Study on the Analysis of Excavator Boom: A Review

Janmit Raj[1], Gaurav Saxena[2], Rishi Kumar[3], Kuldeep Kaushik[4]
[1]*Research Scholar, Department of Automobile Engineering, Rustam Ji Institute of Technology, Tekanpur, Gwalior (M. P.)- 475005, India*
[2]*Assistant Professor, Department of Automobile Engineering, Rustam Ji Institute of Technology, Tekanpur, Gwalior (M. P.)- 475005, India*
[3]*Visiting Faculty, Department of Mechanical Engineering, The M. S. University of Baroda, Vadodara (Gujarat) - 390002, India*
[4]*Assistant Professor, Department of Mechanical Engineering, Malwa Institute of Technology and Management, Gwalior (M.P.) - 475001, India*

Abstract

In view of the significant increase in the research activity on the excavator boom in last few years, the present article is an attempt to identify and highlight the various researches that are most relevant to excavator boom. A review of the reported studies in the field of FEA (Finite Element Analysis) consisting the design of the boom, structural analysis, fatigue analysis, modal analysis, shape optimization and CAD/CAE system integration with the required softwares for carrying out the analysis work with an emphasis on the publication in the last 13 years (2002-2015). This literature progressively discuss about the softwares, research methodology and the outcome of the discussed researches and is intended to give the readers a brief variety of the researches carried out on the excavator boom.

Keywords*: Excavator Boom, CAD/CAE, FEA.*

I. INTRODUCTION

Excavator is one of the most important machinery in engineering machinery. Excavators are widely used in construction, railway, water conservancy, mining and other industries [7], [24]. There are many variations in hydraulic excavators. They may be either crawler or rubber-tire-carrier-mounted, and there are many different operating attachments. Hydraulic excavator consists of three main parts: undercarriage, upper structure and the working device, the upper structure rotates on the undercarriage [16].

The working device of a hydraulic excavator mainly consists of boom, arm, bucket, boom cylinder, arm cylinder, and bucket cylinder [11]. The boom and boom cylinder composed the boom linkage mechanism, the arm and arm cylinder composed the arm linkage mechanism, and the bucket and bucket's cylinder and linkage mechanism composed bucket mechanism, all of above linkage mechanism connected with each other by means of pin-hinged [14]. Excavator digs, elevates, swings and dumps material by the action of its mechanism of working devices [12]. The working devices of excavator works under complex load in serve working conditions [16]. So the strength, reliability and durability of excavator working device are directly related to the excavator's working performance and efficiency, therefore the strength and reliability research of the working device has a very important significance [20]. The schematic view of the hydraulic excavator is shown below in Fig.1:

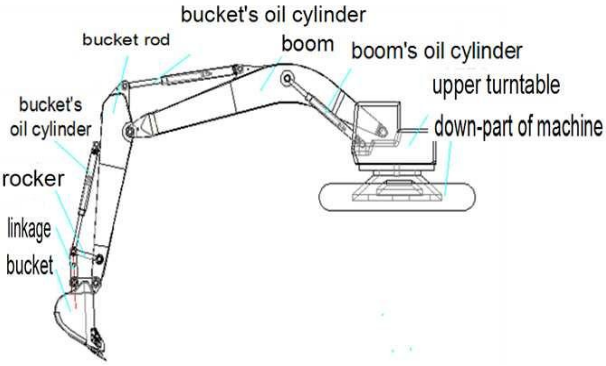


Fig.1: Schematic of Hydraulic Excavator

II. BRIEF DESCRIPTION TO EXCAVATOR BOOM

The excavator boom is one of the key components of an excavator, and also main load bearing part. The boom of an excavator is a box shaped rectangular section structure [4]. The boom structure is mainly made of thin steel plate welded together which forms a complex geometry [11]. Excavator boom under the main frame is consist of four primary plates viz., cover plate, under cover plate, left plate and right plate excluding other supporting plates such as front fork plate, Strengthening Plate, Boom cylinder connecting seat plate, Arm cylinder connecting seat plate[6], [8], [17]. The shape and size of all the plates vary from manufacturer to manufacturer. Stiffeners and reinforcement plates are welded inside to increase the strength and to prevent the excessive deformation which leads to failure of the boom [24]-[25]. The root of the boom is hinged with a pin at the front end of the middle platform. Boom cylinder executes the lifting movement to the boom [10]. The performance of the boom in an excavation work, determines the overall-performance, safety and reliability of an excavator.

III. PREVIOUS RESEARCHES

A. Introduction

The boom of an excavator is complex structure to examine [19]. During operation, the boom of an excavator is stressed throughout their life. In the process of operation, boom is subjected to tensile, compressive, torsional and bending loads due to lifting and digging process whereas it is subjected to shock load due to excessive vibrations in the operation of the hydraulic excavator [12].so it is necessary to study about the stress-strain distribution, fatigue and vibration characteristics of the boom under above mentioned conditions [26].

Most of the researches on excavator are done in past few years with the development of finite element method (FEM). Several researchers have conducted a variety of different analysis on excavator boom using different FEA tools. The primary aim of the article is to provide a comprehensive literature review of researches conducted on excavator boom which emphasis on design, structural analysis, fatigue life prediction, modal analysis and structural optimization through CAD/CAE system integration of excavator’s boom with the focus on analysis technique employed and the outcome of the study.

B. Work Carried Out on the Design of Boom

For selection of an excavator the weight is an important parameter. Mobility, operation flexibility and fuel consumption can be reduced by using light weight design approach while keeping the same required strength. Some researcher attempted to reduce the weight of the boom. Sun Jianghong and Pan Shangfeng [1] carried out basic research on the lightweight design of the excavator boom, but they didn’t give any specific lightweight design scheme. In further work the Luigi Solazzi [2] was the first who proposed the use of aluminium alloy in place of steel alloy for excavator boom and arm. Assuming the load conditions conducted finite element analysis at five operating conditions. The weight of final geometry was reduced by 50% while price increased about € 2.500–3.000. The Table 1 below gives the comparison of the weight reduction work carried out by him.

Table I : Comparison of the Total Weight of the Arm

Elements	**Weight (kg) Original Geometry**	**Weight (kg) Optimized Geometry**	**Difference in percentage compared to the original Geometry (%)**
Elements of the arm	2050	1080	-47.3%
Pins	335	195	-41.8%
Bucket (Filled)	2150	2785	+29.5%

An exact year later Srđan M. Bošnjak [3], expressed critics on the research work of above mentioned writer with the shortcomings in its design of the aluminium arm and boom of an excavator. The author suggested him to carry a keen review on his work, towards the improvement in its mathematical modeling, design and weight optimization of the arm and boom of an excavator. The next major study was done by Gedion Habtay Gebremicheal et al. [4] investigated a new design of boom based on semi elliptical section and conducted static stress analysis at different working condition of excavator boom using FEA software ANSYS V.12. The boom model was established in Pro-E Wildfire 4.0 software. The static forces in different working condition were calculated by using UPAS automatic calculation module of VB program. The study concluded that the total weight is reduced by 5% and maximum stress is occurring at upper mid part of boom. The model of the boom prepared, is shown below in Fig.2:

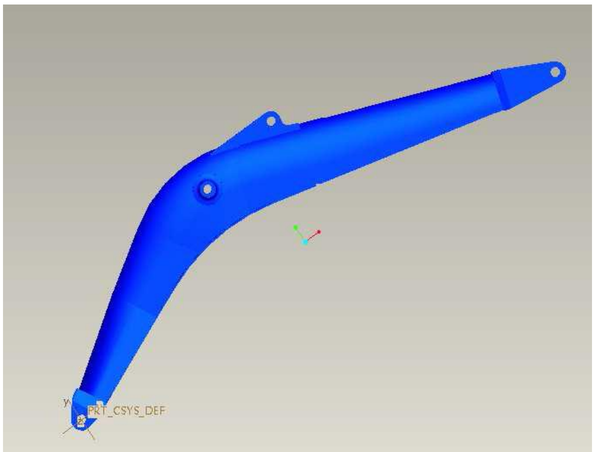
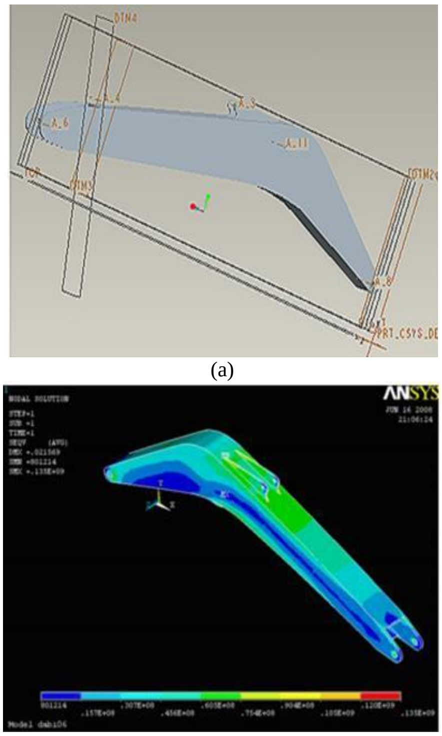


Fig.2: Model of Semi-Elliptical Boom

C. Work Carried Out on the Structural Analysis of Boom

The boom is subjected to complex stress due to in operating conditions [6]. In the process of operation, a variety of force act on the boom [14]. Therefore it is necessary to make the strength calculation of boom under loaded condition [13]. Structural analysis determines the area under high stress and the most dangerous working condition of each structural member [8]. Structural analysis in important in design, provides a basis for structural optimization and improve service life [15].

The researcher Yu Shuo et al. [5] showed static analysis of actual (without internal ribs) and improved boom (with internal ribs welded on the sides) model using finite element method. The study revealed that high stress concentration is occurring at arm cylinder hinge point and exceeding the limit of yield stress in the boom cylinder hinge point in actual model. While in the improved boom model there is significant reduction in maximum stress, only about 60% of the original. In the further work Li Dan et al. and Zhu Chun-Hua et al. [6]-[7] conducted the FEA of excavator boom at the dangerous operating state of boom using ANSYS software, including the calculation model for determining the load at each hinge point. Pro/E software was used to build the finite element model of boom. The results depicted that the boom in this condition does not have a large displacement, the maximum stress is less than the allowable stress of the material and deformation is in the elastic range. The model Geometry and the analysis window of the boom is shown below in Fig.3:



(a)

(b)

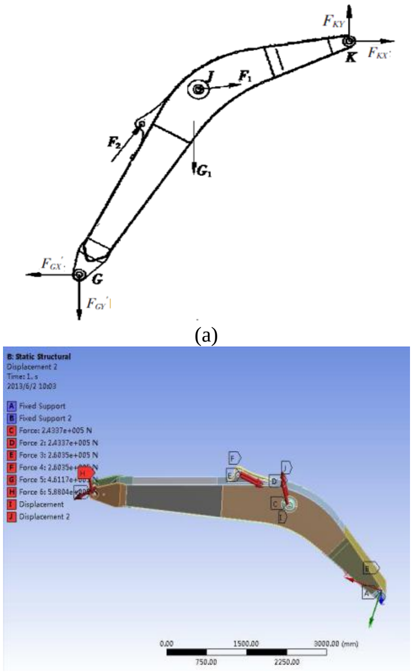
Fig.3: (a) Geometric Model of the Boom in Pro/E, (b) Stress Contour of the Boom in ANSYS

In another study Qinglu Shi et al. [8] tested the excavator boom strength in three working conditions and compared them in finite element software. The maximum tangential digging force of bucket teeth in these working conditions were calculated by EXCA 10.0 software. The Load and Moment at all hinge joint were calculated for three working condition from the result obtained from EXCA 10.0 software. The analysis concluded that under offset and transverse load there were high bending stresses which can cause damage to boom structure. An advance step in this field was done by researchers Zhou Hong Bing et al. [9] by developing the programing code using MATLAB for calculating the forces acting on each hinge joint in multi-work conditions. And performed static structural analysis using finite element software ANSYS. In analysis D-H coordinate transformation method was applied to obtain boundary load for finite element analysis. The study revealed that maximum equivalent stress is lower than the yield stress of material and eccentric loading condition is found to be relatively dangerous. Lin Mingzhi and Xing Shuxin [10], a year later, they compared two boom models. A geometric model of both, one with thick steel plate and rear wing web while the other model having thin steel plates and baffles welded inside was prepared using PRO/E and the same is analysed in the ANSYS. The obtained result shows that second model has much lower stress distribution than the first one. Further the team of researchers Huang Meimei et al. [11] investigated boom in two working conditions, imposing hydraulic cylinder force as load in ANSYS software tool package, UG software was used to build the 3-D model of boom. The study concluded that the Primary peak value of stress concentration is at shaft housing and the secondary peak value of stress concentration is at area around hinge point of boom cylinder. In addition to this, heat affected zone during welding of plates is more prone to damage. In extensive research Qing Xin Ding et al. [12] depicts the work on statics simulation and strength calculation of excavator boom in all poses. For calculating all poses and hinged forces acting in boom under all poses parametric model of boom was built using the MSC Adams software. Further finite element analysis was carried out in ANSYS software tool package for calculating maximum equivalent force in all states. In the analysis force in longitudinal plane is considered. The study concluded that the strength of boom is within required limit under all poses. The table 2 below depicts the max. equivalent stress distribution in the boom under different condition:

Table Ii : Max. Equivalent Stress of the Boom Under Partial Positions and Orientations

Pose	1	2	3	4	5	6	7	8	9	10
Equivalent stress [MPa]	202.98	189.953	165.917	138.817	122.246	101.193	78.397	126.89	172.684	202.548

The Li Zhi-Jie et al. [13], in his work, gave a brief study on how the tool of finite element can be used? to get quick and fast results for estimating the stresses of the excavator boom. In this research excavator using bucket digging operation mode is considered for load calculation and given mathematical equation for load calculation at each hinge point. The boom model were built in PRO/E software, Meshing and analyzing is done in ANSYS Workbench 13.0 software package. The result shows that the maximum equivalent stress is in safe limit and the maximum deformation is within the elastic limit. A year later Zhang Ju-Gui, et al. [14] determined that the static strength of excavator boom using ANSYS Workbench. In this article the geometric model was established in Pro/E, The torque balance equation was applied for load calculation; three typical working conditions were analyzed by imposing constraint on the basis on actual operating condition. The results of finite element analysis showed that the static intensity of the boom is preferable. The maximum stress mainly occurred in the hinge point connecting the boom cylinder with the boom and the hinge point connecting the boom to the base. The Fig.4 below shows the stress, constraint & load condition of the boom prepared & analyzed by him.



(a)

(b)

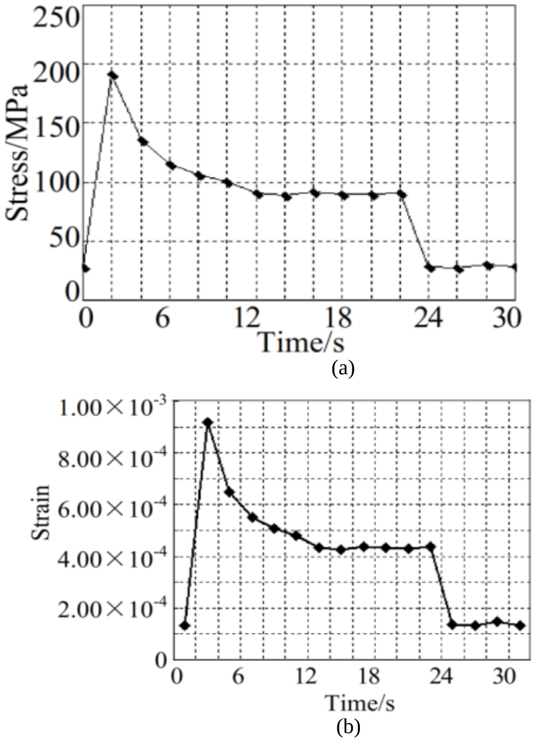
Fig.4: (A) Stresses Condition of the Boom, (B) Constraint and Load Condition of the Boom

In the more recent study by Shengbin Wu and Xiaobao Liu [15] suggested four improvement for reducing stress concentration along with the improvement in high stress distribution by comparing main and improved model of boom, Through finite element analysis of a hydraulic excavator's boom in ANSYS Workbench. The force at all hinge point of boom was calculated by applying moment balance and force balance principle. The analysis shows that high stress is appearing in the top and bottom cover plates which near the rear seat and the junction of boom cylinder's connecting seat and the bottom plate. The practices demonstrate that the proposed improvements can reduce the destruction of hydraulic excavator's boom. Afterward Shiva Soni et al. [16] carried out the static analysis of excavator boom. The modeling and simulation of boom was done in Autodesk INVENTER software which was interfaced with the softwares like Altair Hyper-mesh for meshing and ANSYS FEA tool for the analysis. The study reported that the stress is under acceptable limit.

D. Work Carried out on the Fatigue Analysis of Boom

The boom of hydraulic excavator is main stressed component during operating states. According to market feedback a local fatigue crack were developed in boom after a period of use. Fatigue is the most common mode of failure in excavator boom [19]. During excavation, cyclic loading results in generation of cyclic stress which causes fatigue failure [20]. Therefore, it is necessary to analyze the fatigue of excavator boom to prevent breakdown and to provide guidance for servicing during repair and maintenance. [17]

In important study Du Lin [17] examined the fatigue reliability of dangerous point of excavator boom using the RBF neural network response surface method in MATLAB. The author used Manson-Coffin formula for calculating fatigue life of each cycle based on rain flow and strain history. After each cycle, the fatigue damage was calculated and the corresponding fatigue life was calculated by Miner cumulative damage theory. The fatigue analysis is based on the results obtained from stress and strain analysis in ANSYS software. The research revealed that boom structure has satisfactory fatigue reliability, which can be calculated by the curves in the Fig.5 given below:

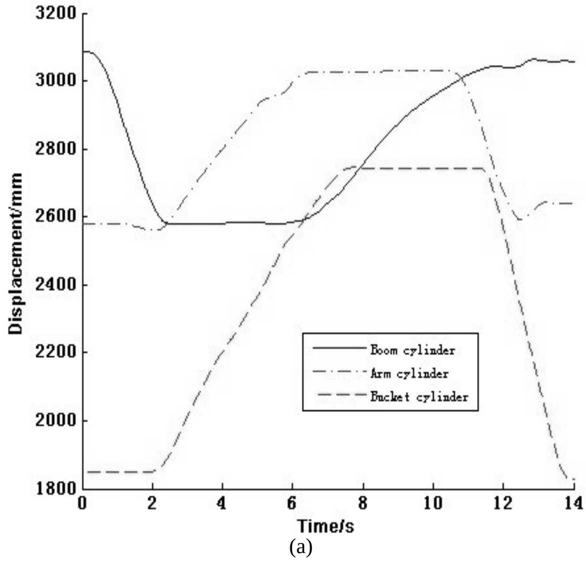


(a)

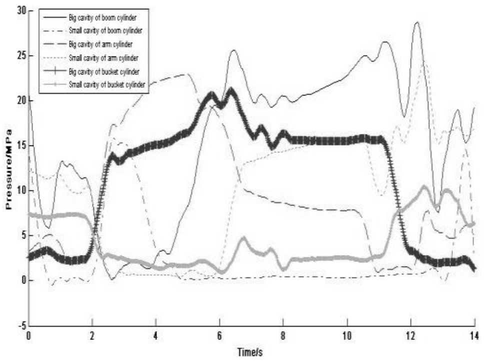
(b)

Fig.5: (a) Stress Time History, (b) Strain History

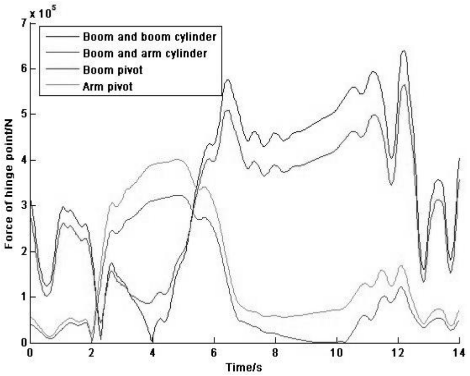
The major study in the field of fatigue was done by the Zhang Weiguo et al. [18], which performed the experiment and dynamic simulation on 6 Ton compact excavator for fatigue analysis of its working device. The pressure and displacement sensors were used to record the displacement of each hydraulic cylinder for getting the displacement curve. These curves were used to perform dynamic simulation. The 3-D geometric model was built in solid modeling software PRO/E and the dynamic simulation of model was carried out in ADAMS software for obtaining the working loads on hinge points. The fatigue analysis of boom and arm was carried forward in the software MSC Fatigue. The study concluded that the area of boom and arm where stress concentration is high, have lowest fatigue life. The Fig.6 shown below shows the displacement, pressure and force curves.



(a)



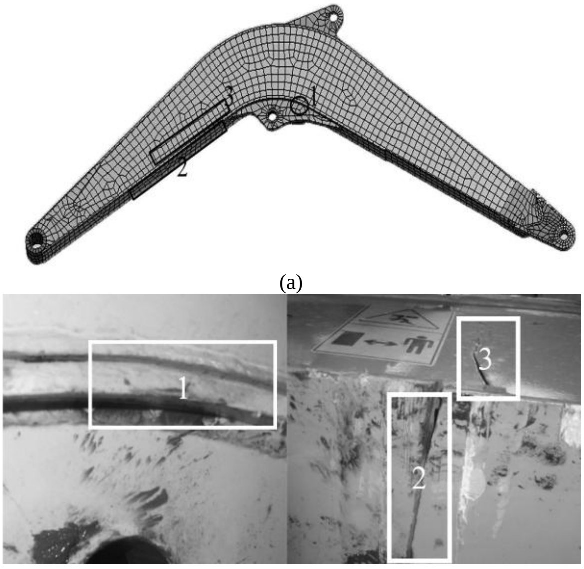
(b)



(c)

Fig.6: (a) Displacements of Cylinders, (b) Pressures of Cylinders Cavities and (c) Force of Boom Hinge Point

Aiming to shortcoming in other method of predicting fatigue in all operating states Qing Xin Ding et al. [19] presented EFLB-ALL method for predicting the fatigue life of excavator boom under all operating states, loading conditions applied closer to actual working situations. In the research the stress-time history were obtained by a simulation method with virtual prototype technology, parameterized finite element model were established and a virtual program was created to calculate maximum principle stress at each point in every state. This method calculates the minimum life and most dangerous operating states of boom. Based on analysis result author identified 3 danger zones and these zones found identical to actual destruction crack, which was depicted in Fig.7 below:



(a)

(b)

Fig.7: (A) The Three Risk Areas of the Boom and (B) Schematic Diagram of Actual Crack Region of the Boom

Further the Tao Jiang and Xiongbin Liu [20] carried out further study and validated its work with the work of Zhang Weiguo et al. [18] and predicted the fatigue life of a 20ton-hydraulic excavator boom in actual loading conditions by performing analysis in

MSC Fatigue software with the integration with the programming software package MATLAB. The result of analysis shown that the front end of the support plate at the hinge point of arm cylinder and boom is prone to fatigue damage because of higher stress concentration and therefore this part of boom have shortest fatigue life of 16.6 years which meet the requirement of the excavator fatigue life. Now in the most recent work Xie Yu-Ting et al. [21] recently modeled excavator's boom in solid-works for analyzing stress- strain and fatigue life using FE-SAFE software, meshing was done using Abacus software. The applied constraints were consistent to actual load conditions and a load cycle of 43seconds was applied to the finite element model for analysis. In the analysis strain test technology was used to measure load time history at each point. After analyzing the test result shown that the maximum stress appearing in the boom was less than the yield strength of material and the fatigue life were considerably high.

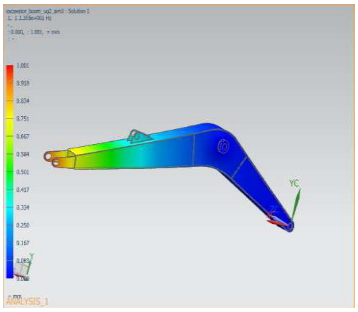
E. Work Carried out on the Modal Analysis of Boom

During excavation, excavator experiences immense mechanical shock and vibration. The mechanical shock and vibration are the main cause of structural damage [25]. Therefore in order to reduce vibration, shock and to improve service life is it necessary to conduct modal analysis. This type of analysis is used to find vibration characteristic of the structure. The main task of this analysis is to study the free vibration of undamped system, especially to determine the natural frequency of the structure. This type can provide a basis for improving the stability and reliability of the hydraulic excavator [24]. The study conducted by G. Wszołek (2004) [22] is considered as one of the major attempt in modal analysis of excavator. He wrote the GRAFSIM program by using the SIMULINK simulation module from MATLAB software, which analyze the vibrational characteristics of the excavator's virtual prototype under applied excitation in three conditions. While Du Wenjing [23] put forward an integrated finite element analysis on the working equipment and introduced the FEM for the contact analysis which was then successfully analyzed. Based on the FEA software ANSYS, the 3D models of the main components (boom and bucket rod) of the working equipment for a certain type of hydraulic excavator were, pre-stress modal, analyzed, for obtaining the inherent frequency and the modal vibration mode of boom and bucket rod under four different kinds of working conditions. Compared to the excavator's working frequency, the natural frequencies and the vibration modes are analyzed, which provided important modal parameters to the corresponding analysis for excavator's boom and bucket rod as well as a basis for the optimization design for the analysis on total vibration and structural dynamic characteristics. Later Jing Zhang et al. [24] presented the modal analysis of excavator boom imposing constraints to actual working conditions at five modes which is shown in Table 3 below. The first four orders are of bending vibration and the fifth of torsional deformation, using ALGOR software, 3-D model was built in Pro/E software package. After analyzing the model result shown that the first order and the fourth order show larger displacement. In these two frequencies maximum displacement points are prone to damage.

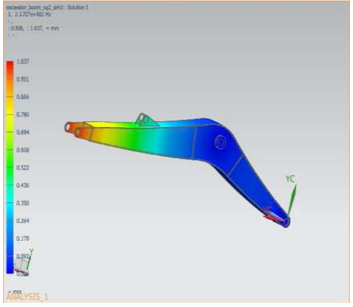
Table III : Frequency and Vibration Mode of Each Order

Model	Frequency(Hz)	Vibration Mode
1	17.05	Bending vibration in XZ plane
2	33.52	Bending vibration in XY plane
3	92.53	Second order bending vibration in XZ plane
4	149.11	Second order bending vibration in XY plane
5	226.15	Torsion

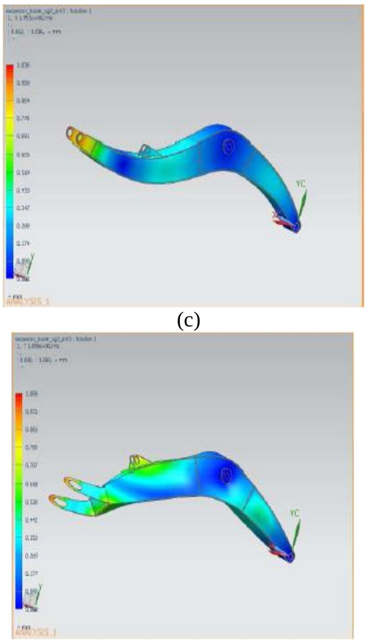
In the further research Minglong Wang and Aimin Ji [25] extracted the natural frequency of excavator boom at 10 states in free state and constraint state using finite element software ANSYS. The simulation model of excavator boom was built using UG software. The article reported that every mode of constraint state were lower than that of Free State. The boom structure is safe from resonant condition. The contour of four example modes of vibration of modal analysis is shown below in Fig.8.



(a)



(b)


(c)

(d)

Fig.8: Contours of Modal Analysis [4]. (a) 1st Modal, (b) 2nd Modal (c), 6th Modal and (d) 7th Modal

In the more recent study Caiyuan Xiao and Zhang Guiju [26] analyzed the geometric model of boom and bucket rod in FEA software ANSYS to determine the natural frequency and main mode of vibration by evaluating the first 6th order modal. The result presented in the work showed that the maximum deformation of boom occurring at the middle position of the hinge point between the front plate and the boom cylinder and also at area near hinge point between the rear plate and the boom cylinder.

F. *Work Carried out on the Optimization of the Boom Structure*

As from the analysis point of view it is well understood that the optimization of any structure is a very important work for a researcher to perform in its field. . So in that context Chen Jian et al. [27] derived mathematical model taking thickness of plates as variable for optimization of excavator boom considering the influence of the stiffness sensitivity and stress sensitivity in optimization process. In the research the design model iterated 10timesfor obtaining optimal solution. In the optimized design of model in same condition the Von Mises stress reduced by 12.9% and local rigidity is increased by 29.8%. On the other hand Haiyan Hua et al. [28] presented cultural algorithm method for structural optimization of excavator boom. The aim of this research was to minimize the volume of boom, more uniform stress distribution under four different working conditions. In this study OCVRAC KB method was used for constraints expression and handling.

Later in the further research Ding Qingxin et al. [29] introduced Latin Hypercube method implementing pareto method for identifying optimization variables for optimal design of boom structure. For optimization multi-island genetic algorithm (MIGA) and sequential quadratic programming technique were applied and an optimized design with 15% mass reduction was obtained. In order to achieve the automatic exchange of data between various software the author prepared an integrated platform of software such as ANSYS, Hyper-Mesh, UG and ISIGHT. Aiming to shortcoming in the exiting genetic algorithms for structural optimization of boom structure Hua Haiyan and Lin Shuwen [30] constructed a new mechanism for excavator boom structural optimization called KEGA (Knowledge evolution Genetic Algorithm) which is shown below in Fig.9. For optimization this method uses new knowledge-based selection, crossover and mutation operator to integrate the optimal process knowledge and domain culture. In this research author used eight kinds of testing algorithms with different genetic operator to conduct structural optimization. Author concluded that KEGA mechanism effectively improved the optimal searching. In the most recent work Xiao Cai-Yuan et al. [31] simulated excavator boom model for calculating stress –strain and weight optimization of boom in OPT module of ANSYS software under two typical working condition. The 3-D model of boom was created in design software Pro/E. After optimization author concluded that the boom structure is fail safe and the total mass has been reduced by 9.7%.

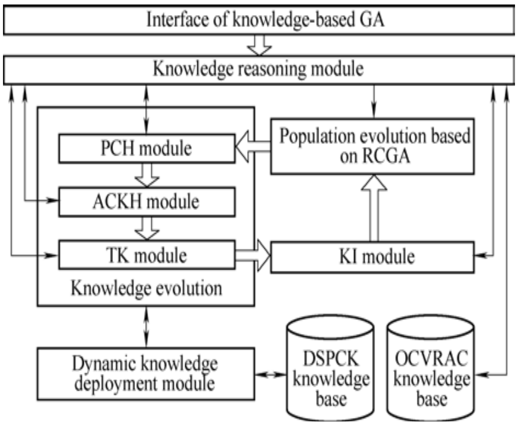


Fig.9: Basic Framework of KEGA Mechanism

G. *Work Carried out on the CAD/ CAE Technology Integration*

The modeling and analysis of excavator working devices is complex and time consuming process [33]. The complexity can be reduced to certain extent by using CAD/CAE Technology Integration [34]. CAD/CAE technology includes customizing and modifying parameter values of geometric modeling and analysis of working devices of excavator. This integrated platform will speed up modeling and analysis process [32], [35]-[36].

The researcher Lirong Wan and Zaichang Wang [32] integrated CAD/CAE technology in excavator research by presenting conjoint simulation, Modeling the excavator working devices in Pro/E software, simulating model in ADAMS software to get the excavating trajectory envelope and carried out strength analysis and optimization of boom structure in ANSYS software. The Research indicated that the stresses are in safe working limit and there is a scope of plate thickness optimization. In the next work Zhu Kun and Ji Ai- Min [33] demonstrated parametric design of excavator boom. In the research, using secondary development tools of UG and C++ language the control program and user interface dialog were created. To access database and to modify parameters of geometric model the dialogue frame of C-Data Choice and C-Insert Dialog were established. The development of this system led to simplify the modeling and modifying parameter of the excavator boom. Later Shen Zhen-Hui and Yang Shuan- Qiang in their work [34] they integrated VC++ program and ANSYS, Pro/Engineer environment to achieve automated structural finite element analysis of excavator boom. The study is based on the Pro/TOOLKIT, APDL commands and the "Batch" operation mode of ANSYS. The study further extended by Kun Zhu et al. and AIMIN JI et al. [35], [36] for CAD/CAE technology integration in the design of the excavator boom. This integration technology was based on three steps. Firstly, integration of design database with CAD/CAE for exchanging and transmitting model between CAD and CAE systems. Second was the establishment of boom model based on parametric method. Thirdly was the development of secondary codes of ANSYS Workbench and customization of analysis platform for boom. The ideal model will be get by modifying parameters according to result. In the Further study author summarized the defects in the process of CAD/CAE integration, the integrated design and analysis system including parametric boom design, database design, detailed feature simplification of boom, mid-surface extraction and structure analysis is also studied.

IV. CONCLUSION

This work has provided a comprehensive literature review of existing various research work carried out in terms of design, stress, fatigue, vibration, optimization analysis and CAD/CAE integration of excavator boom. An effort has been made to comprise all the important contributions to this area and highlighting the most pertinent literature available for investigating the excavator boom. The concluding remarks and future work from the current literature survey are as follows:-

1. From the review of available literature on excavator boom it is apparent that nearly all the research conducted has been purely simulation based on finite element method.
2. Most of the FEA of excavator boom was done in various operating states, simulating actual working conditions in software. The studies shown that mostly higher stress concentration occur at bottom plate of the boom near boom cylinder connecting seat. The forces at each hinge point were calculated mathematically.
3. The FEA is a useful tool since they provide accurate results to access strength and fatigue life of the excavator boom. The same can also be applied for shape optimization of boom.

ACKNOWLEDGMENT

The review presented in this work is by no means complete but it gives a comprehensive representation of different finite element techniques applied to the analysis of the excavator boom. The author wishes to apologize for the unintentional exclusions of missing references and would appreciate receiving comments and pointers to other relevant literature for a future update.

REFERENCES

[1] Sun Jianghong and Pan Shangfeng, “Design Integration and Lightweight Analysis of hydraulic Excavator based on Pro/ Engineer.” CAD/CAM and Manufacturing Information, Vol. 10, pp. 56-59, 2009.

[2] Luigi Solazzi, “Design of Aluminium Boom and Arm for an Excavator.” Journal of Terramechanics, Vol. 47, pp. 201–207, 2010.

[3] Srđan M. Bošnjak, “Comments on Design of Aluminium Boom and Arm for an Excavator.” Journal of Terramechanics, Vol. 48, pp. 459–462, 2011.

[4] Gadion Habtay Gabremicheal, Jin Chen, Muhammed Farooq Zaman and Xiaoping Pang, “Design and Finite Element Analysis of Hydraulic Excavator Boom.” Advanced Materials research, Vol. 538-541, pp. 568-571, 2012.

[5] Yu Shuo, Zhan Jianzong and Kong Zhe, “Finite Element Analysis on Hydraulic Excavator's Boom.” Construction Machinery and Equipment, Vol. 33 (11), pp. 8-10, 2002.

[6] Li Dan, Zhou Zhi-Hong and Liu Rui-Hua, “Finite Element Analysis Based on ANSYS for Boom of Hydraulic Excavator.” Construction Machinery, Issue 17, pp. 74-78, 2008.

[7] Zhu Chun-Hua, Guo Xiu-Yun and Wang Shao-Lei, “Finite Element Analysis of Working Equipment of Excavator Based on Pro /E and ANSYS.” Coal Mine Machinery, Vol. 30 (4), pp. 73-75, 2009.

[8] Shi Qinglu, Zhang Fusheng and Lian Jin Yi, “Comparison Analysis for Boom Strength of Excavators.” Construction Machinery, Vol. 40, pp. 40-43, 2009.

[9] Zhou Hong-Bing, Wang Hui-Ke**,** Guo Xin-Hua and Wang Shi-Huai, “Static Strength Analysis of Working Device of Hydraulic Excavator Based on MATLAB and ANSYS.” Journal of Guangxi University (Natural Science Edition), Vol. 34 (06), pp.774-779, 2009.

[10] Lin Mingzhi and Xing Shuxin, “Finite Element Analysis to Boom of a Hydraulic Excavator.” Construction Machinery and Equipment, Vol. 41, pp. 43-45, 2010.

[11] Huang Meimei, Zhao Tingting and Zou Huaqiang, “Finite Element Analysis of Boom of Hydraulic Excavator.” Modern manufacturing technology and equipment, Issue 01, pp. 19-20, 2011.

[12] Qing Xin Ding, Kun Liang Hui, Jian Wen Chen, Juan Chen, Ying Cheng Tian and Jia Yu, "All-Pose Statics Simulation and Strength Calculation of the Excavator Boom." Advanced Materials Research, Vol. 199-200, pp. 232-238, 2011.

[13] LI Zhi-Jie, Wen-Quan Ming and Shi Guang-Lin, "Finite Element Analysis of Hydraulic Excavators based On Workbench." Journal of Guangxi University of Technology, Vol. 23 (03), pp. 32-35, 2012.

[14] Gui Ju Zhang, Cai Yuan-Xiao, Qing-Tan and You Yu-Mo, "Finite Element Analysis of Working Device for Hydraulic Excavator." Journal of Chemical and Pharmaceutical Research, Vol. 5 (12), pp. 123-128, 2013.

[15] Shengbin Wu and Xiaobao Liu, "Improvements for a Hydraulic Excavator's Boom." Applied Mechanics and Materials, Vol. 490-491, 510-513, 2014.

[16] Shiva Soni, L S. Ahirwar., Reliance Jain and Ashish Kumar Shrivastava, "Simulation and Static Analysis on Improved Design of Excavator Boom." International Journal of Emerging Technology and Advanced Engineering, Vol. 4 (3), pp. 49-55, 2014.

[17] Du Lin, "Fatigue Reliability of the Excavator's Boom." Machinery, Vol. 37 (05), pp. 25-27, 2010.

[18] Zhang Weiguo, Quan Long, Cheng Hang and Yang Jing, "Fatigue Analysis on Working Device of Excavator Driven by Practical Load." Transactions of the Chinese Society for Agricultural Machinery, Vol. 42 (05), pp. 35-38, 2011.

[19] Qing Xin Ding, Ying Cheng Tian, Juan Chen, Jian Wen Chen, Kun Liang Hui and Hai Wen Yin, "Estimation of Fatigue Life for Hydraulic Excavator Boom under All Operating States." Advanced Materials Research, Vol. 199-200, pp. 463-469, 2011.

[20] Tao Jiang and Xiongbin Liu, "Fatigue Research on Excavator Boom Driven By Real Loads." Applied Mechanics and Materials, Vol. 488-489, pp. 1057-1060, 2014.

[21] Xie Yu-Ting, Zhang Hua, Yan Wei and Chen Wei-Wei, "Life Analysis and Prediction of Hydraulic Excavator Boom." Coal Mine Machinery, Vol. 36 (2), pp. 143-146, 2015.

[22] G. Wszołek, "Vibration Analysis of the Excavator Model in GRAFSIM Program on the Basis of a Block Diagram Method." Journal of Materials Processing Technology, Vol. 157–158, pp. 268–273, 2004.

[23] Du Wenjing, "Research on Key Technology of Hydraulic Excavator Working Device Design." Changchun: Jilin University, 2007.

[24] Jing Zhang, Jinge Wang and Yuanwen Wu, "The Modal Analysis of Boom Based on PZ-30 Excavator." Fourth International Workshop on Advanced Computational Intelligence, October 19-21, pp. 189-192, 2011.

[25] Minglong Wang and Aimin Ji, "Modal Analysis of Excavator Boom Based on UG and ANSYS." Applied Mechanics and Materials, Vol. 644-650, pp. 255-258, 2014.

[26] Caiyuan Xiao and Zhang Guiju, "Modal Analysis on Working Equipment of Hydraulic Excavator." The Open Mechanical Engineering journal, Vol. 9, pp. 173-180, 2015.

[27] Chen Jian, Zhou Xin, Liu Xin, Ji Qing Hui and Yang Wei, "Optimal Design of Boom Structure for Hydraulic Excavators." Construction Machinery and Equipment, Vol. 39 (07), pp. 19-22, 2008.

[28] Haiyan Hua, Shuwen Lin and Zhenhui Shen, "A New Method of the Constraints Expression and Handling for Excavator Boom Structural Optimization." Advanced Materials Research, Vol. 479-481, pp. 1851-1856, 2012.

[29] Ding Qingxin, Cui Can, Ding Wei, Hui Kunliang and Ye Zuogui, "Optimization Design of Excavator Boom Based on Integrated Software Platform." Mining & Processing Equipment, Vol. 42 (01), pp. 25-29, 2014.

[30] Hua Haiyan and Lin Shuwen, "New Knowledge-based Genetic Algorithm for Excavator Boom Structural Optimization." Chinese Journal of Mechanical Engineering, Vol. 27 (02), pp. 392-401, 2014.

[31] Xiao Cai-Yuan, Zhang Gui-Ju, Mo You-Yu, "Performance Optimization of Hydraulic Excavator Boom Based on Finite Element Technique." Journal of Shaoyang University (Natural Science Edition), Vol. 12 (01), pp. 20-27, 2015.

[32] Lirong Wan and Zaichang Wang, "Application of CAD/CAE Technology in Hydraulic Excavator Working Equipment Analysis." Advanced Materials Research, Vol. 308-310, pp. 2302-2307, 2011.

[33] Zhu Kun, JI Ai-Min, "Research on Parameteric Design of Hydraulic Excavator's Boom." Journal of Mechanical & Electrical Engineering, Vol. 29 (07) 785-789, 2012.

[34] Hui-Zhen Shen and Yang Shuan-Qiang, "Automated Finite Element Analysis Method for Excavator Boom." Value Engineering, Vol.30, pp. 34-36, 2013.

[35] Kun Zhu, Xu Yin and Aimin Ji, "CAD/CAE Integration System of the Excavator Boom." Applied Mechanics and Materials, Vol. 401-403, pp. 81-84, 2013.

[36] Aimin Ji, Kun Zhu, Xinlei Huang and Xu Yin, "Integration Design and Analysis of Excavator Boom Based on CAD/CAE." Journal of Theoretical and Applied Information Technology, Vol. 49 (01), pp. 138-143, 2013.