Design and Optimization of Arm of Excavator

Mr. Altaf S. Shaikh[1]
PG Scholar,Department of Mechanical Engineering,
JSPM's, Rajarshri Shahu School of Engineering and Research Narhe, Pune.
altaf8084@gmail.com

Dr. B.M. Shinde[2]
Head,Department of Mechanical Engineering,
JSPM's, Rajarshri Shahu School of Engineering and Research Narhe, Pune.
bharat_mshinde@yahoo.co.in

Abstract - **Excavators are earth moving equipment normally used for excavating hard rocks and soil below the natural surface of the ground. Because of several working condition excavator parts like bucket, arm, and boom are subjected to high loads. During the excavation operation there is an unknown resistance forces offered by the terrain to bucket teeth. Excessive amount of these forces adversely affected on the machine parts and may fails during excavation operation. Thus, it is necessary to provide not only a better design of parts having maximum reliability but also of minimum weight and cost, keeping design safe under loading conditions. In this paper Finite element analysis (FEA) of existing excavator arm is compared with optimized arm for stresses and deflection. Here FEA approach is applied for the Optimization. This paper discuss about finite element based optimization of excavator arm and thus helped in finding out the most appropriate design of which a prototype is fabricated and tested. This paper includes the study of various iterations of excavator arm and it is found that iteration 4 has sufficient amount of material removed without affecting its strength and finally FEA results is compared with Experimental results.**

Keywords: Excavator arm, static force analysis, Catia V5 R19, FEA, Optimization.

I. INTRODUCTION

A hydraulic shovel of a bucket type excavator is an earth moving machine comprising an upper rotatable chassis mounted on a drivable body with wheel or track and hydraulically powered mechanism consisting of boom, arm and bucket, mounted to the upper chassis as shown in fig.1.
Applications for excavator in India used as a utility machine at large construction sites and urban infrastructure projects as well as the loading of hoppers and trucks, trenching, the cleaning of canals and ditches, general excavation, solid waste management and even demolition and mining work. The useful task of backhoe hydraulic excavator is to free and/or remove surface materials such as soil from its original location and transfer it to another location by lowering the bucket, digging, pushing and/or pulling soil then lifting, swinging The excavation of this task is usually performed by a human operator who controls the motion of the machine manually by using the visual feedback provided through his or her own eyes.

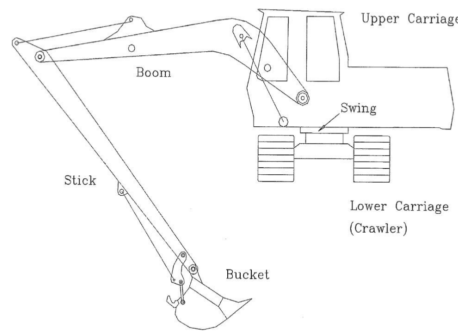

Fig.1: A typical Excavator structure

Fig.2: Existing model (Site image)

Normally excavators are working under worst working conditions. Due to severe working conditions, excavator 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 an equipment of maximum reliability but also of minimum weight and cost, keeping design safe under all loading conditions. The force analysis and strength analysis is important steps in the design of excavator parts.
In this paper, 3D modeling is done in CatiaV5.Meshing and analysis is carried out in Hypermesh and Ansys respectively. Stress and deformation is the output of analysis. Re-designing the arm having rib/truss like structure, following the same procedure for 3D modeling, meshing and analysis. Stress values must be below critical value to ensure that the new design is safe.

Several studies have been done relating to the Design and optimization of various parts of excavator. Following is a list of researchers who has worked in this area of excavator and optimization.

Bhaveshkumar P. Patel, Jagdish M. Prajapati[1] worked on Structural Optimization Of Mini Hydraulic Backhoe Excavator Attachment Using Fea Approach trial and error method. Shape optimization also performed for weight optimization and results are compared with trial and error method which shows identical results. The FEA of the optimized model also performed and their results are verified by applying classical theory. Comparison shows that the variations in results of individual parts are very less and total variation in result is of only 3.93% which reflect that the results of structural weight optimization performed by trial and error method are accurate and acceptable. The differences in results of the Von Mises stresses and the classical theory are very less and we can say that the results are identical and acceptable.

Bhaveshkumar P. Patel and J. M. Prajapati[2] 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.

Sachin B. Bende, Nilesh P. Awate[5] concentrated on the study of the components of the excavator in order to identify the problems faced while performing the lifting and digging operations and to provide a design solution by using CAD-CAE systems. As the present mechanism used in excavator arm is subjected to torsional and bending stresses during lifting and digging operation respectively, because of which failure occurs frequently at the bucket end of the arm. So, the new mechanism of excavator arm is designed and analysis is done at existing digging force and also at newly calculated digging force. Also the bucket volume is increased to compensate for the loss in production due to the reduction in digging force.

Gaurav K Mehta, V.R.Iyer, Jatin Dave[6] performed Finite Element Analysis and Optimization of Excavator Arm. Forces acting on each part of mechanism are obtained by static force analysis of mechanism considering different critical operating conditions. It is found out that the condition in which mechanism is producing maximum digging force is the most critical condition for static force analysis. Each link is analyzed as a free body for this condition and a force coming on each link is obtained. After interpreting FEA results, stresses in bucket, arm, and boom are found to be within allowable stress limit.

R M Dhawale and S R Wagh[7] worked on "Finite Element Analysis Of Components Of Excavator Arm. Various analysis done on components of excavator arm and there are various forces affects on components of excavator arm. Also the bucket volume is increased to compensate for the loss in production due to the reduction in digging force. Increased in bucket volume will also increase the amount material to be fed in the bucket.

Shilpa D. Chumbale Prasad P. Mahajan[8] studied the Failure Analysis and Optimization of Excavator Arm in this new mechanism of excavator arm is designed and the Pro-e software is used for making the 3D model of the excavator arm linkage. By using ANSYS workbench software static analysis of each of the excavator arm component is done at existing digging force and also at newly calculated digging force. Also the bucket volume is increased to compensate for the loss in production due to the reduction in digging force. Dynamic analysis is also done by applying the force acting when the bucket is fully filled with material. The comparison of proposed model with the existing model is done.

Gui Ju-Zhang, Cai Yuan-Xiao, Qing-Tan and You Yu-Mo[9] carried out mechanical analysis in three typical work condition of the working device by using the mechanical theory and method. The static strength finite element analysis of excavator boom was carried through by using ANSYS, from which, the stress and strain deformation contour diagrams of three typical work conditions were obtained. The results of finite element analysis showed that the static intensity of the boom is enough. The study results are of certain guiding significance for working device's optimization design.

II. STATIC FORCE ANALYSIS OF ARM AND BUCKET

In this section calculation for the static force analysis of the bucket and arm excavator for the condition in which the mechanism produces the maximum breakout force has been done. In static analysis one configuration of the mechanism has to be decided first for which the analysis is to be carried out. The maximum breakout force condition is the most critical one as it produces the highest breakout force, and thus for this condition the force analysis is done, and will be used as a boundary condition for static FEA.

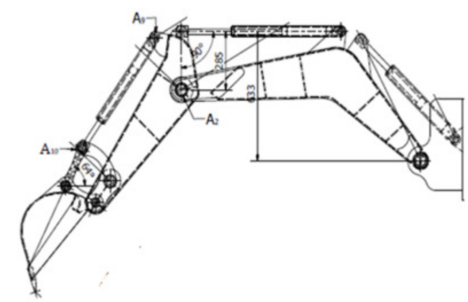

Fig.3: Configuration in which the mechanism is producing the maximum breakout force.

The free body diagram of bucket and arm, with directions and magnitudes of the forces are explained in this section.

A. Bucket static forces:

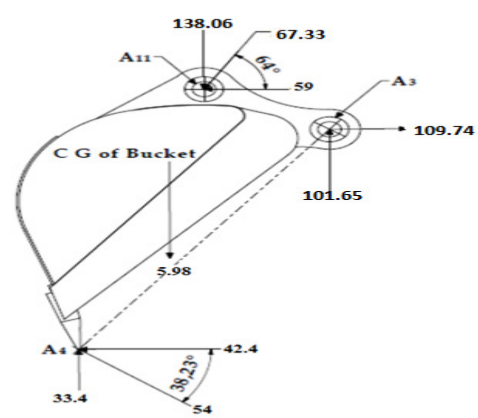

Fig.4: Free body diagram of bucket

All the forces shown in fig.4 are in KN

Maximum bucket digging force is considered to be 54 KN.

CAT 320Dl	General duty excavators
Bucket Digging force	54 KN

Ref: Caterpillar catalogue

Table 1: Static forces on the bucket joints:

Joint of the bucket	Horizontal (X) component (KN)	Vertical (Y) component (KN)
A4	-42.4	33.4
A11	-67.33	-138.06
A3	109.74	101.65

B. Arm static forces:

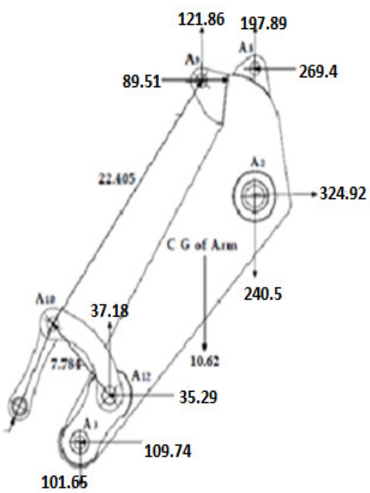

Fig.5: Free body diagram of arm of excavator.

Table 2: Static forces on the arm joints

Joints	Horizontal (X) component (KN)	Horizontal (Y) component (KN)
A3	-109.74	-101.65
A12	-35.29	37.18
A9	89.51	121.86
A8	-269.4	-197.89
A2	324.92	240.5

III. DESIGN AND ANALYSIS OF EXCAVATOR ARM

This design and analysis of excavator arm of includes design and analysis of existing excavator arm. Dimensions of the existing excavator arm have been extracted from the reverse engineering and CAD model has been prepared in CATIA V5. Calculations are done for finding the forces acting on excavator arm. The finite element analysis is carried out by using Hypermesh and ANSYS.

CAD MODELLING:

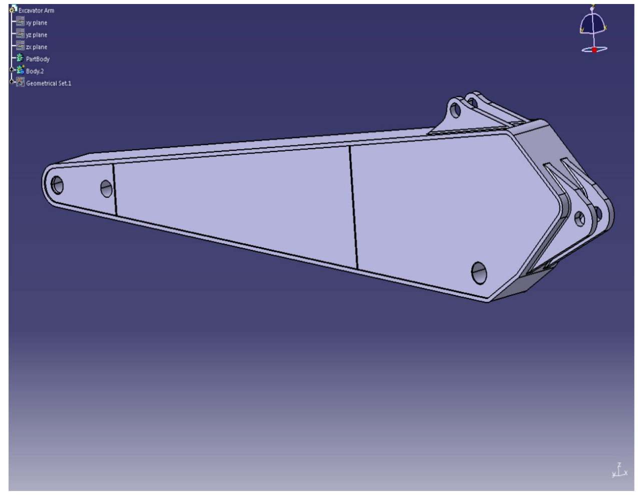

Fig.6: CAD model of arm of excavator drawn in CATIAV5

ANALYSIS:
Analysis is done by selecting appropriate solver and carrying out the operations in various stages to obtain solution. Particularly analysis is carried out in three stages by performing various operations in software.

[A] Preprocessing:

1) Meshing:

In this stage igs file is imported to the meshing software like Hypermesh. The CAD data of the arm structure is imported and the surfaces were created and meshed. Since all the dimensions of arm are measurable (3D), the best element for meshing is the tetra-hedral.

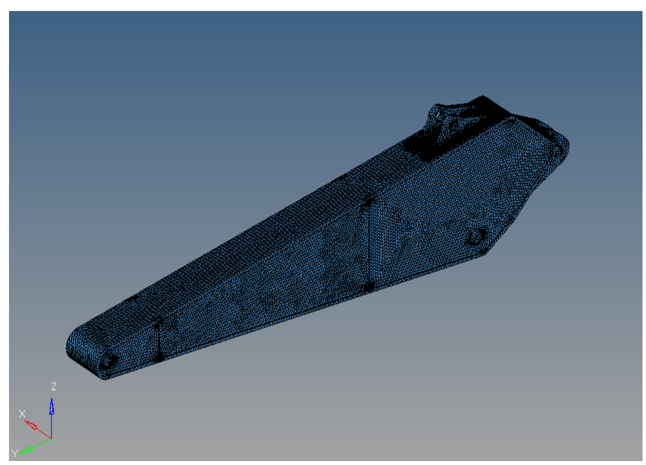

Fig.7: tetra-hedral meshing on arm of excavator

Number of nodes: 73244

Number of elements: 266200

Element size = 4 mm

2) Boundary Condition:

After meshing is completed we apply boundary conditions. These boundary conditions are the reference points for calculating the results of analysis.

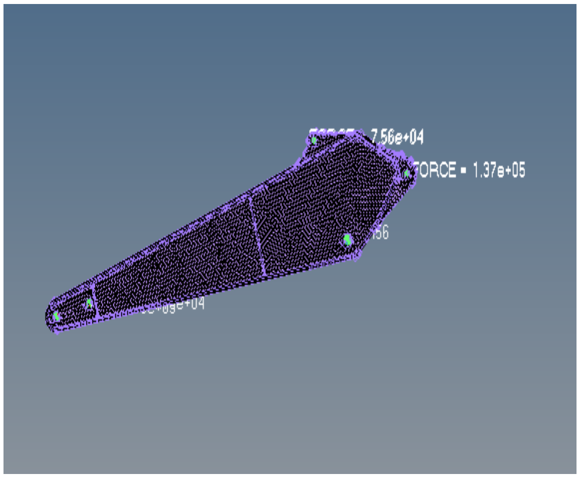

Fig.8: forces and constraints applied in hypermesh.

[B] Solution and Post-processing:

Meshed and boundary condition applied model is imported to the solver. Analysis process starts after applying run in the solver software. Software first calculates the deflection with respect to the boundary conditions applied. Then on the basis of deflection it calculates strain.

Following are the results displayed for stress and deformation:

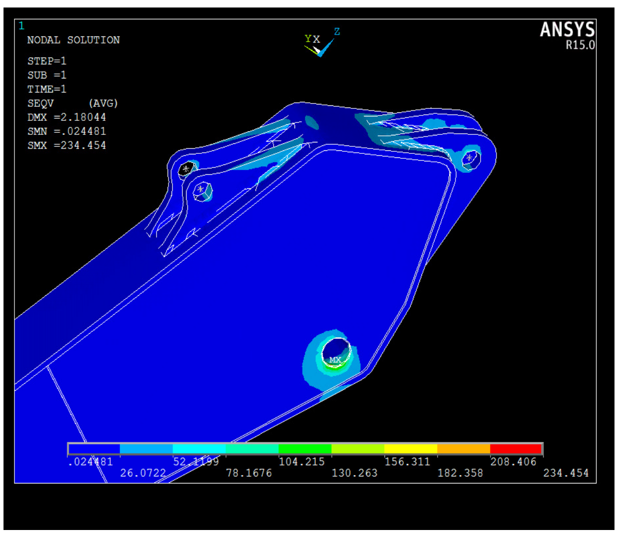

Fig.9: Von-mises stress at joints of excavator arm

Stress value for excavator arm is 234.45 N/mm^2 which is well below the critical value. Hence, design is safe.

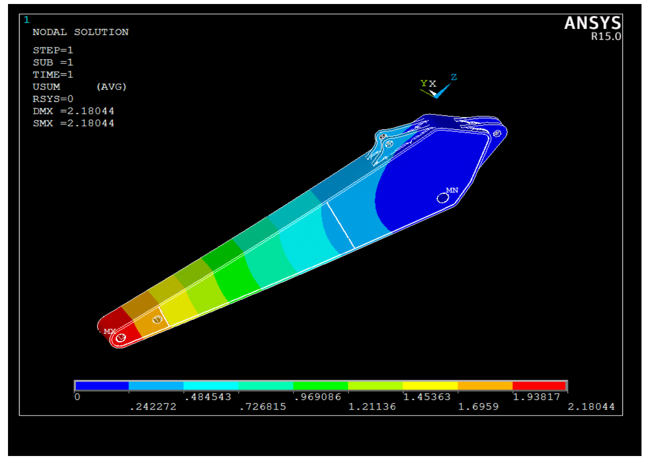

Fig.10: Displacement result for excavator arm

From fig.12, deformation for excavator arm is 2.18 mm

IV. OPTIMIZATION OF ARM

This chapter includes finite element based optimization of excavator arm. The chapter discuss about the analysis of various CAD models of excavator. Modification have been made in CAD model of excavator arm then checked for its stresses and deflection.

CAD model of Optimized arm:

Changes are made in CATIAV5 in existing arm as shown in below figure.

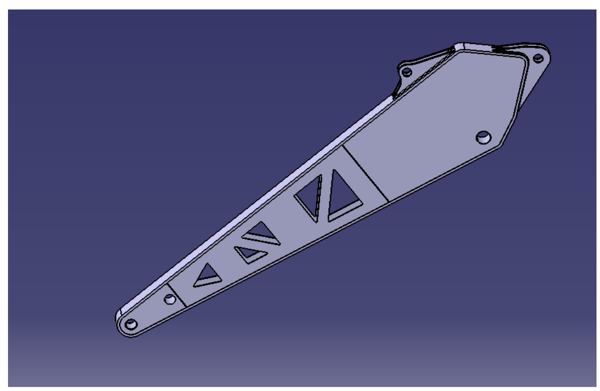

Fig.11: CAD model of Optimized arm

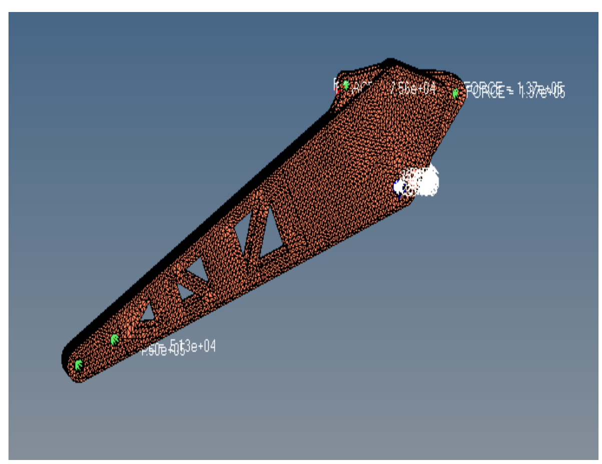

Fig.12: Meshed model and boundary condition application

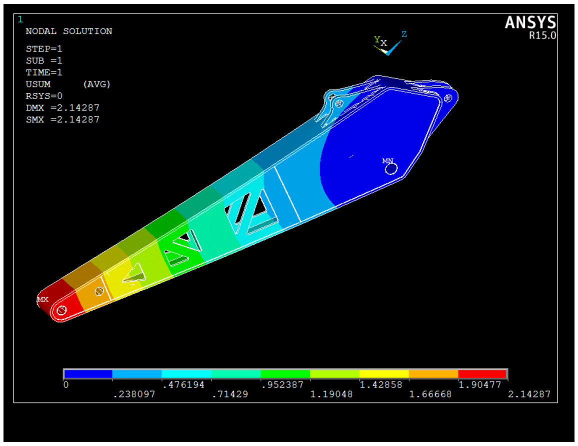

Fig.13: Displacement result for optimized arm

The maximum deformation is found to be 2.14 mm which is very less.

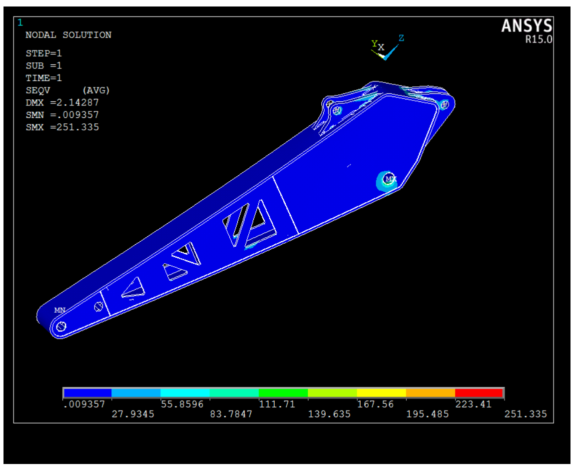

Fig.14: Von-mises stress for excavator arm

The max stress obtained is 251.33 MPa which means the design is safe.

Analysis of arm of excavator has been done for existing design of arm as well as for optimized design.
From the analysis following comparison stresses and deformation are observed.

Table.3: Comparison of stresses, Deformation and weight

	Stress (MPa)	Deformation (mm)	Weight (kg)
Existing	**234.4**	**2.18**	**1082.2**
Iteration 1	210.9	2.10	1065.1
Iteration 2	183.5	2.08	1054.9
Iteration 3	183.24	2.09	1041.5
Iteration 4	**251.33**	**2.14**	**1034.6**

V. EXPEREMENTAL SET UP

Optimized model of arm is fabricated as sown in fig.15 and further testing is done on UTM machine.

Fig.15: Fabricated prototype model of optimized arm

Fig.16: Experimental test setup on UTM

VI. EXPERIMENTAL RESULT & VALIDATION:

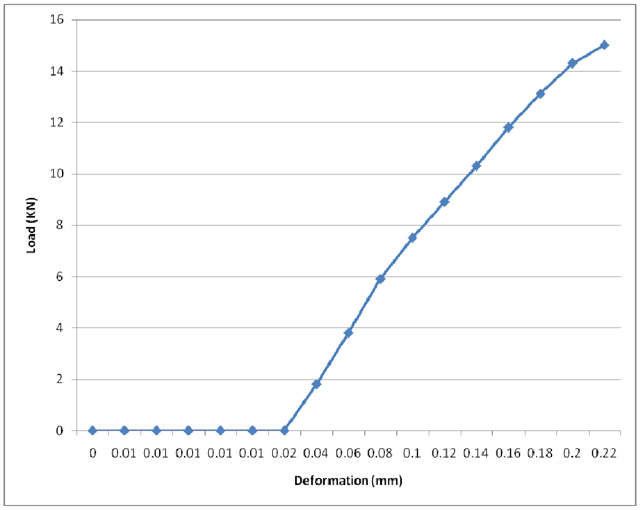

Fig.17: Graph of Load Vs Deformation

The deformation for experimental result is 0.20 mm

As the optimized model is 10 times scaled,
The deformation for experimental result is 0.20×10 = 2 mm

Table.4: Comparison of FEA results with Experimental results

Optimized model of arm	**Deformation in mm**
FEA Result	2.14
Experimental Result	2

Percentage of error

= (0.214-0.20)/0.214= 0.065
= 6.5%

VII. CONCLUSION

1) The forces on the excavator are calculated and the forces flowing to excavator arm are determined
2) The excavator arm is modeled and analysed using software.
3) The analysed part shows there is a scope for optimization.
4) The optimizations of the excavator part is carried out by different iterations and finally the optimized results are obtained.
5) Excavator arm is fabricated and experimentally tested.
6) The FEM results and experimental results are made a comparable study and the validation shows close variance.
7) From comparison of weight of existing model and optimized model it is seen that Overall weight reduction of 5% approximately has been achieved.

Arm of excavator	Existing model	Optimized model
Weight	1082.2 kg	1034.6 kg

REFERENCES

1. Bhaveshkumar P. Patel, Jagdish M. Prajapati[1] “Structural Optimization Of Mini Hydraulic Backhoe Excavator Attachment Using Fea Approach”, machine design, Vol.5(2013) No.1, ISSN 1821-1259
2. Bhaveshkumar P. Patel and J. M. Prajapati[2] “A Review on FEA and Optimization of Backhoe Attachment in Hydraulic Excavator”, IACSIT International Journal of Engineering and Technology, Vol. 3, No. 5, October 2011.
3. Atul Sambhaji Jadhav, Prof. A.P. Shrotri, Swapnil Kulkarni[3] “Design Optimization For Front Loader Tooth Of Earthmoving Equipment”, International Journal of Advanced Engineering Research and Studies, E-ISSN2249–8974
4. Sachin B. Bende1, N. P. Awate[4] “Computer Aided Design Of Excavator Arm: Fem Approach”, Vol. 2, No. 1, January 2013
5. Sachin B. Bende, Nilesh P. Awate[5] “Design, Modeling And Analysis Of Excavator Arm”, International Journal of Design and Manufacturing Technology (IJDMT), ISSN 0976 – 6995(Print), ISSN 0976 – 7002(Online) Volume 4, Issue 2, May - - August (2013), pp. 14-20
6. Gaurav K Mehta, V.R.Iyer, Jatin Dave[6] “Finite Element Analysis and Optimization of am Excavator Arm”, HTC 09
7. R M Dhawale1 and S R Wagh[7] “Finite Element Analysis Of Components Of Excavator Arm—A Review”, International Jouranal of Mechanical Engineering and Robotics Research, ISSN 2278 – 0149Vol. 3, No. 2, April 2014
8. Shilpa D. Chumbale1 Prasad P. Mahajan[8] “Failure Analysis and Optimization of Excavator Arm - A Review”, International Journal for Scientific Research & Development| Vol. 3, Issue 02, 2015 | ISSN (online): 2321-0613
9. Gui Ju-Zhang, Cai Yuan-Xiao, Qing-Tan and You Yu-Mo[9] “Finite element analysis of working device for hydraulic excavator”, Journal of Chemical and Pharmaceutical Research, 2013, 5(12):123-128
10. Caiyuan Xiao and Zhang Guiju[10] “Modal Analysis on Working Equipment of Hydraulic Excavator”, The Open Mechanical Engineering Journal, 2015, 9, 173-180