ScienceDirect

Materials Today: Proceedings

Volume 46, Part 17, 2021, Pages 7099-7104

Numerical and experimental analysis of torsional stress of traditional and modified steering yoke

Anand Koparde [a], Nithin Mithra [b], Sandeep Ahankari [b]

Show more

Share Cite

https://doi.org/10.1016/j.matpr.2020.10.167

Get rights and content

Abstract

Growing competition and inventions in the automobile sector calls for either the modification of available products or the replacement of the old ones by fresh and advanced types of material products. To conserve natural resources along with economizing energy, reducing weight has been the focal point of automobile makers in the current scenario. Due to the added luxury and safety features, the weight keeps on increasing, which in turn reduces fuel efficiency and the total performance of the automobile. Therefore, reducing the weight of the automobile is the prime need of the current automotive industry. Steering yoke is a critical component of a vehicle, which links the suspension and the steering system. The failure study of steering yoke assembly shows that about 71% of failure occurs in yoke part. This paper focuses on optimization of steering yoke by targeting uniformity of stress over the entire structure as an objective function, without compromising with required strength and stiffness of the structure. CAD model of steering yoke was prepared in CATIA. Hyper Mesh was used to prepare a finite element model with ABAQUS solver. Topology optimization with OptiStruct solver, one of the modules of Hyper Works, was utilized for optimization of material. Modifications were done to the geometrical model and was iterated till achieving satisfactory results. It was observed that the stresses were uniformly distributed over the entire structure and maximum stress in the yoke was reduced by 22.8%. The mass of the modified yoke was also reduced by 22.5%. The FEA results of modified yoke were verified by comparing it with torsion experiment to validate the model.

Introduction

For connecting rotating shafts that are coplanar and without coinciding, a universal joint is used. For transmitting motion and power, it is used, and is a positive and mechanical connection. The universal joint assembly has three major components: a cross trunnion and two yokes (flange and weld). Fig. 1 shows the basic single universal joint. The cross trunnion is used to deliver rotation from one yoke to another using four needle pin bearings [1].

The causes of failure and design of power transmission with universal joint have been analyzed by many researchers. Heyes etal. [2] studied about the common failures in automobiles and showed that the transmission system elements' failures cover 25% of all among the automobile failures. Some of the common reasons for failure may be due to faults in manufacturing and design, maintenance or material faults and also user originated faults. Bayrakceken etal. [3] has carried out the fracture analysis of the universal joint yoke and the drive shaft of a power transmission system of an automotive. Metallographic analysis, Spectroscopic analysis and hardness measurements have been conducted for each part, in order to determine the stress condition at the failed section, with the concluding that a failure happens due to the combined action of fatigue process. The crack beginning location of the joint yoke corresponds to highest stress points. The dynamics of the universal joint and its causes of failures have been studied by Vesali etal. [4]. They have collected many damaged samples of the universal joint and carefully examined the damaged sections, nature of the loads acting on the universal joint and on its bearing surface. This study suggested some remedies which can increase the performance and life expectancy of universal joints such as, use of an intermediate spring and damper to reduce the size of the impact load.

Some studies have also been carried out on optimization of geometry for steering yoke. Lingaiah etal. and Wagner etal. [not shown in ref.] have presented the work on optimization of geometry of universal joints by assuming smaller joint angles and neglecting the manufacturing tolerance. The concern of manufacturing tolerance has been added by Hummel etal. 1998 [5] for the design and optimization of cardan joints. As a result, the methodology development and the relationships derivation which were necessary for optimizing the geometry of universal joints with manufacturing tolerances was done. A check was also carried out for the geometric interference between various components. Lozica etal. 2012 [6] have presented the analysis of the effect of the geometric parameters variation on the stress level at joint yoke of the cardan shaft transmitter. As a result, they have concluded that even small variation in shape can lead to significant variation in stress distribution. The stress concentration can be reduced by fillet, but the amount of fillet provided should be optimum. Woischwill et al. [13] proposed a methodology which uses a decomposition approach to find a structure's optimal topology with multiple materials. An optimal joint design was found using different joint types. Zhang et al. [14] developed a framework for optimization of topology where topology along with the shape can be found at the same time using explicit boundary description along with evolution. Gandhi et al. [15] experimented by replacing conventional forged yoke type with sheet metal type. Verification of design was completed successfully to get an articulation angle of 55 degree, 0.19 degree of spring back and a 0.16 degrees maximum twisting angle. A static analysis of steering knuckle was conducted for weight optimization, using ANSYS workbench carried out by Sharma etal. [7]. This study develops the model of steering yoke which achieve 19.35% of weight reduction and maximum stress within control. At the end they have concluded that volume change and it's shape doesn't greatly influence the stiffness of structure. The weight optimization of steering yoke has been performed by Kulkarni etal. [8]. They have performed structural analysis and modal analysis of steering knuckle and developed a finite element model in HyperWorks. They have verified FEA results using analytical calculations and topology optimization to minimize the amount of material to be used was performed by OptiStruct solver.

A number of studies were carried out to study the material optimization of steering yoke. The effects of different composite materials for drive shaft have been presented by Krishna etal. [9] with the help of finite element analysis. They have presented the analysis on the drive shaft of Toyota Qualis with different composite materials like E-Glass epoxy resin, E-Glass polyester resin, E-Carbon and structural steel. As a result they concluded that E-Glass polyester resin induced stress reduction of 18.75% when compared to structural steel. They concluded with the possibility of replacing E-Carbon in place of the conventional material, structural steel, with the advantage of decreasing the weight and stresses which are induced in the drive shaft. Neve etal. [10] has stated that reduction of weight of vehicle is directly proportional to fuel consumption. Researchers in this study have made certain modifications in existing geometry of universal joint yoke and analysed identical boundary conditions with the help of ANSYS. They have carried out analysis on two composite materials such as carbon/epoxy composite, Kevlar/epoxy composite and obtained von-mises stresses, total deformation for each composite material. This study concluded that reduction in weight of universal joint yoke by using composite gives further advantage in increase in fuel economy of vehicle.

Some observation from earlier studies [11] confirmed that integration of robust design tools, (Design for manufacturing) DFM, (Design for manufacturing and assembly) DFMA, FEA and optimization techniques at initial stage of step lock orthotic knee joint design significantly reduces overall manufacturing cost, working stresses, while simultaneously increases the robustness of a joint. Later Tsavdaridis etal. [12] have carried out shape optimization of a rigid joint and concluded that it is necessary to consider the impact of joint stiffness in the process of shape optimization.

In this work, the stress distribution of steering yoke for given input torque condition is analyzed. The objective of this work is to distribute stresses uniformly over the entire structure of steering yoke. Topology optimization technique is utilized to achieve stress uniformity without compromising in strength and load carrying capacity of steering yoke. The main focus in this paper is minimization of the weight of a steering yoke and increasing the life expectancy of the same. What is being put into action involves the shape optimization, by the consideration of the impact created by stiffness in joints. This aspect was not seriously taken care of in some of the earlier works conducted on the analysis of the steering yoke. This shape optimization resulted in finding uniform stress distribution, which in turn proved to improve the life expectancy of the steering yoke.

Section snippets

Modeling and boundary conditions of steering yoke

C1021 was chosen as a material for steering yoke. The mechanical properties of C1021 steel is shown in below Table 1. A CAD model of the steering yoke was developed using CATIA V5. It has an upper steering yoke, lower steering yoke and cross shaft.

Finite element model was developed using Hyper Works. The second order tetrahedral meshing was utilized for meshing the geometry of the solid region. The meshed model of steering yoke with boundary conditions is shown in Fig. 2.

To observe maximum…

FEA stress analysis of traditional and modified steering yoke:-

At the top of the steering yoke,a torque of 350 NM was given and the component was analyzed for von-mises elemental stresses and displacements. The results were obtained from the Hyper View result viewer. Fig. 7 and Fig. 8 shows the stress distribution and displacement distribution of existing steering yoke respectively. The maximum stress observed (172.1 MPa) was far below the material yield strength (228 MPa). The maximum displacement observed (0.2081 mm) was at the location of corners of...

Conclusion

The existing steering yoke is modeled and topology optimization is carried out. The research presented in this paper has an objective of uniformity of stress distribution over the entire steering yoke assembly to avoid failure percentage of an existing steering yoke. The existing steering yoke is analyzed for 350 NM torque with OptiStruct solver, model is iterated until the stresses are observed to be uniformly distributed over the entire structure. The observed maximum stress for the modified...

CRediT authorship contribution statement

Anand Koparde: Software, validation, formal analysis, investigation, Writing-original draft. **Nithin Mithra:** Writing- review and editing. **Sandeep Ahankari:** Conceptualization, Methodology, Project Administration, Writing- review and editing, supervision....

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper....

Recommended articles

References (15)

A.M. Heyes

Automotive component failure

Eng. Fail. Anal. (1998)

H. Bayrakceken *et al.*

Two cases of failure in the power transmission system on vehicles: A universal joint yoke and a drive shaft

Eng. Fail. Anal. (2007)

K. Singh, “Automobile Engineering”, Standard Publishers Distributors, 127-217,...

F. Vesali *et al.*

Dynamics of universal joints, its failures and some propositions for practically improving its performance and life expectancy

J. Mech. Sci. Technol. (2012)

S.R. Hummela *et al.*
Configuration design and optimization of universal joints with manufacturing tolerances
Mech. Mach. Theory (2000)

L. Ivanovic *et al.*
Effect of geometrical parameters variation of the cardan joint yoke on its load capacity
Innov. Automotive Technol. (2012)

M.P. Sharma, D.S. Mevawala, H. Joshi, D.A. Patel, "Static analysis of steering knuckle and its shape optimization", J....

There are more references available in the full text version of this article.

Cited by (3)

Design of Intermediate Shaft Assembly Considering Strength, Rotational Durability and Torsional Durability ↗
2023, Transactions of the Korean Society of Automotive Engineers

Design of Lightweight Steering Intermediate Shaft Which Satisfies Requirements for Static Strength and Fatigue life ↗
2022, Transactions of the Korean Society of Automotive Engineers

Torsional strength analysis of universal joint's ZP-11A due to yokes modification and materials ↗
2022, Journal of Mechatronics, Electrical Power, and Vehicular Technology

View full text