Finite Element Analysis of Excavator Arm

Dhruv N. Patel[1]
[1]M.E Student (CAD/CAM)
[1]Mechanical Engineering Department
[1]A. D. Patel Institute of Technology, New V.V.Nagar, Gujarat, India

Abstract— Excavators are heavy duty earthmoving machines and normally used for excavation task, mechanical analysis was carried out in three typical work condition of the working device by using the mechanical theory and method. The static strength finite element analysis of excavator arm was carried through by using ANSYS, from which, the stress and strain deformation contour diagrams of three typical work condition were obtained so that the hardness can be checked The results of finite element analysis showed that the static intensity of the boom is enough. The maximum stress mainly occurred in the hinge point connected the boom cylinder with the boom and the hinge point connected the boom with the base which played an important part in controlling the strength of the excavator working device The study results are of certain guiding significance for working device's optimization design.

Key words: Excavator, working device, Arm, FEA element analysis, Pro/E, ANSYS, Workbench

I. INTRODUCTION

In the area of globalization and tough competition the use of machines is increasing for the earth moving works, considerable attention has been focused on designing of the earth moving equipments [1].

Today hydraulic excavators are widely used in construction, mining, excavation, and forestry applications [2*]*. Hydraulic excavators also called diggers. There are many variations in hydraulic excavators. They may be either crawler or rubber-tire-carrier-mounted, and there are many different operating attachments. With the options in types, attachments, and sizes of machines, there are differences in appropriate applications and therefore variations in economical advantages [3]

Excavator digs, elevates, swings and dumps material by the action of its mechanism, which consists of boom, arm, bucket and hydraulic cylinders. Bucket is used for trenching, in the placement of pipe and other under-ground utilities, digging basements or water retention ponds, maintaining slopes and mass excavation. Due to severe working conditions, excavator parts are subject to corrosive effects and high load

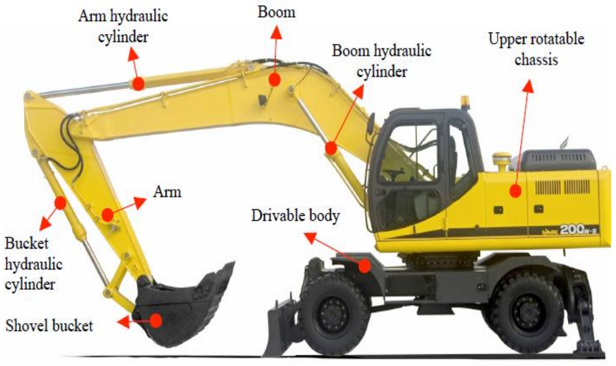

Fig. 1: Excavator

II. DESCRIPTION OF FINITE ELEMENT ANALYSIS

Finite element analysis is a word used in recent language of Mechanical Engineer in which the problems are solved with the use of software using Finite element method to solve the difficult problems where the direct implication of the problem is not always ready in the formula's provided in the standards or the books. The finite element method is a numerical procedure which can be used to solve numerical problems in Engineering. An unsophisticated description of the FE method is that it involves cutting a structure into several elements (pieces of the structure), describing the behaviour of each element in a simple way, than reconnecting elements at "nodes" as if nodes were pins or drops of glue that holds together the elements This process results in the set of simultaneous algebraic equations which can be solved with mathematical formulations for solving the problem for deflection which can be further solved to get the results of strains and stresses. There may be several and thousands of such equations formed which means the computer implication is mandatory.

A. *Working of FEA*

FEA uses a complex system of points called nodes which make a grid called a mesh **Figure 2** This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements.

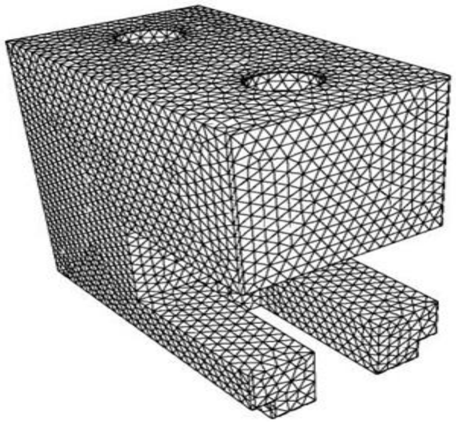

Fig. 2: Tetrahedral Mesh

III. Analysis Of The Arm

Figure 3 shows the static analysis of arm of the backhoe excavator. Figure 5 shows the boundary and loading conditions applied to the arm. Figure 4 shows the mesh view of the arm with 44126 nodes and 8670 elements

Figure 6 shows the results of the Von Misses stresses on arm assembly. As it can be seen from the Figure 6 that the maximum Von Misses stress is acting at the arm cylinder mounting lug and it is 239.39 MPa. Figure 7. shows enlarged view of the arm at which the maximum stress is produced

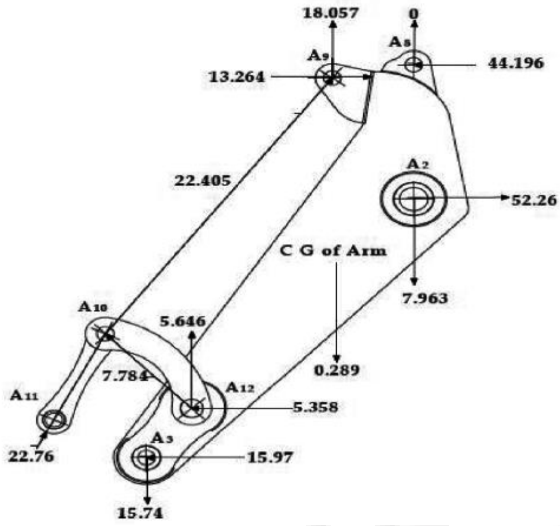

Fig. 3: Static Force Analysis for Arm

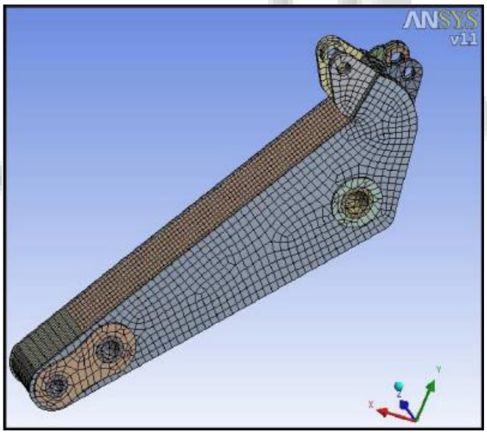

Fig. 4: Mesh View of Arm

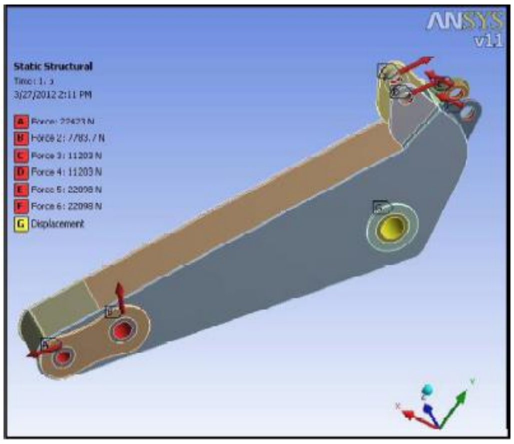

Fig. 5: Boundary Conditions for Arm

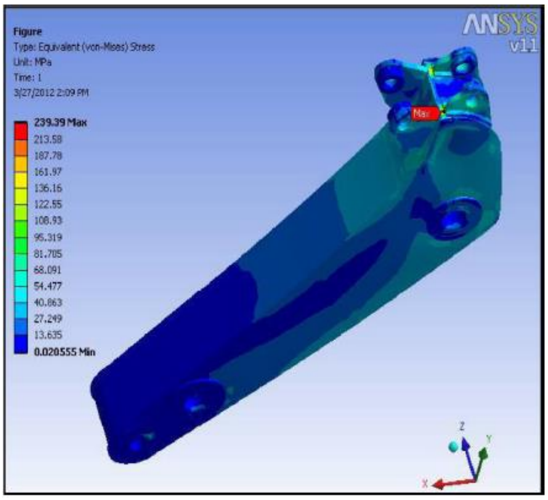

Fig. 6: Von Misses Stresses of Arm

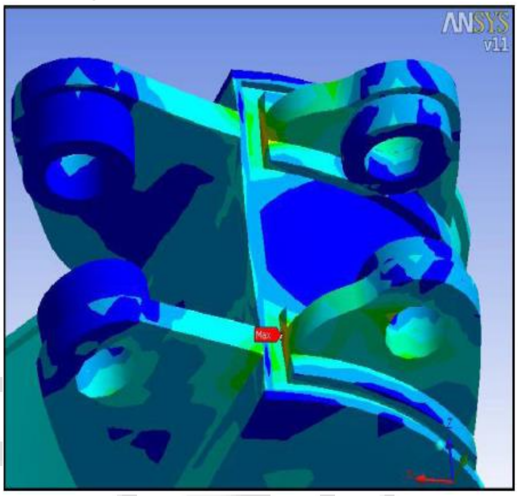

Fig. 7: Enlarged View of Maximum Stresses of Arm

made up of HARDOX 400) is 1000 MPa. Equation (1) yields [σy] = 500 MPa (safety factor = 2), and σVM = 239.39 MPa (Figure 10), so σVM < [σy] and this indicates that the design of the arm is safe. Figure 12 shows the maximum displacement of 0.35945 mm occurs at the free end of the arm (A3 point) which is far less than the minimum plate thickness used in the arm therefore the design of the arm is safe for static condition.

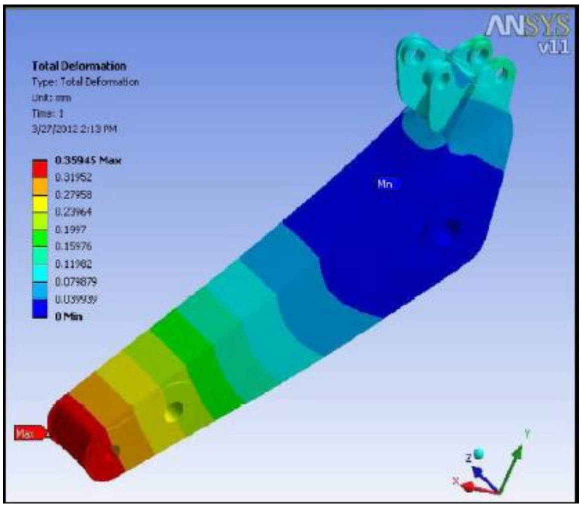

Fig. 8: Displacement of Arm

IV. RESULT

With the results obtained from the Finite Element Analyse by Pro-e Mechanica software is are very safe as per the stress values and the deflections

V. CONCLUSION

From the analysis results, it is proved that the design is safe for the calculated digging force, from the result of running of finite element analysis on an arm, it is seen that there is localized stress concentration in zones where there is discontinuity of material or sudden change in the direction of stress flow. This may arise either due to limitation of meshing logic of the analysis software or due to actual stress concentration in that zone

The conclusion of analysis is listed below

(1) The maximum deformation in arm is 0.35945mm.
(2) The maximum equivalent stress in arm is 239.39 this value is below then ultimate tensile strength so the maximum equivalent stress are within limit.
(3) Factor of safety varies from 2.5 to 12. The arm is overdesigned. There is a future scope in optimization of design of arm.

REFERENCES

[1] A textbook of Theory of Mechanisms and Machines by Amitabha Ghosh
[2] Bhaveshkumar P Patel and Prajapati J M(2011a), “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.
[3] Mehta Gaurav K (2008), “Design and Development of an Excavator Attachment”, p. 1,M.Tech Dissertation Thesis, Nirma University, Institute of Technology, Ahmedabad
[4] Hsin-Sheng Lee, Shinn-Liang Chang and Kuo-Huang Lin, “A study of the design, manufacture and robot control of a pneumatic excavator”, International Journal of Mechanical Engineering Education, 2002pp 345-361
[5] Donald Margolis, Taehyun Shim “Instability Due to Interacting Hydraulic and Mechanical dynamics in Backhoes” Journal of Dynamic Systems Measurement and Control ASME, Vol. 125, 2003, pp. 497-504