International Journal of Scientific & Engineering Research, Volume 8, Issue 4, April-2017
ISSN 2229-5518

Finite Element Analysis of an Excavator Arm using CAE Tool

Asit Kumar Choudhary, Gian Bhushan

Abstract:

Background*:** Excavators are earth moving equipment and the main component for completing its function is arm; which directly affect the working performance and reliability of the excavator. However, using the finite element method for structural analysis of excavator arm is the premise for the structural design of the arm. ***Objective*:** In this paper, we have carried out, finite element analysis of excavator arm by using CAE tools. The tools being used are CATIA, HYPERMESH and RADIOSS LINEAR. The obtained FEA results have been compared with the similar work carried out on the CAD model using ABAQUS. ***Method*:** By using, CAE software Pro/E, the three dimensional (3-D) model of working mechanism in big arm of an excavator, was established. The finite element analysis (FEA) software ANSYS were used to analyze the stiffness, and strength of excavator design. Meshing and analysis was carried out in Hypermesh. ***Result: The analysis results showed that the strength and stiffness for the arm was sufficient and it's was below the critical value. Generally stress values must be below critical value to ensure that the new design is safe. ***Conclusion:*** Finite element analysis using computer-aided design can be used in force and strength analysis, which is an important step in the designing of excavator parts.

Keyword: Computer-aided design; excavator, finite element analysis; hypermesh

1. Introduction

At present, the machines being used for the earth moving works is increasing in the era of globalization and tough competition, as well as extensive attention has been focused on designing of the earth moving equipments [1] . Excavators are typical examples of the heavy duty human operated hydraulic machines and widely used in construction, mining, and excavation applications [2]. By using, CAE software Pro/E, the three dimensional (3-D) model of working mechanism in big arm of an excavator, was established. The finite element analysis (FEA) software ANSYS were used to analyze the stiffness, and strength of excavator design[3].

Author detail

Asit Kumar Choudhary[1], Gian Bhushan[2]

[1] Department of Research and Development, Tata Steel Limited, India

[2]Deparment of Mechanical Engineering, N.I.T. Kurukshetra, India

***Corresponding author:**
Mr. Asit Kumar Choudhary
Researcher
Department of Research and Development,
Tata Steel Limited,
Jamshedpur 831001, India
Email: asit.choudhary1@gmail.com

The literature review indicates the major development in implementation of FEA on various components of excavator. The SDRC's (Structural Dynamics Research Corporation) redesign package has been used for stiffness and strength requirements of an excavator arm [4]. The computer simulation have been used to estimate the load capacities three dimensional multibody modeling of a hydraulic excavator[5]. The control of excavation processes by applying load-independent hydraulic valves was investigated by using two subsystems: a microcomputer and a hydraulic unit (a pump and load-independent valves) [6] .The autonomous control technology, to measure and analyze its motions when it is used for excavation and loading work under the control of an operator has been revealed [7]. The multi-Body model of an excavator and to simulate the prototype testing conditions, which involves the simulation of rigid body systems under the application of cylinder forces and /or motions has been generated[8]. The MBD analysis was performed to check maximum reaction forces at different joints which will cause maximum bending stress in the arm and help in determining in which configuration arm will be under maximum stressed condition [9]. Excavator dynamic models using simplified refined instrumental Variable (SRIV) algorithm, is designed to utilize proportional-integral-plus (PIP) control theory to provide smoother, more quickly and more accurately movement in excavation [10]. Refined 3D FE model of the mechanism explains failure of stress distribution of the working mechanism and provides a new approach for design and analysis[11]. A simple light weight tele-operation system which has been developed for the excavator for dealing with its movement problems[12]. The effect of various parameters on soil-tool interaction, prediction of digging trajectory and excavation forces and for robust design of backhoe mechanism has been discussed[13].

The method has been improved and a new approach was provided for similar product design and analysis. Computer-aided technologies for supporting in tasks such as analysis, simulation, design, manufacture, planning, diagnosis and repair[14].

HyperWorks is an enterprise simulation solution for rapid design exploration and decision-making and provides a tightly integrated suite of best-in-class tools for modeling, analysis, optimization, visualization, reporting, and performance data management. RADIOSS is finite element software, which allows mechanical, structure, fluid, or fluid-structure interaction problems resolution, under dynamic or static solicitations. RADIOSS provides small and large deformation finite element, multi-body dynamics, and sheet metal stamping analysis.

2. Objective

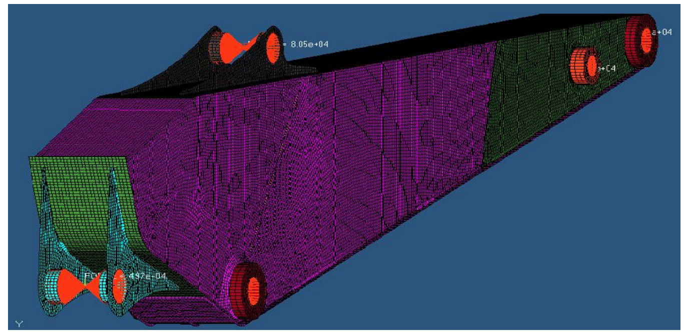
We have carried out, finite element analysis of excavator arm by using CAE tools. The tools being used are Catia, hyperMesh and Radioss Linear. The FEA results obtained have been compared with the similar work carried out on the CAD model using ABAQUS.

3. Methodology

To pursue CAD model generation and finite element analysis, we have used *CATIA V5-R19* and *Altair Hyperworks 9.0.* Altair HyperMesh is a high-performance finite element pre- and post-processor for major finite element solvers, which allows analyzing design conditions in a highly interactive and visual environment.

3.1 FE modeling methodology:

The first step in preprocessing was to prepare a CAD Model of excavator arm and can be used for different loads under similar conditions. CAD modeling of the complete arm was generated by using CATIA software. CATIA is having special tools in generative surface design to construct typical surfaces, which was later on converted into solid. A Solid model of all parts of the structure was assembled to make a complete structure. The process of assembly was very much analogous to general process of fabricating structure while real production. CAD model of our problem consists of four parts, which was assembled together in assembly design to make a complete model. The CAD model of Excavator arm used for analysis is shown in (Figure 1).



3.2. Importing the CAD model:

The model used in this study was prepared in IGES (Initial Graphics Exchange Specification) format which is compatible with all CAD software. After importing the CAD file into Hypermesh, saved in form of hm format.

3.3. Geometry visualization, geometry cleanup and symmetry check

After importing the CAD data, the first step was geometry cleanup to restore proper surface connectivity to the part geometry. After geometry visualization and check it was found out whether there is any symmetry in the model or not. The reflective symmetry has been used in our model.

3.4. Geometry preparation or simplification:

The next step was preprocessing, in which prepares the geometry for analysis. This involves the task of removal of unwanted features, extraction of mid-surfaces, changing the shape of the part for simplify the geometry. Certain details of the shape, such as small holes or blends were not considered for the analysis because of less Importance of the part in the overall assembly. As well as model was represented in a much simpler form that suits the analysis to be performed.

3.5. Refining topology to achieve a quality mesh:

Topological details of the geometry may affect the quality of the mesh created from the surfaces. Split surface and ruled surface operations were performed on excavator arm geometry to produce quality mesh.

3.6. Element length and 2D shell meshing:

Element length was taken as 10 and we have selected manual 2D mesh for meshing all the components. The surfaces are then meshed with algorithms that produce the best quality index (QI). The placement of the nodes on the surface is also optimized to improve the (QI).

3.7. Conversion of 2D (Tria) to 3D (Tetra) and using solid map in 3D option:

The 2D (Tria) to 3D (Tetra)-meshing technique was utilized to mesh the bearing components of excavator arm and meshing the face of the bearing and then using line drag option to create the solid elements.

3.8. Quality checks

After completing meshing part of the model, quality check was performed to check the quality of elements. The quality index panel was used to evaluate the quality of the mesh as well as improve local areas, while the smooth panel was used to achieve larger scale improvements to the mesh quality based on the quality index. The percentage of failed element was almost negligible.

3.9. Element free edges:

Any single quad element has 4 free edges. In this case middle edge was shared and there was no more free edge.

4.0. Load collector-boundary condition:

We had taken six load collector in which one load collector was defining the constrain i.e fixing the bearing and four load collector was defining the forces which acting on the different boundary conditions and one load collector was used to updating the all these four load collector in this one. Assigned different color to each load collector for easy recognition (Figure 2).

5. Design and Analysis of excavator arm:

The results of structural linear static analysis of the Excavator Arm using Radioss linear have been presented. The Figures (2A and B) showed the drawing details and loading conditions of the Excavator Arm for Digging and Dumping conditions respectively.

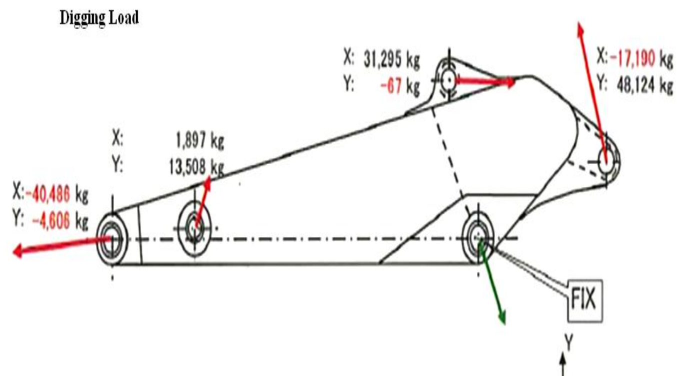


Fig 7.1 Loading Condition for Digging

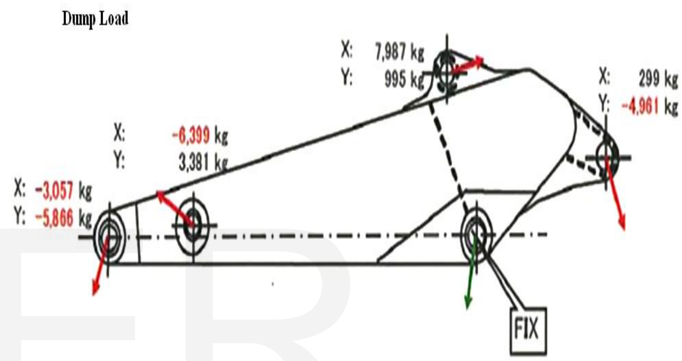


Fig 7.2 Loading Condition for Dumping

The Figures (3A and B) showed the displacement contour of Excavator Arm for Digging and dumping conditions respectively.

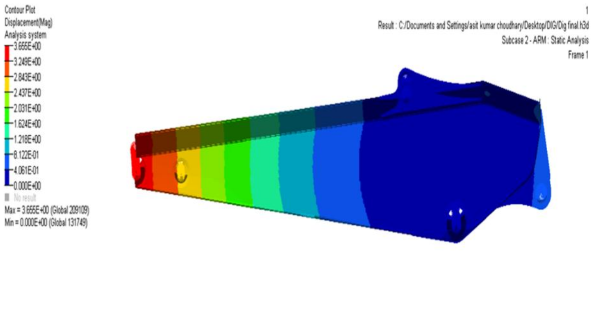


Fig 7.3 Displacement Contour for Digging Operation

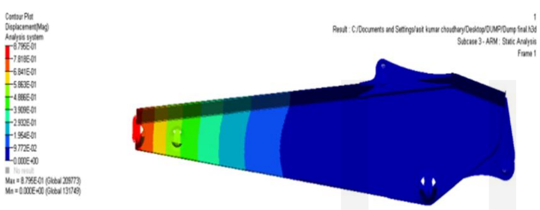


Fig 7.4 Displacement Contour for Dumping Operation

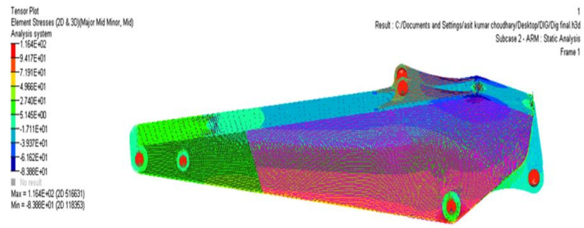


Fig 7.5 Principal Stress for Digging Operation

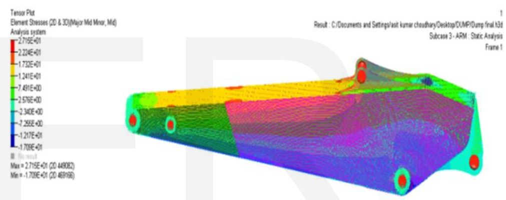


Fig 7.6 Principal Stress for Dumping Operation

The values of maximum displacement for these conditions were 3.65mm and 0.8795 mm which are permissible. The Figures (4A and B) showed the principal stresses of digging and dumping conditions and the values of principal stresses are 116.4MPa and 27.15MPa respectively, which are well below the yield stress of 325MPa.

However, it also observed maximum stress occurs at the free end of the Excavator Arm. The Figures (5A and B) depict that the maximum values of VonMises stresses of Digging and Dumping Conditions and the values are 171.7MPa and 34.75MPa respectively, which are below the yield stress (325MPa) of excavator Arm material i.e M.S.

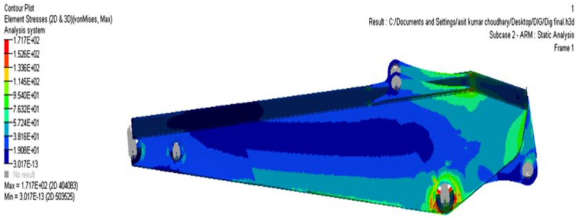


Fig 7.7 VonMises Contour Stress for Digging Operation

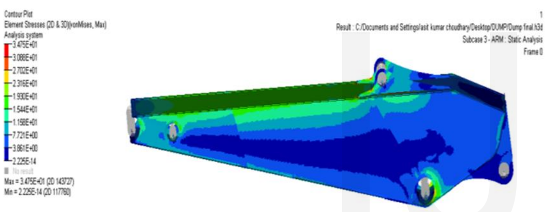


Fig 7.8 VonMises Contour Stress for Dumping Opertaion

The FEA results of the excavator arm have been compared with available ABAQUS results for comparison and showed in the (Tables 1 and 2).

Table 1. The comparison of ABAQUS and FEA Results of Digging Operation.

Sr. No.	Parameters	FEA results using Abaqus	FEA results using Hypermesh and Radioss	Variation
1	Principal Stress	155.1MPa	116.4MPa	24%
2	Displacement	6.9 mm	3.65 mm	47%

Table 2. The comparison of ABAQUS and FEA Results of Dumping Operation.

Sr. No.	Parameters	FEA results using Abaqus	FEA results using Hypermesh and Radioss	Variation
1	Principal Stress	64.8MPa	27.15MPa	58%
2	Displacement	1.2 mm	0.8795 mm	26%

6. Conclusion

From past few decades, the existence of commercial software packages has allowed engineers and analysts to narrow the development process in either products or systems. Finite element method has provides detail analysis of a conceptual design. These tools allow determining forces and stresses that are developed in critical points and allow for modifications for the purpose of meeting the established criterion of the design.

Design of the excavator arm has been modified and analysis of the design was also done. The maximum value of principal stress was developed in excavator arm for digging operation (116.4MPa) and dumping operation (27.15MPa), was well below the yield stress for the given loading condition. It is clearly depicted that the stresses produced are within the safe limit.The maximum displacement observed for digging (3.65mm) as well as dumping (0.8795mm) at free end where the bucket fixed was permissible. The values of the VonMises stresses for digging and dumping were 171.7MPa and 34.75MPa respectively. Generally stress values must be below critical value to ensure that the new design is safe. Therefore, it can be concluded that, finite element analysis using computer-aided design can be used in force analysis as well as strength analysis, which is an important step in the designing of excavator parts.

Acknowledgements:

The authors would like to thank for the financial support provided by national institute of technology, Kurukshetra, India.

Conflict of interest:
The authors declared no conflict of interest.

References:

[1]. Dadhich, S., U. Bodin, and U. Andersson, *Key challenges in automation of earth-moving machines.* Automation in Construction, 2016. 68: p. 212-222.

[2]. Yi, K.-J., *Modelling of On-Site Energy Consumption Profile in Construction Sites and a Case Study of Earth Moving.* Journal of Construction Engineering and Project Management, 2013. 3(3): p. 10-16.

[3]. ZHU, C.-h., X.-y. GUO, and S.-l. WANG, *Finite Element Analysis of Working Equipment of Excavator Based on Pro/E and ANSYS [J].* Coal Mine Machinery, 2009. 4: p. 032.

[4]. Zienkiewicz, O.C. and R.L. Taylor, *The finite element method: solid mechanics.* Vol. 2. 2000: Butterworth-heinemann.

[5]. Yoo, W.-S., et al., *Estimation of maximum lifting load capacities of a hydraulic excavator via multibody computer modeling and simulation.* Journal of Mechanical Science and Technology, 1998. 12(6): p. 1090-1096.

[6]. Budny, E., M. Chłosta, and W. Gutkowski, *Load-independent control of a hydraulic excavator.* Automation in Construction, 2003. 12(3): p. 245-254.

[7]. Yamaguchi, T. and H. Yamamoto, *Motion analysis of hydraulic excavator in excavating and loading work for autonomous control.* 23rd ISARC2006, 2006: p. 602-607.

[8]. Vadhe, R. and V. Dave, *Multi-Body Simulation of Earthmoving Equipment using Motionview/Motionsolve.* Proceeding of the HTC, 2008.

[9]. Filla, R. and J.-O. Palmberg, *Using dynamic simulation in the development of construction machinery.* arXiv preprint cs/0305036, 2003.

[10]. Gu, J. *Design of robotic excavator arm control utilizing proportional-integral-plus*. in *Intelligent Control and Automation, 2008. WCICA 2008. 7th World Congress on*. 2008. IEEE.

[11]. Cui, G. and Y. Zhang. *Integrated finite element analysis and experimental validation of an excavator working equipment*. in *Information Engineering and Computer Science, 2009. ICIECS 2009. International Conference on*. 2009. IEEE.

[12]. Kim, D., et al., *Excavator tele-operation system using a human arm.* Automation in construction, 2009. 18(2): p. 173-182.

[13]. Patel, B.P. and J.M. Prajapati, *Soil-tool interaction as a review for digging operation of mini hydraulic excavator.* International Journal of Engineering Science and Technology, 2011. 3(2): p. 894-901.

[14]. Van Houten, F. and F. Kimura, *The virtual maintenance system: a computer-based support tool for robust design, product monitoring, fault diagnosis and maintenance planning.* CIRP Annals-Manufacturing Technology, 2000. 49(1): p. 91-94.