

Optimization of Excavator Boom with Ribs by FEA Analysis

Mr. Nilesh Panchal¹ *P.G. Student, Mechanical engineering dept. DYPIET Pune,*

Dr. P.M. Ghanegaoker², *Prof. and Dean Mechanical engineering dept. DYPIET Pune.*

Abstract - Excavators are considered as earthmoving equipments and generally used for excavation process. During the process unknown large amount of resistive forces offered by the terrain, stones to the bucket teeth adversely affect on the excavator parts and may fail during excavation process. Design engineer has a challenge for robust design of earth moving equipments which can work against unpredicted forces and under critical working condition. Thus, designers should provide not only a better design of parts that has maximum reliability but also minimum weight and cost, keeping design safe under all operating conditions. In this paper, Finite Element Analysis (FEA) tool is used for analysis and optimization of excavator boom. A modified design is proposed having rib structure with optimization. A 3D model of a boom is drawn by reverse engineering of existing structure in CATIA V5R19, Hypermesh 12.0 is used for meshing, and Ansys R15.0 will be used for solutions.

Keywords – FEA, Boom, Rib structure, Optimization.

I. INTRODUCTION

A bucket type excavator consists of an upper rotatable chassis and a hydraulically mounted mechanism consisting of bucket, boom, and arm which is as shown in fig. 1. Excavator is used as a utility machine for general excavation process, loading of hoppers and trucks, the cleaning of canals and ditches, solid waste management, demolition and mining work. Excavator goes through various motions like digging, pushing or pulling soil, lifting, swinging to carryout excavation task. Human operator controls this motion of the machine manually [1].

As use of this earth moving machine is increasing, more and more attention is provided for increasing its strength and making robust design with reducing cost. Generally, excavator is used to excavate soil by applying the force on the surface of earth using hydraulic system. Force is transmitted from ground then bucket to different parts of excavator.

F. Nilesh Panchal is P.G. student of the Dr. D. Y. Patil Inst. of engg. & tech. Pune. (E-mail: nilesh.ap1988@gmail.com).

S Dr. P.M. Ghanegaoker is Professor and Dean of Mechanical engg. Dept. at DYPIET Pune. (E-mail: pmghanegaonkar@yahoo.com)

Excavator comes in various sizes depending on size of bucket, boom length, arm length and operational working speed. The material considered for the excavator boom is Hardox400. Hardox is also known as high strength steel with a great combination of hardness and toughness. Hardox has the yield strength of 1000 MPa [3].

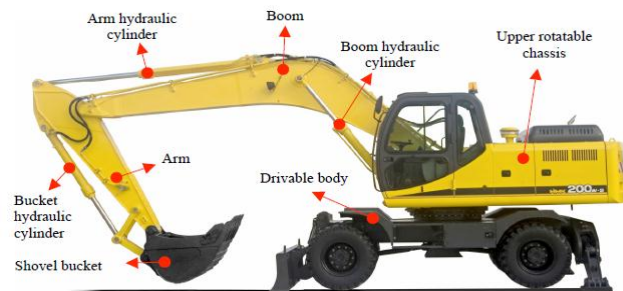


Figure 1: General view of an excavator

II. LITERATURE SURVEY

Amol B. Bhosale, et. all [1], analyzed existing boom structure of an excavator. Forces were calculated and found that maximum von mises stress is below than its yield stress. Thickness of the structure was reduced for optimizing it. Results gave 36.4% reduction in weight.

Janmit Rajet. all [2], highlighted the most relevant work done by researchers in the area of excavator boom. Static structural analysis, fatigue analysis, modal analysis, shape optimization of excavator boom were topic of interest of the authors.

S.SekharBabu, et. all [3], performed structural analysis on bucket of an excavator. The main aim of the author was to design excavator bucket by certain parameters and to improve its life. It was found that ribs have to be changed periodically to protect the bucket.

R M Dhawale, et. all [4], studied various research works in the area of excavator and its attachment. Calculations of forces are done which are further taken for analysis purpose.

This paper is important for researchers who are working in the field of finite element analysis of excavator.

Aimin Ji, et. al [5], a method of integrated design of boom and analysis is put forward by considering all the problems in development of actual product and working process of boom. Integrated design and analysis includes parametric design of excavator boom along with database design, mid surface extraction, static structural analysis. Simplification in structure is done based on parametric calculations.

Bhaveskumar P Patel, et. al [6], has done the analysis on all the excavator components like bucket, boom, arm, swing link. Stresses and deformation were found out. With this, analysis of excavator taking welding consideration is done in which strength of the material for the welding should be less than the strength of the base metal.

Bhaveskumar P Patel, et. al [7], provides a platform for analyzing and optimizing backhoe excavator attachment. In this, boom was modeled as a flexible body by using NASTRAN and the joint reaction forces of a rigid model and a flexible model are compared.

Gaurav K Mehta, et. al [8], found out force acting on each part of excavator like bucket, boom, arm by static force analysis method by considering different operating conditions. It was found that critical condition for mechanism is where the maximum digging force acts. Each component was considered as free body for carrying force acting on it.

III. PROBLEM STATEMENT

Excavators are subjected to high corrosive effects and loads. The excavator mechanism must even work under unpredictable operating conditions. Poor strength properties of the excavator parts like boom, arm and bucket limit the life of the excavator. Therefore, excavator parts should be robust enough to cope with caustic operating conditions of the excavator. The skilled operator is unaware of condition of road, soil parameter and sand force transmitted from soil during excavation process. These forces should be considered for better design of tools, other parts of excavators, and for planning trajectory motion. Excavator has a cyclic motion during excavation. Because of this repetitive nature of work, cyclic stress is developed in the parts of excavator. In today's world, weight is one of the major concern while planning and designing any machine parts. So for reducing the overall price further as for smoothing the performance of machine, optimization is required. Analysis for weight reduction of boom to save material cost and fuel economy throughout the excavation operation under safe loading condition is needed [4, 6, 7].

IV. OBJECTIVES

This project, which will become alternate design for boom, comprise of rib structure to withstand high stresses developed during unpredictable working condition.

1. Structural analysis which determines the area under high stress using FEA.
2. Reduction in material which will reduce weight of structure by optimization and ultimately reduces basic and operating cost.

V. METHODOLOGY

Steps followed for analysis of complete assembly of excavator are as follows

A. Analysis of existing model

1. Reverse engineering of existing model.
2. Generation of 3D model in CATIA V5R19 software.
3. Meshing of assembly, applying connections etc in Hypermesh 12.0
4. Analysis of the assembly is done on Ansys R15.0 to predict the stresses area developed on surface of boom of excavator which is then considered for optimization.

B. Analysis of optimized model

1. Optimization is done in form of iterations by reducing thickness of plates and by adding ribs structure in boom for each iteration and CAD model is done.
2. Meshing and Analysis is done for each iteration of optimized model to check maximum scope of weight reduction.

C. Fabrication, testing

D. Validation (software and experimental)

VI. ANALYSIS OF EXISTING MODEL

Cad modeling:

Dimensions are taken through reverse engineering i.e. through manual calculations. CAD model of bucket, link, boom and arm is done. Figure 2 shows assembly in which components are arranged in such way that maximum digging force will act on bucket.

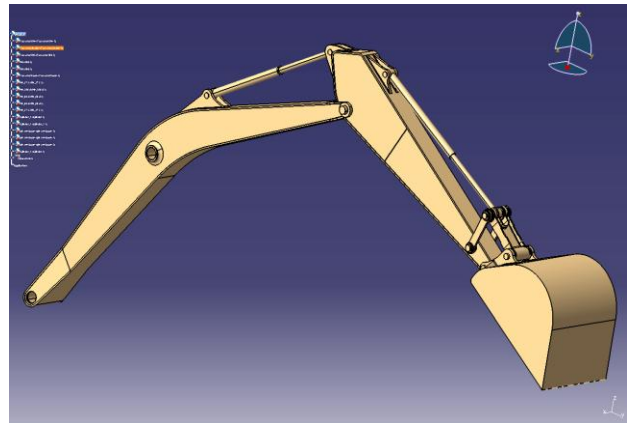


Figure 2: Assembly of excavator in CATIA

Pre processing:

Certain steps in formulating a finite element analysis are meshing, applying of boundary condition, assigning material properties etc. The .igs file is imported to the meshing software like Hypermesh. The CAD data of the excavator structure is imported and the surfaces were created and meshed. Since all the dimensions of boom are measurable (3D), the best element for meshing is the tetra-hedral. Individual component of an excavator is meshed independently and then connected by giving proper connectivity between elements. Element connectivity is a biggest issue in this assembly meshing. Each meshed component should have connectivity with its adjacent

component mesh. After meshing elements are to be checked for Quality i.e. elements have some definite quality criteria which should be met by all elements. An important element criterion for 3D mesh is tet-collapse.

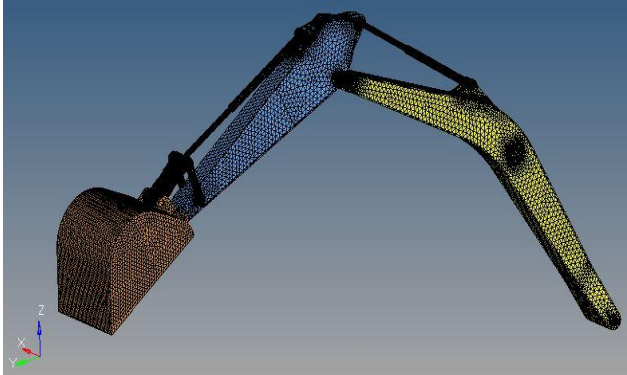


Figure 3: Meshed model of an excavator.

Figure 4 shows maximum digging force is applied of 54kN and constraints are applied to other end of boom.

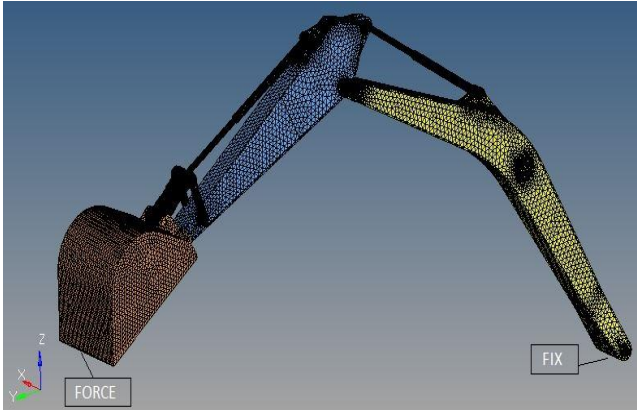


Figure 4: Physical constraints applied on excavator in Hypermesh.

Material properties:

Table 1: Part list of excavator:

Sr No.	Component	Material
1	Bucket	Hardox400
2	Bucket link	Hardox 400
3	Arm	Sailma 450
4	Boom	Sailma 450

Table2: Mechanical properties of Hardox400

Sr No.	Property	Value
1	Density	7473.57 Kg/m ³
2	Modulus of elasticity	210000 MPa
3	Poisson's ratio	0.29
4	Ultimate strength	1250 MPa

Table3: Mechanical properties of Sailma 450

Sr No.	Property	Value
1	Density	7900 Kg/m ³
2	Modulus of elasticity	210000 MPa
3	Poisson's ratio	0.3
4	Ultimate strength	700 MPa

VII. POST-PROCESSING

Analysis and evaluation of the solution results is referred to as post-processing. Post processor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution. Post-processing is done in ANSYS by importing .cdb format of file and then running it. Post-processing will give results of von-mises stress and deformation of excavator assembly. Figure 5 shows Von-misses stress on excavator assembly. Maximum von-misses stress is 626.08MPa which is less than yield value.

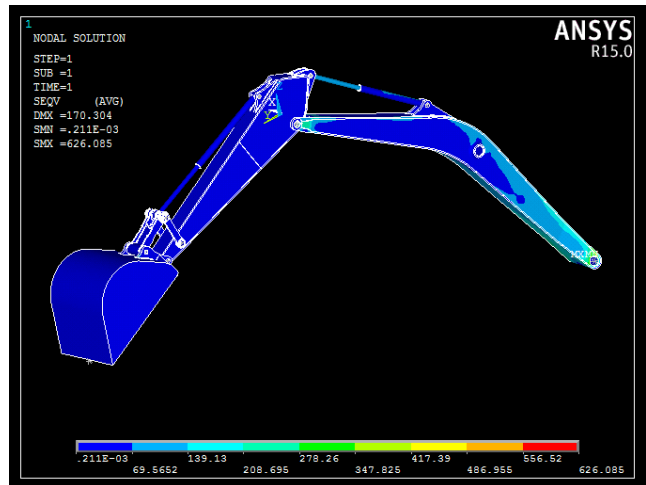


Figure 5: Von-misses stress on excavator assembly.

Deformation comes to be 170.3mm for maximum digging force applied on bucket. Figure 6 shows deformation developed on excavator assembly.

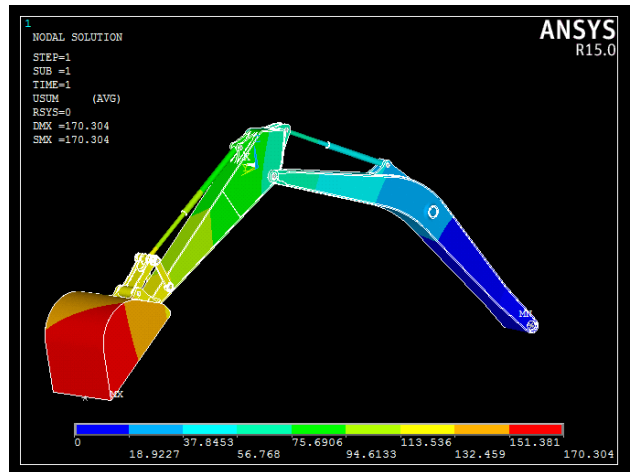


Figure 6: Deformation plot for excavator

VIII. OPTIMIZATION

From the plots of von-misses stress it can be seen that stress is below critical value so there is scope of weight optimization. The aim of this paper is to present a new design of excavator arm that will have better strength. The new design will comprise of ribs to give additional strength to the boom structure. So for weight optimization iterations are done by reducing thickness of boom plates having low stress flow and ribs are provided to maintain strength in boom

which will sustain applied digging force.

A. Iteration I:

Cad modelling: 2 mm thickness is reduced from side plate and ribs are made. Figure 7 shows CAD model of assembly used in iteration I.

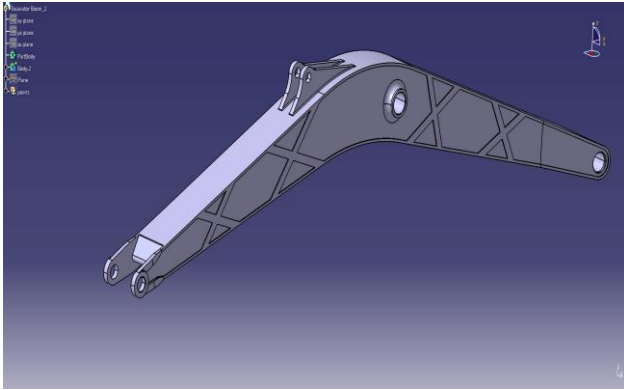


Figure 7: CAD model of boom with rib for iteration I

Meshing:

Figure 8 shows Meshed model of assembly with supporting ribs for iteration I

Meshing details:

Type of element: Tetra-hedral element

No. of nodes: 37699

No. of elements: 120717

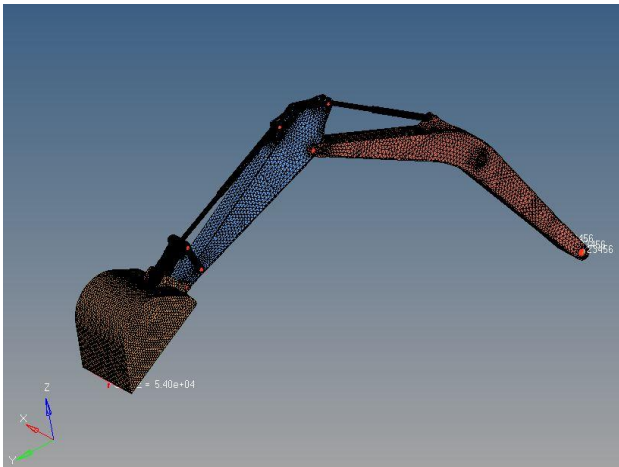


Figure 8: Meshed model of assembly for iteration I

Analysis:

Figure 9 Von-misses stress developed on excavator assembly for iteration I. Maximum von-misses stress is 598.07MPa which is less than yield value.

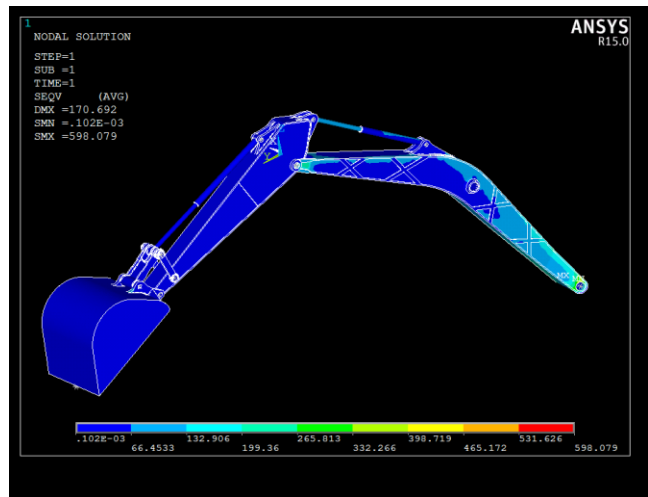


Figure 9: Von-misses stress on assembly for iteration I

Figure 10 shows deformation on excavator assembly for iteration I.

Deformation comes to be 170.7 mm for maximum digging force condition that was applied on bucket.

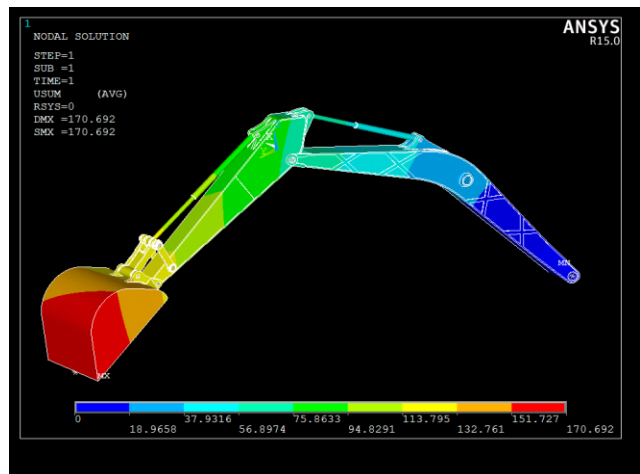


Figure 10: Deformation on assembly for iteration I.

B. Iteration II:

Cad modelling: Total 4 mm thickness is reduced from side plate and ribs are made. Figure 11 shows CAD model of boom.

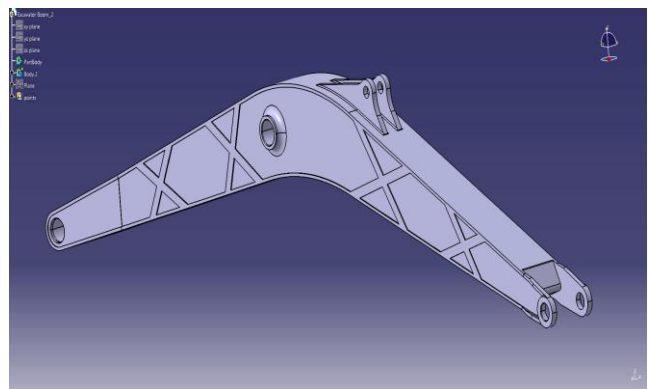


Figure 11: CAD model of boom with rib for iteration II

Meshing:

Figure 12 shows Meshed model of boom with supporting ribs for iteration II

Meshing details:

Type of element: Tetra-hedral element

No. of nodes: 37689

No. of elements: 120619

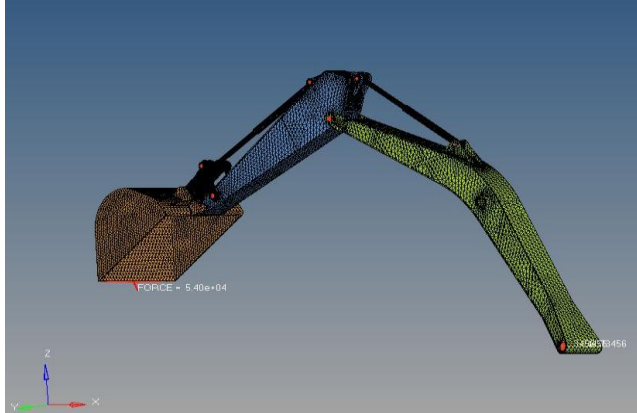


Figure 12: Meshed model of boom for iteration II

Analysis:

Figure 13 Von-misses stress developed on excavator assembly for iteration I. Maximum von-misses stress is 675.37MPa which is less than yield value.

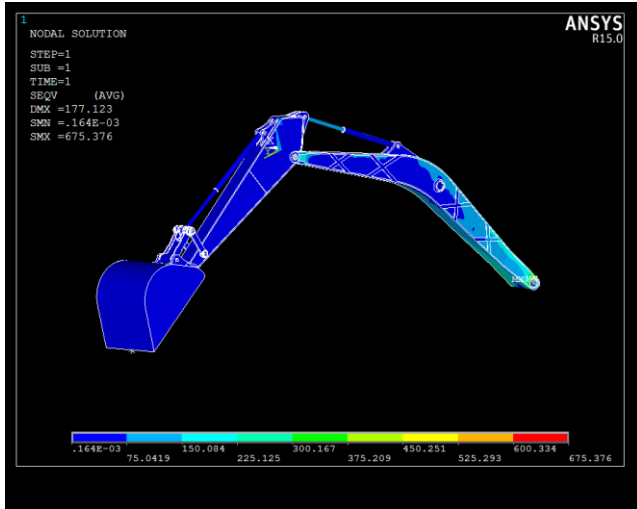


Figure 13: Von-misses stress on assembly for iteration II

Figure 14 shows deformation on excavator assembly for iteration I.

Deformation comes to be 177.1 mm for maximum digging force condition that was applied on bucket.

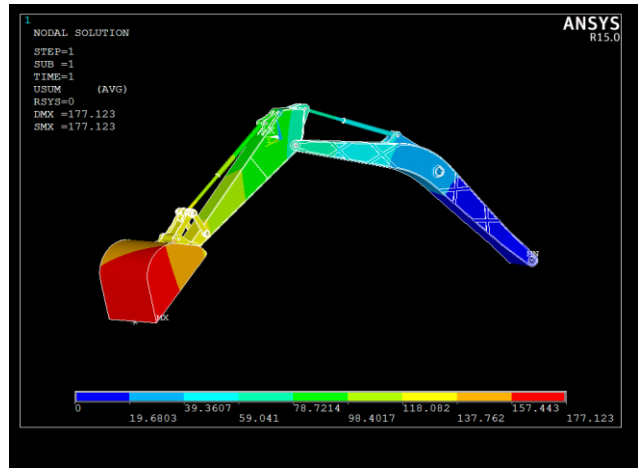


Figure 14: Deformation on assembly for iteration II.

IX. RESULTS AND DISCUSSIONS

Iteration I: Weight of excavator assembly before optimization is 1558.9 kg and after 1st iteration it reduces to 1523.61. Also von misses stresses after 1st iteration is 598.07 MPa which is less than 700 MPa.

Iteration II: Weight of excavator assembly before optimization is 1558.9 kg and after 2nd iteration it reduces to 1474.38. Also von misses stresses after 2nd iteration is 675.37 MPa which is less than 700 MPa.

So, results of above iteration shows there is scope of optimization within safe stress limit.

X. CONCLUSION

Iteration I give 2.23% weight reduction and Iteration II give 5.42% weight reduction within safe limit. More iteration are planned to check maximum scope of optimization within safe stress limit.

XI. PROPOSED WORK

Further iterations with reduce thickness are planned where side as well as upper and lower plates will be taken also by removing some material from low stress region and with adding some additional rib structure to get strength for boom structure wherever necessary.

After getting satisfactory result, scaled down model will be fabricated and tested on UTM for validation.

XII. ACKNOWLEDGMENT

The author would like to thank Dr. P.M. Ghanegaoker, Prof. and Dean of Mechanical Engineering Dr. D.Y. Patil Institute of Engineering and Technology, Pimpri, Pune – 18 for helping in formulate the problem and providing the essential inputs. The author will also like to thank the staff's from the mechanical engineering dept and principal of D.Y.P.I.E.T for helping to complete this work.

REFERENCEES

- [1]Amol B. Bhosale, Dr. Maruthi B H, Dr. Channakeshavalu K, “Optimization Of The Hydraulic Excavator Boom Using Fea Approach”, International Journal For Technological Research In Engineering, ISSN (Online): 2347 – 4718.
- [2]Janmit Raj, Gaurav Saxena, Rishi Kumar, Kuldeep Kaushik, “Study on the Analysis of Excavator Boom: A Review”, SSRG International Journal of Mechanical Engineering (SSRG-IJME) – volume 2 Issue 7– July 2015
- [3]S.SekharBabu, Y. Venu, “Design optimization of excavator bucket using Finite Element Method”, International Journal of Research and Innovation, 1401-1402
- [4]Aimin Ji, Kun Zhu, Xinlei Huang, Xu Yin, “Integration Design And Analysis Of Excavator Boom Based On CAD/CAE”, Journal of Theoretical and Applied Information Technology, ISSN: 1992-8645.
- [5]R M Dhawale and S R Wagh, “Finite Element Analysis Of Components Of Excavator Arm—A Review”, International Journal of Mechanical Engineering and Robotics Research, ISSN 2278 – 0149 Vol. 3, No. 2, April 2014
- [6]Bhaveshkumar P Patel, J M Prajapati, “static analysis of mini hydraulic backhoe excavator attachment using fea approach”, International journal of mechanical engineering and robotics research, ISSN 2278 – 0149, Vol. 1, No. 3, October 2012
- [7]Bhaveshkumar P. Patel, J. M. Prajapati, “A Review on FEA and Optimization of Backhoe Attachment in Hydraulic Excavator”, International Journal of Engineering and Technology, Vol. 3, No. 5, October 2011
- [8]Gaurav K Mehta, V.R.Iyer, Jatin Dave [6] “Finite Element Analysis and Optimization of an Excavator Arm”, HTC 09.
- [9]Juber Hussain Qureshi, Manish Sagar, “Study & Analysis of Boom of Backhoe Loader with the Help of FE Tool”, International Journal of Modern Engineering Research, ISSN: 2249-6645.
- [10]Dhruv N. Patel, “Finite Element Analysis of Excavator Arm”, International Journal for Scientific Research & Development| Vol. 2, Issue 06, 2014| ISSN (online): 2321-0613
- [11]C. P. Motka, Ikbalahemad R Momin, “Development of Backhoe Machine By 3-D Modelling Using CAD Software and Verify The Structural Design By Using Finite Element Method”, International Journal for Innovative Research in Science & Technology| Volume 1 | Issue 8 | January 2015 ISSN (online): 2349-6010