JANUARY 2016

M.Sc. in Mechanical Engineering

ERDEM ÖZSARAÇ

UNIVERSITY OF GAZİANTEP

GRADUATE SCHOOL OF

NATURAL & APPLIED SCIENCES

STRESS ANALYSIS AND IMPROVEMENT OF BOOM AND ARM OF AN EXCAVATOR

M.Sc. THESIS

IN

MECHANICAL ENGINEERING

BY

ERDEM ÖZSARAÇ

JANUARY 2016

Stress Analysis and Improvement of Boom and Arm of an Excavator

M.Sc. Thesis

in

Mechanical Engineering

University of Gaziantep

Supervisor

Assoc. Prof. Dr. M. Akif KÜTÜK

By

Erdem ÖZSARAÇ

January 2016

REPUBLIC OF TURKEY
UNIVERSITY OF GAZİANTEP
GRADUATE SCHOOL OF NATURAL & APPLIED SCIENCES
MECHANICAL ENGINEERING

Name of the thesis: Stress Analysis and Improvement of Boom and Arm of an Excavator

Name of student : Erdem ÖZSARAÇ

Exam Date : 28.01.2016

Approval of the Graduate School of Natural and Applied Sciences

Prof. Dr. Metin BEDİR
Director

I certify that this thesis satisfies all the requirements as a thesis for the degree of Master of Science.

Prof. Dr. M. Sait SÖYLEMEZ
Head of Department

This is to certify that we have read this thesis and that in our opinion it is fully adequate, in scope and quality, as a thesis for the degree of Master of Science.

Assoc. Prof. Dr. M.Akif KÜTÜK
Supervisor

Examining Committee Members	Signature
Assoc. Prof. Dr. Ahmet ERKLİĞ	....
Assoc. Prof. Dr. M. Akif KÜTÜK	....
Assist. Prof. Dr. Erdoğan KANCA	....

I hereby declare that all information in this document has been obtained and presented in accordance with academic rules and ethical conduct. I also declare that, as required by these rules and conduct, I have cited and referenced all material and results that are not original to this work.

Erdem ÖZSARAÇ

ABSTRACT

STRESS ANALYSIS AND IMPROVEMENT OF BOOM AND ARM OF AN EXCAVATOR

ÖZSARAÇ, Erdem
M.Sc. in Mechanical Engineering
Supervisor: Assoc. Prof. Dr. M. Akif KÜTÜK
January 2016, 80 pages

The aim of this study is to analyze boom and arm of excavator and to improve the stresses depending on forces. Besides these stresses remain below the limit values as desires, it is aimed to provide using optimum material. Primarily force analyses is finished. Then the front attachment is modeled and simulated according to critical position of machine. The improvement is applied according to the results. Finite element method (FEM) is used to the engineering problem as a numerical solution methods. Simxpert software program is used in finite element analysis.

Key Words: Excavator, Finite element method, Simxpert

ÖZET

EKSKAVATÖRÜN BOM VE KOLUNUN GERİLME ANALİZİ VE GELİŞTİRİLMESİ

ÖZSARAÇ, Erdem
Yüksek Lisans Tezi, Makine Mühendisliği Bölümü
Tez Yöneticisi: Doç. Dr. M. Akif KÜTÜK
Ocak 2016, 80 sayfa

Bu çalışma, kazıcı iş makinesinin bom ve kol grubunun analiz edilmesi, kuvvetlere bağlı olarak oluşan gerilmelerin iyileştirilmesi için yapılmıştır. Ürünün maruz kaldığı kuvvetlere bağlı olarak istenilen sınır değerinin altında kalmasının sağlanmasının yanı sıra, ürünün bu yükleri optimum hammadde kullanımı ile karşılaması istenmiştir. Öncelikle kuvvet analizi tamamlanmıştır. Bundan sonra ön ataşman, makinenin kritik konumuna göre modellenmiştir ve simule edilmiştir. Sonuçlara göre iyileştirmeler uygulanmıştır. Bu mühendislik problemine sayısal bir çözüm yöntemi olarak Sonlu Elemanlar Yöntemi (SEY) kullanılmıştır. Sonlu elemanlar analizleri Simxpert paket programı ile yapılmıştır.

Anahtar Kelimeler: Kazıcı iş makinesi, Sonlu elemanlar yöntemi, Simxpert.

ACKNOWLEDGEMENTS

The author expresses sincere appreciation to Assoc. Prof. Dr. M. Akif KÜTÜK for his guidance and insight throughout the research. Also, I want to thank my amazing parents. My mother and father support me every time. Last, I wish especially thank my family. My child and my precious wife have made possible and enjoyable in the study.

CONTENT

ABSTRACT....................v
ÖZET....................vi
ACKNOWLEDGEMENTS....................vii
CONTENTS....................viii
LIST OF TABLES....................x
LIST OF FIGURES....................xi
ABBREVIATION....................xv
CHAPTER 1 INTRODUCTION....................1
1.1 General Properties of Excavator....................2
1.2 Excavator Front Attachment....................3
CHAPTER 2 LITERATURE SURVEY....................6
CHAPTER 3 KINEMATIC FORCE ANALYSIS
3.1 Introduction....................12
3.2 Position Analysis....................13
3.2.1 Boom Position....................14
3.2.2 Arm Position....................14
3.2.3 Bucket Position....................15
3.3 Breakout Force Calculation....................17
3.3.1 Force Calculation According to Arm Cylinder....................18
3.3.2 Force Calculation According to Bucket Cylinder....................19
CHAPTER 4 FINITE ELEMENT METHOD AND STATIC ANALYSIS OF BOOM
4.1 Introduction....................24
4.2 Finite Element Method....................24
4.3 Simxpert FEA Program....................26
4.3.1 Introduction....................26
4.3.2 Program Capability....................26
4.4 Static Analysis of Boom....................28
4.4.1 Assumptions....................28
4.4.2 Static Loads....................29
4.4.3 Mesh Properties....................29
4.4.4 Boundary Conditions....................30
4.4.5 Material Properties....................31
4.5 Boom Analysis Results....................31
4.6 Conclusion....................46
CHAPTER 5 STATIC ANALYSIS OF ARM
5.1 Introduction....................47
5.1.1 Assumptions....................47
5.1.2 Static Loads....................47
5.1.3 Mesh Properties....................48
5.1.4 Boundary Conditions....................49
5.1.5 Material Properties....................49
5.2 Arm Analysis Results....................49

5.3 Conclusion....60
CHAPTER 6 DYNAMIC ANALYSIS OF FRONT ATACHMENT
6.1 Introduction....61
6.2 Dynamic Analysis....62
6.2.1 Assumptions....62
6.2.2 Dynamic Loads....63
6.2.3 Mesh Properties....65
6.2.4 Boundary Conditions....66
6.3 Dynamic Analysis Results....66
6.4 Conclusion....76
CHAPTER 7 DISCUSSION....77
FUTURE WORKS....79
REFERENCES....80

LIST OF TABLES

page

3.1 Cylinder forces..23

3.2 Breakout forces..23

4.1 Properties of St52-3...31

4.2 Properties of SAE 1040 steel...31

4.3 Summary table of results...46

5.3 Summary table of results...60

6.1 Cycle time results..65

6.2 Summary table of results...76

LIST OF FIGURES

page

1.1 Front attachment group of excavator....................2

2.1 Schematization of the arm, Solazzi [8]....................8

2.2 Distribution of von mises stresses in the undercarriage....................9

3.1 Figure 3.1 Mechanism of an excavator. (a) Front attachment,
(b) Four bar mechanism....................13

3.2 Mechanism parameters of the boom....................14

3.3 Mechanism parameters of the arm....................15

3.4 Mechanism parameters of the bucket....................16

3.5 Sectional view of cylinder....................17

3.6 Free body diagram while arm cylinder is active....................18

3.7 Free body diagram while bucket cylinder is active....................19

3.8 Maximum bucket breakout force....................22

3.9 Maximum arm breakout force....................22

4.1 Joint forces on the boom....................29

4.2 Application of loads....................29

4.3 Mesh of boom and material properties....................30

4.4 General view of boom model....................32

4.5 General constraints....................32

4.6 First analysis result. (a) Stress distribution of boom
(b) Stress zone of boom..33

4.7 First revision of the boom...34

4.8 Figure 4.8 First revision analysis result.
(a) Stress distribution of revision 1 boom
(b) Stress zone of revision 1 boom...34

4.9 Stress distribution of rev. 2 boom...35

4.10 Third revision of the boom...36

4.11 Third revision analysis result. (a) Stress distribution of revision 3 boom
(b) Stress zone of revision 3 boom...36

4.12 Fourth revision of the boom...37

4.13 Fourth revision analysis result. (a) Stress distribution of revision 4 boom
(b) Stress zone of revision 4 boom...38

4.14 Fifth revision of the boom..39

4.15 Fifth revision analysis result. (a) Stress distribution of revision 5 boom
(b) Stress zone of revision 5 boom...39

4.16 Sixth revision of the boom...40

4.17 Sixth revision analysis result. (a) Stress distribution of revision 6 boom
(b) Stress zone of revision 6 boom...40

4.18 Eighth-Ninth revision of the boom...41

4.19 Eight and ninth revisions analysis results.
(a) Stress zone of revision 8 boom
(b) Stress zone of revision 9 boom...42

4.20 Tenth revision of the boom...43

4.21 Tenth and eleventh revision analysis result.
(a) Stress zone of revision 10 boom
(b) Stress zone of revision 11 boom...43

4.22 Twelfth revision of the boom..44

4.23 Stress distribution of rev. 12-13 boom..45

4.24 Comparing of cases..45

5.1 Joint forces on the arm...48

5.2 Application of loads..48

5.3 Mesh of arm...49

5.4 General view of arm model..50

5.5 First analysis result. (a) Stress distribution of arm
(b) Stress zone of arm..50

5.6 First revision of the arm...51

5.7 First revision analysis result. (a) Stress distribution of revision 1 arm
(b) Stress zone of revision 1 arm...52

5.8 Stress distribution of revision 2 arm...53

5.9 Third revision of arm..54

5.10 Stress distribution of revision 3 arm...54

5.11 Fourth revision of arm...55

5.12 Stress distribution of revision 4 arm...55

5.13 Specific element on thinner plate..56

5.14 Sixth revision of arm...56

5.15 Sixth revision of arm. (a) Stress zone of revision 6 arm
(b) Specific element on revision 6 arm...57

5.16 Seventh revision of arm...58

5.17 Seventh revision of arm. (a) Stress zone of revision 7 arm
(b) Specific element on revision 7 arm...58

5.18 Stress distribution of revision 8 arm...59

6.1 General view of dynamic model..62

6.2 Application of force..63

6.3 Boom cylinder joint motion..64

6.4 Arm cylinder joint motion...64

6.5 Results of first dynamic analysis. (a) Stress zone on the arm side plate
(b) Stress zone on the boom upper plate
(c) Stress zone on the boom side plate...67

6.6 First revision of boom and arm.
(a) Revision 1 on the arm for dynamic analyze
(b) Revision 1 on the boom for dynamic analyze...68

6.7 Results of first revision dynamic analysis.

(a) Stress zone on the revision 1 arm side plate
(b) Stress zone on the revision 1 boom upper plate
(c) Stress zone on the revision 1 boom side plate..69

6.8 Second revision of boom and arm.
(a) Revision 2 on the arm for dynamic analyze
(b) Revision 2 on the boom for dynamic analyze..70

6.9 Results of second revision dynamic analysis.
(a) Stress zone on the revision 2 arm side plate
(b) Stress zone on the revision 2 boom upper plate
(c) Stress zone on the revision 2 boom side plate..71

6.10 Third revision of boom and arm.
(a) Revision 3 on the arm for dynamic analyze
(b) Revision 3 on the boom for dynamic analyze..72

6.11 Results of third revision dynamic analysis.
(a) Stress zone on the revision 3 arm side plate
(b) Stress zone on the revision 3 boom upper plate
(c) Stress zone on the revision 3 boom side plate..73

6.12 Rev. 4 on the arm and boom for dynamic analyze..74

6.13 Results of fourth revision dynamic analysis.
(a) Stress zone on the revision 4 boom upper plate
(b) Stress zone on the revision 4 boom side plate..75

6.14 Comparing of cases...76

ABBREVIATION

FEM: Finite element method

FEA: Finite element analysis

CAD: Computer aided design

DOF: Degree of freedom

D_i: Inside diameter

D_o: Outside diameter

F_{arm}: Force from arm cylinder

P: Pressure

CAE: Computer aided engineering

BCs: Boundary conditions

SPCs: Single point constraints

MPCs: Multi point constraints

ICA: Initial condition analysis

KA: Kinematic analysis

RBE: Rigid bar element

MBD: Multi body dynamic

CHAPTER 1

INTRODUCTION

Construction machines are used in agriculture, industry and natural defense. Also different kinds of equipment can be mounted on it according to work purpose. But these machines are not used for transferring people, animals or freight [1].

Construction machines are generally classified as three main groups. These are;

I. Earth moving machines: Excavator, highwall miner, wheel loader, dozer, grader etc.
II. Industrial machines: Forklift, cleaning machine etc.
III. Road and agricultural machines: Scraper, road reclaimer, compactor, asphalt paver etc.

Industrial machines are generally used for lifting purpose. But there are different machines in this group. Cleaning, concrete mixer, trucks are examples of these. Forklift, stackers, telescopic handler, cranes or elevated work platforms are examples of lifting machines. These machines can be used with different attachments.

Road and agricultural machines are used for highways, agriculture or mining. These machines can be used for fixing rough surfaces, paving, pipelayer, pouring ore from a warehouse or compacting a material like asphalt. Agricultural machines are used for cultivating the field, harvesting, tillage, lumbering. These machines can be used with different attachments.

Earth moving machines can be used for excavating, loading, crushing, demolition or special purpose applications. Backhoe loader, dozer, loaders, excavator are machines used for these purposes. Excavator is a versatile machine. It can be used for digging, loading or as a breaker, grabber, compactor, pallet forks, and auger.

All of these different possible working areas of the excavator make it important. Therefore, this study is made on excavator.

1.1 General Properties of Excavator

Excavator (Figure 1.1) is generally used as heavy construction machine. It is commonly known as hydraulic excavator or crawler excavator.

The classifications are made according to operational weight of machine. Excavator is generally classified as mini, medium and large excavators.

Hydraulic excavator has basically mechanical bodies and hydraulic system. It has welding constructions (chassis, boom, arm, links, bucket etc.), cab, engine, pump, hydraulic actuators, hydraulic motors and idler tracks.

Excavator does not take the movement directly from the diesel engine contrary to the car or truck. Engine produces mechanical energy, the energy is transmitted to variable displacement hydraulic pump via transfer mechanism. The pump works like a heart and transmits the fluid which is feeded from oil tank into various actuator that drive the motion of the excavator. Control valves redirect the oil. The pressurized oil provides the movement of boom, arm or slew. The joystick, placed in the cab, provides controls of the required movement.

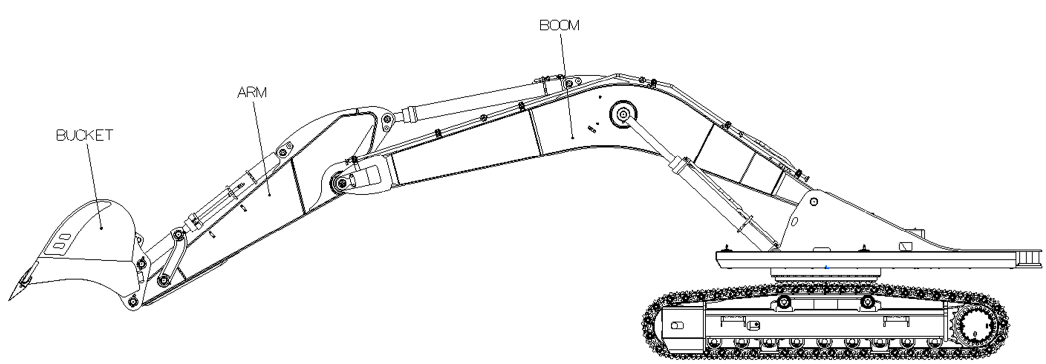


Figure 1.1 Front attachment group of excavator

Excavators moves by means of steel-rubber tracks or it may be tired. It has two frames lower and upper ones. Movement of upper frame is independent of lower frame. The excavators have different using purposes by means of attachments. Digging bucket, auger, grapple, hammer, and ripper may be used as an attachment.

All earth moving machines have been developed to reduce the work done by human and animal power. These machines also ensure a certain standard and quality of work. In the present day, it is very important how to do a work and the cost of that work. There are many parameters affecting the product cost in the earth moving machines. Tooling-mold costs, raw material cost, component selection are example of these parameters. Therefore, the design of a product to be competitive and to be less energy consuming is important. Some of the parameters that affect the energy consumption are;

- Component type
- Design of hydraulic system
- Acclimatization performance
- Weight of attachment

Power is mainly produced by diesel engine for many years. Nowadays the studies have focused on hybrid machines. Temperature and layout of hydraulic parts affect hydraulic system performance. Another important issue is mass of construction or welding parts. Durability of the machines are important that is provided with the use of more material but energy consumption increases with an increase of mass. This contradiction forces the designers to optimization topic. The mass also affects product cost directly and front attachment performance. This study focused on the improvement and lightening of the front attachment of an excavator.

1.2 Excavator Front Attachment

Excavator can be used for different applications. These features are usually achieved by using different attachments. The attachments are mounted in the arm and links. Bucket is assumed as the design attachment. It constitutes one of the major work areas of the excavator. Forces created by the hydraulic cylinders effect on the arm and boom pivot points in the use of this attachment. In this case, strength of boom and arm is very important. Hydraulic cylinder forces will be used in the structure analysis. Finite element method will be used as an analysis method.

Finite element analysis (FEA) is a method that is very powerful and has a wide application area for strength analysis. Strength calculation of complex parts takes a long time. For this reason in practice, it is important to perform the calculations competitively in a short time. It is necessary to simulate the real working conditions when desired to improve a structure and working characteristics (breakout force, digging depth, digging reach at ground etc.) of machine. So, dynamic characteristic of the machine should be set up and included in the improvement process.

First of all, critical loading conditions should be determined according to different piston positions. It is necessary for static force analysis. Then, the model of the parts to be analyzed are created helping with computer aided drawing (CAD) before starting to FEA. In this thesis, models are generated by a commercial CAD program. After generating of solid bodies, the bodies are imported to a commercial finite element program which is called Simxpert. This software uses an interface Nastran for structural analysis and Adams for system dynamics analysis. After the importing process, the model is made suitable for geometric forming. Then, boundary conditions and connection properties are applied. Material properties are assigned.

Connection properties are described according to degree of freedom (DOF). These connections can be defined as fixed, bushing, cylindrical or translational properties. When the model is run as statically, Nastran solver is used. When the model is run as dynamically, Adams motion interface is used.

Firstly, the modeling of boom and arm are prepared. So, solid parts, 2D model geometries and finite element model are prepared. Then, they are analyzed statically. Stress locations are determined. Some improvements and optimizations are applied and the results are compared.

The front attachment components are simulated according to working conditions. The boom and the arm are formed as elastic body. Then, dynamic model is run. The critical stress locations are determined and the improvements are applied.

The aim of this study is to improve and optimize strength of the boom and arm of an excavator and to create a competitive, durable and reliable product. Obtaining a product with minimum cost and maximum performance is important to keep competitiveness.

CHAPTER 2

LITERATURE SURVEY

In this chapter, a literature survey which is related to finite element analysis of earth moving machines is summarized. Excavate, finite element analyze etc. are used in research as keywords.

In recent years, finite element analysis (FEA) is begun to be used frequently. Especially, when analysis of complex parts is important, the FEA greatly simplifies and accelerates calculation processes. Some studies have been made on this topic. These studies were worked on parts under static loads. Because of this reason, it was decided to make a study explaining the difference between static and dynamic ladings.

Oyman [2] worked on the excavator front attachment which consists of boom, arm, bucket and links. He has calculated the forces which effect boom, stick and bucket according to different working conditions and made strength controls by means of Soderberg diagram. If the value is under the diagram, it has been benefitted from Wöhler diagram. It was calculated for lifetime of critical parts. Lifetime was calculated nearly 20 years.

Fıçıcı et al. [3] performed the verification of experimental results with FEA results in constructions of earth moving machines. The experimental strength analysis has made at same position and same boundary conditions. The rosette gauge has used for verifying. Strain gauges have placed to locations which determined as a result of finite element analysis for 35 points. At the end of study, the experimental results have found to be compatible with FEA results.

Yeter [4] analyzed with back (boom, arm, bucket and links) and front (loader, bucket and links) groups of backhoe loader. He used finite element method. Strength and working life of a backhoe-loader parts depend on analysis results in the structure of machine. The 3D models of structures, e.g. loader, bucket, have designed by CAD program and finite element analysis of structures have made by ANSYS Workbench.

Özer [5] studied with structure analysis of backhoe group (boom, arm and links). He studied at different backhoe designs on the same machine. These designs had different working ranges. Different working ranges mean different breakout forces, different reach ground level or different maximum digging depth. Optional attachment properties are determined according to analysis results of different working ranges. The different models which designed by CAD program was analyzed. At the end of analysis, dangerous area was found. Some improvements were applied and safest designs were selected.

İpek [6] aimed to improve loader mechanisms of backhoe-loader machine. Kinematic and force analysis have made. Genetic Algorithm has been used for this improvement by means of Excel macros which determine needs of mechanism problem. Arm breakout force, bucket breakout force, lifting capacity, digging depth and loading height have calculated by this algorithm. As a result of study, arm breakout force, bucket breakout force and lifting capacity have increased.

Sharma [7] made strength analysis of boom and arm of excavator. To make force analysis, Mathcad and Design view have been used. Hypermesh program has been preferred to create finite element mesh model. The Ideas program has been used to apply forces. As a result of this study, the arm and boom have improved.

Yener [8] studied with optimization of excavator boom and designed a program which correlate the user and FEA program, MSC Marc-Mentat. Delphi platform has used for this program according to design parameters. Suitable booms have been selected with different working conditions according to maximum allowable stress and optimum mass. At the end of study, optimum boom has been designed depending on the boundary conditions.

Solazzi [9] studied modification of boom and arm of an excavator (Figure 2.1). Different material was used to boom and arm to lighten the front attachment and at the same time increase the load capacity of bucket. At the end of study, the weight of arm has decreased from 2050 kg to 1080 kg. The load capacity has increased from 1 m^3 to 1.35 m^3. The material price has increased but the productivity per hour also increased.

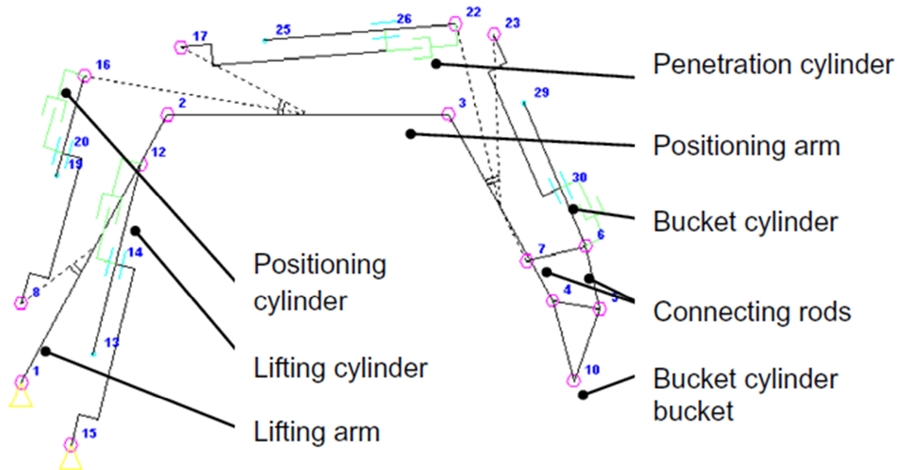


Figure 2.1 Schematization of the arm, Solazzi [8]

Rusinski et al. [10] performed analysis of a mine's loader boom crack. Numerical and experimental approaches have applied. Firstly, a geometrical boom model was created. Then the FEM has performed. Shell elements have been formed. Beam elements have been created for actuators, connectors, bolts and pins. After the structure analysis model has finished, it has run for 18 different cases. The damaged boom has been evaluated by means of macroscope and fractography. At the end of this study, the maximum combined stresses in cases where the jib boom has subjected to torsion forces has concluded to be caused by welding process and found that the loader boom has limited fatigue life.

Bosnjak et al. [11] studied with failure analysis of bucket wheel excavator (BWE). FEM has been applied on tie rod support to see the result of overloading. Firstly, FEM model was prepared by them. Mesh elements were created. DOF was defined. Then external loads was calculated. Also an experimental investigation was made according to material properties and welding process by means of scanning electron microscope (SEM) and light microscope (LM). It was conclusive that the failure of the support was the consequence of superposition of the negative effects caused by inadequate shaping and dimensioning of the support assembly for given load conditions, as well as influences of mentioned defects of the metal weld structure.

Bosnjak et al. [12] also studied analysis of lower frame in the BWE. The cracks

occurred in the undercarriage. Repair and reconstruction were applied. The lower frame has reinforced based on intuition, not research. Then, the undercarriage external load has been determined according to three critical configurations. According to this tree case, FEA has been done and the results have evaluated. The maximum critical stress (Figure 2.2) has occurred in the second case, the lowest stress has occurred in the third case. Redesign has been made by referring to these results.

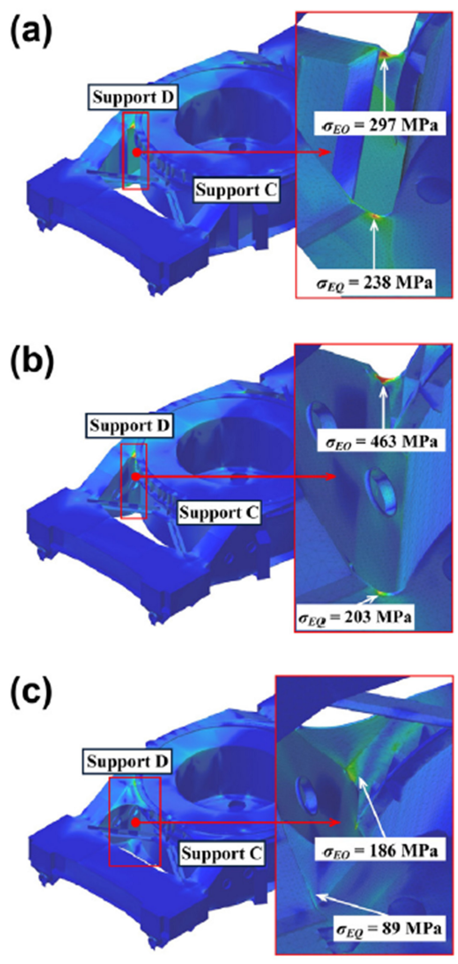


Figure 2.2 Distribution of von mises stresses in the undercarriage

Özbayramoğlu [13] studied with shape optimization of wheel excavator lower chassis. He has developed a computer program to create a parametric FEA of a structure using a commercial program. The parameters have been created in Microsoft Excel. The results have been given with the help of python. The Smart designer has been improved for shape optimization. The shape optimization program used by Uzer [14] has been modified. Then, he has also observed that the Von Mises stresses improved 15 percent without increasing the weight of the structure.

Özünlü [15] studied analysis of excavator and simulated movement of the machine. He has calculated the forces by Recursive Newton-Euler (RNE) and used fourth degree Runge-Kutta method for numerical analysis. A computer program which contains kinematic analysis of mechanics, dynamic analysis of bodies and numerical analysis for simulation of the model has been developed. After the mathematical model has been generated, then dynamic analysis has been performed by RNE.

Kocabıyık [16] performed analysis of excavator cab. FOPS was applied on cab which designed by CAD program. ANSYS software program assisted to him for FEA. AUTODYN has been used to make calculations and monitorize the results. The applying energy was up to 11600 joule. He calculated this energy value according to ISO (3449, 3471, 3164 etc). Maximum displacement was 99.8 mm. This result didn’t effect deflection limiting volume (DLV). At the end of this work, operator cab passed the FOPS test.

Kılıç [17] aimed to develop dynamics model loader system of a backhoe loader. Rigid bodies, joints in the loader mechanism and hydraulic components in the hydraulic system have been modelled and analyzed in MATLAB/Simulink. The simulation results and values which measured on physical system have been compared. The results was consistent with each other. It has been proved that the model can be used to calculate the dynamic loads of the backhoe loader machine. Also, costs could be decreased by using this model in design steps.

Conclusion

Literature survey showed that some studies on earth moving machines are done. These studies focused on structural analysis (static), improvements and optimizations of different parts of the machines. The main purpose of this thesis is to make structural analysis of a new type of excavator just before production to be static and dynamic.

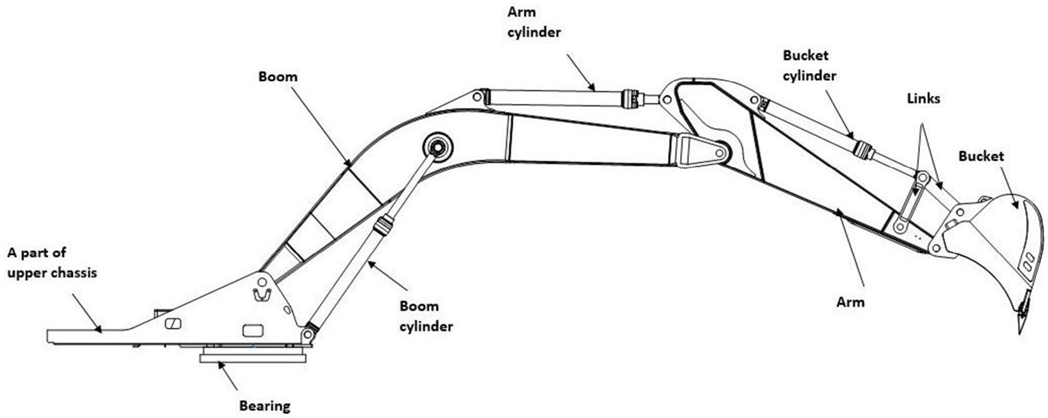
CHAPTER 3

KINEMATIC FORCE ANALYSIS

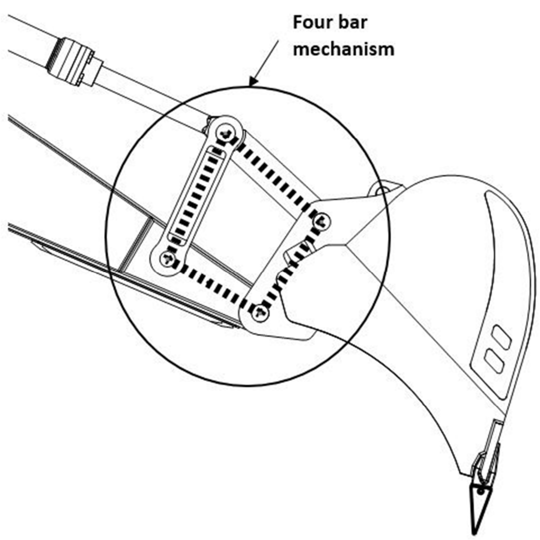
3.1 Introduction

Excavator is a heavy construction machine that is preferred in many applications. Motion capacity of the machine is high due to four DOF. Operator can control the DOF's according to needs of operation. Control of the DOF's is provided by three different cylinders and a rotating gear chain (swing bearing) as shown in Figure 3.1a.

Different motions are provided by hydraulic energy. Hydraulic energy is transported by hydraulic oil. Oil is directed to hydraulic motor or cylinders with the aid of main control valve. For the rotating motion, hydraulic energy is converted to mechanical energy by hydraulic motor. It creates circular motion. Reduction gear box are integrated with hydraulic motor. Reduction gear box is used to decrease circular motion velocity and to increase torque. Pinion gear is at the end of gear box. It rotates swing bearing. So, the upper chassis is rotated independently from undercarriage. For the motion of cylinders, hydraulic energy is converted to mechanical energy by slider-crank mechanism. The rotation of the boom is driven by two simultaneous cylinders. These cylinders are called as boom cylinders. They work as slider-crank mechanism relative to the frame. The rotation of the arm is driven by a different cylinder. It is called as arm cylinder. It works slider-crank mechanism relative to the boom. The rotation of bucket is driven by a cylinder. This cylinder is called as bucket cylinder. It works slider-crank mechanism relative to the arm. The bucket rotation also uses a four bar mechanism (Figure 3.1b).



(a)



(b)

Figure 3.1 Mechanism of an excavator. (a) Front attachment, (b) Four bar mechanism.

3.2 Position analysis

The position analysis is very important step to analyze front attachment. Analysis can be performed for an infinite number of positions. But the important thing is to find the critical position. Critical position means the forces/moments have largest values. When force is applied to point of moment perpendicularly, maximum force that can be applied will be obtained. It is described below how to determine the critical position.

3.2.1 Boom position

Boom cylinder position is first parameter while the boom is analyzing. The point of T and S are pivot points of the boom. Piston side of boom cylinder is on the upper frame and its rod side is on the boom. a1 has a length between T and R points and it changes according to the hydraulic piston cylinder positions (Figure 3.2).

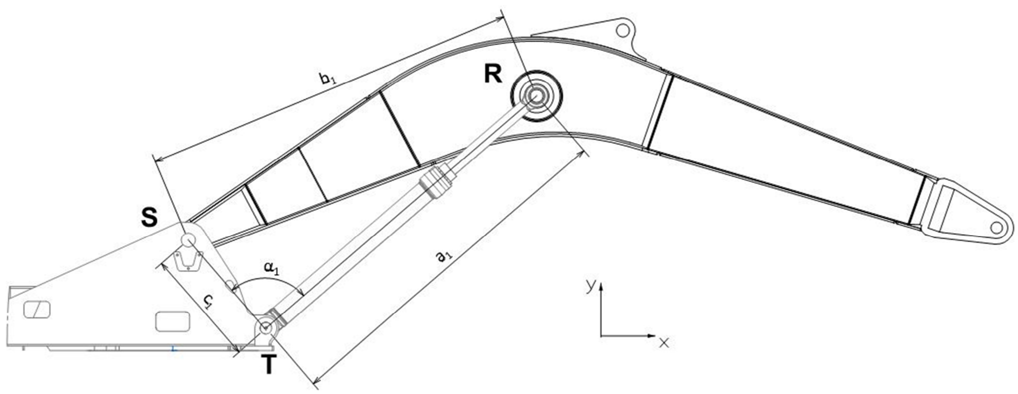


Figure 3.2 Mechanism parameters of the boom

Boom position depends on the cylinder position. Cylinder position can be defined by the angle of α_1 (Figure 3.2). It is calculated as;

$$b_1{}^2 = c_1{}^2 + a_1{}^2 - 2 \times c_1 \times a_1 \times cos\,\alpha_1$$

$$\alpha_1 = cos^{-1}\left(\frac{c_1{}^2 + a_1{}^2 - b_1{}^2}{2 \times c_1 \times a_1}\right) \qquad (3.1)$$

This position can also be found with respect to the coordinate axis.

When the angle of α_1 has certain value (as the input), a_1 can be calculated. So, position of boom cylinder can be determined (as the output).

3.2.2 Arm position

Arm cylinder is important to obtain maximum digging force. The point of M and O are pivot points of the arm. Piston side of the arm cylinder is on the boom and rod side of the arm is on the arm. a2 has a length between M and N points and it changes according to the hydraulic piston cylinder positions (Figure 3.3).

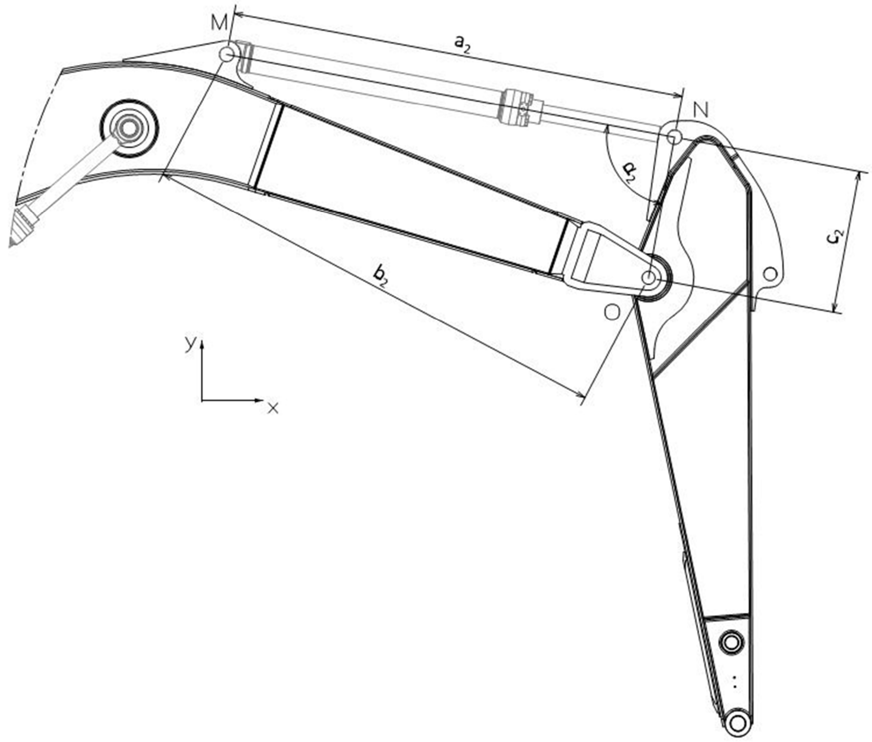


Figure 3.3 Mechanism parameters of the arm

Arm position depends on cylinder position. Cylinder position can be defined by the angle of α_2 (Figure 3.3). It is calculated as;

$$b_2{}^2 = c_2{}^2 + a_2{}^2 - 2 \times c_2 \times a_2 \times \cos \alpha_2$$

$$\alpha_2 = \cos^{-1}\left(\frac{c_2{}^2 + a_2{}^2 - b_2{}^2}{2 \times c_2 \times a_2}\right) \tag{3.2}$$

This position can also be found with respect to the coordinate axis.

When the angle of α_2 has certain value, a_2 can be calculated. So, position of arm cylinder will be determined.

3.2.3 Bucket position

Bucket cylinder affects digging force. The point of G, J and L are pivot points of the bucket and links. Piston side of bucket cylinder is on the arm and its rod side is on the link group. The bucket piston cylinder mechanism actuates four bar mechanism, then the bucket is activated. a3 has a length between G and H points and it changes according to the hydraulic piston cylinder positions (Figure 3.4).

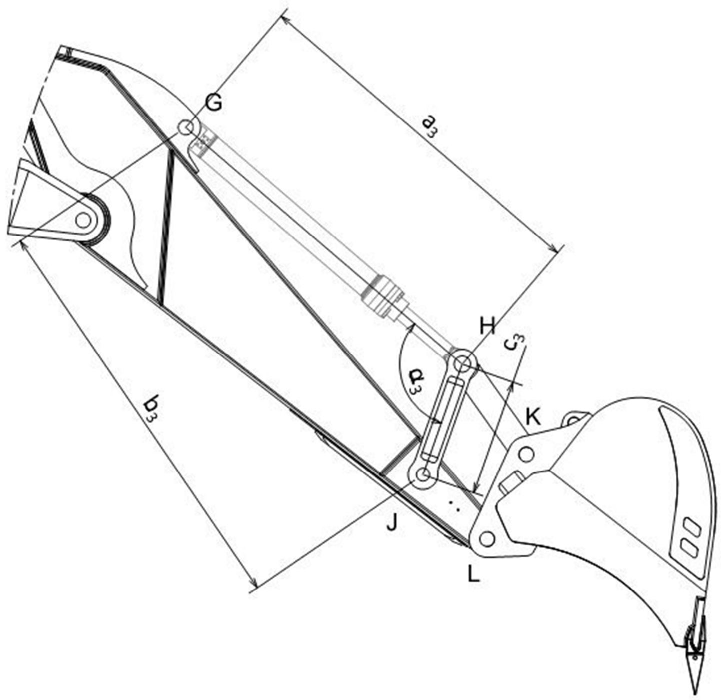


Figure 3.4 Mechanism parameters of the bucket

Bucket position with respect to cylinder position can be defined by the angle of α_3 (Figure 3.4). It is calculated as;

$$b_3^{\,2} = c_3^{\,2} + a_3^{\,2} - 2 \times c_3 \times a_3 \times \cos \alpha_3$$

$$\alpha_3 = \cos^{-1}\left(\frac{c_3^{\,2} + a_3^{\,2} - b_3^{\,2}}{2 \times c_3 \times a_3}\right) \tag{3.3}$$

This position can also be found with respect to the coordinate axis.

When the angle of α_3 has certain value, a_3 can be calculated. So, position of bucket cylinder can be determined.

The position of front attachment can be determined from equations 3.1, 3.2 and 3.3 simultaneously.

3.3 Breakout Force Calculation

Excavator machine can be used for different purposes. Excavation is one of the most important applications for the machine. Breakout force is the most important instrument for excavation.

Hydraulic force is obtained from oil that circulates in a close system after starting the engine. When the operator moves the joystick, oil is directed to the cylinders according to the desired position of the joystick by means of main valve. The cylinders are made of linear motion. Boom is moved through boom cylinders. Arm is moved through the arm cylinders. The position of arm and boom depend on these cylinders. The applicable force depends on mechanism position, diameter of cylinders and the working oil pressure.

Boom cylinder is generally used for bucket unloading position. It means that is used to transport load from one place to another place. Arm and bucket cylinders are used for digging. So, arm and bucket cylinders effect are calculated separately and the results are summed for break out force.

Working pressure and area of cylinder are necessary to calculate the force created by the cylinder. Large diameter means larger area (Figure 3.5). Thus, inner diameter is used to calculate area.

Figure 3.5 Sectional view of cylinder

$$F = P \times \pi \times \frac{{D_i}^2}{4} \qquad (3.4)$$

D_i is the inside diameter and P is the working pressure.

3.3.1 Force Calculation According to Arm Cylinder

When analyzing the front attachment, the forces in Figure 3.6 which are applied to points of A, D, E, C and B should be known. Degrees of freedom of these points should also be known.

$$F_{arm} = P \times \pi \times \frac{D_{arm}^2}{4} \tag{3.5}$$

$$F_{boom} = P \times \pi \times \frac{D_{boom}^2}{4} \tag{3.6}$$

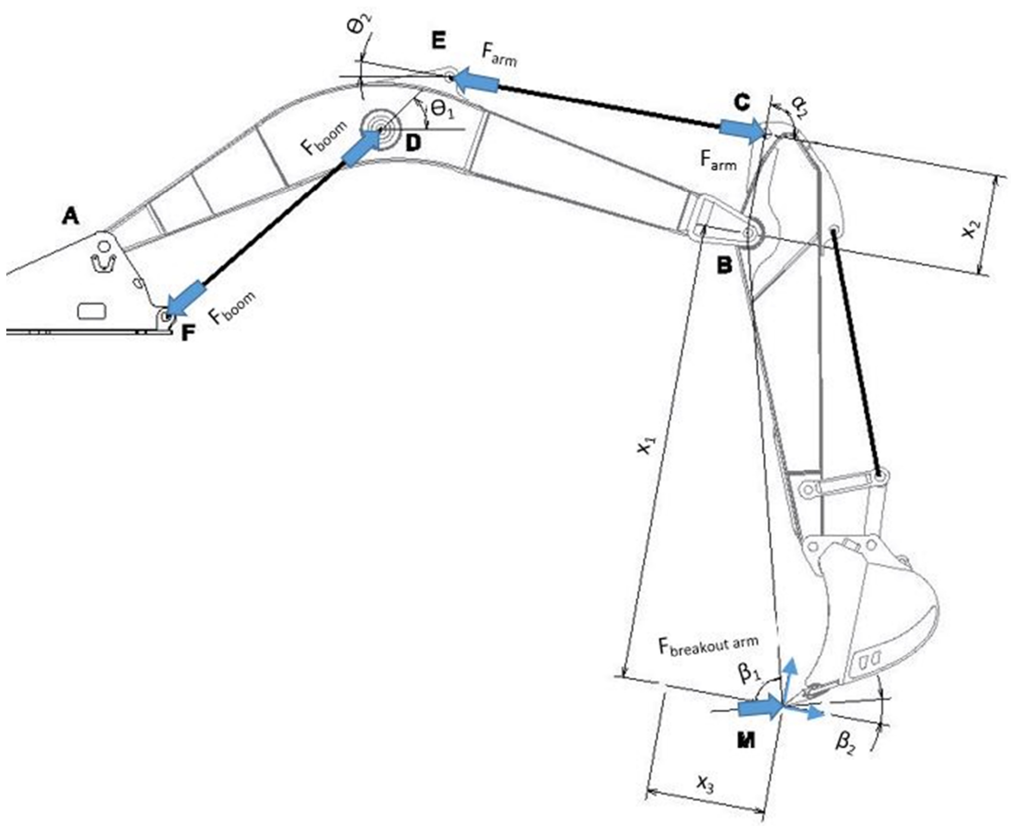


Figure 3.6 Free body diagram while arm cylinder is active

When the boom is analyzed statically, the values of F_{boom}, F_{arm}, Θ_1, Θ_2 should be known. In addition, when the breakout force from arm cylinder is analyzed, the values of α_2, β_1, x_1, x_2 and x_3 should be known.

To calculate $F_{breakout\ arm}$;

The moment point becomes location of B. So, breakout force of arm is calculated with the aim of below formula (3.6).

$$F_{arm} \times x_2 - F_{breakout\ arm} \times \cos\beta_2 \times x_1 - F_{braekout\ arm} \times \sin\beta_2 \times x_3 = 0$$

$$F_{breakout\ arm} = \frac{F_{arm} \times x_2}{(\cos\beta_2 \times x_1 + \sin\beta_2 \times x_3)} \tag{3.7}$$

3.3.2 Force Calculation According to Bucket Cylinder

After the breakout force of arm is calculated, bucket force is another important parameter that should be calculated. The forces affected to points of G, H, J, L, and K should be known in Figure 3.7. Degrees of freedom of these points should be also known.

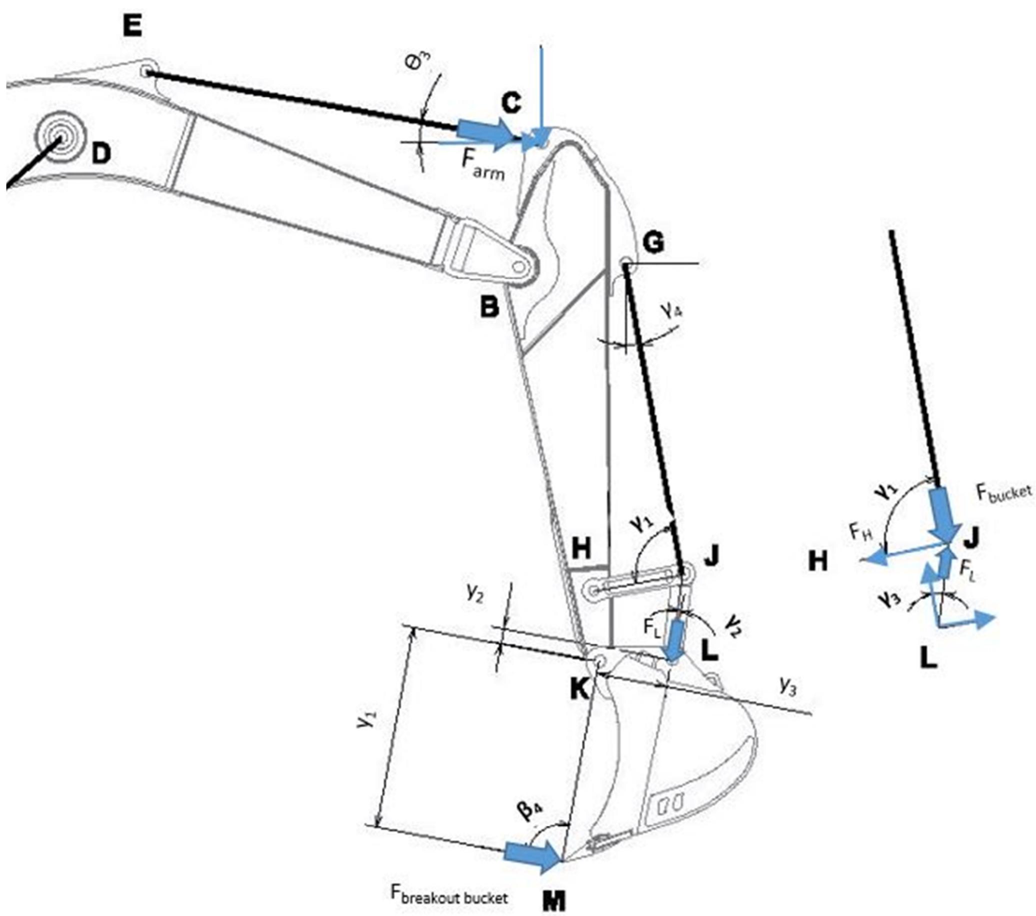


Figure 3.7 Free body diagram while bucket cylinder is active

$$F_{bucket} = P \times \pi \times \frac{D_{bucket}^2}{4} \tag{3.8}$$

When the arm is analyzed statically, the values of F_{bucket}, F_{arm}, γ_4, Θ_3, F_H should be known. In addition, when the breakout force from bucket cylinder is analyzed, the values of $\gamma_{1,}$ $\gamma_{2,}$ $\gamma_{3,}$ y_1, y_2, y_3, β_4, F_L should be known.

To calculate $F_{breakout\ bucket}$;

The moment point becomes location of K. So, breakout force of bucket is calculated with the aim of below formula (3.9).

From static equilibrium of point J;

$$F_{bucket} = F_L \times \cos\gamma_3$$

$$F_L = \frac{F_{bucket}}{\cos\gamma_3} \tag{3.9}$$

$$F_L \times \cos\gamma_2 + F_L \times \sin\gamma_2 - F_{breakout\ bucket} \times y_1 = 0$$

$$F_{breakout\ bucket} = \frac{F_L(\cos\gamma_2 + \sin\gamma_2)}{y_1} \tag{3.10}$$

Where

F_{arm}: force of arm cylinder

F_{boom}: force of boom cylinder

Θ_1: horizontal angle of the boom cylinder force

Θ_2: horizontal angle of the arm cylinder on point E

α_2: angle between EC and CB

x_2: dimension between point C and point B

x_1: dimension between point M and point M

x_3: dimension between point M and point B

β_2: angle of $F_{breakout\ arm}$ according to the coordinate axis

Θ_3: horizontal angle of the arm cylinder force on point C

γ_1: angle between GJ and HJ

γ_2: angle of F_L according to the coordinate axis

γ_3: angle of F_L according to the coordinate axis

γ_4: vertical angle of the bucket cylinder

y_1: dimension between point K and point M

y_2: vertical dimension between point L and point K

y_3: horizontal dimension between point L and point K

β_1, β_4: breakout force according to SAE

Breakout force changes continuously according to the cylinder position while excavating operations are made. This force calculation is defined through SAE J1179 handbook. This standard defines maximum breakout force as "*The maximum digging forces are the digging forces that can be exerted at the outermost cutting point. These forces are calculated by applying working circuit pressure to the cylinder(s) providing the digging force without exceeding holding circuit pressure in any other circuit. Weight of the components and friction are to be excluded from these force calculations*". SAE standard describes how to calculate the breakout force for the bucket and arm cylinders. Standard implies: '*The maximum bucket tangential force, 'V', (Figure 3.8) is the digging force generated by the bucket cylinder(s) and tangent to the arc of radius 'C'. The bucket shall be positioned to obtain the maximum output moment from the bucket cylinder(s) and connecting linkage.'* This calculation is for bucket cylinder. For the arm cylinder, standard implies '*The maximum arm/dipperstickforce, 'W' (Figure 3.9) is the digging force generated by the arm/dipperstick cylinder(s) and tangent to the arc of radius 'B'. The arm/dipperstick shall be positioned to obtain the maximum output moment from the arm/dipperstick cylinder(s) and the bucket positioned as in the case of maximum bucket tangential force.'* [18]

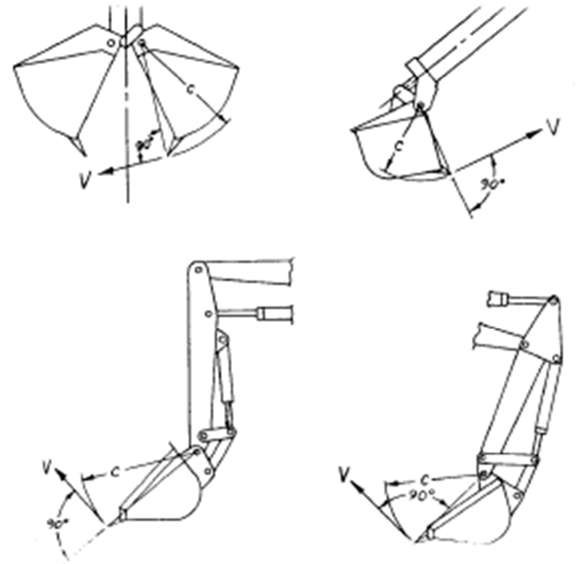


Figure 3.8 Maximum bucket breakout force

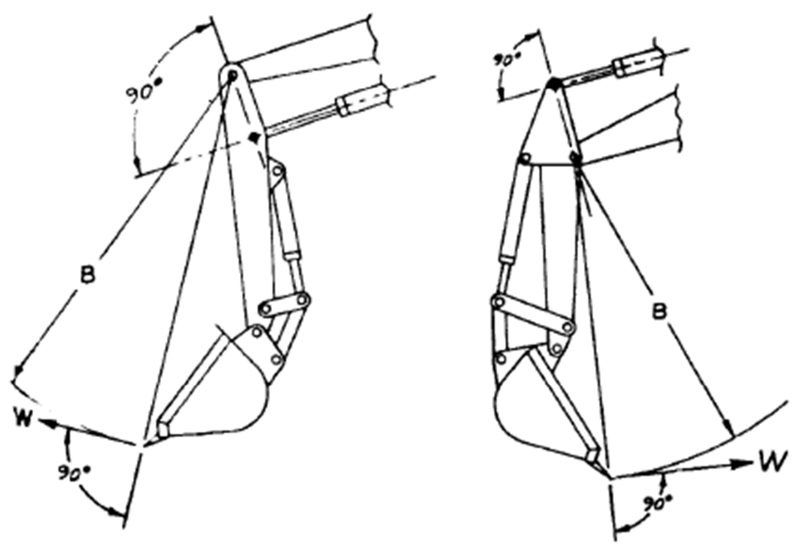


Figure 3.9 Maximum arm breakout force

In this chapter, position analysis and kinematic analysis has been focused. The results that are obtained in this section are very important initial steps for the structural analysis. Direction, magnitude and location of forces to be applied to solid models during analyzes will be based to results given in this chapter.

Cylinder forces and breakout forces that are calculated according to SAE J1179 are shown;

Table 3.1 Cylinder forces

Cylinder Extension Forces (kN)	
Boom	539
Arm	704
Bucket	539

Table 3.2 Breakout forces

Breakout forces (kN)	
Arm active	150
Bucket active	200

CHAPTER 4

FINITE ELEMENT METHOD AND STATIC ANALYSIS OF BOOM

4.1 Introduction

In this chapter, a general knowledge about FEM and Simxpert software will be given. In addition, boundary conditions (BC), information about Nastran and Adams programs that are used by Simxpert interface are explained. Static analyze of boom will be carried out mainly in this chapter.

4.2 Finite Element Method

The finite element method offers virtually unlimited problem generality by permitting the use of elements of various regular shapes. These elements can be combined to approximate any irregular boundary. In similar fashion, loads and constraints of any type can be applied. Problem generality comes at the expense of insight-a finite element solution is essentially a stack of numbers that applies only to the particular problem posed by the finite element model. Changing any significant aspect of the model generally requires a complete reanalysis of the problem. Analysts consider this a small price to pay, however, since the finite element method is often the only possible method of analysis. The finite element method is applicable to all classes of field problems, including structural analysis, heat transfer, fluid flow, and electromagnetics [19].

Finite element method is a numerical solution method to find solutions to various engineering problems. With this method, complex problems and geometries are sub-divided into small details (elements). These small elements facilitate the analysis.

It is very important to be competitive in the free market. FEM is widely used to speed up the solution rather. Three factors are important to be competitive;

- ✓ Fast solutions
- ✓ High quality
- ✓ Cheaper product

These targets can be achieved with the help of computer aided engineering.

Prior to use of FEM certain steps should be applied for design stage. These steps to define can be listed as follows;

I. Coordinate systems
II. Model geometry
III. Finite elements
IV. Loads
V. Boundary conditions
VI. Material properties

First of all, the product should be created in two or three dimensions by means of computer aided design (CAD). Then, the product is imported by FEM software. Throughout present study the software that is called Simxpert is used. Components are combined by welding more than one part. If the part is defined as a solid model, surface process is not performed. Solid mesh is generated on solid model. This application is generally used for parts that have complex geometries. When the part has not complex shape, surface process is applied. With this process, mid surfaces are obtained as two dimensional and these surfaces are combined. It is important how the parts are combined. Combination is made according to the welding process; method of welding is butt welding. Then, thickness and material information are assigned to planer parts. After geometries are prepared, mesh model is generated. Generating the mesh model, checking quality of meshing should be done. The elements can be skew, tapered or free (does not connected other mesh elements). These kind of undesirable elements are determined and regulated. These elements should be minimized. Then materials and properties are assigned. The following information is needed at this step;

- Material properties: Isotropic, orthotropic etc.
- Young’s modulus
- Shear modulus
- Poisson’s ratio
- Density

After the material properties are assigned, loading and boundary conditions (BCs) are generated. The forces of cylinders are used for loading. The boundary condition is described according to machine working properties. After constraint types, loads are applied and the finite element model is hence ready to run.

4.3 Simxpert FEA Program

4.3.1 Introduction

Simxpert is a unified computer aided engineering environment for product simulation that enables manufacturers to accelerate the speed and accuracy of simulation, increase design productivity, and bring better products to market faster. It also provides assistance with pre-processing, modeling, solving, post-processing, results and reporting [20].

4.3.2 Program Capability

Simxpert software deals with the following topics [20];

- Perform comprehensive linear and nonlinear finite element analysis
- Layout advanced FEA testing for stress, thermal, vibration, and contact events
- Predict working loads arising from system interactions

- Use multibody dynamics to analyze the behavior of different components, connections, and forces in complete mechanical systems
- Study thermal performance by utilizing steady-state heat transfer, free/forced conduction, and thermo-mechanical analysis
- Perform large deformation, highly nonlinear, short duration transient dynamic analyses for both structural impact and coupled fluid-structure problems such as fan blade out scenarios and stent deployment in arteries
- Ensure safety and crashworthiness by using Simxpert's numerous tools
- Unique capabilities including dummy positioning, seat belt routing, airbag deployment, and explicit impact analysis
- Simulate control systems for: electrical, thermal, hydraulic, gas dynamics, and numerous other applications

While the model geometry are preparing, combining of welding parts are made. Fillet, groove and butt welds are widely used during combining of large parts, like chassis, boom or bucket. Fillet and groove welding are achieved by combining with each of the two parts. But butt welding is different. It is achieved by means of RBE2 rigid connectors in Nastran. So, model geometry is prepared without changing of part geometry.

Rigid Bar Element (RBE2): The RBE2 element defines a rigid body whose independent degrees of freedom are specified at a single point and whose dependent degrees of freedom are specified at an arbitrary number of points. The RBE2 element does not cause numerical difficulties because it does not add any stiffness to the model. The RBE2 element is actually a constraint element that prescribes the displacement relationship between two or more grid points [19]

The Newton-Raphson Method is widely used by ADAMS/Solver and is an important numerical algorithm to understand in order to produce models that lead to robust and fast simulations. To make initial condition analysis (ICA); position, velocity, force and acceleration initial condition analysis are made in Adams/Solver. To make kinematic analysis (KA); position, velocity and acceleration level kinematic analysis are made in Adams/Solver [19].

In MSC Nastran, constraints divided into two basic groups. They are single point constraints (SPCs) and multipoint constraints (MPCs).

The finite element model was created with shell elements. These elements are including Quad and Tria elements.

Some terms that are used in FEA;

Stiffness matrix: it describes the element's displacement response for a given load [19].

Rotation: Displacement about a coordinate axis. A grid point has three rotational degrees of freedom, one about each axis [19].

Translation: Direct, linear displacement along a coordinate axis. A grid point has three translation degrees of freedom, one along each axis [19]

Joint motion: Prescribe translational or rotational motion on a translational, revolute or cylindrical joint [19].

4.4 Static Analysis of Boom

In this section, boom is analyzed statically. If necessary, improvements and reinforcements are applied.

4.4.1 Assumptions

- It is assumed that material behavior is linear elastic.
- Displacements are small.
- Structure is in static equilibrium.
- Structure is unloaded prior to defined loads.

➤ Structure is free of initial and residual stress.

4.4.2 Static Loads

Two cylinders are effective in this analyze. They are boom and arm cylinders. Forces of F_{arm} and F_{boom} are calculated from formulation 3.4 and 3.5.

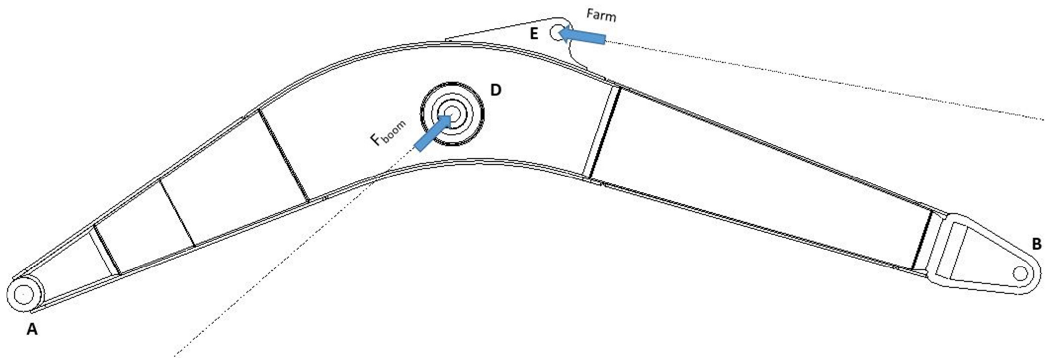


Figure 4.1 Joint forces on the boom

Loads are applied according to angle of the forces (Figure 4.2).

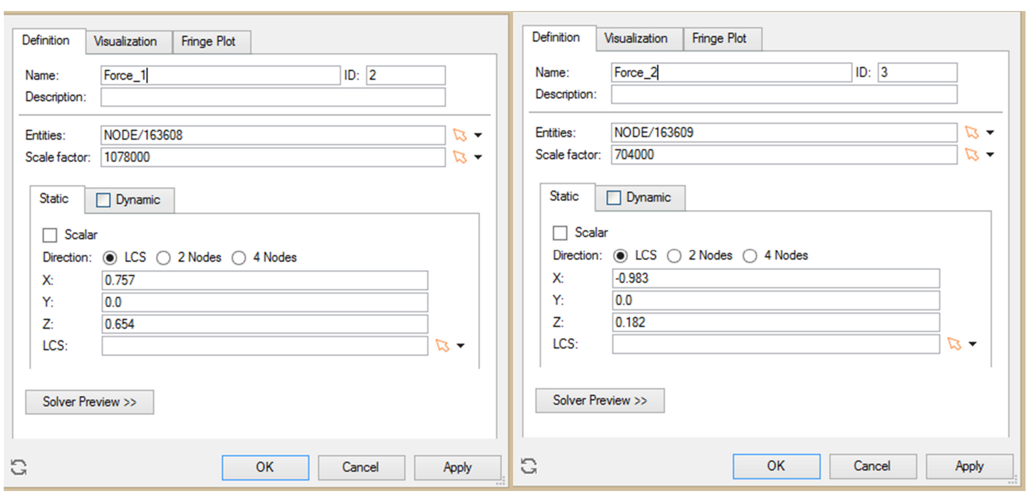


Figure 4.2 Application of loads

4.4.3 Mesh Properties

Surface mesh is used via Nastran. Mesh is applied automatically (Figure 4.3a). The boom model has 164539 shell elements (%98.6 quad, %1.3 tria).

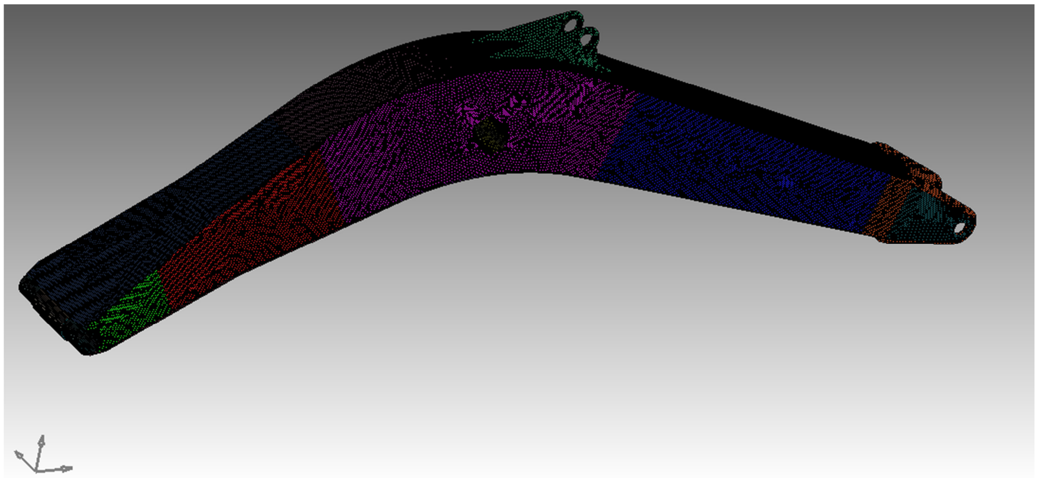


Figure 4.3.a Mesh of boom

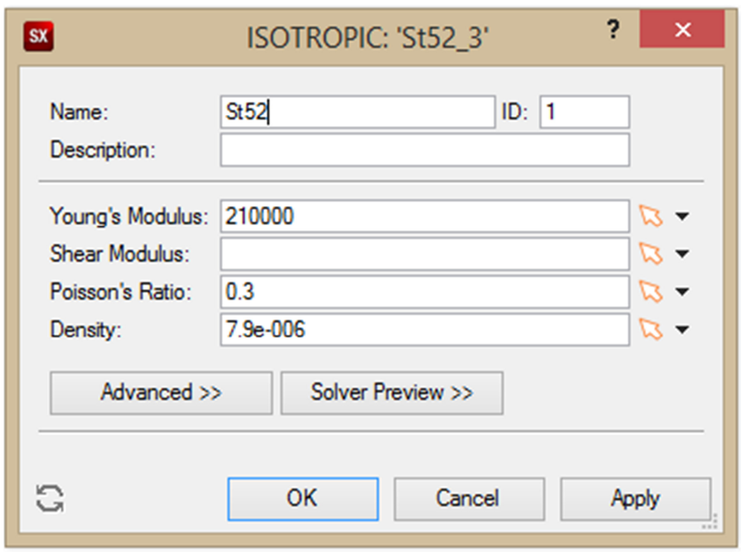


Figure 4.3.b Material properties of boom

4.4.4 Boundary Conditions

Points A and B are supports. They are general constraints. There is no freedom of x,y,z translations, they are fixed. But there is one freedom of y rotation (pin connection), the other x,z rotations are fixed.

Points E, D are load points shown in figure 4.1. Forces are applied as calculated in Chapter 3.

4.4.5 Material Properties

The material of front attachment is steel (St52-3) and material of pins are SAE 1040 steel. Each property is assigned as shown in figure 4.3b. The properties of these materials are shown in the table 4.1 and 4.2.

Table 4.1 Properties of St52-3

Material	Lineer Isotropic
Young's Modulus	210 GPa
Poisson's Ratio	0.3
Density	7.9e-006 kg/mm³
Tensile Yield Strength	355 MPa
Compressive Yield Strength	355 MPa
Tensile Ultimate Strength	520 MPa

Table 4.2 Properties of SAE 1040 steel

Young's Modulus	200 GPa
Poisson's Ratio	0.29
Tensile Yield Strength	415 MPa
Tensile Ultimate Strength	590-620 MPa

4.5 Boom Analysis Results

The boom cylinder and arm cylinder diameters are known. α_1 and α_2 are 90 degrees. The boom is analyzed statically.

After creating of model geometry, mesh model, loads and BCs are defined and then finite element model is run. Results are investigated in terms of Von-Misses stress distribution.

The boom model (figure 4.4) has two general constraints. One of these constraints is for chassis connection. It has one degree of freedom (figure 4.5). The second constraint is for arm connection. It has one degree of freedom (figure 4.5). Both constraints have rotational freedom.

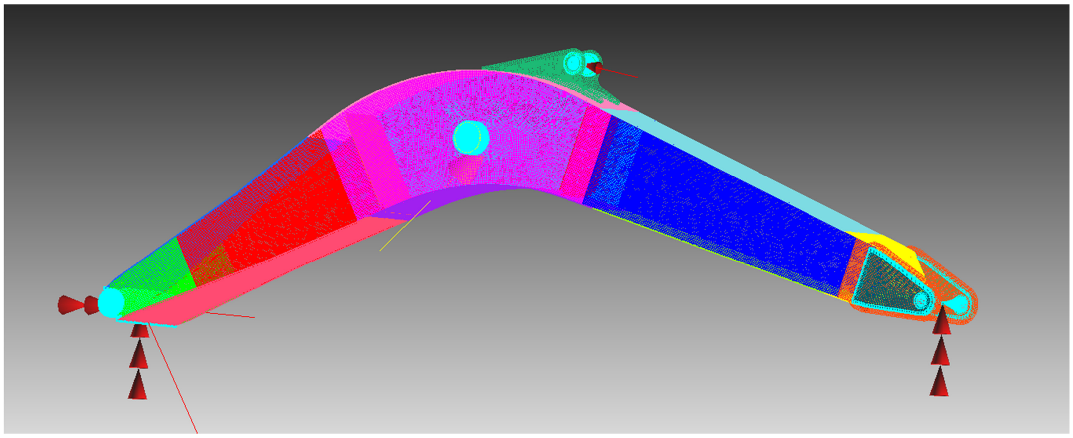


Figure 4.4 General view of boom model and material

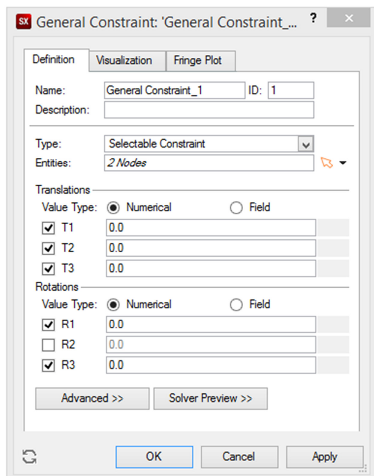
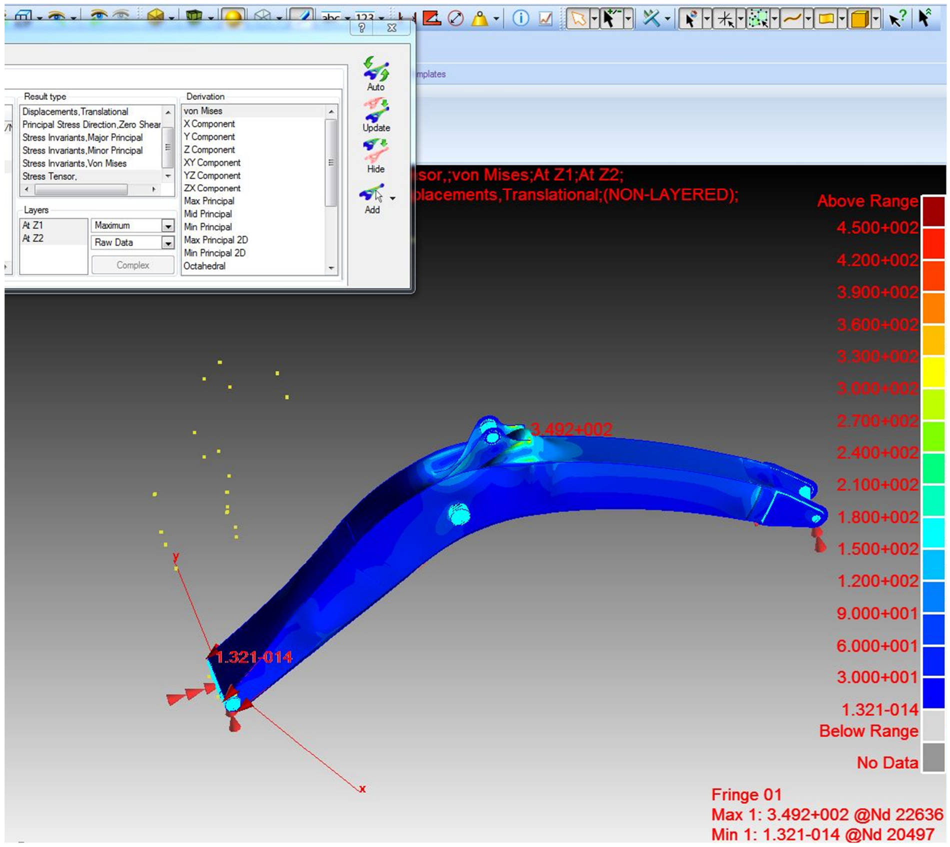
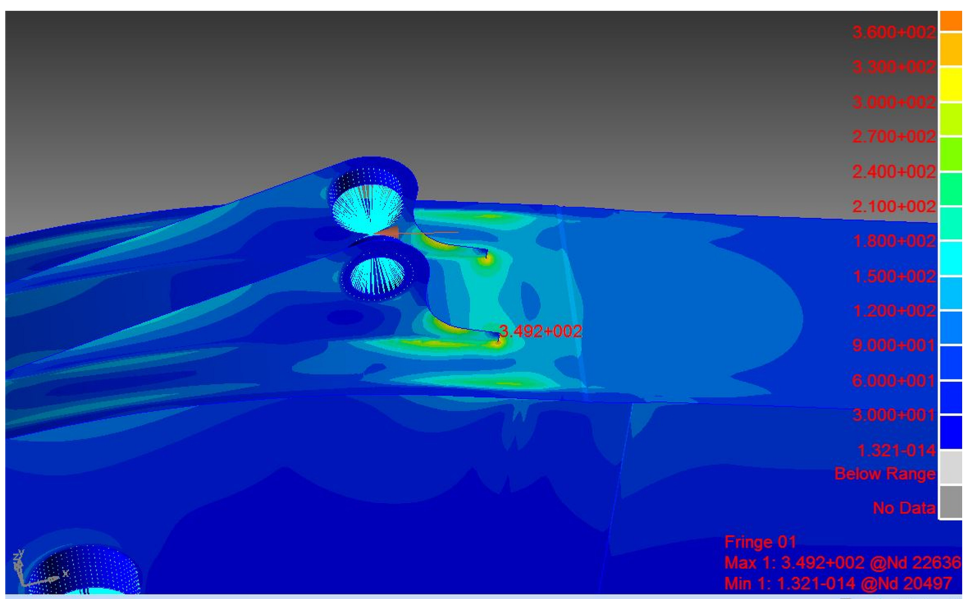


Figure 4.5 General constraints

Arm and boom cylinder forces (from equations 3.5 and 3.6) are applied on the boom. Investigating the analysis results, critical areas that have high stress values are observed (Figure 4.6). Some revisions are performed according to analysis results. After reinforcing, analysis is performed again. The reinforcing process is continued until the stress values become acceptable. The results are shown below;



(a)



(b)

Figure 4.6 First analysis result. (a) Stress distribution of boom (b) Stress zone of boom

First analysis results showed that high stress (349 MPa) occured between arm cylinder connection plate and boom upper plate.

Modifications made on the first revision are as follows;

The geometry of cylinder connection plate is changed. Edge radius in the marked area is increased (Figure 4.7). In this way, it is aimed to reduce stress concentration.

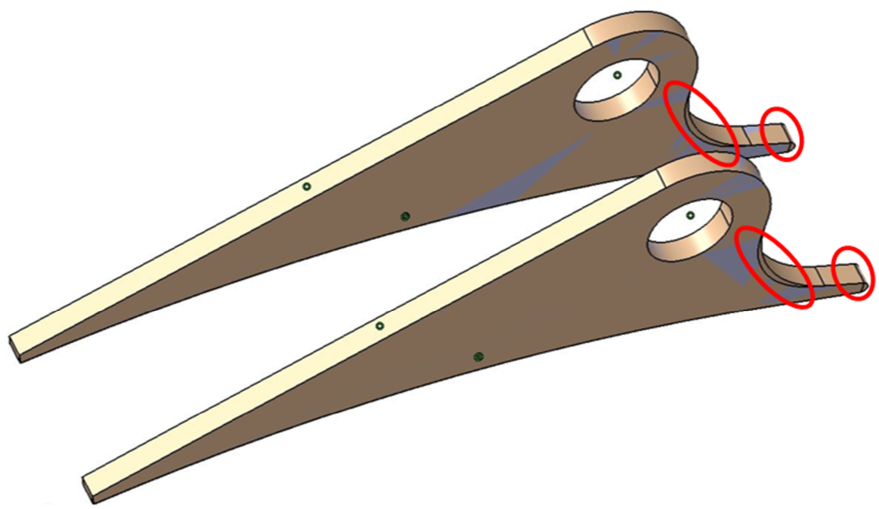
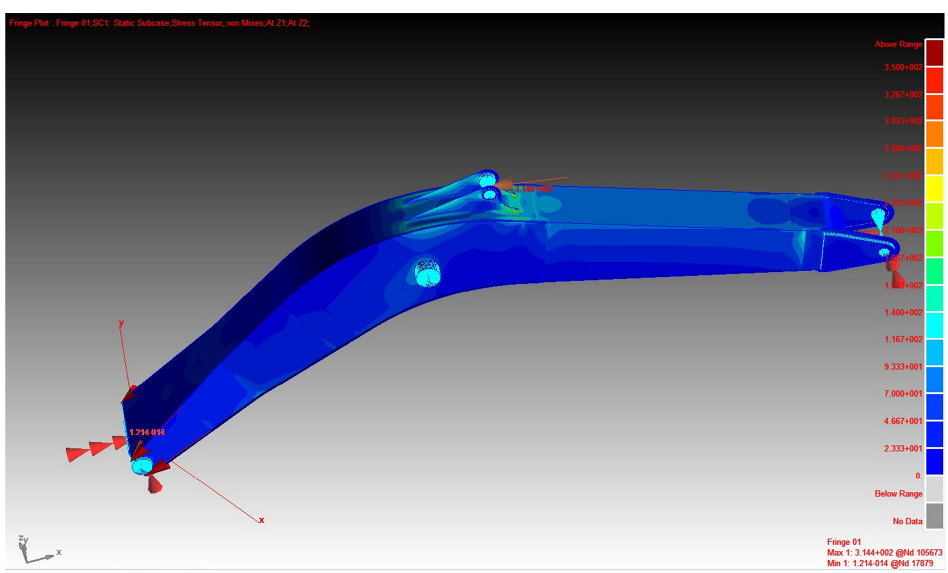
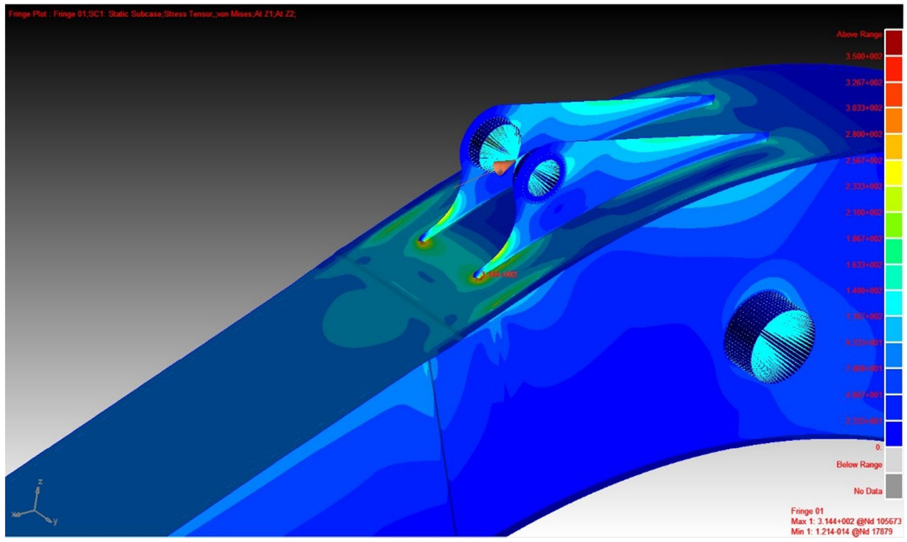


Figure 4.7 First revision of the boom



(a)



(b)

Figure 4.8 First revision analysis result. (a) Stress distribution of revision 1 boom (b) Stress zone of revision 1 boom

After application of first revision, the stress value is decreased (Figure 4.8) to 314 MPa. Hereby, revision 1 was useful but stress remains high.

Changes made on the second revision are as follows;

Thickness of cylinder connection plate is increased to 34 mm, about 20 percent. Changes in stress are observed depending on the thickness of plate.

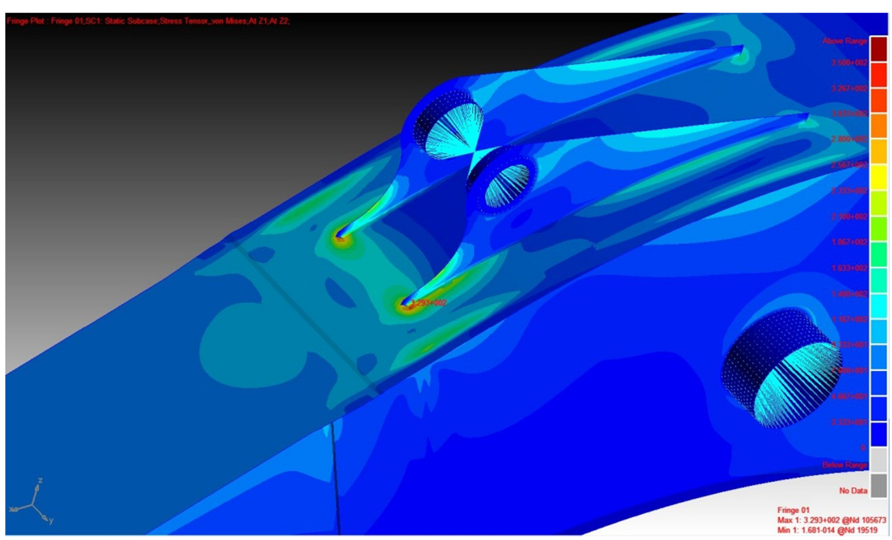


Figure 4.9 Stress zone of rev. 2 boom

The stress value is increased to 329 MPa compared to rev. 1 (Figure 4.9). So, the revision 2 is canceled.

Changes made on the third revision are as follows;

The geometry of cylinder connection plate is changed. The end of the part (1) is extended and the radius (2) is increased (Figure 4.10).

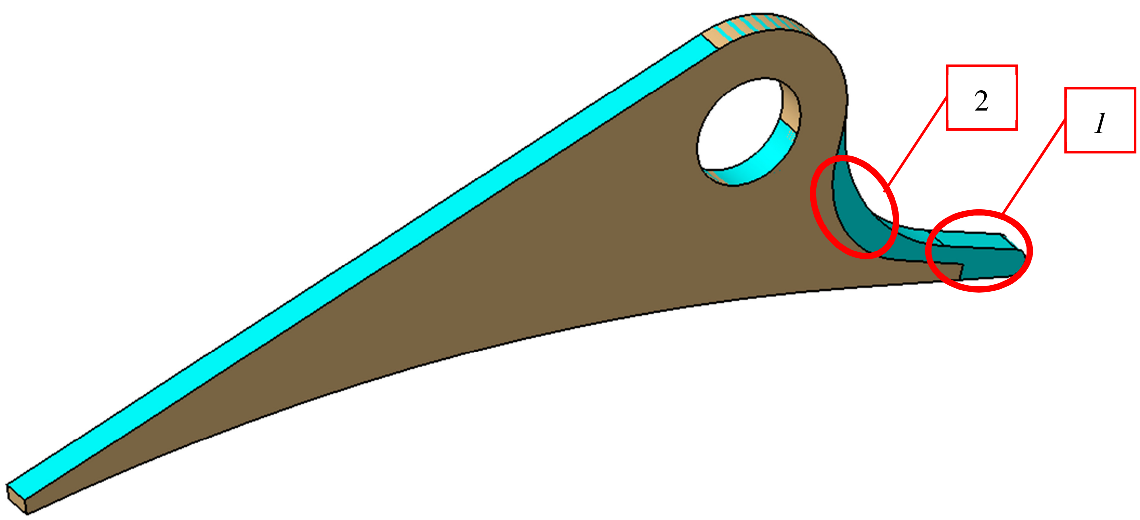
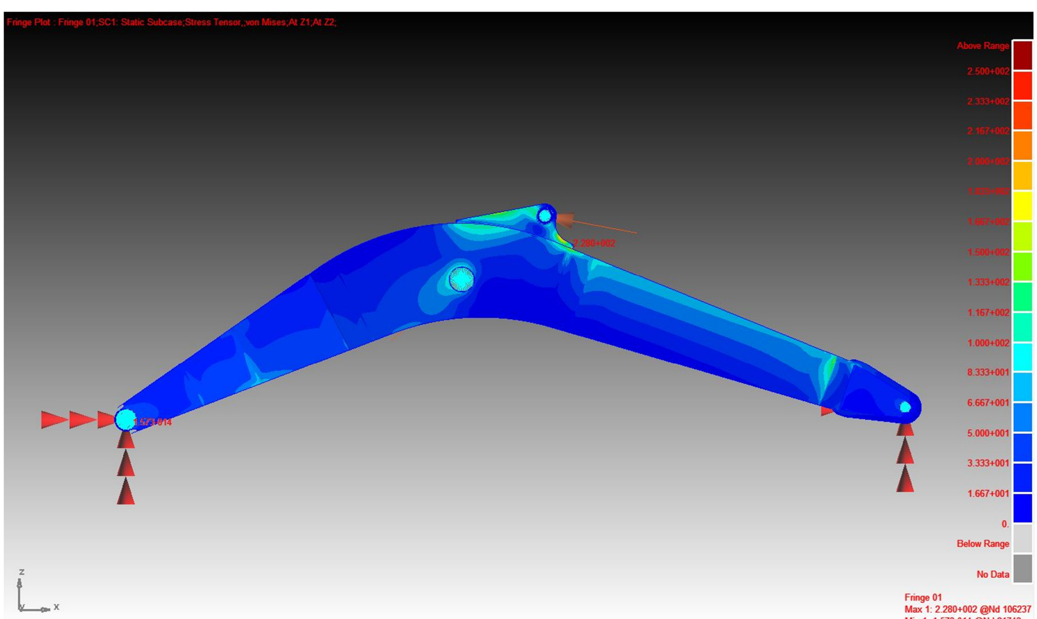
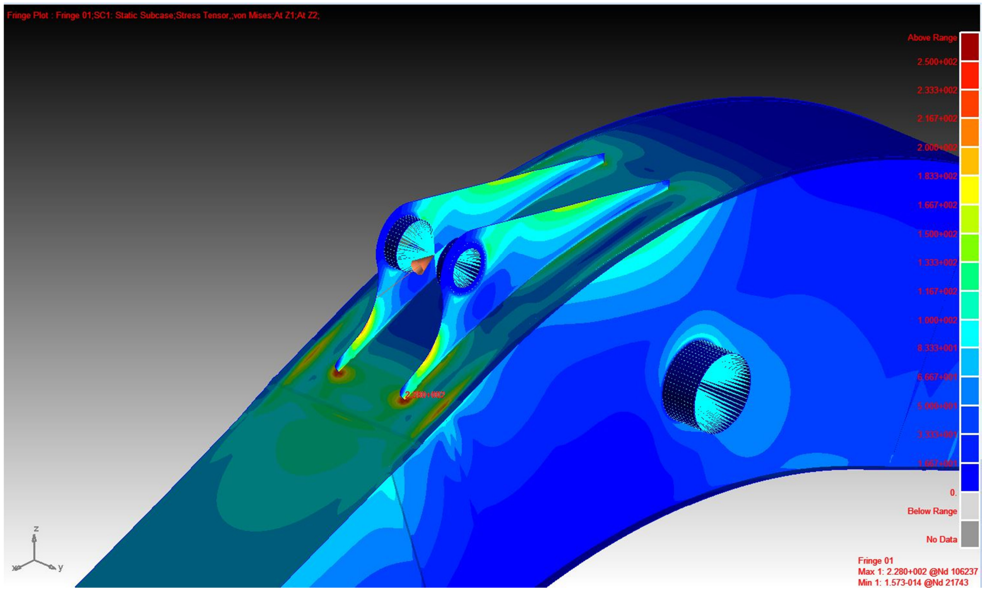


Figure 4.10 Third revision of the boom



(a)



(b)

Figure 4.11 Third revision analysis result. (a) Stress distribution of revision 3 boom (b) Stress zone of revision 3 boom

When the revision 1 compared to revision 3, the stress of the third revision is decreased (Figure 4.11) to 228 MPa. Thus first revision is canceled and third revision is accepted.

Changes made on the fourth revision are as follows;

In revision four, the radius (1) is increased (Figure 4.12).

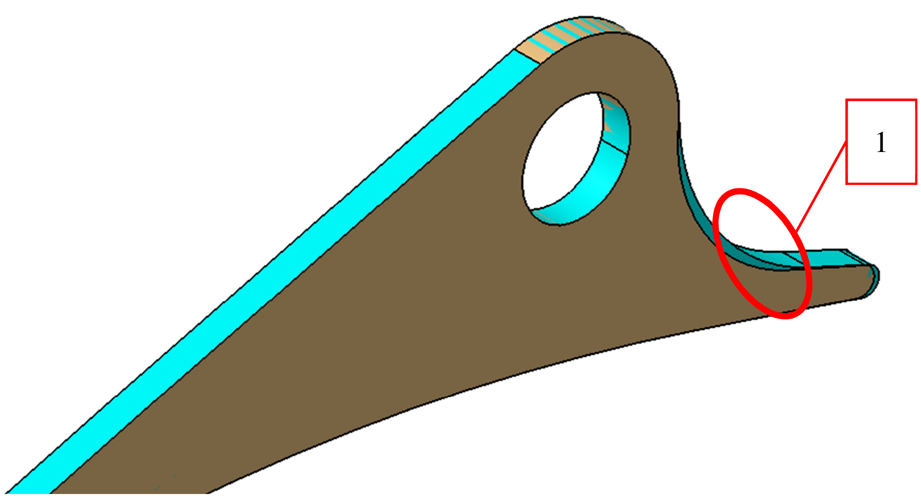
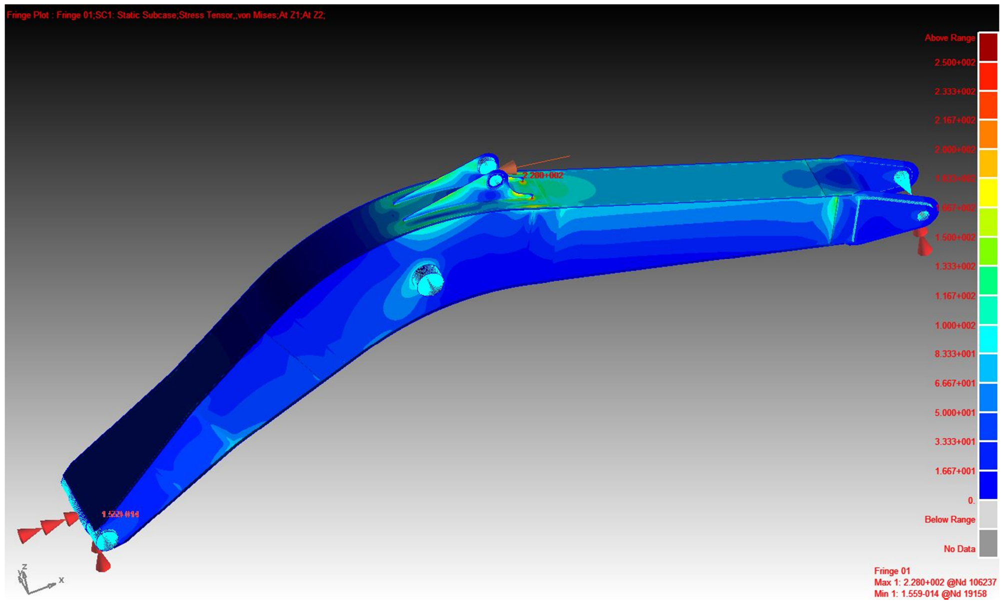
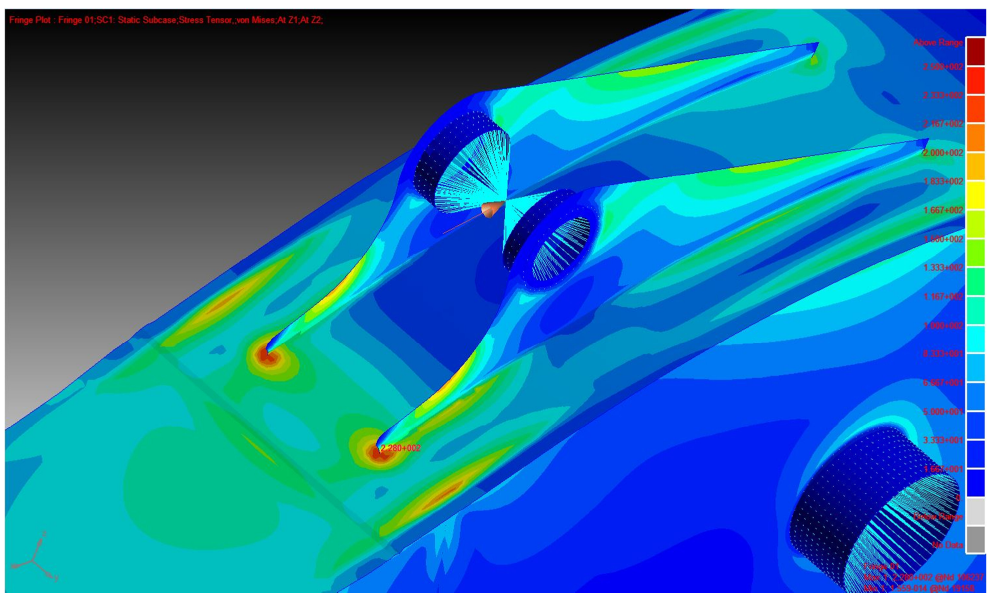


Figure 4.12 Fourth revision of the boom



(a)



(b)

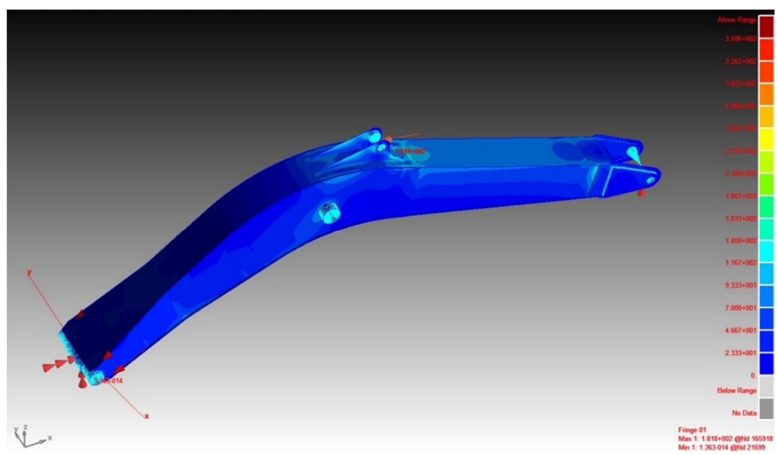
Figure 4.13 Fourth revision analysis result. (a) Stress distribution of revision 4 boom (b) Stress zone of revision 4 boom

When the revision 4 compared with revision 3, the stresses of the fourth revision are same (Figure 4.13). Thus fourth revision is canceled.

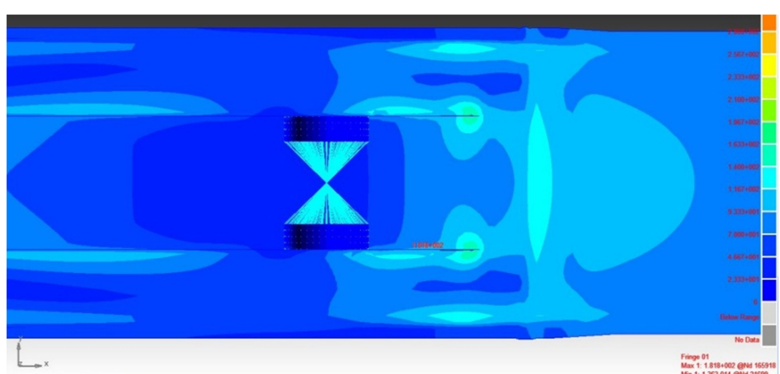
Changes made on the fifth revision are as follows;

Here, thickness of the marked part is increased to 20 mm, about 25 percent (Figure 4.14).

Figure 4.14 Fifth revision of the boom



(a)



(b)

Figure 4.15 Fifth revision analysis result. (a) Stress distribution of revision 5 boom (b) Stress zone of revision 5 boom

Stresses in the critical area are decreased (Figure 4.15) to 220 MPa with applied modification. According to the results, the third revision is canceled and the fifth revision is accepted.

Changes made on the sixth revision are as follows;

The geometry of cylinder connection plate is changed. The end of the part is removed (Figure 4.16).

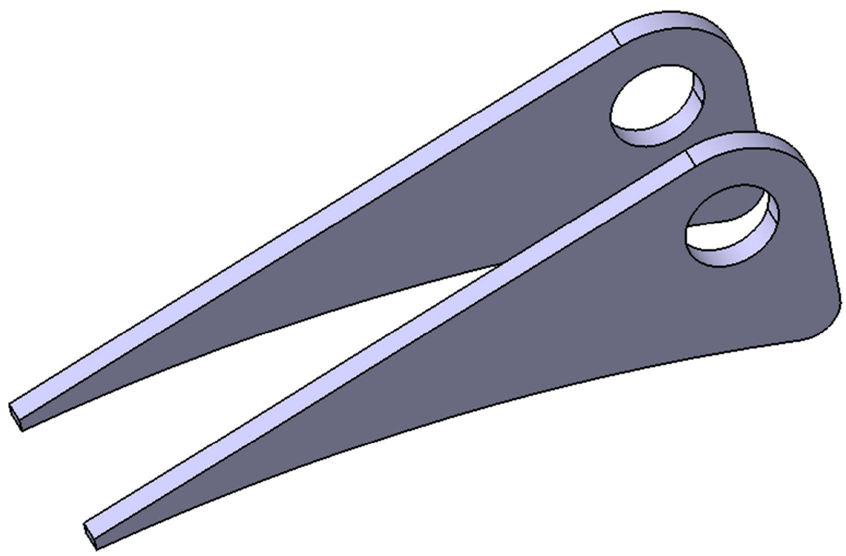
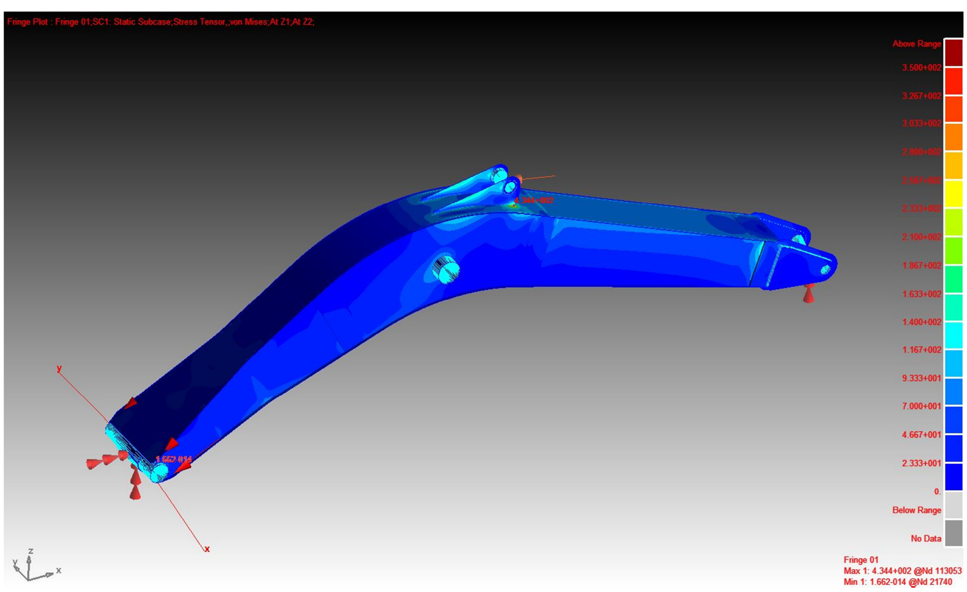
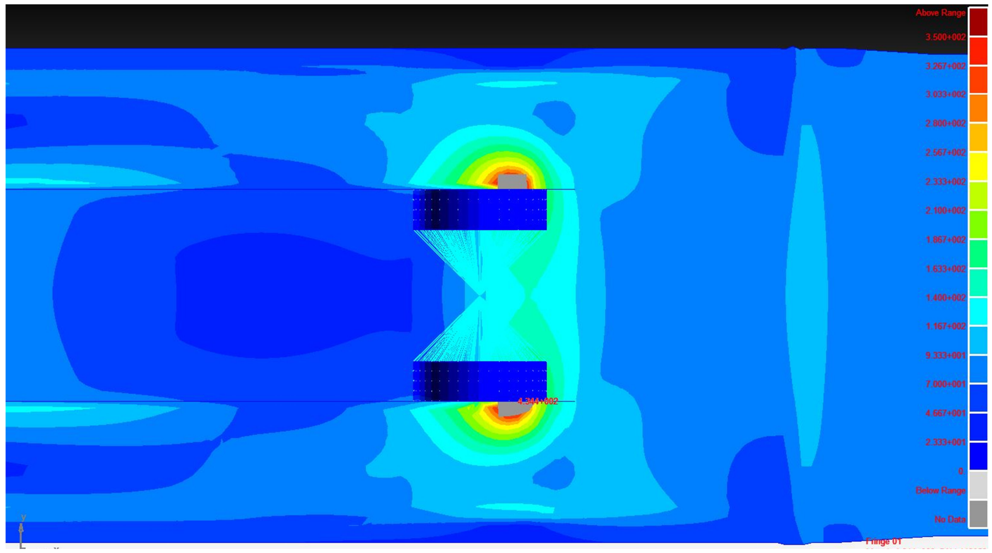


Figure 4.16 Sixth revision of the boom



(a)



(b)

Figure 4.17 Sixth revision analysis result. (a) Stress distribution of revision 6 boom (b) Stress zone of revision 6 boom

This revision affected the results. The stress considerably increased (Figure 4.17) to 434 MPa. For this reason, this revision is canceled.

Changes made on the seventh revision are as follows; Thickness of the plate is changed in revision 5 is increased 10 percent. But the results did not change considerably hence the rev. 7 is canceled.

Changes made on the eighth and ninth revisions are as follows. The geometry of radius is changed. Revision 8 is shown in Figure 4.18.a. Revision 9 is shown in Figure 4.18.b.

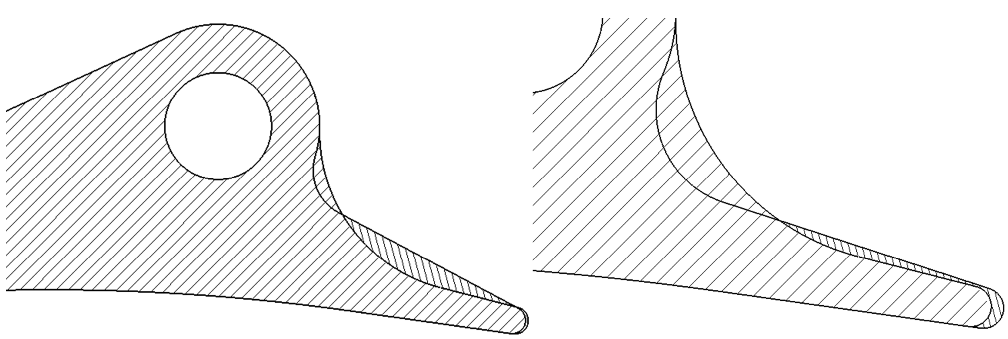
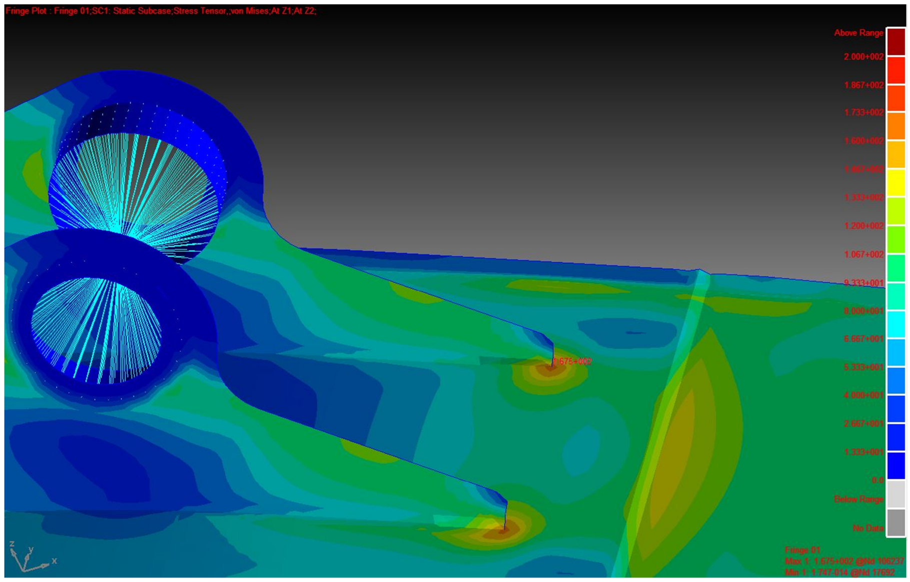


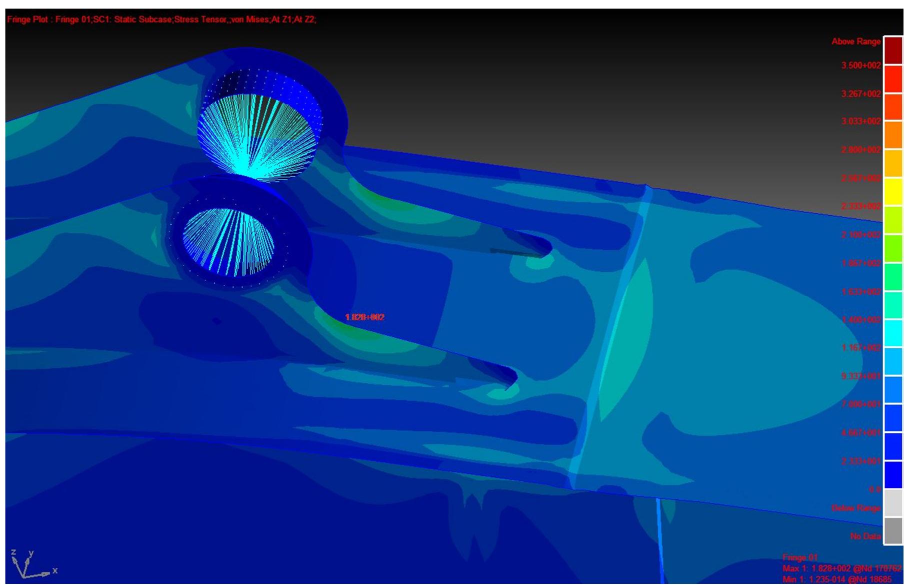
Figure 4.18.a Eighth revision

Figure 4.18.b Ninth revision

Figure 4.18 Eighth-Ninth revision of the boom



(a)



(b)

Figure 4.19 Eight and ninth revisions analysis results. (a) Stress zone of revision 8 boom (b) Stress zone of revision 9 boom

The result of rev. 9 did not change significantly (Figure 4.19). The result of rev. 8 affected positively. The revision 8 are accepted.

Changes made on the tenth and eleventh revisions are as follows;

In this revision, reinforcing part is added to the cylinder connection part as welded (Figure 4.20). Thickness of reinforcing part is decreased to 4 mm, about 50 percent in rev. 11.

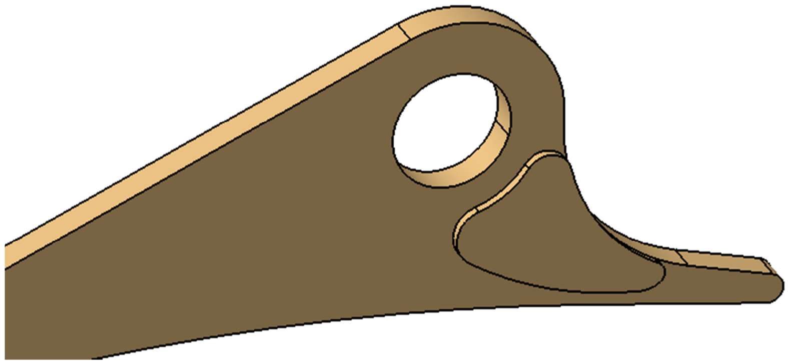
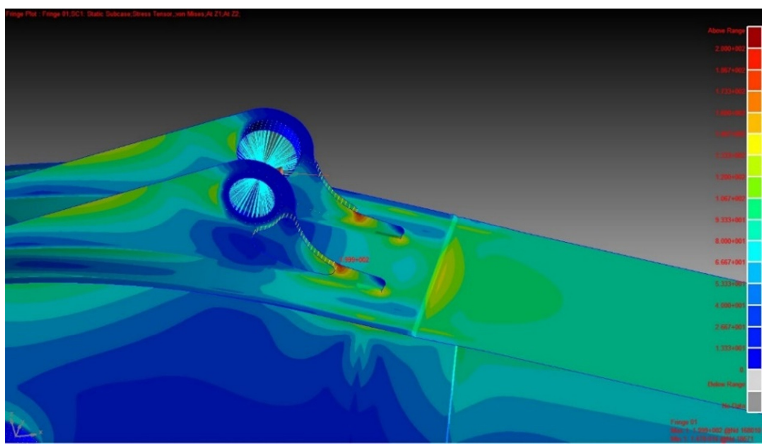
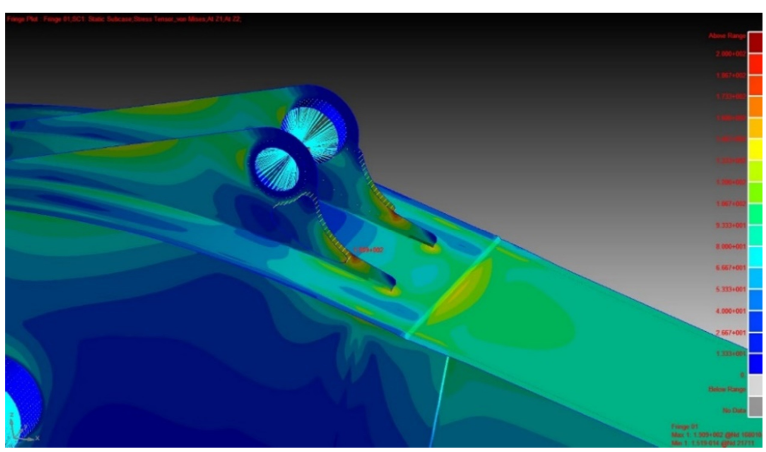


Figure 4.20 Tenth revision of the boom



(a)



(b)

Figure 4.21 Tenth and eleventh revision analysis result. (a) Stress zone of revision 10 boom (b) Stress zone of revision 11 boom

The stresses are increased to 200 MPa (Figure 4.21). So, the revisions are canceled.

Changes made on the twelfth and thirteenth revisions are as follows;

The geometry of reinforcing part is changed (Figure 4.22). The aim is to distribute the load. Thickness of new reinforcing part is doubled.

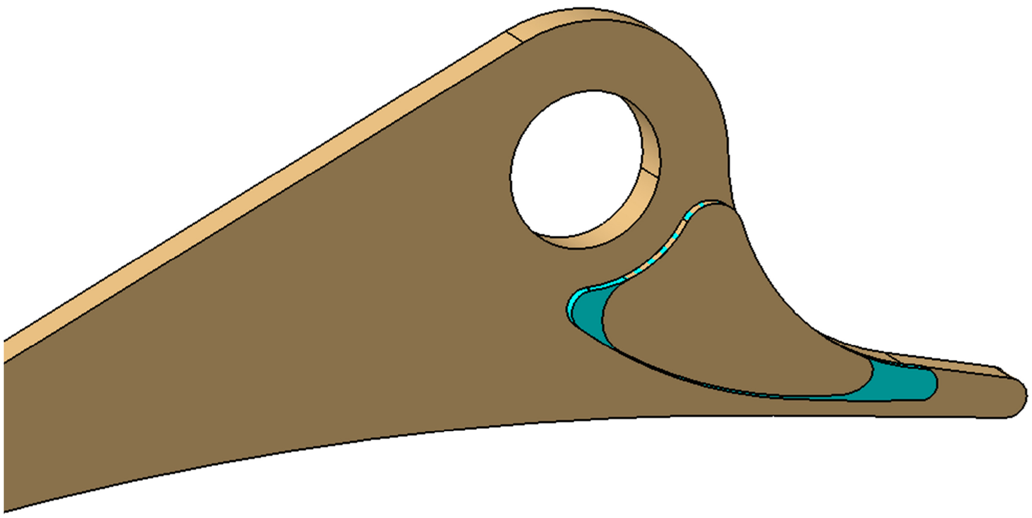


Figure 4.22 Twelfth and thirteenth revision of the boom

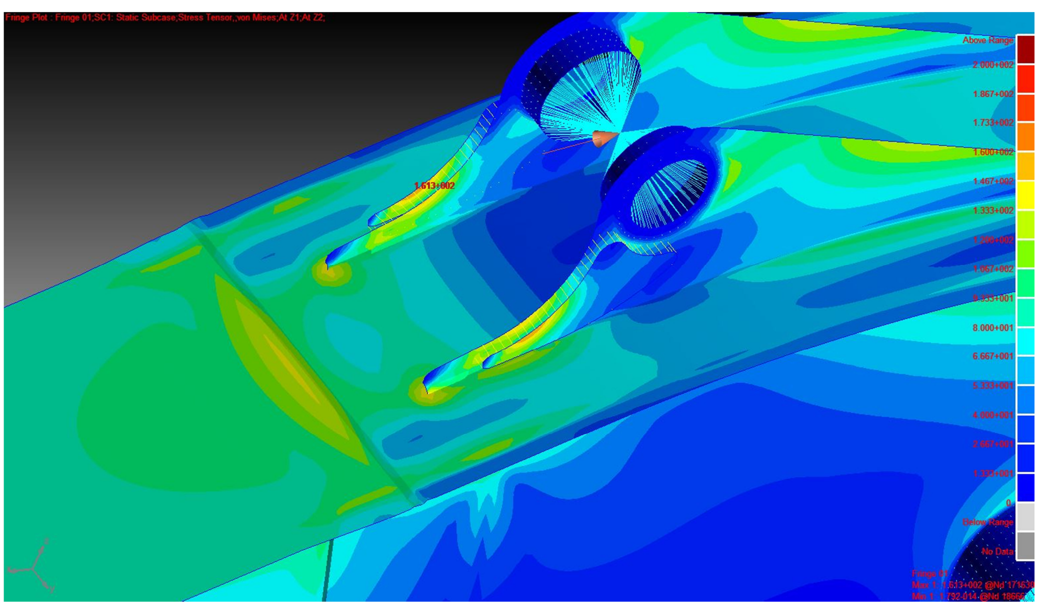


Figure 4.23.a Stress zone of rev. 12 boom

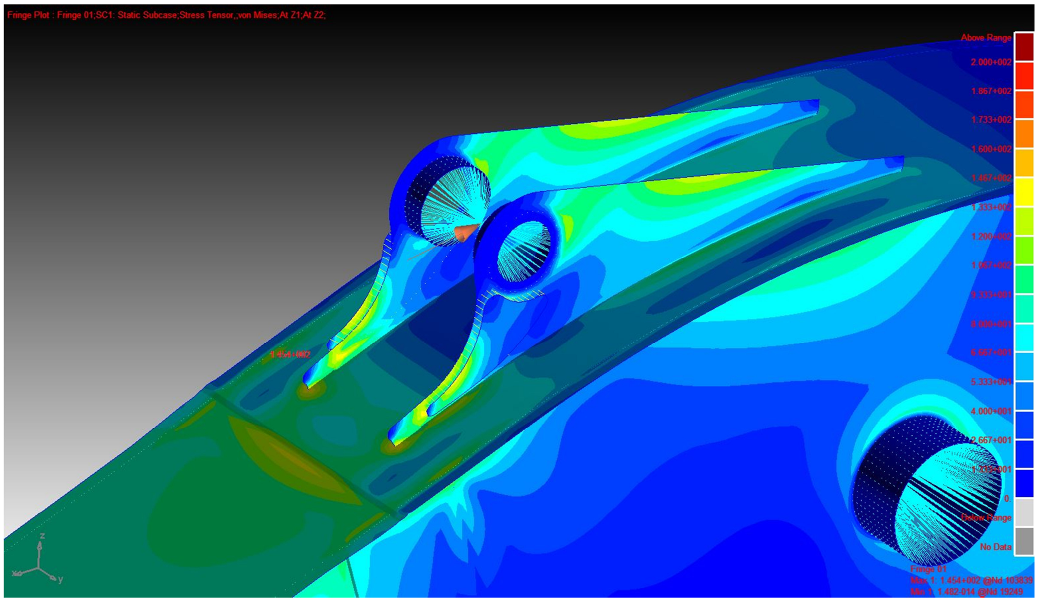


Figure 4.23.b Stress zone of rev. 13 boom

Revision 13 is accepted as final revision in static analysis (Figure 4.23). The stresses are decreased to acceptable values.

The difference between first and last case is as follows;

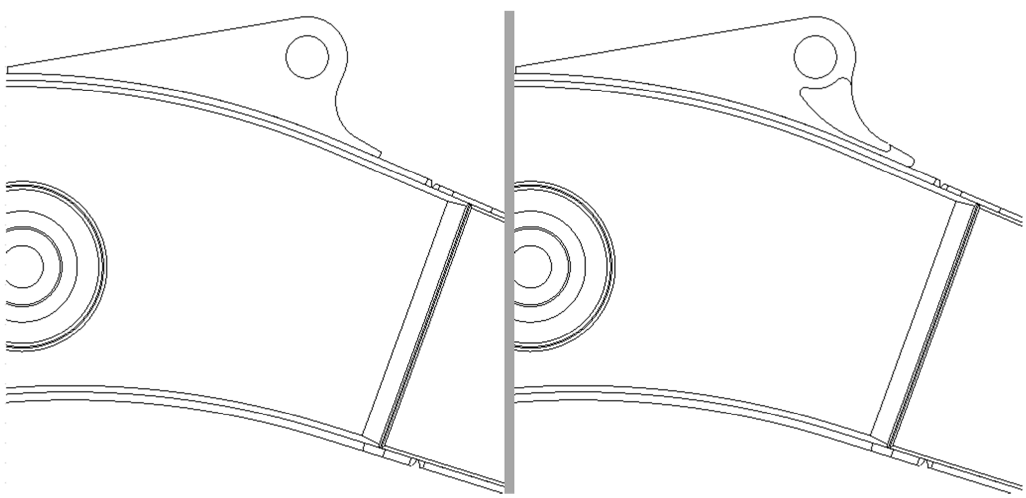


Figure 4.24 Comparing of cases

The results of the modifications are summarized at Table 4.3 below.

Table 4.3 Summary table of results

Reinforcement	Stress value (MPa)	Improvement (%)	Factor of Safety
First situation	349	-	1.02
Rev. 1	314	10.0	1.13
Rev. 2	329	5.7	1.08
Rev. 3	228	34.7	1.56
Rev. 4	228	34.7	1.56
Rev. 5	219.6	37.1	1.62
Rev. 6	434	-24.4	0.82
Rev. 7	181.8	47.9	1.95
Rev. 8	167.5	52.0	2.12
Rev. 9	182.8	47.6	1.94
Rev. 10	199.9	42.7	1.78
Rev. 11	190.9	45.3	1.86
Rev. 12	161	53.9	2.20
Rev. 13	145	58.5	2.45

This table helps comparing the results. It shows improvements gradually and change in stress values. Also, this table emphasizes effects of revisions in terms of factor of safety.

4.6 Conclusion

Importance of analysis is observed before the practice in engineering applications. The preliminary design that is ready to be given to mass production is detected high Von-Mises stress distributions. The analysis of boom is carried out according to the FEM steps. Highly stressed parts are determined between boom upper plate and arm cylinder connection plate. Some improvements are applied to minimize non-acceptable stress. Result of this study, the stress values are decreased gradually. Sheet thickness, material property, part geometry is shown to affect the design. At the end of this study, all stress levels are reduced below the allowed value and static analysis has been finalized.

CHAPTER 5

STATIC ANALYSIS OF ARM

5.1 Static Analysis of Arm

Front attachment of an excavator consists of many parts. In this chapter, static analysis of arm and its results are carried out with applied modifications.

5.1.1 Assumptions

- It is assumed that material behavior is linear elastic.
- Displacements are small.
- Structure is in static equilibrium.
- Structure is unloaded initially
- Structure is free of initial and residual stress.

5.1.2 Static Loads

Two cylinders are affective in this analyze. They are bucket and arm cylinders. Forces of F_{arm} and F_{bucket} are calculated from formulation 3.5 and 3.8. The forces that affect the joints are shown in figure 5.1. Applications of loads are shown in figure 5.2.

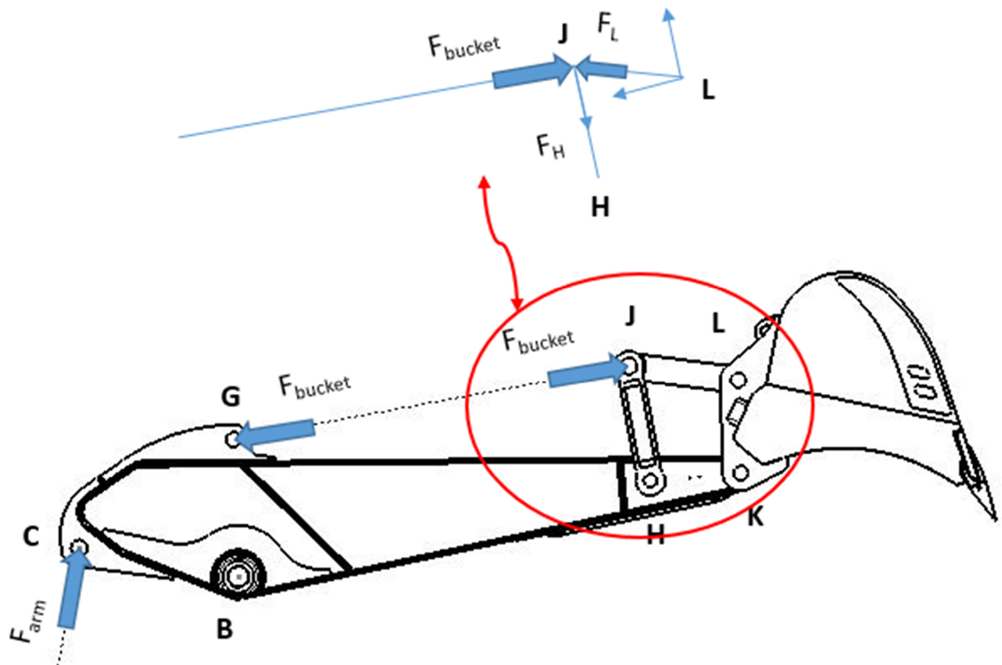


Figure 5.1 Joint forces on the arm

Loads are applied according to angle of the forces.

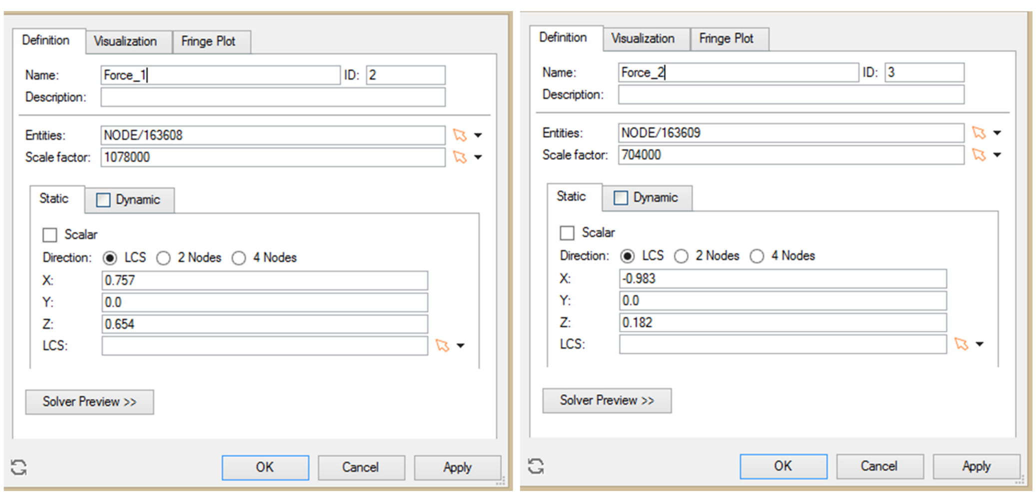


Figure 5.2 Application of loads

5.1.3 Mesh Properties

Surface mesh is used with the aid of Nastran. Mesh is applied automatically (Figure 5.3). The arm model has 98993 shell elements (% 98.1 quad, % 1.9 tria).

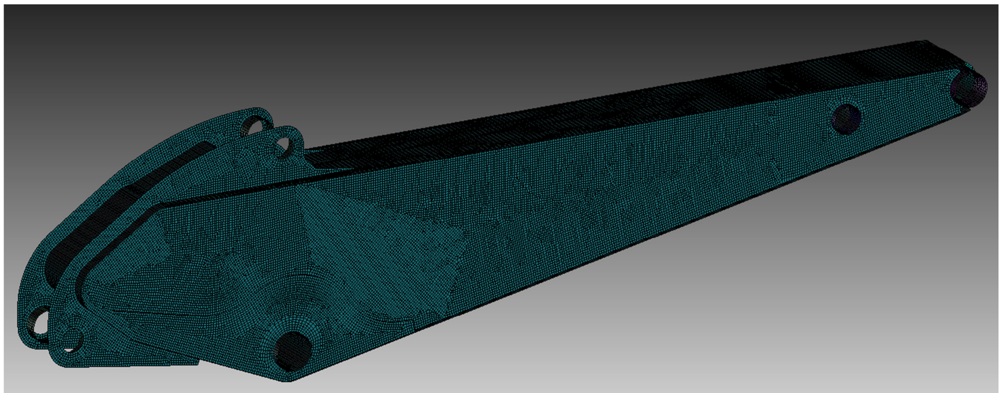


Figure 5.3 Mesh of arm

5.1.4 Boundary Conditions

Points B, K are supports. They are general constraints. There is no freedom of translations along x,y,z axes but there is one freedom of rotation (pin connection) about y axis, the other x,z rotations are fixed.

Points C, G, H are load points shown in figure 5.1. Forces are calculated in section 3.3.1 and 3.3.2.

5.1.5 Material Properties

The material of front attachment contains St52-3 and material of pins are SAE 1040 steel. Each properties are assigned as shown in figure 5.1. The properties of these materials were given in the table 4.1 and 4.2.

5.2 Arm Analysis Results

The arm cylinder and bucket cylinder diameters are known. α_3 and α_2 are 90 degrees. The arm can be analyzed statically. After creating of model geometry, mesh model, loads and BCs, finite element model is run. Results are investigated in terms of Von-Misses stress distribution. Von-Mises stress is widely used in industrial applications. It is used to control whether the material withstands a certain load. Control is made by comparing the resulting stress to yield stress.

The arm model (figure 5.4) has two general constraints. One of these constraints is for boom connection. It has one degree of freedom. The second constraint is for bucket connection. It has one degree of freedom. Both constraints have rotational freedom.

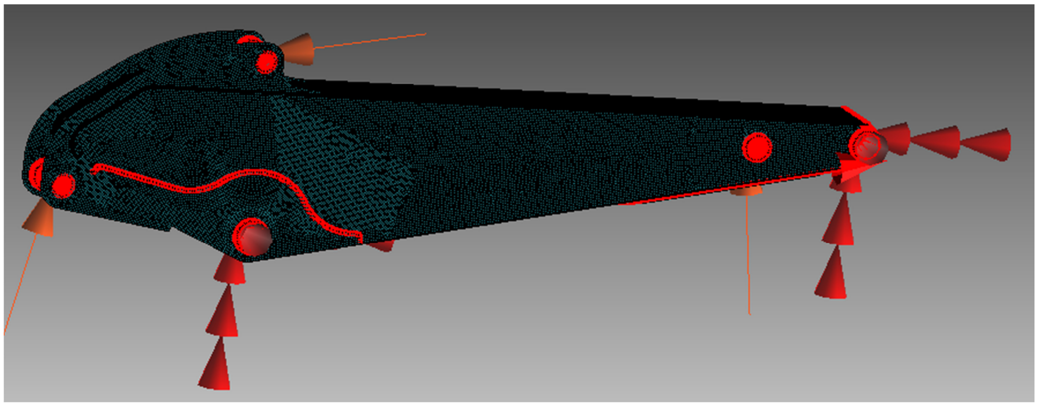
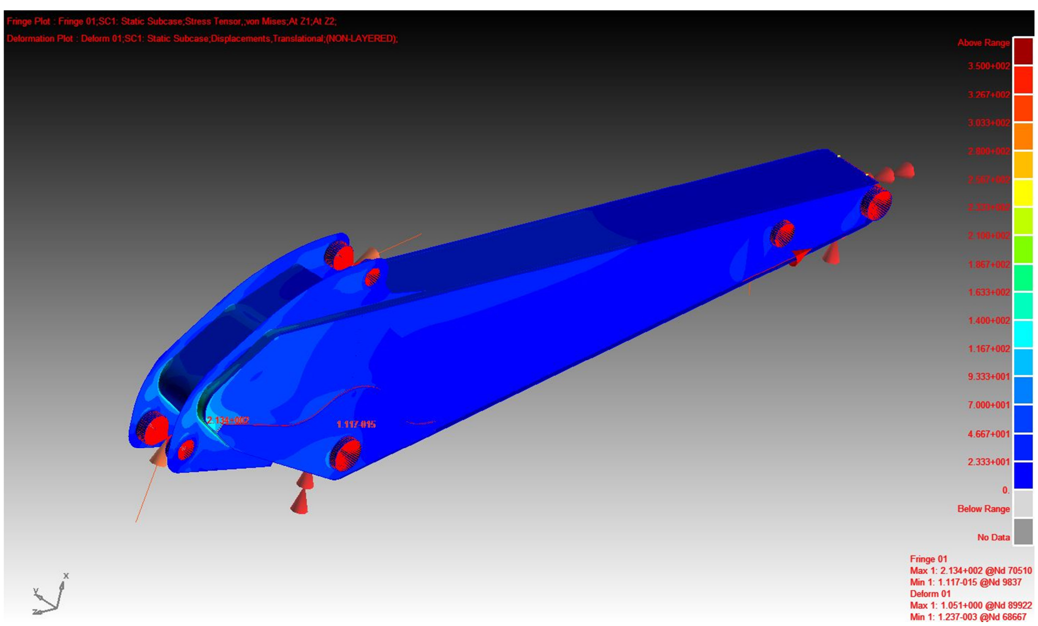
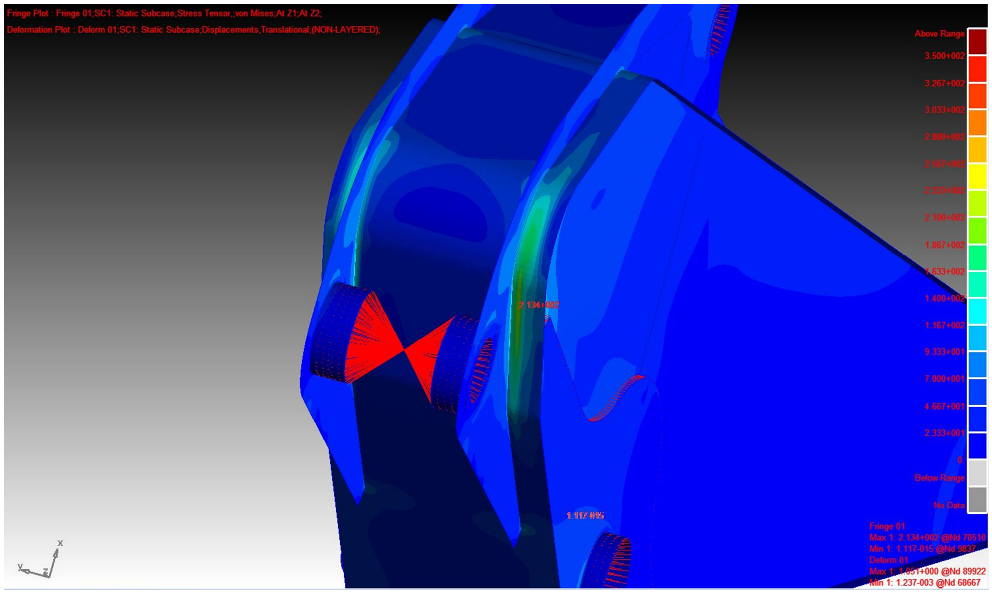


Figure 5.4 General view of arm model

Arm and bucket cylinder force (from equations 3.5 and 3.8) are applied to boom. Some revisions are performed according to analysis results. The results are shown below;



(a)



(b)

Figure 5.5 First analysis result. (a) Stress distribution of arm (b) Stress zone of arm

First analysis results showed that high stress (213 MPa) occurs between arm cylinder connection plate and arm back plate (Figure 5.5).

Changes made on the first revision are as follows;

Thickness of cylinder connection plate that is pointed red arrow is changed (Figure 5.6). It is increased to 36 mm, about 20 percent.

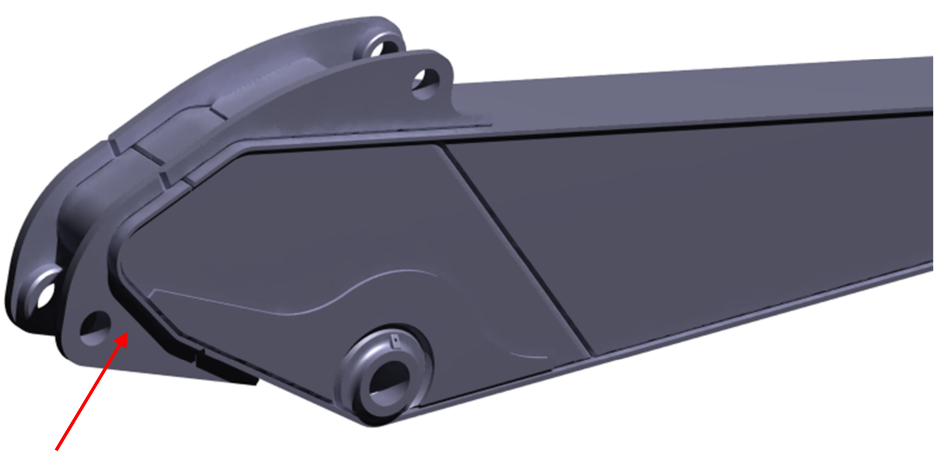
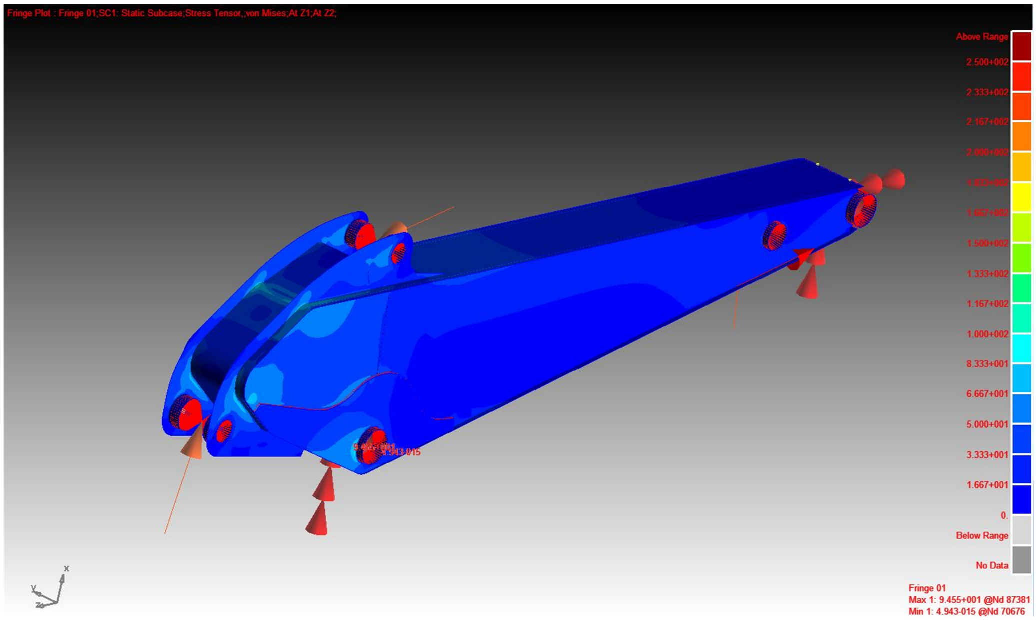
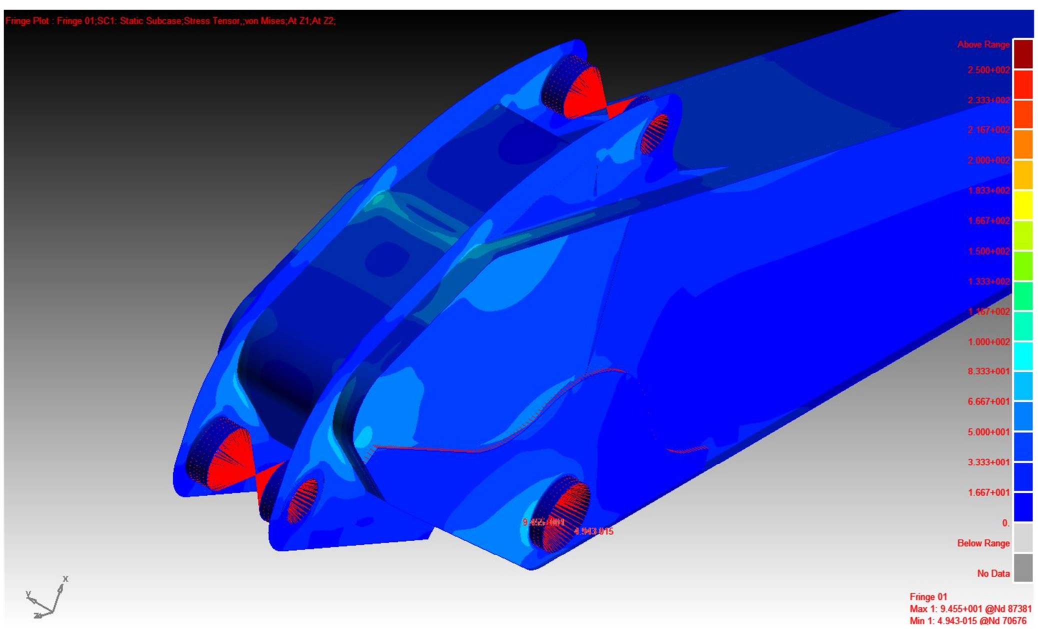


Figure 5.6 First revision of the arm



(a)



(b)

Figure 5.7 First revision analysis result. (a) Stress distribution of revision 1 arm (b) Stress zone of revision 1 arm

Stresses in the critical area are decreased to 95 MPa (Figure 5.7). Thus, this revision is accepted.

Changes made on the second revision are as follows;

Thickness of cylinder connection plate that is changed its thickness in revision 1 is increased nearly 7 percent. At the end of this revision, results did not changed stress noticeably (Figure 5.8). So, rev. 2 is accepted and rev. 1 is canceled.

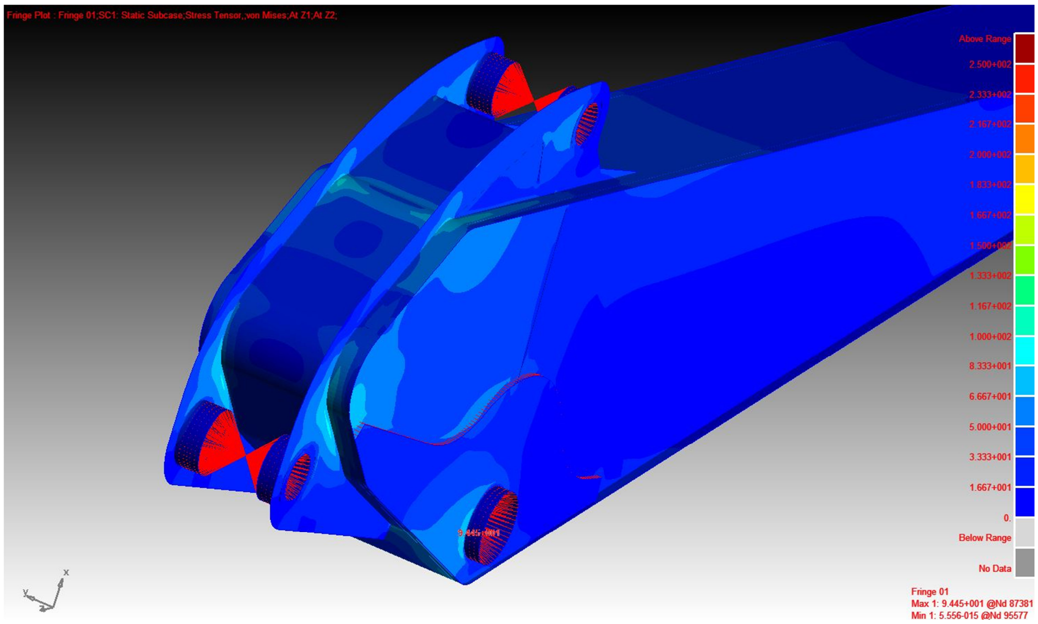


Figure 5.8 Stress zone of rev. 2 arm

When the analysis results are examined, some area seems very safe. In this case, the study on the optimization (opt.) of arm weight is started. Thickness of the parts that are very safe (too low stress) are decreased.

Changes made on the third revision are as follows;

In third revision, the thickness of plate that is marked is thinned to 10 mm, nearly 12 percent (Figure 5.9).

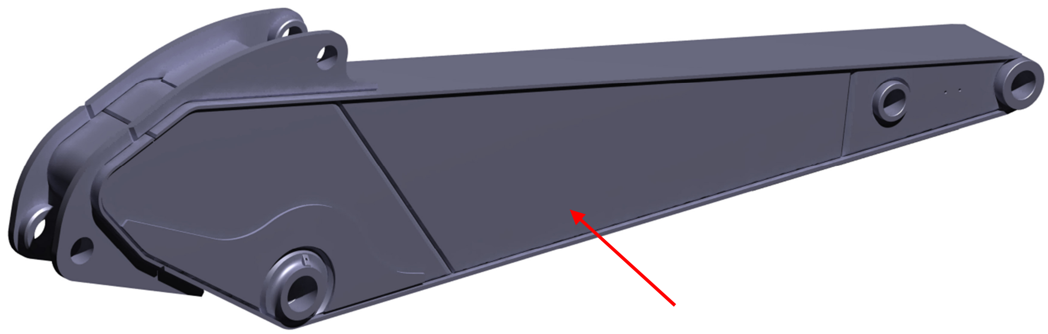


Figure 5.9 Third revision of arm

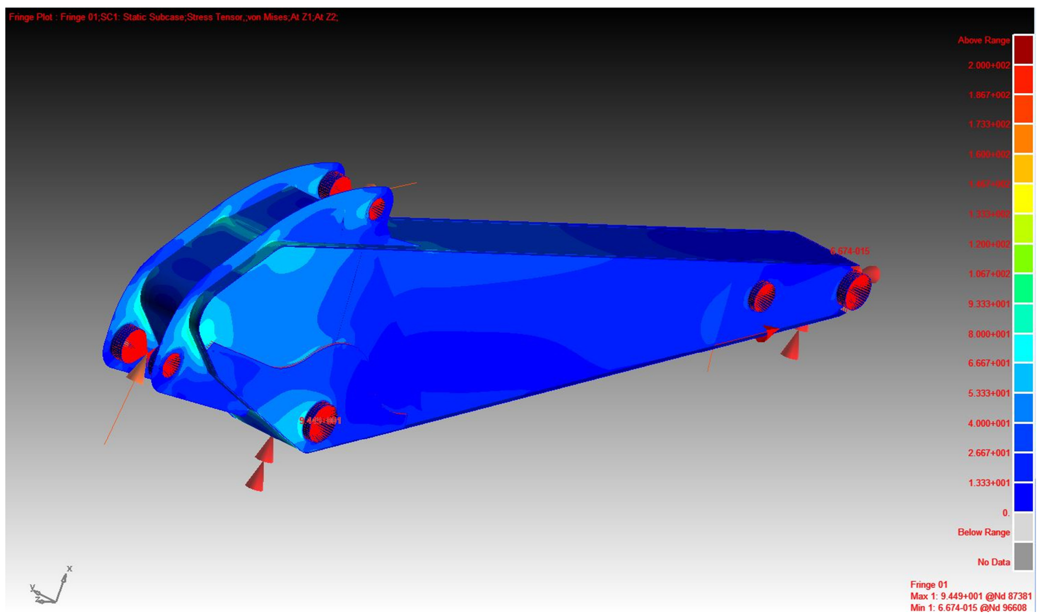


Figure 5.10 Stress distribution of revision 3 arm

The stress in the critical area increased but remained at acceptable level (95 MPa) at the end of this revision (Figure 5.10). So, the third revision is accepted.

Changes made on the fourth revision are as follows;

The thickness of plate that is marked is thinned to 15 mm, nearly 20 percent (Figure 5.11).

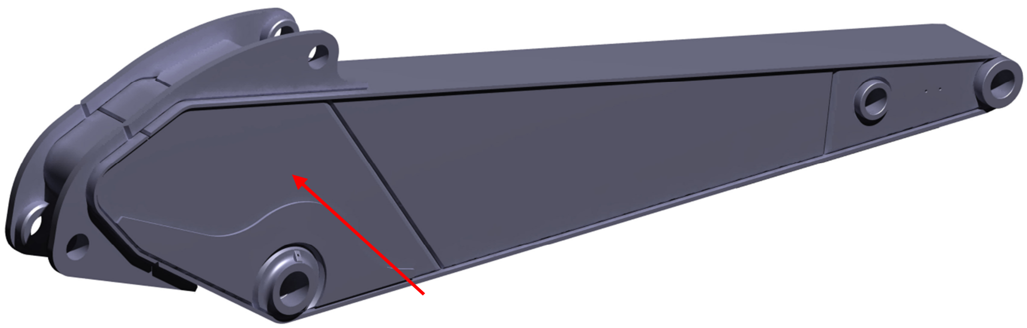


Figure 5.11 Fourth revision of arm

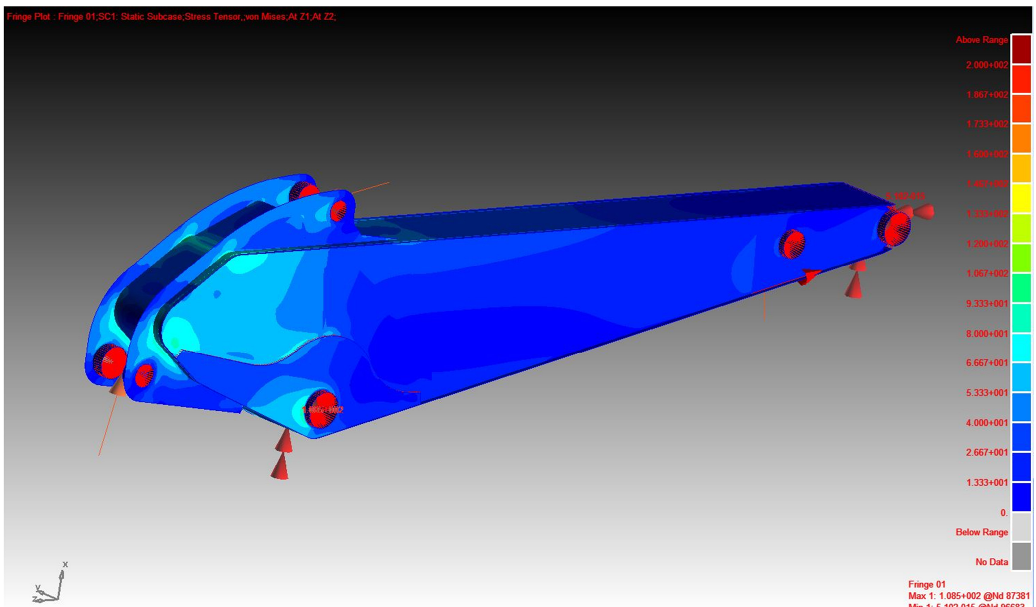


Figure 5.12 Stress distribution of revision 4 arm

The maximum stress increased to 109 MPa. This is approximately 15 percent (Figure 5.12). But the arm model is also in safe area hence the value is acceptable.

Changes made on the fifth revision are as follows;

Plate thickness of revision 4 is thinned 40 percent (Figure 5.13).

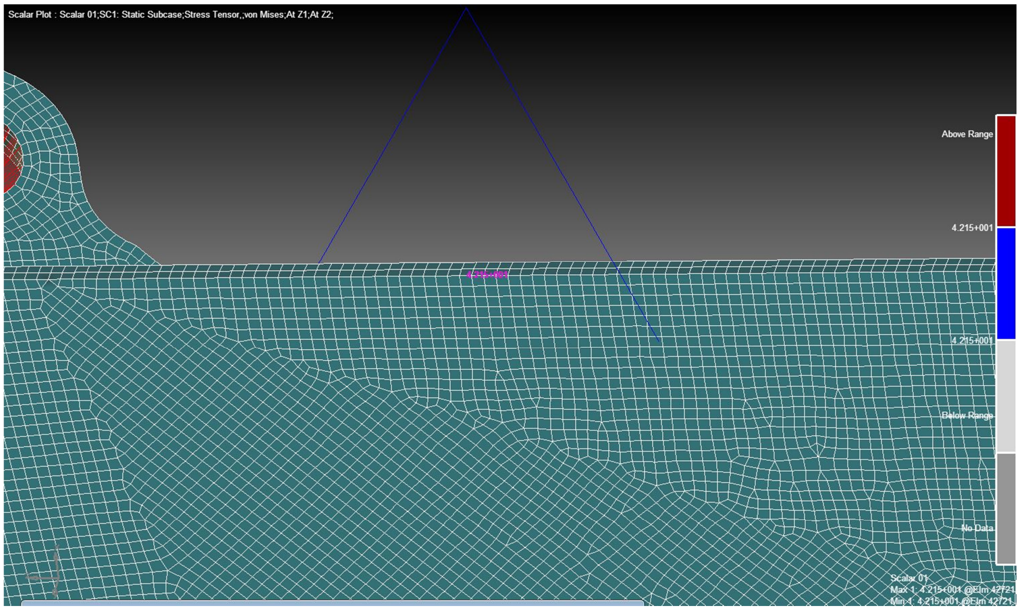


Figure 5.13 Specific element on thinner plate

Maximum stress in the thinner plate did not changed a lot (Figure 5.13). This is approximately 109 MPa. So, the opt. 3 is accepted.

Changes made on the sixth revision are as follows;

The thickness of marked plate is decreased to 12 mm, nearly 20 percent (Figure 5.14).

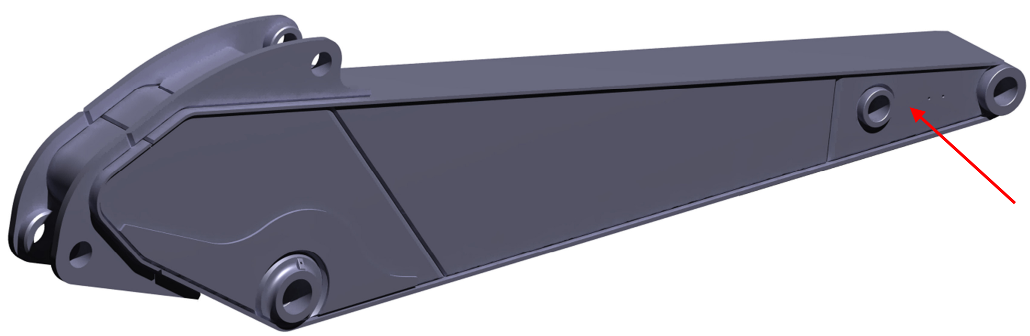
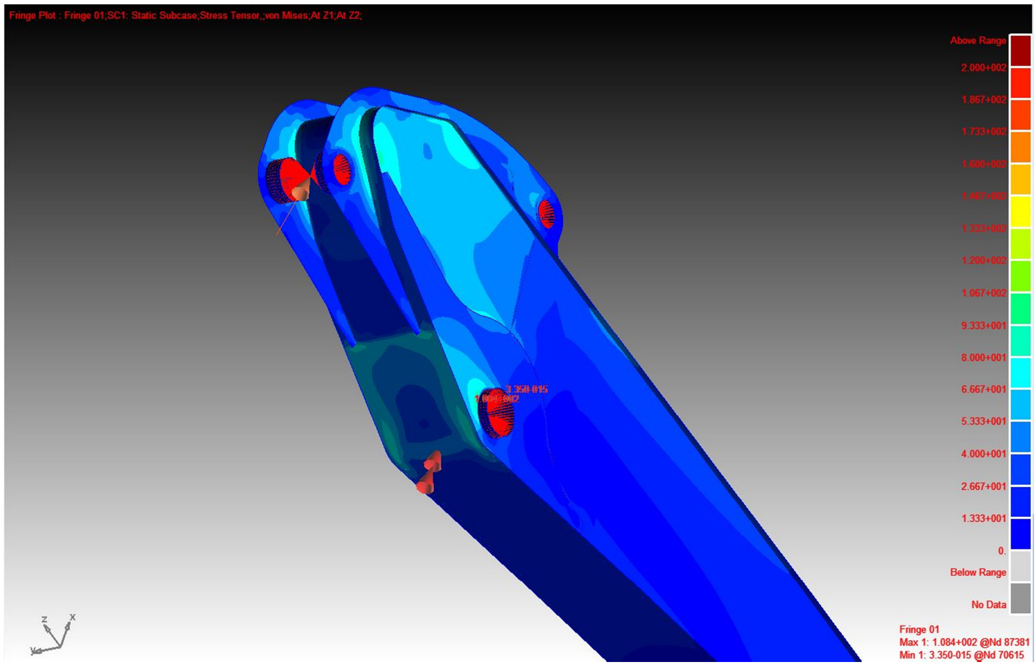
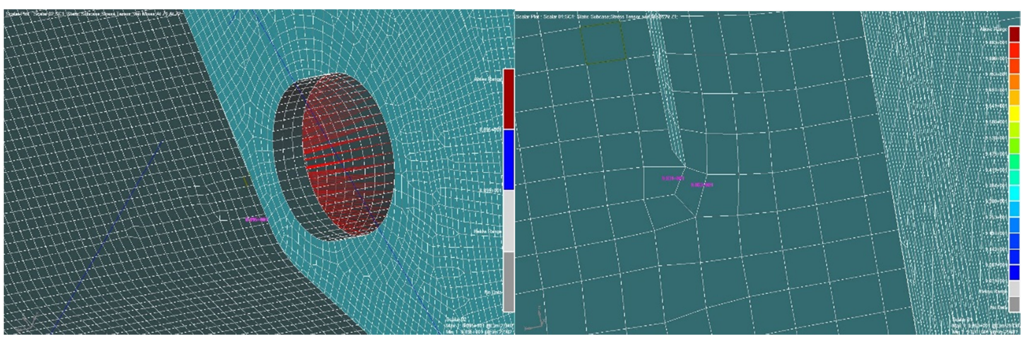


Figure 5.14 Sixth revision of arm



(a)



(b)

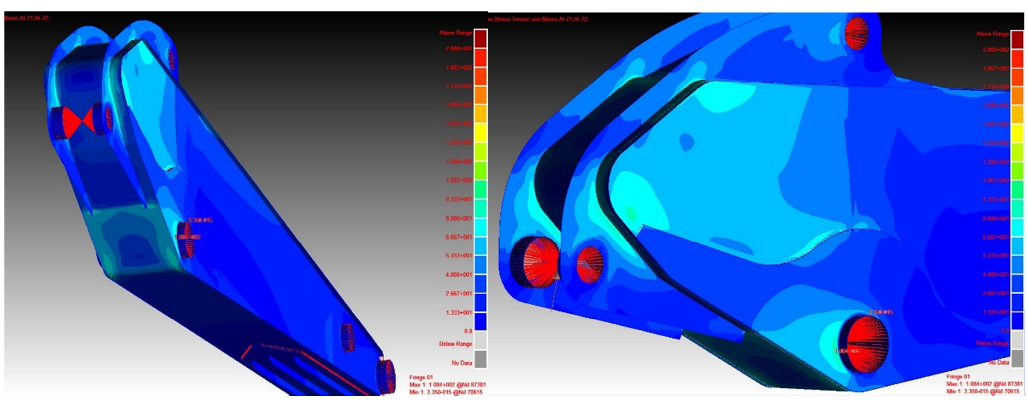
Figure 5.15 Sixth revision of arm. (a) Stress zone of revision 6 arm (b) Specific element on revision 6 arm

The stress on the bottom plate increased to 107 MPa (Figure 5.15). But the values are in acceptable range. Thus, it is accepted.

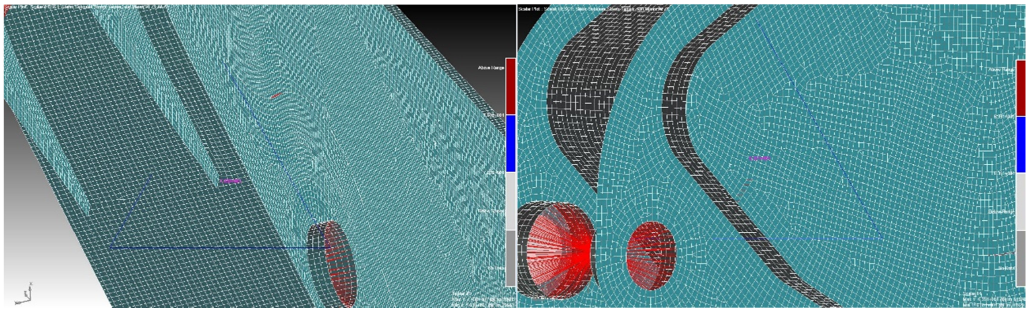
Changes made on the seventh revision are as follows;

Here, thickness of bottom plate that is indicated with arrow is thinned to 12 mm, nearly 20 percent (Figure 5.16).

Figure 5.16 Seventh revision of arm



(a)



(b)

Figure 5.17 Seventh revision of arm. (a) Stress zone of revision 7 arm (b) Specific element on revision 7 arm

The stresses on some locations increased to 108 MPa (Figure 5.17). But the results are still in acceptable range. So the opt. 5 is accepted.

Changes made on the eighth revision are as follows;

Plate thickness of opt. 3 is thinned nearly 33 percent. In addition to that, plate thickness of opt. 4 is also thinned nearly 15 percent.

According to the results of this revision, the stress values are acceptable (Figure 5.18). This is approximately 131 MPa.

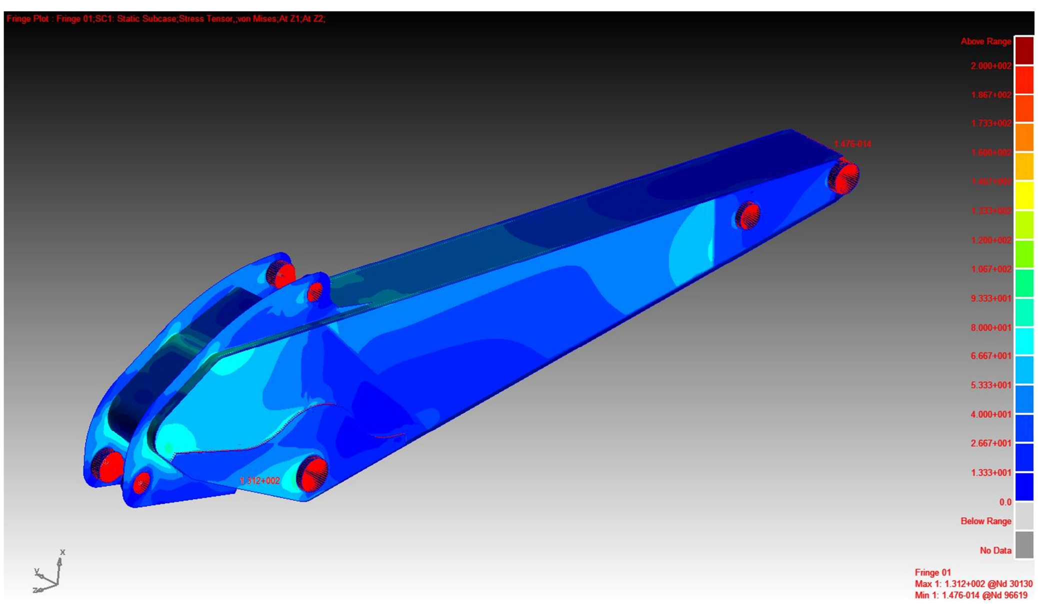


Figure 5.18 Stress distribution of revision 8 arm

Revision 8 is accepted as final model. The results are summarized at Table 5.1.

Table 5.1 Summary table of results

Reinforcement	Stress value (MPa)	Improvement (%)	Factor of safety
First situation	213		1.67
Rev. 1	94.55	55.6	3.75
Rev. 2	94.45	55.7	3.76
Rev. 3	94.49	55.6	3.76
Rev. 4	108.5	49.1	3.27
Rev. 5	108.4	49.1	3.27
Rev. 6	107.6	49.5	3.30
Rev. 7	108.4	49.1	3.27
Rev. 8	131	38.5	2.71

Summary table helps comparing the results. It shows improvements gradually. Optimization results are shown. Also, this table highlights effects of revisions in terms of factor of safety.

5.3 Conclusion

The analysis of arm were carried out according to the FEM steps. Risk parts were determined between arm back plate and rod side of arm cylinder of connection plate. Some improvements were applied to minimize high stress values. Result of this studying, the stress values were decreased gradually. Also, it was observed on some parts that have lower stress value. These parts have high factor of safety. So, some optimizations were applied on these parts. Factor of safety 2.71 was chosen. Because a lighter product was obtained and it is in acceptable factor of safety.

CHAPTER 6

DYNAMIC ANALYSIS OF FRONT ATTACHMENT

6.1 Introduction

Static analysis was completed on the first stage, in the previous two chapters. Critical areas were improved. Some optimizations were applied. But the components or systems are not generally under static conditions in real working conditions. In this chapter dynamic analysis of the front assembly will be carried out to compare with outcomes of the static analyses. Hence the complete design of front attachment of the excavator will be completed.

Dynamic analysis is important to work, observe and check reliability of moving parts. If the analysis is carried on assembly as it work on real case, this is most famous called multibody dynamics (MBD). It is also important when forces act on a moving part is depending on the time. The other important phenomenon is acceleration. Acceleration of front attachment changes while the cylinders move. So, the mass and velocity that is called momentum affects the analysis results. In addition to that, it permits different loading conditions for FEA [20]. Adams/Flex software is utilized at this point.

Adams/Flex provides the technology to correctly include a component's flexibility even in presence of large overall motion and complex interaction with other modeling elements. Greater emphasis has been placed these days on high-speed, light-weight, precise mechanical systems. Often these systems will contain one or more structural components where deformation effects are paramount for design analyses and the rigid body assumption is no longer valid. Adams/ flex allows importing finite element models from most major FEA software packages and is fully integrated with Adams package providing access to convenient modeling and powerful post-processing capabilities [20].

Adams/Flex uses a file that is known as flexible body. Rigid bodies use while the simulation creates. When the simulation is run, the flexible file (modal neutral-format file, mnf) is imported. Flex body use Craig-Bampton method. This method used for reducing the size of a finite element model. Also [21];

- ✓ It combines motion of boundary points with modes of the structure assuming the boundary points are held fixed
- ✓ It accounts for both mass and stiffness
- ✓ Problem size defined by frequency range
- ✓ It allows for different boundary conditions at interface

6.2 Dynamic Analysis

When dynamic model (Figure 6.1) is to be created, the working conditions of the mechanical system are necessary to know very well. Wrong results could be taken if simulation does not generate according to working conditions.

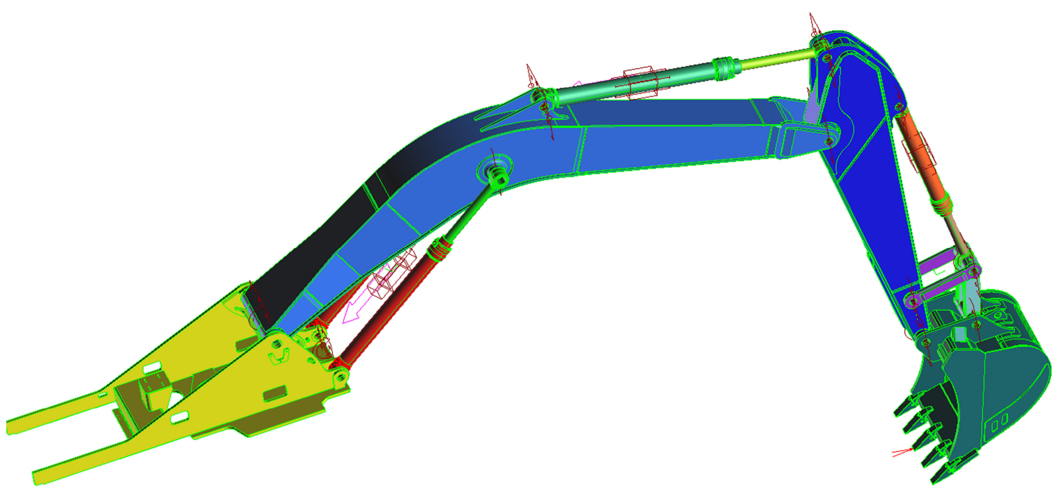


Figure 6.1 General view of dynamic model

6.2.1 Assumptions

- ➢ Material behavior is linear elastic.
- ➢ Deformations are small.
- ➢ Structure is in static equilibrium.
- ➢ Structure is initially unloaded

➢ Structure is free of initial and residual stress.

6.2.2 Dynamic Loads

In order to dynamic analysis, a simulation solid model must be prepared. This simulation must be suitable to actual conditions. Because of this reason, measurement tests are done for cycle time of arm, boom and bucket cylinder.

Arm and bucket breakout forces are calculated with formulations 3.6 and 3.9. These forces must be applied at maximum force position in simulation (Figure 6.2). Teeth are inserted at the end of bucket. The forces are applied at the end of five teeth.

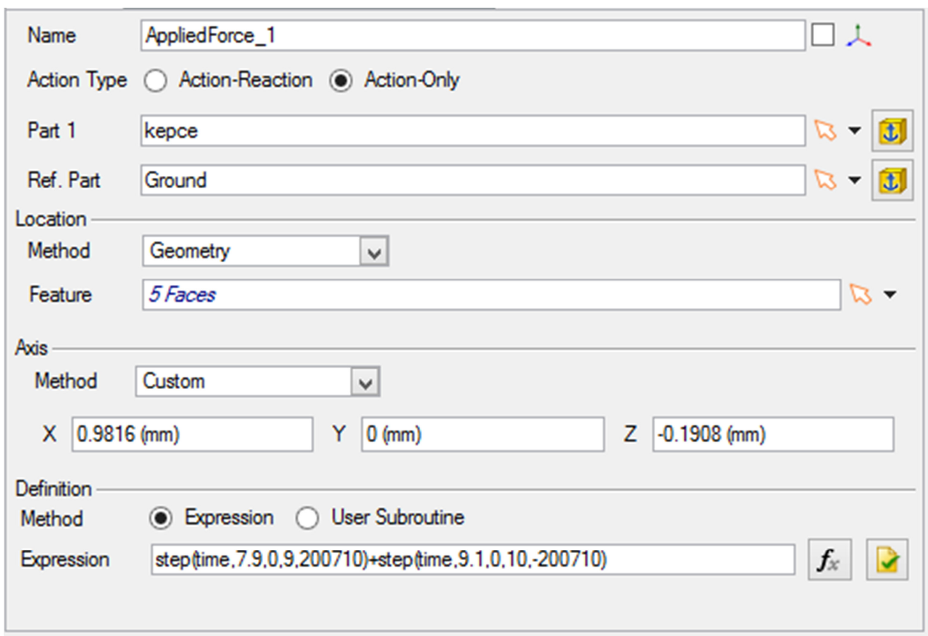


Figure 6.2 Application of force

Three different joint motions are established according to cylinder cycle time and maximum force positions (Figure 6.3, Figure 6.4). These joint motions have properties of translation and displacement motion.

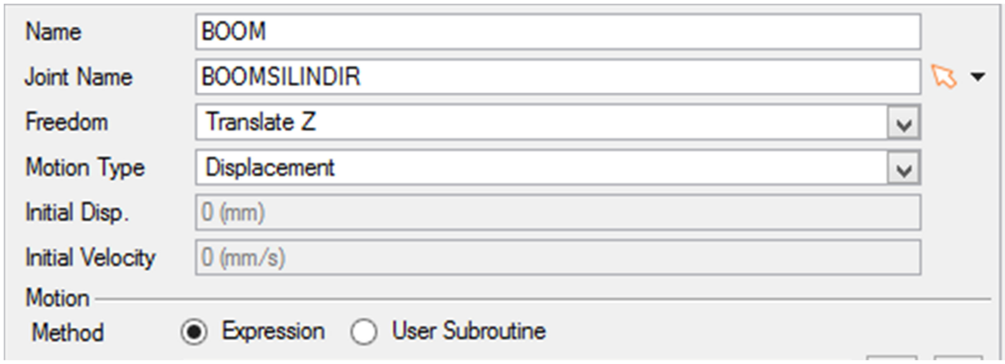


Figure 6.3 Boom cylinder joint motion

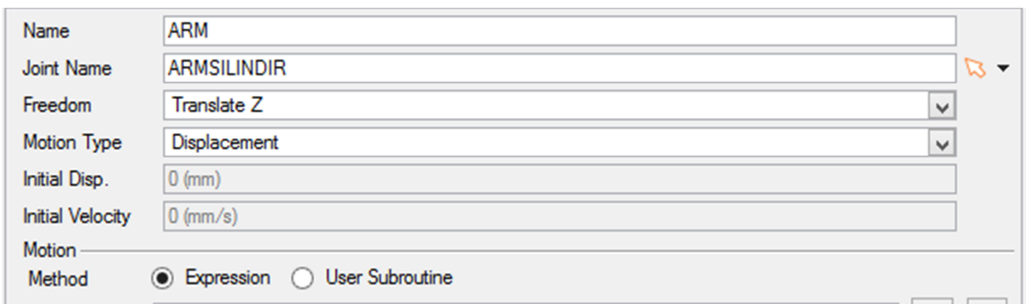


Figure 6.4 Arm cylinder joint motion

The displacement motion is achieved via translational motion. Moment of inertia is important here. So, acceleration of translational movement affects the results. For this reason, some measurements are made on an excavator machine. The cylinder opening and closing speed is measured by chronometer. This is also called cycle time. The measured values are shown in table 6.1.

Table 6.1 Cycle time results

Cylinder	Type of Movement		Results (second)
Boom	Extension (Cylinder opening)	1.	6.99
		2.	7.13
		3.	6.98
		Average	7.03
	Retraction (Cylinder closing)	1.	4.63
		2.	4.75
		3.	4.74
		Average	4.71
Arm	Extension (Cylinder opening)	1.	3.04
		2.	3.34
		3.	3.22
		Average	3.20
	Retraction (Cylinder closing)	1.	3.14
		2.	3.25
		3.	3.18
		Average	3.19
Bucket	Extension (Cylinder opening)	1.	3.06
		2.	3.30
		3.	3.26
		Average	3.21
	Retraction (Cylinder closing)	1.	2.83
		2.	2.76
		3.	2.70
		Average	2.76

The average values are found with use of three measurements. While the simulations are prepared.

6.2.3 Mesh Properties

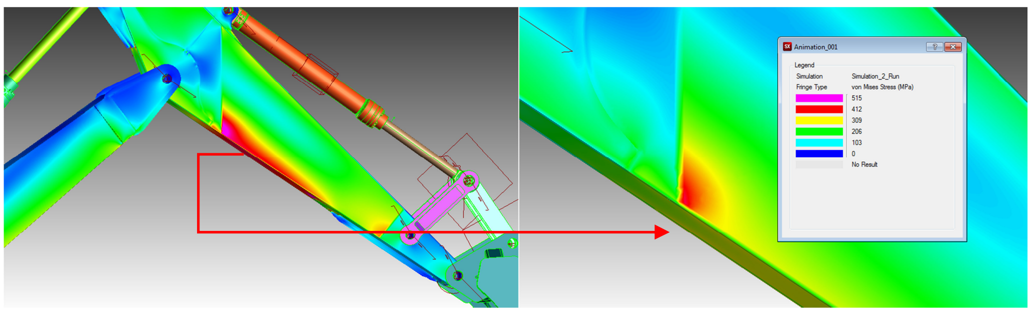
A different mesh application does not apply for dynamic analysis. While the simulation are generating, solid model of components are used. After the simulation is complete, modal neutral files (mnf) are created as output file by the Nastran. This output file gives displacements, velocities, accelerations or loading profiles [19]. The mnf file is used for parts to be examined. The other parts remain solid in the simulation.

6.2.4 Boundary Conditions

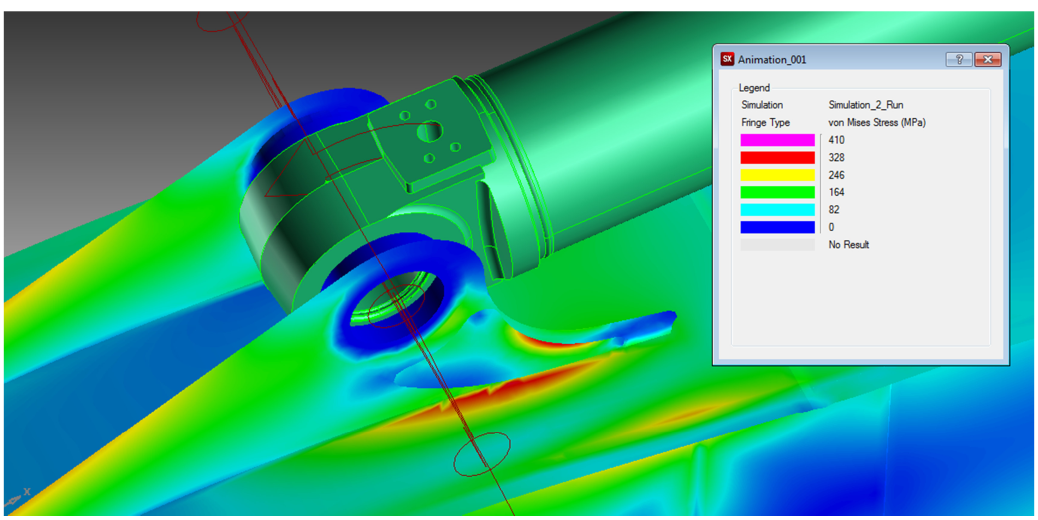
There are fifteen connections and three motions in simulation. Connections consist of twelve revolute and three translational. Motions consist of three joint motions.

6.3 Dynamic Analysis Results

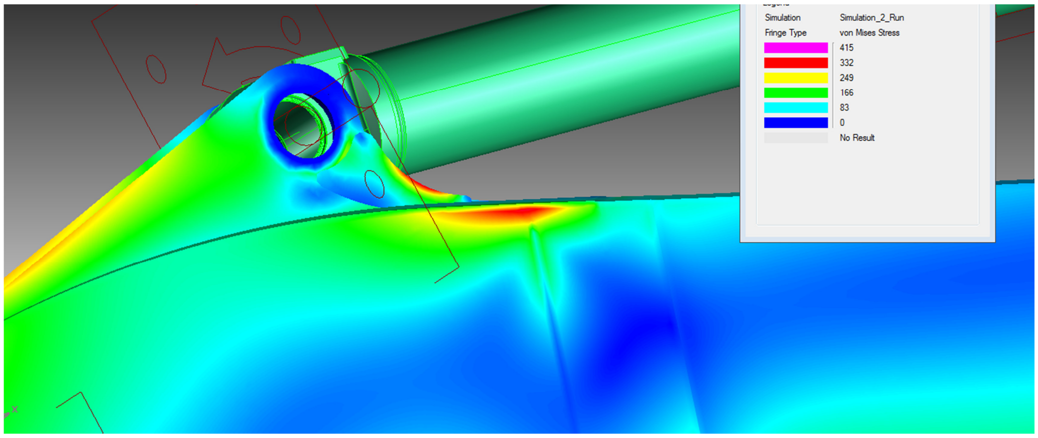
First analysis result showed that the boom and arm is not at safe area (Figure 6.5). These critical regions must be improved. Stress values are 412 MPa for figure 6.5a, 328 MPa for figure 6.5b, 332 MPa for figure 6.5c.



(a)



(b)

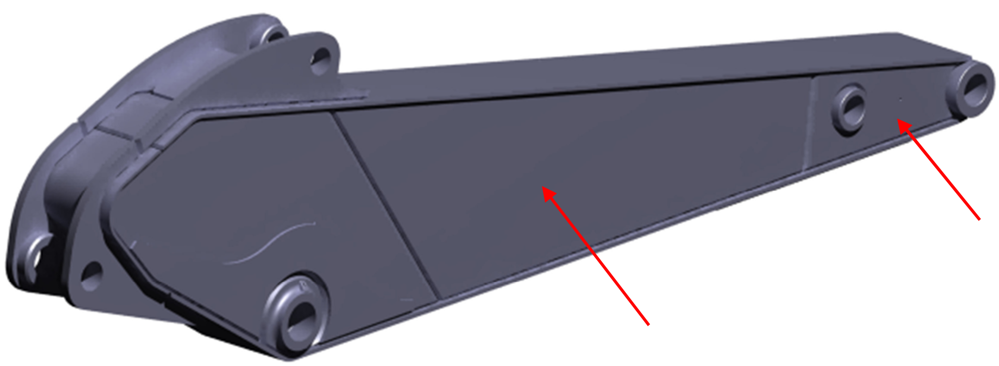


(c)

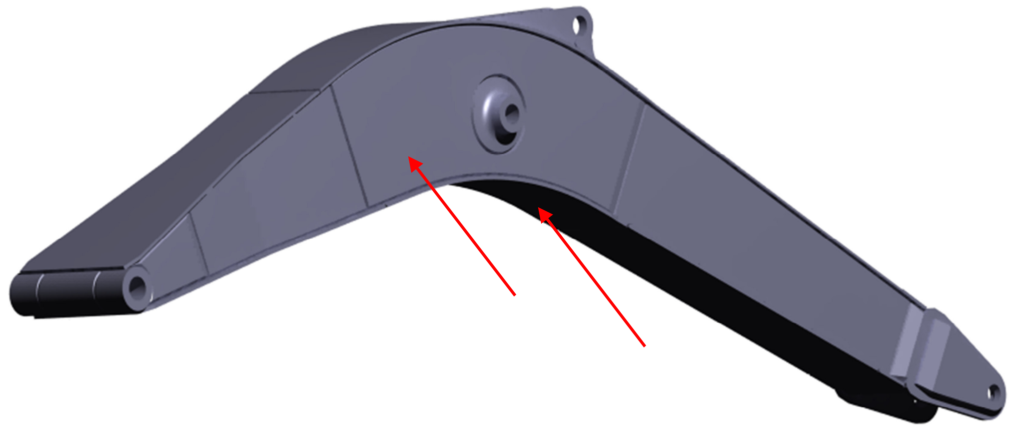
Figure 6.5 Results of first dynamic analysis. (a) Stress zone on the arm side plate (b) Stress zone on the boom upper plate (c) Stress zone on the boom side plate

Changes made on the first revision are as follows;

Thickness of arm left marked plate is increased from 4 to 8 mm. Thickness of arm right marked plate is increased from 8 to 10 mm. Thickness of boom left marked plate is increased from 20 to 22 mm. Thickness of boom right marked plate is increased from 20 to 22 mm (Figure 6.6).

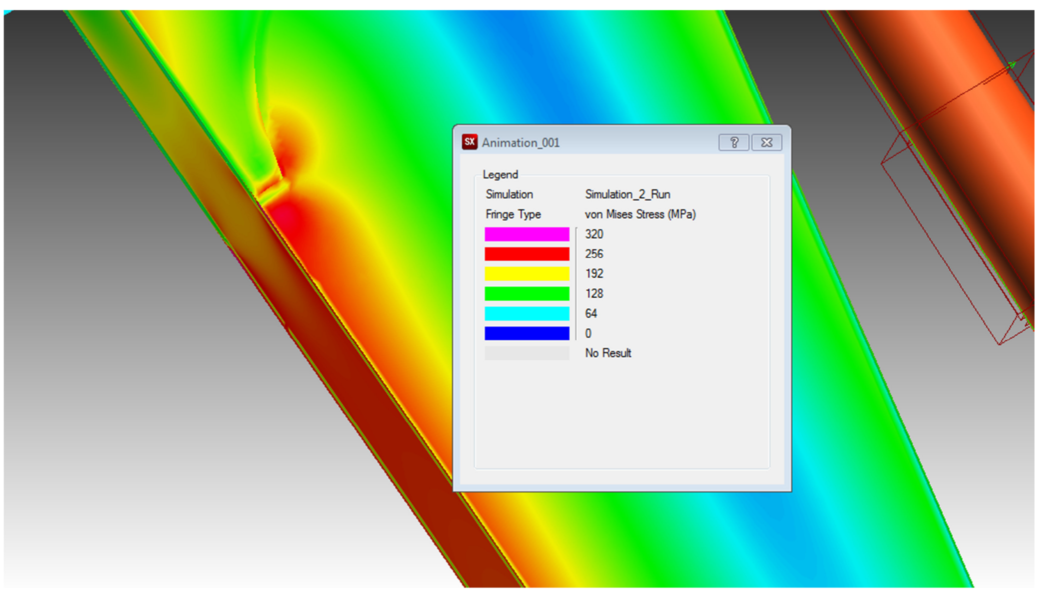


(a)

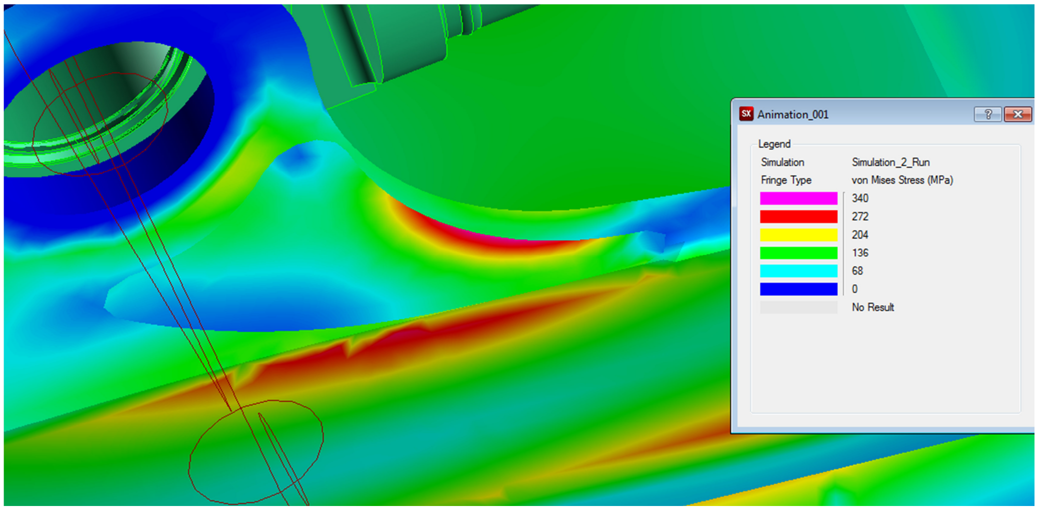


(b)

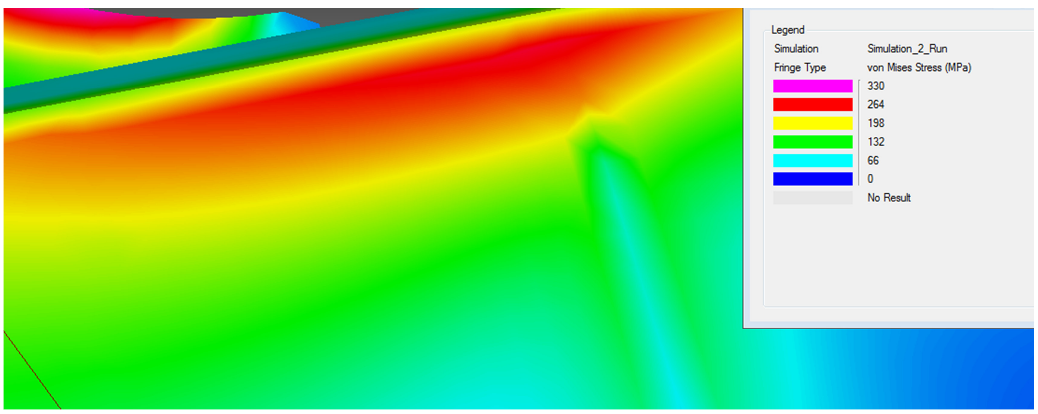
Figure 6.6 First revision of boom and arm. (a) Revision 1 on the arm for dynamic analyze (b) Revision 1 on the boom for dynamic analyze



(a)



(b)



(c)

Figure 6.7 Results of first revision dynamic analysis. (a) Stress zone on the revision 1 arm side plate (b) Stress zone on the revision 1 boom upper plate (c) Stress zone on the revision 1 boom side plate

Rev. 1 decreased the stresses positively (Figure 6.7). But it is not enough. So the improvements are decided to be continued. Stress value is 256 MPa for figure 6.5.a, 272 MPa for figure 6.5.b, 264 MPa for figure 6.5.c.

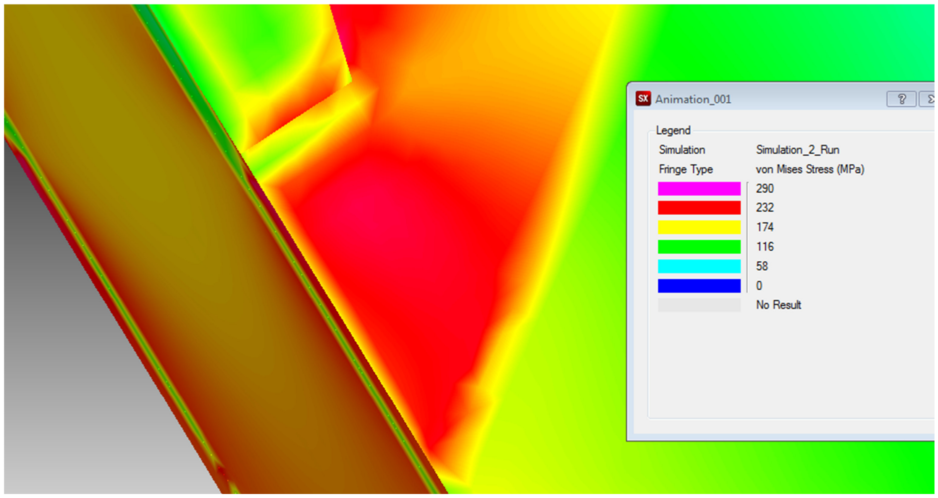
Changes made on the second revision are as follows;

Thickness of arm left marked plate is increased from 8 to 12 mm. thickness of arm right marked plate is increased from 12 to 14 mm. Thickness of boom marked plate is increased from 22 to 24 mm (Figure 6.8).

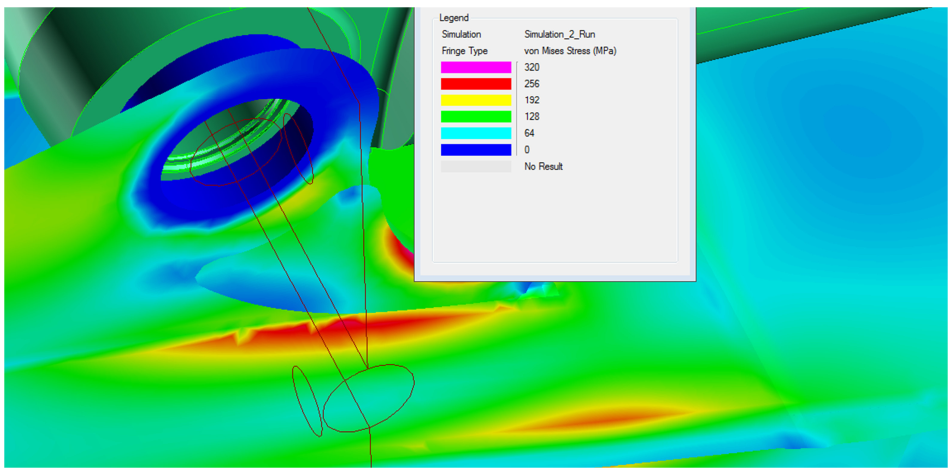
(a)

(b)

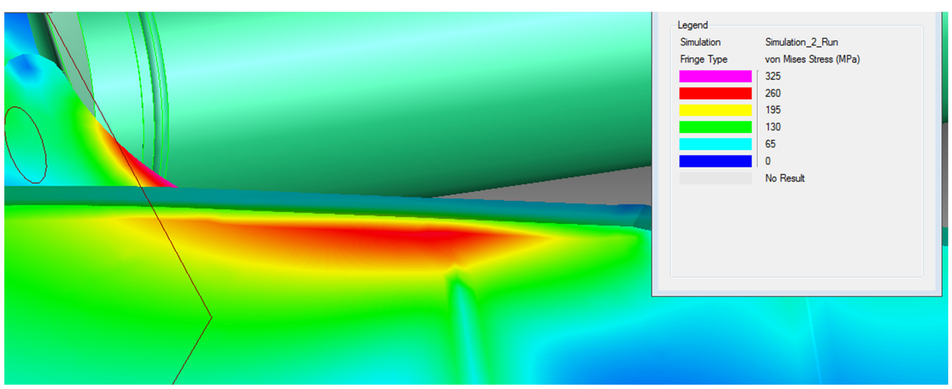
Figure 6.8 Second revision of boom and arm. (a) Revision 2 on the arm for dynamic analyze (b) Revision 2 on the boom for dynamic analyze



(a)



(b)



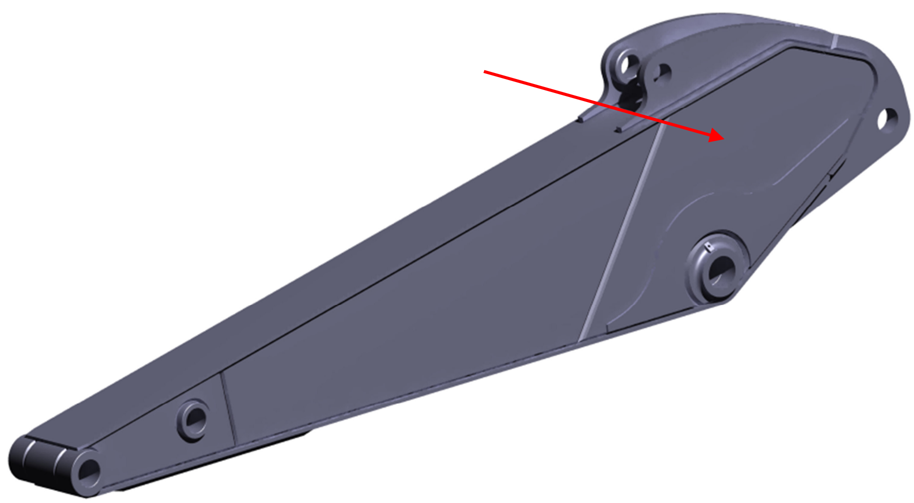
(c)

Figure 6.9 Results of second revision dynamic analysis. (a) Stress zone on the revision 2 arm side plate (b) Stress zone on the revision 2 boom upper plate (c) Stress zone on the revision 2 boom side plate

Revision 2 is affected positively (Figure 6.9). But expected improvement cannot be achieved. Stress value is 232 MPa for figure 6.5a, 256 MPa for figure 6.5b, 260 MPa for figure 6.5c.

Changes made on the third revision are as follows;

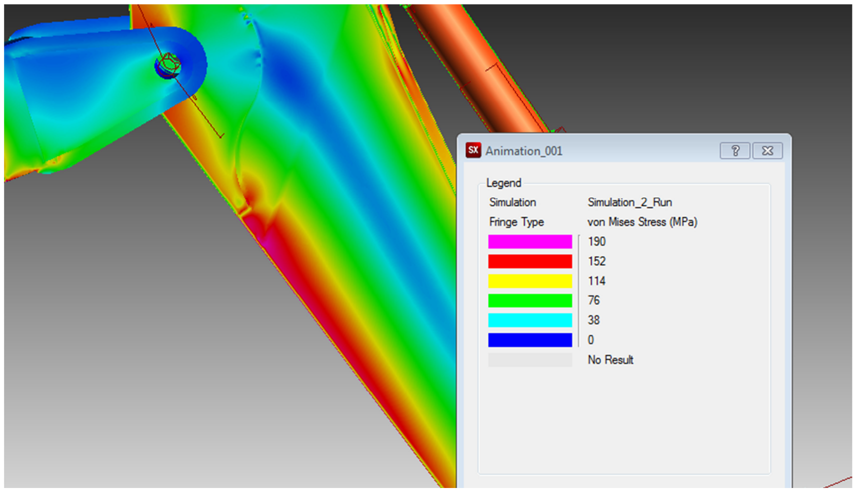
In the revision, thickness of arm plate is changed to 14 mm (Figure 6.10).



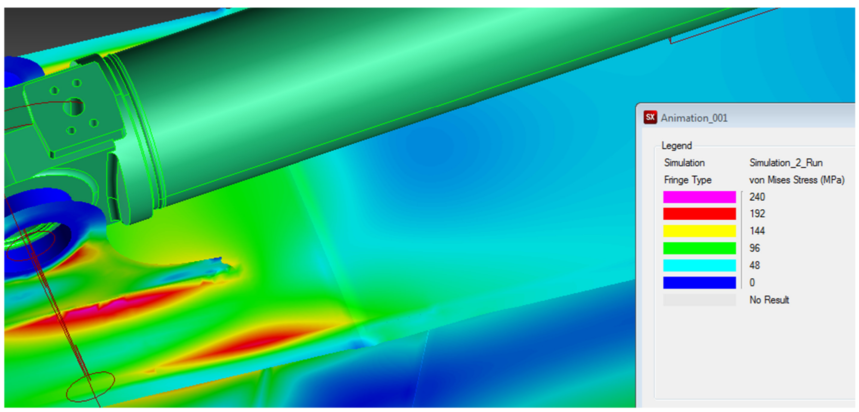
(a)

(b)

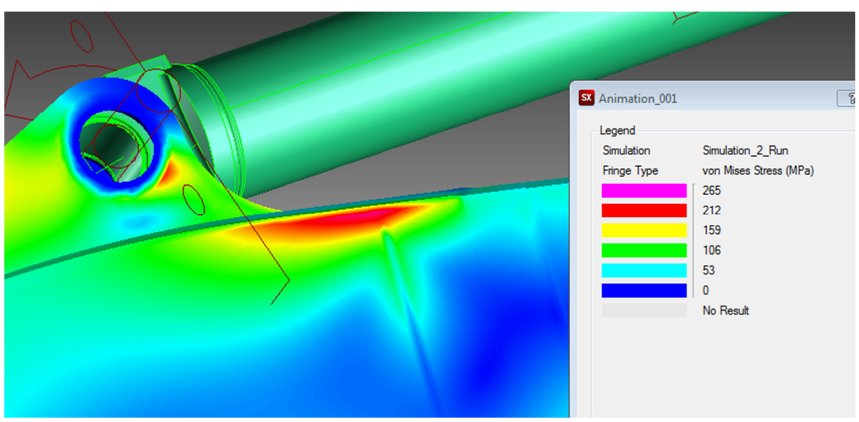
Figure 6.10 Third revision of boom and arm. (a) Revision 3 on the arm for dynamic analyze (b) Revision 3 on the boom for dynamic analyze



(a)



(b)



©

Figure 6.11 Results of third revision dynamic analysis. (a) Stress zone on the revision 3 arm side plate (b) Stress zone on the revision 3 boom upper plate (c) Stress zone on the revision 3 boom side plate

Results of revision three are 152 MPa for figure 6.5a, 192 MPa for figure 6.5b, 212 MPa for figure 6.5c.

Changes made on the fourth revision are as follows;

Thickness of below marked plate was increased (Figure 6.2). At the end of this revision. The results approached to the safe zone.

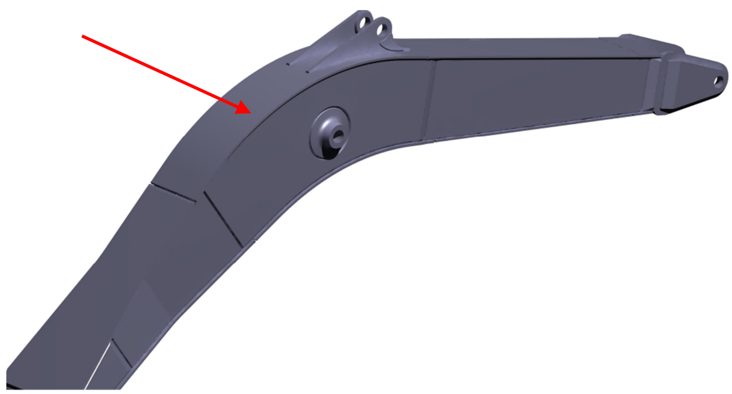
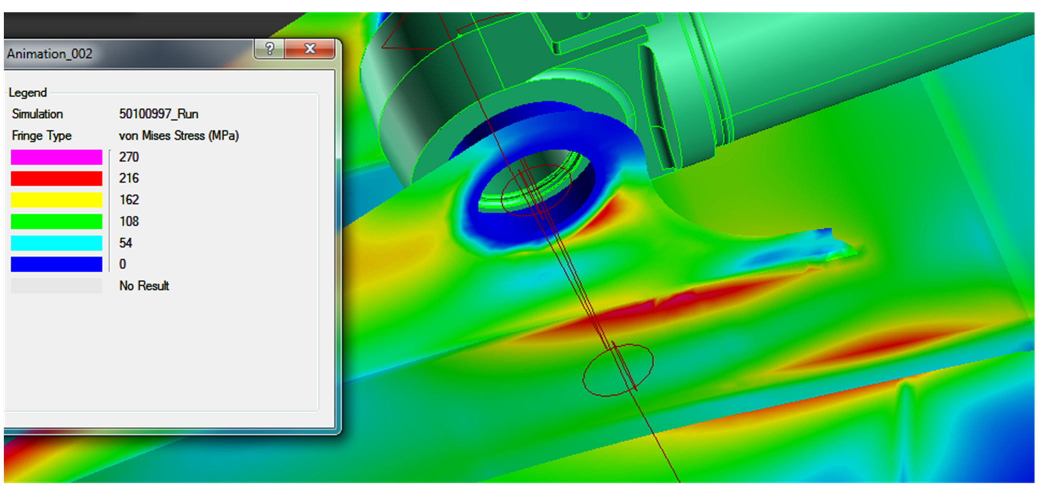
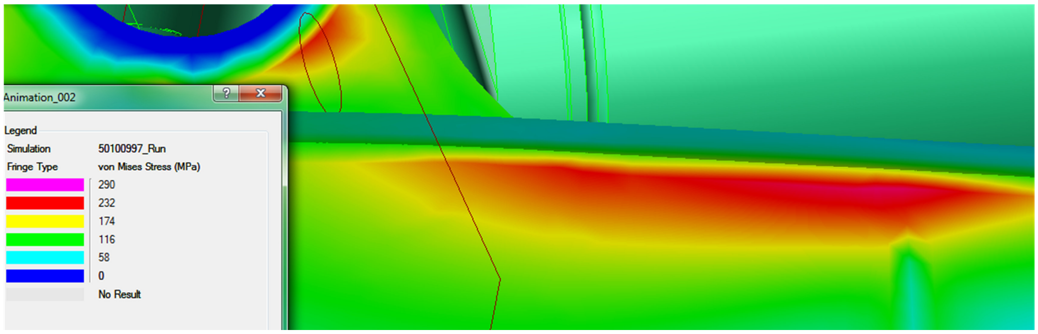


Figure 6.12 Rev. 4 on the boom for dynamic analyze

According to the results, rev. 4 was canceled (Figure 6.13). Final design is rev. 3 for dynamic analysis. Stress value is 216 MPa for figure 6.5a, 232 MPa for figure 6.5b.



(a)



(b)

Figure 6.13 Results of fourth revision dynamic analysis. (a) Stress zone on the revision 4 boom upper plate (b) Stress zone on the revision 4 boom side plate

The difference between first and last case is as follows;

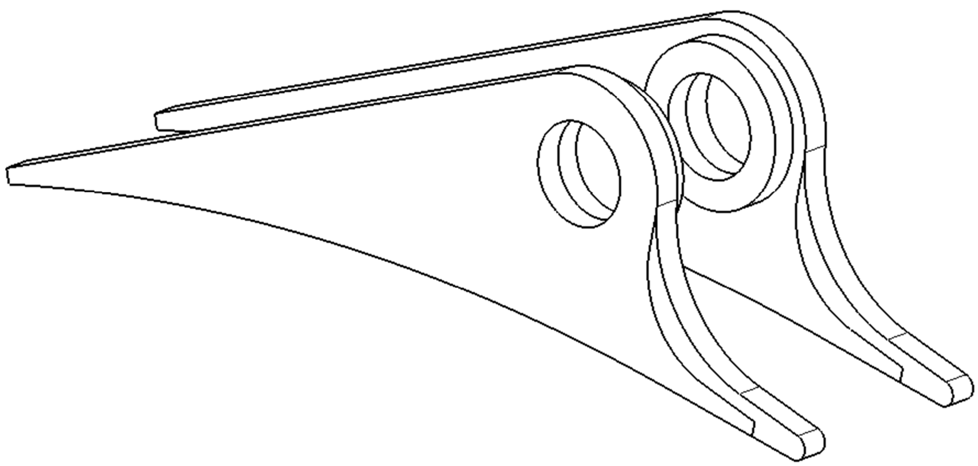


Figure 6.14 Comparing of cases

The results are summarized by a table, Table 6.2. The results can be compared according to this that table.

Table 6.2 Summary table of results

Description	Revisions	Stress (Mpa)	Factor of Safety
Stress zone on the arm side plate	First situation	412	0.86
	Rev. 1	256	1.39
	Rev. 2	232	1.53
	Rev. 3	152	2.34
Stress zone on the boom upper plate	First situation	328	1.08
	Rev. 1	272	1.31
	Rev. 2	256	1.39
	Rev. 3	192	1.85
	Rev. 4	216	1.64
Stress zone on the boom side plate	First situation	332	1.07
	Rev. 1	264	1.34
	Rev. 2	260	1.37
	Rev. 3	212	1.67
	Rev. 4	232	1.53

6.4 Conclusion

There are important points to be considered before starting a dynamic analysis. Modeling and simulation should be created as close as to reality. Firstly, BCs, cycle times of cylinders and application of forces were determined. Then, a simulation was generated based on working conditions of front attachment. In the first situation, the results indicated the difference between static and dynamic analysis. High stress values were observed in some regions which were accepted during static analysis. Some improvements were applied to these critical regions. Hence stress values were decreased gradually. After application of the modifications according to dynamic analysis, the front attachment become ready for manufacturing and safe operation.

CHAPTER 7

CONCLUSIONS

Structural analysis is very important for earthmoving machines. Components of these machines are moving parts. They are exposed to high loading during working. Thus, structural analysis should be done for reliable designs. Therefore, some studies have been done during this thesis. Studies were focused on static analysis initially. After that, dynamic analysis carried out.

In this study, strength analysis was performed on front attachment of the crawler excavator. In order to analyze welding parts, boundary conditions and forces must be determined. Front attachment was solved as kinematical. FBD was generated. First, boom and arm were analyzed with FEM by means of Simxpert interface that includes Nastran. Then these welding components were analyzed with FEM by means of Simxpert interface that includes Adams. Nastran was used for static analysis. Adams was used for dynamic analysis.

FEM which is numerical method was used in the structural analysis. It has chosen for strength analysis. It can give the results in a short time. Improvements can end up in a short time. Cycle time tests were also carried out experimentally.

At the end of static analysis, some critical stress values were observed. Thereupon, some improvements were applied. According to boom analysis results, maximum stress value was 349 MPa and factor of safety was 1.02 in the first situation. After improvements were applied, maximum stress value was decreased to 145 MPa. Factor of safety was increased to 2.45. According to arm analysis results, maximum stress value was 213 MPa and factor of safety was 1.67 in the first situation. After improvements were applied, maximum stress value decreased to 131 MPa. Factor of safety was increased to 2.71.

At the end of dynamic analysis, some critical stress values were observed. As a result of that, some improvements were applied. Maximum stress value in the boom was 328 MPa and factor of safety was 1.08 in the first situation. At the end of boom improvements, maximum stress value was decreased to 192 MPa. Factor of safety was increased to 1.85. Maximum stress value in the arm was 412 MPa and factor safety was 0.86 in the first situation. At the end of arm improvements, maximum stress value was decreased to 152 MPa. Factor of safety was increased to 2.33.

Parts in the dynamic work conditions expose to stress because of their own inertia which should be applied to structural analysis. This is why it was needed to dynamic analysis.

The results indicated importance of dynamic analysis when to be analyzed the moving parts. Dynamic analysis has shown that high stress values occur on the front attachment. Factor of safety is increased by improvements. As a result of this work, a safer product was provided.

FUTURE WORKS

In the future studies, working on front attachment’s kinematic is planned. In this wise, better breakout force and reduce of material usage will be aimed. Sweeping motion can also be considered during design and analyses. The sweeping force is going to apply to bucket. Hence it is going to apply to boom and arm also. Including sweeping force to analyses will yield complete real case analyses. Under these complete real case situations, application of material optimization procedure will result an optimum light weight but also a durable design.

REFERENCES

[1] Republic of Turkey General Directorate of Highways 4199/1. (1983). Highway traffic law. 73.p. Ankara.

[2] Oyman V. (2005). Ekskavatör kollarının tasarımı, a Major Project Report for the Degree of Master of Science in Mechanical Engineering, Yıldız Technical University.

[3] Fıçıcı F., Yener M., Arıkoğlu T., Söylemez E. (2003). İş makinaları sempozyumu ve sergisi.

[4] Yeter E. (2009). Improvement of backhoe-loader back and front arms, a Major Project Report for the Degree of Master of Science in Mechanical Engineering, Gaziantep University.

[5] Özer S. (2007). Kazıcı-yükleyicilerde kazma mesafesine bağlı olarak bom-stik grubunun tasarımı, a Major Project Report for the Degree of Master of Science in Mechanical Engineering, Mersin University.

[6] İpek L. (2006). Optimization of backhoe-loader mechanisms, a Major Project Report for the Degree of Master of Science in Mechanical Engineering, Middle East Technical University.

[7] Sharma A. (2005). Kinematics and finite element analysis of excavator attachments, Dr thesis, Nirma University of Science and Technology Institue of Science, Ahmedabad, 94s.

[8] Yener M. (2005). Design of computer interface for automatic finite element analysis of excavator boom, a Major Project Report for the Degree of Master of Science in Mechanical Engineering, Middle East Technical University.

[9] Solazzi L. (2010). Design of aluminum boom and arm for an excavator, *Journal of Terramechanics*, 47, 201-207.

[10] Rusinski E., Moczko P., Czmochowski J. (2008). Numerical and experimental analysis of a mine's loader boom crack, *Automation in Construction*, 17, 271-277.

[11] Bosnjak S. Zrnic N., Simonovic A., Momcilovic D. (2009). Failure analysis of the end eye connection of the bucket wheel excavator portal tie-rod support, *Engineering Failure Analysis*, 16, 740-750.

[12] Bosnjak S., Petkovic Z., Simonovic A., Zrnic N. (2013). 'Designing-in' failures and redesign of bucket wheel excavator undercarriage, *Engineering Failure Analysis*, 35, 95-103.

[13] Özbayramoğlu E. (2008). Shape optimization of wheeled excavator lower chassis, a Major Project Report for the Degree of Master of Science in Mechanical Engineering, Middle East Technical University.

[14] Uzer C. (2008). Shape optimization of an excavator boom by using genetic algorithm, a Major Project Report for the Degree of Master of Science in Mechanical Engineering, Middle East Technical University.

[15] Özünlü Ö. (2009). Dynamic modeling of an excavator during digging and simulating the motion, a Major Project Report for the Degree of Master of Science in Mechanical Engineering, Middle East Technical University.

[16] Kocabıyık R. (2010). Bir iş makinesi kabinine uygulanan dinamik analizlerin bilgisayar ortamında simülasyonu, a Major Project Report for the Degree of Master of Science in Mechanical Engineering, Yıldız Technical University.

[17] Kılıç B. (2009). Dynamic modelling of a backhoe-loader, a Major Project Report for the Degree of Master of Science in Mechanical Engineering, Middle East Technical University.

[18] SAE. Society of Automotive Engineers. (1995). Hydraulic excavator and backhoe digging force, J1179 SAE Handbook Volume 3, 40.355.

[19] MSC. MacNeal-Schwendler Corporation. (2012). MSC getting started with MSC Nastran user's guide.

[20] MSC. MacNeal-Schwendler Corporation. Software platform for MSC Simxpert, *www.mscsoftware.com.*, 27.01.2016.

[21] Gordan S. (1999). The Craig-Bampton Method, FEMCI Presentation, NASA Goddard Space Flight Center, May 6, http://analyst.gsfc.nasa.gıv/FEMCI/craig_bampton/.