ISSN : 2455-9679
Impact Factor : 1.389

IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

"REVIEW ON FEA AND OPTIMIZATION OF BACKHOE LOADER IN HYDRAULIC EXCAVATOR "

Swapnil Singh[1] , Lijo verghase[2]

[1]*M. Tech Scholar, Mechanical Engineering, Oriental Institute of Science & Technology Bhopal,(M.P.), India*

[2]*Assistant Professor, Lijo Verghase, Mechanical Engineering Dept., Oriental Institute of Science & Technology, Bhopal,(M.P.), India*

ABSTRACT

Excavators are used primarily to excavate below the natural surface of the ground on which the machine rests and load it into trucks or tractor. Due to severe working conditions, excavator parts are subjected to high loads. The excavator mechanism must work reliably under unpredictable working conditions. Thus it is very much 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. It can be concluded that, force analysis and strength analysis is an important step in the design of excavator parts. Finite Element Analysis (FEA) is the most powerful technique in strength calculations of the structures working under known load and boundary conditions USING CAD software. In general, computer aided drawing model of the parts to be analyzed must be prepared prior to the FEA. It is also possible to reduce the weight of the mechanism by performing optimization task in FEA. This paper provides the platform to understand the Modeling, FEA and optimization of backhoe excavator attachment, which was already carried out by other researchers for their related applications and it can be helpful for development of new excavator attachment.

KEYWORDS : *Backhoe, Excavator, FEA, Optimization, CAD.*

I. INTRODUCTION

In the era 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 loads.

II. PROBLEM FORMULATION

Due to severe working conditions, excavator parts are subject to corrosive effects and high loads [3]. The excavator failure problems of backhoe components and getting optimized design without compromise with its strength and performance during digging operation. mechanism must work reliably under unpredictable working conditions. Poor strength properties of the excavator parts like boom, arm and bucket limit the life expectancy of the excavator. Therefore, excavator parts must be strong enough to cope with caustic working conditions of the excavator [4]. The skilled operator also cannot know about the terrain condition, soil parameters, and the soil-tool interaction forces exerted during excavation operation are required to find because these forces helpful for better design of the tool, backhoe parts and for trajectory planning [2]. Normally the exactor is working under cyclic motion during excavation process. Due to this repetitive nature of work, cyclic stresses are developed in the parts of backhoe attachment. High level of stresses can cause the damage of critical parts of excavators and it will adversely affected on productivity of machine. Now a day weight is major concern while designing the machine components. So for reducing the overall cost as well as for smoothing the performance of machine, optimization is needed.

III. UTILITY OF FEA AND OPTIMIZATION FOR BACKHOE ATTACHMENT

The strength analysis is an important step in the design of excavator parts. Finite Element Analysis (FEA) is the most powerful technique in strength calculations of the structures working under known load and boundary conditions. One can determine the critical loading conditions of the excavator by performing static force analysis of the mechanism involved for different piston displacements. The boundary conditions for strength analysis will be determined according to the results of static force analysis. In general, computer aided drawing (CAD) model of the parts to be analyzed must be prepared prior to the FEA. Preparation of the CAD model can be done either using a commercial FEA program or using a separate commercial program, which is specialized for CAD. Structural optimization for strength is a popular subject in modern engineering design. It has been widely used to obtain an optimum strength/material mass ratio for structures under specified load conditions [4]. The FEA and optimization is versatile tool for designing the backhoe attachment in hydraulic excavator. The next section includes the research work carried out by other researchers in the same field. Structural model of micro- excavator digging arm shown in figure.1

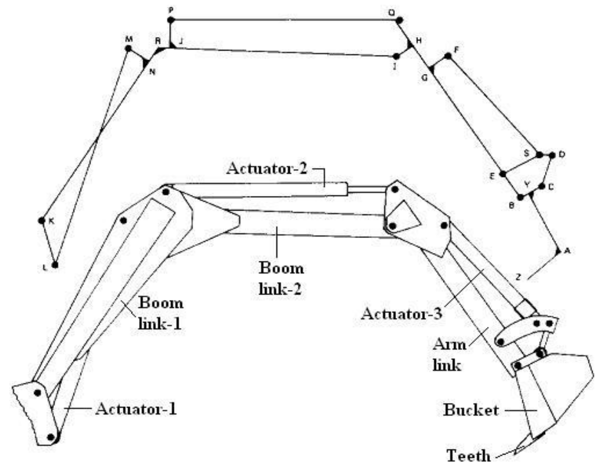


Fig. 1 Structural model of Powerful 360WT micro- excavator digging arm

Finite element analysis is an important part of the overall design process, serving to verify or validate a design prior to its manufacture. Because finite element analysis is a simulation tool, the actual design is idealized, with the quality of the idealization dependent on the skill and experience of the analyst. Naresh N. Oza, had carried out the FEA and optimization of Earth moving attachment as backhoe in 2006. Accuracy of results is dependent on choice of elements, number of nodes, selection of proper material, boundary conditions, applied loads and expertise of the analyst. Principal steps od Finite Element Analysis are Creation of geometry and its cleanup, specify material and element properties, meshing of geometry in into nodes and elements, apply the loads and boundary conditions, and finally carry out the solution and post processing results. After getting the results, interprets the results and do required corrective steps on it to fulfill the requirement of the problem. They have done the EFA of the boom, arm and bucket by following the standard practice of analysis and carry out the solution for stress and deflection analysis, finally the results are compared with the results obtained from the MathCAD. Optimization for weight is also carried by them and reduces the weight of arm from 180 Kg to 154 Kg and stresses reduced from 386 MPa to 263 MPa. The weight of the bucket is reduced from 165 Kg to 156 Kg, and the developed stresses are within the limit [5]. The computational modeling techniques and computer programs developed for the structural design analysis of a micro excavator digging arm mechanism under static or quasi-static loading conditions are outlined by MA Bromfield and WT Evans in 1988. The computer programs allow the design engineer to analyze the forces and stresses at numerous locations on the digging arm, which can assume various

ISSN : 2455-9679
Impact Factor : 1.389

geometric configurations. The computer theory was used to develop an integrated CAD software package to allow the design engineer to carry out structural analysis and design optimization calculations on ADAMS simulation and the experiment are also compared. The result of the flexible multibody dynamics analysis showed that the operating frequencies were about 4 ~ 5Hz. For the good result of the flexible multibody dynamics analysis using the ADAMS, it is necessary to set up the step size up small. The calculated results were also compared to the physical experiment. The calculated results were in a good agreement to the experimental results and were acceptable for the design purpose [8]. Yefei Li, Xianghong Xu and Qinying Qiu, in 2006, have presented an application of Grid-enabled computing technologies in the field of engineering design using Finite Element Method (FEM). A Grid-enabled analysis environment with self-developed codes provides easy access to computational and database capabilities to enhance the engineering system based on FEM results. The aim is to obtain better lighter and cheaper designs by using Finite Element Method based computational analysis models with easy-to-use grid environment. Data structure provides a high level abstraction to heterogeneous data involved in a FEM-based analysis process. Various search algorithms exposed in this, the Powerful 360WT micro excavator which is shown in figure 1. Product development times and costs have been reduced as a result of using the CAD software. The results showed good correlation between theoretical and experimental stresses, considering the many simplifications that were made in the modelling technique. But to integrate the hydraulic system and structure for optimum performance was not taken for study. The computer modelling technique has been developed for static or quasi-static loading situations but it could be extended to the dynamic situation by introducing mass, inertia, and acceleration terms [6]. Ram Vadhe and Vrajesh Dave, in 1993 have developed a multi-body model of an excavator and to simulate the prototype testing conditions. Multi-body simulation involves the simulation of rigid body system under the application of cylinder forces and/or motions. The link to be designed is considered as a flexible body. Two cycles of digging and dumping operations are simulated to determine the reaction forces generated at each joint and stresses generated on the flexible body. The boom was modeled as a flexible body using the NASTRAN program and the joint reaction forces of a rigid model and a flexible model are compared. A finite element model of the boom, shown in Figure 2, consists of 4,880 nodes and 4,101 elements (Hex element 1128, Wedge element 364, Quad element 2601, Tri element 8). External nodes are selected at the points at which kinematic joints are located.

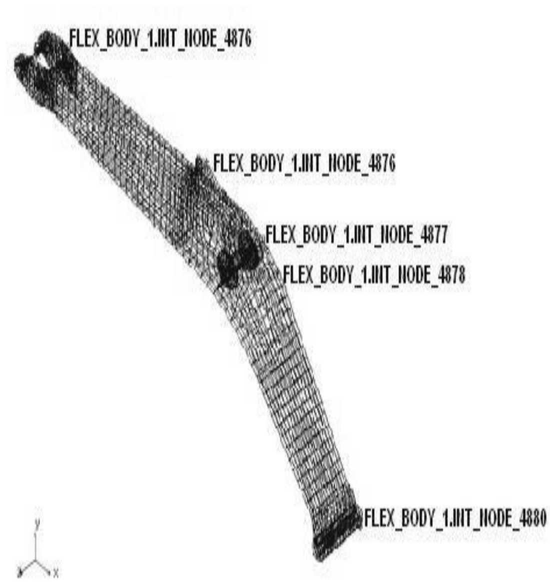


Fig. 2 The Finite Element model of the boom

Luigi Solazzi, in 2010, have carried out study on the boom and the arm of an excavator in order to replace the material, which they are usually made of, with another material. In particular, the study wants to substitute the steel alloy for an aluminium alloy. This change lightens the components of the arm, allows to increase the load capacity of the bucket and so it is possible to increase the excavator productivity per hour. Figure 5 shows the stress state. It has been assumed that the material used to make the arm is the steel alloy S355 JO EN 10025.Finite element meshing shown in figure 3.

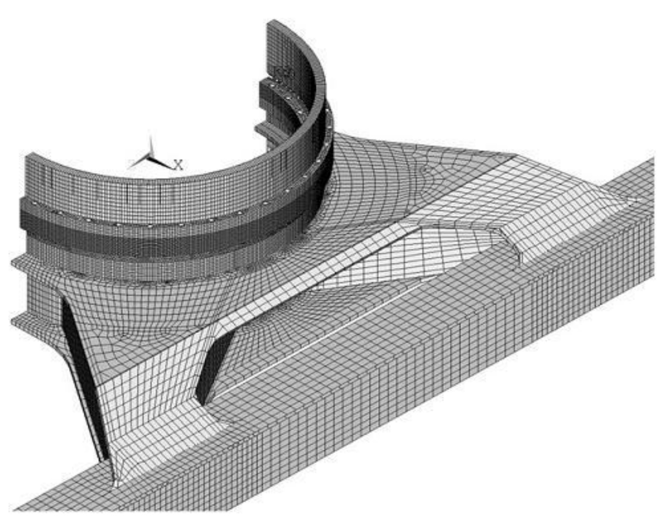


Fig. 3 Digital model — finite element grid (1/2 of the model)

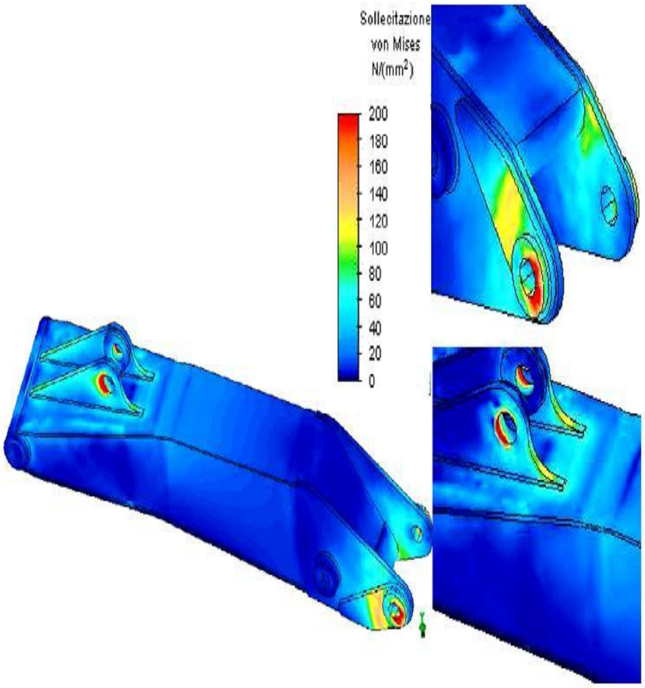


Fig. 4 Stress state (Von Mises) in the first element

The evaluation of the new geometry of the arm with the different material has been studied in order to obtain at least the same safety factor and deformability of the original geometry (steel alloy) and for the new geometry (aluminium alloy). On the basis of the relationships state above between the geometry of the steel alloy panel and the geometry of the aluminium alloy panel, for each component of the arm has been developed a new cross-section. With this cross-sections it has been numerically modeled the whole element of the arm. The consequent step is to perform the FEM analyses in order to verify both the safety factor and the flexibility of the component as regards to the original value. This last operation has been repeated iteratively until the goal has been achieved. It is shown in figure.4

Based on this study they achieve that the total weight of the arm is reduced of about 50% and the capacity of the bucket increased of about 30%.

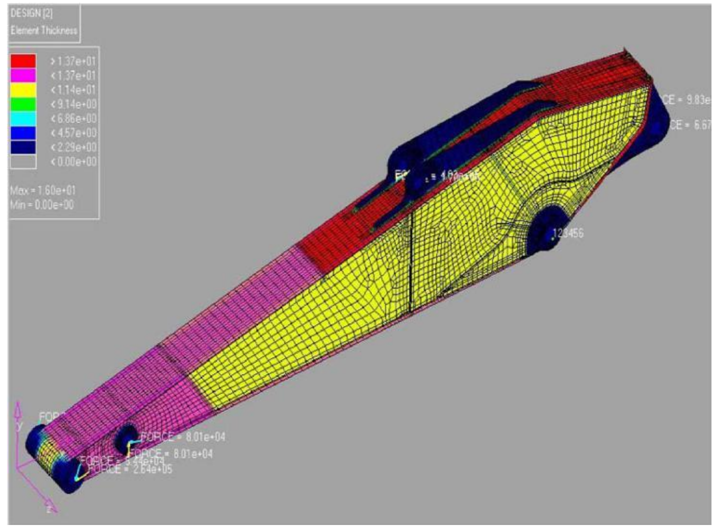
IV. PROBLEM FORMULATION

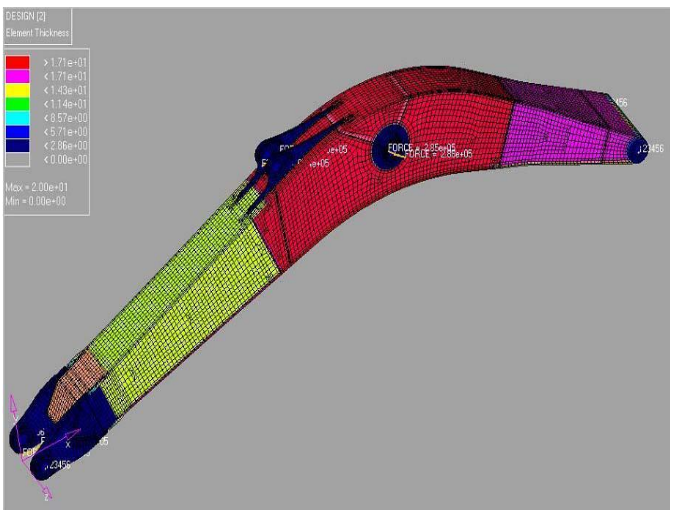
J. Mottl has described ‘Voting Method’ for optimization of the weight of an excavator in 1992. He has carried out optimization for all parts of the excavator such as the chassis, cabin, jib, etc. with consideration as non-linear programming problem [12]. Mehmet Yener, in 2005, Parameterization of boom geometry is done to add some flexibility to interface called OPTIBOOM. Optimization of boom carried out for HMK 220LC model excavator. It is shown in figure.5, figure 6 and figure 7.

In general there are 3 types of structural optimization techniques: sizing, geometrical and topology optimization. Out of these three techniques, topology optimization may give better results by changing the initial topology. Starting from an initial design, more than 100 alternative designs were created and compared with each other in terms of boom mass and maximum von Mises stresses. 21 shape parameters have been changed to obtain new boom geometries and the best design has been found at 90th design alternative. Maximum von Mises stress value at the 90th boom is 146 MPa while it is 186 MPa in the initial boom shape. Maximum von Mises stress has been reduced by 21.5 %. Mass of the 90th model is 1454 Kg and mass of the initial boom is 1403 kg. thus, mass of the boom increases only by 3.6 % [4]. Yefei Li, Xianghong Xu and Qinying Qiu, in 2006, have described a Grid-enabled analysis with self-developed codes provides easy access to computational and database capabilities to enhance the engineering system based on FEM results. The aim is to obtain better lighter and cheaper designs by using Finite Element Method based computational analysis models with easy-to-use grid environment. Data structure provides a high level abstraction to heterogeneous data involved in a FEM-based analysis process. Various search algorithms exposed in this environment enable complex search strategies to be constructed within the environment. They have explained a strategy of combining a design of expert system platform and at genetic algorithm (GA) with gradient descent search is used on design and checking of the hydraulic excavator. Optimization is carried out using genetic algorithm followed by a local search on the best point found using the Lagrange interval search method from self-developed codes. The results then used to build CAD model for validating the stress and displace distribution of the whole parts of working equipment using the ANSYS model. It shows that the maximum stress of the working equipment is still under safe state, and results of optimization is as shown in figure 7, thus the structure design object has to been made compromise. As we know, a typical Finite Element Analysis (FEA) problem is composed by three essential steps: Modeling, mesh generation and solution, and post-process. A parametric geometry model suitable for FEM analysis is first generated using CAD software Pro/Engineer. The subsequent mesh is generated with minimum reference to the geometry information, as only the top-level entities in the CAD model are referenced in the meshing. There are two

ISSN : 2455-9679
Impact Factor : 1.389

typically mesh tools used, either direct access provided to CAD parts and assemblies in their native mode, or third-party standard exchange data formats used. They have utilized interface between them which produce ANF file format as exchange geometry. Generation is processed in ANSYS batch mode. Volume mesh is then generated by applying parameters on adaptive meshing. The structure FEA solver ANSYS is then used to solve the problem, the model used in this work is an assembly static analysis. A three dimensional working equipment optimization problem is presented here. The problem is both complex and has a high computational cost. The goal is to reduce the weight and manufacture cost of hydraulic excavator, FEM is adopted here to analysis the whole structure's stress and displacement distribution [9].





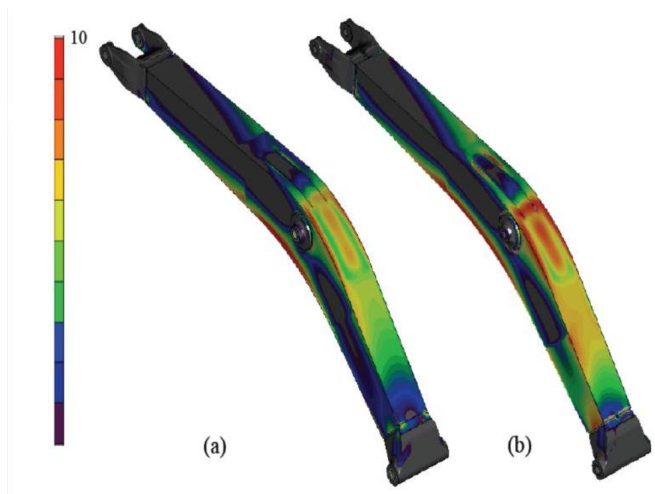


Fig 7 Isometric view of the boom models representing stress distribution

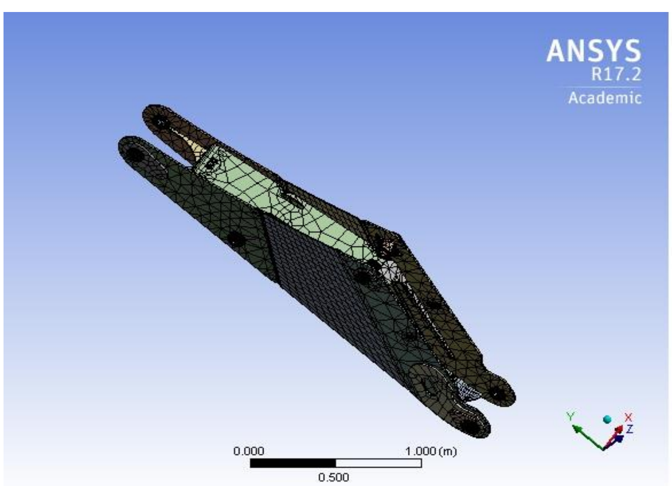


Fig. 5 Size Optimization of Boom

V. CONCLUSION

The FEA and optimization is versatile tool for designing the backhoe attachment in hydraulic excavator. To carry out the modeling and FE analysis of an excavator, various software used by researchers like PRO-ENGINEER, ADAMS, NASTRAN, CATIA, ANSYS, Hypermesh, Abaqus, I-deas etc. according to their ease of user friendliness and accuracy of results. By conducting FEA it is very easy to identify weak components through strength analysis of excavator attachment and corrections are possible in early stage of design. OPTIBOOM software developed by Mehmet Yener in 2005 and optimization of boom carried out.

REFERENCES

1. Mehta Gaurav K, "Design and Development of an Excavator Attachment" M. tech Dissertation Thesis, Nirma University, Institute of Technology, Ahmedabad, May 2008, pp. 1.

2. Bhaveshkumar P. Patel, Dr. J. M. Prajapati, "Soil-Tool Interaction as a Review for Digging Operation of Mini Hydraulic Excavator", International Journal of Engineering Science and Technology, Vol. 3 No. 2, February 2011, 894-901.

3. Bhargav J Gadhvi, B. P. Patel and P. M. Patel, "Development of a Controller for Mini Hydraulic Excavator as a Review", Proceeding of National Conference on Recent Advances in Manufacturing, SVNIT, Surat, 19th - 21st July, 2010, pp. 198-203.

4. Mehmet Yener, "Design of a Computer Interface for Automatic Finite Element Analysis of an Excavator Boom", M.S. Thesis, The Graduate School of natural and Applied Sciences of Middle East Technical University, May 2005, pp. 1-4, 68-69.

5. Nareshkumar N. Oza, "Finite Element Analysis and Optimization of an Earthmoving Equipment Attachment – Backhoe", M. tech Dissertation Thesis, Nirma University, Institute of Technology, Ahmedabad, May 2006, pp. 31-68.

6. MA Bromfield and WT Evans, "Computer modelling of microexcavator", Computer Aided Design, Butterworth & Co (Publishers) Ltd., Vol. 20, No. 9, November 1988, pp. 549-554.

7. Ram Vadhe and Vrajesh Dave, " Multi-Body Simulation of Earthmoving Equipment using Motionview/Motionsolve", Driving Innovations with Enterprise Simulation, L & T e-engineering Solutions, 1993, pp. 1-5.

8. Jun-Yong Park, Heui-Won Kim and Wan-Suk Yoo, "Matching of Flexible Multibody Dynamic Simulation and Experiment of a Hydraulic Excavator", Proceedings of ACMD, 2004, pp. 459-463.

9. Yefei Li, Xianghong Xu and Qinying Qiu, "FEM-Based Structure Optimization with Grid-Enabled Analysis Environment"

10. Proceedings of the 6th World Congress on Intelligent Control and Automation, IEEE, Dalian, China June 21 - 23, 2006, pp. 6915-6919.

11. Tadeusz Smolnicki, Damian Derlukiewicz, and Mariusz Stańco, "Evaluation of load distribution in the superstructure rotation joint of single-bucket caterpillar excavators", Automation in Construction, Elsevier, 2008, pp. 218-223.

12. Luigi Solazzi, "Design of aluminium boom and arm for an excavator", Journal of Terramechanics, Elsevier, Vol. 47, 2010, pp. 201–207.

13. J. Mottl, "Excavator optimization using the voting method", Computer Methods in Applied Mechanics and Engineering, No. 98, North-Holland, 1992, pp. 227-250.

14. Bipin N. Patel, "Finite Element Analysis and Optimization of Arm and Boom of Excavator", M. Tech. Thesis, Institute of Technology, Nirma University of Science and Technology, Ahmedabad, May 2007, pp. 40-72.