with the same load conditions and constraints as that of old wheel hub for Aluminium alloy 6061. To validate the structural integrity of the newly designed wheel hub, a finite element analysis was performed using ANSYS 14 as a qualitative test to ensure that the design did not have any inherent stress concentrations or fatal flaws.

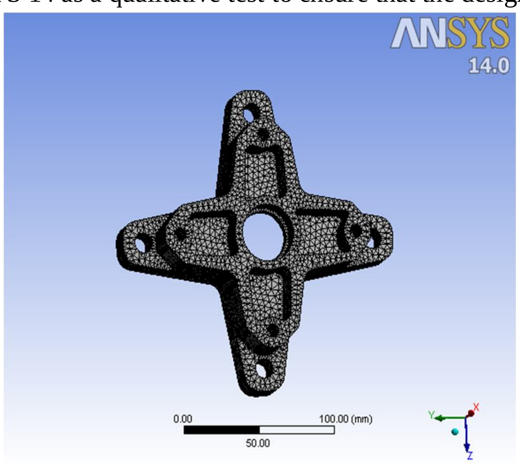


Fig.15 Mesh of optimized wheel hub

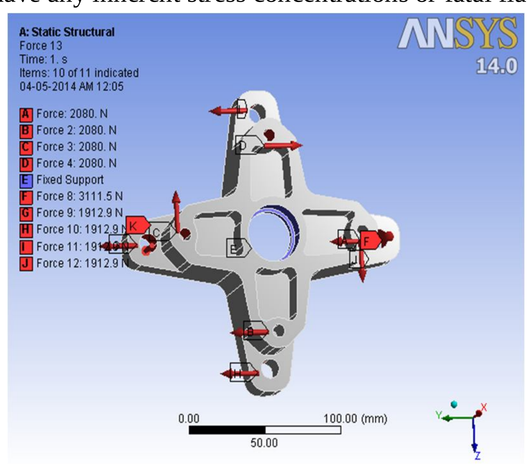


Fig.16 Loads and Constraints

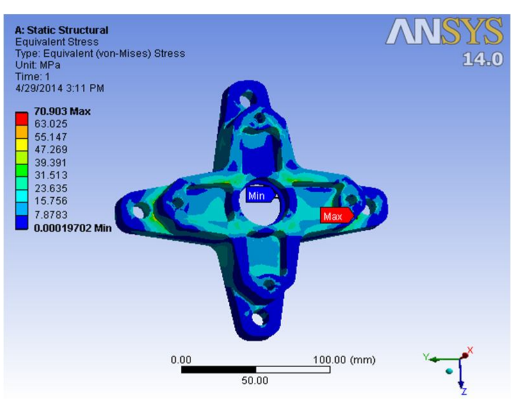


Fig.17 Stress distribution on optimized wheel hub

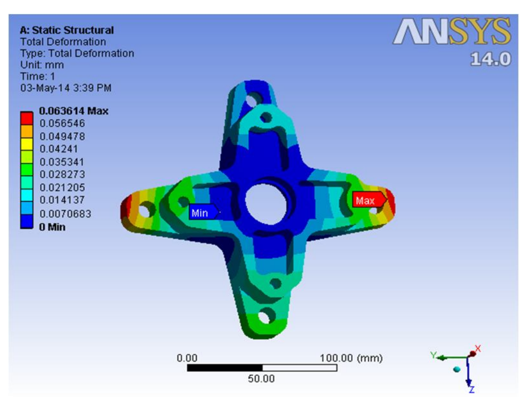


Fig.18 Deformation in optimized wheel hub

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue X, Oct 2018- Available at www.ijraset.com

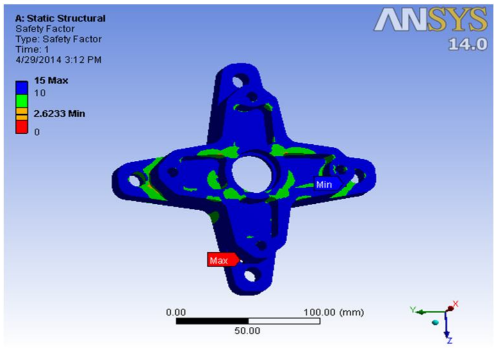


Fig.19 Stress Safety factor for optimized wheel hub

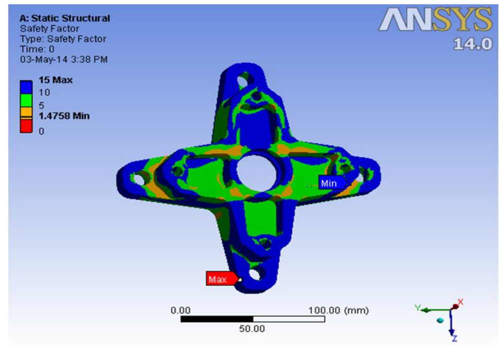


Fig.20 Fatigue safety factor for optimized wheel hub

A. Observations

Aluminium Alloy 6061 : The maximum stress acting on hub is increased from 38.494 MPa to 70.903 MPa and the minimum factor of safety for stress in optimized component became 2.6233.The maximum deformation in hub is increased from 0.026318 mm to 0.063614 mm and the minimum factor of safety for fatigue became 1.2157 in the optimized hub. So, we can say from the results that the new optimized design is safe. The mass is reduced from 1.1806 kg to 0.76851 kg. So the new optimized design is lightweight.

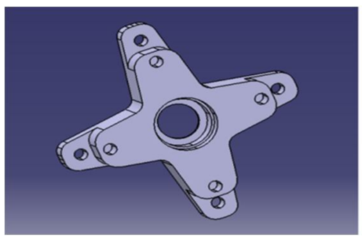


Fig.21 Old wheel hub

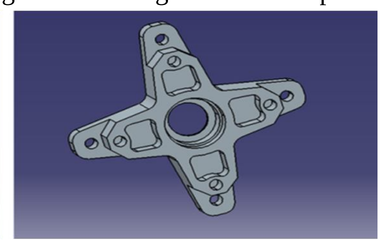


Fig.22 Optimized Wheel hub

VIII. CONCLUSION

The following conclusions can be drawn from the analysis conducted on Front Wheel Hub of COEP SUPRA FSAE car in this study:

1) FEM analysis results an efficient and simple method of achieving stresses for different loading conditions according to forces applied to the wheel hub from the dynamic analysis.
2) Topology optimization of wheel hub resulted in 34.9 % weight reduction for wheel hub with aluminium alloy as material. Stress and displacement increases slightly but they are within safe limit.
3) Table shows the comparison of Mass, stress and deformation of optimized wheel hub (aluminium alloy 6061) with the original wheel hub.

TABLE III . Comprison Of Optimized Wheel Hub With The Original Wheel Hub

Sr. No	Parameter	Old Wheel hub	Optimized Wheel hub	Change	% change
1.	Mass	1.1806 kg	0.76851 kg	-0.41209 kg	-34.9%
2.	Stress	38.494 MPa	70.903 MPa	32.409 MPa	84%
3.	Deformation	0.026318 mm	0.063614 mm	0.037296	141%
4.	Material Cost	Rs.450	Rs.300	-Rs.150	-33.33%
5.	Stress safety factor	4.8319	2.6233		
6.	Fatigue safety factor	2.2393	1.2157		

4) The new optimized design of wheel hub is validated with original load conditions and constraints and it is found to be safe.
5) CAE analysis and optimization tools save development time and reduce costs in the conceptual design phase for new parts. The product development process becomes faster and more efficient using optimization tools.

IX. ACKNOWLEDGEMENT

I would like to thank Dr. Vipin Kumar Tripathi for his guidance and constant supervision & also for his support in completing the project. I would also like to thank my colleagues from COEP SUPRA Team Octane Racing who provided insight and expertise that greatly assisted the project.

REFERENCES

[1] Idris Karen, Ferruh Ozturk “Re-design of a failed clutch fork using topology and shape optimization” Elsevier, Material and design 31;2010;3008-1014
[2] Gerhard Fischer , Vatroslov V. Grubisic “Design criteria and durability approval of wheel hub” SAE international, USA 11-16-1998 technical paper,pp 4-8
[3] Grubisic, V., Fischer, G., and Heinritz, M “Design Optimization of Forged Wheel Hubs For Commercial Vehicles”, SAE Technical Paper 841706, 1984, doi:10.4271/841706 pp 2-7
[4] S.Dhar “Fracture analysis of wheel hub fabricated from pressure die aluminum assembly theoretical and applied fracture mechanics” , vol 9 feb 1988,pp 5-11
[5] Shang, Shixian (Robert) “Finite element modeling of dynamic impact and cornering fatigue of cast aluminum and forged magnesium road wheels” Proquest dissertation and thesis 2006
[6] Thomas D. Gillespie “Fundamentals of vehicle dynamics Society of automotive engineers”,2008,pp.21-40,45-75,795-831
[7] CATIA V5 R19 Users Guide:2010
[8] Ansys 14 Users guide.
[9] S.S Rao “Introduction to optimization in Engineering Optimization Theory and practices”, New age International Publication,2003,pp.1-10