ISSN 2277–5528
Impact Factor- 4.015
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & MANAGEMENT

Design and Implementation of Tractor Front Wheel Hub by the Help of Finite Element Analysis

Rohit Pardhi and Prof. P. K. Sharma
Department of Mechanical Engineering, NIIST, Bhopal, (M.P.), India

Abstract

In this thesis to study the work undertaken developing technique for the design of tractor front wheel hub for increased reliability of hub in whole working condition &weight reduction of wheel hub with the help of finite element analysis. Also increase the life of taper roller roller bearing & oil seal which are generally used in front tractor wheel hub. For this protection& smooth running, we design a cover to prevent the oil seal from mud& foreign material. Design a channel, for proper lubrication separation in hub & lubrication breathing system in running conditions. The weight and dimension of the hub must be small as possible because of the unsprungweight which further reduce the rotational mass. We use for better result which used in physical scenario. The objective was to reduce the mass of the wheel center and that has been achieved. With respect to the finite element formulation, the next important parameters are the mesh density and normal contact stiffness factor if augmented Lagrange is used. The mesh density affects the contact stiffness and the whole reaction force mostly in center where taper bearing in attach.The second goal of this study is to find out all the problem and changes to make a successful working of the wheel hub and reduce the weight of hub.

1.1 Introduction

Engineering components with optimum use of material and easy manufacturability is a direction where previous simulation through finite element method is found to be very useful. Front wheel hub of Tractor is one of the major and very main components and needs very good design as this part experiences the worst load condition of the whole tractor. The objective of this paper was to analysis the new design of the front axle of tractor, the results of the existing design were taken as basis for comparison with results of the proposed models. Based on the finite element analysis results, redesign was carried out for the front axle for weight optimization and easy manufacturability

1.2 Background

Hub is a structural part of the tractor which support the structural load of the tractor. Front wheel assembly system consist of axle beam, steering system, stub axle & wheel hub. Primary function of the hub is to support the weight of the tractor. It absorbs the shock due to undulated road surface. It absorb the torque applied on it due to braking of vehicle. The hub is the centre portion of the wheel and is the part where the wheel is attached to the suspension through the wheel carrier (or knuckle). The spokes radiate out from the hub and attach to the rim. The rim is the outer part of the wheel that holds the tyre.[15]

1.3. Main parts of wheel hub

Wheel – Refers to either the remaining major part of the detached wheel/tire assembly, or to the metallic part of the wheel, distinct from the attached tire. Also refers to complete wheels (i.e. without missing parts separated by fracture), with or without tires mounted on them. The wheels referred to in this report are stud-piloted wheels.

Rim – Refers to the outer part of the metallic wheel, originally welded at manufacture onto the center or hub portion to form the metallic wheel.

Hub – The central portion of the metallic wheel, originally welded at manufacture into the rim to form the wheel or the hub is the center of the wheel and typically house a bearing and is where the spokes meet. [16]The hubless wheel (also known as a rim rider or centerless wheel) is a type of wheel with no center hub.Morespecifically.the hub is actually almost as big as the wheel itself.The axle is hollow following the wheel at very close tolerance.

Spider – A ring-shaped piece that was originally integral to the wheel hub at the center of the metallic wheel. The circumferential fracture passing through the lug holes separated this piece from the remainder of the hub. The spider remained attached to the tractor after separation of the remainder of the wheel took place.

Spider Area – The central, substantially flat, area of a wheel hub that contains the lug nut holes.

Lug nuts – Also called outer cap nuts, these screw onto what are here called studs (or inner cap nuts) to hold the outer wheel of a pair of dual wheels onto the tractor.

Studs – Also called inner cap nuts, these screw onto inboard fasteners and hold the inner dual wheel on. The lug nuts or outer cap nuts screw onto the studs or inner cap nuts.[16]

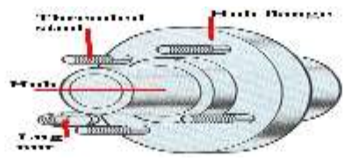


Figure 1–1Hub Basic Parts

Figure 1–2Model of Tractor Front Wheel Hub

- **Hub Failure/Problem Identification**

There are four primary causes of hub failures. They are lack of lubrication, overloading the vehicle, installing the axle nut too tight, or installing the axle nut too loose. Usually, hub failures are progressive and will produce some evidence of the impending failure. Evidence of a failing hub can include leaking hub seals, tire tread wear anomalies, sounds, smells, smoke, and steering wheel feedback.

1.4 Objective of this work:

To optimize the rear axle hub ensuring a proper way of material distribution, using Optistruct without affecting its functional/working conditions. The optimized design has to meet the requirements with respect to stress, displacement and natural frequencies. To create simulations of various wheel center designs that focus on reducing the mass of the current design.

1.5 Literature Review

Dilip K Mahanty et al.(2013) Engineering components with optimum use of material and easy manufacturability is a direction where prior simulation through finite element method is found to be very useful. Front Axle of Tractor is one of the major and very important component and needs very good design as this part experiences the worst load condition of the whole tractor. The objective of this paper was to analyse the new design of the front axle of tractor for Thirteen (13) different Certification Test load conditions. The existing design has no field failure reports; so the results of the existing design were taken as basis for comparison with results of the proposed models. Based on the finite element analysis results, redesign was carried out for the front axle for weight optimisation and easy manufacturability. This led to five proposed designs of the front axle which were evolved based on the above objectives. The proposed designs were evaluated for selected worst load cases of the existing design. The finite element analysis of new models yielded displacements and stresses close to the existing design. The increase in stresses were close to 15 % for all five models. The increase in displacement was not significant but all the new designs conceived had met the structural requirement. It was also observed that for the proposed designs there was a significant reduction in weight (approximately 40 %) and the proposed models did not involve a lot of welding, thereby significant savings of manufacturing was observed. The components used in the assembly were also found to be cost effective like smaller diameters bearing, smaller knuckle size etc. The reduction in cost of production and weight significantly reduced the cost of the new design of Front Axle. This analysis work showcases the use of finite element analysis as a method for reduction of cost in terms of materials and manufacturing.

Ravikant et al (2017) "Finite Element Analysis of Front Axle of Farm Tractor Using CAE Tools" Front Axle is attached to the front side of the Tractor and is used in the process of steering the machine towards right or left and is one of the major and very important components. Designing of the components is very important aspect, as this part experiences the worst load condition of the whole tractor. Our objective of the work is to carry out FEA analysis of front axle. The 3D model of front axle was generated in CATIA and then imported in ANSYS workbench. In this work static analysis of the front axle of a tractor has been taken as a case study. The Von Mises stress, strain and total deformation obtained for the same loading condition and compared with the existing results

Mohd Rehan et.al (2013) Pretending Through Ansys Of Residual Stresses In Cast Iron Tractor Hub This work presents a study and implementation of the pretending of residual stresses in molding. The objects of study are a cast iron tractor Hub part (provided by the company ESCORT, Faridabad) and an optimizedversion of the Hub resulting from the application of a topology optimization process. The models are solved through an uncoupled thermo-mechanicalsolidification analysis, performed both in the FE commercial software Ansys and the results.

The results shows that the part subjected to the topology optimization process develop less residual stresses than its original Hub. The implementation of residual stress analysis during the design of castings can lead to important improvements on the mechanical behavior of the final parts on aspects as crucial as fatigue life. Therefore, we strongly recommend the use of this type of numerical simulations as part of the design routine of casted parts. As well, the difference in the residual stress development of parts that has and has not undergone topology optimization procedures, suggest the benefits of the inclusion of shape optimization in the design process.

1.6 DESIGN AND ANALYSIS OF JOHN DEERE-USA CLUTCH HUB

The 3D model of clutch hub was drafted using CATIA V5 software and analysis of the hub was done for static loading condition. This project finds the maximum stress in failure region during operation. This project also suggests the design modification to the company to improve the lifetime of the clutch hub. As from the analysis result of the actual hub the maximum value of stress is very near to the theoretical endurance limit. Due to which the crack is formed on the hub during the earlier period of the testing. In order to increase the life of the hub, some of the modifications are done on the hub to reduce the maximum stress.

Shantanu Ramesh Shinde et al (2015) 'Advancement in Simulation of Front Axle of Tractor**'** the past few decades have witnessed rapid technological growth in the area of automobiles. One way of improving ones status in market is to keep the products price as low as possible, but with good reliability. This can be done by using CAE tools like FEA. FEA reduces the cost of testing. Previously the analysis of front axle was done without assembling the wheel hub, so the correlation of FEA results with experimental results was about 14-17%. After lots of study it was predicted that if the wheel-hub is assembled with the front axle and the forces are applied on the wheelhub,the results would be closer to the original values. The objective of this paper is to perform finite element analysis to the front axle system of tractor *with wheel-hub* for Bump test and Endurance structural test- forward (EST-FWD),Endurance structural test- reverse (EST-REV) to these two test load conditions and validation within 10% of actual stress measured experimentally**.**

HUB BASICS AND DESIGN

1.7 Classification

Based on GVWR commercial vehicles are classified as follows

- Light commercial vehicles (0-6.5 tonnes)
- Medium commercial vehicle (6.5-11.8tonnes)
- Heavy commercial vehicle (11.8 tonneson wards)

1.8 Basic Description of Wheel Hub

Wheel center is a revolving segment which offers help to the haggle in easymovement.It is the segment whereupon the wheel and brake mounts; it is fitted over the wheel bearings. A center point get together contains the wheel bearing, latches, seal and the center to mount the wheel. It may likewise contain the automated stopping device framework, wheel speed sensor which makes them exceptionally costly now and again. The course finished which the wheel center is mounted is again finished the axle. Single wheel application implies the back pivot contains just a solitary wheel on its either side.[15]

1.8.1 Methodology

Computational approach- Following software tools will be used

- **Pro-e** for modelling
- **Altaire** for analysis

- Experimental set-up-using tractor front wheel hub

Figure 3–1Hub for a single wheel application

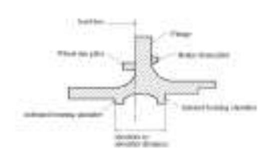


Figure 3–2Hub for a dual wheel application

- **Test fixtures**

Figure 3–3Test fixtures

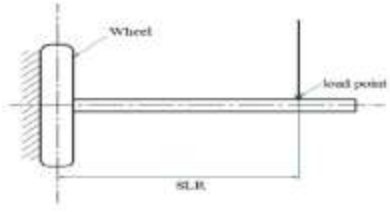
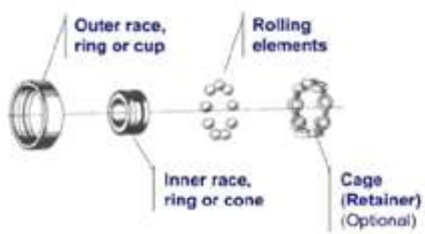


Figure 3–4Fixture with wheel fixed

Bearings-Wheel center orientations are frail parts in wheel driving framework in view of its terrible state of burdens. Two run of the mill outrageous working conditions for vehicle, specifically braking with turning and driving with turning, and dissected heaps of wheel center point direction. The outline parameters of wheel center point orientation are dissected independently, with the motivation behind finding the key parameters in them, which affected the power of bearing, lastly these essential parameters for auxiliary plan have been streamlined, the power of wheel center direction has been diminished by 45%.As the center point is a rotator segment, we utilize rolling contact heading. In moving contact direction, the contact between bearing surfaces is moving as opposed to sliding as incase sliding contact bearings. The favorable position of a moving contact orientation over a sliding contact bearing is that of a low starting friction. Because of this low grinding offered by moving contact bearings, these are otherwise called the counter erosion direction. Erosion in course prompts warm, higher torque expected to defeat the rubbing, wear all of which at last prompts the deterioration in execution of the heading.

Nomenclature



Calculations

- **Hub**
  - Wheel nut torque
  - Pin wheel contact stress
  - Wheel to hub friction
  - Thermal expansion
- **Bearings**
  - Loads for cornering
  - Bearing to upright interference
  - Bearing hub interference[18]
  - Reaction forces acting on the bearings
    - RA=F (z/y -1)
    - RB=F (z/y)
  - Pressure acting on the bearings
    - PA= RA/ A
    - PB= RB/ A
    - Moment arm = [0.7 (slr) + d]
    - Test load = moment/moment arm
    - Moment = [0.7 (slr) + d](S)(L)
    - Slr = static loaded radius
    - S = load acceleration factor
    - L = load rating of the hub (In this application the load rating being1.75 tonnes).
    - d = load offset value.

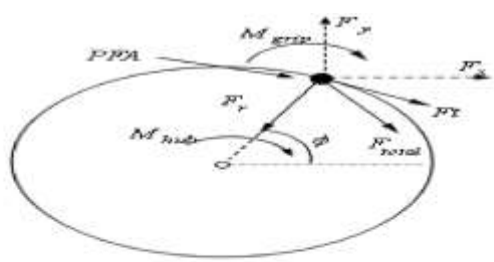


Figure 3–8Force Applied In Hub

ISSN 2277–5528
Impact Factor- 4.015

The Basic Cross Section

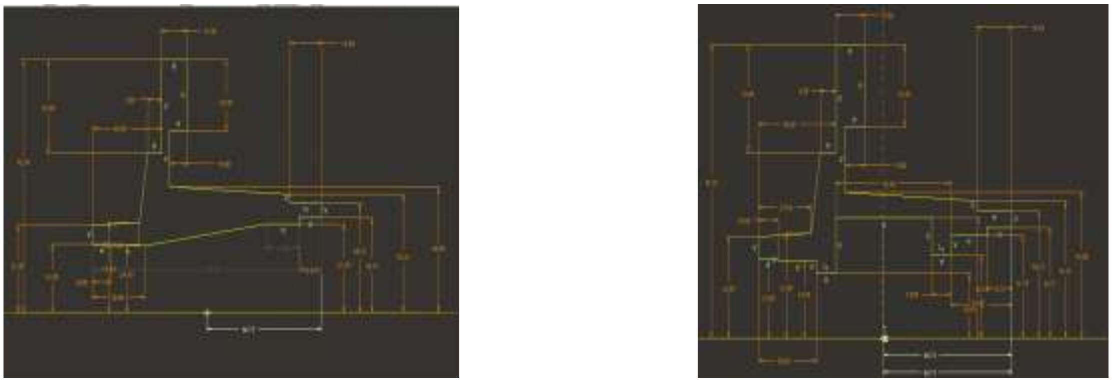


Figure 3–9Basic Cross Section Of Hub Front And Side View

2. The model cut section

Figure 3–10Cut Section Of Hub Front And Side View

Observation

Need for optimization exercise

To diminish the kerb weight and cost of the vehicle, it is required to streamline raise pivot wheel center point. Henceforth FE enhancement has been completed for conceivable zones of material diminishment. Furthermore the center being a pivoting segment which is a piece of unsprung mass diminishing the weight would mean better ride

Benefits of this advancement work out

Figure: Hub-cup

Figure: front-wheel-hub messy

Figure: front-wheel-hub Sonalika

Essential assembling contemplations has been considered while touching base at the advanced shape 10% weight sparing in the enhanced model Increase the characteristic recurrence of the hub center point The comparative plan idea can be reached out to other vehicle demonstrate in view of outline confirmation and approval trials.

Solid Modeling:

Strong Modeling is geometrical portrayal of a genuine protest without losing data the genuine question would have. It has volume and in this way, on the off chance that somebody gives an incentive to thickness of the material, it will have mass and inactivity. Dissimilar to the surface model, on the off chance that one makes an opening or cut in a strong model, another surface is naturally made and the model perceives which side of the surface is strong material. The most helpful thing about strong displaying is that it is difficult to make a PC demonstrate that is questionable or physically non-feasible.

Outline of center point by Solid Modeling:

The accessible center point configuration programming's are numerical in nature for the best possible demonstrating of the center point. Committed center plan programs play out the counts, which are important to make the genuine profile of the center. Be that as it may, CAD/CAM applications can do this in seconds to create a right center profile rapidly and effortlessly because of their graphical nature. They are graphical displaying apparatuses and there are a limited number of figurings they can perform and a limited number of focuses they plot along the circle.

Computer aided design frameworks estimated shapes, for example, center point by characterizing focuses along a bend and after that basically associating those focuses with a straight lines. The more focuses you can plot, the littler the lines are utilized to draw the bend. While they can plot a significant number of focuses along the bend, approaching the center, there is dependably a blunder because of the requirement for the product to inexact utilizing focuses and lines.

Devoted center point configuration programs permit to make a center point that is inside the AGMA or ISO quality rating. Truth be told the measures are now in corporate into numerous center plan programs. For application where exact center are a bit much as in the center plan itself, strong demonstrating frameworks are exceptionally helpful. Strong demonstrating is a decent downstream apparatus, useful for characterizing instrument ways for EDMs, lasers and different frameworks that can draw information from CAD frameworks. Strong displaying is likewise the reason for stereo lithography and other quick prototyping frameworks.

These abilities and applications make present day CAD/CAM frameworks, for example, Pro/Engineer extremely control full building plan device with an extraordinary arrangement to offer for the originator. The greater part of this adaptability is conceivable by the procedure called parametric demonstrating.

.**Modeling Hub InPro/Engineer**

Figure 4–1Pro-E Model Section of Hub Front and Side View

FINITE ELEMENT METHOD

Pre-Processing
-Geometry Definition
-Mesh Generation
-Simulation setup by Hyper mesh
Calculation
Solution of the governing differential equation

Post-Processing
-Result visualization
-Result preparation for comparison
-Result comparison
Solidification
Stress analysis
Strain

Table 5.1 Simulation Procedure

5.7 PROCEDURE

The simulation procedure steps are the following diagram:

Figure 5–2Optimization flowchart

Simulation Setup

Here a step by step procedure to setup and run first the thermal simulation and then the stress simulation in **Abaqus** is presented and commented

- Assembly
- Importing the models
- Materials definition
- Sections definition
- Sections assignment
- Mesh element type

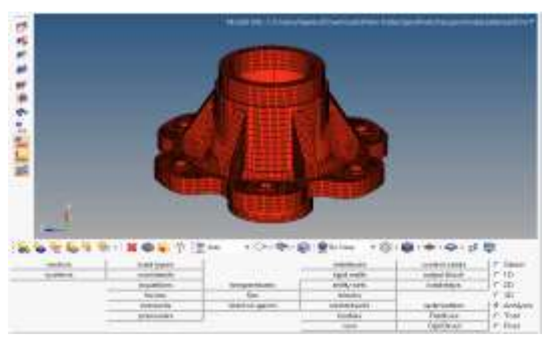


Figure 6–1Analysis model

Displacement

In finite element analysis the solution function used to represent the behavior of field variables in each element are called "displacement function" or "interpolation function". These functions are usually polynomial of linear, quadratic and cubic form. The accuracy of the solution can be improved by increasing the order of the polynomial.

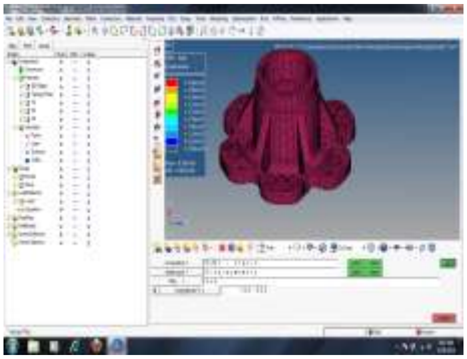


Figure 6–2Displacement Plot

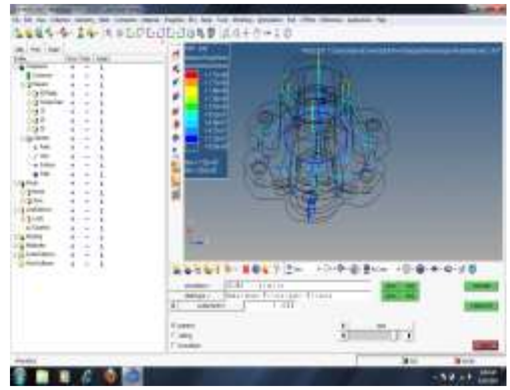


Figure 6–3 Intensity Plot

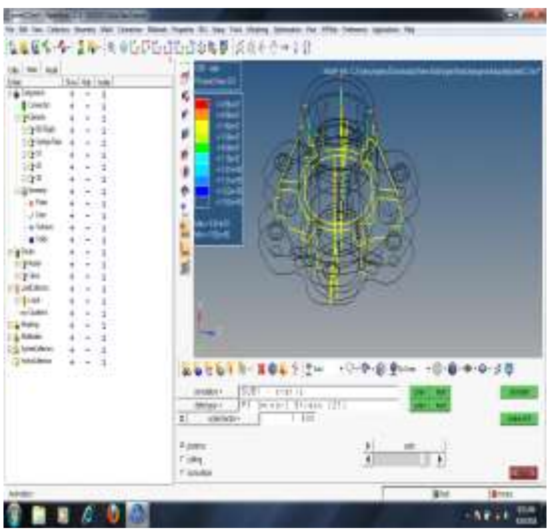


Figure 6–4Maximum Principal Stress

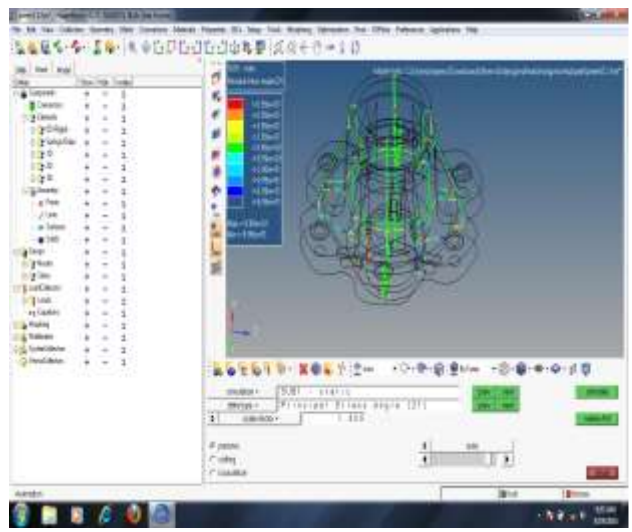


Figure 6–5Minor Stress Plot

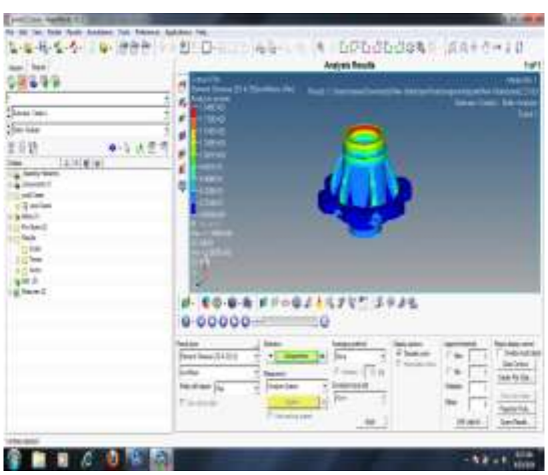


Figure 6–6Principal Stress Plot

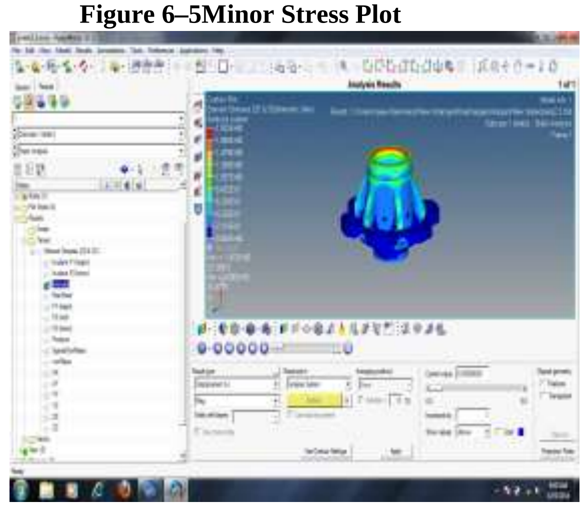


Figure 6–7Von Misses Stress Plot

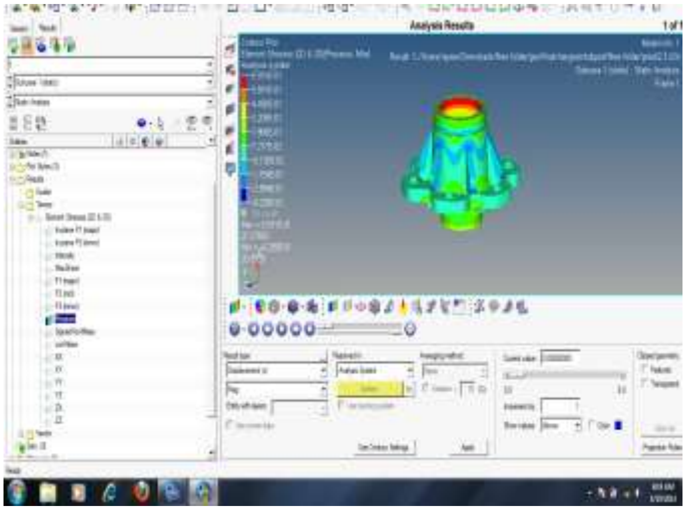


Figure 6.8 Pressure stress plot

RESULTS & DISCUSSION

In this thesis to have a look at the work undertaken developing technique for the layout of tractor the front wheel hub for increase reliability of hub in entire working condition &weight loss of wheel hub with the help of finite element evaluation. also growth the lifestyles of taper curler roller bearing & oil seal that are generally used in the front tractor wheel hub.for this safety& clean running, we layout a cowl to save you the oil seal from mud& foreign material. layout a channel, for correct lubrication separation in hub & lubrication breathing machine in jogging conditions. the load and size of the hub need to be small as possible due to the unsparing weight which similarly reduce the rotational mass. we use for higher result which used in physical state of affairs. the objective changed into to lessen the mass of the wheel middle and that has been carried out.

with respect to the finite detail formulation, the next vital parameters are the mesh density and regular touch stiffness aspect if augmented lagrange is used. the mesh density influences the touch stiffness and the whole response force generally in middle in which taper bearing in attach.

the second one goal of this examine is to find out all of the hassle and adjustments to make a a success running of the wheel hub and decrease the burden of hub.

Remolding the preliminary area as a tapered cantilever has enabled utilization of optistruct essential in the optimization method utilizing topology optimization. for a hub, the preliminary design area is adequately modified to be comparable to a cantilever segment predicated on the engineering judgment. Utility of optimization technique utilizing optistruct hub. for this reason, it's miles required to prepare geometry for appearing optimization to get the high-quality results from the put in force.

in this profile resulted in a weight reduction of 2.five kg in line with axle. The design consequences designate that changes may be made to reduce the mass of the wheel middle.

the implementation of residual pressure evaluation at some stage in the layout of castings can cause consequential amendments at the mechanical demeanor of the final additives on elements as vital as fatigue life. therefore, we vigorously advocate the utilization of this type of numerical simulations as a thing of the design ordinary of casted components. as properly, the difference inside the residual strain development of components that has and has not gone through topology optimization processes, recommend the benefits of the inclusion of shape optimization in the layout process.

the pressure, displacement, and safety elements are all well in the layout criteria, however the design standards avert extra consequential reductions inside the mass of the wheel middle. it's miles now just 0.656 kg and similarly adjustments could now not constitute a paramount reduction for the value and effort required.

Conclusion

Reshaping the initial domain as a tapered cantilever has enabled utilization of Optistruct critical in the optimization process utilizing topology optimization. For a hub, the initial design space is adequately modified to be akin to a cantilever section predicated on the engineering judgment. Application of optimization technique utilizing Optistruct hub. Hence, it is required to prepare geometry for performing optimization to get the best results from the implement.

On this profile resulted in a weight reduction of 2.5 Kg per axle. The design results designate that modifications can be made to reduce the mass of the wheel center.

The implementation of residual stress analysis during the design of castings can lead to consequential amendments on the mechanical demeanor of the final components on aspects as crucial as fatigue life. Consequently, we vigorously recommend the utilization of this type of numerical simulations as a component of the design routine of casted components. As well, the difference in the residual stress development of components that has and has not undergone topology optimization procedures, suggest the benefits of the inclusion of shape optimization in the design process.

The stress, displacement, and safety factors are all well within the design criteria, but the design criteria avert more consequential reductions in the mass of the wheel center. It is now just 0.656 kg and further modifications would not represent a paramount reduction for the cost and effort required.

Future work

This thesis can be an interest for researchers, instructors and postgraduate students who have great exuberance to work more on tractor wheel hub. It may give enlightenment about the characteristics of hub and evoke pervious works of sundry bodies that are involved in wheel hub research and engenderment. Furthermore this study contributes to a better hub design, avail technological institutions and all those who are intrigued with wheel hub. More work can be done to ameliorate this study and to obtain better output. Generally, the following areas are worthy for further research in the light of this thesis.

- Further three dimensional numerical method of investigation and study can be conducted on the analysis of bending and contact stresses for wheel hub.
- Further numerical method of investigation and study can be conducted on the whole wheel hub with all elements in the system including shaft and bearing.
- Further numerical method of investigation and study can be conducted on hub in mesh under dynamic condition with and without cracked surface pitting or wear.
- The bending and contact stress analysis of hub composed of composite materials utilizing three-dimensional finite element analyses can be recommended as future work.

REFERENCES

1. Analysis and weight reeducation of a tractor front axle –Dilip K Mahanty,Vikash Manohar,Bhushan S Khomane,Swarender Tata consultancy service India Swarpudgata,International Auto Limited India
2. O-OS-Poster1_Topology_optimization_of_Rear_AshokLeyland
3. C.S. KRISHNAMOORTHY ‘Finite Element Analysis Theory and Programming’ , Tata Mcgraw Hill. (1994)
4. SAE Car Wheel Center Exercise. Autodesk Inventor® Parametric Study
5. Version 2011 © European Aluminium Association
6. MoDESIGN AND ANALYSIS OF JOHN DEERE-USA CLUTCH HUB
7. Tao.Narang B.S. Automobile Engineering, Khanna Publication
8. Comparative Evaluation of Tractor Trolley Axle by Using Finite Element Analysis
9. Approach Sanjay Aloni PG(M.Tech CAD/CAM) Student of Department of Mechanical engineering, YCCE, Nagpur
10. Design consideration and durability approval of wheel hub-SAE international 11-16-1998; technical paper, author: Gerhard Fisher,Vatroslov V. Graubisic
11. Fracturer analysis of wheel hub fabcricated from pressure die aluminium assembly ;theoretical and applied fracturer mechanics, vol 09-02-1988 ;author :S.dhar
12. TFinite element modelling of dynamic impact and cornering fatigue of cast aluminium and forged magnetism road wheels.prequest dissertation and thesis 2006 ;author:Shang Shixian(Robert Braestrup,.
13. .Kutz M.Mechanical engineering handbook New York Wiley;1986
14. Klebanonov BM Barlam DM Nystrem FE ,machine elements-life and design ,Boea Raton :CRC press:2008
15. Design of wheel hub assembly of a single wheel hub application for commercial vehicle-Madhusudan Reddy,Manjunath M M,Shridhar Patil

16. R.S. Khurmi and Gupta" Theory of Machine".
17. Design and Analysis of Front Axle for Two Wheel Drive Tractor R. Oyyaravelu, K. Annamalai, M. Senthil Kumar, C.D. Naiju, Joel Michael
18. Load Analysis of Wheel Hub Bearing in Wheel Driving System with Exterior Rotor Motor- Jia Wu, Lu Xiong
19. Tim Stress Analysis and Safety Prediction of Alloy Wheel Hub- Wen Xue Qian, Xiao Wei Yin, Li Yang Xie
20. An anliyisis of the iteraction between the front and rear axle of a wheel four wheel drive tractor and its contribution to power delivery efficiency. ByIanto John Guy
21. Design of a half shaft and rear wheel hub assembly for a race car.Prof.Rahul Kumar,Prof B.K..
22. Prof. B.D.Shiwalkar " *Design of Machine Elements*" Central Techno publications, second edition,2011,pp.(19.1-19.20)
23. Prof. B.R.Kharde and Prof. B.R.Borkar "Stress Analysis of Gearbox"IRACST – Engineering Science and Technology Vol.2, No. 3, June 2012.
24. Bhandari V.B. " Design of Machine Elements" Tata McGraw Hill Publication , second edition-2010.pp (703-718)
25. S.S.Deshpande, N.S. Gokhale, "Practical finite element analysis", Finite to Infinite ,second edition, 2009, pp.111-207.
26. Zhang, J.J., Esat, I.I., Shi, and Y.H., Load Analysis with Varying Mesh Stiffness, Computers and Structures, 70,pp.273-280, 1999.
27. Prof.P.Seshu "Finite Element Analysis "PHI Learning Pvt .Ltd.,2003
28. Non linear analysis of farm tractor Front axle using ABAQUS,CAE,Dr. Sanjan Pawar ,Dr.Prakash Kamath Mr.Rajesh Upadhya
29. Modeling method for Complex Stucture System in Finite Element Simulating Analysis ; Xue Ling Zhang ,Ya Hui Hu ,Shu Feng Chai
30. Finite elemnt method by S.S.Rao
31. Fundamentals of finite elemnt analysis published by McGraw-Hill companies,lnc.1221 Avenue of theAmericas New York,NY 10020
32. Ali jafari, Majid Khanali, Hossein Mobli and Ali Rajabipour, "Stress Analysis of Front Axle of JD 955 Combine Harvester under Static Loading", Department of Agricultural Machinery, University of Tehran, Karaj, Iran*FEB,2006.
33. Ravikan ,Finite Element Analysis of Front Axle of Farm Tractor Using CAE Tools Volume 5 Issue VI, June 2017 IC Value: 45.98 ISSN: 2321-9653, International Journal for Research in Applied Science & Engineering Technology (IJRASET)