SHAPE OPTIMIZATION OF AN EXCAVATOR BOOM BY USING GENETIC ALGORITHM

A THESIS SUBMITTED TO
THE GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES
OF
MIDDLE EAST TECHNICAL UNIVERSITY

BY

CEVDET CAN UZER

IN PARTIAL FULFILLMENT OF THE REQUIREMENTS
FOR
THE DEGREE OF MASTER OF SCIENCE
IN
MECHANICAL ENGINEERING

JUNE 2008

Approval of the Thesis

"SHAPE OPTIMIZATION OF AN EXCAVATOR BOOM BY USING GENETIC ALGORITHM"

Submitted by **CEVDET CAN UZER** in partial fulfillment of the requirements for the degree of **Master of Science in Mechanical Engineering** by,

Prof. Dr. Canan Özgen
Dean, Graduate School of **Natural and Applied Sciences** ________________

Prof. Dr. Kemal İder
Head of Department, **Mechanical Engineering** ________________

Prof. Dr. Eres Söylemez
Supervisor, **Mechanical Engineering, METU** ________________

Examining Committee Members:

Prof. Dr. Suat Kadıoğlu (*)
Mechanical Engineering, METU ________________

Prof. Dr. Eres Söylemez (**)
Mechanical Engineering, METU ________________

Prof. Dr. Haluk Darendeliler
Mechanical Engineering, METU ________________

Doç. Dr. Bora Yıldırım
Mechanical Engineering, METU ________________

Kadir Geniş (M.S.)
Hidromek Ltd. Sti. ________________

Date: ________________

(*) Head of Examining Committee
(**) Supervisor

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 fully cited and referenced all material and results that are not original to this work.

Name, Last name : Cevdet Can UZER

Signature :

ABSTRACT

SHAPE OPTIMIZATION OF AN EXCAVATOR BOOM BY USING GENETIC ALGORITHM

Uzer Cevdet Can
M.S., Department of Mechanical Engineering
Supervisor: Prof. Dr. Eres Söylemez

June 2008, 109 pages

This study concerns with the automated structural optimization of an excavator boom. The need for this work arises due to the fact that the preparation of the CAD model, performing finite element analysis and model data evaluation are time consuming processes and require experienced man power. The previously developed software OptiBOOM [35], which generates a CAD model using a finite set of parameters and then performs a finite element analysis by using a commercial program has been modified. The model parameter generation, model creation, analysis data collection and data evaluation phases are done by the Python and Delphi based computer codes. A global heuristic search strategy such as genetic algorithm is chosen to search different boom models and select an optimum.

Keywords: Finite elements, genetic algorithm, heuristic methods, shape optimization, excavator

ÖZ

EKSKAVATÖR BOMUNUN GENETİK ALGORİTMA KULLANILARAK ŞEKİL OPTİMİZASYONUNUN YAPILMASI

Uzer Cevdet Can
Yüksek lisans, Makina Mühendisliği Bölümü
Tez Yöneticisi: Prof. Dr. Eres Söylemez

June 2008, 109 sayfa

Bu çalışma ekskavatör bomunun otomatik olarak optimize edilmesiyle ilgilidir. Katı model oluşturulması, sonlu elemanlar analizinin yapılması ve çok sayıdaki modelin değerlendirilmesi, zaman alıcı işlemler olduğundan ve deneyimli işgücüne gereksinim duyduğundan dolayı böyle bir çalışmaya ihtiyaç duyulmuştur. Önceden geliştirilmiş olan, otomatik olarak ekskavatör bomunun sonlu elemanlar analizini yapmaya yarayan OptiBOOM [35] bilgisayar yazılımı otomatikleştirilmiştir. Model parametresinin yaratılması, modelin oluşturulması, veri toplanması ve değerlendirmesi tamamiyle Python ve Delphi tabanlı kodlar tarafından yapılmaktadır. Genetik algoritma gibi global sezgisel arama stratejilerinden bir tanesi, birçok farklı model arasında arama yapılabilmesi için kullanılmaktadır.

Anahtar kelimeler: Sonlu elemanlar, genetik algoritma, sezgisel metodlar, şekil optimizasyonu, ekskavatör

To My Family and Arzum

ACKNOWLEDGMENTS

I wish to express my deepest gratitude to Prof. Dr. Eres SÖYLEMEZ for his guidance, advice, criticism, encouragements and insight throughout the research.

I would like to thank my friends Ferhan FIÇICI, Taner KARAGÖZ, Ender YILDIRIM, Erkal ÖZBAYRAMOĞLU and Levent İPEK for their suggestions and comments.

I would also like to express my appreciation to general manager of Hidromek Ltd. Company, Mr. Hasan Basri Bozkurt for his support.

I have furthermore to thank Prof. Dr. Eres SÖYLEMEZ for accepting me to Hidromek R&D team and thank all my friends working in Hidromek R&D Department.

This study was supported by Hidromek Ltd. Comp.

TABLE OF CONTENTS

LIST OF FIGURES

FIGURES

ABBREVIATIONS

ABBREVIATIONS

FEA	: Finite Element Analysis
FEM	: Finite Element Model
GA	: Genetic Algorithm
LP	: Linear Programming
NLP	: Nonlinear Programming
TS	: Tabu Search
ESO	: Evolutionary Structural Optimization
CAD	: Computer Aided Design
CAO	: Computer Aided Optimization
SAE	: Society of Automotive Engineers
IIW	: International Institute of Welding
FAT	: Fatigue Class
GUI	: Graphical User Interface

CHAPTER 1

INTRODUCTION

A hydraulic shovel of a bucket type excavator is an earth moving machine comprising an upper rotatable chassis mounted on a drivable body with wheel or track and hydraulically powered mechanism consisting of boom, arm and bucket, mounted to the upper chassis.

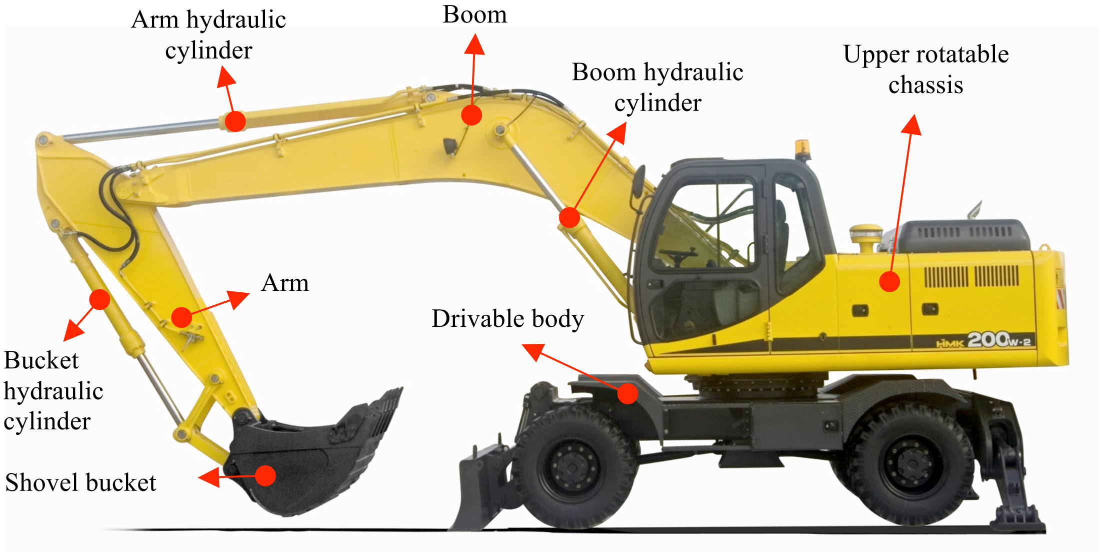


Figure 1.1 - General view of an excavator

The mechanism is actuated by the help of hydraulic cylinders. The machines are widely used for the digging, lifting and cleanup purpose. Trench digging in the application of placing pipes, digging applications in construction areas, rearranging face of the earth are some examples for the use of excavators.

Excavators can also be used for tasks other than digging. In such cases different end attachments can also be used. One common application is breaking rocks, for which a breaker is used instead of a bucket (Figure 1.2).

Figure 1.2 - General view of an excavator with a breaker.

An excavator boom consists of an upper chassis mounting bracket, an arm mounting bracket at the tip end of the body, an arm cylinder connection bracket welded on the upper plate and a boom cylinder boss placed in the middle of the vertical side plates. The boom body is constructed with upper, lower and vertical side plates welded to each other at right angles to form a rectangular cross section. Additionally, reinforcement plates may be connected to form a closed box section in pursuant of the design criteria (Figure 1.2).

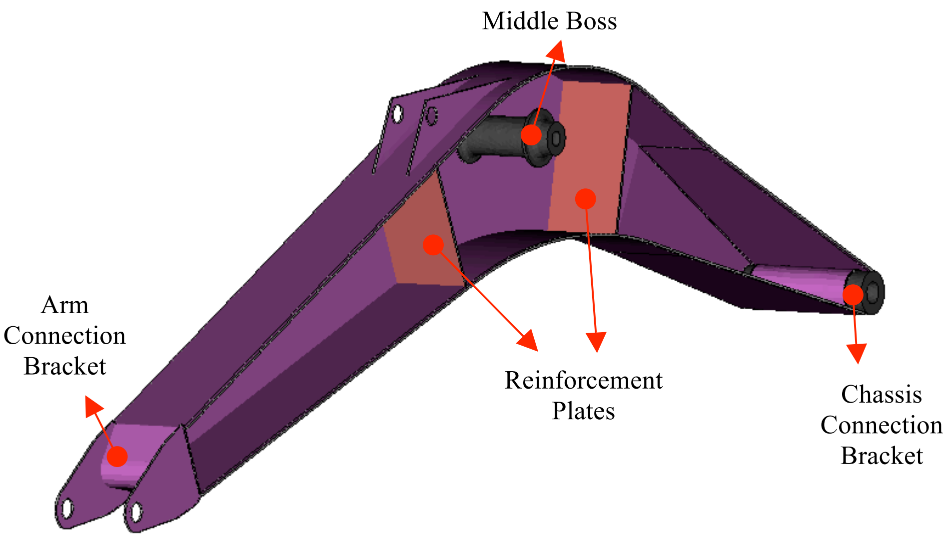
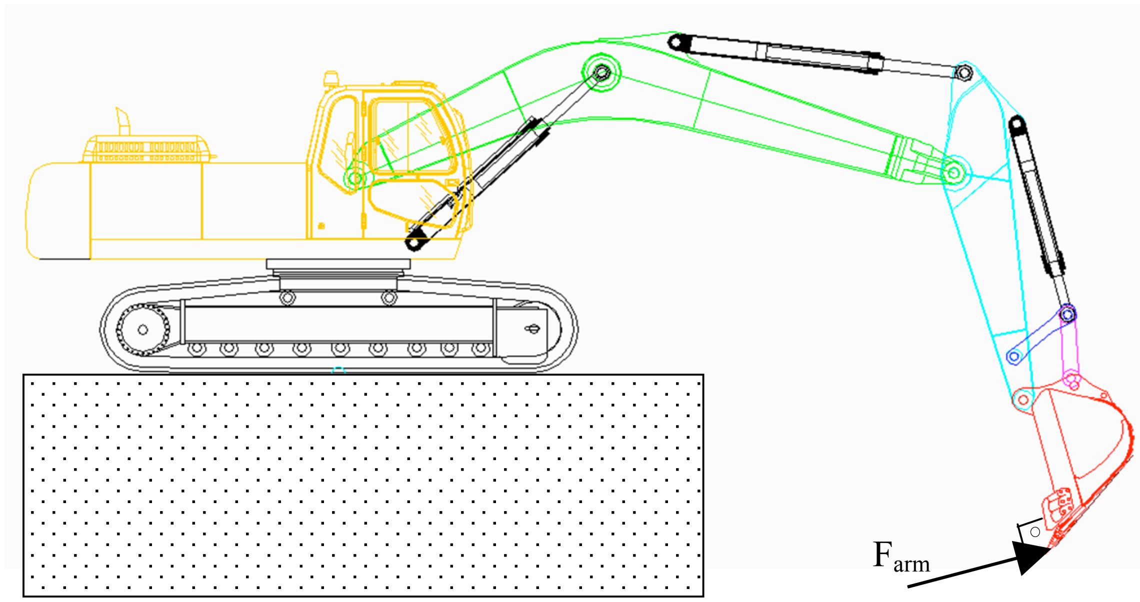
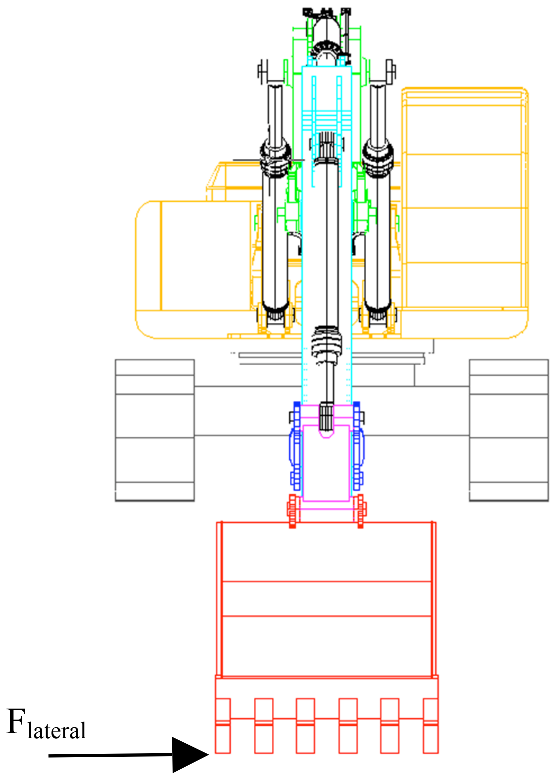


Figure 1.3 - General view of an excavator boom with reinforcement material.

The boom, arm and chassis are subject to different high loading conditions throughout their life span. The fatigue loading of these parts determine the life expectancy of an excavator. In boom design most severe digging position is chosen. The boom, arm and bucket are exposed to F_{arm} arm breakout force (calculated in accordance with SAE J1179 [28]) at the bucket tip during excavation of the earth (Figure 1.4.a) and also lateral force ($F_{lateral}$) perpendicular to the length-wise direction of the arm and bucket at the bucket tip during sweeping the earth (Figure 1.4.b). These mentioned loads result in bending and torsional moments in the boom structure. In figure 1.5, deformed shape of an excavator boom which is subjected to bending and torsional moments can be seen.



(a)



(b)

Figure 1.4 – External forces on the bucket. (a) Arm breakout force. (b) Lateral force.

Figure 1.5 – Deformation shape of the excavator boom which is subjected to arm breakout force and lateral force.

The boom should be designed such that Von Mises stress will not exceed allowable design stress value to ensure aimed fatigue life and at the same time, mass of the boom will be minimized as much as possible to minimize operating costs of excavator such as fuel consumption and digging cycle time. Hence, the aim is to minimize the weight of an excavator boom while limiting stress values in a predetermined range.

Employing finite element method (FEM) became popular in recent years to enhance the precision of the stress evaluation. Finite element method is a powerful tool to solve complex structural systems. Definitely, it is possible to obtain very realistic solutions at the end of the analysis of the structure [34], but the accuracy of the final result is markedly dependent on well defined boundary and load conditions.

In finite element analysis (FEA), creation of the model, describing the boundary and load conditions of the structure is a lengthy, time consuming process. An automated procedure is required if this needs to be performed repetitively as in the case of optimization. An in-house optimization software OptiBOOM is employed in this study [35]. OptiBOOM was developed by Mehmet Yener in Hidromek R&D department. The program is capable of modelling and analyzing the structure automatically by using pre-defined parameters. The program recognizes the boom shape in terms of angles, lengths, radii and thickness. In fact, program creates an interface between the user and commercial softwares Msc. Marc® and Msc. Marc Mentat®. The program prepares Mentat procedure files which contain all geometric, material and boundary information. The Borland Delphi® based OptiBOOM code has been modified and new features has been added in this study.

Implementing an analytical method in such an optimization problem is not feasible because of too many model and constraint parameters. Therefore, heuristic methods such as Genetic Algorithm (GA) make it possible to develop a preliminary design. Genetic Algorithm is capable of searching a large space by handling arbitrary kinds of constraints and objectives which are embedded into a fitness function. Fitness function evaluates each design in terms of the stress and mass outputs with some weighting factors.

In this study, besides Borland Delphi®, another programming language Python™ is utilized in the modelling, post-processing and developing Genetic Algorithm phases. Msc. Marc Mentat® provides a programmable interface through Python programming language. Python™ is a dynamic open source so it includes a large library which has been developed by different users, throughout the world.

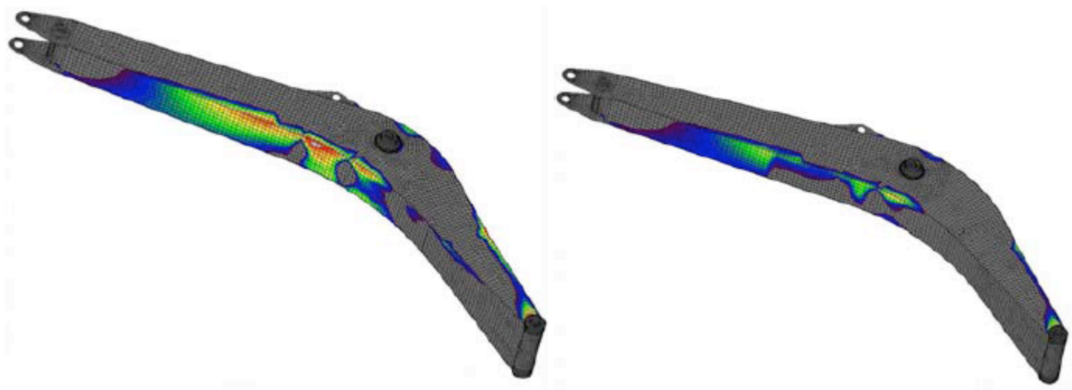
CHAPTER 2

LITERATURE REVIEW

Searching the best outcome of any real process creates the idea of optimization concept. The correct formulation to solve a given problem forms the 50% part of the total effort. Well defined optimization procedures should be evolved to reach the best solution of the problem [16]. In the last decade, the importance of the optimization topic in the industry increased in touch with the cruel commercial [25]. The efficiency and reliability of manufactured goods mostly depend on geometry. Hence, structural optimization subject became popular among the applied mathematicians and engineers [14]. In the first part of this chapter, studies about the design of structural parts which are existent on excavators are discussed and the second part covers an optimization review.

2.1 Design Studies on Excavator Parts

There are many studies in literature and also in Hidromek R&D department about the design of excavator booms and other welded structures. Yener [35] parametrized boom geometry and developed Delphi® based computer program (OptiBOOM) that is capable of automatically creating FEM of excavator boom by using parametric geometry information and running Msc. Marc® analysis program to solve created model. OptiBOOM shortens FEM creation and analysis time, and assists the designer in improving the structure's weight and strength. Yener created more than 100 alternative boom designs and compared with each other in terms of mass and stress. The boom design which has been chosen as final design has weighed 3.6% more than initial design but maximum Von Mises stress of the final design has decreased 21.5% according to initial design. Figure 2.1 shows Von Mises stress maps for initial design and improved design of HMK 220 LC excavator boom.



(a) (b)

Figure 2.1 – Von Mises stress map for initial design (a) and improved design (b). [35]

Usually butt and fillet welds are used in construction of excavator boom. Fillet welding constitutes approximately 80% of the total welding operation in an excavator boom construction. Mostly welding type and quality determines the life span of excavator boom due to fatigue.

Fıçıcı [10] modelled three dimensional surface cracks (Figure 2.2) in fillet welds to be able to predict fatigue life of such type of welded connections. Fıçıcı calculated stress intensity factors around the crack front for the test specimens which are subjected to bending and axial loads.

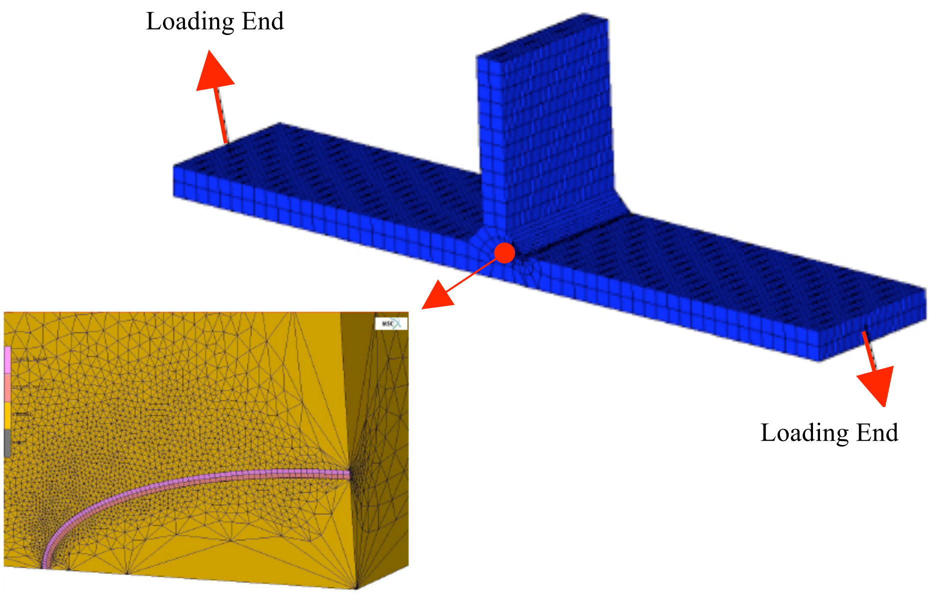


Figure 2.2 – Modelled surface crack on fillet weld. [10]

Karagöz [19] also modelled three dimensional surface cracks in fillet welds. J integral values have been calculated for such cracks in T welded joints of construction machinery. Karagöz used sub modelling technique in finite element method.

Figure 2.3 shows that a fillet weld region is selected from the excavator boom and the region is modelled in detail again.

It should be noted that all welds implies initial flaws in the form of root gaps and may include other initial flaws potentially depending on production quality. Stress intensity factor calculation done by Fıçıcı and J integral calculation done by Karagoz are required to make successful predictions for fatigue life of welded structures.

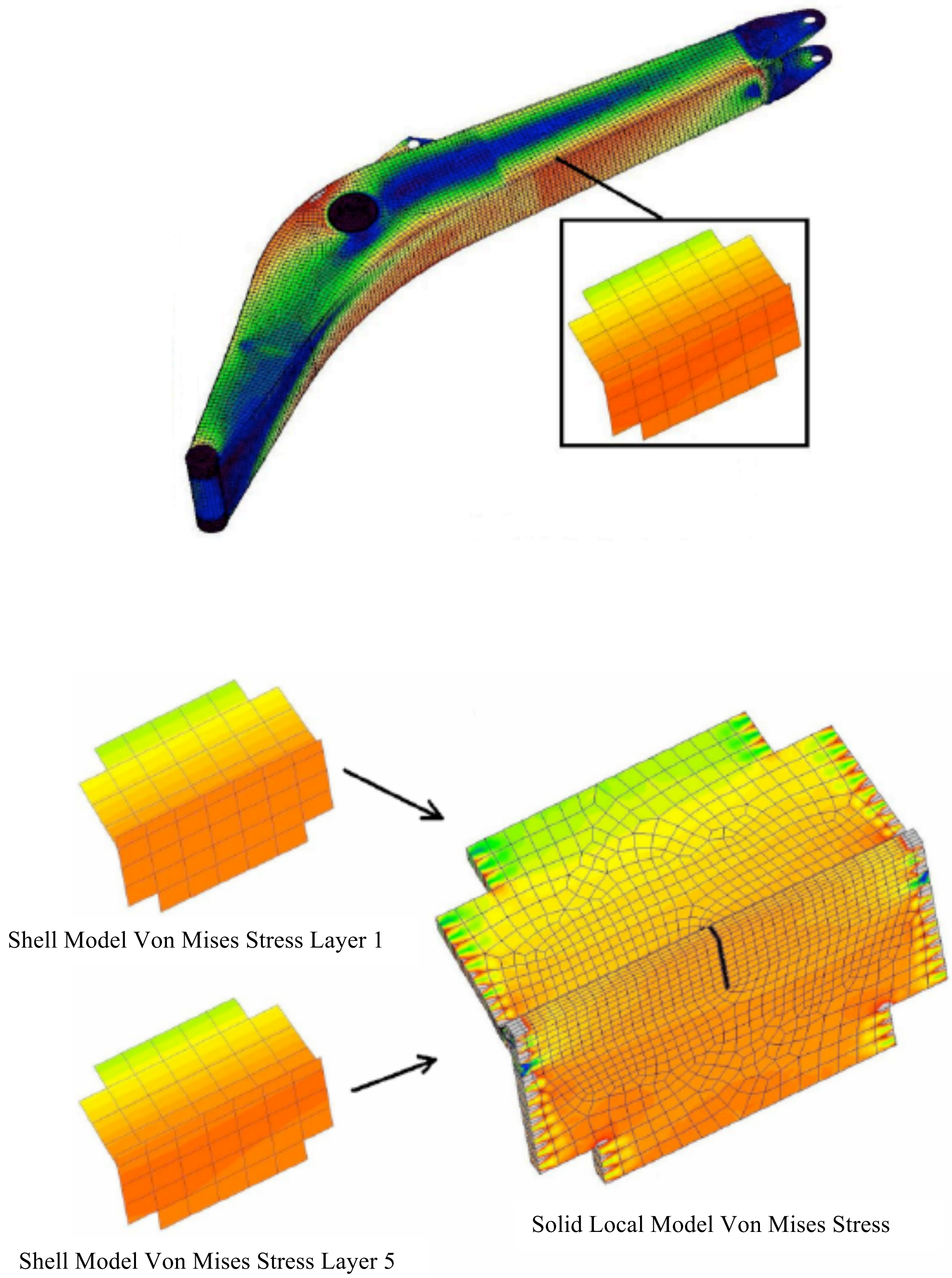


Figure 2.3 – Crack modelling on fillet weld by using sub modelling technique. [19]

Another design study on construction machineries has been done by Volvo Excavators AB and Alfgam Optimering. Carlgren et al. has aimed to improve fatigue durability of middle boss of excavator boom [2]. Geometrical modifications are applied to the structure in order to decrease stress intensity factor in weld root gap. They constructed a finite element model as the parent boom model and also a verification model including arm and bucket models with lower order shell, bar and beam elements (Figure 2.4).

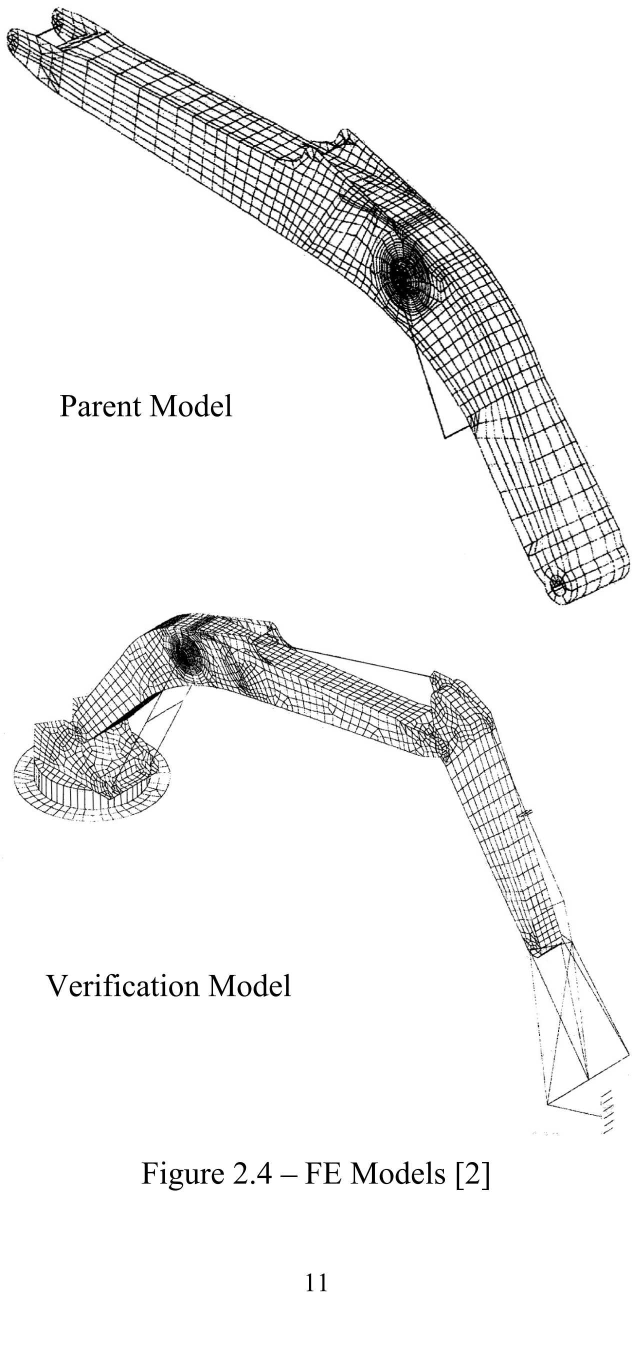


Figure 2.4 – FE Models [2]

The load cases applied in finite element model have been determined by observing stress data collected while the excavator is working during real applications in the field. To idealize the operating conditions, a set of theoretical load cases has been defined, however, only most frequent and most critical load cases LC01 and LC04 has been considered to decrease analysis time (Figure 2.5).

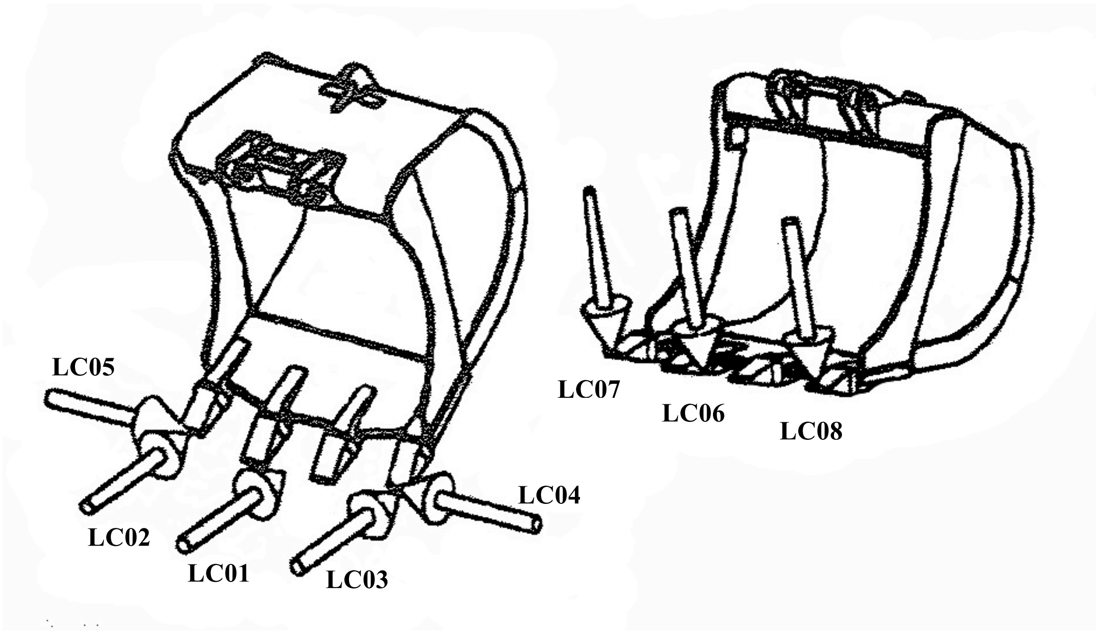


Figure 2.5 – External forces on the bucket [2].

Due to large deformations at bucket tip, position and direction of force applied at bucket tip changes. Hence, slewing torque change versus bucket force has been determined by measurement and calculation. Also strain gauge measurements are performed on the boom intermediate part (Figure 2.6). Calculated and measured strain values are compared and essential modifications are applied to parent model to use it for further analysis (Figure 2.7).

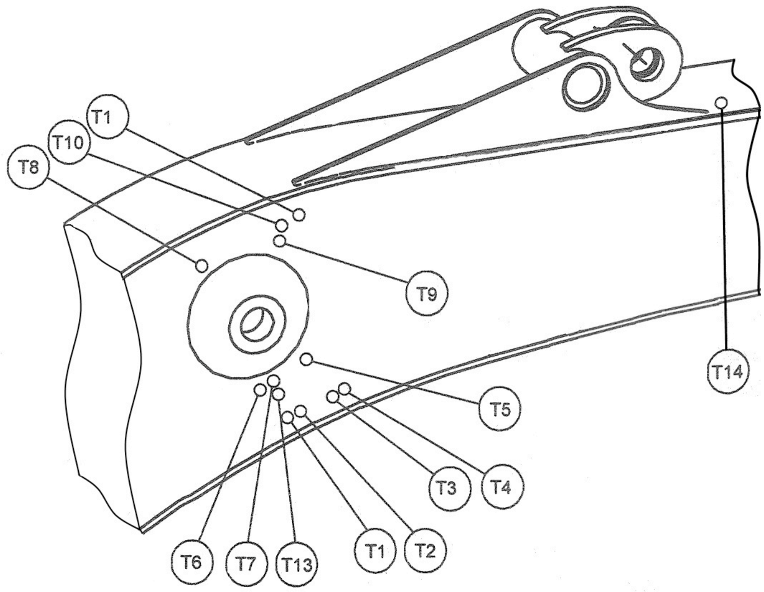


Figure 2.6 – Strain gage position on the boom [2].

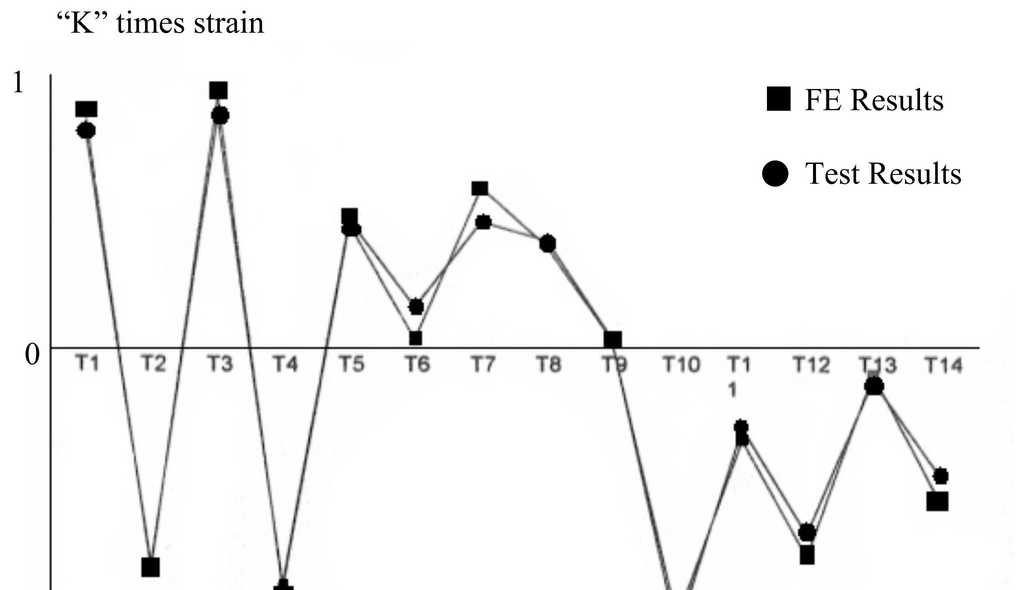


Figure 2.7 – Comparison of calculated and measured values of strain (K = scaling constant) [2].

Since the stress intensity factor is proportional to a linear combination of nodal displacements, optimization problem has been formulated as:

$$\sigma^e \leq \sigma_{crit}\ ,\ \sigma_{crit} = \text{allowed stress}$$

$$x_j \in X_j\ ,\ \mathrm{j} = 1,\mathrm{N}$$

$$X_j = \{\underline{x}_j \leq x_j \leq \overline{x}_j\}$$

In here, σ^e is Von Mises stress at related coordinate, σ_{crit} is the allowable maximum Von Mises stress and x represents the coordinates of the j[th] design variable. It is possible to control the cross section of the middle boss, including the diameter and wall thickness (Figure 2.8).

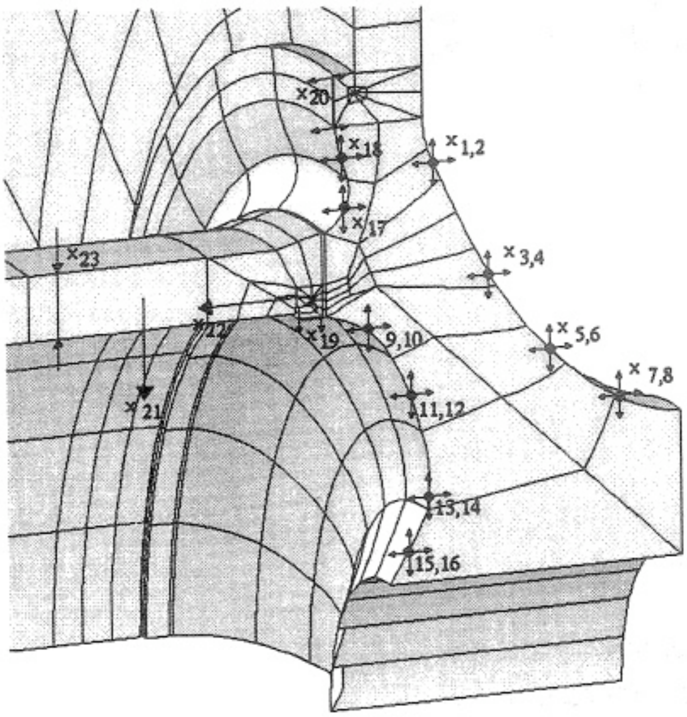


Figure 2.8 – Shape variables controlling the cross section of the middle boss [2].

Structural optimization code OASIS-ALADDIN has been employed in this study. The geometry of the middle boss has been changed as shown in figure 2.9. The weight of the part has been reduced by 7%. Stress intensity factor in load case LC01 has been reduced by 42% and 49% respectively for the inner and outer welds and for load case LC04 the corresponding result is 67% and 72%. Consequently, fatigue durability of the structure has been increased significantly.

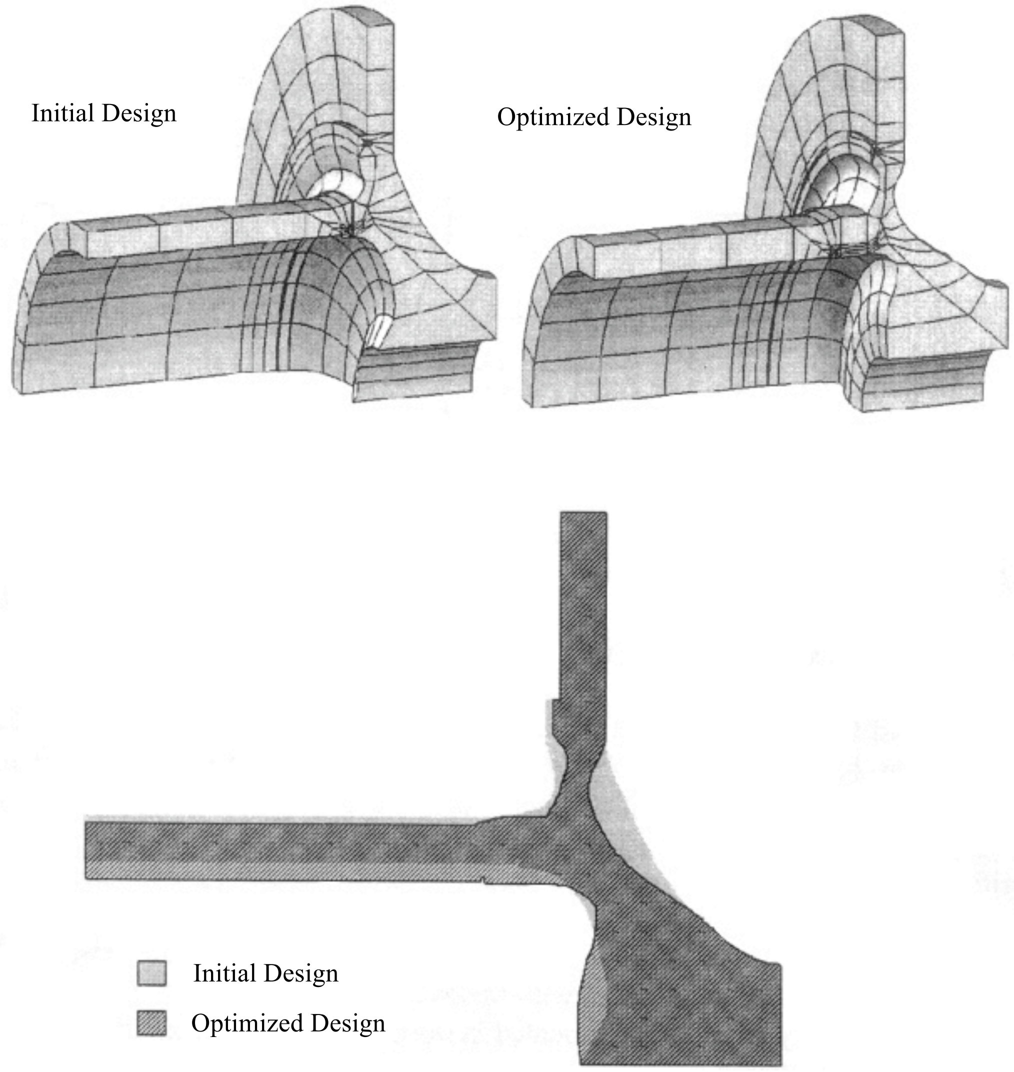


Figure 2.9 – Comparison between initial design and optimized result [2].

2.2 Optimization Review

The optimization issue might be classified as two main titles that are called linear (LP) and non-linear (NLP) programming. The construction of the optimization problem including linear functions is called Linear Programming. Limited number of structural design problem is available to apply the LP because objective functions of the most of the structural optimizations are extremely non-linear [12].Therefore; this study focuses on NLP.

In a complex structural design, constructed by using a high number of design parameters, the derivative information to calculate the output data and clear relations between design parameters may not be available. In such a case the structural design space will include more than one minima or maxima. Further, design optimization tends to even worse cases if the design variables have discrete values [12]. Because of these limitations, one should prefer Heuristic Search Methods. Some of common methods are Tabu Search, Simulated Annealing, Neural Network and Genetic Algorithms.

2.2.1 General Formulation

An optimization problem can be formulated as [16];

Minimize $f(x) = f(x_1, x_2, ..., x_n)$

Subjected to $g_i(x) = g_i(x_1, x_2, ..., x_n) \leq 0$, i=1 to n

$h_j(x) \equiv h_j(x_1, x_2, ..., x_n) = 0$, j=1 to p where x denotes a design vector with components x_i, i=1 to n and, $g_i(x)$ and $h_j(x)$ are considered to be inequality and equality constraints. $g_i(x)$ may be converted as,

$$g_i(x) = g_i(x_1, x_2, ..., x_n) \geq 0 \qquad \text{i=1 to n}$$

for the maximization problems.

2.2.2 Tabu Search

Tabu Search (TS) starts as the known local search methods do [11]. The search process goes on iteratively until a convergence criterion is satisfied. After this point, TS introduces a strategy to modify the neighbourhood to a new one. To realize this, TS uses special memory structures to discover the design space. The way of solutions encountered over a specified horizon are forbidden to related neighbourhood and classified as tabu. The tabu list forces the search to find new points in the design space.

2.2.3 Simulated Annealing

The main idea of simulated annealing is based on the analogy of annealing of solids. Annealing is a heat treatment process to change the material properties such as strength and hardness. In cooling (or solidification) process of pre-heated solid, the atomic structure reaches an energy level according to cooling rate. It is observed that during the natural annealing process system may have a high energy even at low temperature. Therefore, simulated annealing is able to achieve global minima and escape from local minimum points. The Boltzmann's equation is employed to decide the energy level of the system.

$$P(\Delta E) = e^{\frac{-\Delta E}{k_b T}}$$

P(E) changes in the interval [0,1]. The algorithm perturbs the atom position at a given temperature and computes the change in the energy. If the P(E) of perturbed atom is less than the previous P(E) value, the perturbed state is accepted. The analogy between the annealing and the optimization is constructed by replacing E with an evaluation function in terms of position defining parameters in the space. The temperature, T becomes a control parameter [12, 18].

2.2.4 Neural Network

A neural network is a signal processing system. It is assumed that there are signal carrying neurons connected to each other and after a training process weights between neurons are determined to compute output. A neuron processes several inputs that may include its own output and produce only one output. This may be called as a short term memory element. All neurons are connected each other via links with weights which represent the importance of the connection in the evaluation. This may be thought as a long term memory. By this way system responses can be estimated for any input. This procedure decreases the amount of computational effort such as FEA by deciding tendency of the optimization previously.

2.2.5 Genetic Algorithm

2.2.5.1 Background

Genetic algorithm is the most popular method of evolutionary optimization methods. Genetic algorithm mimics the main mechanisms of the theory of evaluation that are mutation, natural selection, adaptation and gene flow. The method regenerates new populations evolving over generations. While doing this, it allows the individuals to survive or not according to their calculated payoff information. Genetic algorithm is able to search a large multi-dimensional solution space so the method is capable of creating a solution set including the global maxima (or minima) and also the local maxima (or minima). Another advantage of the GA is that method does not require derivative information so it may be applied into many applications easily.

The GA terminology is mostly based on the theory of evolution. Here the basic terms are explained. The mentioned payoff information is called fitness number in the terminology of genetic algorithm and includes the adaptivity information of the individual. A string in GA carries the information that characterizes the individual in form of binary or real numbers. The same logic thought for chromosomes is also applicable for the strings. For instance, parameters defining the shape of a structure might be listed in a string by converting each parameter into binary format so this would be the genetic definition of that model.

Population is the collection of produced individuals. The individuals may lie on anywhere of the design space. Also, the first population created at the very beginning of the optimization phase is called as initial population.

Each binary or real number is called as gene and the list of the genes form string as mentioned above.

2.2.5.2 Determination of Design Space

Most of the designers roughly decide design boundaries based on their engineering intuition. However, the optimization boundaries may fail in implementation of GA in problems. In such a case, bit mutation and crossover performance decreases dramatically and generally a global optimum solution could not be found [12]. Directly applicable, simple methods deciding the design boundaries do not exist but there are studies related to this problem. Two of them are mentioned in the following paragraph as being example.

To overcome this problem, Chen and Lin proposed two methods [3]. In one of the studies, Taguchi orthogonal array is employed to do experiments for the design spaces in which best topologies determined. The obtained information is used to find the optimum design boundary. The second method covers the predicting the boundaries by using co-work of GA and neural networks. Mainly, neural network is a signal processing system and it is used to predict the system responses. GA searches a set of design space and finds optimum solutions. Neural network uses the design data as a signal and analyzes the data. Therefore responses of the other individuals might be predicted. Most of the complex structural design problems require big amount of computational effort such as finite element analysis so neural network codes are run rather than running finite element analysis.

Rasheed et al. followed a different way and they tried to make the mechanisms of natural selection more controllable rather than use of random crossover, mutation [26]. They presented four methods to achieve an effective GA. First two methods propose different crossover techniques, third one is a type of mutation and the fourth one is the screening mode. In the first method, Line Crossover, two parents are selected and a line between two parents is constructed. Then a point is chosen along this line or it's extensions from each side (Figure 2.1).

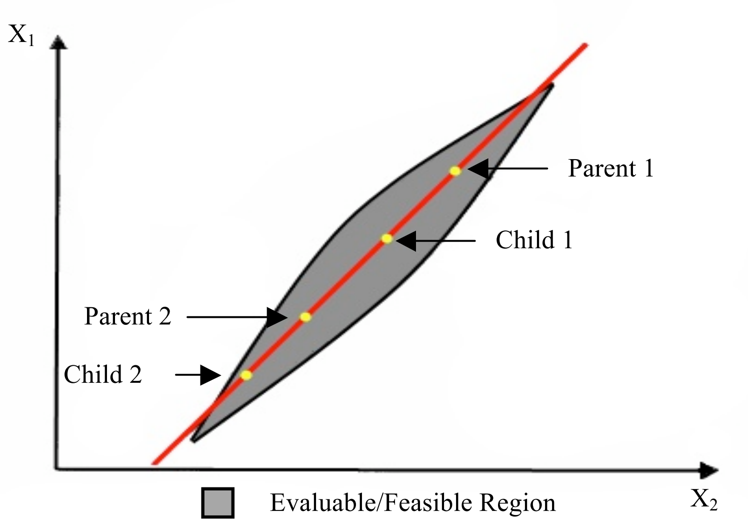


Figure 2.1 – Line crossover behaviour [26]

The second method gives a step by step procedure to determine the second candidate for the crossover after the selection of the first one by using standard method. *Q* function is defined in terms of fitness values of two candidates and distance between them. The aim is to maximize the *Q* function while changing the information coming from the second candidate. Therefore, search process will continue to drive in valuable solution space. In the shrinkage method, the idea is creating a perturbation function, in terms of value of the parameter and stage number of the optimization process, that decreases as the optimization process goes on. Therefore, at the beginning of the optimization, large mutations will occur and as the number of stages increase, the relatively small mutations will occur. The last method, screening method, defines a threshold value and chooses one candidate. Then it searches for the second candidate at the neighbourhoods. If the fitness value of the neighbour is better than the threshold value, crossover is processed. The above mentioned methods were employed in the design of a supersonic transport aircraft design.

2.2.5.3 Genetic Operators

A basic GA consists of three common genetic operators. These are reproduction, crossover and mutation. The flowchart of the mechanism is depicted in figure 2.2. In the following paragraphs the detailed descriptions of these genetic operators will be made [7, 18]

Evaluation: At evaluation stage, the fitness function defining the worth of the string in the population is calculated. By this way, individuals in the population are sort in order of strongest to weakest. Only in this stage the information directly comes from the problem itself.

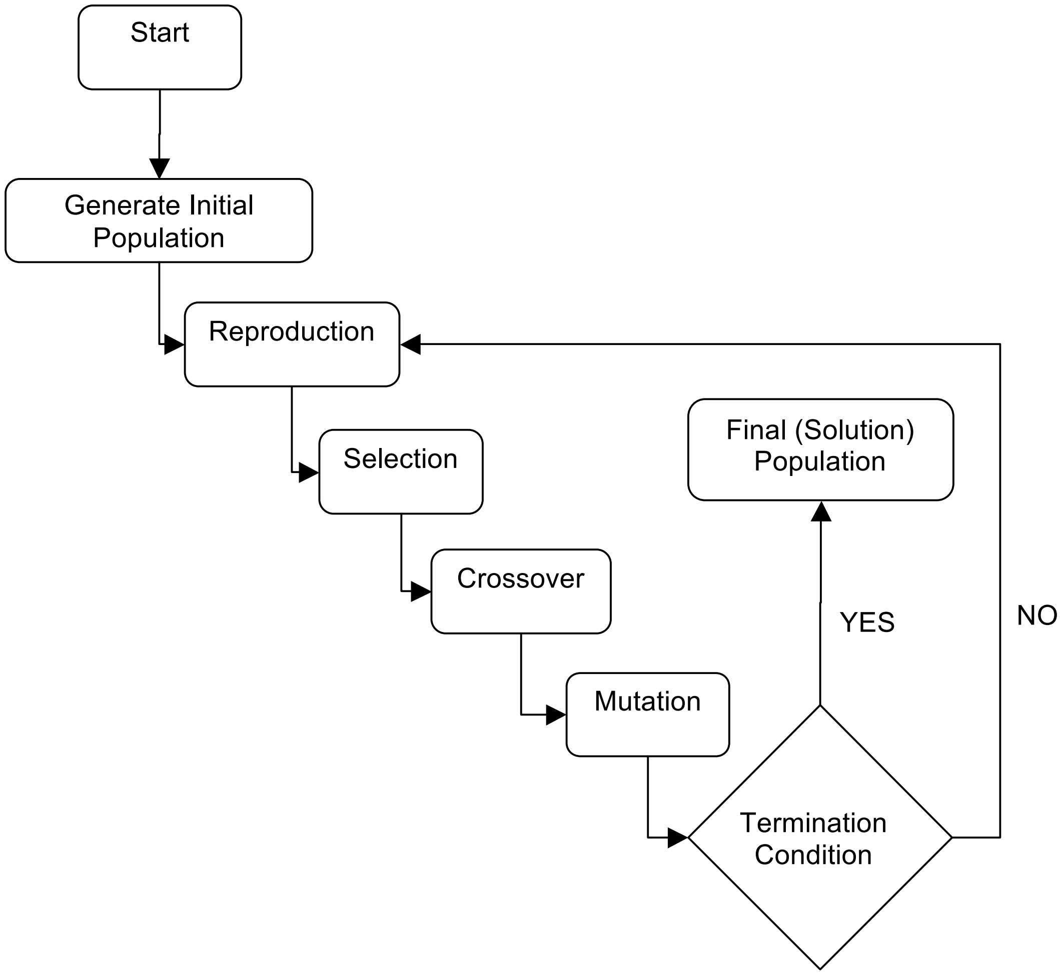


Figure 2.2 – Flowchart of Genetic Algorithm

Reproduction: Actually, reproduction is the stage where the Darwinian survival of the fittest one theory is mimicked. There are a number of ways to implement. The most popular and easiest way is using a biased roulette wheel. Each string is represented as a slot sized in proportion to its fitness [7]. The roulette wheel is rotated artificially and the ball on the roulette wheel falls into one of the pockets on the wheel. This process is continued until a set of strings is formed. These arbitrarily selected strings are the candidates of the crossover stage. As it is expected, the strings with higher fitness values have more chance to be selected at each rotation of the roulette wheel.

Crossover: The created set of strings is a population of parents. Two steps are proceeded in crossover stage. First, the selected parents are mated randomly and then the crossover stage is run with the determined couples of parents. As it was mentioned previously, the strings may be in binary or real format. In case of using binary format, the genes are exchanged at randomly determined section of the string. Otherwise, a weighted mean function in terms of mated genes and some constants is calculated to find the real value of the new gene.

Mutation: In some cases, the genetic algorithm diverges to local optimums. To overcome this problem, randomly selected genes are altered randomly. In this process the useful genetic information can be lost. Hence, a good intuition is required to control the rate of mutation.

2.3 Shape Optimization

Shape optimization is a part of optimal control theory or it can be thought as a part of structural optimization. As a matter of fact, optimal control theory serves to structural optimization. The geometrical facets are reasonably effective in the determination of the efficiency and reliability of the structure [14]. Shape optimization finds the optimal shape, minimizes the cost function while satisfying given constraints. Shape optimization may be thought as being three parts [14]. These are;

Sizing Optimization: One of the sizes of the structure, which is strongly effective on the geometry, is optimized.

Shape Optimization: The shape of the structure is optimized but topology is kept.

Topology Optimization: Shape of the structure is optimized without keeping topology.

2.3.1 Evolutionary Structural Optimization (ESO)

The main idea of the evolutionary structural optimization method is removing inefficient material from the structure by using the predefined criteria. The finite element analysis method is generally used with ESO. The element, which is assessed as an inefficient element is removed from the FEM analysis. The objective function is defined as a combination of the criteria assigned for each element generally. A factor is defined for each element to measure the contribution of that element towards the overall performance of the structure. The assigned fitness number carries a great importance. After the analysis of the structure is done, ESO determines the contribution factors for each element and deletes the elements whose contribution factors are less than a reference value. The reshaped structure is reanalyzed and the contribution factors for each element are determined again. The process is finished when the resulting structure achieved a predefined convergence criterion. A set of solutions is obtained after the optimization process. In fact the ESO is a stress based topology optimization method. However, simplicity and applicability of the method make it adaptable for many other fields such as buckling, heat conduction and thermo-elastic problems. There are two extended version of ESO exists recently. *Morphing ESO* is an algorithm for sizing optimization problems and *Nibbling ESO* is an algorithm in which the material removal process is only allowed in a predefined boundary [6].

2.4 Sensitivity Analysis

The method defining how the constructed model is sensitive to altering design parameters is known as post-optimality analysis or sensitivity analysis. Sensitivity analysis clears the uncertainties associated with parameters and increases the accuracy of the design's optimality [1]. Most of time, designer is not able to predict how much design parameter accuracy is enough for the related model. Sensitivity analysis makes it possible to know and controls the effect of each parameter on the design. Also, sensitivity analysis helps the designer to understand the behaviour of the system of the model in the course of changing design parameters.

2.5 Previous Works on Shape Optimization

Taguchi method is one of the most popular statistical search approach, not discussed previously. Lee and Kwak employed Taguchi Method as a zero-order optimization method [22]. The idea behind the Taguchi method is providing the best quality level by employing the controllable factors that constitute the quality while minimizing the effect of noise. In order to use the Taguchi Method as an optimizer, three prescribed initial level values for each design are appointed as starting values and the level values are updated after an optimum level combination is reached. Lee and Kwak implemented the method on a computer aided design (CAD) and finite elements based code. The developed optimization code is applied to the designs of micro-gyroscope and boom structure of an excavator.

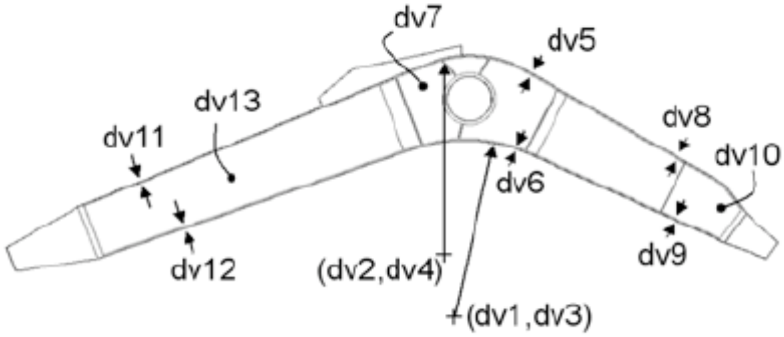


Figure 2.3 - Design variables of excavator boom [18]

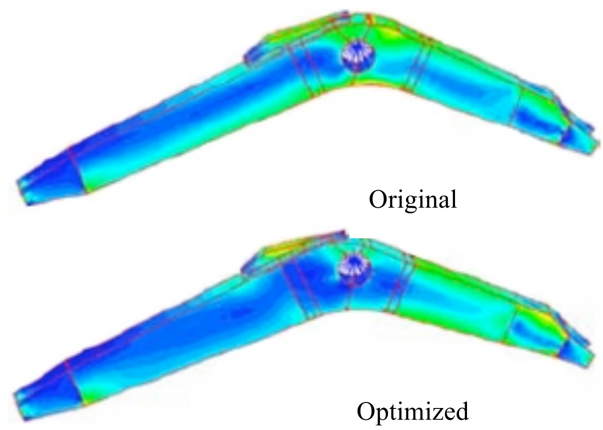


Figure 2.4 - Stress distribution of excavator boom [18]

In the design of an excavator boom structure, the method of modelling the structure and application of the boundary conditions are almost same as it is done in this thesis. Lee and Kwak succeeded the shape optimization of an excavator boom by using the proposed method and likewise, this thesis tries to find the optimum shape of an excavator boom by employing genetic algorithm.

Biological growth method is another shape optimization method, firstly introduced by Mattheck [24]. The idea of inspiring by the nature in shape optimization of engineering parts is a very common issue. Mattheck manifests that although CAO

(Computer Aided Optimization) is a powerful tool used in the industry, some useful simplified methods that decrease the dependence to FEM codes, are available. Mattheck mimics the growth phases of trees that grow in such a manner that stress peaks are reduced and surface stresses are distributed uniformly. Mattheck employed pocket calculator and graphic methods in this work.

Mattheck claims that bending stress which tries to deform the contour of the notch is the notch stress. Therefore overall stresses consist of bending and nominal stress components. One could try to optimize the contour of the notch shape by increasing the nominal stress while decreasing the bending stress and vice versa. In this study sum of the nominal and bending stresses are taken constant. The mentioned method is called “Pocket Calculator Method”.

Mattheck mentions the “method of small triangles” as the graphical method. The idea is increasing the angle of the notch by applying triangles and repeating the same procedure to the high notch stress region. The accuracy of the method is proved by the applications of the method to the torn of a rose and claw of a bear.

Tekkaya and Güneri developed a parametric study by employing biological growth method [30]. The aim in their study was to minimizing the Von Mises stress at any point along the optimization domain by changing the shape of the optimization domain. Seven parameters are set during the execution of the method. These parameters are optimization boundary, optimization region, expansion coefficient, reference stress, conversion factor with units degrees temperature per stress, Young’s modulus and magnification factor. In this study a biaxially loaded plate having a hole at its centre is considered as the problem. Commercial finite analysis program MARC is employed in the study. The figure 2.5 depicts how the shape of the design domain changes and figure 2.6 shows the change in the distribution of the Von Mises stress along the boundary over the iterations.

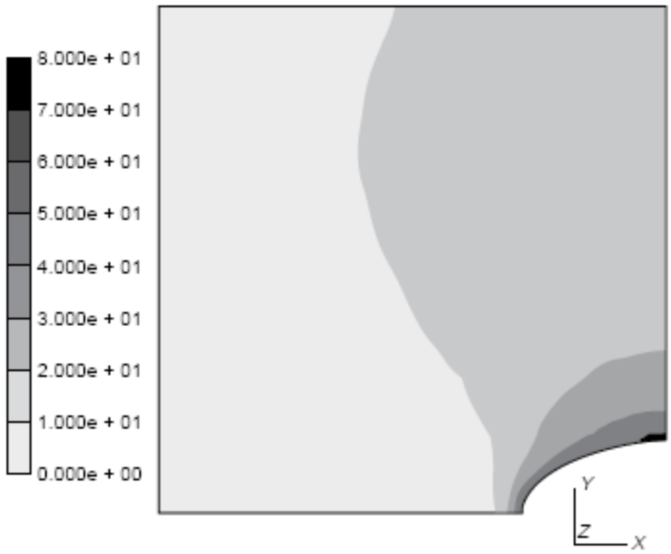


Figure 2.5 – Hole geometry and Von Mises stress distribution (in MPa) after 6 iterations for 40 MPa reference stress and 500 magnification factor [30]

Also it is possible to minimize the mass of the plate by assigning reference stress value higher than the global minimum value. Tekkaya and Güneri presented that biological growth method is a powerful tool at any stage of the design process. It can be run in the existing finite or boundary element programs easily. As mentioned before, the main purpose of the method is to reduce stress peaks along the optimization boundary. Therefore the method serves to the benefit of fatigue and fracture firstly.

One may require reaching an optimum design using the parameterized shape of the related model. A generic description of the model is indispensable and will be controlled by a set of design variables. Each new model presents particular properties depending on selected design variables and is convenient to generic description.

Too many variables causes for the creation of complicated design problems that increases the time required for solution. Therefore, it is good to use an effective small parameter set for behoof of automatic design process [31].

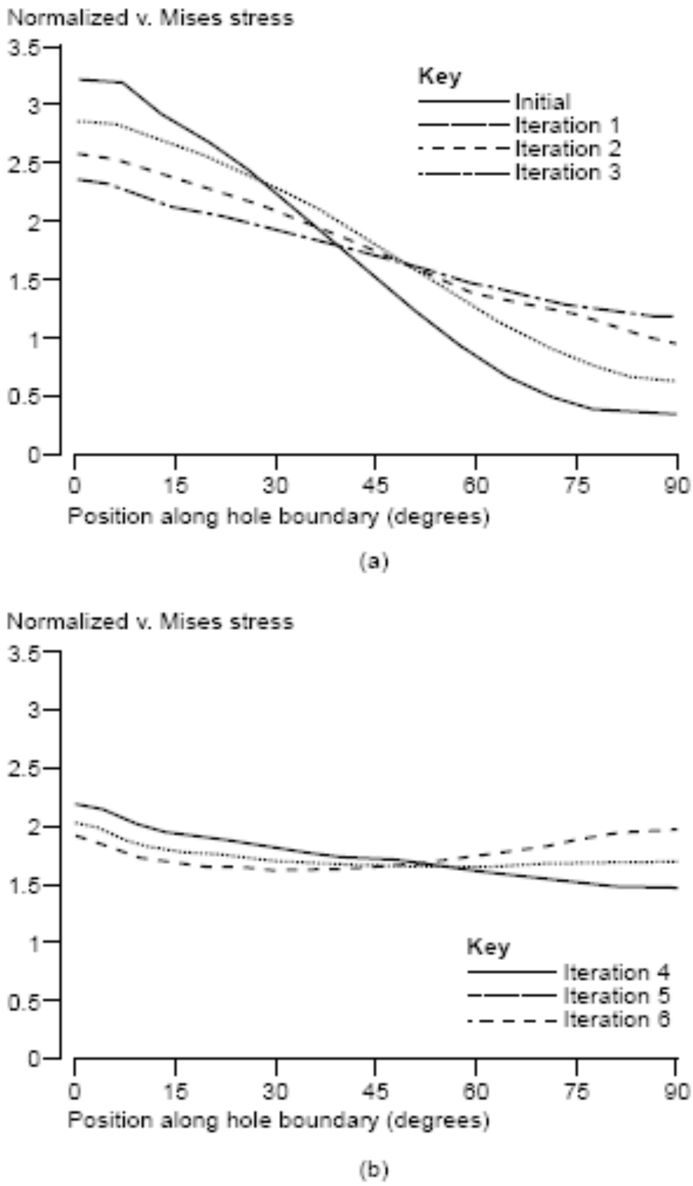


Figure 2.6 – Normalized v. Mises stress distributions along the hole boundary. (a): first three iterations, (b): last three iterations [30]

Wang parameterized a can design to find the optimum bead shape, bead placement and wall thickness against the axial load and panelling while satisfying minimum mass criteria [32]. The can wall is deformed due to high external pressure and this event is called as panelling. It is observed that beads on cans are effective structural shapes to increase the panelling performance. Beside this, cans are also deformed against axial loads in real life because of the system of stocking of filled cans. It is investigated that the bottom layer cans receive high axial loading. To overcome this problem, this parametric optimization study is done.

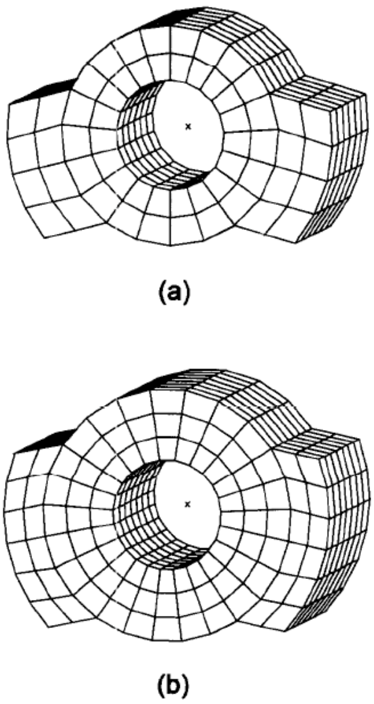
Loading conditions are determined by comparing real life loading conditions and FEM results. Although explicit solutions of FEM give more accurate results, implicit solvers are preferred in order to decrease the time required for the solution. Some correction factors are employed to fit the real world and FEM results. Can diameter, bead depth, can wall thickness and height of the can are assigned as the shape parameters.

Optimization design space is constructed in terms of material thickness and bead depth which are design variables and the medium is divided into quadrants by the axial load and paneling requirements. Previous works and engineering intuition make it possible to create such a design space and optimal solution region.

Another parametric approach is also introduced by Jae and Young in order to achieve the optimum shape design of an engine mounting rubber using parametric approach [20]. To decrease the disturbances such as random shocks from the road, excitations from the rotating imbalanced parts on the engine, not only the rubber properties and placements of the rubber parts are enough, but also the shape of the rubber part is effective. Jae and Young constructed a parametric model including an objective function which is in terms of the three directional desired stiffness and computed stiffness values and also the weighting factors. The aim is to minimize the objective function, namely it aims to drive the stiffness values in three directions to the desired stiffness values. The shape model includes six parameters in total; however four of them take place in the optimization process.

A sequential unconstrained optimization method, penalty function is adapted into design process. Basically, the method converts the constrained optimization problem to a set of unconstrained optimization problem and penalty terms are placed into the objective function. Therefore, in case of the violation of constraints, penalty parameters penalize the objective function. This means larger the violation, larger the penalty and vice versa. The penalty parameters are redefined and minimized at each iteration. The process is ended when there are not significant improvements through the optimum point.

Stiffness values are computed in two FEA phases such that first analysis is related to vibration and second is to static gap. Then optimization code is run and model satisfaction is checked. The iteration is continued until the convergence criterion is said to be ok. An optimum solution for the mounting rubber is shown as in figure 2.7.



(a)

(b)

Figure 2.7 - Finite element models of the initial (a) and optimized (b) configurations [20]

Most of the designers have not preferred a commercial CAD program since the topic of the structural optimization became popular. However, the recent developments of the feature based and parameterized CAD modelling made the direct usage of CAD possible in the optimization processes. Edwin Hardee et al. developed a methodology of employing CAD models in the optimization process directly [13]. The momentous part of this approach is to designate the design velocity fields according the data given on the CAD model. The design velocity field aims to characterize the change of the finite element nodes with respect to the change of the arbitrary design parameters [21]. Hardee et al. put a mixed a method that combines finite difference and boundary element methods in order to compute the design velocity fields. Pro/MESH and Msc/PATRAN is available to create the mesh of the structures in this study. To overcome the difficulties of using FEA based finite difference method to calculate the design velocities, p- and h- version FEA which are related directly to the existing Pro-E CAD parameters. The developed method is applied to optimize the shape of a turbine blade. Design sensitivity coefficients for Gauss stress points and tip displacements with respect to four parameters are computed. Sensitivity coefficients has been verified and very accurate results has been obtained in the study. Two optimization problems, minimization of the blade mass while keeping the stress values below a designated limit and minimization of the airfoil tip while keeping the mass under a designated limit are carried out. In the first application, at the optimum the cost function reduced from 15 567.40 mm3 to 14 428.5 mm^3 and all stresses are below the limit. The tip displacement minimization in the second problem has been concluded with reduction less than the 60% of original design and 14 760 mm^3 volume.

Li et al. used discrete variable method in the developed stress based ESO [23]. They decided the presence or absence of the element by a binary decision method. A stress sensitivity number based on finite element analysis is calculated to estimate the stress change due to element removal or addition. The optimal stress distributed design is achieved by gradually adding or removing elements in order of lowest to highest stress sensitivity number. The structure is discretized using a dense mesh. The removal or addition condition is assigned as binary numbers (0 or 1) and determined by comparing the stress sensitivity number with the highest by employing a rejection ratio. FEA database holds all the element information in it while including only elements that will form the instant structure in the analysis phase. Hence, the solutions do not provide the displacement information which is used in the calculation of the stress sensitivity analysis. To overcome this problem, a fictitious displacement is created by employing an extrapolation technique. The information required for extrapolation is obtained from the neighbourhood elements because element removal or addition always happens at the boundaries by attaching or detaching to other elements. A study on a typical fillet weld shape is demonstrated. LxH weld rectangular domain is assigned as the initial volume. At the end of the analysis minimum Von Mises stress condition is achieved at the volume of around 12% of the initial domain.

The topic of Fracture Mechanics uses the ESO widely also. There are many accomplished studies in the literature [6, 4, 5, 17].

CHAPTER 3

PARAMETRIZATION OF THE EXCAVATOR BOOM

3D geometry of excavator boom is required to be constructed in a fast manner to be able to search a larger design space in optimization. Hence, geometry of boom model is defined in terms of points, angles, radii, and lengths by constructing mathematical relations between them. In the study done by Yener [35] an excavator boom has been defined in terms of varying and fixed parameters. A computer program which is capable of computing the relation among the parameters and performing pre-process phase of the FE analysis has been developed. By this way it became possible for the designer to analyze several FE models in a short time. The computer code developed by Yener [35] has been modified and extended to incorporate an optimization algorithm. The parametric definition of the boom model is discussed in this chapter.

3.1 Geometry Parameters

There are 15 geometry parameters which are required to create the 2D side form of the boom model (Figure 3.1). α_1, α_2, α_3, and α_4 are parameters indicating the angles of upper and lower plates and, R_1, R_2 represent curvatures of the upper and lower plates at the middle section of the boom. Point p_1, p_2, p_3, and p_4 are starting points of upper and lower plates. In otherwords, upper and lower plates are connected (welded) to arm and chassis mounting brackets at these points.

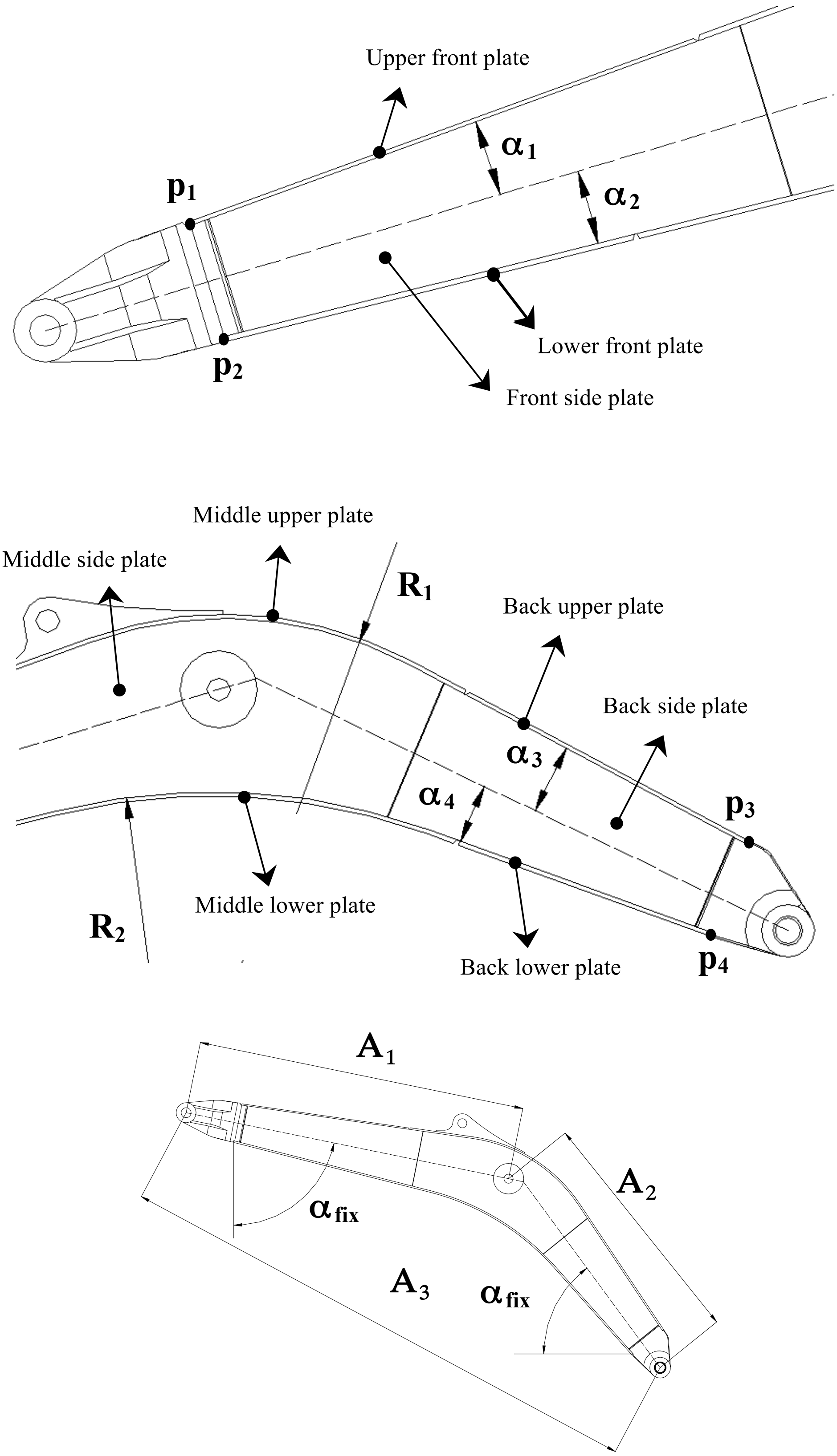


Figure 3.1 – View of α_n, R_n, p_n, A_n and α_{fixn} parameters and name of plates which are used in construction of a boom.

In addition to mentioned p_n points, α_{fix1}, α_{fix2} and A_1, A_2, A_3 are required to define positions of mechanism joints (arm, chassis connections and boom hydraulic cylinder connection).

Thickness of plates, length of side plates, reinforcement plate positions are other required parameters used in optimization phase. t_1, t_2, t_3, t_4, t_5, t_6, t_7, and t_8 form parameter set of thickness and L_1, L_2 are the parameters of reinforcement plate positions and L_3 and L_4 are the parameters designating length of side plates (Figure 3.2).

3.2 Assumptions

Generally, forging or casting is preferred for the production of arm and chassis brackets, and middle boss, so that these structures have high strength properties and are not determinative directly in fatigue life determination of boom. Also, parameters (p_1, p_2, p_3, p_4, A_1, A_2, A_3, α_{fix1}, and α_{fix2}) related to mechanism remain same in order to keep predetermined mechanism envelope and digging forces same. Hence, shapes and positions of arm and chassis brackets and middle boss, and parameters of mechanism joints are not concerned in optimization.

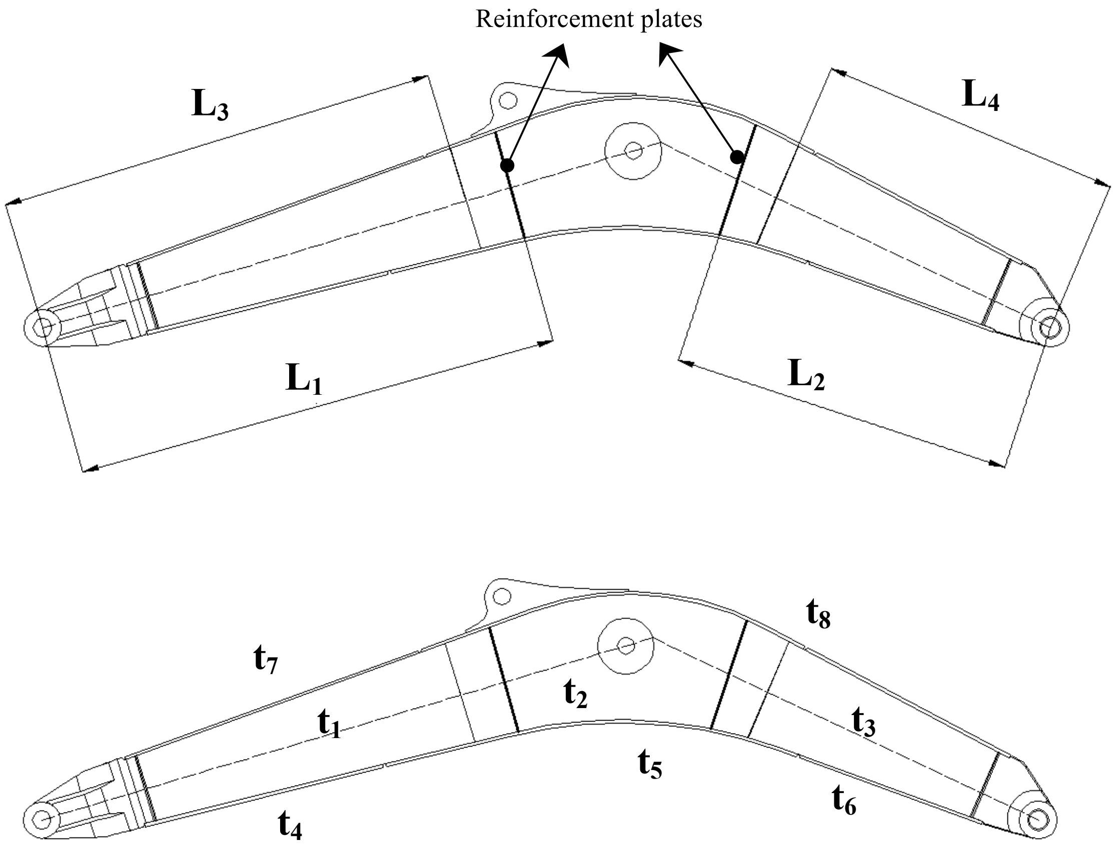


Figure 3.2 – Representation of L_n and t_n parameters.

Software creates the boom models by starting from existing FE models of these parts instead of modelling them repetitively. Because of this, altering position parameters of these parts and mechanism joints is unnecessary and these parameters do not contribute to optimization process.

Secondly, start and end positions of some plates are restricted harshly because of limitations in production phase. For instance, if one desired to weld the lower plate in two pieces, the intersection of plates should be positioned in Z position as shown in figure 3.3 always.

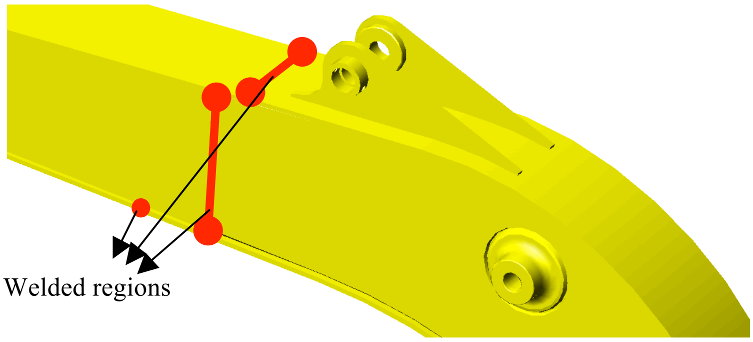


Figure 3.3 – Z type welding in excavator boom

If designer includes the lower and upper plate positions in optimization process as variable parameters, many infeasible models will be created. Thus, instead, designer determines fixed positions for the intersections of upper and lower plates and then alternates the position of intersection of side plates between these fixed positions. Distance, L_1, between the welds of upper and side plates, and distance, L_2, between the welds of lower and side plates should not be smaller than minimum distance, L_{min}, in order to avoid any possible overlapping of residual areas of welds (Figure 3.4). In Hidromek, L_{min} is kept nearly as 80~100 mm generally.

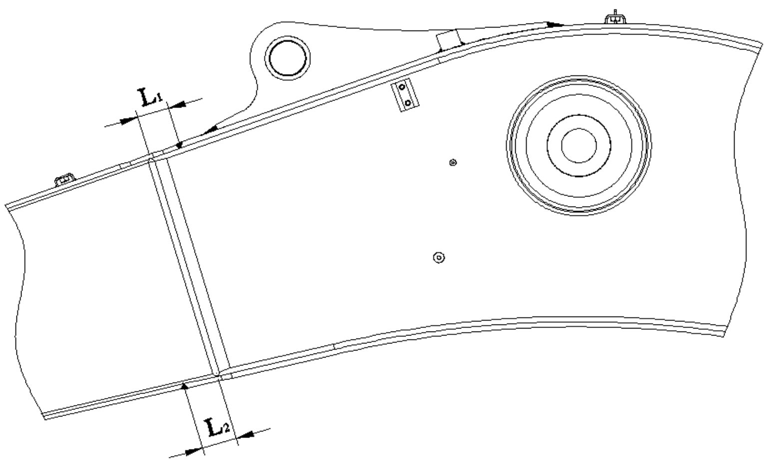


Figure 3.4 – L_1 and L_2 distance between welds.

CHAPTER 4

FINITE ELEMENT MODELING

Finite element method is a powerful technique to solve complex strength problems such as shear or bending stress calculation on the boom cross section while excavator is digging the earth.

In a boom, all parts are manufactured from sheet metal except the middle cylinder boss and rear bushings which are forged or cast. For these parts, tetrahedral solid elements are used; whereas for the sheet metal parts quadrilateral thick shell elements are preferred.

Since the optimization process spends a huge computational time for the finite element analysis, the element size is a crucial parameter in the finite element method. The element size should be determined such that analysis should maintain accurate results within minimum computational time. Previously, Yener made a convergence check to find the appropriate element size [35]. HMK220 LC excavator boom analysis was carried out using both 21473 and 51929 elements (average element size for the models are 40 mm and 20 mm respectively). The stress results showed a difference in between 0~7.3% and the time consumed for the computation of the model with 21473 elements is approximately 50% less than the time consumed for the other model. Hence, 40 mm mean element size is selected on this study.

The finite element model used in this study does not include any special tying between shell and solid elements, and any weld between the sheet metals are not taken into account. Although, the fatigue life of the design is directly related to the fatigue life of the welds, stress information regarding welded regions is not available directly due to the lack of correct nodal connections. The handicap is overcome by the help of approximations discussed in section 5.2.

4.1 Load Cases

The load cases applied in the finite element analysis should be able to simulate real loading on the excavator boom. A full loading and discharging cycle consists of infinite number of loading conditions. The study done by Carlgren et al. defines a set of theoretical load set that idealizes a full digging cycle in the analysis [2]. His set consists of 8 basic load cases related to the break out, lifting, swelling and emptying phases. The point and direction of application of each load cases are represented in figure 4.1.

Actually this type of multitudinous load case definition is hard to implement in an optimization process because of the requirement of large computational time. Therefore, it is sufficient to take most frequently used load cases into consideration in the analysis [2]. The load cases determined for this study are almost equivalent for the load cases LC01 and LC04 shown in figure 4.1. Arm breakout force and lateral force dependent to the slewing and emptying actions are the clear definitions of the mentioned load cases respectively in this study.

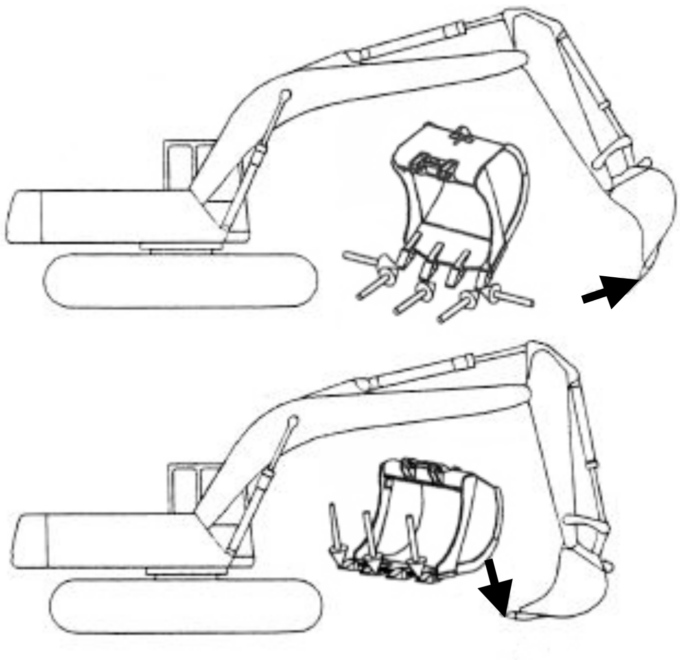


Figure 4.1 – Load-cases for excavator boom. External forces on the bucket [2]

4.1.1 Arm Breakout Force Calculation

Infinite number of resultant forces may be exerted due to activation of the arm and bucket cylinders, at the bucket tip as the load cases. To idealize the loading condition, the declared arm digging force calculations instruction by the SAE J1179 [28] is applied.

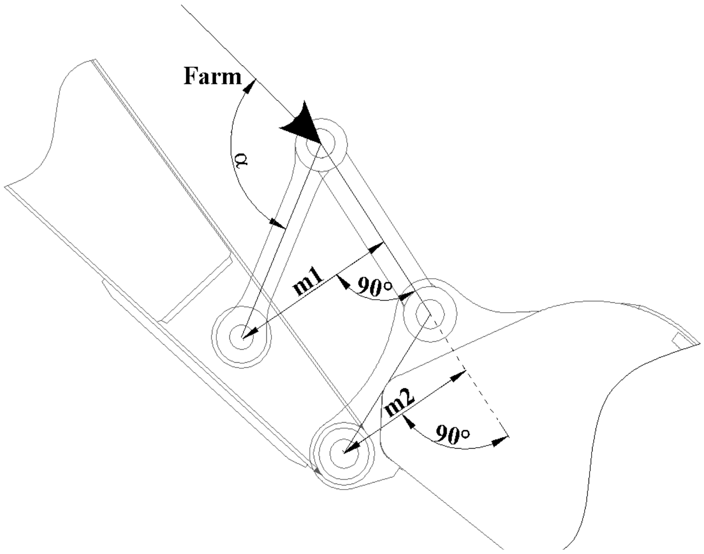


Figure 4.2 – Notations used in the moment calculation at bucket mounting.

$$M = F_{arm} \cdot \sin(\alpha) \cdot \lambda \cdot \frac{m_2}{m_1}$$

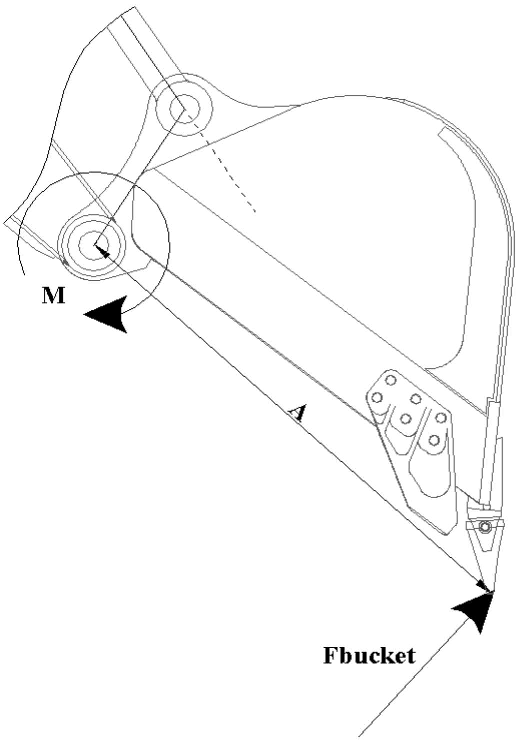


Figure 4.3 – Notations used in the force calculation at the bucket tip.

$$F_{bucket} = \frac{M}{A}$$

The variation of the bucket cylinder length with respect to bucket force is shown graphically in figure 4.4 [35]. s3 and F represent bucket cylinder length and bucket breakout force respectively. It is seen that bucket cylinder length should be set to 74% of the total bucket cylinder length in order to obtain maximum bucket breakout force. After the bucket is positioned, the position of the arm is arranged to maintain maximum moment.

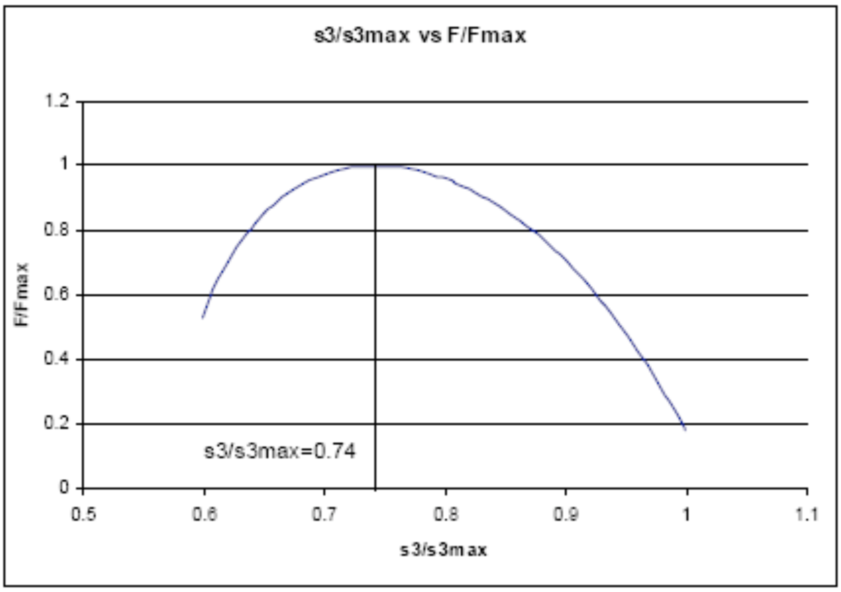


Figure 4.4 – s3/s3max vs F/Fmax [35]

Maximum arm breakout force mentioned as F_{arm} in figure 4.5, occurs while the arm cylinder acts on the arm at perpendicular angle. Figure 4.6 shows the change of arm cylinder length respect to arm breakout force.
S2 and F represent arm cylinder length and arm breakout force respectively. It is seen that arm cylinder length should be set to 73% of the total arm cylinder length in order to obtain maximum arm breakout force.

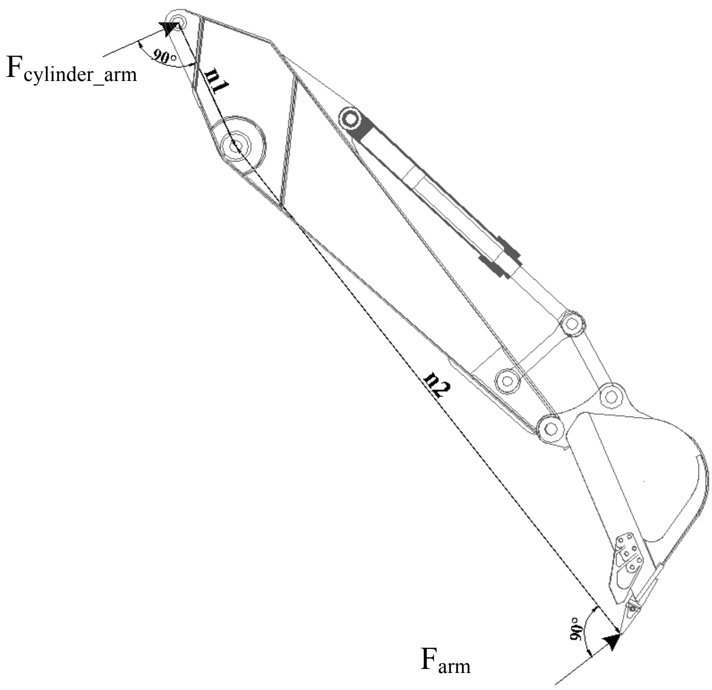


Figure 4.5 – Maximum arm breakout position

$$F_{arm} = F_{cylinder\_arm} \cdot \frac{n_1}{n_2}$$

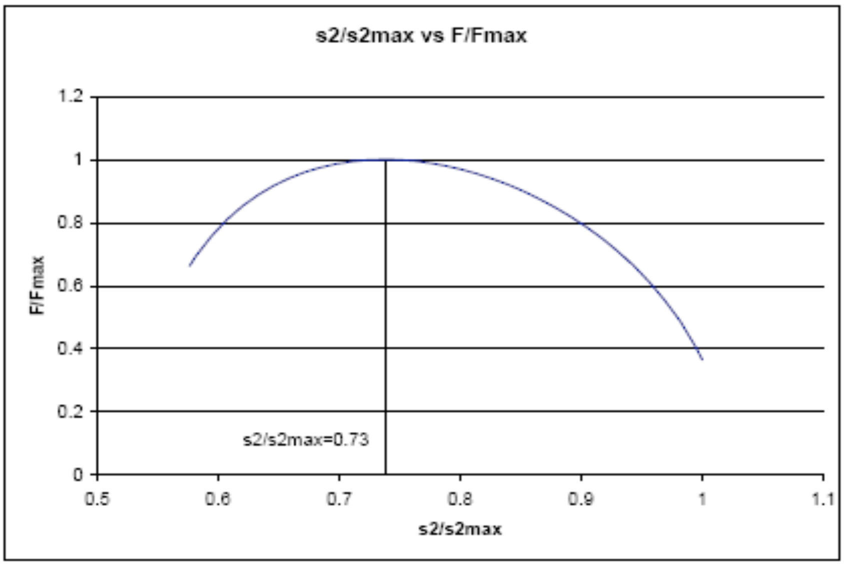


Figure 4.6 – s2/s2max vs F/Fmax [35]

4.1.2 Lateral Force

The excavator boom is subjected to torsional and lateral forces when the main body turns around for loading the dipped up earth and sand into a truck or similar carrier [2, 29]. The inertial forces, mainly caused by the loaded bucket, create the mentioned forces on the boom structure. Therefore, lateral force can be calculated as,

$$F_{lateral} = m \cdot a$$

where a is the acceleration which is measured for a loaded bucket during truck loading simulation and m is the total mass of the bucket and load in it.

Also excavator operators use side of bucket for sweeping the earth and shoving rocks by rotating the upper chassis by means of a hydraulic motor. Hydraulic motor applies a moment M to upper chassis and this moment induces a force $F_{lateral}$ at the bucket tip (Figure 4.7).

Designer should be aware of such lateral loads and should determine appropriate lateral load to use in design phase.

4.2 Boundary Conditions

The correct application of the boundary condition is important as much as the load cases to obtain physically reasonable results. Displacements and rotations are restricted on the upper chassis mounting of the boom structure in FE model allowing rotation in z axis that enables the boom to oscillate at vertical direction.

Boom cylinders and arm cylinder are replaced by linear elements which represent the same cross sectional properties. Although the hydraulic cylinders have ability to move in axial direction in practice, axial movement is not allowed in the analysis. Therefore any hydraulic discharge due to high pressure is neglected and only elasticity of the line elements are taken into consideration. It should be noted that same type real cylinders are used in the strain gauge test phase of the boom structure. The types of pivot connections of line elements are also the same as those used in upper chassis mounting point except for the displacements being allowed for all directions.

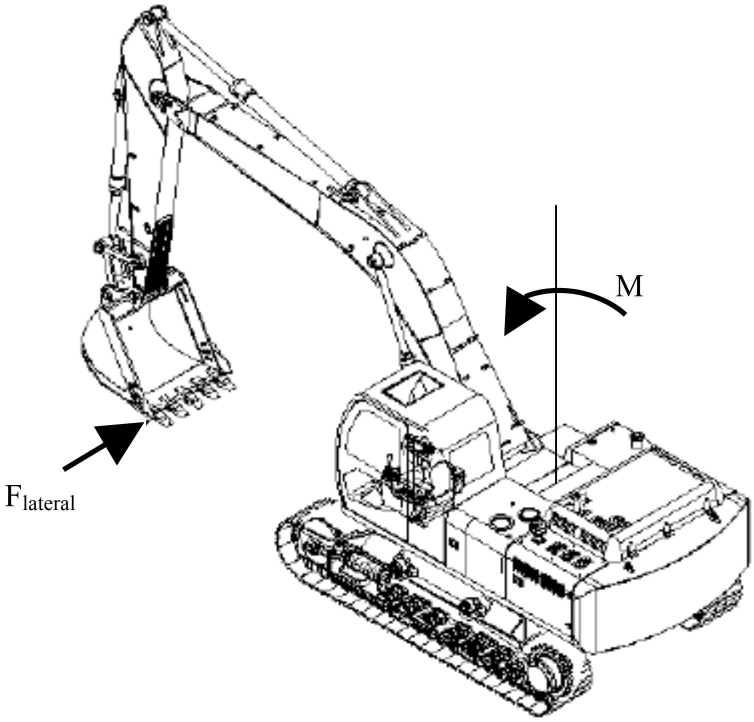


Figure 4.7 – Moment created by hydraulic actuator induces lateral force $F_{lateral}$ at the bucket tip. [35]

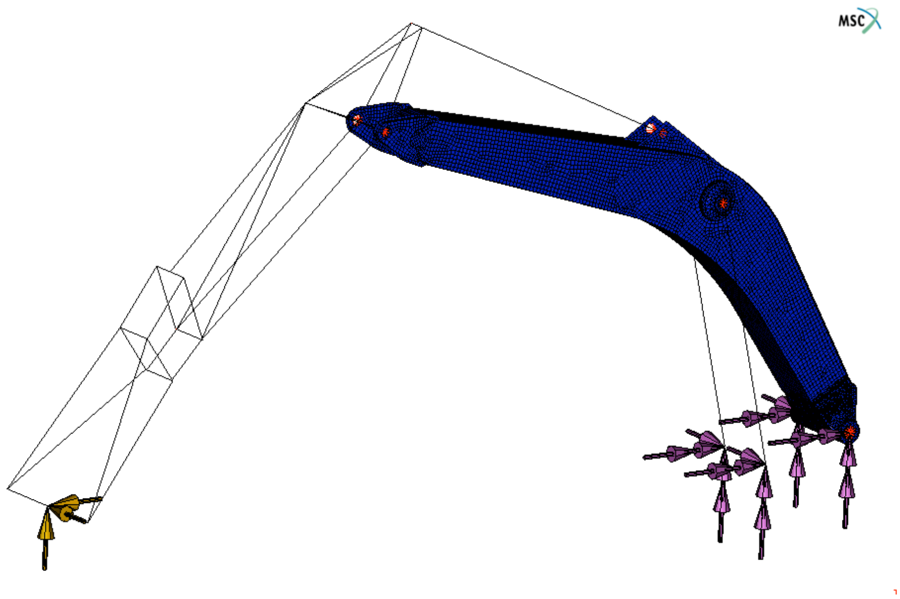


Figure 4.8 – Presentation of boundary and load conditions

CHAPTER 5

DESIGN CRITERIA OF EXCAVATOR BOOM

An excavator boom which is in the form of banana shape from the side view comprises a chassis bracket, boom cylinder boss and an arm connection bracket. All these parts are assembled with sheet metals such that a hollow rectangular tube is obtained from the welded construction. Forging, casting or sheet metal forming may be preferred for the manufacture of the boss and brackets.

The hollow rectangular section is supported with reinforcements placed parallel to the cross section. The reinforcements increase the structural rigidity markedly against the torsional loads and prevent side plate buckling. The figure 5.1 shows the effect of the reinforcement for the deformation of the cross section.

However, attaching the reinforcement materials in the welding construction is a challenging issue and the quality of the reinforcement welds may be poor depending on the manufacturing facilities and the quality of the weld operators. Also, there have been initial crack already due to the existence of weld. Therefore, the welding standards suggests lower allowable stresses in order to maintain required fatigue life due to poor quality welding and this in return increases the weight of the structure. Also, reinforcement itself is an extra weight source in the structure.

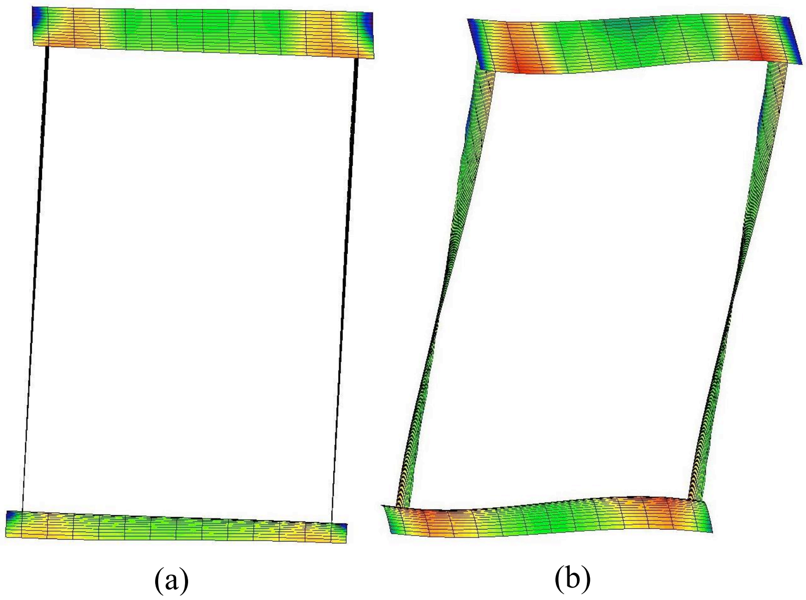


Figure 5.1 – Partial cross sectional deformation view of an excavator boom structure (a) with reinforcement (b) without reinforcement

Actually decreasing the thickness of the sheet metal and increasing the cross sectional area of the hollow boom structure is a logical way to design a light weight boom. But, the rigidity of the surface in the outward direction decreases proportionally to the third power of a ratio of the reduction of the plate thickness as the thickness of the plates is reduced [29].

Figure 5.2 depicts the generated deformation against the torsional and lateral forces if the thickness of each plate is reduced. The structural rigidity drops dramatically and angular deflection (α_{def}) at corners increases. Reinforcement materials support the structure in such a case and they are very effective to decrease the angular deflection.

Actually, the design of the boom is markedly dependent on a good engineering intuition, good knowledge of using the welding standards and successful application of the stress methods on finite element analysis.

5.1 Basic Principals of Stress Calculations on Welded Constructions

As it is mentioned before, the FE model of the boom structure does not include welding design information. Hence, nominal stress approximation is accepted as the method to obtain stress values around the welded region comparable with weld design codes.

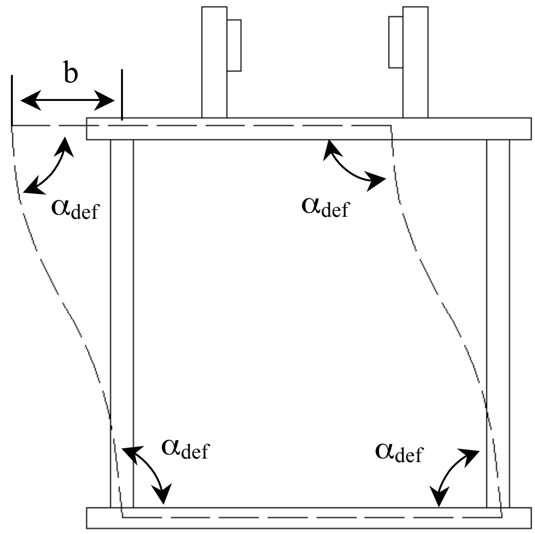


Figure 5.2 – Deformation of the cross section due to lateral and torsional forces. α_{def}, and b are the amount of the deformation. b represents directional deflection while α_{def} is angular deflection. [29]

The nominal stress can be reckoned away from the weld region. Therefore, the stiffness of the weld does not contribute to nominal stress and because of this it is not required to design the weld in the finite element model [8]. In the "Fatigue Design of Welded Joints and Components" published by International Institute of Welding

[15], there is a set of recommendations which gives guidelines for the welds by means of stress limitations and fatigue life. The recommendations present the fatigue resistance according to the nominal stress by tables of structural details in terms of a set of S-N curves.

5.1.1 Nominal Stress

The stress calculated on the net cross section of the structure disregarding the local stress raising effects of the any joints and structural detail but regarding the macro geometrical effects is called Nominal stress. The nominal stress may vary over the cross section depending on the loading condition and the macro geometrical effects. Thus, macro geometrical effects are required to be taken into consideration in the calculation of nominal stresses.

5.1.2 Calculation of Nominal Stress

The theory of structural mechanics may be used to determine the nominal stress in simple structures. For the complicated hyperstatic structures and the structures including macro geometric discontinuities that are unable to be solved analytically, finite element method is employed [15]. It should be noted that all stress raising effects of the structural details are ignored. Commonly principal stresses are used as the nominal stresses in finite element analysis.

It is assumed that the nominal stress must vary linearly [8] if no external loads exist in the area under study. The below graph (Figure 5.3) is drawn in order to determine the nominal stress in a finite element model. The main idea of the approach is that if a linearity of the stress values along a path rises, the geometric component of the total stress becomes zero. In such a case the nominal stress is achieved by extrapolating the linear stress curve towards to the weld toe.

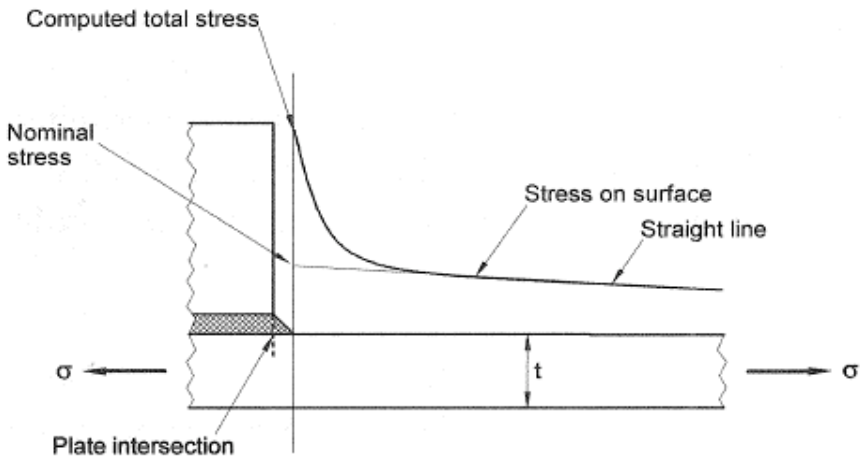


Figure 5.3 – Extrapolated nominal stress in welded joint.[8]

The figure 5.4 shows an example for nominal stress calculation by using finite element technique.

The extrapolation line must lie perpendicular to the expected path of the crack. In other words, the line must lie parallel to the load direction. It is momentous to remember that stress path should be long enough to see the behaviour of the graph clearly. In case of parallel loading along the weld path, nominal stress can be found directly from the finite element model.

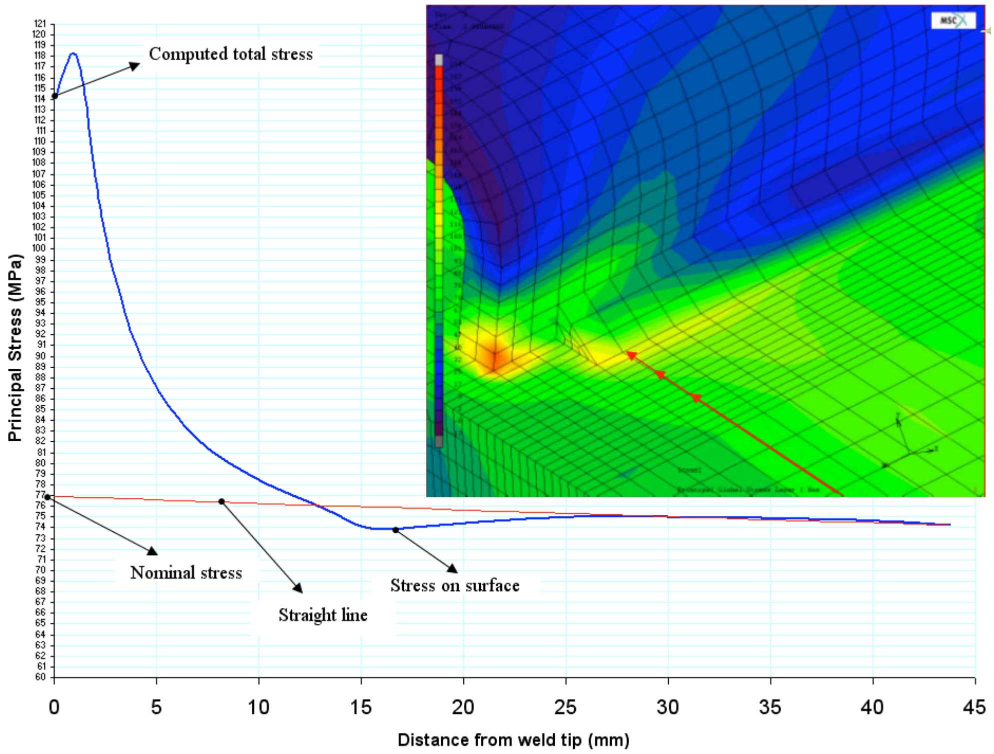


Figure 5.4 – Extrapolated nominal stress in welded joint.

Briefly, one should be aware about the three issues mentioned below [8];

- The region of interest must not be too complicated by reinforcement materials and intersecting plates and the nominal stress should be defined clearly.
- The joint type and loading condition must exist in one of the fatigue classes available for nominal stresses
- The structure should maintain the defined limits in the fatigue class exactly without any defects and misalignments.

5.2 Determination of Stress Limitations

Fillet and butt welds are the most frequently used weld joint types in the construction of the boom structure. The perpendicularly oriented plates are joined with the fillet welds and the plates closing the upper and lower openings of the rectangular hollow structure are joined to each other with butt welds. The stress requirements will be discussed and presented by focusing on the critical regions of the welded construction. Previously done fatigue experiments, strain gauge measurements and the comparison of the data come from these works with the computational environment and the recommendations declared by the authorities, creates the substructure of this section.

Three sample parts eligible for the comparison with the recommendations are taken into observation. Furthermore, these parts are chosen from such locations in that numerous failures occur in the fatigue experiments and also in real life. Figure 5.5 shows the regions that will be examined.

Region 1 is a sample for two plates welded perpendicular to each other, region 2 is the welded joint connecting the reinforcement material to the lower plate and region 3 is the sample of butt weld making the connection of two lower plates.

5.2.1 Region 1: Double Sided Fillet Weld

The structure formed by the perpendicularly welded plates at region 1 is similar with the shape declared in the IIW recommendation by the number of 321, 322 and 323. The figure 5.6 depicts the detailed pictures of the structure.

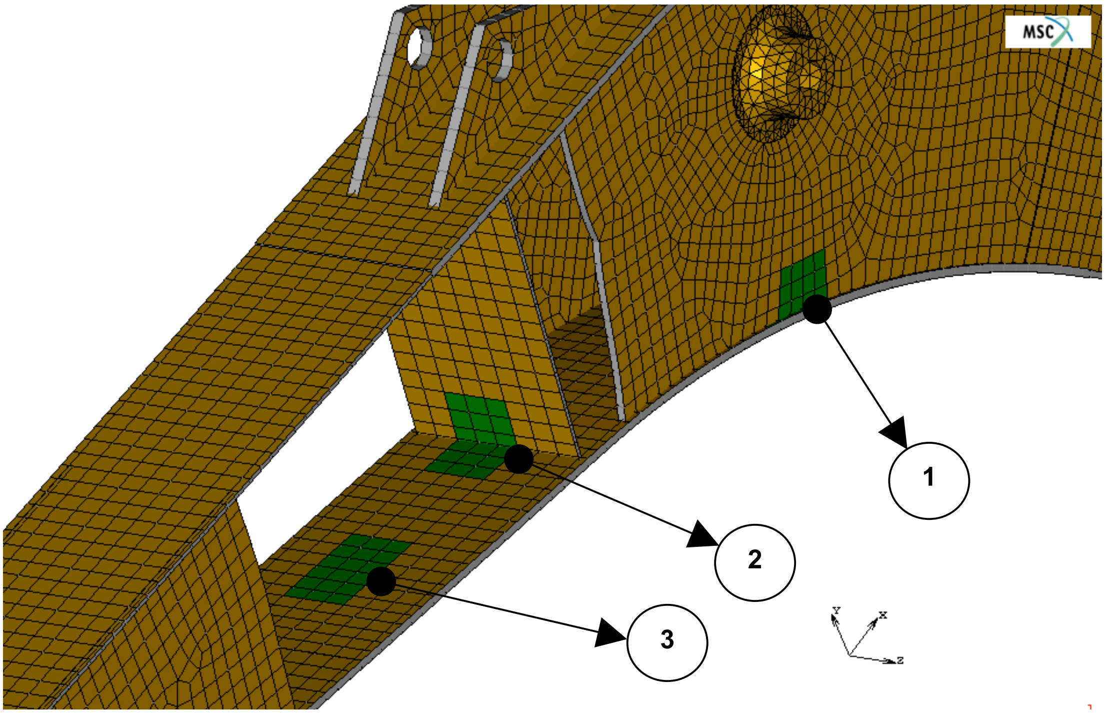
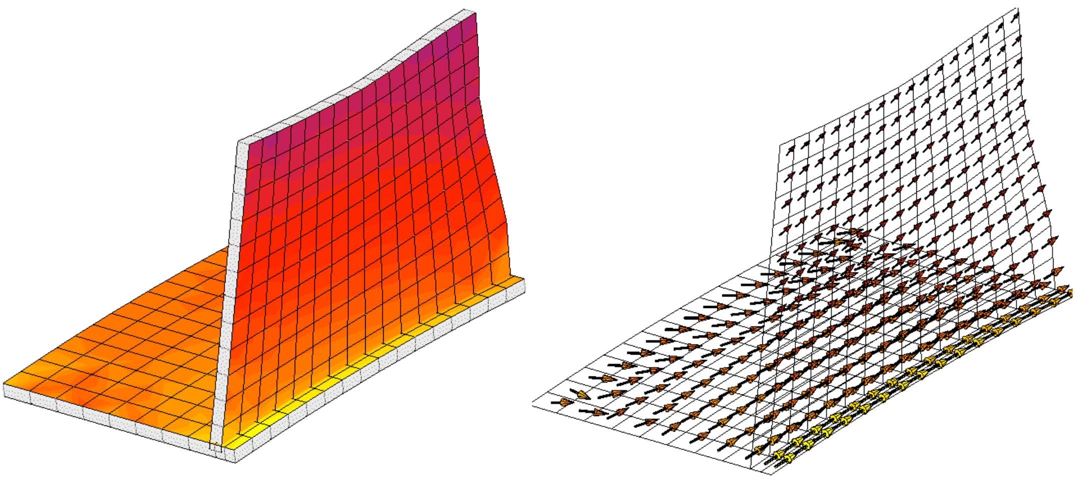


Figure 5.5 – Representation of three typical weld joint on the boom structure.



(a) (b)

Figure 5.6 – (a) represents a detailed picture of region 1 and (b) represents the main loading direction of the part

Also the main loading direction on the mentioned part exactly satisfies the loading detail given on the recommendation. The manufacture processes create differences among the three fatigue classes (Figure 5.7). Different applications exist for different designs in Hidromek production facilities also. But, the detail 323 is preferred among the others since stress value obtained from S-N curve (Figure 5.8) related to detail 323 at aimed fatigue cycle ensures the design to be on the safe side.

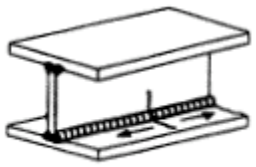
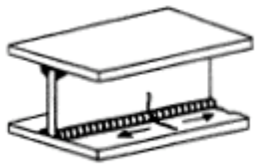
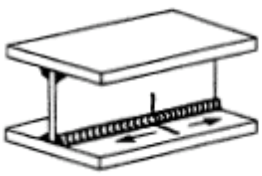
321		Continuous automatic longitudinal fully penetrated K-butt weld without stop/start positions (based on stress range in flange) NDT	125
322		Continuous automatic longitudinal double sided fillet weld without stop/start positions (based on stress range in flange)	100
323		Continuous manual longitudinal fillet or butt weld (based on stress range in flange)	90

Figure 5.7 – IIW Fatigue Resistance Codes for Steel [15]

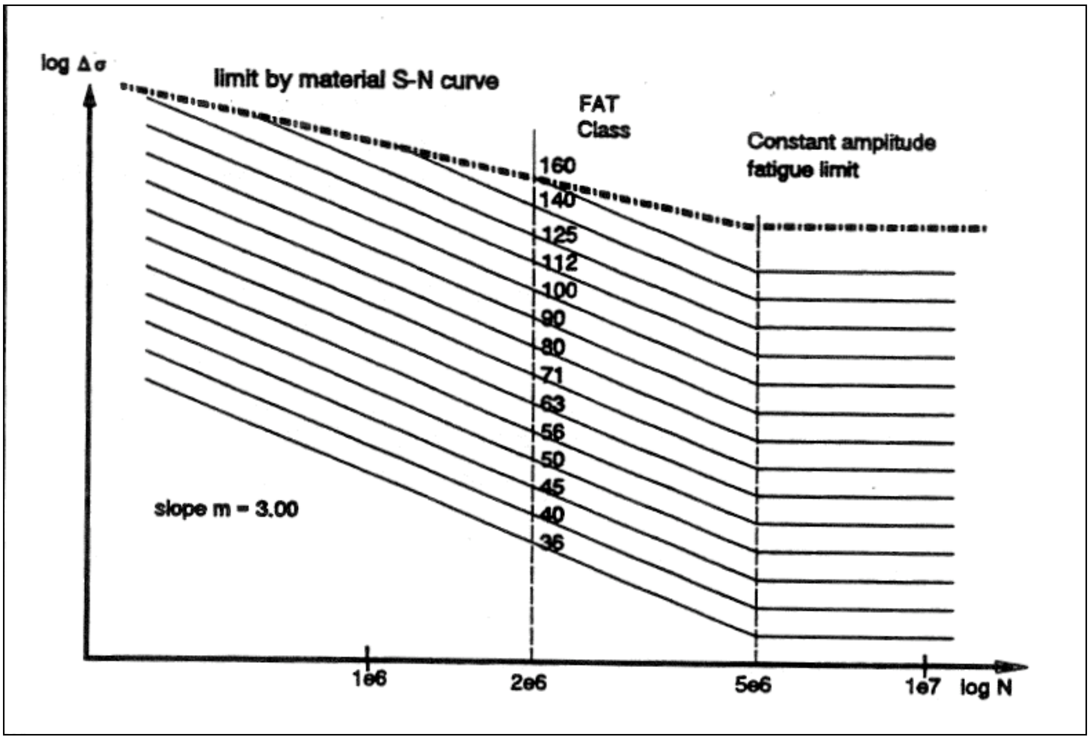
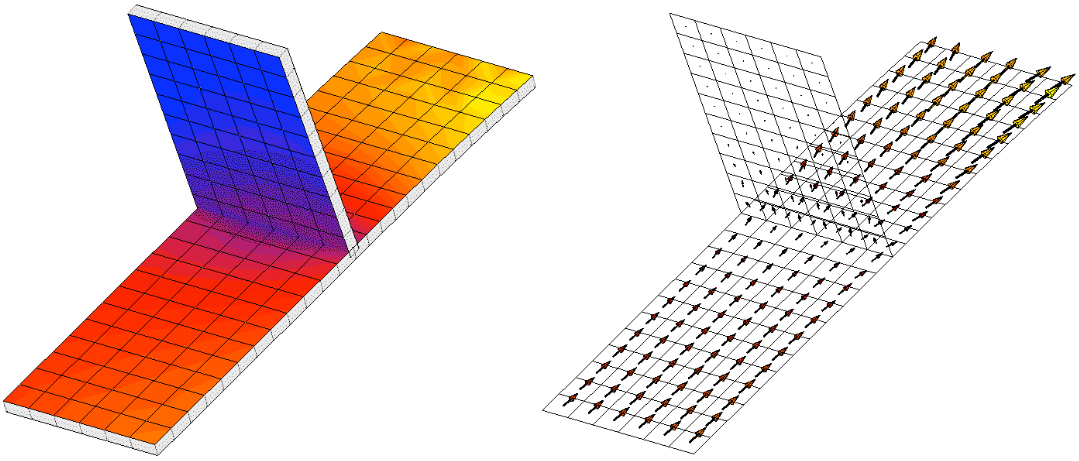


Figure 5.8 – S-N Curve for steel [15]

5.2.2 Region 2: Transverse Non Load Carrying Attachment

The reinforcement and lower plate connection can be discussed as a non-load carrying part because the reinforcement plate side of the part does not carry almost any load as it can be seen clearly in figure 5.9.



(a) (b)

Figure 5.9 – (a) represents a detailed picture of region 2 and (b) represents the main loading direction of the part

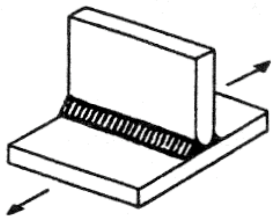
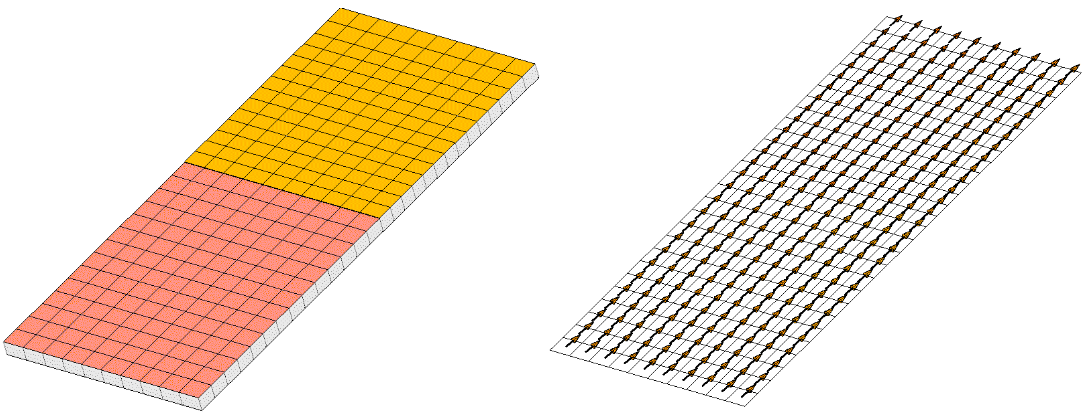
511		Transverse non-load-carrying attachment, not thicker than main plate K-butt weld, toe ground Two-sided fillets, toe ground Fillet weld(s), as welded Thicker than main plate	 100 100 80 71

Figure 5.10 – IIW Fatigue Resistance Code for Steel [15]

The structural detail of 511 given in figure 5.10 satisfies the shape requirement desired for the part given in figure 5.9. The thickness of the reinforcement plate is never greater than the welded lower plate and welds can be thought as fillet welds as welded. Hence, the FAT can be chosen as 80.

5.2.3 Region 3: Transverse Butt Weld

Butt Weld is used to make the side by side connection of two plates (Figure 5.11). The plates are positioned in flat before the welding process and the weld ground flush is applied partially at the tips of the weld line. The transition between two plates with different thickness is maintained with at least 1:4 slope. Under these circumstances, structural detail 222 given in figure 5.12 can be used and FAT can be chosen 100 for slope 1:5.



(a) (b)

Figure 5.11 – (a) represents a detailed picture of region 3 and (b) represents the main loading direction of the part

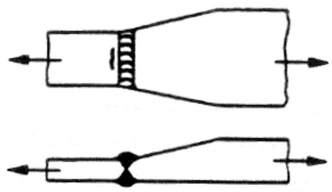
222		Transverse butt weld made in shop, welded in flat position, weld profile controlled, NDT, with transition in thickness and width: slope 1:5 slope 1:3 slope 1:2 For misalignment see 3.8.2	 100 90 80

Figure 5.12 – IIW Fatigue Resistance Code for Steel

Three different sample parts representing three different weld joint types are examined in order to deduce a comprehension of stress limitation in the design of the boom structure. To develop such an understanding on the stress limitation, determined stress values and engineering intuition is required. In the light of the mentioned explanations, mean and maximum stress limitations are determined for each critical region as the design criteria of the boom structure. In addition, it should be noted that the principal stress values are obtained from the finite element analysis. However, in order to make the average stress goal comparative with the yield criteria, Von Mises stress is used in the design phase. It is possible to achieve Von Mises stress goals at all the desired locations disregarding the tensile or compressive loading distinction. Thus, this handicap caused by the static loading is overcome.

5.3 Other Design Limitations

Excavator boom has the ability to swing vertically at the upper vehicle mounting and turn relative to the lower body. Therefore, some geometrical limitations should be considered to prevent the boom not to have interference with the other parts.

Also some other limitations exist depending on the productivity of the boom. These limitations may be listed as;

The manufacturers always classify thickness of the plates as the integer values and most of the thickness values are even numbers. Actually any plate at desired thickness may be manufactured, but the cost will increase. Therefore, a design completed by disregarding the availability of the plates in the market would not be reliable.

Another issue depending on the productivity is that difference between the thickness of the plates welded to each other should be in the limits declared by the directives not to decrease the joining performance of the welds.

Positioning of the plates should be taken into consideration during the design phase. For instance, it is not possible to place reinforcement plates into the rectangular hollow structure of the boom effectively if the lower and upper plates are considered to be single piece without any weld.

CHAPTER 6

PROBLEM DEFINITION

In chapter 3 the geometry of the boom has been defined by using fixed and variable parameters. 18 variable parameters, which are contributing to the design of the boom, are called design parameters. Objective function and design constraints will be defined in terms of design parameters in this section. It should be noted that the design problem in this study is minimizing the mass of design while satisfying stress limitations and acceptable geometry.

6.1 Design Variables

As previously mentioned, 18 varying parameters producing the boom geometry have significant influence on the design of the boom. Mass and stress values are sensitive remarkably to the variation of these parameters. Hence, these parameters are most suitable to use in boom design.

Design variables can be represented in form of a vector (X) which is a column containing the design variables.

$$X = [\alpha_1 \quad \alpha_2 \quad \alpha_3 \quad \alpha_4 \quad R_1 \quad R_2 \quad L_1 \quad L_2 \quad L_3 \quad L_4 \quad t_1 \quad t_2 \quad t_3 \quad t_4 \quad t_5 \quad t_6 \quad t_7 \quad t_8]^T$$

The parts, when manufactured may show variations in the dimensions due to the quality of the machines used and welding deformations. Therefore, one should be aware of the production sensitivity of the plant where the designed boom will be produced. It is useless to perform long computations to increase design precision if the production facility is not capable of manufacturing at same precision with the design. A second issue is that the sheet-metals are considered to be used in design are only available in limited thickness. Hence it is useless to select thickness values from a continuous range of values. Depending on the mentioned problems, the variables are taken as discrete-valued design variables. The sensitivities of the parameters a_1, a_2, a_3 and a_4 is assumed 0.01^o and for R_1, R_2, L_1, L_2, L_3 and L_4 parameters, sensitivity is set to 1 mm considering the sensitivity in Hidromek production plant. The available sheet metal thickness in Hidromek reserves are 8, 10, 12, 14, 15, 16, and 2 mm increments (i.e. 18, 20, 22 …) so the values of the thickness variables (t_1, t_2, t_3, t_4, t_5, t_6, t_7, and t_8) can only be chosen from this discrete-set of values.

6.2 Objective Function

Stress and mass, which are the two most important parameters, determine the performance of the boom design. The limitations for the stress values have been discussed previously. Hence it would be reasonable to minimize the mass while satisfying the stress limitations. Mass can be described as;

$$Mass = m(X, \rho)$$

Here *m(X, ρ)* is the mass function of the boom. *X* designates the vector of design variables as mentioned before and ρ is the density of the used material in production of the boom. One can obtain different boom models in shape by altering design parameters of array *X*. Material required for the construction of each alternative will show variations in volume and in mass.

Generally steel is the preferred material in boom production. Since type of the available steel material is limited in the market, changing value ρ is not feasible to reduce the weight of the boom. In Hidromek st34 and st52 type steels are generally used and st52 is preferred for the boom construction.

To find the mass, commercial CAD program Msc. Mentat® is employed. The prepared pre-processor is capable of using Msc. Mentat automatically to prepare the CAD model of the boom and calculate the total mass.

Since it is aimed to obtain a lightweight boom which satisfies stress requirements, mass function, *m(X)*, can be selected as objective function, *F(X)*. The objective function can be described as;

$$F(X) = m(X, \rho)$$

It is important to note that this objective function will be modified to an unconstrained objective function in section 6.4 after the design constraints are defined in section 6.3.

6.3 Design Constraints

As previously discussed, limitations of stress values and design vector should be defined well in minimization process of the boom mass. Constraints restricting the design variables directly are called as *side constraints* and constraints related on performance or behaviour requirements such as stress limitations are called *behaviour constraints*. Side constraints for the boom design problem can be described as;

$$X_i^l \leq X_i \leq X_i^u \text{ , } i = 1, 2, \ldots, 18$$

Then side constraints can be written as;

$$g_i^s \equiv X_i - X_i^u \leq 0 \text{ , } i = 1, 2, \ldots, 18$$

$$g_i^s \equiv -X_i + X_i^l \leq 0 \text{ , } i = 1, 2, \ldots, 18$$

Side constraints define the boundaries of the design space. In other words, one can guarantee to obtain a boom model by using parameters chosen from the given ranges. The upper and lower bounds of variables should be defined carefully by observing the overall design of the boom including placement of the boom on the chassis, placements and geometries of other attachments, envelope of the attachments in 3D space and producibility of the boom. For instance, in figure 6.1 it is seen that the crawler prevents the boom to rotate at lower position. Because of this, α_4 angle should not exceed a determined maximum value, α_{4max}, to have no interference between the boom and the crawler.

It is also possible to create constraints defining the relations among the parameters. For example one may require holding the angle between the back upper and back lower plates at a desired value, so the side constraint should be defined as;

$$g_i^s \equiv \alpha_3 + \alpha_4 - \alpha^{desired} = 0$$

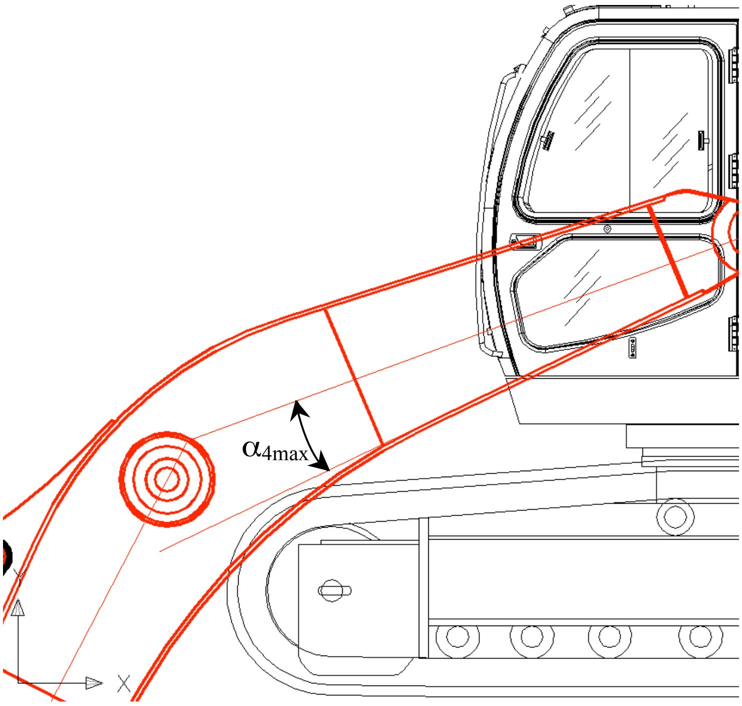


Figure 6.1 – View of a boom model. Angle of back lower plate should be limited by α_{4max}.

The developed computer program lets the user to construct said constraint equations easily. One can directly enter the variable bound information through the user interface for the designated eighteen variables (Figure 6.2).

The other constraint equations defining the relations among the parameters can be defined in the computer program by using simple Microsoft Visual Basic 6® codes. The computer program uses independent Visual Basic codes for optimization process so this makes it possible to modify and add new constraints for the optimization problem.

The critical locations on the boom such as high stress regions may vary according to boom type. For instance stress components caused by side loads increase significantly as the distance between the middle boss and the arm connection bracket increases. Then the designer may prefer to use reinforcement material to increase the rigidity. In such a case the high stress region may shift to somewhere else. Hence the designer should be aware of the critical locations on the boom structure by virtue of design experiences and engineering intuition. The number of critical locations is dependent on the decision of the designer and also the designer will make decision of stress requirements for each location separately. Stress limitations for the selected regions can be written as;

$$\sigma_{lower} \leq \sigma_i^{\max}(X) - \sigma_i^{aimed} \leq \sigma_{upper}\,, \qquad i = 1, 2, \ldots, n$$

where $\sigma_i^{\max}(X)$ and $\sigma_i^{required}$ are the maximum stress and aimed stress at the i^{th} region on the boom. Required stress is specified by using the approaches discussed in chapter 5. *n* represents the total number of the regions from where the stress data is collected. σ_{lower} and σ_{upper} bound the difference between the required stress and calculated stress.

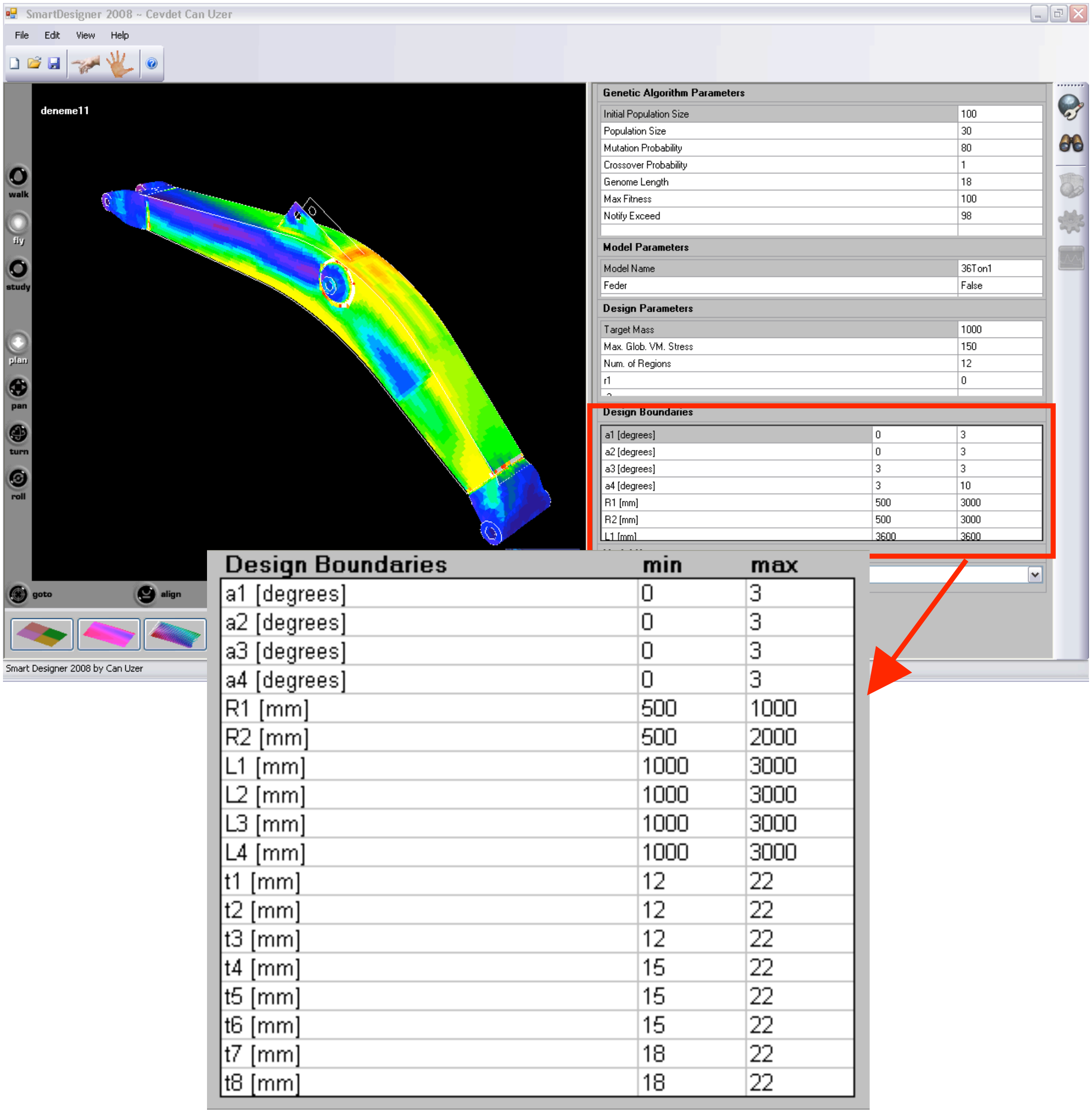


Figure 6.2 – View of design boundaries section in user interface.

So that desired stress value falls in an acceptable stress range rather than equal to a strict stress value. Approximately the error among the strain measurements and FE results is 10 %. By considering this situation, it is useless to search the design space for exact stress values. Otherwise the objective function becomes less manageable and it may be impossible to find an optimum. The behaviour constraints can be written as;

$$g_{2i-1}^{b} \equiv (\sigma_i^{\max}(X) - \sigma_i^{aimed}) - \sigma_{upper} \leq 0 \quad i = 1, 2, \ldots, n$$

and

$$g_{2i}^{b} \equiv (-\sigma_i^{\max}(X) + \sigma_i^{aimed}) + \sigma_{lower} \leq 0 \quad i = 1, 2, \ldots, n$$

In addition, another stress limitation for the global maximum stress is defined as;

$$\sigma_{global}^{\max}(X) \leq \sigma_{global}^{\lim}$$

where $\sigma_{global}^{\max}(X)$ is the maximum global stress at all over the model and $\sigma_{global}^{\lim}$ is the limit stress that should not be exceeded during the optimization phase. Then behaviour constraint equation can be written as;

$$g^{global} \equiv \sigma_{global}^{\max}(X) - \sigma_{global}^{\lim} \leq 0$$

The required stress values can be set up through the computer user interface as shown in figure 6.3;

6.4 Converting Constrained Minimization Problem into Unconstrained Minimization Problem

Solutions, which do not meet the problem requirements, are marked as infeasible and may be thought as completely valueless (zero fitness value). However, finding feasible solutions is difficult in highly constrained problems [9] and information of feasible solutions is required in optimization.

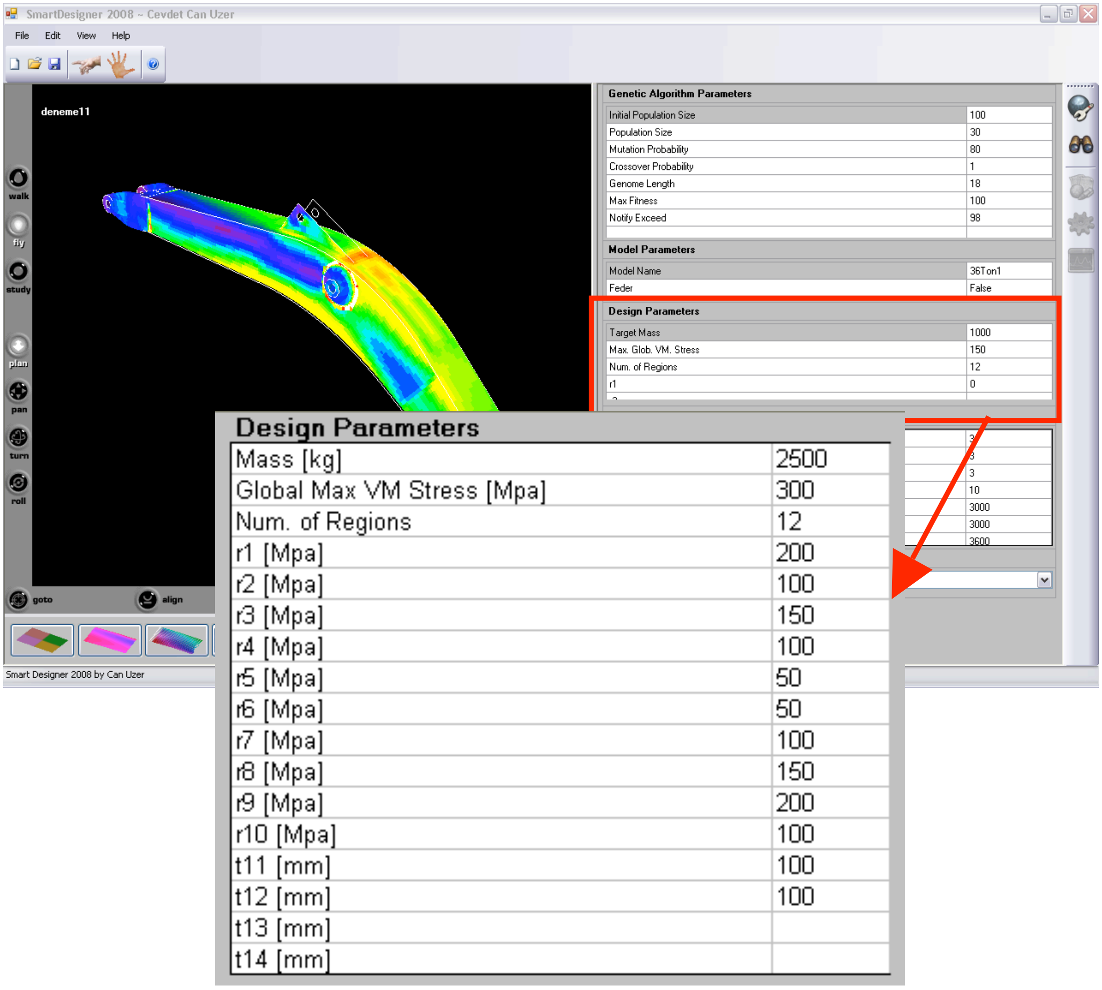


Figure 6.3 – View of design parameters section

Due to that information coming from the infeasible solutions such as models that are not meeting the desired stress values should be also added into evaluation of the solution as a degrading factor. Therefore an unconstrained objective function (penalty function) regarding previously mentioned behaviour constraints should be constructed.

The function can be written as;

$$\phi(X) = F(X) + R \cdot (r^{global} \cdot \langle g^{global} \rangle^2 + \sum_i^{2n} r_i \cdot \langle g_i \rangle^2)$$

where $\phi(X)$ is called as penalty function. r^{global} and r_i are penalty coefficients and sized for each constraint separately. Penalty coefficients should be set such that in case of moderate violations of constraints, the solution will be penalized at significant percentage of value of objective function. For this study r_i are set for 10% of the objective function and r^{global} is set for %50 of objective function. *R* is also a penalty coefficient that adjusts the influence of penalty factors in the penalty function. One can penalize the solutions harshly by directly increasing *R* value without changing r_i values. In general *R* is taken 1 in this study but computer program allows the user to change *R* value for the behoof of optimization. The behaviour constraints with *r* constants are added to increase $\phi(X)$ in case of violating the constraints. The form of penalty function given in here is a widely used one and one can prefer to add other nonnegative power of *g* rather than second degree *g* into penalty function. The bracket function $\langle g \rangle$ means

$$\langle g \rangle = \begin{cases} g, & \mathrm{g} \geq 0 \\ 0, & \mathrm{g} < 0 \end{cases}$$

As a conclusion, the objective function is substituted with the explained penalty function which aims to convert the defined constrained minimization problem into an unconstrained minimization problem and after that the object of the optimization problem is to minimize this new objective function.

CHAPTER 7

SOFTWARE STRUCTURE

In this study *Smart Designer* is used to optimize the shape of an excavator boom by constructing payoff function of design in terms of mass and Von Mises stress. Smart Designer is fully automated intelligent structural optimization software. It is able to make designs for any parametrically definable structural parts. In this chapter, the structure of Smart Designer will be discussed in detail.

7.1 "Smart Designer" Programs

Whole body of Smart Designer is formed by sub programs communicating each other. These programs are

GA Processor: Optimization process of the Smart Designer is managed by this Excel based program.

Msc. Marc Mentat: All the geometrical information is created by this commercial software

Msc. Marc: All the FE analyses are done by this commercial software.

Model Manager: This program includes the parametric definition of the model. It is able to check the correctness of the models, creating and analyzing them by communicating with Msc. Marc Mentat and Msc. Marc

Smart GUI: One can manage the Smart Designer by this interactive user interface. One can define the required design values, side constraints and geometric constraints easily through the Smart GUI and manage the optimization phase notwithstanding the running optimization process. Smart GUI lets the user to verify models and way of optimization by observing numerical and graphical data, and also 3D models of the structures from inside the program. Beside, one can create new models manually and analyze the sensitivity of the models against the change of a parameter.

7.2 "Smart Designer" Structure

In figure 7.1 the general structure of smart designer is shown. It consists of several different programs and supporting scripts and macros. The two main modules of Smart Designer are the optimization and manual designer modules.

User starts to optimize a structural body by defining structural design problem in the Smart GUI. Design constraints, design requirements, other inputs that are specific for the type of the problem and genetic algorithm parameters are entered into Smart GUI by the user (1). Smart GUI communicates with the GA Processor (2) and the information is taken as input by this sub program. In fact, all the essential job definitions for optimization exist in GA Processor. It takes orders from Smart GUI and reports the outputs for Smart GUI directly or saves them as data files.

The genetic algorithm needs model information that comprises stress, mass and geometric data during the optimization goes on running. GA Processor is unable to evaluate structural data; instead, auxiliary programs and codes do that. Set of design arrays are sent to model manager (3), checked whether they are feasible or not for the design and feedback information is sent to GA Processor (4). Beside this, GA Processor is also able to direct model manager to check feasibility of the models, create models and analyze them consecutively (5).

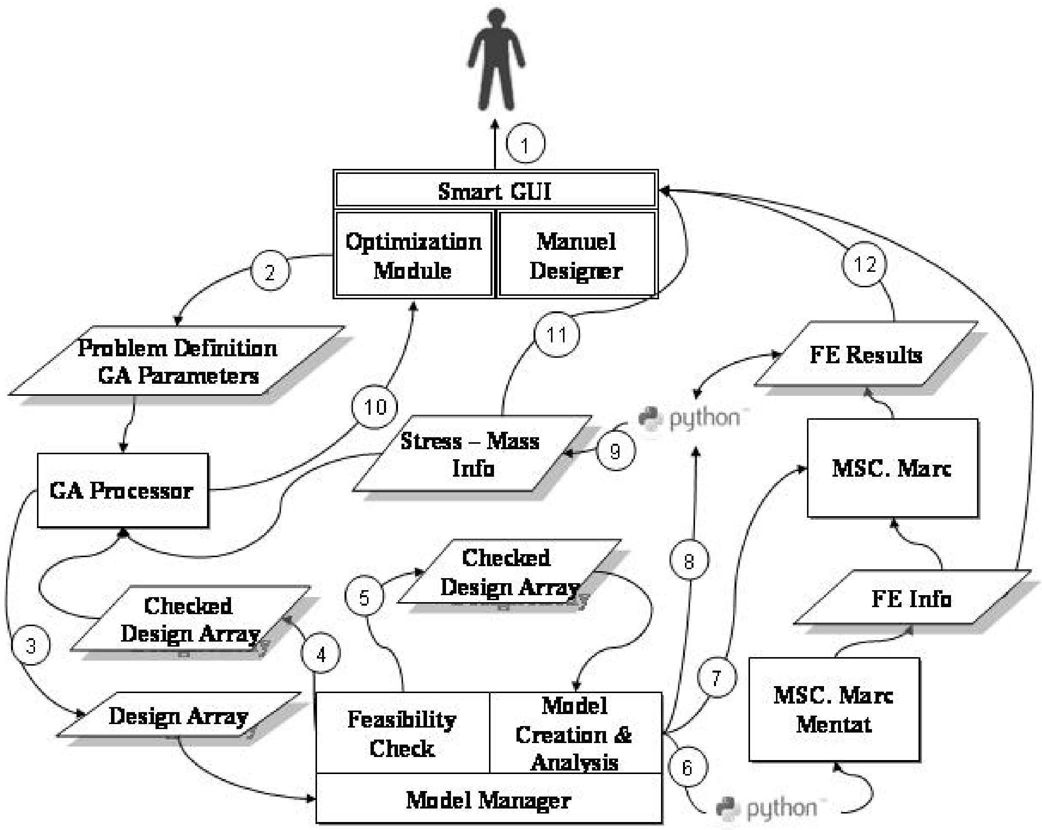


Figure 7.1 – Smart Designer system

Python modules in Smart Designer are vital parts of the program because communication with Msc softwares becomes possible with Python usage. Two special Python modules which are called PyMentat and PyPost, has been developed by Msc. The first module PyMentat helps Python codes to create new command areas that let the user to send Msc. Mentat commands directly to the program and the second module is used to process Msc. Marc result files. For each design, model manager creates Python codes that contain the geometric and FE information of the models and run Python codes to prepare FE models of each design (6). Then, model manager activates Msc. Marc to solve prepared FE models (7) and Msc. Marc stores the FE solution in one of its data storage format (t16 file). At this stage, model manager activates Python code that is using PyPost module to call stress data from Msc Marc result files (8). The collected stress data is stored in standard data files (txt) for multiple accesses of Smart GUI and GA Processor (9).

Smart GUI is able to call and display each type of data created by the auxiliary programs in the optimization phase (10, 11, 12, and 13). By this way user can watch the course of events and control each step easily.

7.3 GA Processor

GA Processor is an Excel and Excel Macro based computer code. The code consists of all the applications related to genetic algorithm in it. This section covers the sub functions of the GA Processor and implementation of genetic algorithm method.

7.3.1 Why Genetic Algorithm

Generating definite relations among the parameters and FE results of a model is quite crucial. Because of this, design engineer should decide intuitively and make successful guesses for further design alternatives to improve the design. Therefore, the issue of finding better is more important for design engineer rather than attain an optimum. By this point of view, this study aims is to develop such computer software that is able to make decisions like a human. The computer software should be able to sense the behaviour of the structure against the design loads and boundaries, and make successful decisions to obtain better designs. Hence finding an exact optimum is not the goal of this study.

Many optimization methods tend to converge for false peaks in a multi peaked design space. Whereas, genetic algorithm uses information coming from a large design area, thus the chance of converging to a local peak is minimized in this method.

Genetic algorithm is interested in payoff values of the input data. It does not matter how the input is evaluated. Therefore, genetic algorithm seems as appropriate method when a completely independent evaluation method such as FEA is considered.

Genetic algorithm guides the search for better results by using probabilistic information that comes from the solution set (population) created by genetic algorithm itself. Hence, genetic algorithm accomplishes to make judgements about the structural designs.

7.3.2 Application of Genetic Algorithm

A basic Genetic Algorithm is employed in this thesis. The algorithm consists of initialization, evaluation, elimination, mutation, reproduction phases. An initial population is created, and then the individuals in the population are evaluated in order to determine their fitness. Then elimination, mutation and crossover operations are applied respectively. The process goes on until a termination condition is satisfied. The figure 7.2 depicts the flow chart of the said operations in detail.

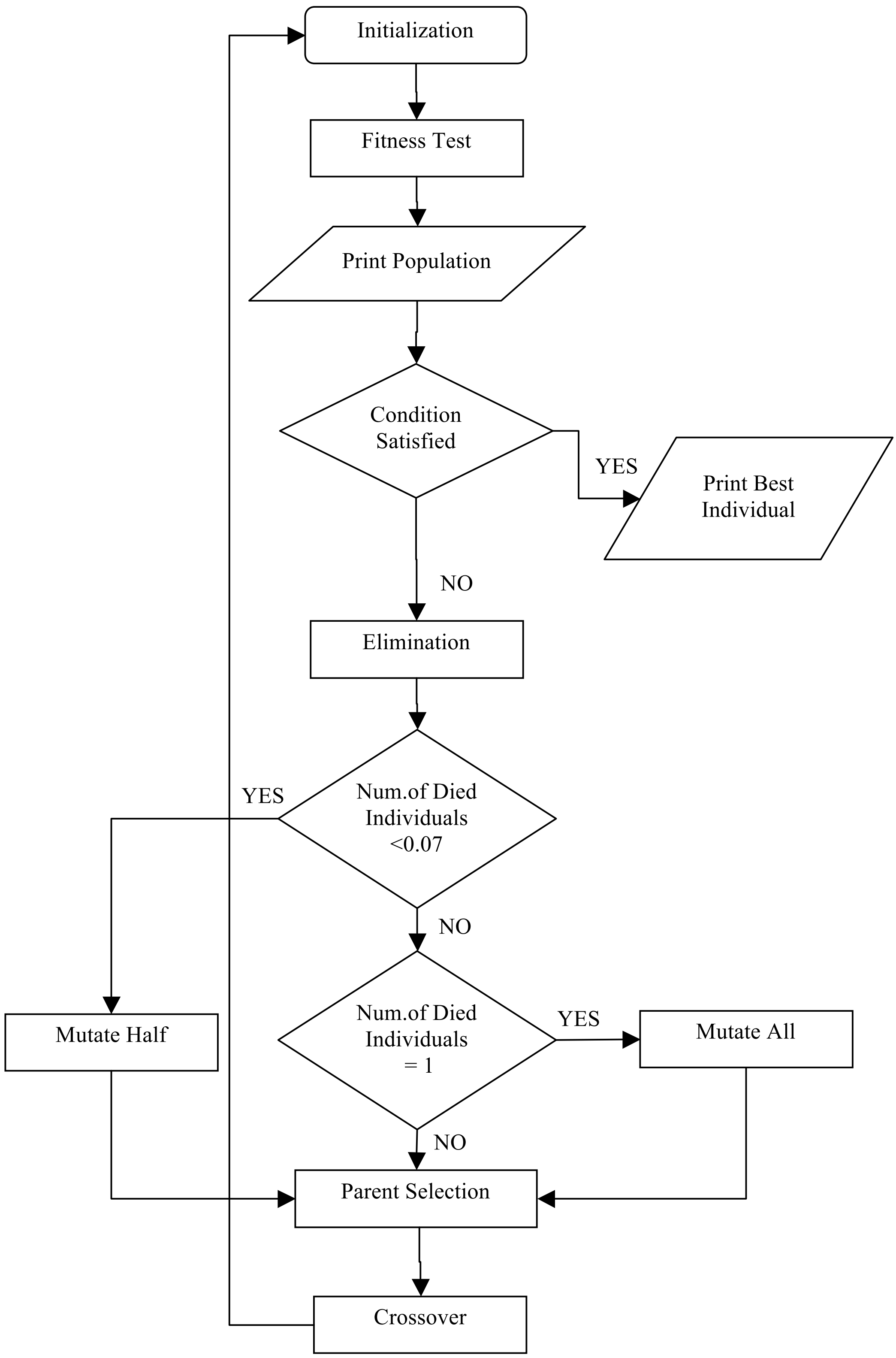


Figure 7.2 - Flow diagram of genetic algorithm process

7.3.2.1 Initialization

Design space is searched randomly and it is expected to catch design regions including possible optimum solutions in the space. This process is performed by taking the geometrical design constraints into account. Most valuable individuals are selected and an initial population is created. The number of individuals in population is determined according to choice of the designer. Initial population includes the starting information required by the genetic algorithm. New generations evolve over the individuals which have higher fitness values in initial population. Because of this tendency of optimization, foresight of the designer is required to neglect the individuals which may not evolve to appropriate design shapes although they have higher fitness values.

7.3.2.2 Fitness Test and Termination Criterion

Previously mentioned penalty function designates the worth of the individual. However, since the object of the penalty function is modifiable, the fitness value of individual is evaluated over 100 points to get clear of units. For this study the fitness function can be written as,

$$f(X) = \frac{m^{aimed}}{\phi(X)} \cdot 100$$

$f(X)$ represents the fitness function, m^{aimed} is aimed mass and $\phi(X)$ is the penalty function as previously mentioned.

Termination condition is dependent on the choice of design engineer directly. In here, desired mass value is prescribed so optimization process quits when the fitness value reaches 100 points.

7.3.2.3 Elimination

Survival criterion for the individual is having a higher fitness value than the mean value of the fitness values of the individuals. Otherwise, individual with low fitness is killed.

7.3.2.4 Selection

It is aimed to create a pool of parents, which are going to breed to produce better individuals. Both elitist and random selection methods are used in this phase. Half of the selected parents are coming from the elitist selection such that they are the fittest individuals of the population. Although elitist selection ensures the population to converge quickly, it may converge to local optimums. Hence, the remaining parents are selected randomly. Rate of elitist selection and random selection might be modified depending on the way of optimization.

7.3.2.5 Crossover

Real number representation is used in crossover operation. Genomes of each selected parents are evaluated with a random number in 1 to 0 range, so a weighted mean is obtained for each genome of new individuals. Random number is set as being same for each couple of parents and same for each genome of individual but it is possible to define different random numbers for each genome in an individual.

7.3.2.6 Mutation

Occurrence of the mutation and the rate of mutation are independent on the parameters related with probability and the constraints of the mutation. In the mutation phase randomly selected genome changes its value to a random value in the range of predetermined limits. Mutation probability determines the chance of happening of mutation at any time as being situation independent. On the other hand, in case of killing less than %7 of the individuals, 50% of the population will be mutated. If all of the individuals are not able to survive and die than all of the population will be mutated.

7.4 Model Manager

Model manager is the section where the entire model related information is created. FE model is prepared and solved by this sub program. Also stress and mass data collection is controlled in here. The sub program consists of Msc. Softwares and other programs specifically prepared according to type of the model.

7.4.1 Msc. Marc Mentat

Msc. Marc Mentat is a platform that users can prepare FE models completely and manage postprocess activities such as generating stress contours, deformed shapes. The most important property of the program used in this study is that the program can read and verify procedure files, which are including sets of commands to realize a job.

7.4.2 Msc. Marc

Msc Marc is a powerful FE solver. The prepared FE models are solved by Msc. Marc. The solver is managed by the shell scripts.

7.4.3 Feasibility Check Module

Randomly selected parameters can not always constitute acceptable structural models. Because of this, a control module should act like a filter to weed out the unsuitable models. One can program a feasibility check module in respect of the needs of related structural design, abiding by creating the appropriate output files required by the other modules. In this study, mathematical model of the boom geometry in the computer code, OptiBOOM, prepared by Yener is rearranged as a feasibility checking module.

7.4.4 Model Creator & Analyzer

This part of the model manager includes mathematical description of the model geometry. The confirmed parameters are processed in here to find the curve and point coordinates in 3D space and procedure files that are containing special command sets for Msc. Marc Mentat are created separately for each model. Besides, this part of the program is responsible for model evaluation process, which includes running Msc Mentat to solve FE data files and managing data collection progress. The program uses shell commands to put the jobs in order. The diagram given in figure 7.3 represents the flow of the data evaluation process.

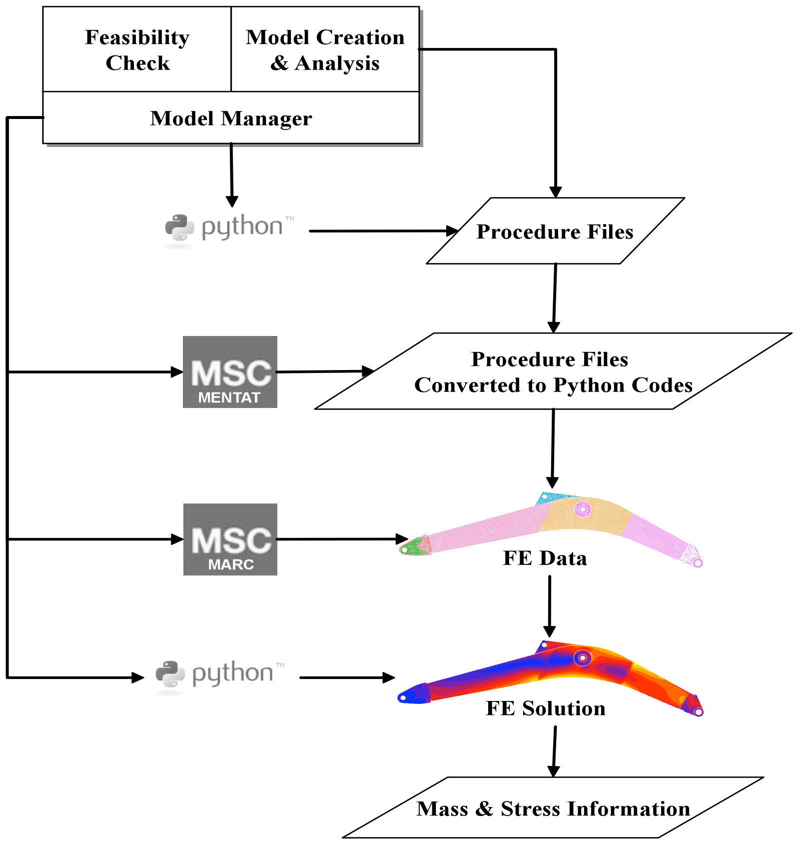


Figure 7.3 – Working diagram of model manager.

7.4.5 Data Collecting Module

Data collecting module is a Python based computer code that is able to communicate to FE result files generated by Msc. Marc directly. The code can manage every kind of data existing in FE result files. Msc. Marc offers large variety of FE results such as stress criteria, strain criteria, temperature, heat flux and so on. This study interests with Von Mises criterion, however type of FE result data can be changed by making a small modification in Python code. Another point that should be stressed is the code works independently from the type of the structure. The code concerns the regions in 3D space declared by the user. The figure 7.4 depicts how the code selects a previously defined region.

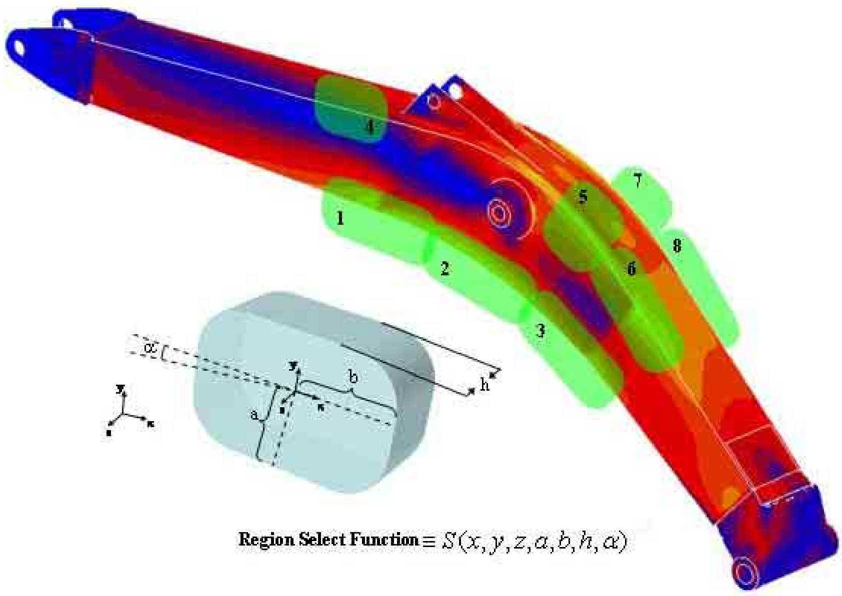


Figure 7.4 – Representation of region selection.

The code searches volumes, which have rounded rectangular profiles with depth in 3D space for each region, and selects nodes falling into search area. One can introduce preferred regions into Python code by using an AutoCAD Visual Basic macro easily.

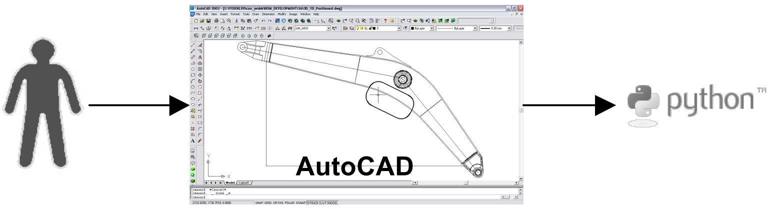


Figure 7.5 – Interrelationship among user, AutoCAD and Python data collecting module.

7.5 Smart Designer GUI

In fact, mentioned parts of the software so far can be controlled by running code parts separately or sending shell commands without requiring a graphical user interface. However, Smart Designer is not designed to use once, it is expected that software should be able to use for many different structural parts and cases on the contrary. Therefore, a GUI that is the only part of the software communicating to human is designed to manage all communication networks between subprograms.

7.5.1 Data Entry

One can load the required data for the optimization process by filling 5 sections through the GUI. These sections are called Genetic Algorithm Parameters, Model Parameters, Design Parameters, Design Boundaries and Model Selecion respectively.

7.5.1.1 Genetic Algorithm Parameters

Data related to genetic algorithm can be entered in here. These parameters are briefly explained below.

Initial Population Size: The number of the individuals existing in initial population. The software searches a number of individuals as designated in here and distinguishes the best ones.

Population Size: The number of individuals created for each generation is entered in here. As a reminding note, genetic algorithm guides the search according to information taken from the population. Therefore, declaring a small population size may fail because of inadequate information. For this study population size is determined as 30 in general.

Genome Length: Genome length designates the size of the array which contains the design parameters.

Max Fitness: One can set maximum fitness value by changing this cell so evaluation of each individual (model) is made over that designated number. In this study maximum fitness value is set to 100.

Notify Exceed: The software finishes the optimization process if the fitness value exceeds the user defined value.

7.5.1.2 Model Parameters

Structure may show diversities in shape according to different applications on the structure. Therefore variations of a structure can be defined in this section. For this study type of excavator boom and decision of using reinforcement material can be set.

7.5.1.3 Design Parameters

This section covers the parameters, which are used in design of the structure. Desired mass value, global and local stress constraints can be set for the boom design in here.

7.5.2 Optimization Initiation and Process Tracing

Designer starts optimization process by creating an initial population. The software randomly creates a set of models at number as designated in i*nitial population size* and reports the ratio of feasible models in initial population. If the user accepts the number of feasible models, program runs to create and evaluate accepted feasible models. After initial population creation is completed, best individuals are selected from the created initial population to create a new population and optimization becomes ready to start with this new population. At this stage, user can observe fitness values of the new individuals and also check starting geometries of the models. User may let the software to run optimization process or decide to re-randomize to obtain a new population.

One can easily trace optimization process easily through Smart Designer GUI. The program lets designer to follow the change of fitness values graphically so success of evolution process can be seen from the graph and designer can be aware of improprieties in optimization process quickly. Also mass, stress and fitness values of best models of each generation are updated on screen and user can observe each generated best model at any time by calling the generation number of it. Besides, user can examine Von Mises stress and principal stress distributions, and geometrical information of the structure in a 3D environment without requiring any other program. Also user can call Msc. Marc Mentat through the program to open models.

CHAPTER 8

CASE STUDIES

In this chapter, four optimum solutions of boom of HMK360 LC excavator will be presented. Initial shape of the boom is determined by considering the current designs of the competitors, using the geometrical requirements of users (i.e. outreach) and the manufacturing facilities available. Due to kinematic considerations, the locations of the pivot points are fixed.

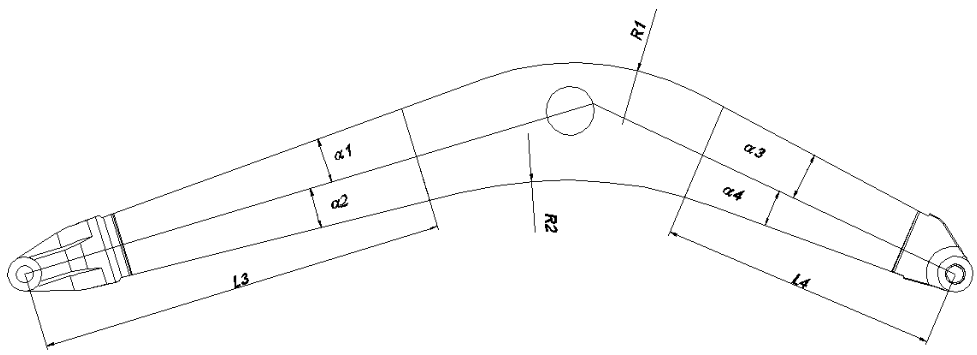


Figure 8.1 – Parametric view of initially designed excavator boom

It is expected to find an optimum solution of the design in the vicinity of the initial design parameter set.

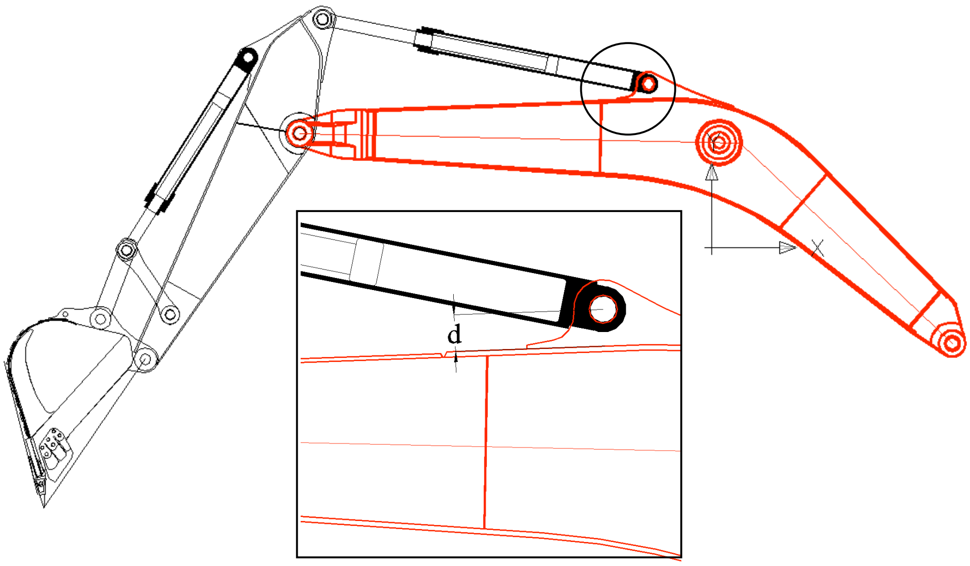


Figure 8.2 – Parametric view of initially designed excavator boom

Angle of upper front plate (α_1) should not exceed an upper value in order not to violate required d value shown in figure 8.2. Initial value of α_1 is used as upper limit for this parameter. Another constraint is limitation of angle between back lower and upper plates. Back plates should be placed at suitable connection angle to chassis mounting bracket otherwise weldment can not be performed effectively. The side constraint can be defined as,

$$\alpha_3 + \alpha_4 < \alpha_{bracket}$$

$\alpha_{bracket}$ is the maximum allowable angle to perform successful weldment of plates to the bracket. Figure 8.3 shows the welded regions on the bracket.

Angle of back lower plate (α_4) should be limited such that the boom should not prevent the rotation of upper chassis completely. In figure 8.4, required d distance between the lower plate and track, and also α_{4max} are depicted.

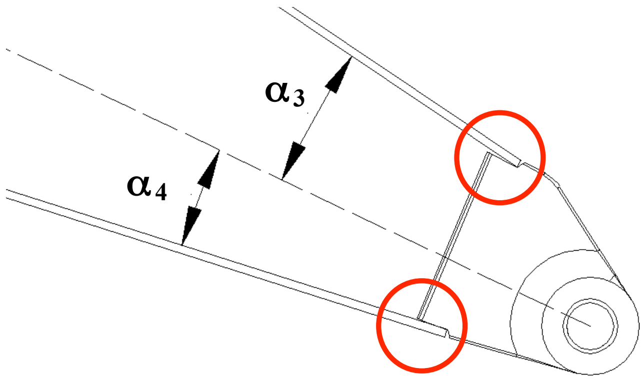


Figure 8.3 – Welded regions on the chassis mounting bracket.

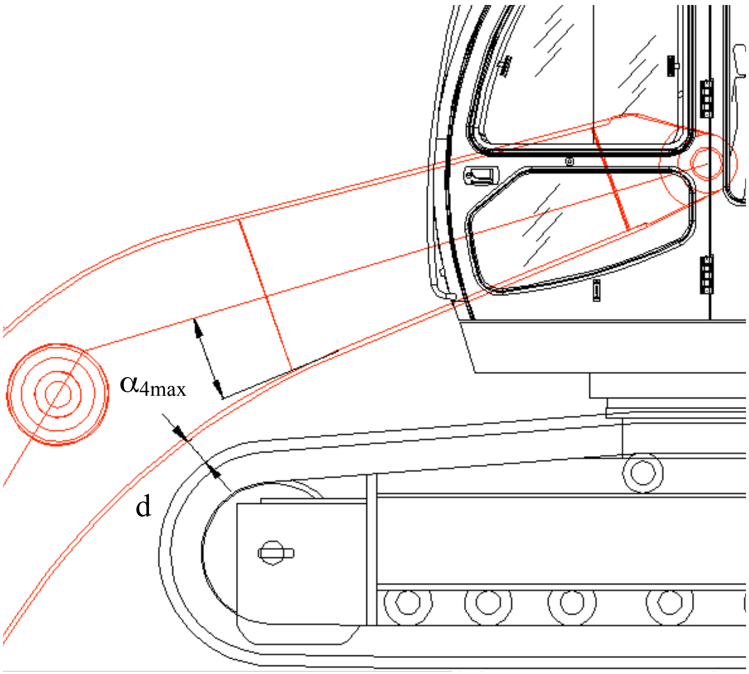


Figure 8.4 – Representation of d and α_{4max} parameters.

Other upper and lower limits of design parameters are set such that parameter set will evolve around the initial parameter set. The upper and lower limits for the design parameters can be defined as;

$$\alpha_{n\_\min} \le \alpha_n \le \alpha_{n\_\max} \ \textit{degrees}, \ n=1 \dots 4$$

$$R_{n\_\min} \le R_n \le R_{n\_\max} \ \textit{mm}, \ n=1,2$$

$$L_{n\_\min} \le L_n \le L_{n\_\max} \ \textit{mm}, \ n=1 \dots 4$$

The subscripts *n_min* and *n_max* designate maximum and minimum values for the angle (α), length (L) and radius (R) parameters respectively. While the angles, lengths and radiuses are continuous variables, the thickness parameters can only take discrete values.

$$t_n = 12, 14, 15, 16, 18, 20, 22 \ \textit{mm}, \ n=1 \dots 8$$

12 regions are selected from the boom as the Von Mises stress control regions (Figure 8.5). For each region required stress values ($\sigma_i^{required}$) are different and can be defined as;

$$\sigma_{lower} \le \sigma_i^{\max}(X) - \sigma_i^{required} \le \sigma_{upper}$$

where σ_{lower} and σ_{upper} represent 10% of the required stress ($\sigma_i^{required}$) so the calculated max stress ($\sigma_i^{\max}(X)$) value (From FE model) should be in range ±10% of the required stress value.

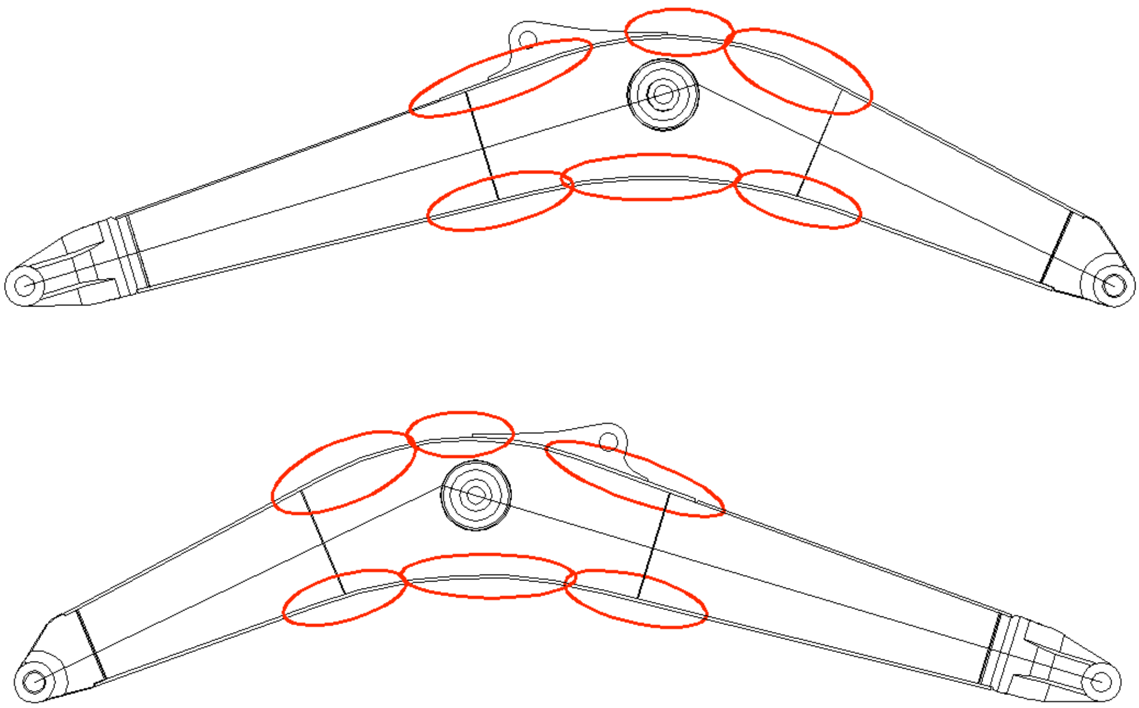


Figure 8.5 – Von Mises stress control regions.

Also maximum global stress ($\sigma_{global}^{\max}(X)$) is limited by a limit stress value $\sigma_{global}^{\lim}$ and this can be defined as;

$$\sigma_{global}^{\max}(X) \leq \sigma_{global}^{\lim}$$

In fact this is a constrained minimization problem and the aim of the optimization is minimizing the objective function which is equal to mass of the boom. The problem is converted to an unconstrained minimization problem and objective function (mass) is reformulated to take the constraints into account. In unconstrained objective function mass is penalized as much as the behaviour constraints are violated. Hence the unconstrained objective function is also called as penalty function. Genetic algorithm evaluates each boom model over the penalty function by comparing it with a predefined target mass value. The function which is used by Genetic Algorithm is called as fitness function and can be defined as;

$$f(X) = \frac{m^{req}}{\phi(X)} \cdot 100$$

f(x) and $\phi(X)$ are the fitness function and penalty function respectively. m^{req} represents the required target mass value and should be defined by the designer. Units of m^{req} and $\phi(X)$ are both kg and $\phi(X)$ is the penalized mass value of the boom design. Genetic algorithm compares this penalized mass value with required target mass value and grades each boom model over 100 by using this fitness function. Hence Genetic Algorithm can evaluate the goodness of each boom model.

The optimization software is capable of creating and solving one FE model at each 45 seconds and a single iteration of a population with 30 individuals approximately takes 20 minutes in a computer with Intel Core 2 Duo T7200 - 2.0 GHz 667 MHz and 4gb ram. In the first run 245 iterations and in the second run 230 iterations are performed. The first run takes approximately 3 days and 9.5 hours and the second run takes approximately 3 days and 5 hours.

Optimization algorithm starts by selecting 30 elite individual from among the initial population. The figure 8.3 shows the change of the fitness value versus generation number. Generation number represents the iteration number. After the 36th iteration, increase speed of the fitness value decreases dramatically as it can be seen from the figure 8.3. The mass versus generation number can be observed to choose appropriate boom designs. The figure 8.4 shows the change of the mass throughout the generations.

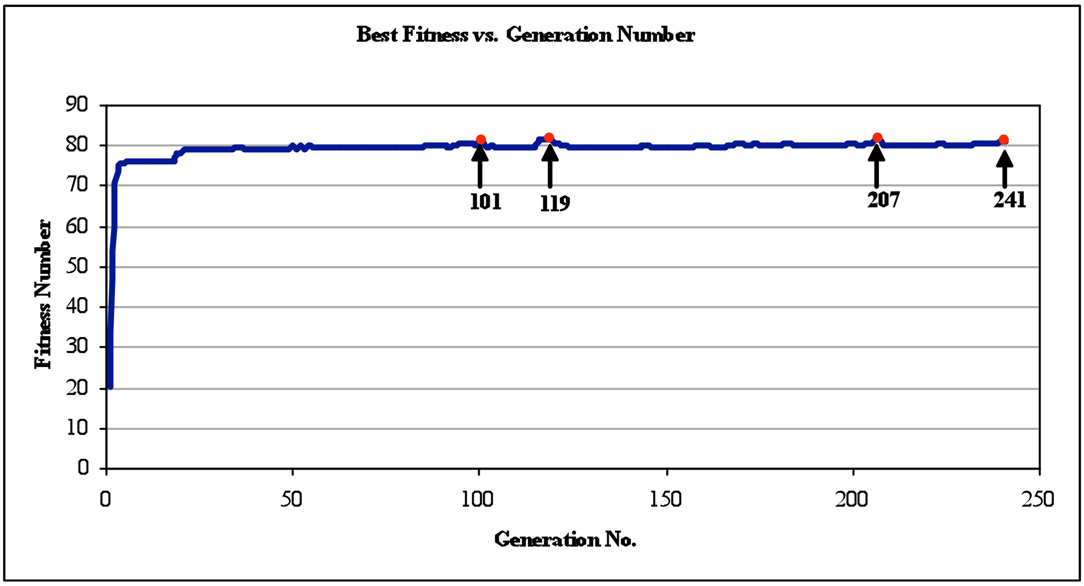


Figure 8.6 – Best fitness vs. generation number graph for optimization run 1.

Mass values of the best models of generations 101, 119, 207 and 241 are less than the other models. All four models do not violate maximum global stress value ($\sigma_{global}^{\lim}$) and have mass only 10% more than the required target mass (m^{req}). The second run continues by taking the best model of generation 101 as the initial model.

Side constraint intervals are narrowed and second optimization run is performed with refined side constraints. Since solution search is performed at a narrowed down search space, final solutions are very similar to each other. Hence, observing the change of mass throughout optimization run and accepting the lightest boom model as the solution is feasible. Model 170 has the highest fitness number among the other boom models. The model is 4.7% lighter than the initial model.

It is seen from figures 8.12 to 8.15 that Von Mises stress distribution over the plates is more homogeneous at optimized boom model. By this way it has been possible to decrease mass of the boom model

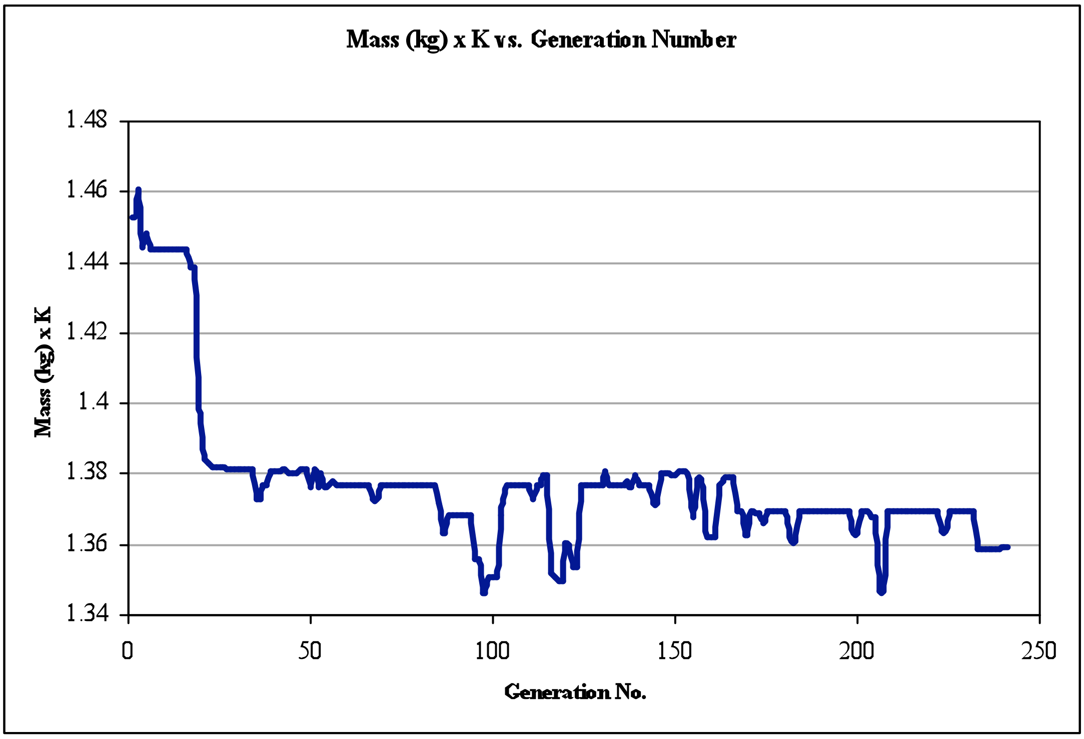


Figure 8.7 – Mass (kg) x K vs. generation number graph for optimization run 1 (K = scaling constant).

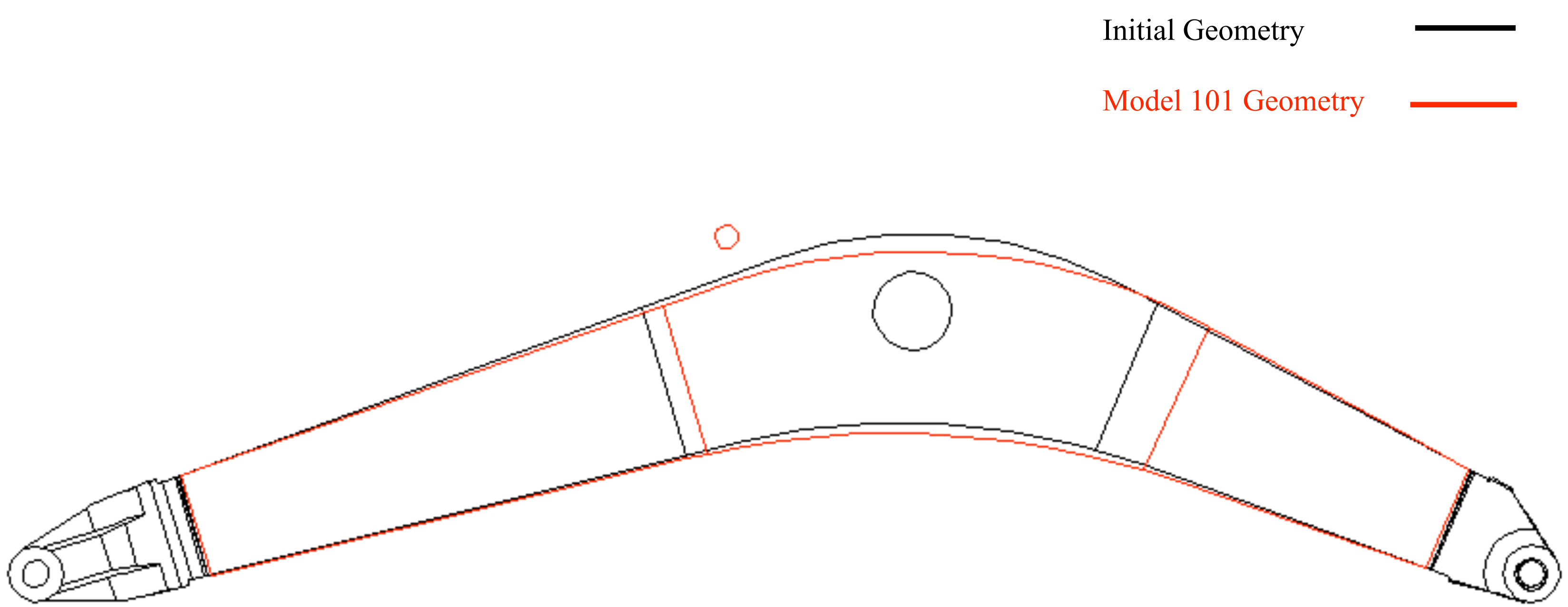


Figure 8.8 – Comparison of initial geometry and geometry of model 101.

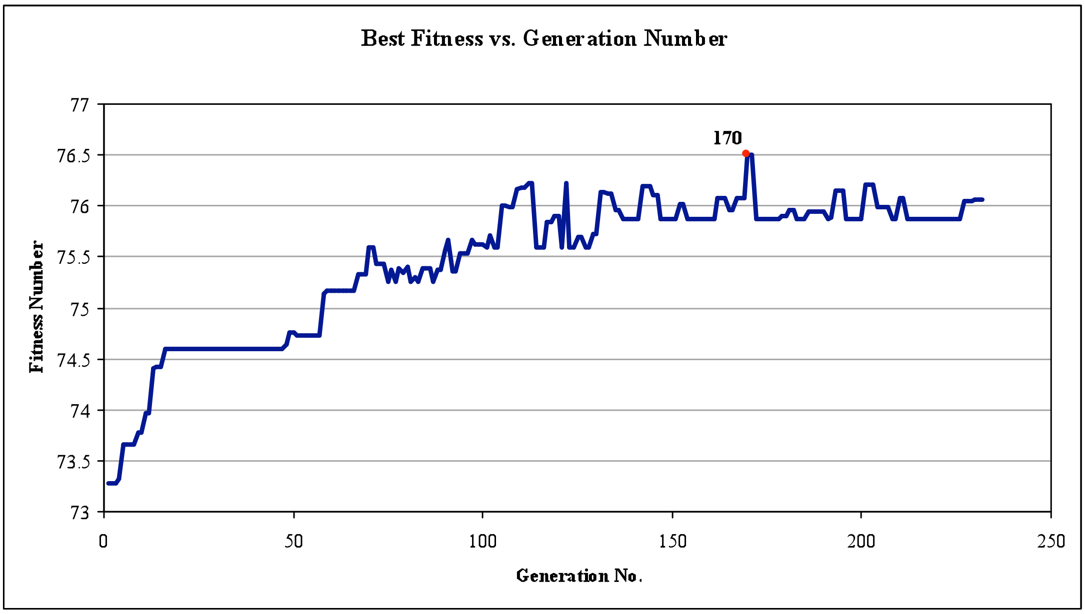


Figure 8.9 – Best fitness vs. generation number graph for optimization run 2.

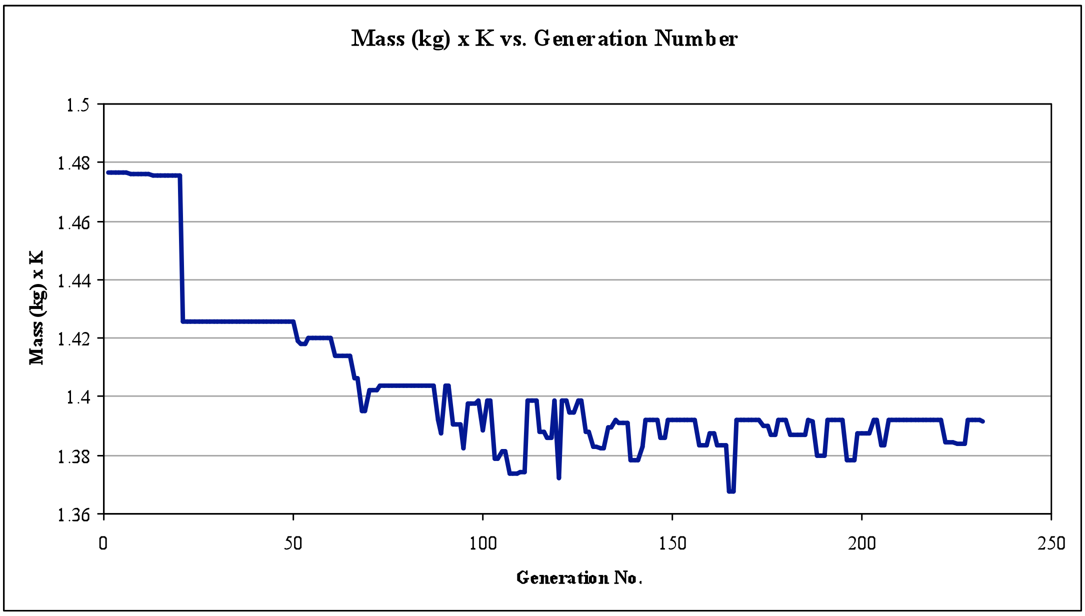


Figure 8.10 – Mass (kg) x K vs. generation number graph for optimization run 2 (K = scaling constant).

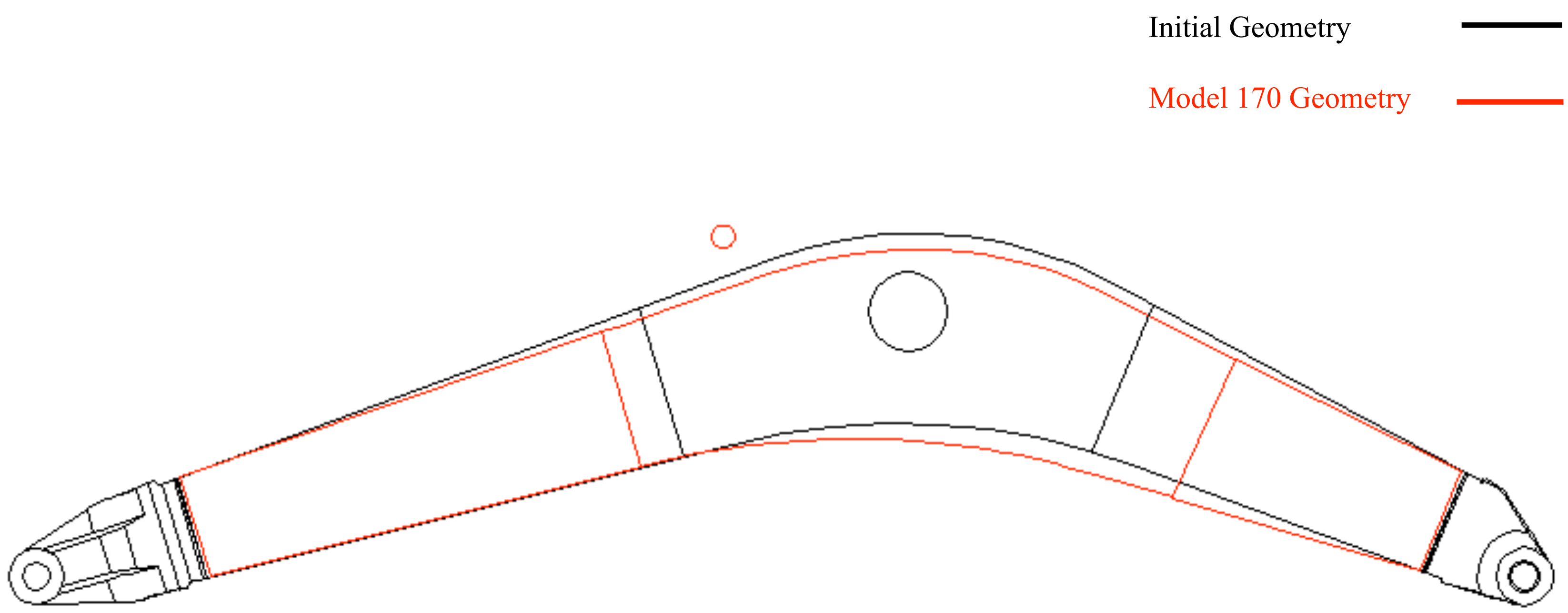


Figure 8.11 – Comparison of initial geometry and geometry of model 170.

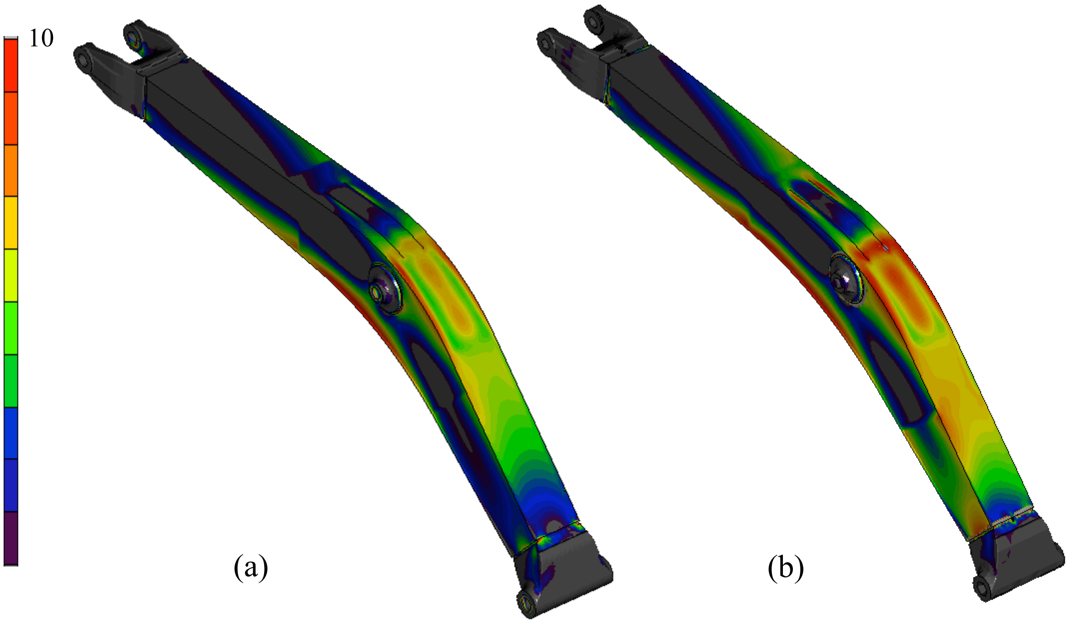


Figure 8.12 – Isometric view of the boom models representing stress distribution over the upper plates. (a) Initial model. (b) Optimized model.

10

(a)

(b)

0

Figure 8.13 – Isometric view of the boom models representing stress distribution over the lower plates. (a) Initial model. (b) Optimized model.

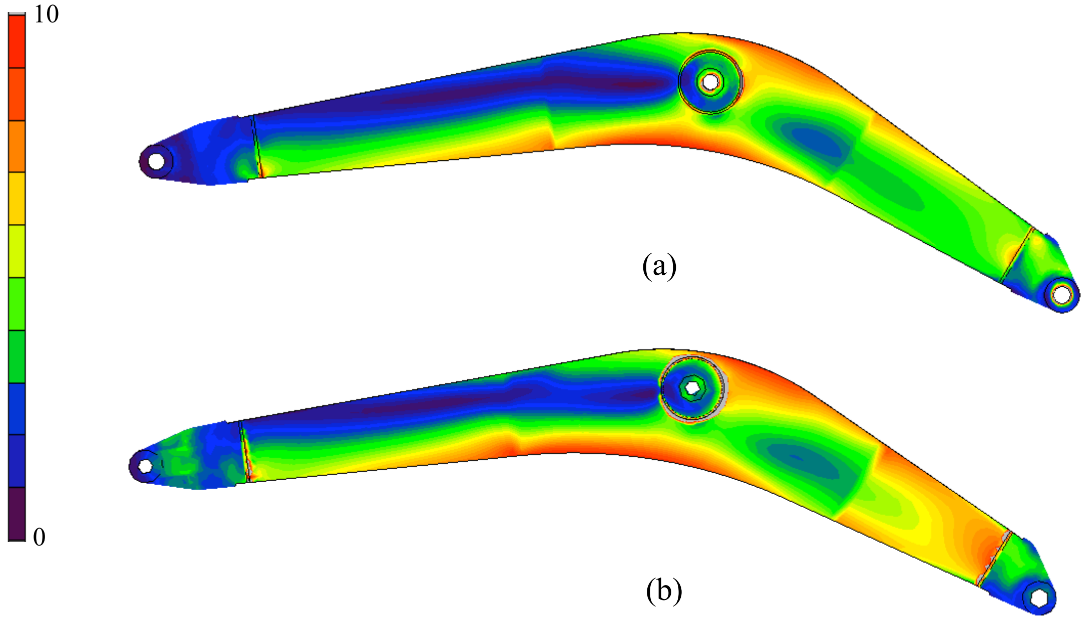


Figure 8.14 –Left side view of the boom models representing stress distribution over the left side plates. (a) Initial model. (b) Optimized model.

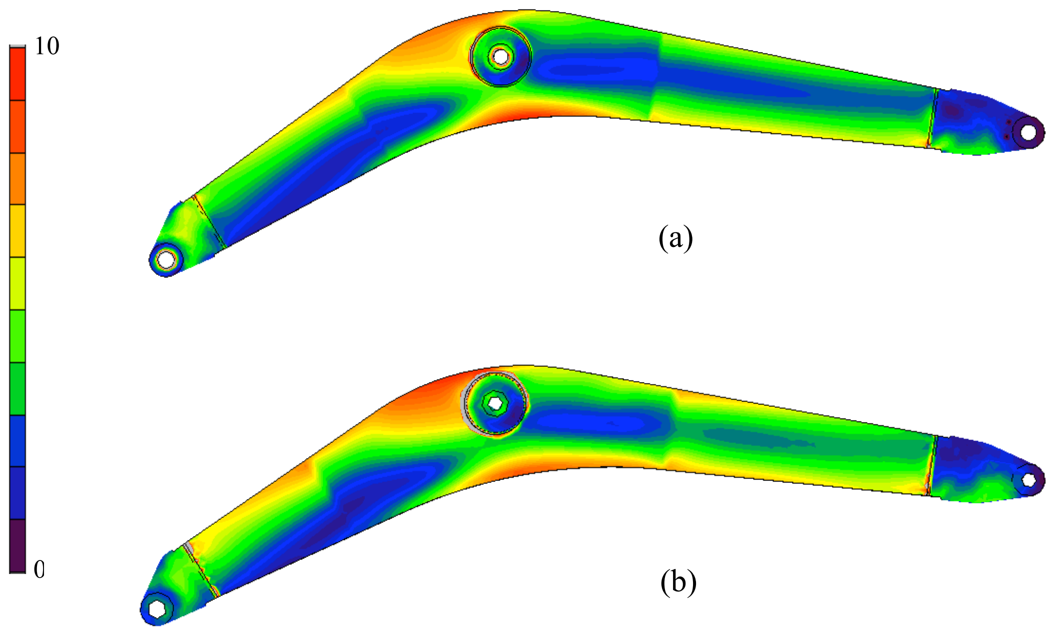


Figure 8.15 –Left side view of the boom models representing stress distribution over the left side plates. (a) Initial model. (b) Optimized model.

CHAPTER 9

DISCUSSION & CONCLUSION

In this study, shape optimization of excavator booms are performed automatically by using Genetic Algorithm method that is embedded in developed computer software.

Excavator booms should be light enough to minimize operating costs and the amount of steel used while keeping stress values under designated stress limits against fatigue failure.

Yener [33] has developed a computer program which is capable of creating finite element model of an excavator boom by using a set of design parameters. However, the program was human dependent and it was not possible to find optimum solutions for the boom design. Hence human factor is replaced with the developed software Smart Designer.

Genetic algorithm method is preferred in this study because Genetic Algorithm requires only payoff values associated with each model so it does not need for auxiliary information. Furthermore, instead of searching exact optimum solutions for complex systems such as excavator boom design, GA searches for better solutions and this makes GA, a humanlike optimization tool.

HMK 360 LC excavator boom model is optimized in this study. Initial design of the boom was 5% heavier than the final design and maximum stress was 10% higher than the Von Mises design stress criterion. Besides, it was observed that stress was concentrated locally over the boom model. After the optimization process more fully stressed boom model was found as the solution. Maximum stress was limited by predetermined global maximum stress value ($\sigma_{global}^{\lim}$) and weight is decreased 4.6% of the initial design. As a result, Smart Designer succeeded to find an optimum solution which is 4.6% lighter than the initial design and satisfying design stress allowance.

In this study optimization process has been completed totally in 6 hours and 14.5 hours. Actually obtained result is not the best one but it is one of the good results which is satisfying design criteria and aimed mass. Hence another designer may desire to continue optimization or finish optimization earlier. It is important to note that optimization always tends to converge for different results so designer may require running different optimization processes by starting with different initial model sets to obtain more solutions for the design.

Smart Designer is model independent optimization software. In other words, software is applicable for every parametrically definable structural model. By changing parametric modeller modules, one can use the software for many different applications.

This study presents a design methodology not only for an excavator boom, but also for other mechanical parts which are subjected to cycling loading. Discussed approaches are applicable for design of other mechanical parts. As mentioned before "Smart Designer" is model independent optimization software, so as future work digging attachments of excavator other than boom and digging attachments of other earth-moving machineries may be designed by using "Smart Designer" in a quick and easy way.

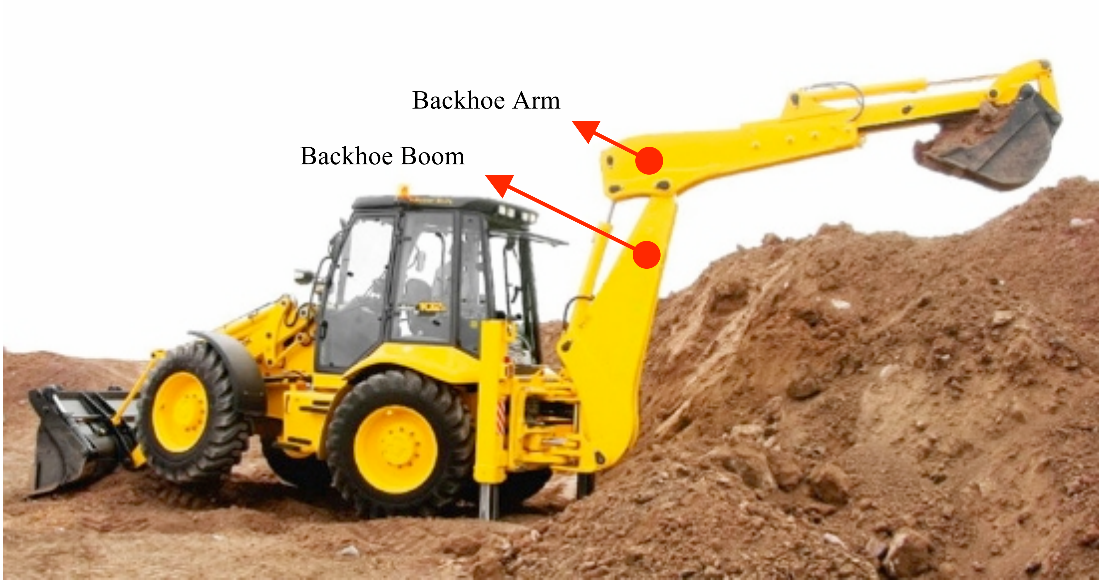


Figure 9.1 – Arm and boom of backhoe loaders are also able to be designed by using "Smart Designer".

In this study, welds haven't been modelled. One may modify parametric modeller for modelling the welds also. By this way, it becomes possible to observe geometric stress or notch stress so design stress allowance for fatigue design may be determined more accurately. Also fracture mechanics may be used in case of employing a parametric modeller which is capable of modelling 3D weld models with cracks.

REFERENCES

[1] Breierova Lucia, Mark Choudhari, An Introduction to Sensitivity Analysis, MIT System Dynamics in Education Project 1996.

[2] Carlgren Patrik, Lidström Per, MackAldener Magnus, Hol Dan, Optimization of Welded Component in Excavator Boom, Welded High-Strength Steel Structures 487-502.

[3] Chen Ting-Yu, Lin Chia-Yang, Determination of optimum design spaces for topology optimization, Finite Elements in Analysis and Design 36 (2000) 1-16.

[4] Das R., Jones R.,Peng, D., Optimisation of damage tolerant structures using a 3D biological algorithm, Engineering Failure Analysis 13 (2006) 362-379.

[5] Das R.,Jones R., Chandra S., Damage tolerance based shape design of a stringer cutout using evolutionary structural optimisation, Engineering Failure Analysis 14 (2006) 118-137.

[6] Das R.,Jones R., Design of structures for optimal static strength using ESO, Engineering Failure Analysis 12 (2005) 61-80.

[7] David E. Goldberg, Genetic Algorithms in Search, Optimization, and Machine Learning, Addison-Wesley Publishing Company Inc., USA, 1989.

[8] Eriksson Asa, Lignell Anna-Maria, Olsson Claes, Hans Spennare, Weld evaluation using FEM, Indutrilitteratur, Stockholm, 2003.

[9] Eriksson Asa, Lignell Anna-Maria, Olsson Claes, Hans Spennare, Weld evaluation using FEM, Indutrilitteratur, Stockholm, 2003.

[10] Ficici F., “Three Dimensional Fracture Analysis of Fillet Welds”, Ms. Thesis Submitted to Middle East Technical University, Turkey, 2007.

[11] Glover Fred, Tabu Search Fundamentals and Uses, US West Chair in Systems Science 1995.

[12] Haftka T. Raphael, Gürdal Zafer, Elements of Structural Optimization, Kluwer Academic Publishers, The Netherlands, 1991.

[13] Hardee Edwin, Chang Kuang-Hua, Tu Jian, Choi Kyung K., Grindeanu Iulian, Yu Xiaoming, A CAD-based design parameterization for shape optimization of elastic solids, Advances in Engineering Software 30 (1999) 185-199.

[14] Haslinger J., Makinen R.A.E, Introduction to Shape Optimization, SIAM, Society for Industrial and Applied Mathematics, Philadelphia, 2003.

[15] Hobbacher A.,Fatigue design of welded joints and components: Recommendations of IIW Joint Working Group XIII – XV, Abington, Cambridge, 1996.

[16] Jasbir S. Arora, Introduction to Optimum Design, McGraw Hill Book Co., Singapore, 1989.

[17] Jones R., Chaperon P., Heller M., Structural optimisation with fracture strength constraints, Engineering Fracture Mechanics 69 (2002) 1403-1423

[18] Karaboğa, Genetic Algorithms in Search, Optimization, and Machine Learning, Addison-Wesley Publishing Company Inc., USA, 1989.

[19] Karagoz T., “A Finite Elements Based Approach for Fracture Analysis of Welded Joints in Construction Machinery”, Ms. Thesis Submitted to Middle East Technical University, Turkey, 2007.

[20] Kim J. Jae, Young K. Heon, Shape design of an engine mount by a method of parameter optimization, Computers and Structures 65 (1997) 725-731.

[21] Korelc Jože, Kristanič Niko, Evaluation of design velocity field by direct differentiation of symbolically parametrized mesh, VIII International Conference on Computational Plasticity 2005.

[22] Lee S. Hoon, Byung M. Kwak, A Versatile Structural Optimization System Based on the Taguchi Method, Korea Advanced Institute of Science and Technology (2002)

[23] Li Qing, Steven P. Grant, Osvaldo M. Querin, Xie Y. M., Evolutionary shape optimization for stress minimization, Mechanics Research Communications, Vol. 26 No. 6 1999 657-664.

[24] Mattheck C., Teacher tree: The evolution of notch shape optimization from complex to simple, Engineering Fracture Mechanics 73 (2006) 1732–1742.

[25] P. Chaperon, R. Jones, M. Heller, S. Pitt and F. Rose, A methodology for structural optimization with damage tolerance constraints, Engineering Failure Analysis 7 (2000) 281-300.

[26] Rasheed Khaled,Hirsh Haym, Gelsey Andrew, A genetic algorithm for continious design space search, Artificial Intelligence in Engineering 11 (1997) 295-305.

[27] Richard L. Fox, Optimization Method for Engineering Design, Addison – Wesley Publish,ng Company, Philippines, 1989.

[28] SAE J1179, “Hydraulic Excavator and Backhoe Digging Forces”, SAE Handbook, Issued 1977-03, Reaffirmed 2002-12.

[29] Sasaki Hidetoshi, Tanaka Toshio, Itoh Tatsushi, Masumoto Nobuyoshi, Method for making a boom of an excavator, Patent number: 6637111, Filing date: Jul 8, 2002, Issue date: Oct 28, 2003, : Komatsu Ltd.

[30] Tekkaya A. Erman, Güneri Alper, Shape optimization with the biological growth method: a parameter study, Engineering Computations Vol. 13 No. 8 1996 4-18.

[31] Ugail H., Wilson M. J., Efficient shape parametrisation for automatic design optimisation using a partial differential equation formulation, Computers and Structures 81 (2003) 2601–2609.

[32] Wang Jyhwen, Design optimization of rigid metal containers, Finite Elements in Analysis and Design 37 (2001) 273-286.

[33] Yener Mehmet, Fıçıcı Ferhan, Söylemez Eres, Genetik Algoritma ve Msc. Marc Mentat Kullanımı ile Ekskavatör Bomunun Yapısal Optimizasyonu, Msc. Software User’s Conference 2004.

[34] Yener Mehmet, Fıçıcı Ferhan, Söylemez Eres, İş Makinaları Konstruksiyonlarının Tasarımında Bilgisayar Ortamında Sonlu Eleman Analizi Metoduyla Mukavemet Analizlerinin Yapılması, TMMOB Makina Mühendisleri Odası İş Makinaları Sempozyumu ve Sergisi 2003.

[35] Yener Mehmet., Design of a Computer Interface for Automatic Finite Element Analysis of an Excavator Boom, Ms. Thesis Submitted to Middle East Technical University, Turkey, 2005.