IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Impact Factor: 6.252
Volume 2, Issue 5, May 2022

Design and Topological Optimization of Wheel Hub

Tejas Dond, Omkar Sutar, Dnyanesh Shelke, Rupesh Vable, Dr. Srihari Notla
JSPM's Rajarshri Shahu College of Engineering, Tathawade, Pune, Maharashtra, India

Abstract: Optimizing automotive components can result in a significant reduction in vehicle weight, greater fuel efficiency and reduced environmental damage. The hub is the vehicle wheel mounting assembly that houses the wheel bearing and supports the terminals. A speed sensor and an ABS sensor are usually connected to the unit via an electrical connection in contemporary automobiles. The function of the wheel hub is to keep the wheel attached to the vehicle while allowing it to rotate freely in the bearing. Since the wheel hub is the only component that really connects the wheels to the vehicle, it is vital to pay special attention to it when optimizing the design. The CATIA software will be used to create a 3D model of the wheel hub. The stress and fatigue life of the wheel hub was calculated using finite element analysis software, then the weight was optimized. ANSYS software will be used to perform static and structural analysis.

Keywords: Topological Optimization

I. Introduction

All suspension components including the wheel hub have high-tension safety components that should not fail during service. Light weight designs are becoming very popular for all parts of a vehicle at low cost. The axle and all suspension components must be designed for loading conditions such as special event loading and bus use for long-distance transport off-road operations to commercial vehicle use and construction sites. As a result, there are different service load spectra for each driving position. Wheel hubs should be designed based on the pressure exerted on the wheels by the operating load during user use. The resulting service load conditions and the stress and fatigue characteristics of the center should be considered. However the integrated shaft and bearing life must be designed to avoid heat loss. This is due to the direct installation of the router.

The objective of the study is to design, optimize and analyze a hub. Hub is a part of wheel assembly of a car. Here we have designed a hub of front wheel assembly. A wheel set assembly consists of a knuckle-stub-hub-brake disk-2 taper bearing-castle nut-cotter pin-lug nut and lug bolts. Hub is rotating part in the assembly where disk is mounted on one side and wheels are mounted on other side. As this part gives a motion to the wheel and affects the overall performance of the car. Hence it is important to design this part by considering various safety considerations. Proper material selection is to be taken in consideration as well as cost is also important. Design is to be done by keeping in mind the safety of the part and manufacturing point of view. As in motorsports weight of the car plays an important role in overall performance of the car. For this purpose, a shape optimization of hub is done which actually gives an idea about the stresses which are induced in it and how much material we can remove so as to reduce the weight and it also gives appropriate factor of safety to the part.

In automobile suspension, a steering upright, also called as a steering knuckle, spindle, upright, or hub, is a component that houses the hub or spindle and connects to the suspension. Wheels are normally connected to hubs via the wheel's face or center. The wheel is connected to the hub and may be easily removed for maintenance due to its robustness. Wheels are normally connected to motors by sliding over and locking into engagement with their shafts, transmitting torque from the motor to the hub and then to the wheel. The contemporary TUV hub is made of mild steel and has been designed.

II. Literature Review

Gerhard Fischer Fraunhofer et al.:[1] The author of this publication examines the essential design and durability variables - operational loads, fatigue characteristics, which are reliant on material and manufacturing technique, and design - as well as the approach for an optimal light-weight design. Finally, the durability approval test procedure, the necessary test facility, the test programmer and requirements, as well as some typical test results, are reviewed. Trends in vehicle development are influenced by competing requirements: on the one hand, the vehicle should have high efficiency and performance while having a low environmental impact (low emission, noise, and waste at the end of its life cycle), while on the other hand, improved reliability and safety should be realised. Finally, these achievements should be accompanied by low manufacturing and possibly low maintenance (service) costs. The operational loading - where stresses during cornering and straight driving over bumpy roads are crucial, the influence of neighbouring components on these stresses, and the fatigue qualities influenced by material and fabrication must all be considered in the design approach. As explained in this study, operational loads can be defined by standardised spectra that are based on vehicle specifications and usage conditions. Improved dependability and safety, on the other hand, should be realized.

Eun Ho Lee and YongSoo Lee et al. [2] This article presents research on the disc connection included hub, which is used to avoid slowing down heat and thermal radiation. Furthermore, it demonstrates that the weight is reduced by combining it with the plate connector, just as the limit increment on the front hub is performed in conjunction with the standard wheel end module. The coordinated hub is designed to extend the life of the disc rotor and reduce the weight of the wheel end module. The temperature of the coordinated hub is determined using apparatus and vehicle testing to evaluate the structure concept of the heat avoidance structure. The test results suggest that the included hub's structural idea is better or equivalent to the benchmark hub in terms of thermal shirking. The findings of the analysis demonstrate that the new proposed hub is well-structured. The fatigue life of the hub is expected to be 200 percent of the test operating life required by the rising speed test. The Biaxial fatigue test is performed in accordance with the SAE specification, and the hub adapts to the essential trial and the 200 percent perseverance test as predicted by the computational inquiry. As a result, the hub exists with 1,000,000km of on-street or 500,000km of rugged terrain, and that's only the tip of the iceberg.

Prasad S Warwandkar, Dubey A, and Paroche S et al. [3] This article discusses the various configuration components to consider when planning gaskets and tops for wheel hub repair. A gasket is installed on the wheel hub face and top is pressed on to it as the wheel closes. When the bolt is tightened, the fixing load is transferred from the hub top to the gasket, causing tension. Because of this stress, the joint is impermeable. To determine the exhibition of the gasket junction, distinct plan parameters such as top thickness and bolt preload are concentrated. The weight stress bend of a gasket is used to test its performance at various top thicknesses. It also demonstrates that thicker gaskets can handle greater mating part irregularities since they pack more tightly. If your ribs are asymmetrical, thicker gaskets are a better option. The capacity of a thicker gasket to fill spine imperfections is determined by the measurement of gasket stress at a particular load. Because compressibility at a certain weight is often represented as a level of the gasket's unique thickness, a thicker gasket with a larger unique thickness truly represents a pack with greater separation. This increased gasket tension suggests that the gasket will fill better into the uneven surfaces of mating ribs, sealing the junction.

[4] **"Cengiz Shevket** et al**.** [4] This article suggests the layout of specialised issues encountered in the building tactics used during the project to minimise the mass and grating of wheel hub units. These features directly enable reduced fuel consumption and associated CO2 emissions while also having a significant impact on vehicle components. Also, show how the project engineers approached the issue, used displaying approaches to re-enact distinct arrangements, directed innovation exploration, selected new breakthroughs, prototyped, and entirely approved genuine products. With the application of the composite hub concept and the optimal use of materials, gauge reduction of 30% and idleness reduction of half on flanged rings can be obtained. Semisolid tossing is a cost-effective method for producing high-quality parts in large quantities. The results can be extended to any application having a pivoting external ring, for example, third age bearing or as an integrated arrangement supplanting gathering of first age bearing and external ring lodging. A rushed life cycle assessment has been performed to consider the environmental impact of a low gauge solution for a typical steel construction.

Rohit B Pawar, Dr. N.K Nath, Dr. S.B Satpal et al **[5]** In this journal literature gradually analyses the research methods, software, and outcomes of the discussed researches, with the goal of providing a broad overview of the research on the wheel hub and upright assembly. The wheel and upright assembly is an essential component of the vehicle suspension system. Knuckle is another name for upright. The hub and upright assembly sustain the vehicle's vertical weight. The hub is an essential component of the wheel assembly system. It's utilised to turn a moving vehicle into a wheel. The sport car's maximum speed. The key factor is maintained by the designer. Create a vehicle with a low weight and high stress capacity. Weight and mass can be minimised using techniques like material selection and a design analysis system.

II. Problem Statement

- For a variety of reasons, the automobile industry is constantly striving to lower vehicle weight. It aids in improving fuel efficiency and thereby lowering emissions. Weight loss is achieved by reducing the size or quantity of pieces, which lowers manufacturing costs.

- The optimization will be carried out by pinpointing the exact place without affecting the material's behaviour or strength.

2.1 Objectives

- Modeling of a wheel hub specimen in CATIA V5 software.
- Static analysis of the wheel hub for stresses and deformation.
- Topology optimization of the wheel hub model in ANSYS 19.
- Optimization is done by selecting proper location on the surfaces without changing the material and behavior of that material.

2.2 Methodology

- STEP 1: This project's work began with a literature review. We gathered a large number of study papers on this subject. We learned about the wheel hub and mounting system for a vehicle after reading these articles.
- Step 2: - The components required for our project are then determined.
- Step 3: - Once the components have been chosen, the 3D model and drafting will be completed using CATIA software.
- Step 4: - With the help of ANSYS and FEA, the components will be analyzed and optimized.

DESIGN OF CAD:

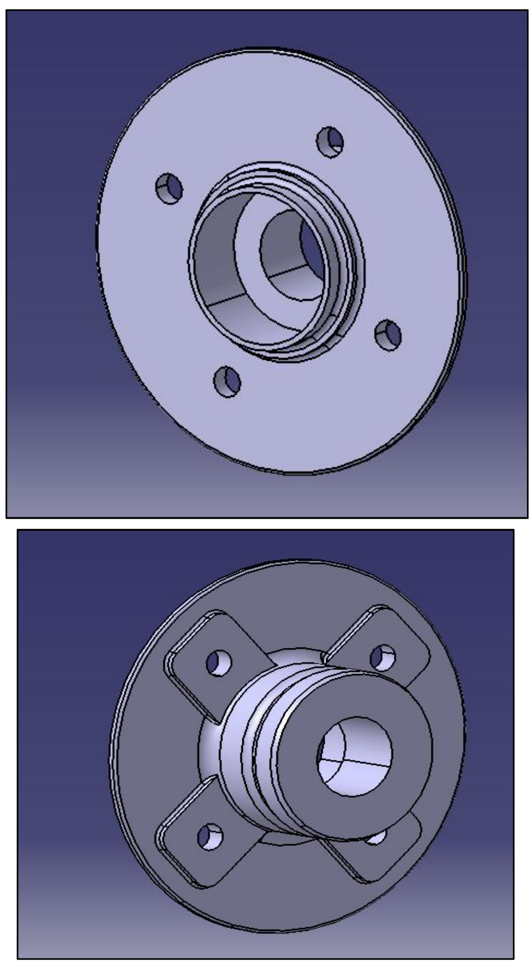


Fig. CATIA design of wheel hub

CATIA CAD

Computer-aided design (**CAD**) is the use of computers (or workstation) to aid in the creation, modification, analysis, or optimization of a design. This software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. Designs made through CAD software are helpful in protecting products and inventions when used in patent applications. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations. The terms computer-aided drafting (CAD) and computer aided design and drafting (CADD) is also used. CAD software for mechanical design uses either vector-based graphics to depict the objects of traditional drafting, or may also produce raster graphics showing the overall appearance of designed objects. However, it involves more than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions.

CATIA is an acronym of (computer-aided three-dimensional interactive application) is a multi-platform software suite for computer-aided design (CAD), computer-aided manufacturing (CAM), computer-aided engineering (CAE), 3D modeling and Product lifecycle management (PLM), developed by the French company Dassault System.

Since it supports multiple stages of product development from conceptualization, design and engineering to manufacturing, it is considered a CAx-software and is sometimes referred to as a 3D Product Lifecycle Management software suite. Like most of its competition it facilitates collaborative engineering through an integrated cloud service and have support to be used across disciplines including surfacing & shape design, electrical, fluid and electronic systems design, mechanical engineering and systems engineering.

FEA (Finite Element Analysis)

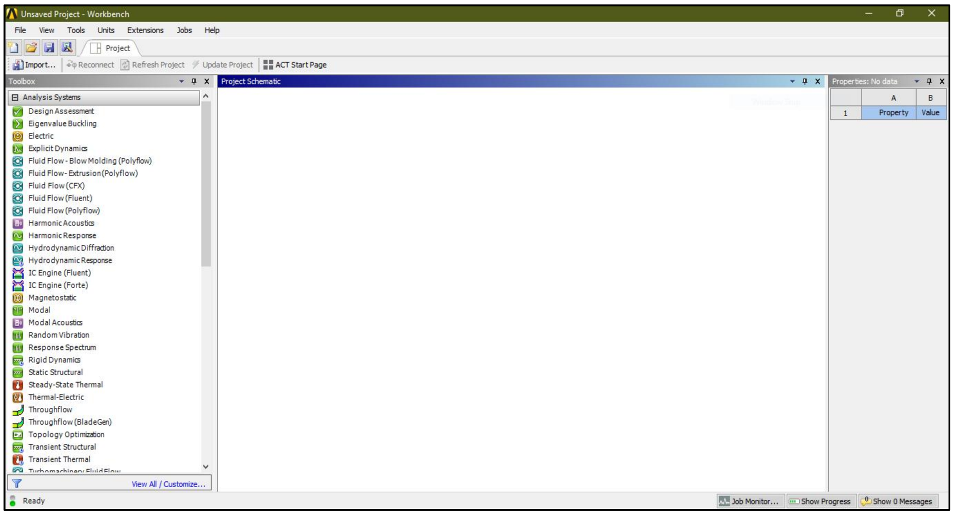
Finite element analysis (FEA) is the process of simulating the behaviour of a part or assembly under given conditions so that it can be assessed using the finite element method (FEM). FEA is used by engineers to help simulate physical phenomena and thereby reduce the need for physical prototypes, while allowing for the optimisation of components as part of the design process of a project. FEA uses mathematical models to understand and quantify the effects of real-world conditions on a part or assembly. These simulations, which are conducted via specialised software, allow engineers to locate potential problems in a design, including areas of tension and weak spots. With the use of mathematics it is possible to understand and quantify structural or fluid behaviour, wave propagation, thermal transport and other phenomena. Most of the processes can be described using partial differential equations (PDEs), but these complex equations need to be solved in order for parameters such as stress and strain rates to be estimated. FEA allows for an approximate solution to these problems. FEA is the basis of modern software simulation software, with the results usually shown on a computer-generated colour scale.

The easiest method to learn FEM is to look at how it is used in practise, which is finite element analysis (FEA). Engineers utilise FEA as a computer tool to perform engineering analysis. It includes the use of mesh generation techniques and FEM-coded software to break down a huge problem into smaller components. The Euler-Bernoulli beam equation, the heat equation, or the Navier-Stokes equations stated as PDE or integral equations are typical underlying physics in FEA, and the divided small pieces of the complex issue reflect different locations in the physical system.

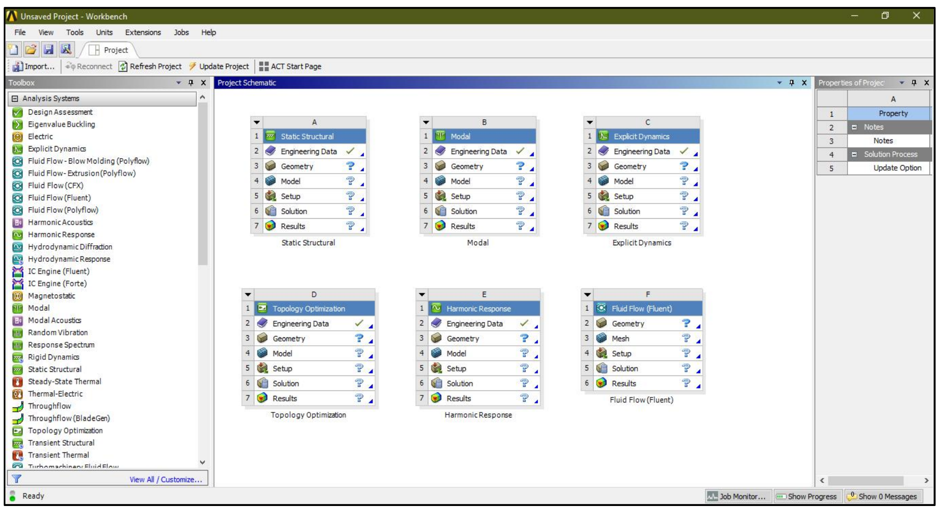
In present research for analysis ANSYS (**An**alysis **Sys**tem) software is used. Basically, its present FEM method to solve any problem. Following are steps in detail

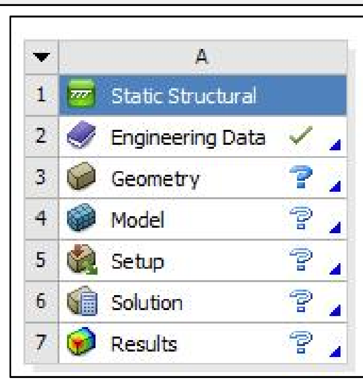
1. Geometry
2. Discretization (Meshing)
3. Boundary condition
4. Solve (Solution)
5. Interpretation of result.

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Impact Factor: 6.252
Volume 2, Issue 5, May 2022



Workbench includes static, modal, harmonic, explicit dynamics, CFD, ACP tool post, CFX, topology optimization, and other sorts of analysis according on the problem





IJARSCT

ISSN (Online) 2581-9429

International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)

Volume 2, Issue 5, May 2022

Impact Factor: 6.252

Material Properties

Table 1 Material properties of structure steel

Properties of Outline Row 3: Structural Steel

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	7850	kg m^-3
4	Isotropic Secant Coefficient of Thermal Expansion		
6	Isotropic Elasticity		
7	Derive from	Young's Modulus and Pois...	
8	Young's Modulus	2E+11	Pa
9	Poisson's Ratio	0.3	
10	Bulk Modulus	1.6667E+11	Pa
11	Shear Modulus	7.6923E+10	Pa

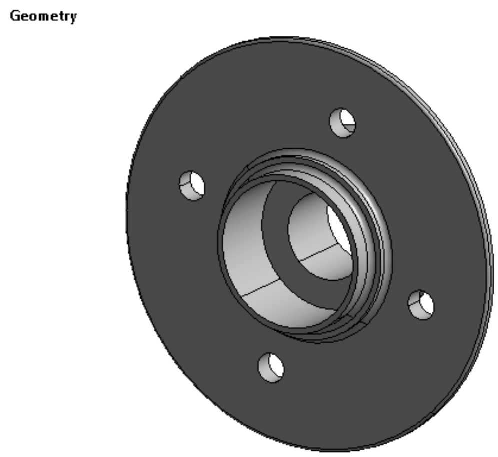


Figure: Catia model of wheel hub.

MESHING

ANSYS Meshing is a high-performance, intelligent, and automated general-purpose product. It generates the most accurate and efficient mesh for Multiphysics solutions. A mesh suitable for a given study can be built for all parts click. For the advanced user of a model with a single mouse who wishes to fine-tune the mesh, full control over the options used to generate it is provided. The power of parallel processing is automatically exploited to cut down on the time it takes to generate meshing.

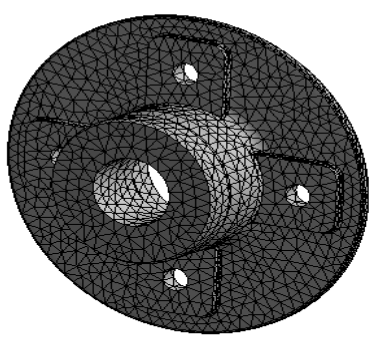


Figure: Mesh of wheel hub.

Details of "Body Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	5.0 mm

Statistics	
Nodes	26221
Elements	15217

After meshing of wheel hub nodes are 26221 and elements 15217.

Boundary Conditions

The setting of a known value for a displacement or a related load is a boundary condition for the model. For a particular node you can set either the load or the displacement but not both. The inside surface of the bolt mounting is given a fixed support. According to the vehicle specifications, an engine torque of (400 to 740) = 580 Nm is used.

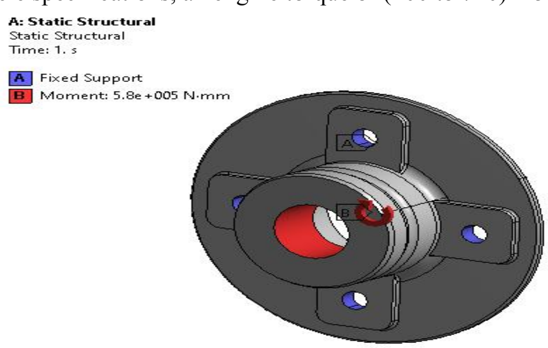


Figure: Boundary condition of wheel hub

Vehicle Hub - Front Hub Module for TUV MAHINDRA Discovery.

Total Deformation

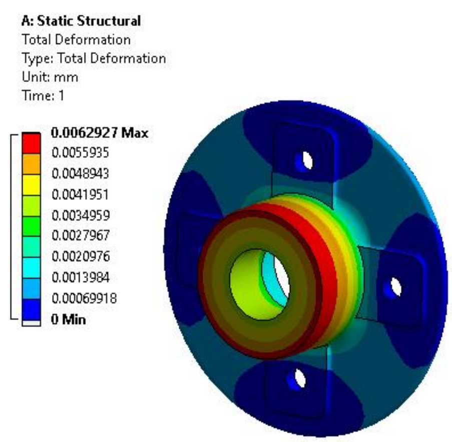


Figure: Total deformation of wheel hub

Maximum deformation under static condition of wheel hub was 0.00632mm

Equivalent stress

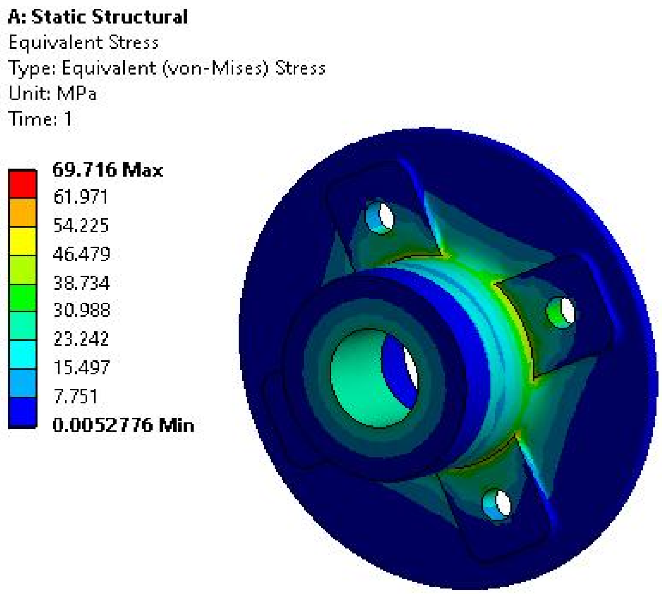


Figure: Equivalent stress **of** wheel hub

Maximum Equivalent stress of wheel hub was 69.716 Mpa.

III. 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 targets.

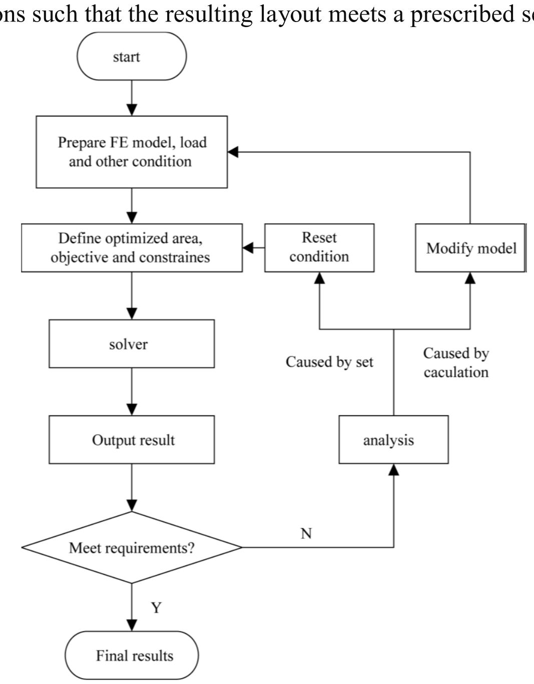


Fig. Process of Topology Optimization

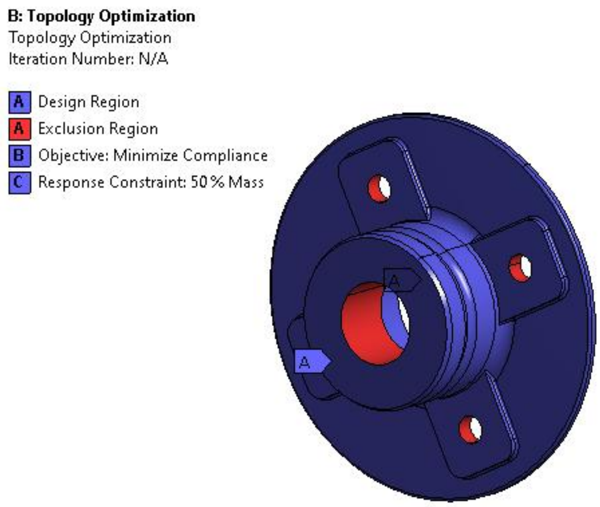


Fig. Details of boundary condition for topology optimization

Topology Element Density

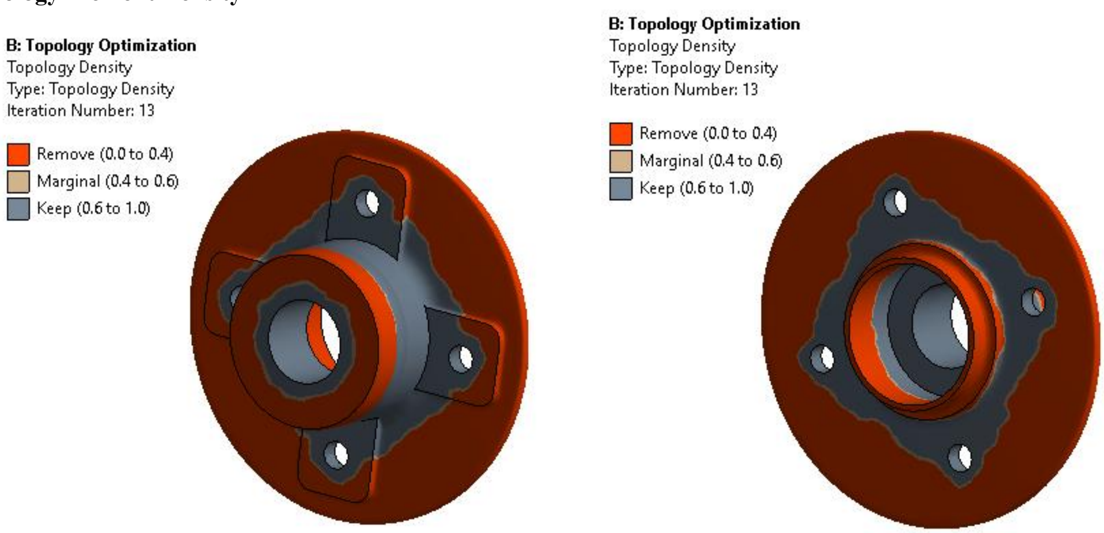
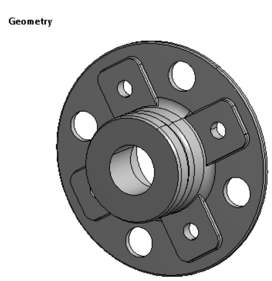


Fig. Topology optimization results

The Area Indicated in Red Region Provide Information About Removal of Material from that Area.

IV. Optimized Design



IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Impact Factor: 6.252
Volume 2, Issue 5, May 2022

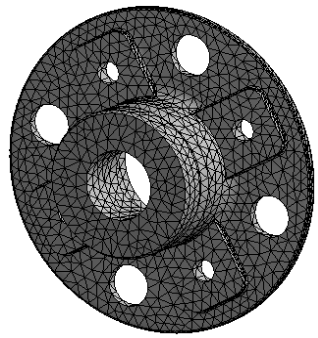


Fig. Details of meshing and final mass value

Details of "Body Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	5.0 mm

Statistics	
Nodes	25959
Elements	15035

Mass	0.98379 kg

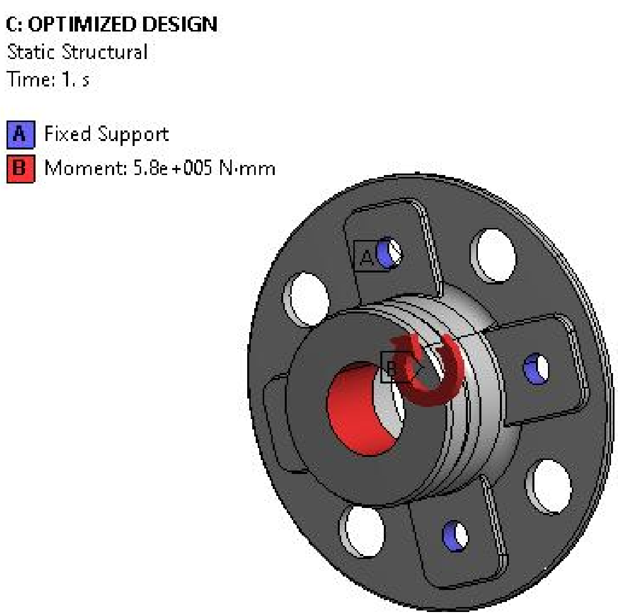


Fig. Details of boundary conditions

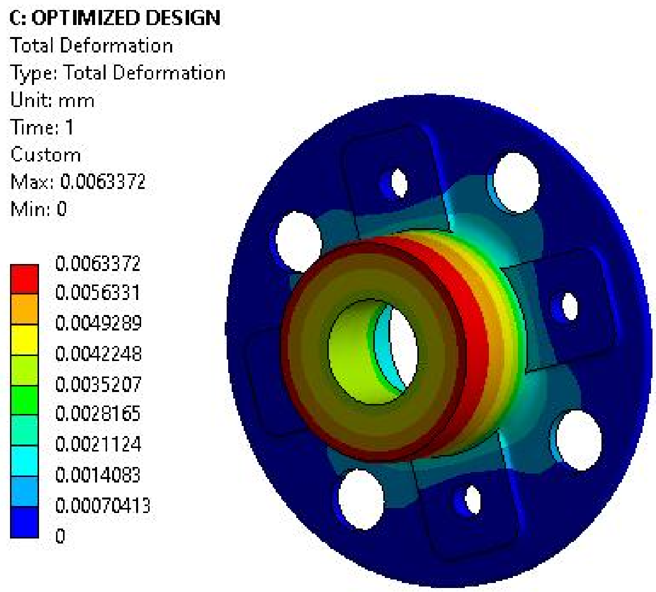
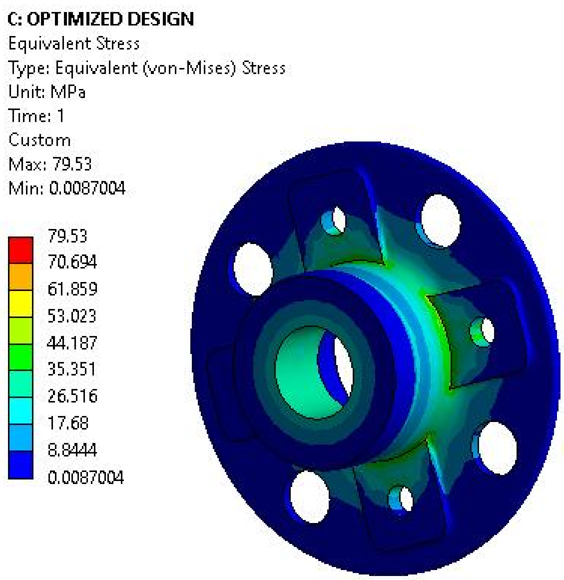


Fig. Total deformation results



-Fig. Equivalent stress results

V. Conclusion

- In the current study, static and topology optimization on wheel hubs is performed to reduce material costs and increase efficiency.
- The red region in topology optimization indicated the material removal area.
- The wheel hub's static structural analysis is performed to determine deformation and equivalent stress. The maximum deformation is 0.0062 mm, and the equivalent stress is 69.716 MPa. The topology optimization technique yields an optimised model.
- A weight loss of approximately 14% is observed.

IJARSCT

ISSN (Online) 2581-9429

International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)

IJARSCT
Impact Factor: 6.252

Volume 2, Issue 5, May 2022

References

[1]. Gerhard Fischer,' Design Criteria and Durability Approval of Wheel Hubs' SAE International by University of Edinburgh, Friday, August 10, 2018

[2]. Eun Ho Lee and Yong Soo Lee, 'Design and Validation of the Integrated Wheel Hub' SAE Technical Paper 2011-01-2170

[3]. Warwandkar P., Dubey, A., and Paroche, S., "Optimization of Wheel Hub Gasket and Cap Design to Prevent Grease Contamination and Bearing Failures," SAE Technical Paper 2017-26-0367, 2017.

[4]. CengizShevket,' Development of Low Friction and Light Weight Wheel Hub Units to Reduce Both the Brake Corner Un-Sprung Mass and Vehicle Co2 Emission' SAE Technical Paper.

[5]. Design and Analysis of Wheel Hub by Rohit B Pawar, Dr. N.K Nath, Dr. S.B Satpal.