Static Force Calculation and Optimization of Excavator Structural Components

[1]Gaurang Rohit, [2]Reena Trivedi, [3]Mitesh Mungla, [4]Atul Deshmukh
[1,2,3] Institute of Technology, Nirma University
[4]TecSo Technology Solution (I) Pvt. Ltd., Vadodara.

Abstract-- **The main objective of this paper is to validate the existing design and redesign the attachments by analyses which are failing under the given operating conditions. During operation of excavator at different positions the stresses induced in the attachments varies. A FEA has been carried out to understand the behavior of the structure. The forces applied in the analysis were calculated by using MathCAD and Design View (DV) software. Finite element analysis of the attachments is done by using Ansys Workbench 11.0.The FEA result of the Dipper/Arm, boom and bucket shows high stress in some regions. For reducing stresses, some design change and also plate thickness of attachments is changed. After this changes the stress in the attachment are coming under allowable limit.**

Index Terms-- **Excavator, Static calculation, Finite element analysis, optimization**

I. Introduction

IN present scenario, development and exploration of the engineering industries are increased. Therefore the attention has been focused on the design of earth moving equipment.

Various Industries like as mining and construction in which earthmoving equipment plays important role for constantly under pressure to improve productivity, efficiency and, safety. With the automation we can increase these all thing. Earlier earth moving components have mechanical components so that the maintenance is also more.

In the tough competition the use of machines is increasing for the earth moving works, considerable attention has been focused on designing of the earth moving equipments. Thus it is very much necessary for the designers to provide not only a equipment of maximum reliability but also of minimum weight and cost, keeping design safe under all loading conditions by careful stress analysis of the machines.

Thus the designer has been charged with the task of providing not only equipment of maximum reliability but also of minimum weight, which can only be justified on the basis of careful stress analysis of the entire structure and a thorough knowledge of material behavior.

During last couple of years companies which manufactures excavator is facing a problem in the attachments and in structural parts. The attachments and other structural parts are failing during operation of excavator, therefore the company is interested in analyzing the attachments as well as structural parts. An attempt is made to find out the displacement and stresses in the excavator attachment for various loading conditions. The main objective is to check whether the attachments and structural parts are in safe condition or not for different loading conditions and in which area the stresses are exceeding the specified limit.

The purpose of this project is to complete a structural analysis of excavator components and optimization. The project consisted of a variety of analysis including static analysis.

In this project, the ultimate goal is to design validation of the excavator components and then if required, using finite element tools optimization will be done**.** To evaluate the stresses in the components for given load cases using Finite Element Analysis.

The forces applied on the models have been calculated from basic machine data and attachment orientation based on max. break-out condition and max. Reach position. The solid model developed in Pro-E and FE Analysis and optimization was performed in Ansys workbench 11.0.

II. Static force Calculation

In calculation of static force analysis for critical operating conditions of the mechanism has been explained. To find the forces at different points of the attachment is very important as it plays a crucial role in the analysis, for getting results close to the actual it is required to have accurate values of forces at all pivot points. The methodology adopted is to find maximum Digging and Break-out force for the given cylinder pressures and this is done using Design View.

There are main two conditions for carried out static calculation. One is maximum torque condition in which the dipper and boom cylinder are keeping at 90 degree. In that we find maximum breakout force at normal to the line of action shown in Fig. 5.1. Second condition is maximum digging force condition in which bucket and boom cylinder are keep at 90 degree. The maximum digging force find out at normal to the line of action.

There are two Load cases which are:
1) Maximum Torque condition
2) Maximum Reach condition

1) Static Force acting on Different Parts

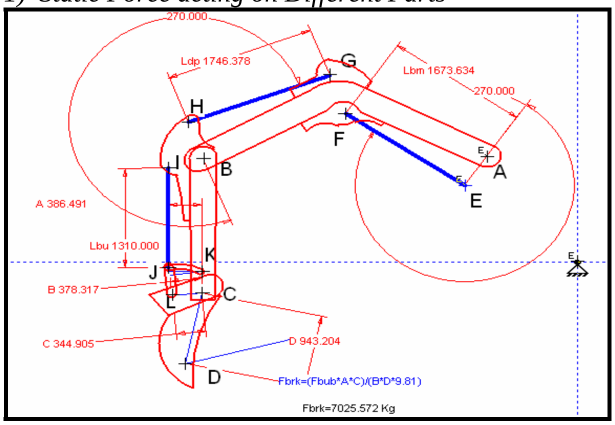


Fig. 1 2-D drawing of attachment in Design View software

TABLE I
FORCE ACTING ON LOWER FRAME

Sr. No.	Part Name	Load (Kg)
1.	Tank	305.02
2.	Rotor	37.49
3.	Cabin + Seat Assembly	250
4.	Engine Assembly	995.79
5.	Counter Weight	950
6.	Weight of the Upper frame	877.424

TABLE II
FORCE ACTING ON UPPER FRAME

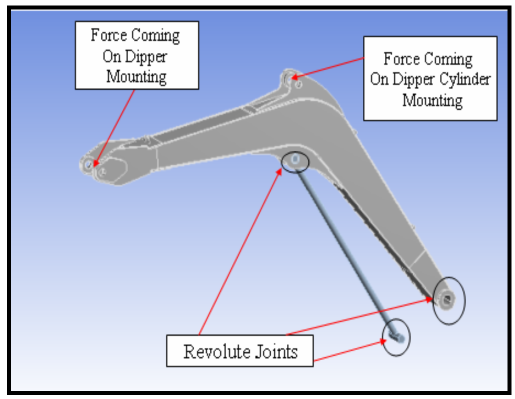
Sr. No.	Part Name	Load (Kg)
1.	Tank	305.02
2.	Rotor	37.49
3.	Cabin + Seat Assembly	250
4.	Engine Assembly	995.79
5.	Counter Weight	950
6.	Force acting by Boom pivot point	
7.	Force acting by Boom cylinder	

TABLE III
STATIC FORCE ACTING ON DIFFERENT COMPONENTS

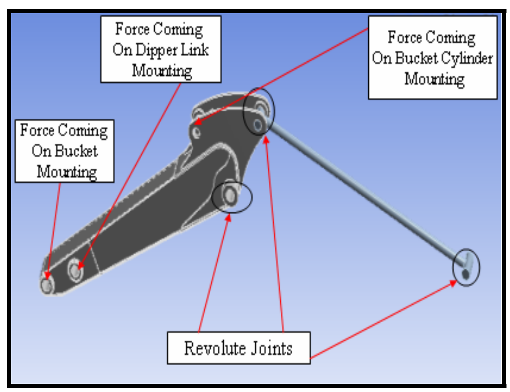
Parts	Pivot Point	Horizontal (X) component(N)	Vertical (Y) Component(N)
Boom	B	-210471.497	-89740.79
	G	177527	67481
	F	179302	-121113
	A	-146358	147485
Dipper	H	-177527	-67481
	I	-131	70405
	K	-8024	1069
	C	-24791	-91363
Bucket	D	-35780	7765
	L	10989.479	-96850.822
Swing Post	Boom Pivot Force	146357.734	-147484.551
	Boom Cylinder Force	-179302.449	121112.918

III. FINITE ELEMENT ANALYSIS

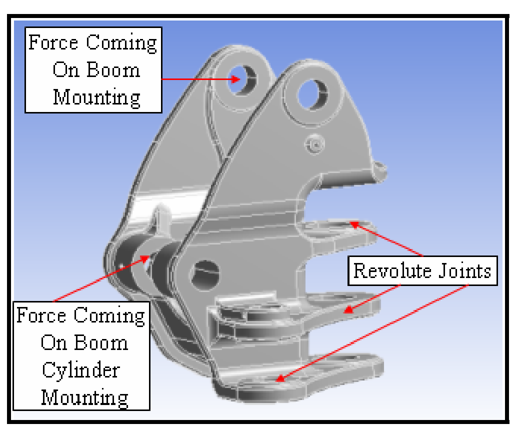
In this section methodology adopted for finite element analysis followed while boundary condition and results of Bucket, Dipper, Boom, Swing post, Upper frame, Lower frame are explained.



(a)

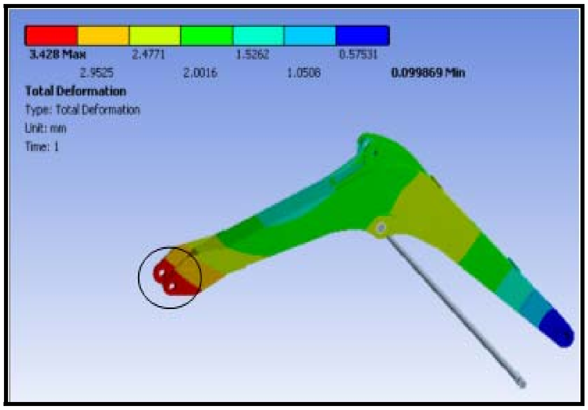


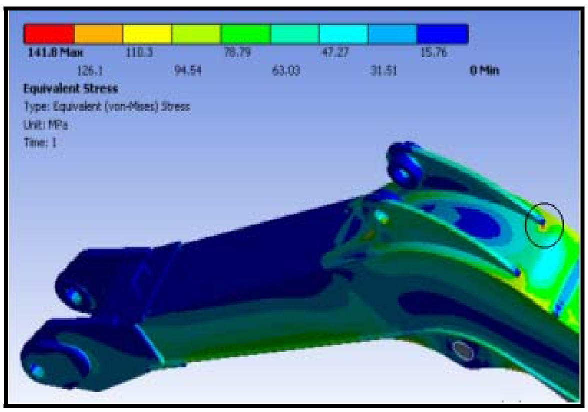
(b)



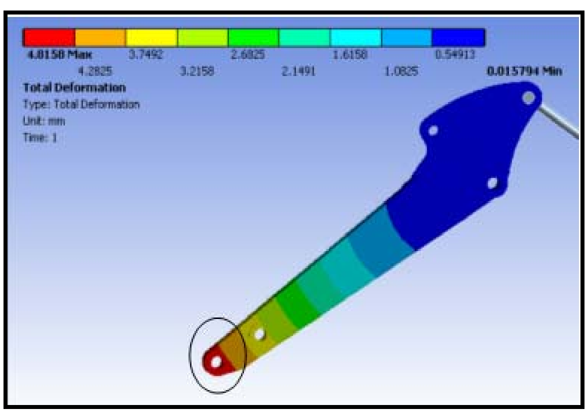
(c)

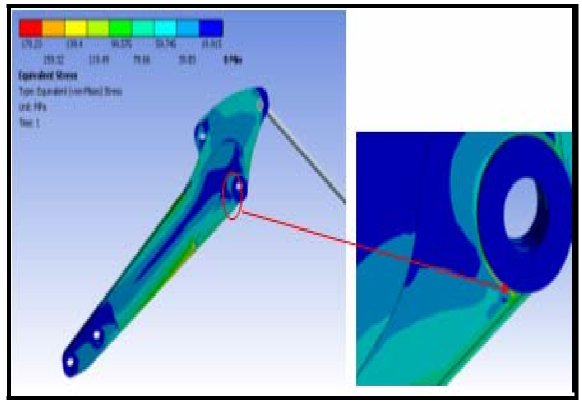
Fig. 2 (a),(b),(c) Boundary condition for Boom, Dipper, Bucket, Swing Post, Upper frame, Lower frame respectively



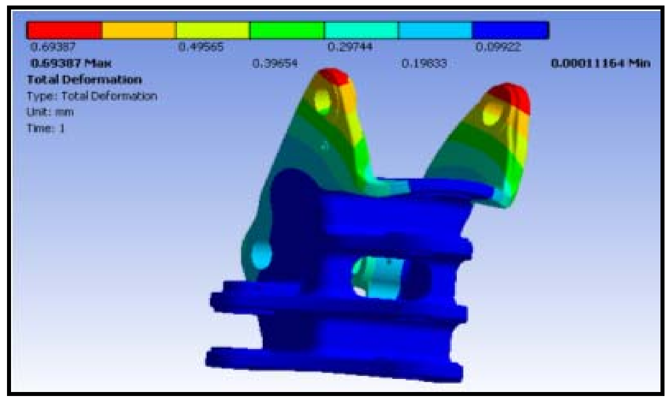


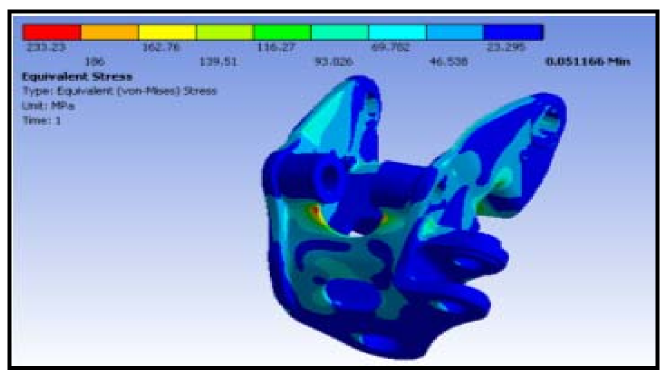
(a)





(b)





(c)

Fig. 3 (a),(b)(c) Maximum displacement & Von Mises stress respectively

In above table we can see the results of the static analysis.

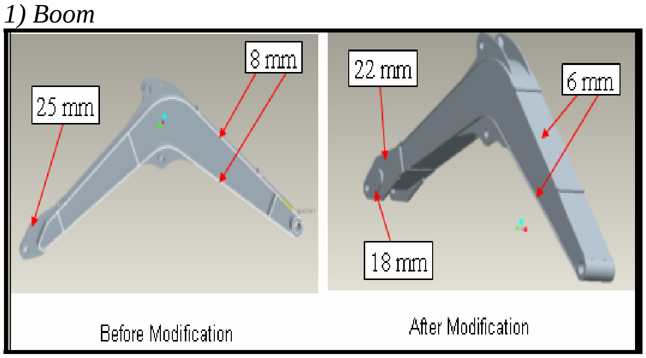
After FEA results I carried out optimization for weight reduction.

IV. Optimization

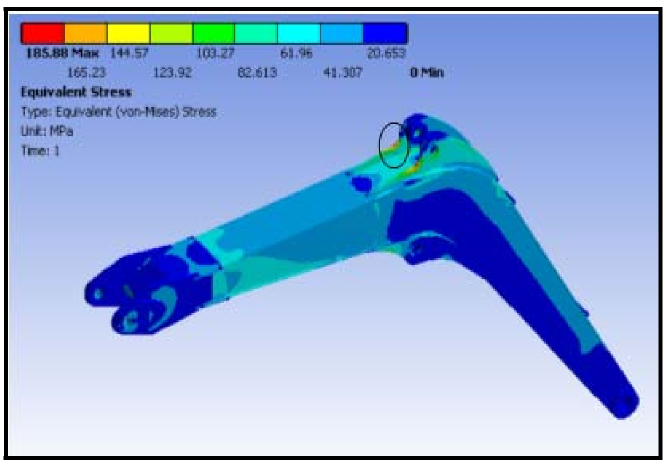
In this section Optimization for dipper, boom, & swing post are discussed. Also, the design modification that has been made after interpreting FEA result is explained.

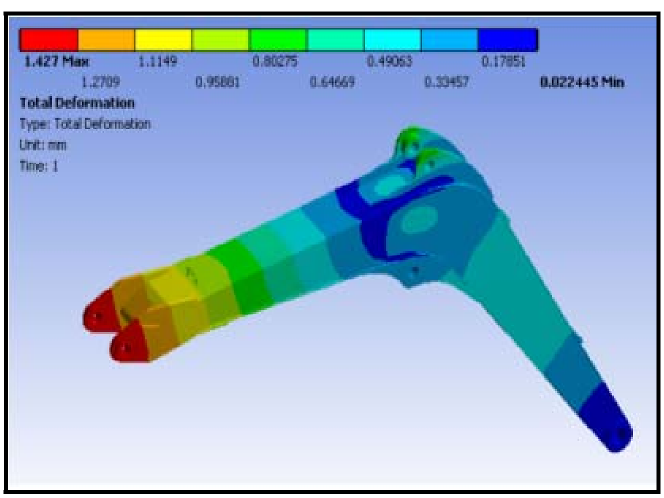
a) Design modification & Results

1) Boom



(a)

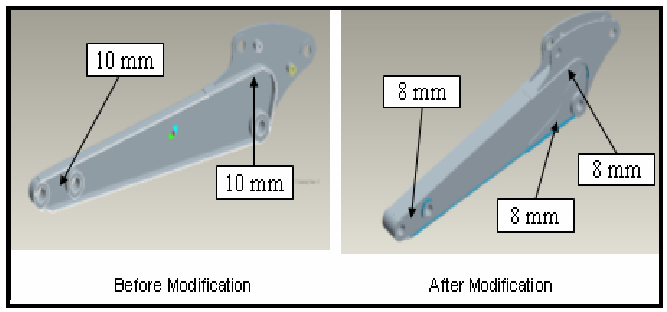




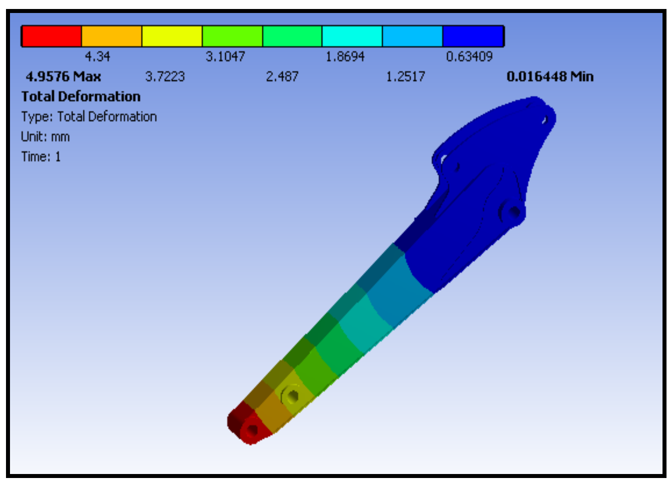
(b)

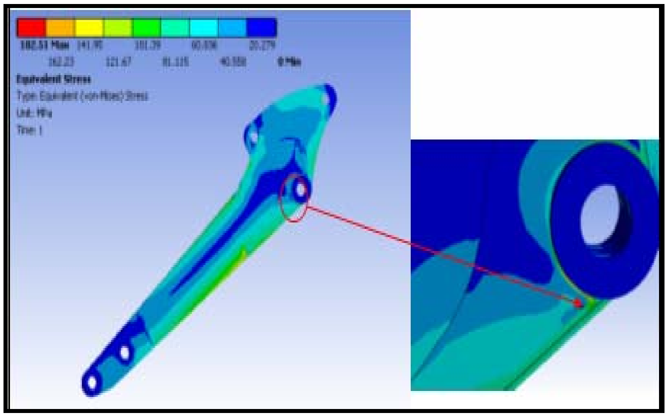
Part Name	Max. Torque Condition Max. Deflection(mm)	Von-Mises Stress(MPa)	Max. Reach condition Max. Deflection(mm)	Von-Mises Stress(MPa)
Boom	2.8485	108.2	1.1337	189.72
Dipper	4.8264	184.44	2.1803	121.8
Bucket	7.1484	275.23	3.8665	148.99
Swing Post	0.6938	233.23	0.5831	196.13
Upper frame	3.2642	235.24	2.3137	199.34
Lower frame	0.0566			70

2) Dipper



(c)



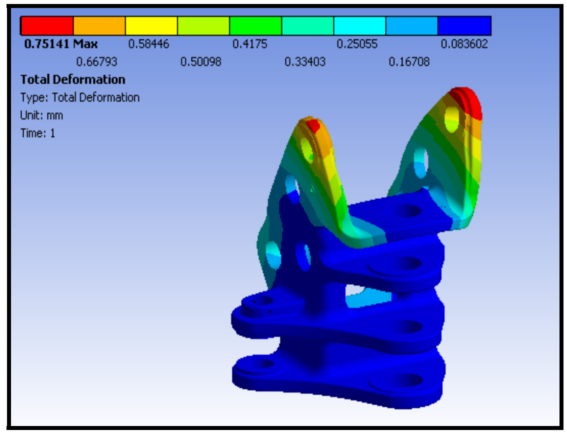


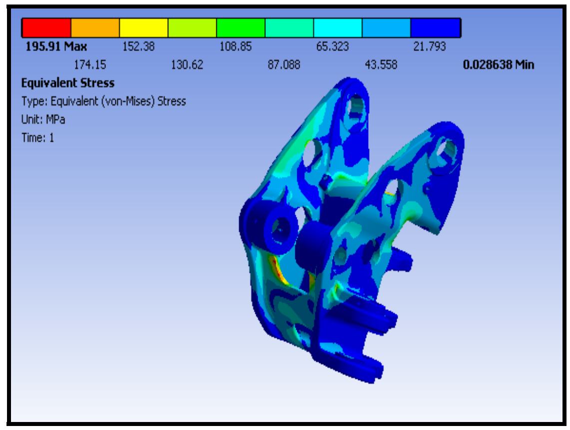
(d)

3) Swingpost

Part Name	Max. Deflection(mm)	Von-Mises Stress(MPa)	Weight Reduction Original weight(Kg)	Reduced weight(Kg)	% Reduction
Boom	1.427	185	341	320	6.16
Dipper	4.8158	179	182	175	3
Swing Post	0.6982	190.36	202	195	3

(e)





(f)

Fig. 4 (a),(b),(c),(d),(e),(f) Modifications & Results of boom, dipper & swing post respectively

Summary:

TABLE IV

V. Conclusion

Finite Element Analysis can be used as a tool to redesign the component if it is already designed by classical design theory. Without making the prototype the loading condition can be simulated and make the necessary changes at the design level, if required for the proper functioning of the component.

In summary we can show the results of analysis before and after optimization respectively. After optimization I reduced the stress as well as weight of the components.

VI. References

[1] Workspace Analysis and Maximal Force Calculation of a Face-Shovel Excavator- by F. Geu Flores A. Kecskemethy A. Pottker

[2] Modelling and Control Of Excavator Dynamic During Digging Operation- by A. J Koivo, M. Thoma, E Kocaoglan and J. Andrade–Cetto

[3] Structural Optimization based on Discrete Element- by Shinji Nishiwaki

[4] Daryl L. Logan, "A First Course in the Finite Element Method"

[1] Optimal control of an excavator bucket positioning- by E. Budny, M. Chłosta, W. Gutkowski