

Modification of Support of Front Axle for Reliability

Mr. Sharadchandra V Kantute and Mr. Prashant R Wankhede
Lecturer, Bharati Vidyapeeth Institute of Technology, Navi Mumbai, India
sharad2555@gmail.com and prw1977@gmail.com

Abstract: *The front axle support is the part which holds the engine and also gives support to it and lies between engine and front axle . According to the present market demand of off highway vehicle the low cost and light weight vehicle is in demand to fulfill the requirement of cost efficient vehicle. In this paper analysis of front axle support is done for study of stress generated in the component and then after optimization of its shape and according to the shape its weight will also reduced. Considering the effect of forces acted on such a heavy parts in designed by casting having dynamic loads of less frequency with greater amplitude may cause great damage to the component. According to the production techniques of components in tractor front axle requires a properly designed support with high stiffness. The design of component was modeled in Creo parametric 2.0 and the analysis was performed in solid works. Shape optimization technique is used for performing optimization cause miserable reduction in weight of connecting rod. The optimized component is 10.35% lighter compare to initial design*

Keywords: Axle, Front axle support (FAS), Computer Aided Design (CAD), Finite Element Analysis (FEA)

I. INTRODUCTION

The finite element method is applied in solid works. We have used CAE to shorten this development thereby reducing the tests. CAE tools are widely used in the automotive industries. In fact, their use has enabled the automakers to reduce product development cost and time while improving the safety, comfort, and durability of the vehicles they produce. Heavy duty vehicles like Dumpers which are widely used for many purposes in different fields having prime mover component as engine and idle component as axle etc. front axle support is one of the idle component in the Vehicle which lies between the engine and the front axle. The basic function of front axle support is to carry the loads coming out from the engine which is vertically downwards and to give support to the engine. It also hold the engine and the axle and to carry out the loads coming out as a reaction force from the ground. So this type of idle part plays an important role to give a complete model to the whole body. So its design is very important with regard to the bearing of all load carried by them.

II. MATERIAL

The basic requirement of material to design this front axle support is it must have good shock resistance capacity also have good compressive strength. For the fulfillment of this requirement grey cast iron and nodular cast iron are the best options.

Table-1 Nominal composition of grey and nodular cast iron

Grey cast iron	C- 3.4%	Si- 1.8%	Mn- 0.5%	-	-
Nodular cast iron	C- 3.4%	P- 0.1%	Mn- 0.4%	Ni- 1.0%	Mg- 0.06%

Since grey cast iron has less tensile strength and shock resistance than steel, but its compressive strength is comparable to low and medium carbon steel and Tiny amounts of magnesium or cerium added to these alloys slow down the growth of graphite precipitates by bonding to the edges of the graphite planes. Along with careful control of other elements and timing, this allows the carbon to separate as spheroid particles as the material solidifies. The

properties are similar to malleable iron, but parts can be cast with larger sections. So considering above two properties nodular cast iron is best for manufacturing front axle support.

III. DESIGN PARAMETER

Table-2 Parameters for materials

PARAMETER	GREY CASTIRON	NODULARIRON
Young's Modulus	$6.61781e^{+010}$ N/m ²	$1.9e^{+011}$ N/m ²
Poisson's Ratio	0.27	0.27
Tensile Yield Strength	151658000 N/m ²	413613000 N/m ²
Density	7200 kg/m ³	7300 kg/m ³
Thermal Expansion	$1.2e^{-005}$ /K	$1.2e^{-005}$ /K

IV. CAD MODELLING AND ANALYSIS

CAD Modeling is the base of any project. Finite Element software will consider shapes, whatever is made in CAD model. The model of the Front axle support is prepared by using CREO PARAMETRIC 2.0 software. Figure1, 2, 3 below shows the design of front axle support, mesh model of FAS and deflection produced after analysis

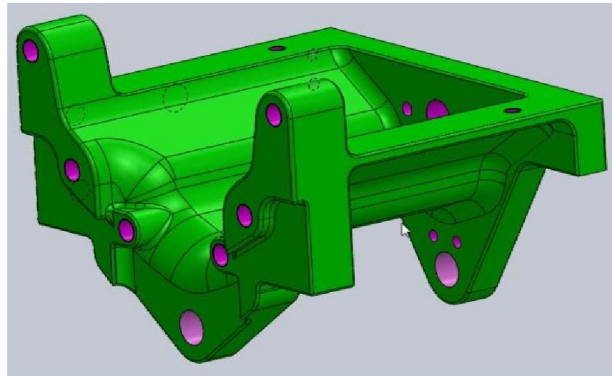


Figure-1 Design of front axle support

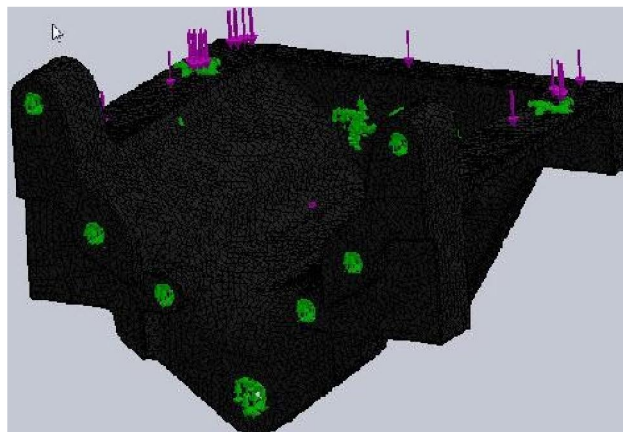


Figure-2 Mesh model of FAS

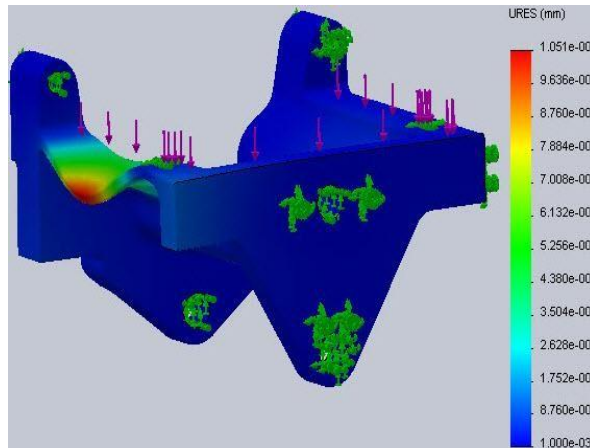


Figure-3 Deflection

Optimization Approach

The objective of optimization technique is to minimise the mass of the front axle support and reduces the cost of production. The front axle support is subjected to tensile load on the upper span of the bracket while using factor of safety of around 2.0 maximum stress of front axle support is monitored and it is not over the allowable stress. The load acting on the span of it is around 20KN. The optimization technique methodology flowchart is shown in fig 6.

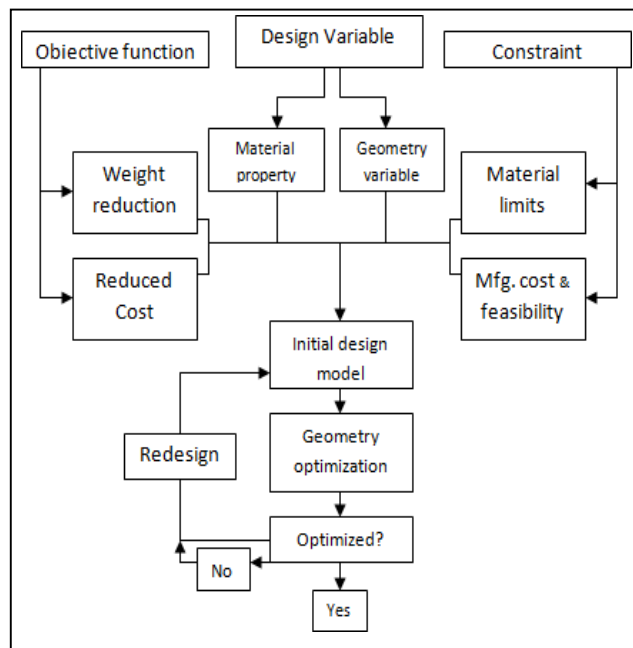


Figure-4 Optimization process chart

Optimization of Support of Front Axle

The optimization of the front axle support carried out using optimization technique. The optimization focused on the uncritical sections which need to be reduced. From the shape optimization theory it is suggest the unnecessary shape and design of front axle support. The result of shape optimization of front axle support is shown in fig 6. The main objective is to minimize the weight of the front axle support as well as the total production cost. The implementation of these optimizations is to find out the best design and shape of the front axle support to improve the performance and the strength especially at critical positions Figure 6 shows the positions where changes is necessary.

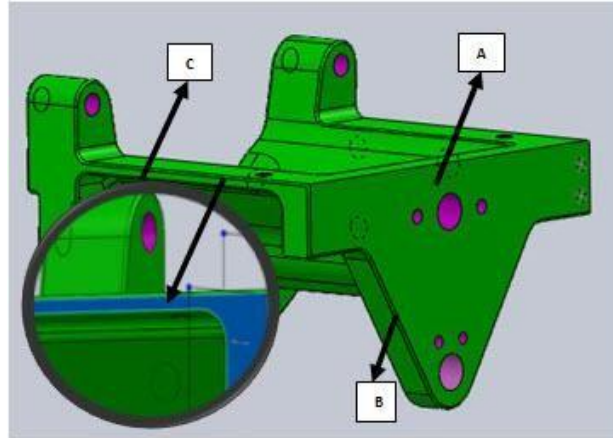


Figure-5

In the above figure A shows the area where there is no direct loading or any attachment is made, B shows the broader thickness which can reduce and C shows variable of thickness which can be optimized. After optimization following results were obtained as below.

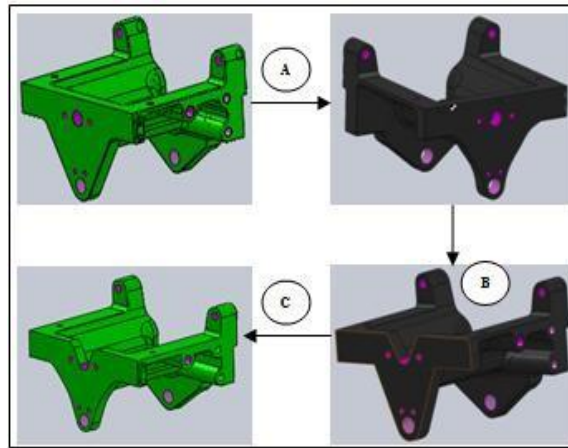


Figure-6

In above figure A-C shows the optimization of original shape to final optimized shape.

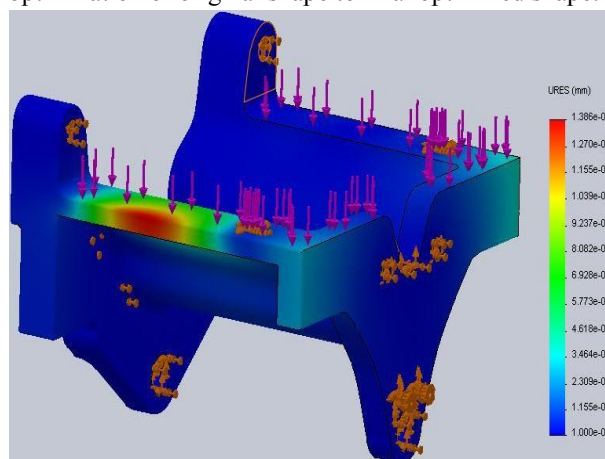


Figure-7 Deflection after analysis

Table 3 shows the comparison between initial and optimize designs on maximum principles stress and mass of front axle support. Optimize front axle support was choose as the best optimize design due to the lowest occurred stress and

mass. Eventhough the mass of optimized bracket is not the lowest, but the decision was also based on the maximum stress which is 196.39 MPa.

Table-3 Comparisons

S.NO.	DESIGN	MASS(Kg)	PERCENTAGE REDUCTION
1.	Original	31.28	-
2.	Optimized	28.04	10.35%

V. CONCLUSION

By the finite element analysis method and the assistance of solid works software, it is able to analyze for stress and strain. In this study we tried to simulate real condition by notice to all of effective force on front axle support. Shape optimization were analyzed to the front axle support and according to the results, it can be concluded that the weight of the optimized design is up to 10.35% lighter and has more strength than initial design of front axle support. Material optimization approach will be considered for future research.

REFERENCES

- [1] Ritesh Kumar Dewangan, Manas Patnaik, Narendra Yadav/ International Journal of Engineering Research and Application (IJERA) ISSN: 2248-9622 vol.2, Issue 4 July-August 2012, pp.457-460
- [2] Vinod Kumar Verma, Dinesh Redkar, Arun Mahaja, HTC 2012. Weight optimization of front axle support by optistruct technology application.
- [3] Rahman, M.M., Ariffin, A.K., Abdullah, S., Noor, M.M., Bakar, R.A. and Maleque, M.A. 2008b. Finite element based fatigue life prediction of cylinder head for two- stroke linear engine using stress-life approach. Journal of Applied Sciences, 8(19):3316-3327.
- [4] Rahman, M.M., Ariffin, A.K., Jamaludin, N. and Abdullah, S. 2007. Effect of nitrating treatment on fatigue life for free piston linear engine component using frequency response method: a finite element approach. Structural Durability and Health Monitoring, 3(4): 197-209.
- [5] Mirehei, A Zadeh, H.M., Jafari, A. and Omid M.2008. Fatigue analysis of connecting rod of universal tractor through finite element method (ANSYS).Journal of Agricultural Technology. 4(2): 21- 27.Rahman, M.M., Ariffin, A.K., Abdullah, S.,Noor, M.M.and Bakar, R.A. 2009b.