International Journal of Analytical, Experimental and Finite Element Analysis
Vol. 7, Issue. 2, July 2020, pp 53 - 60
https://doi.org/10.26706/ijaefea.2.7.20200612

Design and Finite Element Analysis of Differential Cover for Rear Drive axle of a Light Commercial Vehicle (LCV)

Nikita A Duble[1]
nikitaduble.duble@gmail.com

A. D. Diwate[2]
addiwate@gmail.com

[1] ME Student, Department of Mechanical Engineering, JSPM Narhe Technical Campus, Pune, Maharashtra, India.

[2] Associate Professor, Department of Mechanical Engineering, TSSM BSCOER Pune, Maharashtra, India.

Abstract - **This work is intended to design differential cover based on existing cover. The cover is checked for structural stability by performing finite element analysis. Earlier observed issues like premature cover failure, bolt loosening (carrier to cover) and oil leakage from cover mating surface are rectified through finite element analysis. This is done by performing multiple FEA iterations by changing wall thickness, size of hole and number of holes. Fatigue life of differential cover obtained by finite element method is validated by experimental method. The model chose is that of a light commercial vehicle which has a gross axle weight rating 1050 kg. The cover material is SAPH 440 (Steel Automotive Pickled Hot-rolled and 440 MPa minimum tensile strength).**

Index terms - Differential cover, Drive axle, Gross axle weight rating

I. Introduction

The rear axle assembly is used on rear-wheel drive vehicles. This assembly is the final leg of the drive train. The axle assembly contains the drive head unit, the axle shafts, and the axle housing. Axle assemblies are exposed to substantial burdens from the motor and street. They are toughly developed and only occasionally come up short. In a axle assembly, motor force enters the drive pinion gear from the drive shaft gathering and differential pinion burden/spine. The drive pinion gear, which is in work with the ring gear, causes the ring apparatus to turn. The collaboration of the ring and drive pinion gears turns the force stream at a 90° point [1]. The distinction in the quantity of teeth on the ring and pinion gears causes a decrease gear proportion. This diminishes turning speed, while expanding torque. Power from the ring gear courses through the differential case, insect apparatuses, and side ringing's to the drive axles. The drive axles move power from the differential unit to the back wheels. Bearings and housing are main sub part of axle. Their function is to support and align the differential unit and drive axles. Bearings and housing are designed in such a way that they will not fail under hard usage. Seals and gaskets also play important role in safe working of axle assembly. Seals are used at the input to the drive head and in wheel hub [2]. Gaskets are used at housing mating surface, such as between the differential cover and the housing, to provide a tight seal from the outside. Differential covers play significant role in keeping your wheels turning. It allows easy servicing of differential fluid, create a sealed area that protects gears and gear fluid from dirt and moisture. Diff covers should likewise support the auxiliary unbending nature of the differential itself with the goal that the housing doesn't flex and misshape under burden. It should sustain unavoidable force and impacts. And it must dissipate heat to keep oil temperature within permissible limit inside drive head [3].

Research Article
Published online – 30 July 2020

Cite this article – Nikita A. Duble and A. D. Diwate, "Design and Finite Element Analysis of Differential Cover for Rear Drive axle of a Light Commercial Vehicle (LCV)", *International Journal of Analytical, Experimental and Finite Element Analysis*, RAME Publishers, vol. 7, issue 2, pp. 53-60, 2020.
https://doi.org/10.26706/ijaefea.2.7.20200612

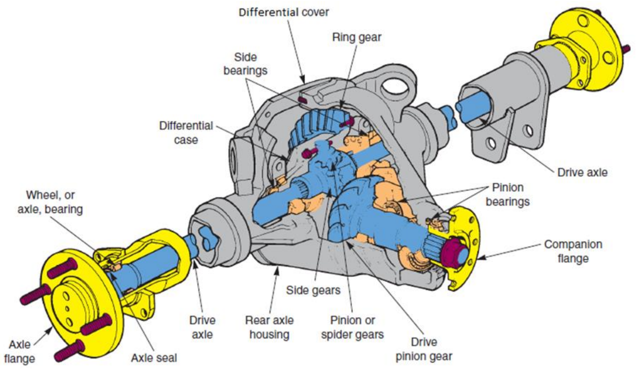

Figure 1: Rear Drive Axle [3]

In this paper the literature study has been carried out on various factors of differential covers. Using the knowledge from literature review, we can know the previous work undergone and its results and conclusion. Our intention is to carry forward the work to implement a new aspect of research, which can be useful for industries for their future projects.

Solidworks is used for creation of CAD model of axle housing. Cosmos software is used for examining stress, strain in axle housing. Whenever stress changed crack observed in axle housing. Then design is updated as per previous observations. The life cycle was improved to over 5.0×10^5 and satisfied the design requirement. Results provided from tests were compared with the analyses also. The method of finite element analysis turned effective in giving fatigue life of axle housing. This provide reference for the design of axle housing [1].

Many efforts are taken to develop the front axle design by considering the noise and vibration analysis at static and dynamic loading conditions. The chose model is a light commercial vehicle (LCV) with a vehicle load of around 5-10 tons. It has drop forged steel type depending upon the extent of total load the LCV experiences. The failure of LCV axle (front) under dynamic and static loading conditions is big concern to both goods and human lives, hence it is important to inspect the structural integrity of the axle to sustain. Axle beam is analyzed to obtain factor of safety and maximum deformation for vehicle static load, braking torque and turning force. Determination of load capacity of axle is motive of this research. Modal, static and transient analysis are performed on axle beam. Pro-E WildFire5.0 is used for creation of axle geometry whereas analysis is performed in ANSYS14.5. A mesh is created utilizing the product to evaluate the quality and ability of the item to get by against all powers and vibrations [2].

A basic idea about drive axle and its related components are given. It states that axle is a straight shaft that is fixed in location, it is combined with bearing or brushing use to mount rotating wheel or gears. The wheel or apparatus can be appended to it with an inherent outfitting or bushing. An orientation or brushing fits inside the focal point of the haggle it to turn without influencing the hub itself. The motivation behind pivot is to tie down the wheels or riggings to explicit areas comparative with different wheels or apparatus. The wheels would not stay fixed in position and the power and vehicle would make the wheel twist level. In vehicle two kinds of differential are utilized. Which are front pivot and back hub. The force created by the motor is moved to the wheels through grip, gear box, all-inclusive joints, propeller shaft, last drive, differential and back axles [3].

The design of structural modeling is usually based on the different geometric function. Since every component has a definite life span, it is necessary to calculate its core parameters. To discover the life expectancy of segment, the part should be as information boundary to the Finite Element Analysis. The Finite Element Analysis is only a numerical technique for tackling Engineering and Mathematical issues. The Analysis of the axle casing of tractor is done by using ANSYS. Various boundary conditions are imposed in order to get stress, strain and deformation plot. These broke down outcomes help to overhaul the back packaging of tractor. The optimized design is made in PRO/E. During updating the segment, different standards' taken in the genuine field must be considered. The analysis of the updated segment is carried out by giving various boundary conditions for both materials ‘SG 500” and “SG 200”. The basic outcomes like stress, strain is examined. The outcomes showed updated segment is safe. By the use of CAD/CAE tools better material is suggested for tractor axle casing [4].

Wu Zhijun et. al. describes about finite element analysis of Dong Fang Hong 40 tractor's drive axle. Finite element analysis is done in ANSYS workbench. Optimized model is created based on multi-objective fuzzy reliability. Applying hereditary calculation to settle the plan model, multi-objective fuzzy dependability vigor configuration model was changed into a few single-objective fuzzy unwavering quality structure models. Favorable parameters obtained by applying deflection constraint, static strength constraint, fatigue strength constraint, dimension constraint and fuzzy boundary condition constraint. After comparison of new design with old one mass was reduced by 20.54%, the sensitivity was also cut down by 33.08%. It is observed that reliability and robustness were improved. Joined with hereditary calculation, the fuzzy dependability vigor plan strategy has evident focal points than conventional strategies, and the ideal outcomes have been effectively applied [5].

Structure and advancement of axles weight is the significant for assembling and keep up the heaviness of axles with higher unbending nature under working conditions. A few endeavors for advancement of axles has been done by numerous scientists thinking about the static boundaries to run the segment for more secure working conditions. The weight decrease of the drive has a significant job in the overall weight decrease of the vehicle, on the off chance that it very well may be accomplished without increment in most and decline in quality and unwavering quality. It is conceivable to accomplish structure of front hub with less weight to build the first. The fundamental goal of this paper is to concentrate on to consider the different techniques favored for structuring and streamlining the axles for more secure working under different limit conditions [6].

For the investigation of front hub an examination was contrasted and partitioned in to two stages. In the initial step axle was design by analytical method. For this vehicle determination, its gross weight, payload limit, slowing down force utilized for subject to issue to discover the rule stresses and avoidance in the pillar has been utilized. In the subsequent advance front axle were demonstrated in CAD programming and examination in ANSYS programming [7].

Improvement of joint of axle is significant region to address the huge disappointment pace of fixing in the paddy land applications. The bolted joint associations require legitimate consideration and examination at the structure stage for a safeguard activity in administration [8].

Khairul Akmal Shamsuddin et. al. investigates an untimely disappointment that happened because of the higher stacking limit of the substantial vehicle is examined. To decide the explanation of the disappointment, a CAD model of the housing was created. The mechanical properties of the lodging material were resolved from the producer manual. By utilizing the given information, stress circulation investigation performed by utilizing limited component programming and weariness life are anticipated. Plan upgrade arrangements were proposed to expand the weariness life of the housing [9].

II. Methodology

A. Literature Survey

Research paper can help us in following areas; assistance on CAD model, finite element model in CAE, software tools required, the conditions required for applying various Constraints and how the loads are applied, these papers can also help us in selection of the material.

B. Measurement and Design

i. Measurement of dimensions of counter part of differential cover.
ii. Study the design parameters for modelling cover.
iii. Based on the dimensions measured and study of design parameters prepare basic outline of differential cover.
iv. Taking reference from above the 3D model will be created in Catia.
v. Saving the CAD model in the “.igs” format for Hypermesh [5].

C. FEA and Analysis

i. Meshing the CAD model of differential cover in the Hypermesh (preprocessor).
ii. Existing material Assignment to the cover.
iii. Calculation of boundary conditions
iv. Apply the Boundary conditions to the meshed model of cover.
v. Static analysis of cover model with boundary conditions using ANSYS solver (postprocessor).

D. Optimization

i. Change the thickness of cover and change hole size (carrier to cover).
ii. Static analysis of cover with new selected thickness and hole size.

E. Fabrication, Experimental validation, and Result

i. Prototype making
ii. Suitable experimentation.
iii. Comparing results from FEA and experimentation for verification [1].

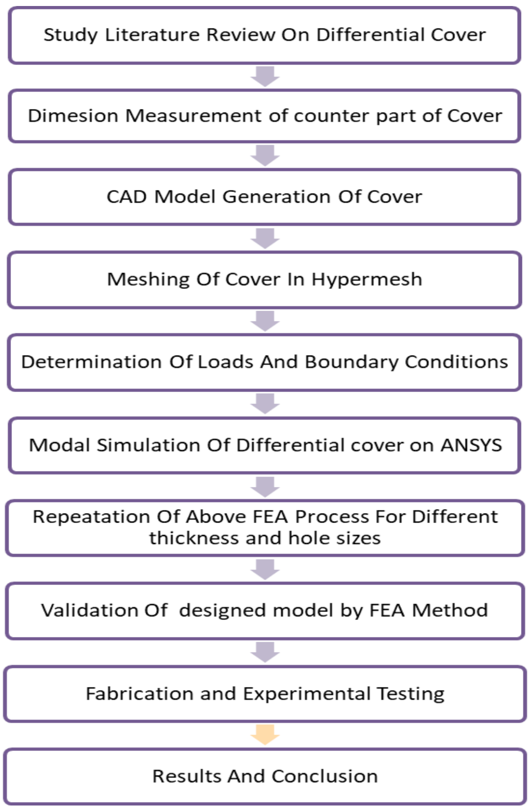

Figure 2: Methodology flowchart

IV. Design And Finite Element Analysis

A material for differential cover is selected as SAPH440 and finite element analysis has been carried out as per material properties given in table 2. The chemical composition of SAPH440 material is given in table 1.

TABLE I
CHEMICAL COMPOSITION

Chemical composition					
Grade	C	Si	Mn	P	S
SAPH440	<0.21	<0.3	<1.5	<0.025	<0.025

TABLE II
MECHANICAL PROPERTIES

Material	Yield Strength (Mpa)	Tensile Strength (Mpa)	Strain Hardening Index	Yield Ratio	Elongation (%)
SAPH440	315	440	0.17	0.643	33.3

A. CAD Model Generation

Based on dimensions obtained from existing cover and its counterpart optimized differential cover is modelled in CATIA.

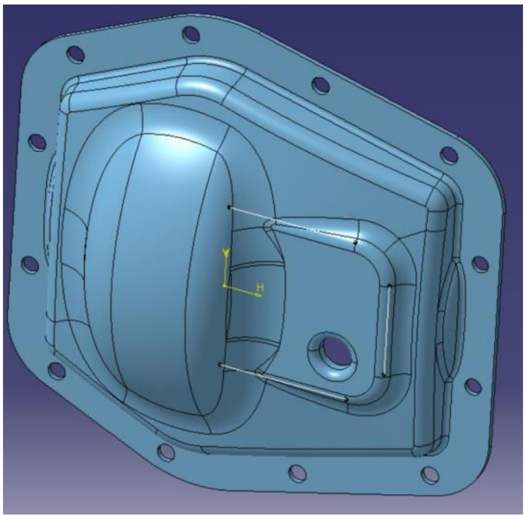

Figure 3: Optimum designed 3D model

B. Meshing

In order to carry out a finite element analysis, the model must be divided in to a number of small pieces known as finite element. In simple term, a mathematical net or “mesh” is required to carry out a finite element analysis. If the system is 1D in nature, we may use line element to represent our geometry and to carry out our analysis. If the problem is two dimensions, then a 2D mesh is required. Corresponding, if the problem is complex and a 3D representation of the continuum is required, then we use a 3D mesh. A very fine mesh creates the hardware space problem because the computation becomes voluminous. Out of several kinds of elements, solid elements are used for meshing a 3D component. The shape of solid elements can be a Hexahedron, Wedge, Tetrahedron or a Pyramid as per requirements. Figures 4 shows the configuration and the number of nodes required for each element.

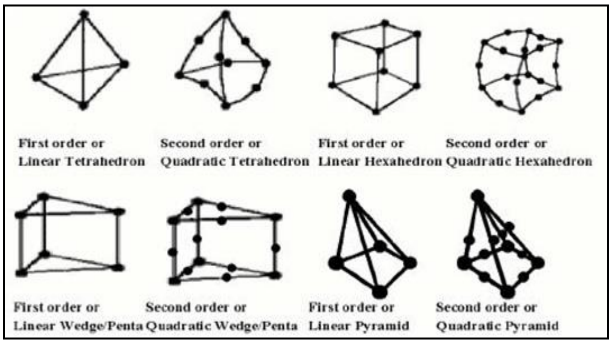

Figure 4: Solid Elements [6]

For meshing, file is imported in the meshing software HYPERMESH. The CAD data of differential cover is imported, and the surfaces were created and meshed. Cover is meshed using 3D tetrahedral element. For meshing a tooth, linear tetrahedron element is used. Very fine meshing has been selected to get accurate results. The size of the element is chosen to be 1 mm. Number of nodes generated is 1991152 and number of elements generated is 1180999. Figure 5 shows mesh model of differential cover.

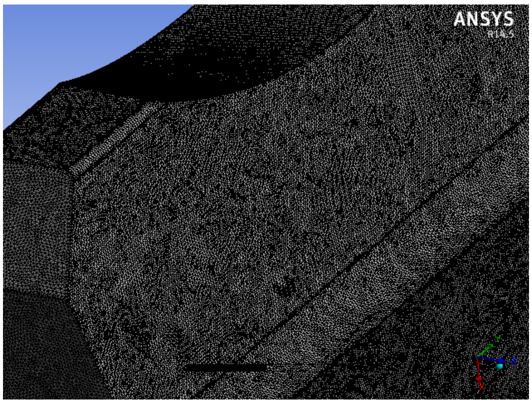

Figure 5: Meshing of differential cover

C. Load cases for fatigue analysis

a) Gross axle weight rating = 1050 kg
b) Vertical fatigue test - Vertical force: 2G to 3G
c) Vertical and braking test - Vertical force 1G to 2.8G (tension) at track points and braking force 0 to 2G at tire rolling radius in rearward direction only.
d) Vertical and cornering test – Vertical force 0.5G to 1.5G and cornering force 0.25G to 0.75G

D. Loading and boundary conditions

Boundary condition is applied in half portion. Axle model is symmetrical. Static analysis is performed for three cases which are vertical load, vertical + Braking load and Vertical + Cornering load with respective boundary conditions in order to calculate cycles achieved before failure [6].

i. Vertical load

This condition demonstrates loading of vehicle when it is standstill. For this vertical test, the model has been constrained at the wheel center point and 3G load (3×1050 = 3150 kgf) has been applied at the suspension pad where leaf spring will be mounted in the vehicle. The vertical load will be applied on both suspension pads simultaneously and numbers of cycles will be noted.

ii. Vertical and braking load

To simulate the condition of subjecting the vehicle to loading and braking, the above condition is applied. For braking load condition, the model has been constrained at the suspension bolt holes and at the top face of spring pad in vertical direction alone. At wheel center point vertical 2.8G load (2.8×1050 = 2940 kgf) is applied and at tire rolling radius braking load 2G (2×1050 = 2100 kgf). The combination of vertical and braking load is phased as follows: [6]

At the point when vertical force is at is at least value, the braking force is at its minimum.

At the point when vertical force is at is at most extreme value, the braking force is at maximum in backward direction.

iii. Vertical and cornering load

At the point when the vehicle is taking a turn right or left because of the heaviness of the vehicle and payload, vertical burden will follow up on the axle simultaneously because of the turning impact cornering load likewise applied on a level plane from side of the axle. For cornering load condition, the model has been limited at the spring bolt openings and at the spring cushion top face in the vertical bearing alone. At the wheel center point the vertical 1.5G load (1.5×1050 = 1575 kgf) and at tire rolling radius cornering load 0.75G (0.75×1050 = 786 kgf) is applied [1]. The combination of vertical and Cornering loads is phased as follows:

At the point when vertical force is at is at least value, the cornering force is at its minimum.

At the point when vertical force is at is at most extreme value, the cornering force is at maximum at inside direction [7].

iv. Various loading points

The figure shows that the loading point on the axle as vertical load on the spring pad combined vertical load at wheel track point and braking load at tire rolling radius which is perpendicular to the axle axis. Also combined vertical load at wheel track point and cornering load at tire rolling radius which is in line to the axle axis.

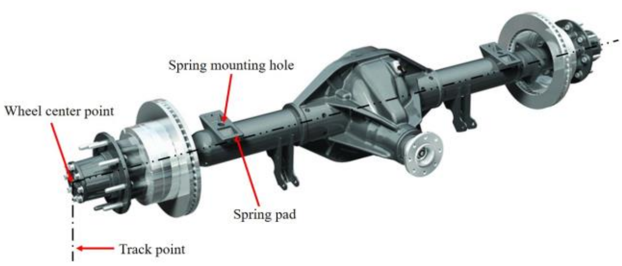

Figure 4: Loading locations [3]

v. Loading condition for gap analysis at cover mating surface

Force acting due to pre-stressed bolt to the cover is considered. Here a standard value of force divided by total number of bolts is applied on bolt locations of cover [5].

E. Analyzing Bolt Pretention

As the bolt size has been changed for differential cover, it is important to carry out bolt pretension analysis. Bolt pretension analysis is carried out in ANSYS. Below table shows the bolt pretention values. The threshold value for pretention is 100. All the values are below 100 hence selected bolt size is correct [5].

TABLE 3

BOLT PRETENTION VALUES

Bolt	Pretension value
Blt_1	83.9
Blt_2	83.9
Blt_3	84.6
Blt_4	82.9
Blt_5	83.7
Blt_6	84.3
Blt_7	84.5
Blt_8	93.4
Blt_9	83.2
Blt_10	84.2
Blt_11	83.5
Blt_12	83.8

F. Fatigue analysis of optimized design

The number of cycles achieved by optimized differential cover is 365050 cycles. Fatigue life of cover is more than 300000 cycles which is general requirement.

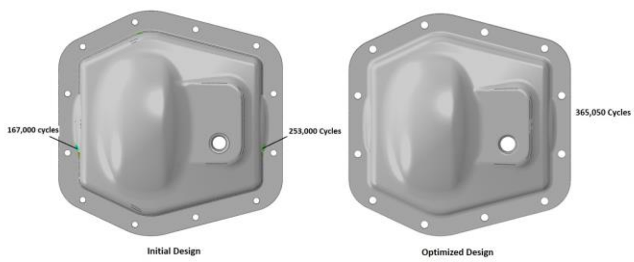

Figure 5: Life for fatigue analysis

The below figure shows leakage area on cover mating surface. Grey portion is indication of presence of oil leakage. For initial design oil leakage is observed whereas optimized design has not shown leakage area.

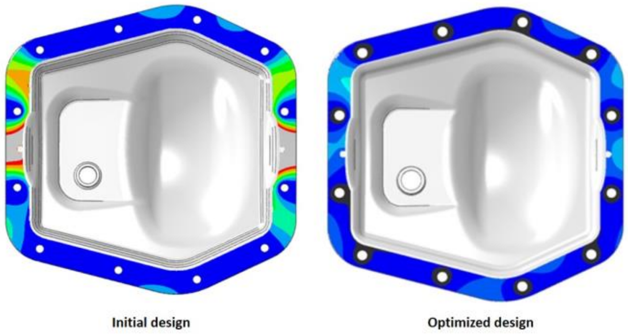

Figure 6: Oil leakage plot

TABLE 4
FATIGUE LIFE FROM FEA

Load case	Fatigue life cycle	Target life cycle
Vertica load	490587	300000
Vertical and braking load	10850	6000
Vertical and cornering load	16771	10000

V. EXPERIMENTATION

Below photo shows test setup for differential cover. The differential cover has been tested for fatigue by applying different load cases. Fatigue life cycle obtained by experimental fatigue test are given in table 5.

Figure 7: Experimental test setup [7]

TABLE 5
FATIGUE LIFE FROM EXPERIMENTATION

Load case	Fatigue life cycle	Target life cycle
Vertica load	457870	300000
Vertical and braking load	9981	6000
Vertical and cornering load	15790	10000

VI. RESULTS

The FEA and experimental fatigue life cycles are shown in below table.

TABLE 6
FATIGUE LIFE FROM FEA AND EXPERIMENTATION

Load case	FEA Fatigue cycle	Experimental Fatigue cycle	Target life cycle
Vertica load	490587	457870	300000
Vertical and braking load	10850	9981	6000
Vertical and cornering load	16771	15790	10000

VII. CONCLUSIONS

FEA and experimental results for fatigue life of cover have close resemblance. Both FEA and experimental fatigue cycles has achieved target life cycles. This shows optimized design of differential cover is reliable. Bolt pretention analysis values are below threshold value hence proving optimized bolt size is correct. Experimental testing of differential cover shows increase in numbers of bolt hole has eliminated oil leakage from cover mating surface. The referenced designing improvement process in this paper end up being helpful in diminishing the advancement time and cost by using CAD/CAE tools. This avoids rework at later stage. Experimental testing of cover validates optimized design differential cover suggested by FEA.

REFERENCES

[1] Sivasakthi D, R. Prabhu, “Tractor Rear Axle Casing”, *International Journal for Research in Applied Science & Engineering Technology (IJRASET)*, Volume 5, Issue IV, April 2017.

[2] Siddarth Dey, P.R.V.V.V Sri Rama Chandra Murthy. D, P.Baskar . "Structural Analysis of Front axle beam of a Light Commercial Vehicle (LCV)", *International Journal of Engineering Trends and Technology (IJETT)*, V11(5), pp. 208-213, May 2014. https://doi.org/10.14445/22315381/IJETT-V11P241

[3] Lalit Kumar, Chandrakant Singh, Bhumesh Kumar Dewangan, Prakash Kumar Sen, Shailendra Kumar Bohidar, “Study on the Front Axle and Rear Axle Attached to Differential System”, *International Journal for Innovative*

Research in Science & Technology, Volume 1, Issue 7, December 2014.

[4] Lu, S. K., Su, J. H., Liao, S. D., Su, J. Q., Wang, B., Yu, L., Jiang, Y. L., & Wen, S. H., "Finite Element Analysis on Fatigue Failure Prediction of a Rear Axle Housing of Vehicle Based on Cosmos", *Applied Mechanics and Materials*, Vol 121–126, 843–847, 2011. https://doi.org/10.4028/www.scientific.net/amm.121-126.843

[5] Wu Zhijun, Zhang Lihua, "Optimal Design for Rear Drive Axle House Based on Fuzzy Reliability Robust Analysis", *International Conference on Chemical, Material and Food Engineering (CMFE-2015),* Atlantic Press, pp. 875 - 880, 2015.

[6] Amol A. Sangule, Prof. Dalwe D.M., "A Review on Modeling and Analysis of Front Axle of Alto Maruti-800 LMV Car for Weight Reduction", *International Journal for Research in Applied Science & Engineering Technology (IJRASET)*, Volume 7, Issue IV, Apr 2019. https://doi.org/10.22214/ijraset.2019.4069

[7] Pravin R.Ahire, Prof. K. H. Munde, "Design and analysis of front axle for heavy commercial vehicle", *International Journal Of Engineering And Computer Science*, Volume 5 Issues 7 July 2016, Page No. 17333-17337. https://doi.org/10.18535/ijecs/v5i7.27

[8] Chetan D. Papat, Idris Poonawala, S.M.Gaikwad, "Design of axle housing bolted joint by analytical method", *IOSR Journal of Mechanical and Civil Engineering*, Volume 11, Issue 4 Ver. VI, Jul- Aug. 2014, PP 55-60.

[9] Khairul Akmal Shamsuddin, Mohd Syamil Tajuddin, Mohd Nurhidayat Zahelem, "Stress Distribution Analysis of Rear Axle Housing by using Finite Elements Analysis", *The International Journal of Engineering and Science*, Volume 3, Issue 10, Pages 53-61, 2014.