Juber Hussain Qureshi, Manish Sagar/ International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com
Vol. 2, Issue 3, May-Jun 2012, pp.1484-1487

The Finite Element Analysis of Boom of Backhoe Loader

Juber Hussain Qureshi *, Manish Sagar**

*(M.E. Scholar, Mechanical Engineering Department, MITS, Gwalior, MP, India)

** (Asst. Prof. Mechanical Engineering Department Madhav Institute of Technology & Management, Gwalior, MP, India)

ABSTRACT

A Backhoe machine is used to lift tones and tones of load, thus some amount of pressure is to be developed for providing the necessary force by the cylinder to lift this load. FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. Finite element Analysis of the boom is made followed by the results of Dynamic study of the Boom of the machine. In this paper researcher provides the platform to understand the Modelling and FEA of Boom of Backhoe Loader, which was already carried out by other researchers for their related applications and it can be helpful for the development of boom of backhoe loader.

Keyword: - Boom; FEA; Backhoe; Dynamics.

1. INTRODUCTION

Backhoe-Loader is most preferred for excavation and earth moving due its versatility. Many construction companies consider the backhoe to be the workhorse of earthmovers. The backhoe is used to dig up hard material, handle cohesive soil and pieces of rocks generated by blasting. It is attached to the back of the tractor. It easily lifts heavy loads. The backhoe gets its name from the placement of the scooper bucket assembly on the back of the tractor. The backhoe consists of three pieces – the boom, the stick or dipper, and the bucket. The boom is the upper arm piece. The stick is the lower arm piece. The bucket is the digging and lifting tool at the end.

Figure 1 Backhoe Loader

2. DISCRIPTION OF FINITE ELEMENT ANALYSIS

Finite element analysis is a word used in recent language of Mechanical Engineer in which the problems are solved with the use of software using Finite element method to solve the difficult problems where the direct implication of the problem is not always ready in the formula's provided in the standards or the books. The Finite element method is a numerical procedure which can be used to solve numerical problems in Engineering.

An unsophisticated description of the FE method is that it involves cutting a structure into several elements (pieces of the structure), describing the behaviour of each element in a simple way, than reconnecting elements at "nodes" as if nodes were pins or drops of glue that holds together the elements.

This process results in the set of simultaneous algebraic equations which can be solved with mathematical formulations for solving the problem for deflection which can be further solved to get the results of strains and stresses. There may be several and thousands of such equations formed which means the computer implication is mandatory.

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

Working of FEA

FEA uses a complex system of points called nodes which make a grid called a mesh **Figure 2** .This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements.

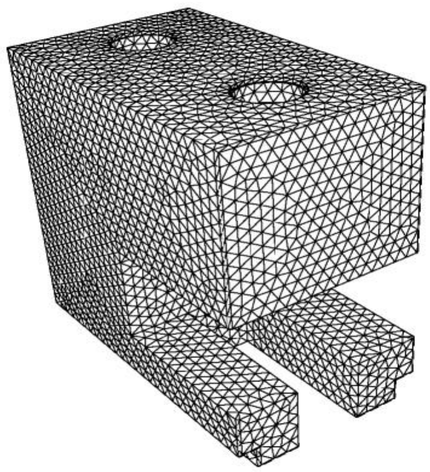

Figure 2 Tetrahedral Mesh

3. RESEARCH METHODOLOGY & EXPERIMENTAL WORK

Finite element modelling is one of the major subjects of Computer Aided Engineering where the importance of Finite element method is sub-divided in following rules of problem solving methodologies.

3.1 Preprocessing: The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub-regions, or elements, "connected at discrete points called nodes."

3.2 Analysis: The dataset prepared by the preprocessor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations

$$\mathbf{K}_{ij}\mathbf{u}_j = \mathbf{f}_i$$

where u and f are the displacements and externally applied forces at the nodal points

The formation of the K matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types.

3.3 Post-processing: In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results

The use of Finite element is the second step considered in the Finite element analysis which falls after modeling of the part/assembly to be analyzed. It is important to take decision before starting the Finite element modeling to ensure the results are correct and your model behaves properly as thought based on engineering principles. The Few types of modeling used are illustrated below:

3.4 Shell Modeling**:-**Shell element is used where the modeling style is very critical and the models cannot be modeled in the form of a surface, or say the joining of the intersecting surfaces cannot be used for producing the thickness.

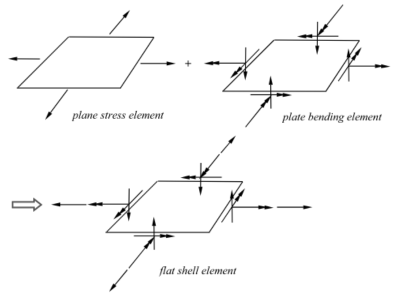

Figure 3 Shell Modeling

3.5 Solid Modeling**:-**Here the modeling done for the boom is solid modeling, because of the critical welding geometries involved. When the modeling is performed of solid type the most common used mesh types are Tetrahedral Mesh Elements and Hexahedral Mesh/Brick Elements.

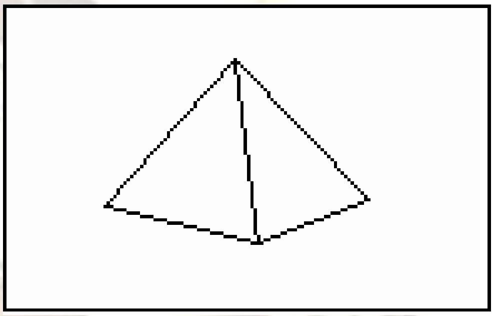

Figure 4 Solid Modeling

3.6 Beam Modeling: **-** A beam is a one-dimensional idealization that, in three dimensions, represents a structure whose length is much greater than its other two dimensions. You create a beam by specifying the cross-section shape and position, the degrees of freedom at the beam ends, and the location of the beam with respect to the axis where Mechanica applies the beam load. Mechanica sees beams slightly differently in native mode and FEM mode.

3D Modeling of Boom of Backhoe Loader

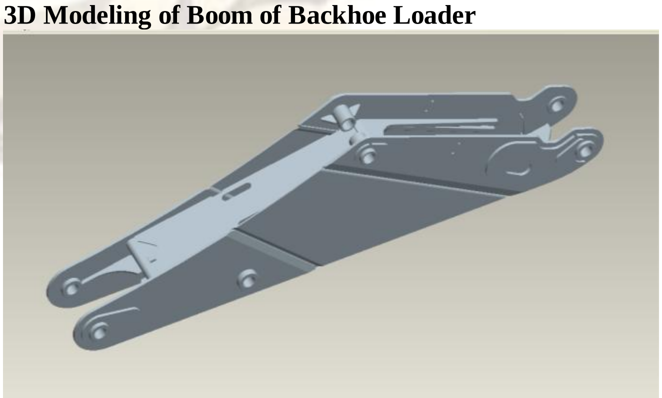

Figure 5 Cad model of Boom

Meshed 3D model of Boom of Backhoe Loader

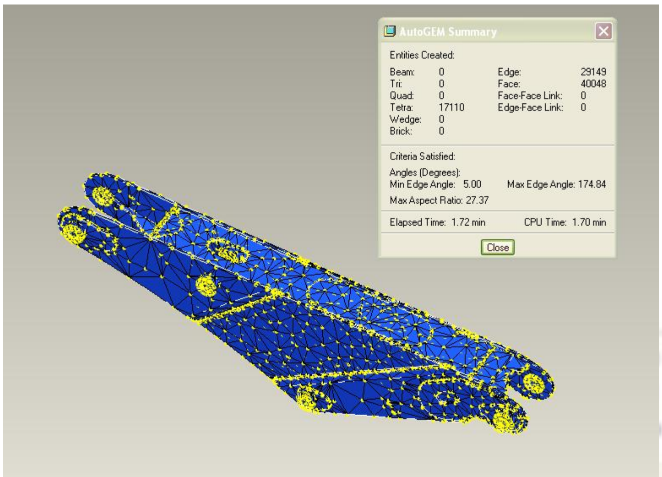

Figure 6 Tetrahedral Meshed Model of Boom

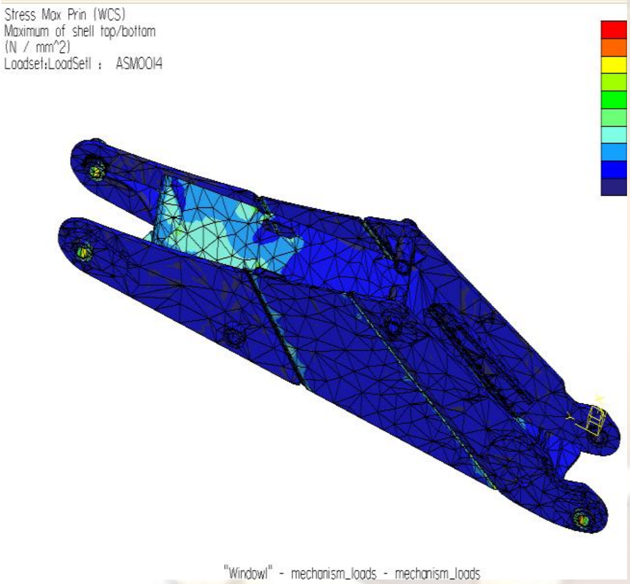

Figure 7 Max Principal Stresses for Mechanism Load
The Maximum Principal Stresses are shown fig.7 where the stress limits in the green zone is of 30-32 MPa while that in the red zone is of 45-48 MPa.

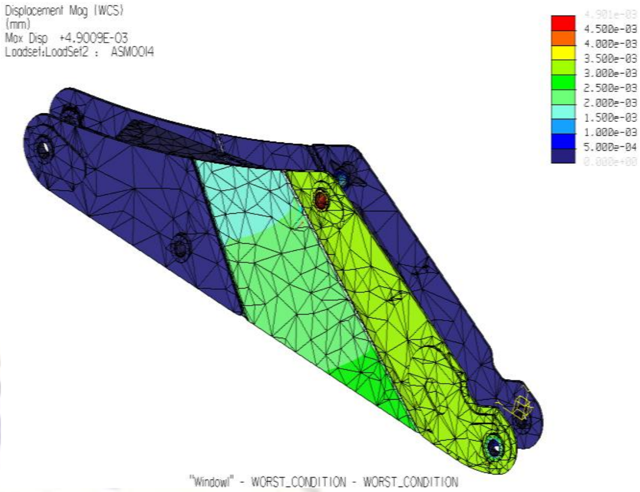

Figure 7 Results of Displacement for Total force

The Maximum Displacement seen in the structure is only 4.9E-04, thus the structure can be considered to be very stiff under this type of loading condition

4. RESULT

With the results obtained from the **Finite Element Analyse by Pro-e Mechanica software is** are very safe as per the stress values and the deflections.

5. CONCLUSION

As Inertia plays a big effect while performing a dynamic analysis which is completely dependent upon the time in our case which can only be assumed for the cycle to complete. The max effect of inertia can be plotted on the graph especially for new shapes of boom, to get the safe results of stresses resulting in the safe life of the boom.

REFRENCES

[1] Mehta Gaurav K, “Design and Development of an Excavator Attachment” M. tech Dissertation Thesis, Nirma University, Institute of Technology, Ahmedabad, May 2008, pp. 1.

[2] Bhaveshkumar P. Patel, Dr. J. M. Prajapati, “Soil-Tool Interaction as a Review for Digging Operation of Mini Hydraulic Excavator”, International Journal of Engineering Science and Technology, Vol. 3 No. 2, February 2011, 894-901.

[3] Bhargav J Gadhvi, B. P. Patel and P. M. Patel, “Development of a Controller for Mini Hydraulic Excavator as a Review”, Proceeding of National Conference on Recent Advances in Manufacturing, SVNIT, Surat, 19th - 21st July, 2010, pp. 198-203.

[4] David A Bradley, Derek W Seward, “Developing Real-Time Autonomous Excavation – The Lucie”, Proceedings of the 34^{th} Conference on Decision & Control, IEEE, New Orleans, LA - December 1995, pp. 3028 – 3033.

[5] C.F. Hofstra, ,A. J. M. van Hemmen, ,S. A. Miedema, J. van Hulsteyn, “Describing the position of backhoe dredge buckets”, 1999.

[6] S. S. Rao, P.K. Bhatti, “Probabilistic approach to manipulator kinematics and dynamics”, Reliability Engineering and System Safety,Elsevier Science, 2001, pp. 47-58.

[7] Hsin-Sheng Lee, Shinn-Liang Chang and Kuo-Huang Lin, “A study of the design, manufacture and remote control of a pneumatic excavator”, International Journal of Mechanical Engineering Education, 2002, pp. 345-361.

[8] Fuad Mrad, M. Asem Abdul-Malak, Salah Sadek, and Ziad Khudr, “Automated excavation in construction using robotics trajectory and envelop generation”, Engineering, Construction and Architectural Management, Blackwell Science Ltd., 2002, pp. 325-335.

[9] Donald Margolis, Taehyun Shim “Instability Due to Interacting Hydraulic and Mechanical Dynamics in Backhoes”, Journal of Dynamic Systems, Measurement and Control, ASME, Vol. 125, 2003, pp. 497-504.