ISSN 2395-1621

Modeling, Static Analysis and Optimization of Backhoe Attachments of Excavator by Using CAE Tools

#1Jaydeep Patil, #2Tejaskumar Bharsakale, #3Anil Sawant, #4Sumit Shinde, #5Swapnil Bhoite

[1]Patil.jaydeep015@gmail.com,
[2]tejasbharsakale@gmail.com,
[3]anil982265@gmail.com,
[4]sumitshinde178@gmail.com,
[5]swapnilbhoite93@gmail.com

#1Assistant Professor, Mechanical Engineering,
#2345Mechanical Engineering

Savtribai phule university, pune India
Padmabhooshan Vasantdada Patil Institute of Technology, Bavdhan- Pune Maharashtra, India

ABSTRACT

Excavators are high power machines used in the mining, agricultural and construction industry whose principal functions are digging (material removing), ground levelling and material transport operations. Backhoe attachment is rear part of excavator machine. The backhoe attachment is subjected to static as well as dynamic forces. This project work includes static force analysis and design modification of backhoe assembly of excavator by using FEA. Initially CAD model of actual working model is created in CAD software package CATIA V5. Maximum breakout forces acting on bucket teeth were calculated analytically and these are used as boundary condition for FEA analysis perform using HYPERMESH as a pre-processor and ABAQUS as a solver. Results of analysis were interpreted in HYPERVIEW and modifications were made in initial model by trial and error method. Various designs were compared to each other in terms of mass, maximum displacement, maximum von-mises stresses and optimum model is selected from available alternatives. Modified design selected has critical stresses with in permissible limit with reduced weight.

ARTICLE INFO

Article History

Received: 7th June 2016

Received in revised form :

7th June 2016

Accepted: 9th June 2016

Published online :

10th June 2016

***Keywords*- Excavator, Backhoe, CAD, FEA.**

I. INTRODUCTION

Development of any country is mainly depending upon industrial sector, agriculture, construction; transportation etc. Heavy vehicles are the backbone of the entire above sector hence these vehicles should work properly in all working condition. The efficiency of these vehicles depends upon type of loading, operating condition duration of use and maintenance. To increase life of vehicle static as well as dynamic analysis of above condition should be done. These heavy vehicles include multiaxel Trucks, Cranes, Bulldozers, Tractors and Road rollers etc.

Today in the machine age when the use of machines is increasing for the earth moving works, considerable attention has been focused on designing of these earth moving equipments. Achievement of an ambitious and rapidly growing rate of industry of earth moving machines is assured through the high performance construction machineries with complex mechanism and automation of construction activity. Bulldozers, scrapers, motor graders, excavators and other machines are widely used for most arduous earth moving work in engineering construction. Thus it is very much 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 by careful stress analysis of the machines. This arises a scope for developing some standard methodology to calculate the loads on the machine and carrying out stress analysis of various parts of earth moving machineries [1]

II. 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 more sophisticated description of the FE method regards it as piecewise polynomial interpolation. That is over an element a field quantity such as displacement is interpolated from values of the field quantity at nodes. [2]

III.MODELLING

Finite element analysis is performed on CAD model. Though modeling is not part of FEA but it is considered as prerequisite for finite element analysis. Accuracy of Results produced by FEA tool depends on accuracy of CAD model created by CAD software package CAD model should represent the actual geometry of object to be analyzed.
Modeling of backhoe attachments of excavator is done in CAD software package CATIA V5. While modeling every part reference of original dimensions is taken. Detailed dimensions are taken from manufacturers catalog and actual measurement according to that CAD model representing approximate actual model of backhoe attachments.

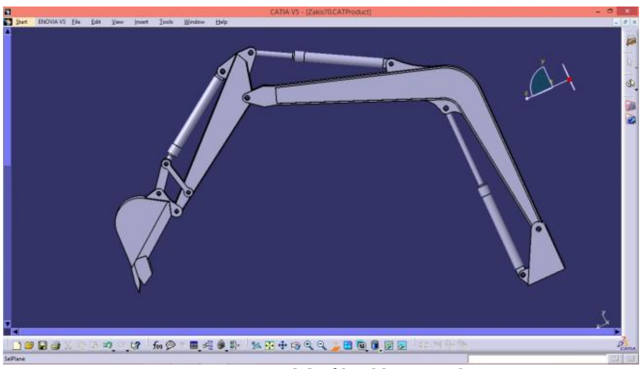

Figure 1 CAD Model of backhoe attachment

IV.BOUNDARY CONDITION CALCULATION

i. Bucket capacity calculation

To select bucket from various buckets available for the model bucket capacity is calculated.
As shown in fig 2 the heaped capacity can be given as,

$V_h=V_s+V_e$ .... (1)

Where, Vs is the struck capacity, and Ve is the excess material capacity. Struck capacity can be calculated as,

$V_s = P_{area}\left(\frac{(Wf+Wr)}{2}\right) = 0.2586\ m^3$........................ (2)

Excess material capacity can be calculated as,

$V_e = \left(\frac{L_B * W_f^2}{4}\right)-\left(\frac{W_f^3}{12}\right) = 0.06858\ m^3$........ (2)

From (1), (2) and (3) bucket capacity for the proposed 3D backhoe bucket model comes out to be 0.3271 m^3.

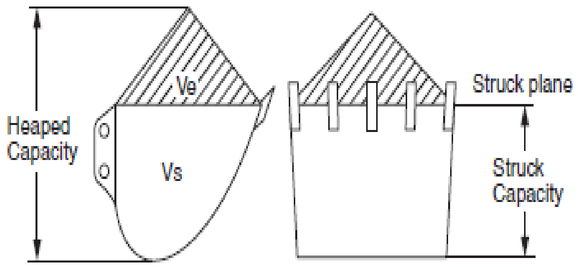

Fig 2 Bucket capacity

This value of bucket capacity is used for selection of bucket weight.

Table1. Bucket Capacity

Heaped capacity	Width	Weight
0.24m^3	500mm	231Kg
0.27m^3	600mm	251Kg
0.32m^3	700mm	286Kg
0.38m^3	800mm	306Kg

ii. Cylinder pressure calculation

To find the force boundary condition we have to calculate cylinder pressure. From specification we have 3 variable displacement axial piston pump out of which bucket and arm cylinder pump have flow 68 lit/min and boom cylinder pump have flow 52.5 lit/min and model have fluid power 39 Kw (52 hp).

HP = P*Q/1714.......................... (4)

Where,

H.P. = Fluid power

P= Maximum cylinder pressure, psi

Q=Maximum oil flow of pump, gpm

Bucket cylinder pressure

$P = \frac{HP*1714}{Q}$

Bucket cylinder pressure = 53.205MPa

Bucket Cylinder Force = Working Pressure *end area of bucket cylinder

= 175565.859 N

Bucket Curling force (Fb) = $\frac{\text{Bucket cylinder force}}{d_D}\left(\frac{d_A * d_C}{d_B}\right)$

Bucket Curling or Brackout Force (Fb) = 48 KN

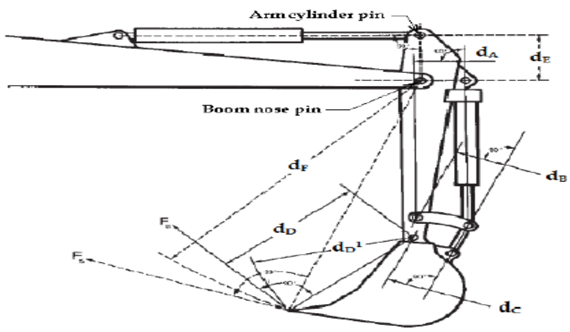

Fig. 3 Digging Forces by Standard SAE J1179

Arm cylinder pressure
P = HP*1714/Q
= 41.07 MPa
Arm cylinder Force (Fs) = Working pressure* end area of arm cylinder
= 174990.44 N

Arm crowd force (FS) = $\frac{\text{Arm cylinder force} * d_E}{d_F}$

Arm crowd force (FS) = 33.26 KN

In static analysis one Configuration of the mechanism has to be decided first for Which the analysis is to be carried out. From all the Configurations, 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 [3]

V. FINITE ELEMENT SIMULATION

i. Geometry clean-up

FEA tool uses geometry created in CAD software package. While modeling object some errors remain in the geometry and some are created while importing geometry in CAE software. While importing, CAE software may miss some data. These errors include duplicate surfaces, free edges, suppressed edges, non manifold edges, theses errors are represented by different colours in HYPERMESH.
As we know only a perfect error free geometry can produce correct result. Any of the above mentioned error can take away results from accuracy. So it is important to remove all these errors, to remove these errors we can use quick edit option and also some surfaces are created according to requirement. All errors are corrected till all parts of geometry turns green colour, which represents shared edges which is our requirements. Following figure shows cleaned geometry after removing all errors.

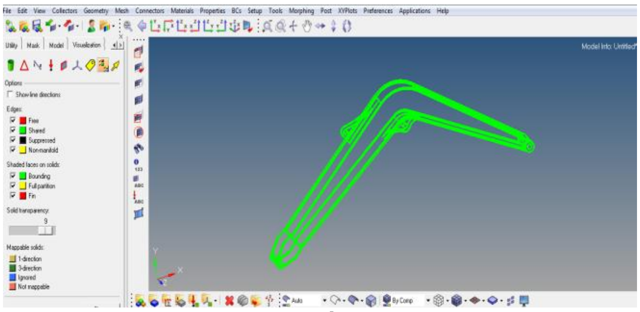

Fig.4 Geometry Clean-up

ii. Meshing

FEA uses a complex system of points called nodes which make a grid called a mesh. 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. [2]

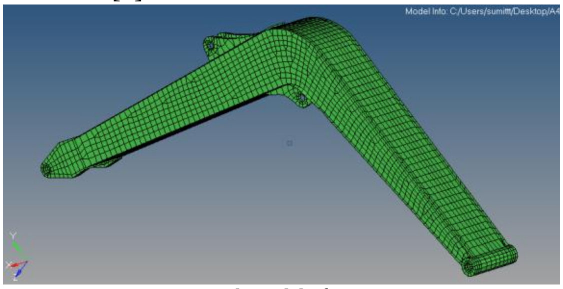

Fig.5 Mesh model of Boom

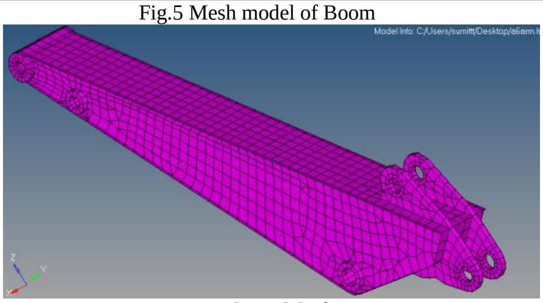

Fig. 6 Mesh model of Arm

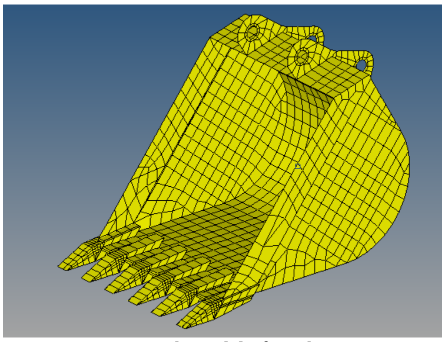

Fig. 7 Mesh model of Bucket

Table 2 Nodes and Elements summery

Component	Nodes	Elements
Boom	5942	4456
Arm	3931	2225
Bucket	14896	12595

After quality meshing of components material properties were assigned to the mesh model.

iii. Material Properties

AISI 1040 is the material used for the backhoe attachments of excavator.having allowable yield strength 415MPa. AISI 1040 carbon steel has high carbon content and can be hardened by heat treatment followed by quenching and tempering to achieve good properties.
Mechanical properties of AISI 1040
Yield strength- 415 mpa
Bulk Modulus- 140000 mpa
Elastic Modulus- 210000 mpa
Poissons Ratio- 0.27-0.30
Density- 7.845 g/cc

iv. Boundary Condition

As calculated earlier Maximum Breakout force 48 KN at angle of 38.23◦.The force is applied such that it will be distributed uniformly on all teeths. Considering that this force will transmit to arm, boom and supporting structure. The supporting structure is fix in all direction.

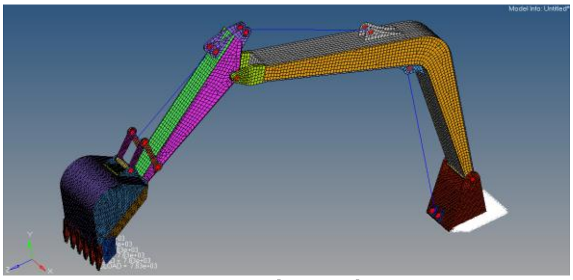

Fig. 8 Boundary Conditions

VI. RESULT AND DISCUSSION

1. Von-mises stresses

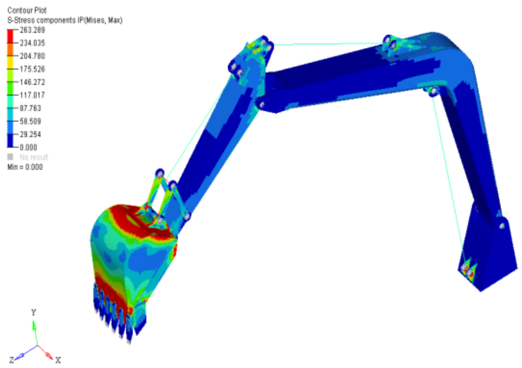

Fig . 9 Von-Mises Stresses in Backhoe

Maximum von-mises stresses were seen in bucket and at the cylinder hinge supports which are 263.289Mpa and Minimum stresses were seen in boom which are far below allowable stress value which is 415MPa.

Table 3. Result Summery

Parameter	Maximum	Allowable
Von-mises stress(MPa)	263.289	415

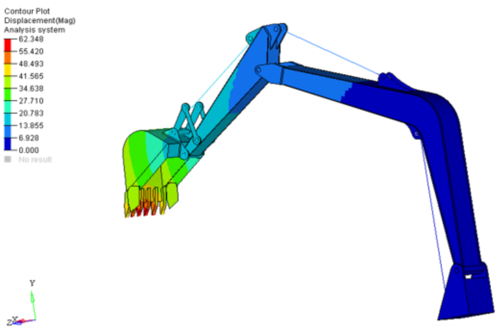

Fig.10 Displacement of backhoe

Above results are obtained after solving model for applied boundary condition in solver abaqus. It is observed that maximum displacement is 55.420mm which occure in Bucket and Minimum displacement is 6.928mm. displacement in arm is in the range from 6.928mm to 20.783mm. After studying displacement results we came to know that displacement is minimum in boom. For the displacement in boom stresses produce are less than allowable stresses. So model is safe for applied boundary conditions but the difference between actual stresses and allowable stresses is more hence we can go for optimization. So that size can be changed with reduced material requirement and consequently cost can be reduced.

OPTIMIZATION

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. As stated in previous section stresses induced in boom are less than that of bucket and arm. So we can go for optimization of boom by modifying boom design keeping stresses induced in boom within permissible limit.[4]

i. Optimization of Boom

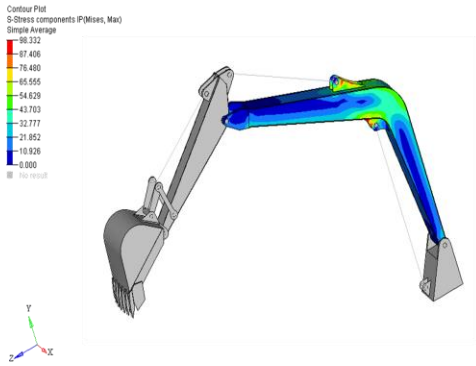

Fig 11 Von-Mises Stresses In Boom

Above figure shows von-mises stresses in boom, we can see that maximum stresses induced in boom are 87.406-98MPa.323MPa. These stresses only seen in cylinder hinged support. Excepting cylinder support bracket stresses induced in remaining boom body are far below yield stresses. This triggers the scope for optimization of boom.

For optimization starting from initial design various alternative design are created and compared with each other in terms of mass, maximum displacement, maximum von-mises stresses and finally a best design is selected for boom in which boom body. thickness is reduced from 20mm to 18mm. following are the results obtained after optimization.

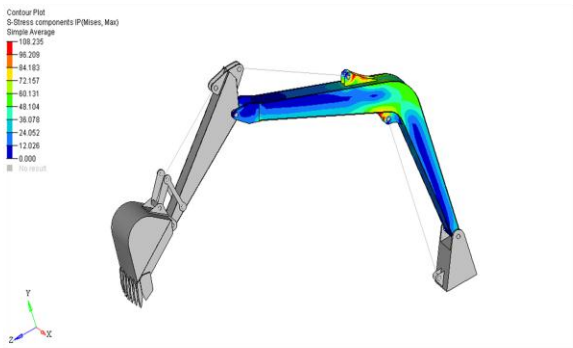

Fig 12 Von-mises stresses in optimized model of boom

The effect of optimization of boom on entire backhoe assembly is shown in following fig.

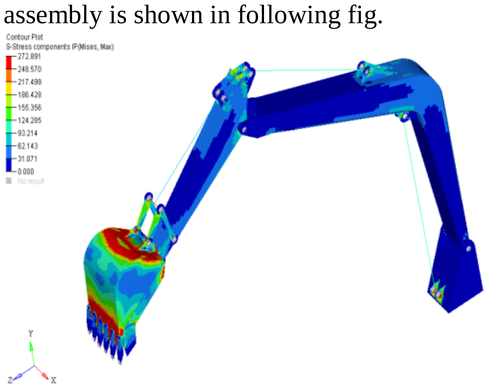

Fig. 13 Von-mises stresses in optimized model

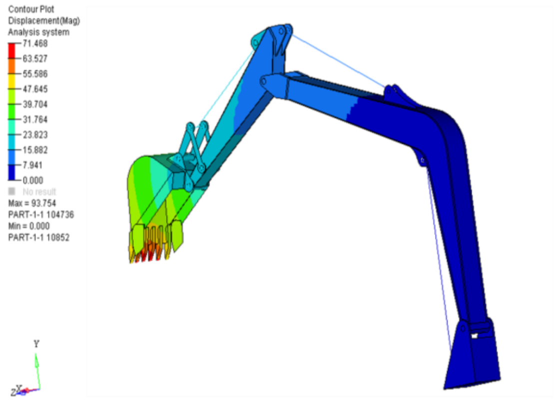

Fig 14 Displacement in optimized model.

As we can see maximum von-mises stresses increased from 263.289MPa to 272.891MPa and Maximum displacement increased from 62.368mm to 71.468mm, but due to reduction in thickness of boom overall weight of backhoe assembly reduced from 2183Kg to 2098Kg. i.e. Total weight reduction is 85Kg, We can see that even after reducing thickness stresses induced remain below yield stresses, so we can say that optimization of boom is justified.

VII. CONCLUSIONS

This paper deals with static force analysis and optimization of backhoe assembly of excavator. By using finite element analysis. Initially literature and data collection is done to understand approach and methodology to perform analysis. CAD Model is generated by using CATIA-V5 FEA technique is used to simulate the operating conditions of mechanism without actually making the prototype. After interpreting FEA results Stresses in Boom, Bucket and Arm are found to be within allowable stress limit. Moreover modification are made in boom model for minimum weight of component without affecting functional requirement.

IX. ACKNOWLEDGEMENT

The work presented in this paper is by no means complete but it gives comprehensive representation of finite element technique applied to to the static force analysis of backhoe attachments of excavator. Authors wishes to apologize for the unintentional exclusion of missing referances and would appreciate receiving comments and pointers to other relevant literature for future update.

REFERENCE

1] C.P. Motka, "Development of backhoe machine by 3D modelling using CAD software and verify the structural design by using finite element analysis", IJIRST, volume 1, ISSUE 8, January 2015, ISSN-2349-6010

2] Juber Hussain Qureshi and Manish Sagar (2012), "The Finite Element Analysis of Boom Of Backhoe Loader", International Journal Of Engineering Research And Applications (Ijera), Issn: 2248-9622.

3] Bhaveshkumar P. Patel And J. M. Prajapati, Evaluation Of Bucket Capacity, Digging Force Calculations And Static Force Analysis Of Mini Hydraulic Backhoe Excavator, Machine Design – The Journal Of Faculty Of Technical Sciences, Vol.4, No.1, 2012

4] Bhaveshkumar P. Patel And J. M. Prajapati, Structural Optimization Of Mini Hydraulic Backhoe Excavator Attachment Using Fea Approach, International Journal Of Mechanical Engineering And Robotics Research, Submitted, 2013. PP 43-56

5] Bhaveshkumar P. Patel And J. M. Prajapati, 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, Pp 505 – 511.

6] 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, Pp 894- 901.

7] Rahul Mishra And Vaibhav Dewangan (2013), "Optimization Of Component Of Excavator Bucket",Ijsret, Vol. 2. No. 2, Pp. 076-078.

8] Dhruv N. Patel1 Finite Element Analysis of Excavator Arm, IJSRD - International Journal for Scientific Research & Development| Vol. 2, Issue 06, 2014 | ISSN (online): 2321-0613

9] E. Budny, M. Chłosta, W. Gutkowskiexperiment On A Stable Motion Of The Backhoe Excavator Bucket,

10]Manisha P. Tupkar, Design And Analysis Of An Excavator Bucket, International Journal Of Scientific Research Engineering & Technology (Ijsret), Issn 2278 – 0882 Volume 4, Issue 3, March 2015

