Impact Factor 6.1

Journal of Cyber Security

ISSN:2096-1146

Scopus

DOI

Google Scholar

More Information
www.journalcybersecurity.com

Crossref doi Google Scholar scopus

Designing and Simulation of wheel Hub and Upright of Vehicle: A Review

Kiranpreet Singh Kang, Shubham Jha, Jaswinder singh
[1]A.P Department of Mechanical Engineering
[2]B.E Student, Department of Mechanical Engineering, Chandigarh University
shubhamjha2128@gmail.com, jahluwalia18@gmail.com

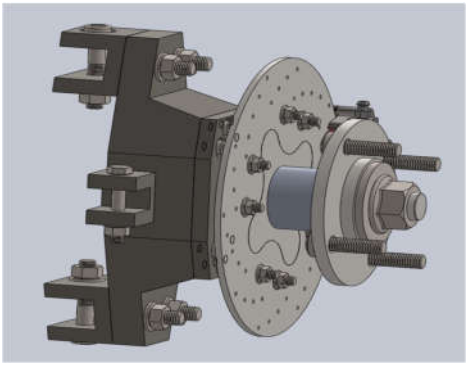
Abstract

This paper deals with various methodologies adopted by present researcher and the designers for analysis of wheel hub and upright assembly with main objective of optimization of the vehicle. This review includes the design ,analysis and production of uprights without compromising the strength,the software used for the design and analysis purpose are Anysis,Solidworks,Hypermesh. The review includes key areas of researches as minimizing the weight of the car by reducing the weight of the uprights and hub without compromising the strength and stress distribution by performing static load analysis and fatigue load analysis using CAD and Anysis. This literature discusses about the research methodology and softwares and the outcomes of the researches and is ought to give the readers a brief variety of the literature carried out on the wheel hub and upright assembly
***Keywords**: Design and Simulation, wheel hub, Anyysis, Solidworks*

I. Introduction

Wheel hub and upright assembly is an important part of the vehicle suspension system which permits the steering arm to turn the front wheels and support the vertical weight of the vehicle **[2, 20]**. Uprights are also known as the knuckle or spindle. It assembles with the tires and a spindle that rotates in a stable plane of motion by a suspension assembly. The force on the hub and the uprights are cyclic in nature due to the steering **[26].** To gain the maximum speed for the sports car the weight has to be minimized, while designing it should be kept in mind that it should not decrease its strength and strees distributions due to the removal of the material **[9]**.

The design of wheel hub and upright are one of the key factors in improving the weight or mass of the vehicle. In **fig.1** basic type of hub is shown. The mass reduction without compromising the strength of the wheel hub and upright assembly has done by the researchers and designers to improve the weight or mass of the vehicle. Weight or mass of the vehicle can be reduced by selection of material, designing ,simulation or analyze method **[10]**. Steering uprights are tends to subjected to varying load during its life, leads to fatigue loads,permanent loads **[13]**. The upright and hub assembly also transfers the whole weight of the vehicle onto the wheels of car, which lead to stress on mounting points.



Front upright and hub assembly

II. Brief Description To Hub And Upright

In automotive suspensions, there are four uprights two at the front known as the steering upright while the rear contains two known as the driven uprights, the steering upright is also called as steering knuckle.

Fig. 1. Hub as reported by ref. **[4]**

The wheel and tire assembly attach to the hub and spindle of the knuckle where the tire/wheel rotates while being held in a stable plane in motion by the knuckle/suspension assembly **[3]**. As shown in **fig.2** of double- wishbone suspension system, the knuckle has attached to the upper mounting point and bottom mounting point. The wheel assembly is shown attached to the knuckle at centre point. Suspension systems in any vehicles uses different kinds of links, arms, and joint to let the wheels move freely, front suspension allows the front wheel to turn **[27].** Steering spindle assembly, which has two separate or one complete parts attached together in one of these links. Hub has the part attached to upright, the purpose of a wheel hub is to attach a wheel to a motor shaft. Hubs are used to attach to the wheels are typically attached to hubs via the wheels face on its centre. The wheel is attached through fasteners to hub due to a good strength and can be easily be taken out for storage or servicing. Hub should be capable of rigidly supporting its share of its total weight of a vehicle without failure in during its calculated life span. If the hub geometry and material selection are not adequate, it will break assembly which will cannot be repaired any more.**[29].**

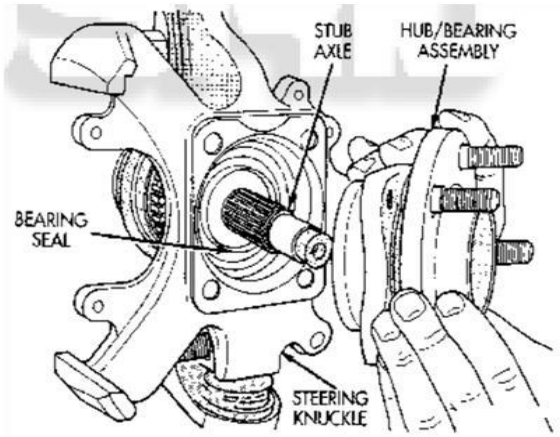


Fig. 2.Component of wheel hub and upright as reported by re**ference. [26]**

III. Previous Research's

3.1 Introduction

The hub and upright assembly is a complex structure for examine **[9].** The hub and upright are stressed throughout their expected life during load. In the process of operation, hub and upright assembly is subjected to various analysis like fatigue, cornering force, braking force, bump, impact load and combination of all other forces due to bump and cornering process, whereas it is subjected to sload due to excessive vibrations in the hubs and uprights in off road conditions **[20].** It is necessary to examine about the stress-strain distribution, fatigue and vibration characteristics of the a upright and hub assembly under above mentioned conditions **[17].** Most of the researches on the upright hub and in past few years are done using Finite Element Methods (FEM). Several researchers have conducted a variety of different analysis on the hub and upright assembly using different FEA tools. The primary aim of this article is to provide a comprehensive content review of researches conducted on the hub and up- right assembly which focuses on design, structural analysis, fatigue life prediction, structural optimization through CAD/CAE system integration of the hub and upright assembly with the focus on the analysis technique employed in making.

3.2 Work carried out on the design on upright and hub

For the selection of a hub and upright assembly, weight is an benefecial parameter. Fuel consumption(intake) can be reduced by sometimes by using light weight design without compromising strength. Some researcher attempted to reduce the weight of the hub and upright,significant study to reduce the upright weight, was done by **Suhaimi [1]**,through static analysis some parts of upright are eliminated maintaining permissible structural strength through stages such as designing, analysis, fabrication and fitting .

3.3 Work carried out on the structural analysis of upright and hub assembly

The hub and upright assembly was subjected to complex stress due to carrying load reacted by the wheels, which are transmitted to the chassis through A-arms indirectly from the upright. Hence, nature of the load is bending **[1].** This process, a variety of force act on the hub and upright **[20].** Therefore, it is necessary to make the strength calculation of hub and upright under load condition **[26].** Structural analysis determines the area under high stresses and provides basis in structural optimization which improves service life.

Song and Lee [6] discussed Reliability-Based Design Optimization (RBDO) of the automotive knuckle component under bump and brake conditions. The probabilistic design minimized the weight of a knuckle component subjected to stresses, and frequency constraints in order to meet the given target reliability. Later on **Babu et al. [7]** achieved load analysis in two steps, first modeling of steering upright as per design parameters and also analysis considering the loads and boundary end conditions. Then finding out the minimum stress area. In another research **Rangababu et al. [8]** carried out analysis on the three materials viz. GCR15, Steel and Low Alloy Martensitic Chrome Steel, with the help of conventional model and Fiat model. He upgraded both conventional model and Fiat model. The results obtained for von misses stress, strain and total deformation values had compared as shown GCR15 (alloy steel) was suggest as most suitable material.Statical analysis and optimization of upright was done by **Prajwal [9]** through FEA software Hypermesh,Anysis. After optimizations of the weight was reduced up to 5% and stress concentrations were reduced up to 15%. In context researchers, **Tagade et al [10]** extended the scope of static analysis of previous researchers, analysis geometric modeling was done on CREO 2.0 and ANSYS. The analysis concluded with mass reduction of uprights by 67% maintaining factory of safety was in between 3 to 4.

Using similar methodology **Sharma [11]** had used CREO2.0 for geometric modeling and later carried out Static analysis on ANSYS WORKBENCH. In analysis of knuckle,load applied due to braking torque on caliper mounting, longitudinal reaction due to andvertical reaction due to vehicle weight and steering reaction was considered as applied boundary conditions. The final reduction in mass was 19.35%, maintaining factor of safety between 3 to 4.**Fig. 4** shown detail of meshing of upright model.

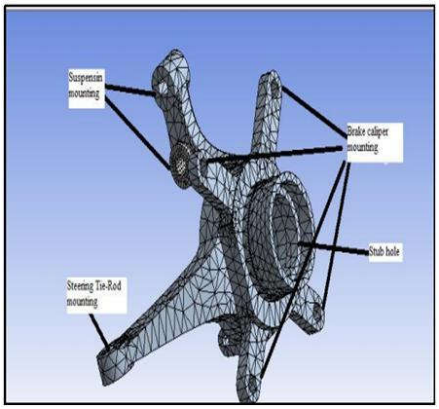
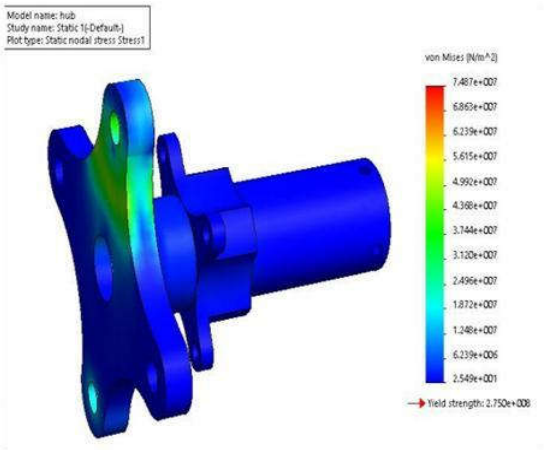


Fig. 4. Detail And Meshing of the Upright Model as reported by **[11]**

Fig. 5. Design Of Hub as the presented by **[12]**



3.4. Work carried out on the fatigue analysis of hubs and uprights assembly

The hub and upright are subjected to cyclic loading conditions, which were results in generation of cyclic stress ,ultimately fatigue failure **[11, 14].** Therefore, it is necessary to analyze the fatigue of hub and upright assembly from the prevent premature breakdown, failures and fractures. It takes place when applied stress get so large in that material can no longer endure stresses and strains. In classic the structural analyses, failure predictions are based solely on the material strength or in yield strength. Durability analysis goes beyond this, evaluating failure based on repeated or simple or complex loading **[26].**

3.5. Work carried out on the analyzation of the hub and upright assembly

As from the analytical point of view, it is well understood that the optimizations of any structures is a very important work for a researchers performing analysis in his field. A significant work in optimization of uprights was done by **Shelar and Prof. Khairnar [18]** by using methodology based on durability and optimized design through probabilistic model of designs variables (doe. The obtained data give reduction in weight by 9.195% with stresses 23.67%. Another experimental

analysis of optimization through fatigue life had done by **Sivananth [19]**, geometric model were developed on SOLIDWORK and analysis was carried out on HYPERMESH. In analysis comparisons of two materials SG (iron alloy) and Al alloy steering arm was done. Regional impact forces of 17.5 KN and 35 KN were applied at varying speed. For the velocity of 2 m/s the deflection varies between 3.5 mm and 4.39mm ,which does not affect upright ,but in case of SG iron for velocity of 4m/s, 8.6mm deflection was obtained similarly by 12 mm of deflection was obtained for Al alloy which largely affects steering knuckle

Pawar and Pathak [21] used Catia for CAD modeling and HyperMesh for structural analysis. Various structural parameters such as the nodal displacements, stress distribution and fatigue parameters like damage and fatigue lifes were analyzed on the same solver. S-N curves were analyzed at high cycle fatigue, where the material was subjected to cyclical stresses that are predominantly within the elastic ranges. Test cycle constraint of material survival for more than 1000 cycles was applied for evaluating results.

3.6. Work carried out on the CAD/ CAE technology integrations

Design and development of a wheel hub and upright assembly requires lightweight and sturdy components to move fast **[7].** Optimization through CAD/CAE is a best process to simplify a design to reduce its weight, manufacturing cost without compromising the initial and strength of a part [**12].**

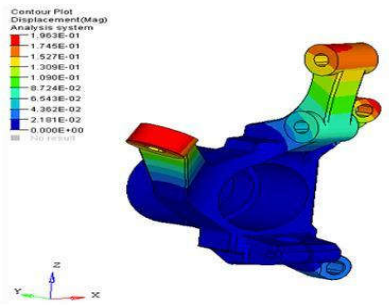


Fig. 9.Displacement Contour Of the Modified Model as per presented by ref. **[20]**

.

IV. Conclusion

This work has provided a vast literature review of existing researches carried out in terms of designs, stress, fatigue, optimizationand analysis of the hub and uprights assembly of sports car. An overall efforts has been made to comprise all the important contributions to this area and highlighting the most pertinent contents available for investigating the hubs and knuckle assembly. The concluding remarks and future work from the present literature survey are as follows:-

- From the review of available research on hubs and uprights assembly, it is apparent that most of the research conducted is purely on simulation based on finite element method.
- The FEA is a useful tool since it provides an accurate results to access strength and fatigue life of the hub and uprights. The same can also be applied for shape optimization of hub and uprights
- The review shows that hubs and uprights are designed for different types of vehicle according to their use. Applied boundaries conditions are calculated analytically.

References

[1]. Suhaimi,Khalis bin.”Design And Fabrications Of An Uprights With Brake Caliper Mountings For the Formula Versality Race Car”
dissertation, april 2011.

[2]. Kulkarni, V. R. and Tambe, A. G.;” Optimization and Finites Element Analysis of the Steering Knuckle”, Altair Technology
Conferences, 2013.

[3]. Razak I.H.A, Yusop M.Y.M, Yusop M.S.M, Hashim M., Modeling, Simulations And Optimization Analysis Of Steering Knuckles
Component For Race Cars, International Journal of Research in Engineering and Technology,Volume: 03 Issue: 11, eISSN: 2319- 1163 — pISSN: 2321-7308, Nov- 2014, Pp.221-225.

[4]. Dyapa, Anudeep Reddy and Shenoy,Vishal.,”Design And Analysis Of Uprights Of An FIA Related Cruisers Class Solar Electric Vehicle”,” International Journal of Engineering Research and Technology (IJERT)”, ISSN: 2278-018, oct,2014, Pp.1062-1066.

[5]. Gill, Simren., Hay,Philip., Mckenna, patrick., and Thibodeau, Philippe;”Final Design” Dalhouse’s Formula SAE ” April 8,2011.

[6]. Song, Chang Yong. and Lee, Jongsoo., Reliability-Based Designs Opti- mization Of Knuckle Components Using Conservative Method Of Moving Least Squares Meta-Models, Probabilistic Engineering Mechanics, vol.26 (2011),Pp. 364-379.

[7]. Babu,B., Prabhu,M., Dharmaraj,P. and Sampath,R.,” Stress Analysis On Steerings Knuckle Of The Automobile Steering System”, IJRET: Inter- national Journal of Researches in Engineering and the Technology,Vol.3,NO.3, eISSN: 2319-1163 — pISSN: 2321-7308, 2014, Pp .363-366

[8]. Rangababu. Daavuluru., Depti, and K., Ramana, B.”Optimal Designs And Strenghths Analysis Of Wheel Hubs Using Different Types
Of Materials” Journals Of Technological Advances And Scientific Research; VOL.1 /ISSUE 04/OCT-Dec 2015;Pp. 353-359.

[9]. Prajwal,Pathri Bhargav. ;”Designs Optimization Of Formula One Student Sports Car Upright Using Hypermesh”, International Journal Of Mechani- cal And Industrial Engineerings(IJMIE), the ISSN No. 2231-6477, vol-2, issue-1, 2012, Pp.54-59.

[10]. Tagade, Poonam P., Sahu, Anil R. and Kutarmare H. C.,” Optimizations And Finite Element Analysis Of Steerings Knuckle” International Journal Of Computer Applications International Conference On Quality Up- Gradation In theEngineering, Science And Technology (ICQUEST2015), (0975 8887).

[11]. Sharma,Mahesh P., Mevawala, Denish S., Joshi, Harsh., and Patel,Devendra A. ,” Static Analysis of Steering Knuckle and Its Shape Optimizations ”, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE),2014, e-ISSN: 2278-1684, p-ISSN: 2320-334X , Pp. 34-38.

[12]. Das, Abhijeet., ”Virtual Prototype Of Uprights Assembly Of A Race Car For SUPRA SAEINDIA Competition”, International Journal Of Mechanical And Industrial Technology Vol. 2, Issue 2), ISSN 2348-7593, October 2014 - March 2015, Pp: 79-86 .

[14]. Bhokare,Kiran S., Kakandikar,G. M.,Kulkarni, Swapnil S., ”Predicting Fatigue Of Steerings Knuckle Arm Of A Sport Utility Vehicle While Deploying Analyticals Techniques Using CAE”, International Journal of Scientific Research and Management Studies (IJSRMS),ISSN: 23493771 Volumes 1 Issue 11, Pp: 372-381

[15]. Mahendra L. Shelar and Prof. H. P. Khairnar, Design Analysis and Optimization of Steering Knuckle Ussing Numerical Methods and Design of Experiments Volume 2, Issue 3, ISSN: 2321- 9939, 2014, Pp. 2958-2967.

[16]. Sivanath,V. , Vijayaragana S., and Aswathawan R., ”Fatigue and the impacts analysis of automobiles steering knuckles under operation load cases” Altair Technology Conference india 2015.

[17]. Dumbre,Purushottam.,Prof Mishra,A. K., and Aher,V. S., ”Structural Analysis of steerings Knuckle for Weight Reductions , ”International Journal of Emerging Technology and Advanced the Engineering”, Vol. 4, No. 6, eISSN 22502459 , April-June, 2014, Pp.552-557.

[18]. Pawar, Abhishek,, and Pathak, Charudatta., Fatigue Lifes Estimation of a Steering Knuckle”, International Engineering Research Journals (IERJ) Special Issue 2, ISSN 2395-1621, 2015, Pp. 1036-1040.

[19]. Y. Kojima, Mechanical Cae In Automotive Designs R D Review Of Toyota Crld, Vol. 35, No.4, 2000.

[20]. Qu Yuan., Zhang Linbo, Xu Youzhong., Zhang Guanliang., And the Wu Shenrong., ”Practical Approach of Durability Evaluation for theSuspension Knucklse at Different Design Phases”, Chery Automobiles Co., Ltd ,”SAE International” 2009-01-1411.

[21]. R.Prem Raj ,and K.Palpandi, ”Static Analysis and Topology Optimiza- tions of Steering

Knuckles by Using Finite Element Method" ,International Journal of Innovative Researches in Science, Engineering and the Technology Vol. 4, Special Issue 13, ISSN:- 2347-6710, December 2015, Pp 374-380.

[22]. R. Premraj M. Chandrasekar K. Arul kumar "Free Vibrations Analysis And Optimizations Of Streeing Knuckle" South Asian Journal of the Engi- neering and Technology Vol.2, No.22, 2016, Pp. 157 172.

[23]. Patel Akash A.,Fatigues or Durability Analysis of Steering Knuckles", International Journal for Scientific Researches Developments, Vol.1, Issue9, ISSN: 2321-0613, 2013

[24]. Chavan, Kamlesh Lalasaheb., Deodas, Sanjay., Kulkarni,Swapnil Shashikant.,"ass Reductions For the Steering Knuckles Arm In A Suspension System Through Topology Optimization In Cae ", International Journals of Advanced Engineering Research and Studies /IV/I/Oct.-Dec,2014,E- ISSN22498974 ,PP 34-35.

[28]. Vishwakarma, Preeti., and Kanungoo, Mukeshs.," Finites Element Anal- ysis Of Chervolet Fronts Hub With The Help Of Inventer" ijite vol. 02, issue -02, ISSN : 2321-1776. Feb. 2010, Pp. 8-16.

[29]. http://www2.mae.ufl.edu/designlab/Lab

[30]. Yusop, M.S.M, Lazim, M.S.M, Razak, I.H.A, Hashim, M.F " Model Developments And The Shape Optimizations Analysis Of A Rear Knuckle For Race Car ",International Journal of Researches in Engineering and Technology vol: 03 Issue: 12, eISSN: 2319-1163 — pISSN: 2321-7308, Dec-2014, Pp:(172-177).