ISSN 2347 - 3983
Volume 4, No.4 April 2016
International Journal of Emerging Trends in Engineering Research
Available Online at http://www.warse.org/IJETER/static/pdf/file/ijeter01442016.pdf

ANALYSIS OF FOUR WHEELER FRONT AUTOMOTIVE AXLE THROUGH FINITE ELEMENT ANALYSIS

Rohit Bharat Patil[1], Girish Lonare[2],
[1] Bharti Vidhyapeet College of Engineering, India, rohit.b1988@gmail.com
[2] Bharti Vidhyapeet College of Engineering, India, girishlonare1977@gmail.com

ABSTRACT

In this study, front dead axle of a vehicle has been examined. The axle analysis in the paper is done with help of ANSYS software. The scope of the paper is to find out whether the axle is capable to work under given loads. Then static strength and dynamic characteristics of rear axle are analyzed in five typical load cases. The results show whether there is any scope for optimization of the axle in order to decrease the weight so as to increase the average of the vehicle keeping the system intact and risk free. Using ANSYS software can avoid expensive and time-consuming development loops, so the design period is shortened.

Keywords
Modeling, meshing, Analysis, ANSYS.

1. INTRODUCTION

The axles serve to transmit driving torque to the wheel, as well as to maintain the position of the wheels relative to each other and to the vehicle body. The axles in a system must also bear the weight of the vehicle plus any cargo. The front axle beam is one of the major parts of vehicle suspension system. It houses the steering assembly as well. About 35 to 40 percent of the total vehicle weight is taken up by the front axle. Hence proper design of the front axle beam is extremely crucial [1]. During last few decades due to global economic scenario optimum vehicle design is major concern. The conventional prototype testing is a versatile technique for evaluation of performance but it is time consuming, very costly and reproduction of test results and optimization of design is very difficult. Finite Element Analysis (FEA) is widely used in automobile sector for this purpose due to its versatility in scientific evaluation, reproducibility of the results.[1] Their structural strength, stability etc, are hardly evaluated resulting in reduced passenger safety, fuel efficiency etc. with increased possibility of maintenance, damages etc. Hence there is a need to ease of optimization etc with very low levels of cost implications [8]. To accomplish the need to design a moderate car, the structural engineer will need to use imaginative concepts. The demands on the automobile designer increased and changed rapidly, first to meet new safety requirements and later to reduce weight in order to satisfy fuel economy requirements. Experience could not be extended to new vehicle sizes, and performance data was not available on the new criteria. Mathematical modeling was therefore a logical avenue to explore. Most recently, the finite element method, a computer dependent numerical technique, has opened up a new approach to vehicle design. During the vehicle operation, road surface irregularity causes cyclic fluctuation of stresses on the axle, which is the main load carrying member. Therefore it is important to make sure whether the axle resists against the fatigue failure for a predicted service life. Axle experiences different loads in different direction, primarily vertical beaming or bending load due to curb weight and payload, torsion. Due to drive torque, cornering load and braking load. In real life scenario all these loads vary with time. Vertical beaming is one of the severe and frequent loads on an axle Due to their higher loading capacity; solid axles are typically used in the heavy commercial vehicles. During the vehicle service life, dynamic forces caused by the road surface roughness produce dynamic stresses and these forces lead to fatigue failure of axle housing, which is the main load carrying part of the assembly. Performing physical test for vertical beaming fatigue load is costly and time consuming. So there is a necessity to build FE models which can virtually simulate these loads and predict the behavior, even though the FEA produce fairly accurate results [1].

1.1 LOADS ON AXLE

When the vehicle is not in motion, the only job that the axle has to do is hold the wheels in proper alignment and support part of the weight. When the vehicle goes into motion, the axle receives the twisting stresses of driving and braking. When the vehicle operator applies the brakes, the brake shoes are pressed against the moving wheel drum. When the brakes are applied suddenly, the axle twists against the springs and actually twists out of its normal upright position. In addition to twisting during braking, the front axle also moves up and down as the wheels move over rough surfaces. Steering controls and linkages provide the means of turning the steering knuckles to steer the vehicle. As the vehicle makes a turn while moving, a side thrust is received at the wheels and transferred to the axle and springs. These forces act on the axle from many different directions. You can see, therefore, that the axle has to be quite rugged to keep all parts in proper alignment [1].

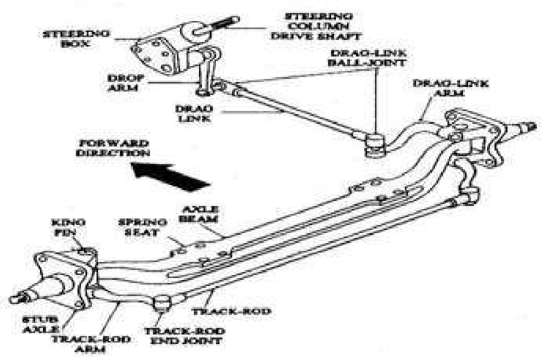

Figure 1: Layout of Front Axle

2 LITERATURE REVIEW

2.1 Stress Distribution Analysis of Rear Axle Housing by using Finite Elements Analysis

A premature failure that occurred due to the higher loading capacity of the heavy vehicle is studied by Khairul Akmal Shamsuddin and his associates for rear axle housing in his experiments. By using the given data, stress distribution analysis performed by using finite element software and fatigue life are predicted. Design enhancement solutions were proposed to increase the fatigue life of the housing. [6]

2.2 Design and analysis of front axle of heavy commercial vehicle

The approach of Ketan Vijay Dhande, Prashant Ulhe in their project has been divided into two steps. In the first step front axle was design by analytical method. For this, the vehicle specifications, its gross weight and payload capacity in order to find the stresses and deflection in the beam has been used. In the second step front axle were modeled in NX-CAD and meshed in HYPERMESH software module. The meshed model was solved in ANSYS software. [2]

2.3 Structural Analysis of Front axle beam of a Light Commercial Vehicle (LCV)-(2014)

The present work done by, Siddarth Dey, P.R.V.V.V Sri Rama Chandra Murthy. D, P.Baskar aims to determine the load capacity of the front rigid axle of a LCV and determine its behavior at static and dynamic conditions. This paper analyses the static, transient and modal analysis of the front axle beam. The geometry of axle is created in Pro-E WildFire5.0 software which is imported to ANSYS14.5. A fine congregate finite element model (meshed) is generated using the software to assess the strength and capability of the product to survive against all forces and vibrations. [1]

2.4 Static and Dynamic Strength Analysis on Rear Axle of Small Payload Off-highway Dump Trucks

Aiming at early breakage of rear axle in small payload off-highway dump trucks, Ji-xin Wang, Guo-qiang Wang, Shi-kui Luo, Dec-heng Zhou generated a 3D solid finite element model including nonlinear hydropneumatic suspension has been created by means of ANSYS software. Then static strength and dynamic characteristics of rear axle were analyzed in five typical load cases. According to the analytical results, the weak locations of rear axle were obtained and the modified design has been determined [3].

2.4 Weight Optimization of Front Axle Support by OptiStruct Technology Application (2012)

The front axle requires a properly designed support with high strength and stiffness. In the research carried out by Vinod Kumar Verma *Deputy Manager (R&D, CAE) Mahindra Engineering Services a* baseline analysis was carried out on the complete front axle assembly to extract the stress and displacement in the system. Then the Topology Optimization was been done on FAS for better material distribution. The optimization result gives the guideline for material removal and developing an optimized front axle support. In the paper, they present a methodology for designing a front axle support using OptiStruct module of HyperWorks. A mass reduction of 12% with desired strength and considering the manufacturing feasibility is achieved with 50% of design cycle time by performing the optimization analysis. [5]

2.5 Analysis and Weight Reduction of a Tractor's Front Axle

In this paper by Dilip K Mahanty, Vikas Manohar, Bhushan S Khomane, Swarnendu Nayak to analyse the new design of the front axle of tractor for Thirteen (13) different Certification Test load conditions. The existing design had 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 was 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 [4].

2.6 Failure Analysis of Induction Hardened Automotive Axles

The present work done by C. Kendall Clarke, Don Halimunanda, aims to provide a fractography/ fracture mechanic approach to making the determination of when the axle failed. Full scale tests on axle assemblies and suspensions provided data for fracture toughness in the induction hardened outer case on the axle. These tests also demonstrated that roller bearing indentions on the axle journal, cross pin indentation on the end of the axle, and axle bending can be accounted for by spring energy release following axle failure. Pre-existing cracks in the induction hardened axle are small and are often difficult to see without a microscope. The pre-existing crack morphology was intergranular fracture in the axles studied. An estimate of the force required to cause the axle fracture can be made using the measured crack size, fracture toughness determined from these tests, and linear elastic fracture mechanics. The axle can be reliably said to have failed prior to rollover if the estimated force for failure is equal to or less than forces imposed on the axle during events leading to the rollover[7] .

3. FINITE ELEMENT ANALYSIS

Using ANSYS software can avoid expensive and time-consuming development loops, so the design period is shortened. [3]

3.1 CAD model

Figure 3.1 gives the 3D CAD model of the front axle which is imported into the ANSYS software using importing tool. The model was prepared in Creo Parametric.

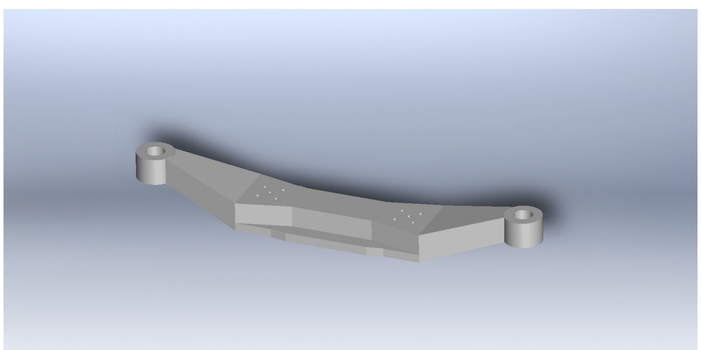

Figure 3.1: CAD Model

3.2 Boundary Conditions

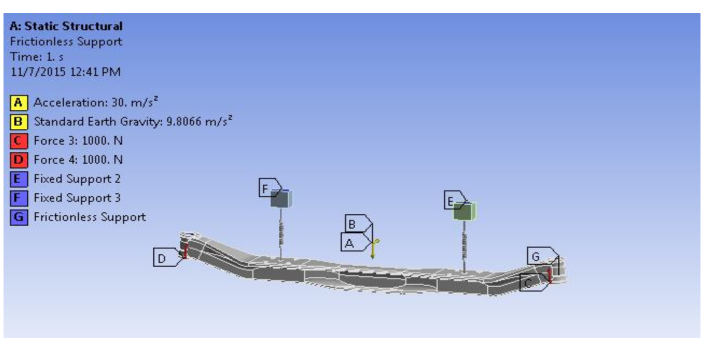

Figure 3.2: CAD Model with boundary Conditions

The springs shown in the model (figure 3.2) are equivalent to the leaf springs attached to the axle. The stiffness of these springs are assumed to be 66000 N/m and has been calculated keeping in mind the dimensions and number of springs and the span length between them

No. of Leafs = 7

Width of each leaf ~ 40 mm

Thickness of each leaf = 6 mm

Span Length = 800 mm

For simplicity and reduce computational time the leaf spring has been replaced by 2 rigid blocks(figure 3.2 F & E) provided fixed support and preload of 50000 N which account for the weight of the vehicle. The forces 1000 N(figure E & F) account from a reference considering shock coming from the roads .A 3g acceleration is considered to account for braking force acting on the axle when breaks are applied on the vehicle. Frictionless support is provided t in account of accommodating the king pin which will avoid the deflection in lateral direction.

3.3 Meshing

Figure 3.3 shows the meshed CAD model of our front axle. A very high mesh quality is being used in order to get more precise values. The details relating to the meshing are mentioned below in Table 1.

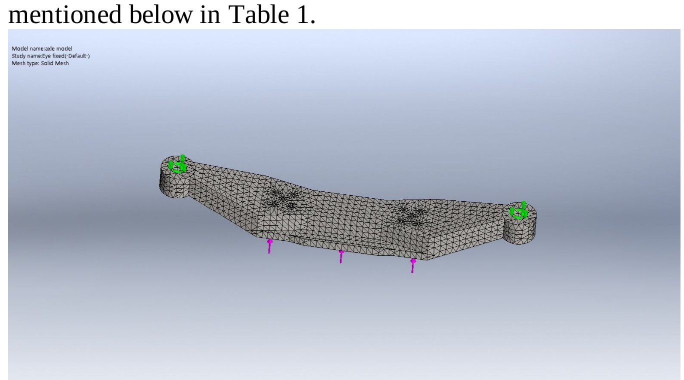

Figure 3.3: CAD Model Meshing

Table 1: Meshing Properties

Study Name	Eye Fixed (Default)
Mesh Type	Solid Mesh
Mesher Used	Standard Mesher
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian Points	4
Element Size	3.69681 mm
Tolerance	0.184841
Mesh Quality	High
Total Nodes	33928
Total Elements	21864
Maximum Aspect Ratio	12.736
Percentage of elements with aspect ratio < 3	97.3
Percentage of elements with aspect ratio > 10	0.439
% of distorted elements (Jocobian)	0
Time to complete mesh(hh.mm.ss)	00.00.02
Computer Name	Rohit

4 RESULTS

4.1 Total Deformation

As shown in figure 4.1 it is found that maximum deformation obtained is at the king pin joint where the area is red showing the area in concern.

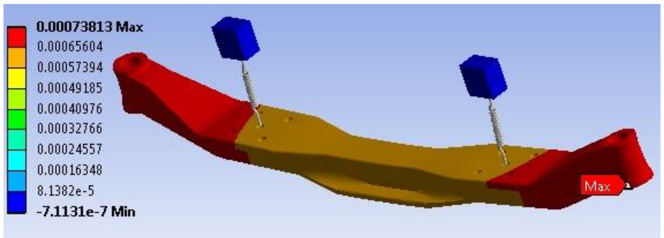

Figure 4.1: Total Deformation

4.2 Von-Mises Stress Distribution

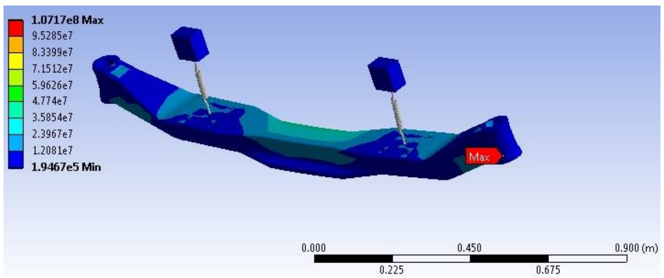

Figure 4.2: Von-Mises Stress distribution

Figure 4.2 gives the von-Mises stress distribution of the front axle. It is found that maximum stress obtained is at the king pin joint.

4.3 Factor Of Safety

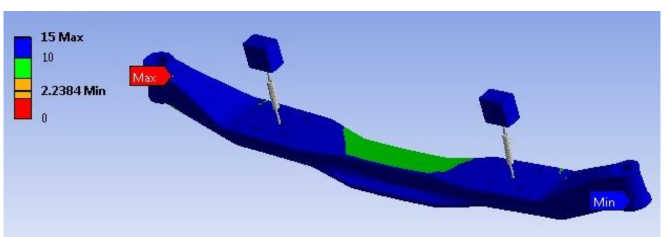

Figure 4.3: Factor of Safety

From the figure 4.3 it is found that very high factor of safety is seen through the analysis in the blue region.

5 CONCLUSIONS

From the blue color it is seen that there is a very high factor of safety in these regions and hence optimization can be performed on this geometry.

6 FUTURE SCOPES

This analysis can be extended to Dynamic condition that is when eye is fixed to chassis with pin joint. That is eye has movement in that. There is a need to consider that movement for really working conditions

REFERENCES

1. 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), (IJETT) – Volume 11 Number 5 - May 2014
2. Ketan Vijay Dhande, Prashant Ulhe, DESIGN AND ANALYSIS OF FRONT AXLE OF HEAVY COMMERCIAL VEHICLE, (IJSTM)- Volume No.03, Issue No. 12, December 2014
3. Ji-xin Wang, Guo-qiang Wang, Shi-kui Luo, Dec-heng Zhou, Static and Dynamic Strength Analysis on Rear Axle of Small Payload Off-highway Dump Trucks.
4. Dilip K Mahanty, Vikas Manohar, Bhushan S Khomane, Swarnendu Nayak- Analysis and Weight Reduction of a Tractor's Front Axle-Tata Consultancy Services, India-Swarup Udgata -International Auto Limited,India
5. Vinod Kumar Verma- Dinesh Redkar- Arun Mahajan-Weight Optimization of Front Axle Support By OptiStruct Technology Application-Mahindra rise
6. Khairul Akmal Shamsuddin, 2Mohd Syamil Tajuddin,3Megat Mohd Amzari Megat Mohd Aris,4Mohd Nurhidayat Zahelem, Stress Distribution Analysis of Rear Axle Housing by using Finite Elements Analysis , (IJES)-Volume 3,Issue 10,Pages 53-61,2014
7. C. Kendall Clarke, Don Halimunanda, Failure Analysis of Induction Hardened Automotive Axles, ASM International 2008, 16 May 2008
8. "Criticality Analysis of Passenger Bus Body Structural Design Using Finite Element Method", Sanlosh B. Belure, Salish S. Kadam and S. B. Wadkar; National Conference on computational methods in Mechanical Engineering; Osmana University, Hyderabad; September 16, 2008.