# Analysis of Excavator Bucket Using Ansys Software

## Swapnil D. Joshi[1], Ashish K. Gedekar[2], Prof. Chaitanya G. Deshmukh[3]

[1,2]*Student Dept. of Mechanical Engineering , Dhamangaon Education Society's College of Engineering and Technology, Dhamangaon Rly . , Maharashtra, India.*

[3]*Assistant Professor, Dept. of Mechaqnical Engineering , Dhamangaon Education Society's College of Engineering and Technology, Dhamangaon Rly . , Maharashtra, India.*

---------------------------------------------------------------------***---------------------------------------------------------------------

**Abstract -** Excavator bucket life is the very important issue, because it fails to complete its designed life. The major reason behind this failure is its working environment and modes of operations. Excavator buckets are designed to work in worst conditions, situations for long duration of time. During its working bucket undergoes with great amount of stresses, loads, jerks, deformations and it would be very difficult to withstand in such situations. Hence the excavator bucket material is tough. However this selection is not sufficient for designed bucket life. In bucket observations, we found cracks and tear areas on buckets which failed during working. Excavator Bucket always undergoes the heavy loading conditions. Hence the life of bucket is less than the designed life span. It all happened only due to the vibrations and the heavy loading conditions. Deformation and stresses induced in bucket structure must be studied well so that the life of bucket can be improved. It can be done by conducting Load test by using CAE software. Now a day's CAE softwares are more reliable and giving more approximate results. The sudden jerks, stresses, deformations are usually accurse during working. Due to sudden jerks and impact with rocks or tough materials, vibrations induces. This vibrations are measured in the form of natural frequency of a bucket. Continuous impact of such vibration range may affect the bucket life. And hence the cracks can be developed which leads to failure of bucket. There is a particular range of frequency up to which bucket can withstand. But beyond that range bucket may fails. Hence the study of resonant frequency also an important aspect for excavator bucket.

*Key Words***:**  Load test, Vibration Analysis, ANSYS, CAE Tool, Natural Frequency, Shape Optimisation

## 1.INTRODUCTION

A bucket (also called a scoop to qualify shallower designs of tools) is a specialized container attached to a machine, as compared to a bucket adapted for manual use by a human being. It is a bulk material handling component.The bucket has an inner volume as compared to other types of machine attachments like blades or shovels. The bucket could be attached to the lifting hook of a crane, at the end of the arm of an excavating machine, to the wires of a dragline excavator, to the arms of a power shovel or a tractor equipped with a backhoe loader or to a loader, or to a dredge. Excavator bucket is the important part which is responsible for work. It is used for digging, trolley felling, Heavy duty work etc. ANSYS is the original (and commonly used) name for ANSYS Mechanical or ANSYS Multiphysics, general-purpose finite element analysis software. ANSYS, Inc actually develops a complete range of CAE products, but is perhaps best known for ANSYS Mechanical & ANSYS Multiphysics. The academic versions of these commercial products are referred to as ANSYS Academic Research, ANSYS Academic Teaching Advanced, Introductory etc. All of these products are general purpose finite element self contained analysis tools incorporating preprocessing (geometry creation, meshing), solver and post processing modules in a unified graphical user interface (GUI). For the remainder of this appendix, when we state "ANSYS", we mean the FEA capability of either the commercial or academic ANSYS Inc. products discussed above. Further we are limiting our discussion to the To study stress analysis of excavator bucket with the help of ANSYS 14.5 as a CAE tool traditional (or "classic") user interface of ANSYS, not the ANSYS Workbench environment. One of the advantages of "ANSYS" is the user-programmable capability. The ANSYS Command Language contains several thousand commands relating to creating geometry, mesh, boundary conditions, solver settings & many other features. A subset of these commands are termed ANSYS Parametric Design Language (APDL), and focus on managing parameters, macros, if-then-else branching, do-loops, and scalar, vector and matrix operations. For example, if multiple analyses are required with different values of parameters, do-loops can be used to change parameters, or the Optimization module (/OPT) can be used to automatically manage parameters & populate a response surface. There are two different modes of performing finite element analysis in ANSYS: batch and interactive modes. The batch mode requires an input file of commands and executes it from the command line, while the interactive mode turns on the graphical user interface and shows the result of each user action (picked from a menu or typed as a command) in a graphics window. The batch mode is best suited for those experienced with the ANSYS Command Language. In this tutorial, only the interactive mode will be explained.

---

## 2.LITERATURE REVIEW

The idea for Ansys was first conceived by John Swanson while working at the Westinghouse Astronuclear Laboratory in the 1960s. At the time, engineers performed finite element analysis (FEA) by hand. Westinghouse rejected Swanson's idea to automate FEA by developing general purpose engineering software, so Swanson left the company in 1969 to develop the software on his own. He founded Ansys under the name Swanson Analysis Systems Inc. (SASI) the next year, working out of his farmhouse in Pittsburgh. Swanson developed the initial Ansys software on punch-cards and used a mainframe computer that was rented by the hour. Westinghouse hired Swanson as a consultant, under the condition that any code he developed for Westinghouse could also be included in the Ansys product line. Westinghouse also became the first Ansys user. By 1991 SASI had 153 employees and $29 million in annual revenue, controlling 10 percent of the market for finite element analysis software. According to the engineering design revolution, the company became "well-respected" among engineering circles, but remained small. In 1992, SASI acquired Compuflo, which marketed and developed fluid dynamics analysis software. In 1993, Mr. Swanson sold his majority interest in the company to venture capitalist firm TA Associates. Peter Smith was appointed CEOand SASI was renamed after the software, Ansys, the following year. Ansys went public in 1996, raising about $46 million in an initial public offering. By 1997, Ansys had grown to $50.5 million in annual revenue. In the late 1990s, Ansys shifted its business model. It focused less on selling software licenses and corresponding revenue declined. However, revenue from services increased more dramatically. From 1996 to 1999, profits at Ansys grew an average of 160 percent per year. In February 2000, Jim Cashman was appointed CEO. Current CEO Ajei S. Gopal was appointed in early 2017. The first commercial version of Ansys software was labeled version 2.0 and released in 1971. At the time, To study stress analysis of excavator bucket with the help of ANSYS 14.5 as a CAE tool the software was made up of boxes of punch cards, and the program was typically run overnight to get results the following morning in 1975, non-linear and thermo-electric features were added. The software was exclusively used on mainframes, until version 3.0 (the second release) was introduced for the VAXstation in 1979. Version 3 had a command line interface like DOS. In 1980, apple II was released, allowing Ansys to convert to a graphical user interface in version 4 later that year. Version 4 of the Ansys software was easier to use and added features to simulate electromagnetism. In 1989, Ansys began working with Compuflo. Compuflo's Flotran fluid dynamics software was integrated into Ansys by version 5, which was released in 1993. Performance improvements in version 5.1 shortened processing time two to four-fold, and was followed by a series of performance improvements to keep pace with advancements in computing. Ansys also began integrating its software with CAD software, such as Autodesk. In 1996, Ansys released the DesignSpace structural analysis software, the LS-DYNA crash and drop test simulation product, and the Ansys Computational Fluid Dynamics (CFD) simulator. Ansys also added parallel processing support for PCs with multiple processors. The educational product Ansys/ed was introduced in 1998. Version 6.0 of the main Ansys product was released in December 2001. Version 6.0 made large-scale modeling practical for the first time, but many users were frustrated by a new blue user interface. The interface was redone a few months later in 6.1. Version 8.0 introduced the Ansys multi-field solver, which allows users to simulate how multiple physics problems would interact with one another. Version 8.0 was published in 2005 and introduced Ansys' fluid–structure interaction software, which simulates the effect structures and fluids have on one another Ansys also released its Probabilistic Design System and Design Xplorer software products, which both deal with probabilities and randomness of physical elements. In 2009 version 12 was released with an overhauled second version of Workbench. Ansys also began increasingly consolidating features into the Workbench software. Version 15 of Ansys was released in 2014. It added a new features for composites, bolted connections, and better mesh tools. In February 2015, version 16 introduced the AIM physics engine and Electronics Desktop, which is for semiconductor design. The following year, version 17 introduced a new user interface and performance To study stress analysis of excavator bucket with the help of ANSYS 14.5 as a CAE tool improvement for computing fluid dynamics problems. In January 2017, Ansys released version 18. Version 18 allowed users to collect real-world data from products and then incorporate that data into future simulations. The Ansys Application Builder, which allows engineers to build, use, and sell custom engineering tools, was also introduced with version 18. Released in January 2020, Ansys R1 2020 updates Ansys' simulation process and data management (SPDM), materials information and electromagnetics product offerings. In early 2020, the Ansys Academic Program surpassed one million student downloads. In November 2020, South China Morning Post reported that Ansys software had been used for Chinese military research in the development of hypersonic missile technology.

## 3.AIMS AND OBJECTIVE RESEARCH

Aims:
i)     Life improvement of excavator bucket by performing different CAE analysis.
ii)    Stress Analysis to find the possible deformation and stress concentrated areas.
iii)   Vibration/modal analysis of bucket for obtaining natural frequency value.
iv)    Shape optimization of Modified Excavator Bucket.
v)     Redesigning of bucket.

Objectives:
i)     Comparisons of existing bucket and modified bucket.
ii)    Analysis of modified and optimized bucket.
iii)   Tabulated data formation for three different loading conditions.
iv)    Conclusion on the basis of obtained CAE results.

## 4. STRUCTURAL ANALYSIS ON EXCAVATOR BUCKET

MATERIAL PROPERTIES REQUIRED:

i) Poisson's Ratio

When applying a load in a certain axis direction and consequently deforming in that direction, the ratio of deformation on the opposite side is called Poisson's Ratio. Easily put, when pulling out the tensile specimen, the proportion of the reduced amount of deformation in the vertical direction from load with respect to the increased deformation amount in the load direction. Basically, it can be used at the elastic region because of the assumption of Hooke's Law, theoretically having a value less than 0.5 greater than zero. Common carbon steel's Poisson's Ratio is 0.28 to 0.3, and rubber is almost close to 0.5. Our flagship product, F20-03 is about 0.39 or so. The structural analysis must be entered in the required Poisson's Ratio, most resin is often difficult to analyze because its Poisson's Ratio is not well known

ii) Yield Stress

As described above, yield stress is the point at which permanent deformation is drastically increased when an external force is increased beyond the elastic limit. As for metallic material, the maximum stress of the elastic limit is the same as yield stress. If you remove the load in the tensile test using general specimens, the maximum stress value returning to the initial point of the SS curve becomes yield stress; even it returns, it has an error with the permanent deformation, Mechanics of Materials recommend using the offset yield stress that allows 0.05 to 0.3 percent of permanent strain.

iii) Density:

Since density is mass per unit volume, the density of a metal can be calculated by submerging it in a known amount of water and measuring how much the water rises. This rise is the volume of the metal. Its mass can be measured using a scale.

## 5. ANSYS AS A CAE TOOL

ANSYS software is used to design products and semiconductors, as well as to create simulations that test a product's durability, temperature distribution, fluid movements, and electromagnetic properties. ANSYS was founded in 1970 by John Swanson. Swanson sold his interest in the company to venture capitalists in 1993. ANSYS went public on NASDAQ in 1996. In the 2000s, ANSYS made numerous acquisitions of other engineering design companies, acquiring additional technology for fluid dynamics, electronics design, and other physics analysis. ANSYS develops and markets finite element analysis software used to simulate engineering problems. The software creates simulated computer models of structures, electronics, or machine components to simulate strength, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, and other attributes. ANSYS is used to determine how a product will function with different specifications, without building test products or conducting crash tests. For example, ANSYS software may simulate how a bridge will hold up after years of traffic, how to best process salmon in a cannery to reduce waste, or how to design a slide that uses less material without sacrificing safety. Most ANSYS simulations are performed using the ANSYS Workbench software,which is one of the company's main products. Typically ANSYS users break down larger structures into small components that are each modeled and tested individually.A user may start by defining the dimensions of an object,and then adding weight, pressure, temperature and other physical properties.Finally, the ANSYS software simulates and analyzes movement, fatigue, fractures, fluid flow, temperature distribution, electromagnetic efficiency and other effects over time. ANSYS also develops software for data management and backup, academic research and teaching. ANSYS software is sold on an annual subscription basis.

## 6. STEPS TO PERFORM STRUCTURAL ANALYSIS

To perform structural analysis of Existing Excavator Bucket, following steps are to be performed.

Step 1: Open ANSYS Workbench 14.5 Software and select structural analysis option from analysis setting menu.

Step 2: Set material Properties in engineering data module for excavator bucket.

Step 3: Import .igs format file of existing excavator bucket into design modeler module.

Step 4: Apply material property to imported bucket geometry.

Step 5: Perform Meshing operation in model module. This process is also called as descretisation process.

Step 6: Apply Boundary Conditions on Excavator Bucket.

Step 7: Select required result type from solution menu. Step 8: Solve the analysis.

Step 9: Save required Results.

## 7. The Method of Stress Analysis:

• The analysis of stress process within a body entails the willpower at all points of the body of the scale of 9-stress components. In other way, it is the willpower of the interior allocation of stresses. An option technique used in stress analysis is the purpose of the interior allocation of strains.

• Professionals can build a part for serving a function while reducing the quantity of material used to make it for reducing the cost. To make sure that this specific part can work as required without breaking, these parts must be tested on the quantity stress it can take. Within SolidWorks Stress Analysis, an individual can create a part with a specific material or shape and perform a stress analysis of the concerned part.

• The main part of analysis includes fixing the nature of loads acting on a structure, including tension, density, trim, torsion, winding or an amalgamation of some of these loads. When force is applied frequently almost all materials will get ruptured or failed at a lower stress than they should act likewise under a stable shape load condition. The analysis process to determine stress under these specific cyclic load conditions is called fatigue analysis and it is often applied for determination of stress level in an aerodynamic structural system.
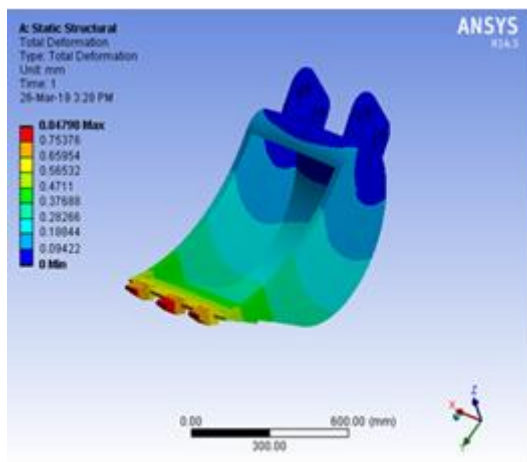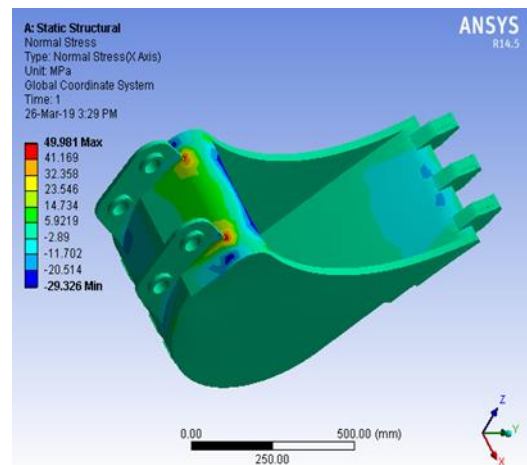


**Fig:** Total Deformation



**Fig:** Shear Stress

## 8. RESULTS AND DISCUSSION

**Table:** Results Values

| Sr. No. | Result | Value |
|---------|--------|-------|
| 1 | Total Deformation | 0.89 mm |
| 2 | Equivalent Stress | 78.817 MPA |
| 3 | **Shear Stress** | **25 MPA** |
| 4 | Normal Stress | 62.21 MPA |
| 5 | Stress Intensity | 81.625 MPA |

## 9. CONCLUSION

The stress at the tip of the teeth is calculated 112.98 MPA and stress due to shearing of rivet 157.47 is calculated. Percentage error between analytical result and Ansys result are 13.69 % and 6.72 %. As per the above analysis, it is suggested that the bucket used for the excavation purpose should be properly checked for its application on the basis of the soil strata. And considering the failure of the tooth and rivet due the impact loading, it is very much economical to change the tooth assembly and also the inclination and thickness of the tooth. ¬ Structural analysis was carried out on the excavator bucket at different widths of teeth such as 25 mm, 30 mm, 35 mm, 40 mm and 45 mm. And the analysis was carried out on three types of materials named Stainless Steel, AISI-1045 and TI Carbide and the action of various stress and strains on the excavator bucket at various loads were investigated. The best combination of parameters like Von misses Stress and Equivalent shear stress, Deformation, shear stress and weight reduction for excavator bucket were done in ANSYS software.

## REFERENCES

[1] Miloš Tanasijević, "Quality of Service Evaluation for Bucket Wheel Excavator", Faculty of Mechanical Engineering, BelgradeFME Transactions (2007) 35, 141-148

[2] Nedeljko Vukojević, Fuad Hadžikadunić, "Evaluation Of Stress-Strain State Of Repaired Bucket Wheel Excavator Sh630", Faculty of Mechanical Engineering, Fakultetska br.1, 72 000 Zenica, B&H

[3] Santosh Gudagunti, " Implementation of Lean in Excavator Bucket Manufacturing Industry", Department of Industrial Engineering Lawrence Technological University Southfield, MI 48075, USA

[4] Janmit Raj, "Study on the Analysis of Excavator Boom: A Review", Research Scholar, Department of Automobile Engineering, Rustam Ji Institute of Technology, Tekanpur, Gwalior (M. P.)- 475005, India Volume 2 Issue 7–July 2015

[5] Vishwajeet A. Patil, " Fatigue Analysis and Design Optimization of Excavator Bucket", Department of Mechanical Engineering, Vishwakarma Institute of Technology, Pune, India

[6] Mehul Kumar A Patel, "A CRITICAL REVIEW ON KINEMATICS OF HYDRAULIC EXCAVATOR BACKHOE ATTACHMENT" , Int. J. Mech. Eng. & Rob. Res. 2015

[7] Arjun Kundu, "Analysis of Excavator Bucket Teeth Using FEM", Assistant Professor, Department of Mechanical Engineering, Rungta Engineering College Bhilai, Chattishgarh, India Vol. 6, Issue 4, April 2017

[8] K.Sathishkumar, "Design and Analysis of Hardness Improvement on Excavator Bucket Teeth", Assistant Professor, Department of Mechanical Engineering, Sree Sakthi Engineering College, Karamadai - 641 104, Coimbatore, India Vol-3 Issue-2 2017.

[9] Dr. S.C. Kongre, "Modeling and Static Analysis of Backhoe Excavator Bucket", HOD Mechanical Engineering, Acharya Shrimannarayan Polytechnic, Pipri, Wardha Vol.4, No.3, March 2016.

[10] Prof.A.Sivasubramaniam, "Design and Analysis of an Excavator Bucket", Associate Professor, Department of Mechanical Engineering Saveetha School of Engineering, Saveetha University, Chennai, India Vol. 7, Issue 7, ( Part -3) July 2017, pp.20-23.

[11] Manisha P. Tupkar, "Design and Analysis of an Excavator Bucket", M Tech (CAD/CAM), Department of Mechanical Engineering Rajiv Gandhi Collage of Engineering Research and Technology Chandrapur (MS) INDIA Volume 4, Issue 3, March 2015.