Proceeding of NCRIET-2015 & Indian J.Sci.Res. 12(1):089-092, 2015 ISSN: 0976-2876 (Print)
ISSN: 2250-0138 (Online)

OPTIMIZATION OF FRONT END LOADER OF CONSTRUCTION TRACTOR

BHEEMRAO M. KAMBLE[1]

Department of Mechanical Engineering, BKIT Bhalki, India

ABSTRACT

Excavator is a typical hydraulic heavy-duty human-operated machine used in general versatile construction operations, such as digging, ground leveling, carrying loads and dumping loads. Normally front end loader of the construction tractor works under the worst working conditions. Due to severe working conditions, front end loader parts are subjected to high loads and must work reliably under unpredictable working conditions. Thus, it is necessary for the designers to provide not only a equipment of maximum reliability but also of minimum weight and cost, keeping design safe under all loading conditions. The force analysis and strength analysis are important steps in the design of excavator parts. Finite element analysis is the most powerful technique used to evaluate strength of the structures working under high loads. This paper focuses on solid modeling of the front end loader of tractor and its F.E analysis for strength evaluation. It also includes the effect of the developed stresses in front end loader parts.

KEYWORDS: Loader, Force, Solid Modelling, Front End Loader

A front end loader of JCB tractor which is also known as loader, bucket loader, front loader, front end loader, scoop loader, is a type of tractor, that has a front-mounted square wide bucket connected to the end of two arms to scoop up loose material from the ground, such as dirt, sand and move it from one place to another. A loader is commonly used to move a stockpiled material from ground level and deposit it into an awaiting dump truck. Hydraulic system is used for operation of machine while digging or moving the material.

Several tractor manufacturers like JCB, Case Corporation, FIAT, Komatsu ltd, Volvo construction equipment etc, offers front end loader attachments to construction as well as agricultural tractors. JCB (3DX) loader is most preferred for excavation and earth moving due to its versatility [Juber Hussain Qureshi, 2012]. The Front end loader of the JCB is for loading with high mechanization. It can be efficient and convenient to operate in harsh environment and narrow roadways [Bhaveshkumar P Patel, 2011]. The front end loader is usually two degrees of freedom mechanism with a bucket which can be raised and lowered. The loader assembly may be a removable attachment or permanently mounted. Often the bucket can be replaced with other devices or tools. By studying force performances loader working mechanism the service life of the bucket is increased.

PROBLEM STATEMENT

Due to severe working conditions, front end loader parts are subjected to corrosive effects and high loads. The parts of the front end loader mechanism must work reliably under unpredictable working conditions [Bhaveshkumar P Patel, 2011]. Poor strength properties of the front end loader parts like bucket, arm, and links limit the life expectancy of the front end loader. The skilled operator also cannot know about the terrain conditions, soil parameters, and the soil-tool interaction forces exerted during operation are required to find because these forces helpful for better design of the tool, backhoe parts. High level of stresses can cause the damage of critical parts of front end loader and it will adversely affected on productivity of machine. The use of machines is increasing for the earth moving equipments. Thus it is very much necessary for the designer to provide not only a equipment of maximum reliability but also of minimum weight and cost, keeping design safe under all loading condition. Recent trends towards greater automation of excavation machines reflect a larger movement in the construction industry to improve efficiency. Currently, human operators require ten to fifteen years of experience before they can be considered experts. Static force analysis carried out by considering the maximum breakout force condition and static force analysis done for the different parts of the backhoe excavator and can be taken as boundary conditions for static FEA [Bhaveshkumar P Patel, 2011]. Now a day weight is major concern

[1]Corresponding author

while designing the machine components. So for reducing the overall cost as well as for smoothing the performance of machine, optimization is needed.

METHODOLOGY

- Literature review for study of design procedure will be carried out by referring reviewed journals, books, manuals and related documents.
- To study existing design, assembly of front end loader of tractor.
- Force calculation at the joints, stresses will be calculated analytically
- Geometrical model of front end loader of tractor will be modeled to meet the specifications using CATIA software.
- Geometrical model from CATIA will be meshed in ANSYS and force from analytical method will be applied and simulated for results.
- Comparison of simulation result with analytical calculation.

Perform similar structural analysis to support the design modifications.

FEA OF FRONT END LOADER OF JCB TRACTOR

The intension to perform static FEA is to know the stresses developed are within the safe stress limit of the material used for front end loader attachments [Li Xiaohuo, 2012]. Based on this analysis one can identify that the optimization of front end loader mechanism is possible or not.

While analyzing the whole front end loader attachment of JCB, the following FEA procedure is used,

Create the model: The different parts of the front end loader attachments like bucket, arm and links are created in the CATIA software.
Export the model: Model is exported through the IGES file and it should be simplify if required.
Specify Materials: Material properties define the structural characteristics of each part of the assembly for a simulation (FEA). Moreover ANSYS allows a user to change the required material properties of the parts in each simulation, conducting different results.
Meshing: In general, design analysis tools are demanding with respect to the geometric integrity. During the CAD modeling process these integrity requirements may not be met and may even go unnoticed. Examples of integrity errors include small gaps, overlaps or overhangs which can easily be overlooked from the CAD point of view. However they can be very troublesome for mesh. So before meshing the geometry cleanup is done i.e, parts that have very small features with respect to the overall model dimensions and do not play a substantial role in the simulation results can be avoided or suppressed.
Add constraints: structural constraints restrict or limit the displacement of the model. As for an example for static simulation of any part, must apply proper boundary conditions.

Add loads: Structure loads are forces applied to the part during operation. Such loads cause stresses, deformations and displacements in components. While designing the component it is important to know how the component reacts under normal and excessive working conditions.

Analysis of Bucket

The static analysis is utilized to carry out the Finite Element Analysis for various parts of the front end loader parts to know the developed stress level for known boundary and loading conditions.

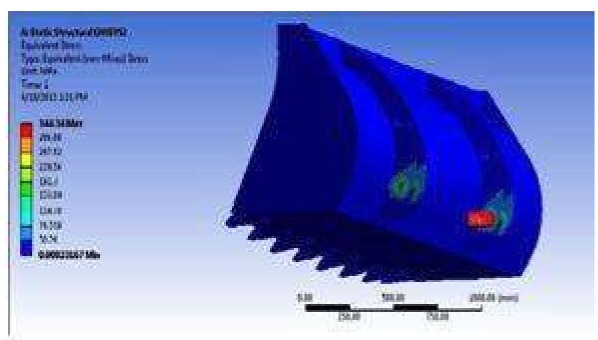

Figure 4.4: Von Misses stress

Figure 4.1 shows the CATIA model of the bucket, Figure 4.2 indicates mesh view of the bucket, Figure 4.3 shows the Boundary and loading conditions applied on the bucket, Figure 4.4 indicates the Von misses stresses developed in the bucket. The Von misses stresses acting on the bucket is 344.34MPa, moreover all parts of the bucket is made up of HARDOX 400 material [Bhaveshkumar P Patel, 2012].

Analysis of Arm and Link

Figure 6.1 shows the mesh view of arm of the front end loader. Figure 6.2 shows the boundary conditions applied to the arm. The maximum von misses stress acting on the arm 117.99Mpa. Yield strength of the HARDOX 400 material is 1000Mpa, Yields [σy] = 500Mpa.(safety factor =2) and σvm = 117.99Mpa, so σvm < σy and this indicates that the design of the arm is safe.

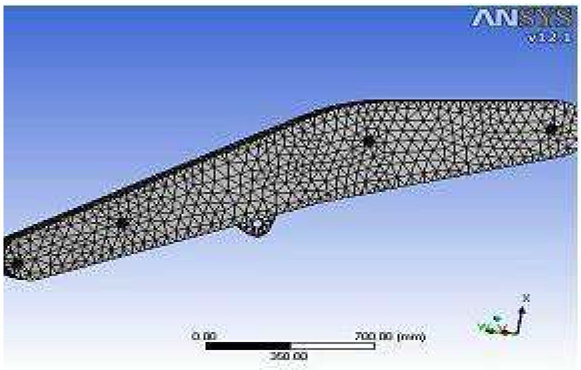

Figure 6.1: Mesh view of arm

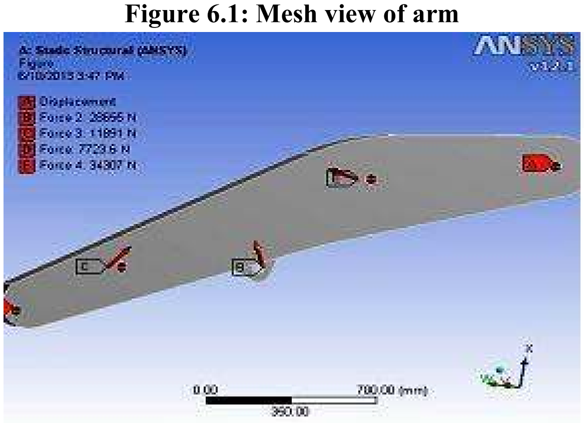

Figure 6.2: Boundary conditions

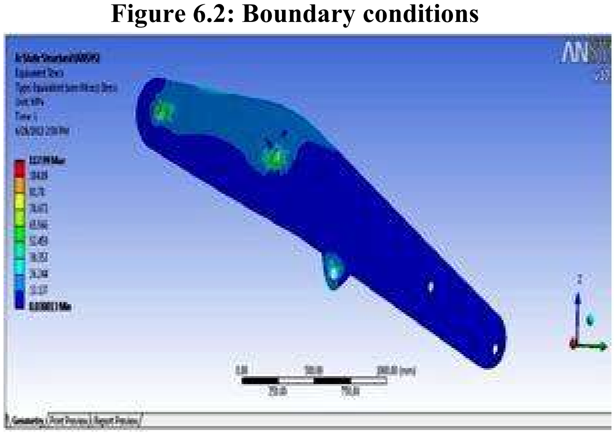

Figure 6.3: Analysis of arm

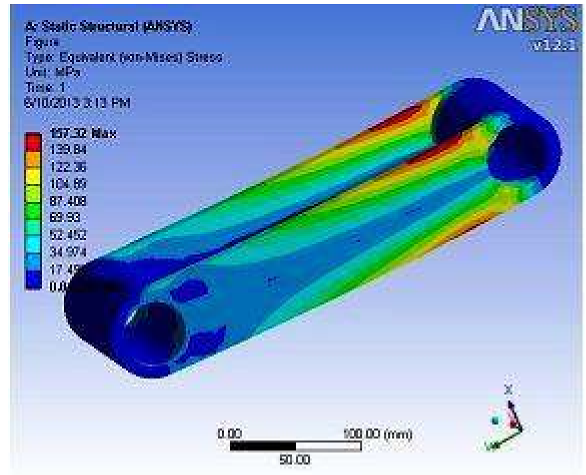

Figure 6.4: link analysis

Figure 6.4 shows the link analysis of the front end loader. The maximum Von misses stress is 157.32Mpa which is less than the yield stress. Hence the design is safe.

REFERENCES

Li Xiaohuo, Chi Qingdi, Mu Yongcheng, Yao Tianyu, "Finite element analysis of working mechanism for full hydraulic backhoe loader", Mechanical engineering college, Liaoning Technology University, 2nd International conference on Electronic and Mechanical Engineering and Information Technology-2012.

Erfan G. Nezami, Youssef M.A Hashash and Jamshid Ghaboussi,"Simulation of front end loader bucket-soil interaction using discrete element method" International journal for Numerical and Analytical methods in Geomechanics, published online 3 january 2007.

N.V.Dhandapani , Dr. G Mohan kumar , Dr K.K.Debnath a Research scholar, Park college of Engg & Tech, Under Anna university, Coimbatore, - 641659, India. "Static analysis of off-high way vehicle chassis support structure for the effect of various stress distributions"

Bhaveshkumar P Patel and Prajapati J M, "A Review on FEA and Optimization of Backhoe Attachment in Hydraulic Excavator", IACSIT International Journal of Engineering and Technology, Vol. 3, No. 5, pp. 505-511, Oct 2011.

Juber Hussain Qureshi ,Manish Sagar, “The Finite Element Analysis of Backhoe loader” , International Journal of Engineering Research and Applications, Vol .2,Issue 3,May-Jun 2012, pp.1484-1487,May-June 2012.

Bhaveshkumar P Patel and Prajapati J M, “A Review on Kinematics of Hydraulic Excavator’s Backhoe Attachment” , International Journal of Engineering science and Technology, Vol.3,No.3, March 2011.

Bhaveshkumar P Patel and Prajapati J M, “Static Analysis of Mini Hydraulic Backhoe Excavator Attachment Using FEA Approach”, International Journal of Mechanical Engineering and Robotics Research, Vol.1,No.3, October 2012.