International Journal on Emerging Technologies **4**(1): 108-111(2013)

ISSN No. (Print): 0975-8364
ISSN No. (Online): 2249-3255

Applying Finite Element Method for stress Analysis and Optimization of Lift arm of Tractor

Sadaf Akhtar* and Mohammad Abbas**

**Department of Mechanical Engineering,* Manav *Rachna International University, Faridabad (HR)*
***Department of Mechanical Engineering, Alfalah school of Engineering and Technology, Dhauj, (HR)*

(Received 05 April, 2013, Accepted 6 May, 2013)

ABSTRACT: The main objective of the work is to carry out the failure reduction and also attempt have been made on weight reduction and cost optimization of the lift arm, which is achieved by changing the design and grade of material of the existing Lift arm. The model is created in PRO-E and then imported in Hypermesh10.0 as an IGES file. Here analysis of Lift arm is taken as a case study.

I. INTRODUCTION

In today's world of competition it is very important to find the best optimum design for manufacturing. Optimization of a product means changes which are made on the product to fulfil customer's demands.

We are trying to reduce the stress involved and weight of the component by using CAE tools. The finite element method is a numerical method for solving engineering and mathematical physics problems. The typical use of this method is to solve the problems in the field of stress analysis, heat transfer, fluid flow, and mass transfer and electromagnetic.

In this method, the domain in which the analysis to be carried out is divided into smaller bodies or unit called as finite elements. The properties of each type of finite element is obtained and assembled together and solved as whole to get solution. Based on application, the problems are classified into structural and non-structural problems.

FEM helps tremendously in producing stiffness and strength visualizations and also in minimizing weight, materials, and costs.FEM allows detailed visualization of where structures bend or twist, and indicates the distribution of stresses and displacements. FEM software provides a wide range of simulation options for controlling the complexity of both modelling and analysis of a system. Similarly, the desired level of accuracy required and associated computational time requirements can be managed simultaneously to address most engineering applications. FEM allows entire designs to be constructed, refined, and optimized before the design is manufactured.

FEA basically takes place in the following three steps: Pre-processing, Analysis, Post processing

The software that we are using in this work are Hypermesh10.0 for pre-processing and ABAQUS 6.11.1, for solving and post-processing.

The current work consists of two types of analysis. In the first part of the study a static analysis is carried out for optimization of the lift arm. The static analysis is carried out under Torsional, Transverse and Chassis Bending load case. In the second part a fatigue failure analysis is carried out to check the fatigue life cycle of the optimized lift arm. A fatigue tool is used to determine the life of the lift arm.

II. OVERVIEW

As this work is based on Finite Element Analysis, so it is required that a component on which analysis can be performed and the results obtained can be compared with the existing one for validation. The component chosen for this purpose is a lift arm which finds widespread applications in vehicles like tractors. A lift arm is a component which is connected to the shaft at one end and other end is attached to top links which are used to connect with various types of implements like harvesters, harrower and they form a simple mechanism that lifts elements of different weight. This work carried out the, optimization of lift arm using the ABAQUS software. For optimization it is important to produce an optimized weight material. The optimization procedure is performed in two phases, phase one is used to change the material of the lift arm while phase two is used for some design parameters of the lift arm. Typical design variables for structural analysis are the width, depth and taper ratios of the arm. As Lift arm is subjected to stresses caused by Gravity, Sudden bump load, Typical tractor load cases like chassis bending, Torsional and transversal loads.

This leads to stresses and deformation in lift arm so a structural analysis of lift arm has been carried out. The stress, strain and deformation contours have been plotted.

III. PROBLEM FORMULATION

The goal of the analysis was to evaluate the strength and fatigue life of Lift Arm.

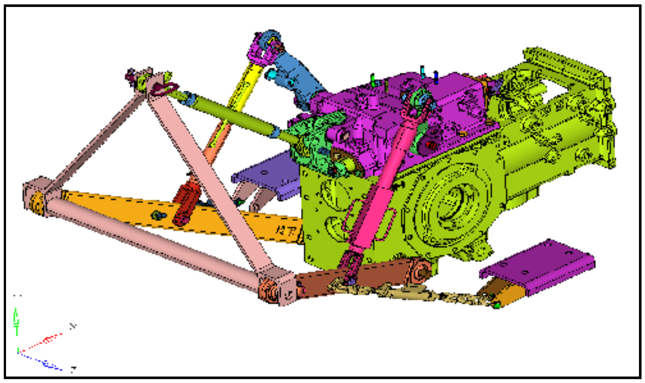


Fig. 1. Hydraulic lift assembly is an .igs format given by designer.

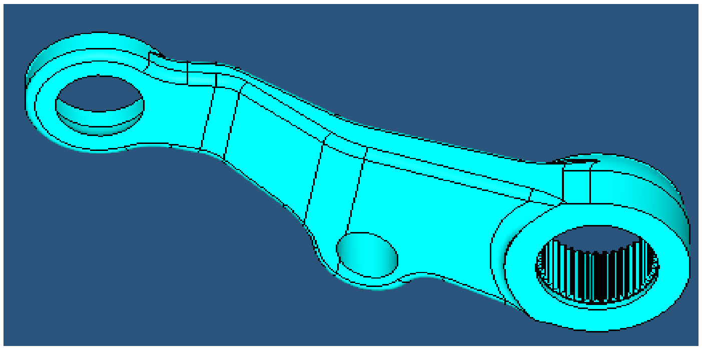


Fig. 2. Design of lift arm in .igs format as given by designer.

Chassis bending and torsion are the two Load Cases considered for analysis.

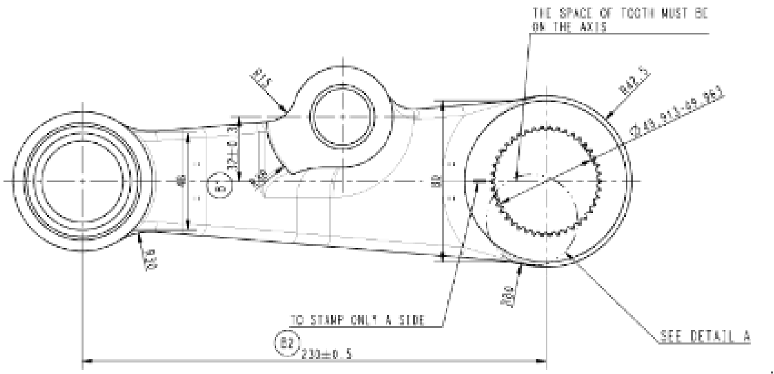


Fig. 3.

The material of the lift arm is ductile cast iron,(MAT2004,GRADE-C) the tensile strength of the material =590MPA.The target life cycle is 5000.

a. Pre-processing

The meshing of the complete hydraulic lift assembly is done using second order tet elements.
Total Number of Nodes:-304830
Total Number of Elements:-197913
Mesh Size:- 4-8mm

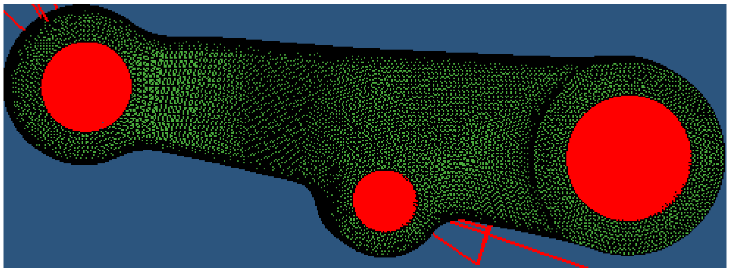


Fig. 4.

b. Analysis

The model has been solved using Abaqus6.11.1 software at different loading conditions and result has been obtained.

Load case 1: Chassis bending Criteria: Fatigue Life Cycle should be > 5000 Cycles

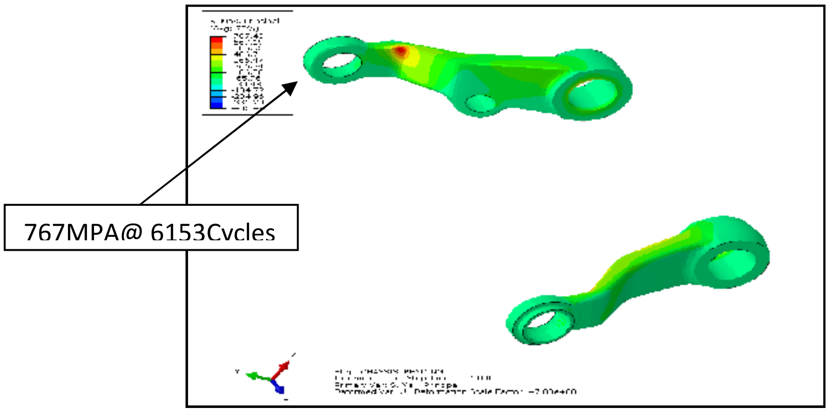


Fig. 5. The criteria was not met.

Load case 2– Torsion Criteria: Fatigue Life Cycle should be > 5000 Cycles

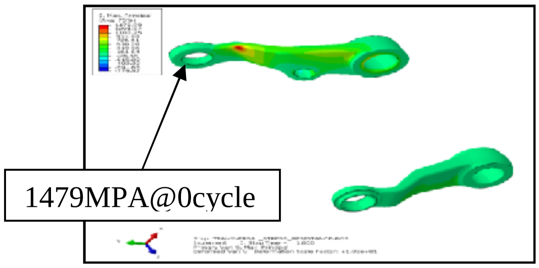
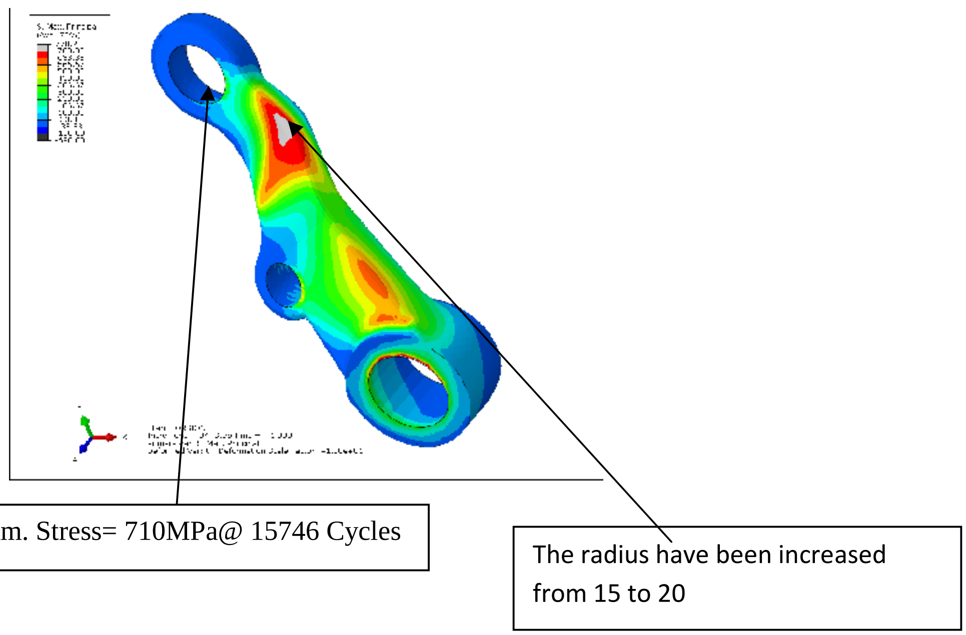


Fig. 6. The criteria was not met.

IV. RESULT

After the analysis for the given target it was noted that the lift arm was failing,it didnot meet the criteria required by the designer. Therefore we decided of changing some design aspects as well we tried changing the grade of the material used.

The new material used is CrNi steel 35CM4BON.The tensile strength of this material is 930MPA.The material has a higher strength than the previous material and is a little cheaper also. We also increased the radius slightly. Analysis was done on the new material using the torsion load case.



V. CONCLUSION

From the above result analysis it can be seen, that the design of lift arm is passing if we change the grade of the material and make slight changes in the design.This research was successful in examining the challenges of applying the analysis and optimization techniques for the interactive stress analysis of a tractor lift arm. This analysis helped in reducing the mass of the lift arm, and considerable amount of money for the company. FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested.

REFERENCES

[1]. Huebner, K.H.; Dewhirst, D.L.; Smith, D.E.; Byron, T.G.The Finite Element Method for Engineers, 4th Ed.; JohnWiley & Sons, Inc.: New York, 2001.

[2]. Rao, S.S. The Finite Element Method in Engineering, 3rd Ed.; Butterworth Heinemann: Boston, 1999.

[3]. Cook, R.D.; Malkus, D.S.; Plesha, M.E.; Witt, R.J.Concepts and Applications of Finite Element Analysis, 4th Ed.; John Wiley & Sons, Inc.: New York, 2002.

[4]. Ward RG. An introduction to the physical chemistry of iron and steel making. London: Arnold; 1962

[5]. Michael J. Ryken, Judy M. Vance, Applying virtual reality techniques to the interactive stress analysis of a tractor lift arm.