Semiconductor Optoelectronics, *Vol. 41 No. 10 (2022), 477-484*
https://bdtgd.cn/

Semiconductor Optoelectronics
ISSN: 1001-5868

Flexible Dynamic Analysis Of Backhoe Excavator Working Device

Alok Sharma
Research Scholar, Chhattisgarh Swami Vivekananda Technical University, Bhilai, India.

Dr. Deepak Sharma
Professor, CSVTU, Chhattisgarh Swami Vivekananda Technical University, Bhilai, India.

Abstract: *An excavator is an off-road vehicle or machine used for several types of construction activities. It has an attachment called a backhoe that is widely used for digging. While digging the earth's massive force works on bucket tips and may bring deformation in it. In this paper an attempt is made to quantify the value of the variables obtained while the all-attached backhoe is considered as a multibody, given in rotation to each joint and simulated in Ansys. The results obtained show that the deformation obtained is small at low rotation and large at high rotation rate. Comparison of the deformation, joint forces and velocity at each set of rotation has been tabulated in x, y and Z directions as well as the overall velocity, total joint force and deformation has also been calculated. This paper provides the base to study and design a multibody system which may undergo deformation so that the end effect may be obtained precisely as required in precision tools and machinery.*
Keywords- Flexible, Dynamics, Backhoe, Excavator, Finite Element Analysis.

1. INTRODUCTION

Backhoe excavators are a very well-liked tool that is frequently used for digging holes, creating foundations, constructing highways, forestry work, gardening, urban construction projects, river dredging, and other hazardous environments. Backhoe excavators are primarily used to make excavations below the ground's natural surface, on which the machine is placed. The backhoe buckets scoop up the ground and move it back toward the machine's body rather than pushing it ahead. The boom and the dipper (arm, crowd, or stick) are connected at one end by the dipper cylinder, which moves the dipper in and out. The end with the loader arms may support a full-width bucket or attachment, whereas the end with the boom and arm combination can swing in a half-circle to allow for digging or adjusting attachments. Multi body system's Design and analysis is a not easy task which requires the acquaintance of computer aided design (CAD), machine design, structural dynamics and Finite element analysis(FEA). Wheel loader vehicle is a category of earth moving machine which lies under the multi body system. This research is based on the reverse engineering concepts, where the wheel loader is designed and analysed to accuracy. Firstly, the dimensions of the wheel loader are measured through vernier calliper and full scale ruler which are directly used for 3D model generation of the wheel loader. SolidWorks 3D CAD Software is used for model creation. Using the real time standards, a few design modifications have been done. Design of lift arms, lift arm cylinders, connecting rod and tilt/bucket cylinder are customized to realtime standards. Kinematics analysis of wheel loader has been performed in SolidWorks. the model has been exported from SolidWorks to MSCVisual Nastran Desktop 4D for kinetics and structural analysis. Structural dynamics Analysis is done through FEM. The findings of the stress study are within the yield stress range for the materials used in the design of the wheel loader. The design verification is done through real time with the factor of safety 1.2. Reverse engineering process is useful to analyze the existing model to improve the design.(N.M. Shah,2004). In this research an effort has been done to design and analyse the rotating bucket of the excavator with the stick and the bucket arm. It focuses on the joint design using the geared motor which provides angular rotation to bucket arm and studies the outcome of digging, bending stresses developed and torsional force on the joint which helps in study the motion of the bucket arm which shows that by finding various reaction forces, a rotary joint can be designed for the excavator arm. It provides the rotation to the arm and enhance the productivity. This is significant to calculate all the forces during designing process, material selection, motor power rating. The excavation could be performed in various position of the bucket(B. Rai, et al., 2013). There are unidentified sources of resistance given by the terrain to bucket teeth during the excavation operation.. The machine parts are negatively impacted by an excessive amount of these stresses, and they may fail during excavation operations. Thus, it is essential to provide a improved design of parts having maximum consistency, minimum weight and cost. Finite element analysis (FEA) of existing excavator arm is compared with optimized arm under given stresses and deflection. For Optimization, FEA approach is applied. Finite element based optimization of excavator arm is helpful in finding out the most appropriate design from which a best prototype is fabricated and tested. Number of iterations of excavator arm is performed out of which it is found that iteration 4 has an adequate proportion of material removed devoid of disturbing its strength and ultimately FEA. (A.S. Shaikhand B.M. Shinde, 2014). When considering the digging force generated by the actuators during in the excavation operation, the bucket in this design plays a crucial role. The evaluation process of bucket volume and the digging forces necessary to prepare the ground for light- and heavy-duty construction activity are the main topics of this article. This technique can be used for autonomous excavation task operation and gives the forecast of digging forces. The excavator mechanism's finite element analysis can be utilised to analyse strength and stress using the evaluated excavation forces as boundary conditions and loading conditions. Static force analysis of the tiny hydraulic excavator attachment is done analytically. This study aims to develop an excavator bucket with a smooth material flow and powerful digging forces. 1. The project's main goal was to analyse and optimise excavator buckets. In ANSYS 15.0 Workbench, the chassis model analysis was completed. An experimental validation was used to validate the findings and confirm the accuracy of the FEA and distortion results. The final thoughts based on the examination of the bucket model and bucket validation at ARAI-2 are presented here. To determine bucket distortion, the bucket model is examined under four different loading scenarios, and bucket deformation is contrasted with standard bucket. It is found that the stresses in the 1.8 cum design are lower than those in the 1.9 cum Current production bucket3 when they are investigated for 1/3 offset and for full offset. When we validated the bucket, we saw that it had greater strength and life than the prior bucket..(S. Lomate1 et al., 2016). The digging forces created by actuators must be stronger than the resistive forces produced by the surface to be excavated in order for CAD-CAE systems to provide a design solution. Productivity and fuel consumption are the two key elements taken into account while building an excavator arm. Because the excavator arm's current mechanism is frequently defective at the bucket end due to bending and torsion loads experienced during lifting and digging operations, respectively. Therefore, a new excavator arm mechanism is created, and a 3D model of the excavator arm linkage is created using the Pro-e software. Each excavator arm element is statically analysed using the ANSYS workbench software at both the previously calculated digging force and the current digging force. Additionally, the bucket volume is raised to make up for the production loss brought on by the drop in digging force. The excavator arm's design has been updated, and an examination of the design has also been completed. The analysis' findings demonstrate that the design is secure for the estimated diggingforce. Productivity and fuel consumption are significant considerations when building an excavator arm. Since lowering the digging force eliminates misalignment of the pin at the bucket end and cracking at the adapter end. However, a decrease in digging force has a direct impact on productivity. To make up for the productivity loss caused by the decrease in digging force, the bucket volume is raised. Additionally, fuel consumption is lower as a result of the decreased digging force. The results of the suggested model are then compared to those of the current model. (S.B. Bende and N.P. Awate,2013).

2. MATERIAL AND METHOD

Excavators are mainly used to dig under the natural ground where the machine is located. Excavator parts are prone to wear due to harsh working conditions. For better results, excavator mechanisms must function efficiently and effectively under unpredictable working conditions. Excavator failure references were used for analysis of current work. First, all relevant data about the machine are collected and aggregated, and based on the data collection his 3D CAD models of the machine components are generated in his 3D design software SolidWorks. This is further analyzed in the same ANSYS module. Rigorous and flexible analysis to know the behavior of the connections between machine components improves design optimization when it comes to withstanding the forces/stresses generated when performing a required activity with sufficient strength. Increase.

1.1 Finite Element Modeling and Analysis

As shown in Figure 1, the fulcrum of the excavator body and work equipment is the origin, the front of the work equipment is the X axis, and the top of the work equipment is the Y axis.

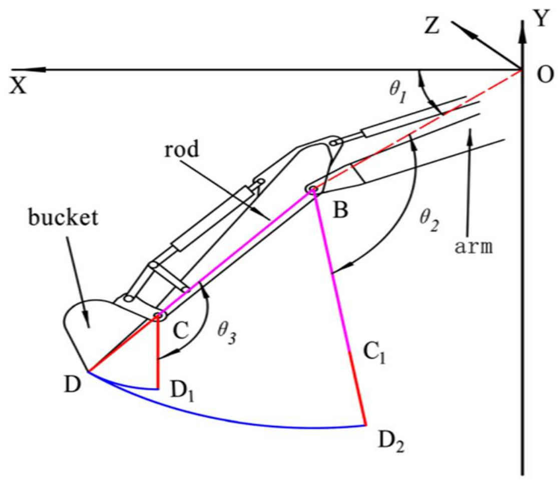

Figure 1. Trajectory of hydraulic excavator

1.2 Modeling of Backhoe Excavator

Modeling and simulation consist of geometry design, pre-processing, and analysis. Geometry modeling consists of drafting the backhoe geometry modelled in Solid works software, as shown in Figure 2. For this purpose, the excavator model of the excavator was actually on site he was measured by JCB-ECO Expert 3Dx and using the dimensions obtained in this way a composite CAD model was developed in Solid works. The model has been imported into ANSYS. The simulation is run at different joints with different sets of rotation angles. Pre-processing consists of defining a 3D model and meshing the model. Assume the length of each link is as shown in Figure 2 and the backhoe is made of mild steel. Boom and stucco lengths are 1816.6mm and 1917.5mm respectively.

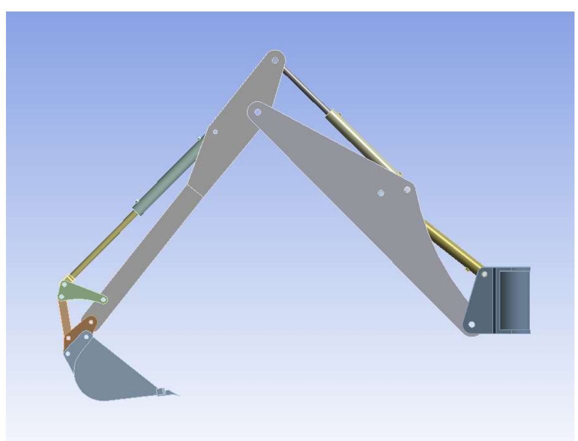

Figure 2. coordinate system at main link

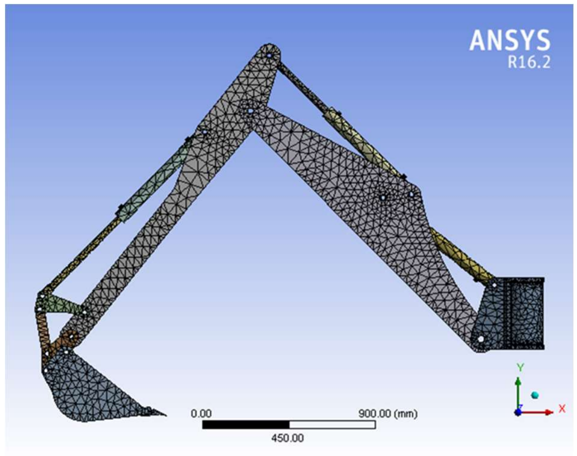

Figure 2. Meshed view of backhoe excavator

Table 1. Fifteen sets of angles provided as input for different joints

S. No.	Joint A	Joint B	Joint C	Joint D
1	0	5	5	5
2	0	10	10	10
3	0	15	15	15
4	5	0	5	10
5	5	5	0	15
6	5	10	15	0
7	5	15	10	5
8	10	0	10	15
9	10	5	15	10
10	10	10	0	5
11	10	15	5	0
12	15	0	15	5
13	15	5	10	0
14	15	10	5	15
15	15	15	0	10

3. Result and Discussions

Dynamic simulation aims to investigate the role of external loads on components and systems when the ratio of internal forces changes. The main research of excavators is to study the dynamic forces of joints. The purpose is to ensure the effectiveness of the design. Multi-body dynamics (MBD) is a very important part of mechanical engineering in general and it is also crucial for computer aided engineering (CAE). Determining kinematic and dynamic behaviour of various mechanical systems is essential in design process of almost every machine in order to meet industrial demands.

A flexible multibody model of the working model is created with dynamic simulation software. The working model is considered as a flexible body for the benefit of flexible body simulation. Buckets, arms, bucket rods, connecting rods, etc. are made rigid. To save effort and time, the strength analysis considers the boom body. The CAD model needs to add constraints to each component at the important points.

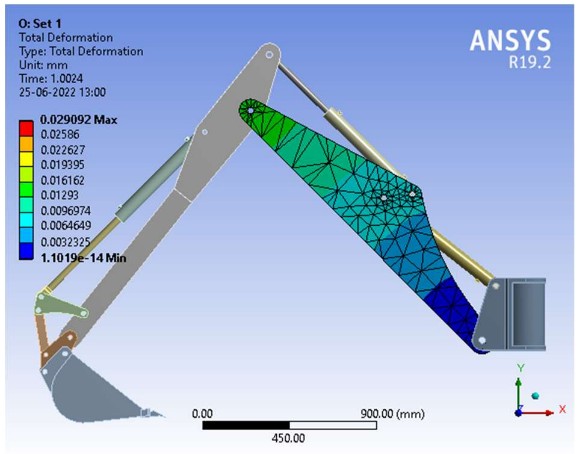

Figure 3. Contour representation of total deformation for set 1

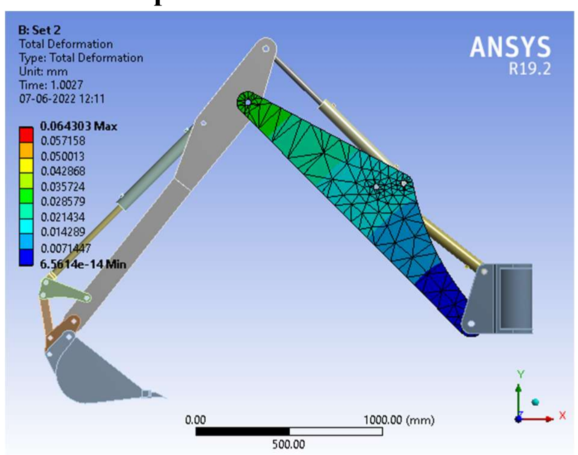

Figure 4. Contour representation of total deformation for set 2

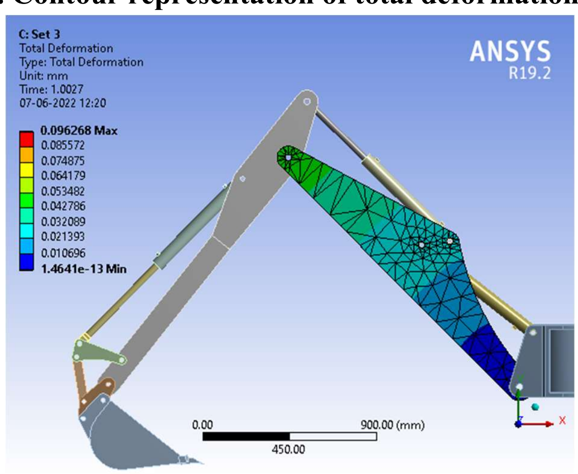

Figure 5. Contour representation of total deformation for set 3

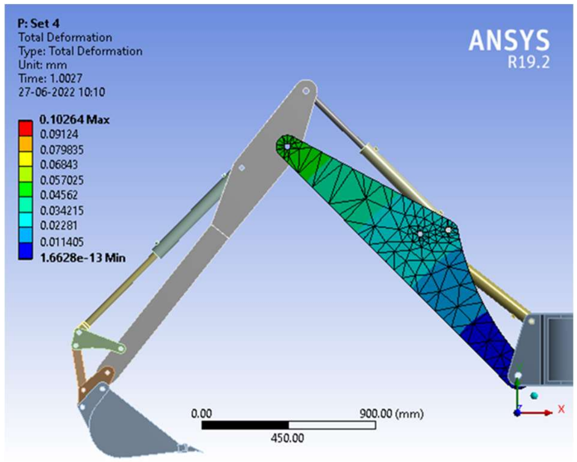

Figure 6. Contour representation of total deformation for set 4

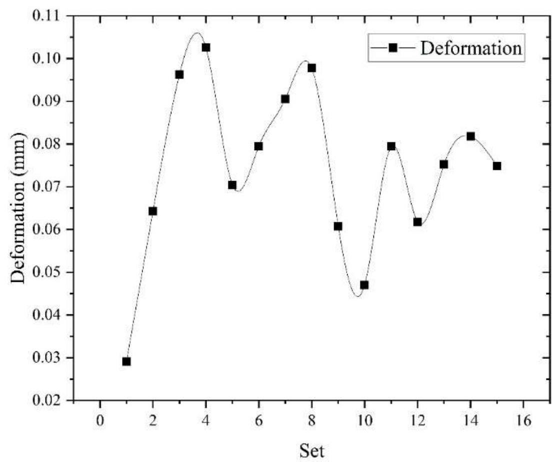

Figure 7. Variation of total deformation on boom with respect to all sets

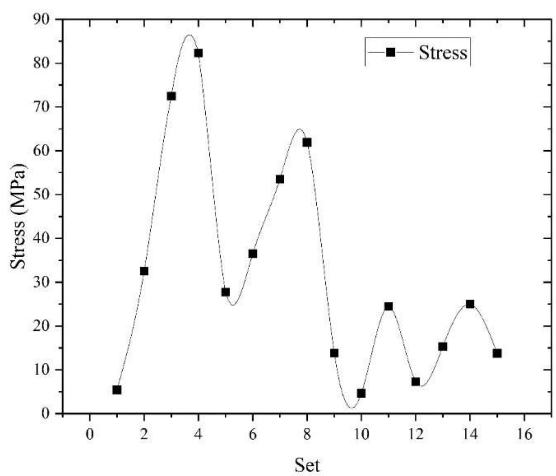

Figure 8. Variation of stress on boom with respect to all sets

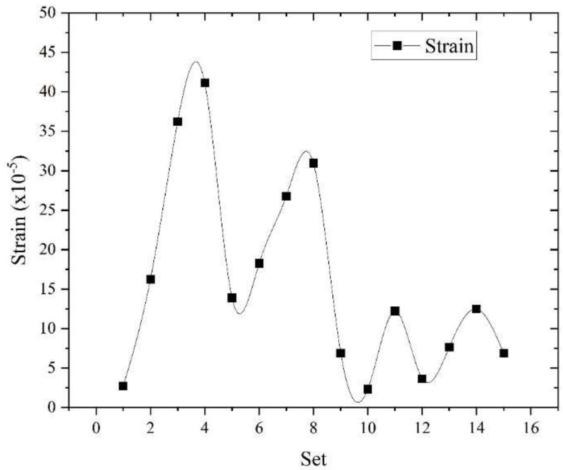

Figure 9. Variation of strain on boom with respect to all sets

4. CONCLUSION

This research work uses virtual prototyping to validate the need for rigid dynamic simulation of excavators. For this-

- The various outputs can further be obtained considering the dipper arm and boom as a flexible body. Again, a comparison can be made to further study the characteristic behavior of the material. After that an overall comparison can be done to find the difference between the two conditions, rigid and flexible, which may predict the behavior of material.
- When rigid simulation was completed, the model was further performed for flexible body dynamic analysis and carried out stresses, deformation and strain simultaneously.
- This obtained results were compared with various sets of angles and it is concluded that the total deformation was obtained minimum in set 4 and maximum in set 1.
- Also, the stresses and strain were obtained minimum in set 4 and maximum in set 1 and 10.

REFERENCES

[1] Yongliang Yuan, E3S Web of Conferences 38, 02020 (2018), ICEMEE 2018.
[2] Y. Li, S. Frimpong. Int. J. Adv. Manuf. Tech. 37, 5(2008).
[3] K. Awuah-Offei, S. Frimpong. Mech. Mach. Theory.42, 8, (2007)
[4] B. He, G. Zhou, S. Hou, et al. Int. J. Adv. Manuf.Tech. 89, 9, (2017)
[5] Y.L. Yuan, L. Du, W. Sun, et al. International Conference on Mechanical Design, (2017)
[6] Tyrell Preiss, University of Saskatchewan Saskatoon, Canada, 2011.
[7] Bhavesh Patel, J. M. Prajapati, International Journal of Mechanical, Industrial Science and Engineering Vol:8 No:1, 2014.