UNIVERSITY OF GAZİANTEP

GRADUATE SCHOOL OF

NATURAL & APPLIED SCIENCES

APPLYING TOPOLOGY OPTIMIZATION UNDER FATIGUE LOADING

M.Sc. THESIS
IN
MECHANICAL ENGINEERING

BY
İLHAN KORAY GEZER
DECEMBER 2011

Applying Topology Optimization under Fatigue Loading

M.Sc. Thesis
in
Mechanical Engineering
University of Gaziantep

Supervisor
Assist. Prof. Dr. M. Akif KÜTÜK

By
İlhan Koray Gezer
December 2011

T.C.
UNIVERSITY OF GAZIANTEP
GRADUATE SCHOOL OF
NATURAL & APPLIED SCIENCES
MECHANICAL ENGINEERING

Name of the thesis: Applying Topology Optimization under Fatigue Loading
Name of the student: İlhan Koray Gezer
Exam date: December 1st, 2011

Approval of the Graduate School of Natural and Applied Sciences

(Prof. Dr. Ramazan KOÇ)
Director

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

(Prof. Dr. L. Canan DÜLGER)
Head of Department

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

(Assist. Prof. Dr. M.Akif KÜTÜK)
Supervisor

Examining Committee Members	Signature
Prof. Dr. İ. Hüseyin FİLİZ	
Prof. Dr. L. Canan DÜLGER	
Prof. Dr. İbrahim H. GÜZELBEY	
Prof. Dr. Mustafa ÖZAKÇA	
Assist. Prof. Dr. M. Akif KÜTÜK	

ABSTRACT

Applying Topology Optimization under Fatigue Loading

GEZER, İlhan Koray
M.Sc. in Mechanical Engineering Department
Supervisor: Asst. Prof. Dr. M. Akif KÜTÜK
December 2011, 51 pages

In this study, an algorithm is developed for topology optimization under fatigue loading conditions. This algorithm is developed under the finite element analysis software ANSYS.

Topology optimization is used to obtain the best material distribution for getting the maximum natural frequency or minimum compliance while decreasing volume of a mechanical part to possible minimum value.

In fact machine parts are operated under the action of repeated or fluctuating stresses. Hence, the design must be under fatigue loadings for correctly simulating the real cases.

Number of studies containing topology optimization according to fatigue loading is limited in literature. Previously developed topology optimization algorithm for static loadings is adapted to fatigue loading in this thesis. The case studies in the literature are compared with results of this study also the statically loaded cases are compared with fatigue loaded cases.

Key words: topology optimization, element removal method, fatigue loading.

ÖZET

Yorulma Yükleri Altında Topoloji Optimizasyonun Uygulanması

GEZER İlhan Koray
Yüksek Lisans Tezi, Makine Mühendisliği Bölümü
Tez Yöneticisi: Y. Doç. Dr. M. Akif KÜTÜK

Aralık 2011, 51 sayfa

Bu çalışmada yorulma yükleri altında topoloji optimizasyonu yapabilen bir algoritma geliştirilmiştir. Algoritma sonlu elemanlar yöntemi kullanan ANSYS paket programında geliştirilmiştir.

Topoloji optimizasyonu; mekanik parçanın hacmini mümkün olan en az değere düşürürken, maksimum doğal frekans veya minimum uyum için en iyi malzeme dağılımını elde etmede kullanılır.

Gerçekte makine parçaları tekrarlı ya da dalgalanan gerilimler altında çalışmaktadır. Bu nedenle tasarımın gerçeğe uygun olarak simüle edilebilmesi için yorulma yükleri altında olması gerekir.

Literatürde, yorulma yükleri altında topoloji optimizasyonu içeren çalışma sayısı sınırlıdır. Statik yüklemeler için daha önce geliştirilen topoloji optimizasyonu bu tezde yorulma yükleri için adapte edilmiştir. Literatürde yer alan çalışmaların sonuçlarıyla bu çalışmada yer alan sonuçlar kıyaslanmıştır. Aynı zamanda sabit yüklemeli parçaların sonuçlarıyla dinamik yüklemeli parçaların sonuçları da kıyaslanmıştır.

Anahtar kelimeler: topoloji optimizasyonu, eleman silme yöntemi, yorulma.

ACKNOWLEDGEMENTS

I would like to express my deepest gratitude and appreciation to my supervisor Assist. Prof. Dr. M. Akif KÜTÜK who inspired and encouraged me in completing this study.

I am extremely grateful to Res. Assist. İbrahim GÖV who helped me with his suggestions throughout the whole study.

Special thanks for Res. Assist. Eyüp YETER for his valuable contributions.

Also special thanks for Güneydoğu Galvaniz General Manager Mr. Abdi KÜLTE who allows me to perform this thesis.

Finally, I would like to express my gratitudes to my family who continually supported me during the study.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS

b	=	fatigue variable
c	=	fatigue variable
k_a	=	the surface factor
k_b	=	the size factor
k_c	=	the reliability factor
k_d	=	the temperature factor
k_e	=	the modifying factor for stress concentration
k_f	=	the miscellaneous – effects factor
n	=	the safety factor
N	=	number of stress cycles
S_e	=	endurance strength
S_f	=	fatigue strength
S_y	=	yield strength
S_{ut}	=	ultimate tensile strength
S_e'	=	mean endurance limit of the rotating – beam specimen
σ_a	=	the stress amplitude
σ_m	=	the mean stress
σ_r	=	stress range
σ_s	=	steady or static stress
σ_{max}	=	the maximum stress
σ_{min}	=	the minimum stress

CHAPTER 1

INTRODUCTION

Nowadays for competition with other companies, prototypes must be designed and produced as fast as possible during the development period. For this reason, using computer based design is the obligatory way for the companies. Analyzing the designs or products on the computer eases the design process. So decreasing the troubles while increasing the quality means faster design.

Designing the optimum shape for minimum weight and determining the critical limit of the stress distribution caused by forces on the model is the most important fact for design on engineering. Therefore, structural optimization is developed for using computer aided design. By using this method, optimal design can be achieved. So designer can think details onwards. By using structural optimization, designing and improving time can be decreased.

Structural optimization is divided into three main groups: size, shape and topology optimization. *Size optimization* is used for determining optimum values of sizes. By using this sizes width, thickness and length of product is changed to get optimum sizes. Structure must be modeled before optimization process. The surrounding and/or inner holes optimization is called as *shape optimization*. Structural topology must be defined before the shape optimization process. After defining of structure, boundary contours are defined by contour parameters such as boundary nodes, lines or planes [1].

Structure must be identified before optimization on shape and size optimization methods. But establishing the design space and boundaries are enough on topology optimization method. These factors are requirements for acceptability of topology method. This method is the most available one to establish smooth structures.

The aim of topology optimization is decreasing the volume of part to find best distribution of elements for getting the maximum natural frequency or minimum compliance. Design space is determining the distribution of materials for the minimum volume for taking the possible maximum rigidity at new volume. Topology optimization can conceptually design prototype, which wasn't determined by initial shape. So, optimization can find the optimal topology using topology optimization in the beginning of a design and the optimal shape or size using shape or size optimization in the second [1].

The machine parts which are going to be optimized are generally failed under dynamic loads. Thus, instead of static conditions; parts are considered to be designed for dynamic conditions for realistic cases.

Necessity of being aware of dynamic conditions during design and accepted importance and advantage of topology optimization during design stage are considered in this thesis together. Hence concept of the thesis was assigned to contain both concepts. Previously developed topology optimization algorithm [1] for statically loaded parts is developed for application to fatigue loaded parts in present thesis.

In Chapter 2, the literature studies about topology optimization, fatigue and topology optimization under fatigue loading are summarized.

General principles of topology optimization, fatigue, topology optimization under fatigue loading and effect of element number on optimization are given in Chapter 3.

Comparison of case studies with literature and the numerical examples are given in Chapter 4. In the chapter, the outcomes compared geometrically as well as numerically.

In Chapter 5, general conclusions from the thesis and proposed future works are presented.

CHAPTER 2

LITERATURE SURVEY

2.1. Introduction

Machine parts are subjected to static and dynamic loads during operation. Most of the mechanical parts, such as gears, cams, shafts, springs, tools and dies are subjected to rapidly changing dynamic loads. These dynamic loads produce cyclic stresses on the part which may cause failure of the part. This type of failure is called as fatigue failure.

If the machine parts are made for more strength according to the fatigue conditions, the stress values of the parts may be decreased. Hence more resistive parts for fatigue may be possible. However, most of today's applications require a careful balance between the strength and weight of the material because of trying to produce goods with minimum material. At this point, necessity of topology optimization under the effect of fatigue loading arises. Decreasing volume of a mechanical part to determine the best material distribution for getting the maximum natural frequency or minimum compliance is called as topology optimization. [1]

In this chapter, available literature studies about the topology optimization are reviewed. After reviewing this content, studies about fatigue loading are given. At the last section, application of topology optimization under fatigue loading is summarized.

2.2. Literature studies about topology optimization

In this section the literature studies about topology optimization are summarized.

G. Steven, O. Querin, and M. Xie (1999) studied Evolutionary Structural Optimization (ESO) for combined topology and size optimization of discrete structures. It is shown, how the ESO concept can be applied to both pin-joined and rigid-joined discrete structures in 2D and 3D with single and multiple load cases. With multiple load cases it has been demonstrated that the evolutionary strategy of increasing the beam size if any of the load case stresses is over target, otherwise decreasing the size, is effective. However, *if bending is the only available load transmission mechanism then the ESO algorithm cannot improve the structure very much.* [2]

Tyler E. Bruns and Daniel A. Tortorelli (2003) have developed a method to systematically remove and reintroduce low density elements from and into the finite element mesh on which the structural topology optimization problem is defined. The material density field which defines the topology and the local 'stiffness' of the structure is optimally distributed via non-linear programming techniques. To prevent elements from having zero stiffness, an arbitrarily small lower bound on the material density is typically imposed to ensure that the global stiffness matrix does not become singular. While that approach works well for most minimum compliance problems, the presence of low density elements can cause computational problems, particularly in structures that exhibit geometric non-linearity's, e.g. in compliant mechanisms [3]

James K. Guest and Jean H. Prevost (2006) have studied on design of microstructures for maximized stiffness and fluid permeability. Periodic materials that are optimized for multiple properties and prescribed symmetries are used for topology optimization. In particular, mechanical stiffness and fluid transport are considered. Effective properties of the material are computed from finite element analyses of the base cell using numerical homogenization techniques. The elasticity and fluid flow inverse homogenization design problems are formulated and existing techniques for overcoming associated numerical instabilities and difficulties are discussed. Those modules are then combined and solved to maximize bulk modulus and permeability in periodic materials with cubic elastic and isotropic flow symmetries. [4]

Z. Kang, X. Wang and R. Wang (2008) have investigated the topology optimization of load-bearing structural components for reducing attitude control efforts of miniature space vehicles. It is pointed out that the attitude control efforts associated with cold gas micro thrusters are closely related to the mass moment inertia of the system that is based on the derivation of the cold gas consumption rate of three-axis stabilization actuators. Therefore, the need to restrict the mass moments of inertia of the structural components is highlighted in the design of the load-bearing structural components when the attitude control performance is concerned. The optimal layout design of the space vehicle structure considering attitude control effort is, thus, reformulated as a topology optimization problem for minimum compliance under constraints on mass moments of inertia. For the optimization problem, numerical techniques are discussed. For the case of a single constraint on the mass moment of inertia about a given axis, a design variable updating scheme based on the Karush–Kuhn–Tucker optimality criteria is used to solve the minimization problem. Mathematical programming approach is employed to seek the optimum constraints. [5]

İ. Göv (2009) has proposed Element Removal Method (ERM) method for topology optimization of 2D parts *under static loading*. Main idea of the proposed method is removal of low stressed material from a structure in an iterative process. In that study, for application of ERM an algorithm is developed in Ansys which is commercial FE program. Results of the developed algorithm are compared with some published results. The results of the developed algorithm are also compared with that of Ansys topology optimization tool. All comparisons yielded satisfying results. Developed algorithm overcomes the problem of topology optimization: long solution time of the method. Time necessary for optimization of parts with high element number is reduced up to 90 %. [1]

2.3. Literature studies about fatigue

Some studies consisting only fatigue loading but not topology optimization are summarized in this section.

H. Bayrakçeken, S. Taşgetiren and İ. Yavuz (2006) have studied on the power transmission system of vehicles that consisted several components which sometimes encounter unfortunate failures. They are focused on the fracture analysis of a universal joint yoke and a drive shaft of an automobile. For those parts, spectroscopic analyses, metallographic analyses and hardness measurements are made. Finite element method was used to analyse the stresses at the failed section. [6]

Seung K. Koh (2009) has studied on the fatigue analysis of an automotive steering link which is very critical for vehicle safety. For determining the local stress and strain distribution of the link, finite element method was used. Strain – gages are used to measure the experimental strains at the critical locations. Calculated local strains and the experimental strains are closely similar. [7]

D. Colombo, M. Gobbi, G. Mastinu, M. Pennati (2009) have analysed an unusual McPherson suspension. Experimental tests and numerical analyses are used for estimating the service life of the component. The fatigue life of the component has been assessed by a defect tolerant analysis. The result of the investigation is that the upper strut mount failure was due to an impulsive load that can not be justified by the static and dynamic loads acting on the component caused by road irregularities and vehicle manoeuvring during usual working conditions. [8]

E.S. Palma, C.L. Petracconi, S.E. Ferreira (2009) have analysed the fatigue behaviour of an automobile body part, according to the standards of performance. A rear trailer tow hook pin of a passenger automobile vehicle is used to perform the methodology of the experiments. Experiments were performed simulating the actual conditions in the customer environment. Strain gages, bonded on assembly critical points were used to measure stress and strain. Besides, stress analysis was also performed using a finite element program. Fatigue analysis is used to access and to compare the fatigue damage imposed during laboratory experiments. [9]

2.4. Literature studies on topology optimization under fatigue loading

The literature studies about topology optimization under fatigue loading are summarized in this section.

G. Florian, E. Weifried and P. Klaus (2003) have compared the optimization criteria stress and strain energy with life time values. The software TOSCA from FE-Design is used for optimization, the fatigue analysis is conducted with the fatigue software FEMFAT from Magna Steyr. By including FEMFAT in the optimization process automatically, it leads to interpretation of static and dynamic loads since fatigue analysis data is processed. The fatigue life prediction is implemented according to the local stress accept which also considers the local stress gradient. In the optimization loop damage values or safety factors are the inputs for TOSCA. The differences in stress based optimization and optimization based on the life time values pointed out on several specimens. [10]

P. Häuβler and A. Albers (2005) are presented a new method for an automated shape optimization of dynamically loaded components in mechanical systems. They are carried out the optimization by means of the results of a durability analysis based on finite elements. Load time histories, which are necessary for durability analyses, are derived from a multibody simulation. The whole optimization loop, which is an iterative procedure, incorporates all these gradual analysis steps and is implemented by the authors in a straightforward, batch-oriented manner using well-known standard software, Nastran. Since the whole process involves several different analysis types, such as multibody system simulation and durability analysis, the resulting setup is rather complex. Some essential aspects of each of the stages involved in the process are explained to provide the reader with the necessary background. [11]

A. Albers and P. Häuβler (2005) have studied a method for the automated optimization of dynamically loaded parts in mechanical systems. Basic aspects regarding the usage of multibody system simulation on the one hand and durability analysis on the other hand was shown. Multibody system (MBS) simulation allowed an adaptation of the loads acting on the part in each iteration. This can influence results of topology optimization significantly since the material distribution varies throughout the optimization process. Also feedback of modified inertia behaviour and stiffness to the overall system is covered by introduction of multibody simulation into the optimization process. In this article it was shown that that technique did not only work in case of shape optimization but also for topology optimization. [12]

M. Mrzygłód and Andrzej P. Zieliński (2006) have investigated multiaxial high-cycle fatigue criteria with respect to their application in structural optimization procedures coupled with finite element codes. As a result of tests carried out for several fatigue criteria, the Dang Van hypothesis was used for the detailed numerical study. It is suggested a way of respective adapting the high-cycle load history. The complete algorithm of the fatigue optimization was illustrated by applying the proposed procedures to vehicle parts which are subjected to high-cycle loadings. The finite element code ANSYS was used in the structural modelling. [13]

M. Mrzygłód and Andrzej P. Zieliński (2006) have investigated optimization of structures working in high – cycle load conditions. The development of a simple application optimization algorithm for such structures was the main object for the authors. The work was concentrated on three principle areas: fatigue of material (with special regard to multiaxial criteria of high – cycle fatigue), parametric optimization of structures, and application of the finite element method. The investigations and numerical implementation of several high – cycle criteria were made and the most convenient one for optimization was selected. The main process of fatigue optimization was preceded by the testing of methods of structural optimization and the preparing the tools for improving the efficiency of the optimization algorithm. This stage includes preparation of software tools based on evolutionary algorithms. In addition, the decision variables were preselected through an investigation of the sensitivity of the objective function on small increments of these variables. The work was illustrated by examples of optimization of mechanical structures working in high – cycle load conditions. [14]

M. Mrzygłód and Andrzej P. Zieliński (2007) have studied on parametric optimization of structures subjected to multiaxial high – cycle fatigue. A large number of load cycles corresponding to infinite fatigue life were taken into account in the analysis and optimization. The authors use simplified form of Dang Van’s criterion for fatigue damage estimation. For needs of optimization, a certain transformation pattern of load time history was assumed, which allowed to considerably accelerate the fatigue analysis process. The proposed methodology has been illustrated by an example of optimization of a car suspension arm. As observed in computational example, the proposed methodology of optimization allowed effectively reducing the mass of the studied structure while maintaining its fatigue durability on an established level. [15]

B. Desmorat and R. Desmorat (2007) have applied topology optimization to discuss an optimization problem of fatigue resistance. Optimizing the shape of a structure in cyclic plasticity combined with Lemaitre damage law is maximized the fatigue lifetime. The algorithm of the topology optimization is detailed and it is given a 3D numerical example. [16]

N. Kaya, İ. Karen and F. Öztürk (2010) have presented a framework for re-design of a failed automotive component subjected to cyclic loading. Topology and the shape optimization approach are used to re-design a failed clutch fork. Stress - life fatigue analysis was conducted to correlate the crack location between the failed component and the simulation model. A new design proposal was determined with the topology optimization approach, and then design optimization by response surface methodology was used to improve the new clutch fork design. [17]

2.5. Conclusions

Literature on topology optimization, the fatigue studies and the topology optimization methods under fatigue loading are reviewed in this chapter. Some problems about the application of those methods are noticed after reviewing these articles. Such as, generally solution times of the problems took 3-4 hours for simple cases. Some applications have very heavy mathematics so that is affecting the solution time. Some of those studies developed on specific constraints so that adding a new constraint is not applicable.

For these reasons; a useful, easy applicable and practical algorithm for topology optimization under fatigue loading is developed in this study. In order to ease the application of the method, a macro is written in Ansys. The basic logic behind this work may be summarized as: *simple mathematics*, *low solution time* and *easy constraint and force application*.

CHAPTER 3

TOPOLOGY OPTIMIZATION UNDER FATIGUE LOADING

3.1. Introduction

In this chapter, main idea of topology optimization and mostly used methods are presented first. Element removal method is used in this study hence basics of this method are presented next. After discussing topology optimization, basics of fatigue phenomena are summarised. At the end of the chapter, application of the element removal method under fatigue loading is presented.

3.2. Topology Optimization

Decreasing volume of a mechanical part by finding the best distribution of material in the design domain for maximum natural frequency or minimum compliance is called as topology optimization. Topology optimization generates optimal shape of a mechanical part for initially defined objectives of the designer. The structural initial shape is generated within a pre-defined design space. In addition, the user provides structural supports and loads prior to optimization. Without any further decisions and guidance of the user, the method forms the structural shape thus providing a *first idea* of an efficient geometry. Therefore, topology optimization is a much more flexible design tool than the classical structural shape optimization, where only a selected part of the boundary is varied. A given amount of structural mass is used to maximize a desired property of the structure, usually stiffness or the lowest eigen frequency.

3.2.1. Main Methods of Topology Optimization

Topology optimization has become popular and has been successfully applied into industrial design since 1988. Many methods have been developed to facilitate and make useful the topology optimization in last decades. Some methods that are commonly used for topology optimization are summarized.

In *material distribution method*, a finite element model is formed for initial shape. Material density or elastic modulus or element thickness are parameterized related to physical properties. Finite element solver then solves for strain and stress distribution. Local von Misses stress distribution is used as information to adjust the elements' thickness. After the adjustment, a new finite element model is created and the process is repeated. Optional trimming process may be performed at specified iterations. The optimum design is obtained when a fully-stress condition is achieved. The thickness adjustment scheme is described next. [18]

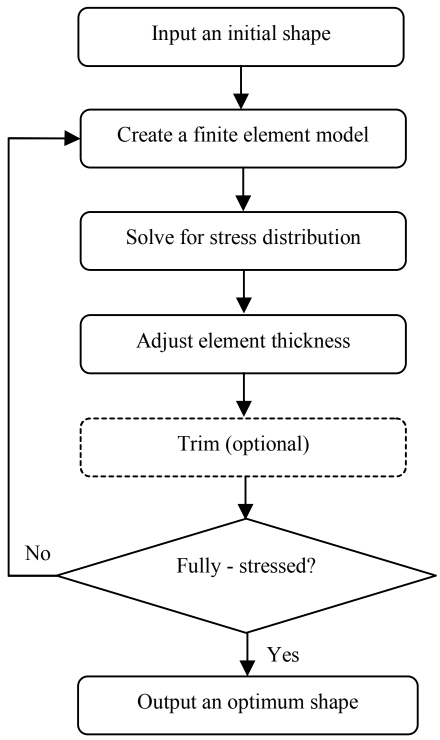


Figure 3.1 Procedure of Material Distribution Method [18]

In practice, a continuum structure is designed to carry the traction applied to the boundary of the structure subject to prescribed displacements imposed on its boundary in the *level set approach*. Design domains of practical structures are often limited and significantly affect the final optimal design of the structures. Structural boundaries under traction and prescribed displacements should be treated as a zero level set in the level set method. [19]

The main idea of the *homogenization method* is to replace the difficult "layout" problem of material distribution by a much easier "sizing" problem for the density and effective properties of a perforated material obtained by cutting small holes in the original homogeneous material. [20]

3.2.2. Element Removal Method (ERM)

Designer must increase numbers of iteration and element to find accurate solution of topology optimization. When numbers of iteration and element are increased solution time increases extremely. ERM is used to obtain better result in a shorter time. The method differs from other topology optimization methods: During the optimization process, low stressed elements are removed from design field. Hence, repeated use of low stressed elements is prevented. This process greatly decreases the solution time as described in the previous study [1]. Topology optimization can be applied to the design domain at design stage or can be applied to the machine element that is already in use. When the optimization of the part is started with the ERM, the following procedure is applied:

i. Design space has to be defined under applied defined loads and boundary conditions.

ii. Limiting values (minimum volume, mass reduction ratio, maximum stress) for optimization must be defined.

iii. Finite element solution and optimization loop is started. In this solution, aim is to find the elements which have comparatively low stress values.

iv. Sensitivity algorithm must be applied to find better element distribution (for preventing porous structure or material islands).

v. Low stressed elements are deleted from design domain and next domain is obtained.

vi. If initially defined limiting values are reached, the loop is ended.

vii. If not, loop continues with finite element analysis for new domain.

Algorithm of the optimization is presented below.

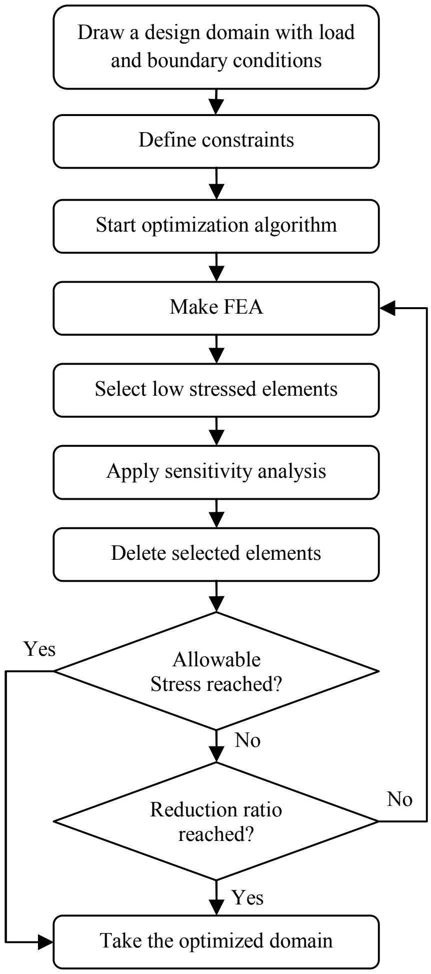


Figure 3.2 Application algorithm of the ERM

A topology optimization tool exists in Ansys. In Ansys, *optimality criteria* algorithm is used for topology optimization. Optimality criteria depend on minimization of compliance. Each element's density values are changed systematically from 0.001 to 1 in every iteration loop. Advantages of the ERM to Ansys are shown in Figure 3.3 and in Table 3.1. [1]

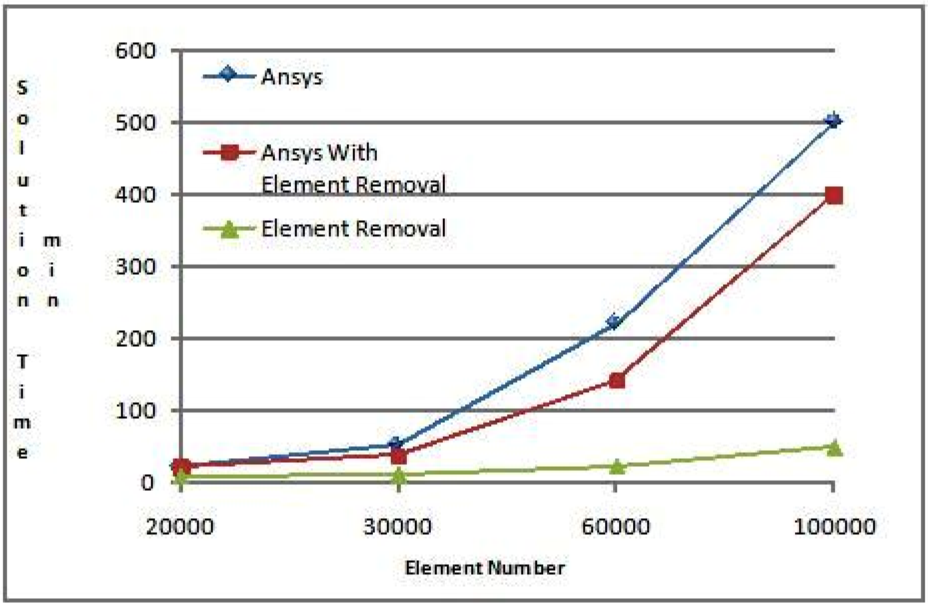


Figure 3.3 Solution times depending on the solution method and the element numbers [1]

Table 3.1 List of solution times depending on the solution method and the element numbers [1]

Methods / **Element Numbers**	**Ansys Opt. Tool**	**Ansys Opt. With Element Removal**	**Element Removal**	**Time Reduction**
20000	22 min.	21 min.	8 min.	63%
30000	52 min.	37 min.	10 min.	80%
60000	220 min.	142 min.	22 min.	90%
100000	500 min.	400 min.	50 min.	90%

Solution of the same problem yields up to 90% reduced time. It is also shown that the results of both methods are similar [1]. The element removal method is used after some modifications under the effect of fatigue loading.

3.3. Fatigue

Topology optimization is considered for static loads generally in the weight minimization problems. Optimization for fatigue load is novel and a few studies exist in this area. The design problem of finding the lightest and safe structure that achieves a given lifetime is solved using the following procedure: determine the lightest structure that can achieve a given lifetime.

Machine parts are generally failed under the action of repeated or fluctuating stresses so that a careful analysis is needed. The actual maximum stresses must be below the yield strength of the material. The distinctive characteristic of these failures is that the stresses are repeated for very large number of times. The name of the failure is *fatigue*. The fatigue failure is sudden and dangerous and it does not give much noticeable warning. It is easy to design according to static failure conditions however fatigue is difficult. It is a very complicated phenomenon.

3.3.1. The S – N Diagram

The specimens are subjected to repeated or varying forces of specified magnitudes for determining strength of the material under the action of the fatigue loads. At the same time cycles or stresses are counted to destruction. While a constant bending load is applied, the number of revolutions for failure is recorded.

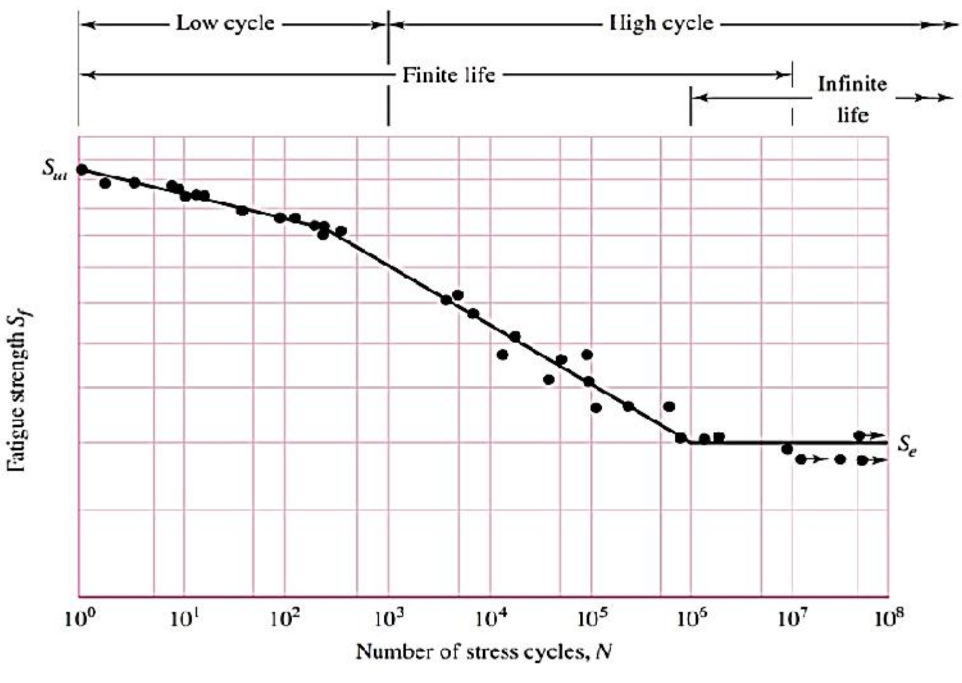


Figure 3.4 An S – N curve diagram plotted from the results of completely reversed axial fatigue tests. Material: UNS G41300 steel normalized; S_{ut}= 800 MPa, max. S_{ut}= 862 MPa [21]

Generally on fatigue failure from N = ½ to N = 1000 cycles is classified as low–cycle fatigue, as shown in Figure 3.4. High cycle fatigue is greater than 10^3 cycles but the region is not clear. It lies between 10^6 and 10^7 cycles for steel.

The mean endurance limit S_e of the rotating – beam specimen is estimated.

$S'_e = 0.5\, S_{ut}$ if $S_{ut} \leq 1400$ MPa

$S'_e = 700$ MPa if $S_{ut} > 1400$ MPa

For cast iron and cast steel;

$S'_e = 0.45\, S_{ut}$ if $S_{ut} \leq 600$ MPa

$S'_e = 275$ MPa if $S_{ut} > 600$ MPa

3.3.2. High – Cycle Fatigue

Using a log – log paper can help to write the equation of the S – N curve line as;

$$\log S_f = b \log N + c \tag{3.1}$$

In the Figure 3.4, it is obviously shown that the line intersects 10^6 cycles at S'_e and 10^3 cycles at 0.8 S_{ut}. If these values are substituted into the equation 3.1, the value of c and b can be found. The results are;

$$b = -\frac{1}{3} \log \frac{0.8\, S_{ut}}{S_e} \tag{3.2}$$

$$c = \log \frac{(0.8 S_{ut})^2}{S_e} \tag{3.3}$$

If N is given, S_f is found.

$$S_f = 10^c N^b \tag{3.4}$$

By the same way; if S_f is given and N is found.

$$N = 10^{-c/b} S_f^{-1/b} \tag{3.5}$$

It must be done a very careful study while determining the endurance limit of the rotating – beam specimen used in the laboratory. The tests are also quite closely controlled conditions. It is not exactly true that expect the endurance limit of the mechanical parts that are obtained in the laboratory.

The factors affecting the endurance limit are shown in Table 3.2.

Table 3.2 Conditions affecting the endurance limit [22]

Material: Chemical composition, basis of failure, variability
Manufacturing: Methods of manufacturing, heat treatment, fretting – corrosion,
Environment: Corrosion, temperature, stress state, relation times
Design: Size, shape, life, stress state, stress concentration, speed, fretting, galling

$$S_e = k_a k_b k_c k_d k_e k_f S_e' \quad (3.6)$$

Where

- S_e= the endurance limit of mechanical element
- S_e' = the endurance limit of rotating beam specimen
- k_a = *the surface factor* that is dependent upon the quality of the finish and the tensile strength,
- k_b = *the size factor* that is depending upon the dimensions, the shape and the method of loading,
- k_c = *the reliability factor* that is outlined an analytical approach for designing a mechanical part subjected to fatigue loads,
- k_d = *the temperature factor* affected the mechanical properties of the material,
- k_e = In mechanical parts the holes, grooves, notches or other kind of discontinuities affects the stress distribution. The effect is called as *the modifying factor for stress concentration*,
- k_f = *miscellaneous – effects factor* that is included all other effects that are made a reduction in the endurance limit.

3.3.3. Fluctuating Stresses

There are some types of stress situations in the loading of the mechanical parts. They are shown below in Figure 3.5.

σ_{min} = minimum stress

σ_{max} = maximum stress

σ_a = stress amplitude

σ_m = mean stress

σ_r = stress range

σ_s = steady or static stress

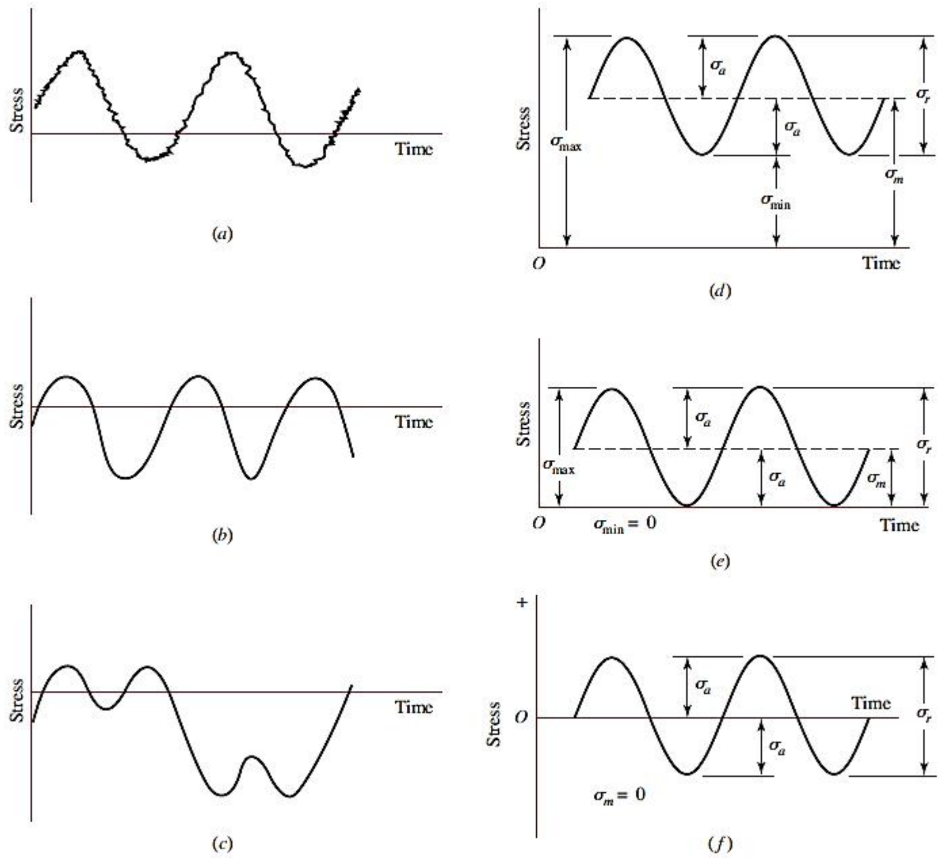


Figure 3.5 Some stress – time relations: (a) fluctuating stress with high – frequency ripple; (b) and (c) non-sinusoidal fluctuating stress; (d) sinusoidal fluctuating stress; (e) repeated stress; (f) completely reversed sinusoidal stress [21]

In this study the applied loads are changing from F to -F. These types of loads create completely reversed stresses similar to state shown in Figure 3.5(f).

3.3.4. Failure Theories

In fatigue problems there are some failure criterias. Some of them are the Gerber parabolic relation, the quadratic or the elliptic equation, Keçecioğlu equation, Chester equation, Dodget equation, Bağcı's equation, modified Goodman equation and Soderberg equation. These criterias are compared in Figure 3.6.

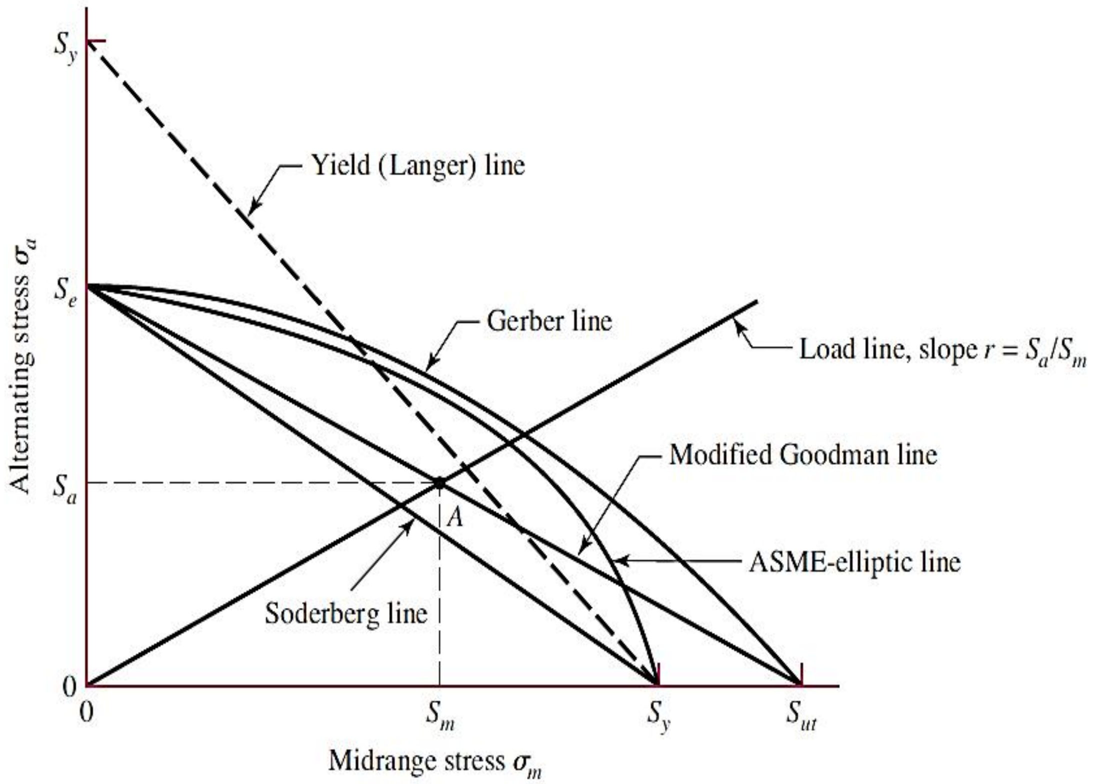


Figure 3.6 Fatigue diagram showing various criteria of failure [21]

The Soderberg failure theory is used in this study as it is the most conservative one among the failure theories. According to Soderberg failure theory, factory of safety is calculated as:

$$n = \frac{1}{\frac{S_a}{S_e} + \frac{S_m}{S_y}} \tag{3.7}$$

Steps of the design for fatigue strength are shown below:

i. First of all, it must be decided that life is infinite or finite.

ii. If life is chosen infinite, endurance limit, S_e must be calculated.
If life is chosen finite, then fatigue strength, S_f must be calculated.

iii. After these calculated values, it must be chosen which failure theory that is wanted to use.

iv. According to these failure theories, the factor of safety or life can be calculated.

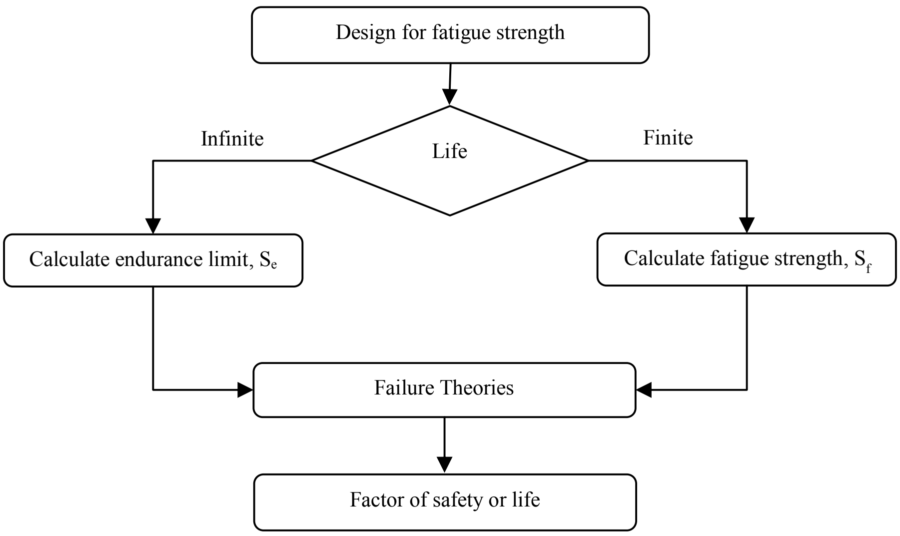


Figure 3.7 Algorithm of design for fatigue strength

3.4. Topology Optimization under Fatigue Loading

Previously developed element removal method [1], was applicable only for topology optimization of parts under static loadings. The method is modified considering fatigue type of loadings. Algorithm is modified to calculate the safety factor. Steps of the optimization under fatigue loading according to volume reduction and safety factor can be listed as:

i. User defines design space under applied loads and boundary conditions.

ii. User defines limiting values for volume reduction ratio and minimum safety factor for optimization.

iii. Optimization process is started.

iv. Finite element solution is done.

v. Each element's safety factors determined. (Soderberg fatigue failure theory is used)

vi. Sensitivity algorithm is applied to find better element distribution for preventing porous structure or material islands.Calculate selection coefficient for each element (set $k=0$ for the element el_i)

	el_{i-2}	
el_{i-1}	el_i	el_{i+1}
	el_{i+2}	

- If el_{i-2} is not void then $k_i = k_i +1$
- If el_{i-1} is not void then $k_i = k_i +1$
- If el_{i+1} is not void then $k_i = k_i +1$
- If el_{i+2} is not void then $k_i = k_i +1$

If $k_i < 4$ then delete $\mathbf{el_i}$ else update safety factor values by using k_i.

vii. Elements with high safety factors are deleted from design domain and next domain is obtained.

viii. Values of volume reduction and safety factor are compared with initially defined values. If one of the limiting values is reached, the loop is ended, otherwise loop continues with new finite element analysis.

Flowchart of the algorithm is given in Figure 3.8.

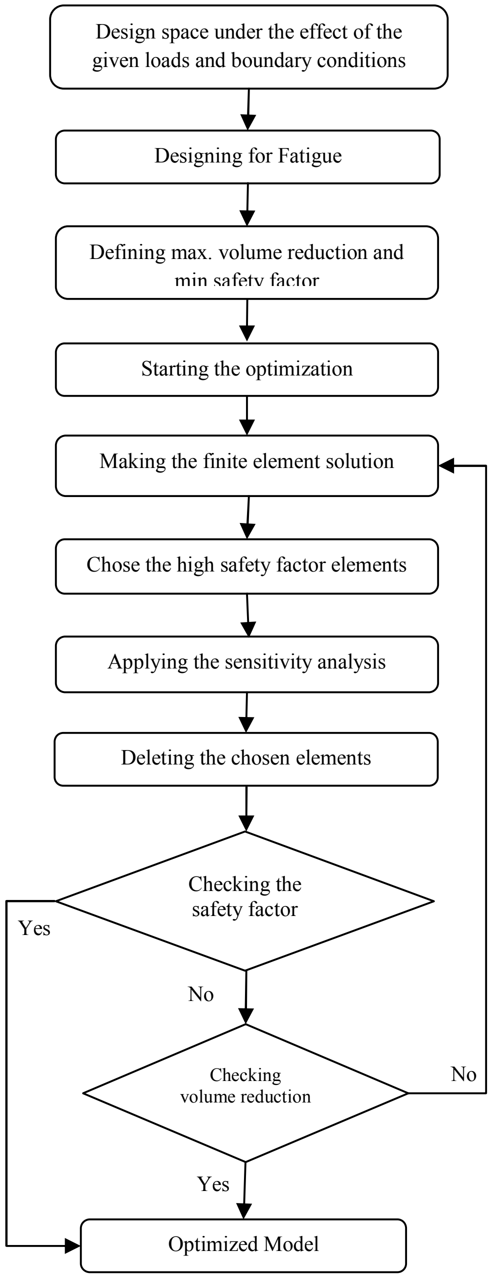


Figure 3.8 Flowchart for applying topology optimization under fatigue loading

Application of the developed algorithm is done on a cantilever beam of Figure 3.9. Application of each step of the algorithm is as follows:

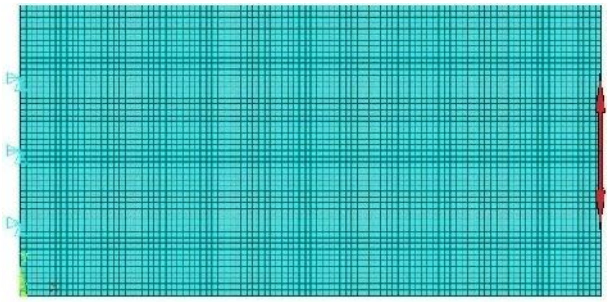


Figure 3.9 Design space under loads

i. Given design domain is modeled using 5000 elements under action of 400 kN downward load and 400 kN upward load. The ultimate strength of the material is S_{ut}=500 MPa and the yield strength of the material is S_y=270 MPa. Domain is fixed from its left side as shown in Figure 3.9.

ii. k factors which are affecting the fatigue strength must be calculated next. For this study all k factors are chosen as 1. For realistic cases these factors must be calculated.

iii. The third step is defining the limiting value of safety factor and volume reduction ratio. For the first case, volume reduction is taken as VR=70% and the factor of safety is taken as n=1.00. For the second case, volume reduction is taken as VR=70% and the factor of safety is taken as n=2.20.

iv. After inserting the required limiting values, the optimization is started.

v. The finite element solution is done. For each element, factor of safety (n) is calculated according to the given formula:

$$n = \frac{1}{\frac{S_a}{S_e} + \frac{S_m}{S_y}}$$

While calculating the endurance limit (S_e) of the given part, all of the k factors are taken as 1.

vi. Algorithm deletes 5% of the design domain at the initial step. The deleted elements are high safety factor elements.

vii. After this removing operation, sensitivity analysis must be applied to the domain before next removing operations. The sensitivity analysis is needed for preventing a porous material distribution and material islands. During application of the algorithm, safety factor of elements and neighborhood of each element are checked and then elements to be deleted are decided. In each loop 2% of initial volume is removed from the design domain.

viii. For the first case (VR=70% and n=1.00) it is reached to desired volume reduction ratio before it is reached to the limiting safety factor. The optimization result is shown as in Figure 3.10.

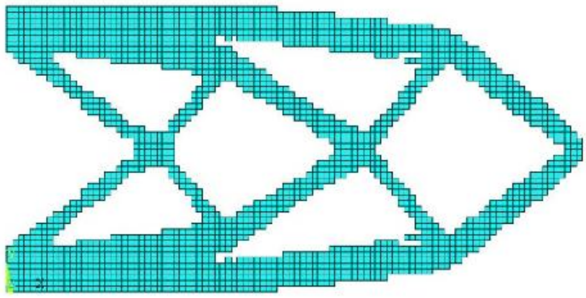


Figure 3.10 Optimized model VR=70% and n=1.00

ix. For the second case (VR=70% and n=2.20) the limiting value of safety factor is reached before the reach of volume reduction limit. The optimization result is shown as in Figure 3.11.

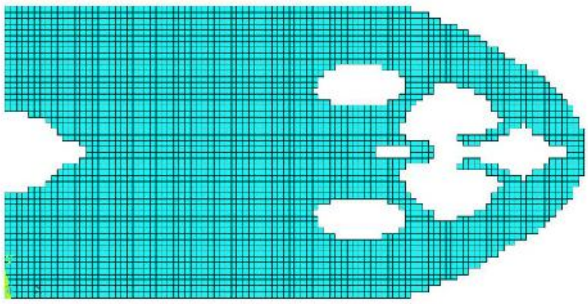


Figure 3.11 Optimized model VR=70% and n=2.20

The developed algorithm is used for two of these constraints which are the volume reduction ratio and the safety factor. Figures 3.10 and 3.11 indicate results for different conditions.

Result given in Figure 3.11 is obtained for safety factor value of 2.20. Verification of this result can be done by hand calculation: For Soderberg fatigue failure criteria factor of safety is calculated in the algorithm with the formula:

$$n = \frac{1}{\frac{S_a}{S_e} + \frac{S_m}{S_y}}$$

The yield strength of the material is S_y= 270 MPa

The ultimate tensile strength of the material is S_{ut}= 500 MPa

The endurance limit of the material is S_e = 0.5 x S_{ut}x k = 250 MPa

The mean strength of the material taken from FEA is S_m = 0 MPa

The alternating strength of the material taken from FEA is S_a = 121.85 MPa

$$n = \frac{1}{\frac{S_a}{S_e}} = \frac{250\ \text{MPa}\,.}{121.85\ \text{MPa}} = 2.05$$

Developed algorithm is checking the safety factor and volume reduction ratio after each finite element solution. Once reaching one of the limiting values, the loop is terminated and the solution is given as the final result of topology optimization. For the problem of Figure 3.11, as the limiting safety factor value (2.20) is reached (2.05) the algorithm is terminated and solution is given.

3.5. Effect of element number

While making the finite element solutions, the design domain is divided into small elements. Decreasing of the element size means that the design domain has more number of elements and more accurate model. But increasing the element number yields definitely increased solution time. Therefore the number of the elements in the design stage must be at an optimum level; high enough for accurate result, not too much for acceptable solution time.

Effect of element number used for modeling is searched on the problem given in Figure 3.9. The beam is modeled using 0.05m, 0.03m, 0.02m, 0.014m and 0.01m element edge length. For the given element edge length, the element number is 800, 2278, 5000, 10296 and 20000. Desired volume reduction is selected as VR = 50%. The selected factor of safety is considered as 1 because only effect of element number is searched now.

Use of 800 elements for the initial model, the optimization results as Fig.3.12.

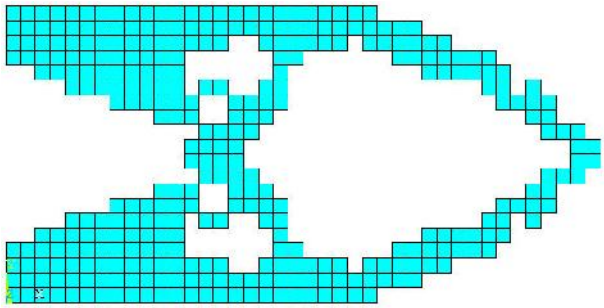


Figure 3.12 Optimization result of 800 elements design space

When the number of the element is changed to 2278, 5000, 10296 and 2000, the optimization yields the results given in Figures 3.13, 3.14, 3.15 and 3.16 respectively.

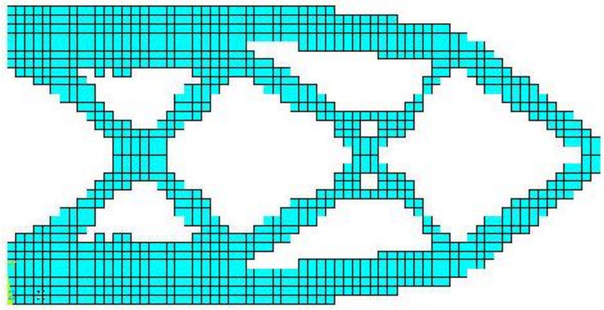


Figure 3.13 Optimization result of 2278 elements design space

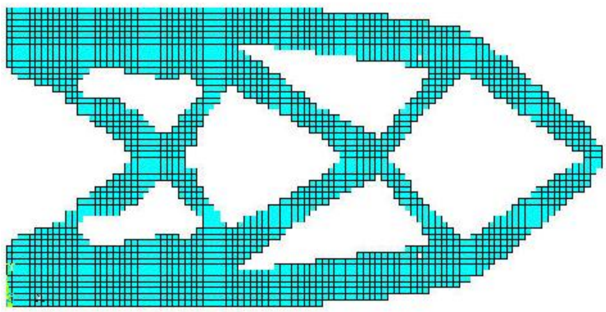


Figure 3.14 Optimization result of 5000 elements design space

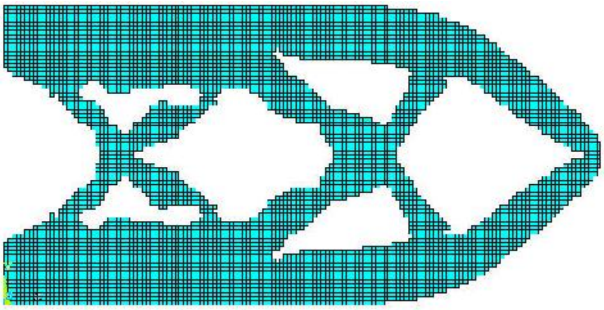


Figure 3.15 Optimization result of 10296 elements design space

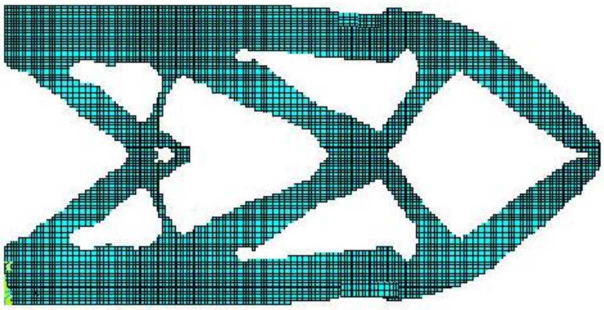


Figure 3.16 Optimization result of 20000 elements design space

The deformation values after optimization for these 5 examples are compared in Table 3.3.

Table 3.3 Variation of deformation with number of elements

Number of Elements	Element Edge Length (m)	Deformation ($mx10^{-6}$)
800	0.05	9.18
2278	0.03	4.26
5000	0.02	4.37
10296	0.014	4.49
20000	0.01	4.16

Use of different element numbers yielded similar results except for 800 elements. After a particular point, element number has no effect on optimization. According to the given deformation values, element edge length is chosen as 0.02m for this model. Choosing smaller element edge length than 0.02m gives similar results but the solution time is increased.

3.6. Conclusion

Application of the element removal method for topology optimization of the parts under static loading was given in the literature. Advantage of the method was proven with given numerical values. The algorithm is adapted for fatigue type of loading during present study.

Before adaptation of the algorithm, basics of the fatigue phenomena are studied. After this step the developed algorithm which is used for topology optimization of machine parts is given under fatigue type of loading.

In this study, algorithm is adapted to do topology optimization according to safety factor. The algorithm is applied on a cantilever beam example. The beam is optimized according to volume reduction and safety factor together. Application results are validated by hand calculation and it is proved that the algorithm is valid for safety factor.

When the optimization is applied to the design domains which have different element numbers, similar optimization results are obtained. The optimum number of element is decided as 5000 for the case study.

Numerical examples are used to compare outcomes of the study in the next chapter.

CHAPTER 4

CASE STUDIES

4.1. Introduction

Case studies are grouped under two headings. In the first part, some examples of the topology optimization under fatigue loading are compared with the static loading examples available in the literature. Only the geometrical results are compared as the numerical values of the solutions are not available.

In the second part, the comparison is made between the examples of the topology optimization under fatigue loading and the static loading. Iteration numbers, maximum deformations and stress values are discussed between the two solutions.

The analyses given in the related sections are done on a computer which has 2 Core 2.26 GHz CPU, 3 GB Ram and Windows 7 operating system.

The used data in the numerical examples are:

- Thickness of element, t = 0.1 m
- Modulus of elasticity, E = 200 GPa
- Poisson's ratio, ν = 0.3
- Ultimate tensile strength, S_{ut} = 500 MPa
- Yield strength, S_y = 270 MPa
- Factors for endurance strength, k = 1

4.2. Comparison of Methods Geometrically

Studies existing in literature are investigated to compare the outcomes of topology optimization procedure proposed in this thesis. However, there is not many fatigue loaded studies which can be compared with the present study. There are some statically loaded studies but the results of these studies are generally visual and there is not any given numerical data.

In this section optimization results which are found in the literature are compared geometrically with the result of the optimization developed in this thesis.

The load is applied to the two end fixed beam from its bottom – center as shown in Figure 4.1. The result of the designed domain according to the level set method is shown in Figure 4.2 and the result of the designed domain according to ERM is shown in Figure 4.3. The same beam is modelled for fatigue loading, and the reversed load is applied from the same point as shown in Figure 4.4. The optimization result under fatigue loading that is shown in Figure 4.5 is similiarly same with that of static loadings.

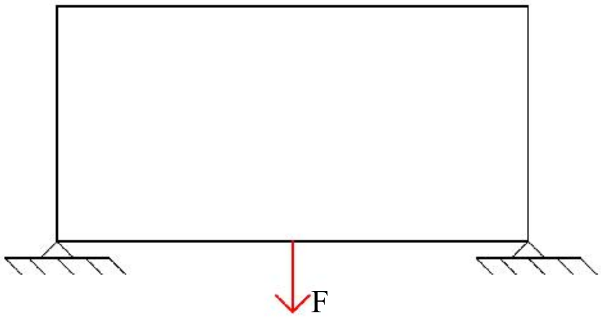


Figure 4.1 Bottom – centre loaded design domain under static loading [1, 23]

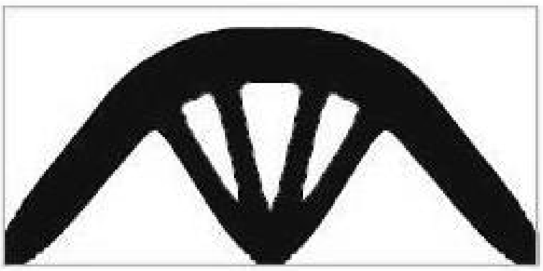


Figure 4.2 Optimization result of level set method [23]

Figure 4.3 Optimization result of ERM under static loading [1]

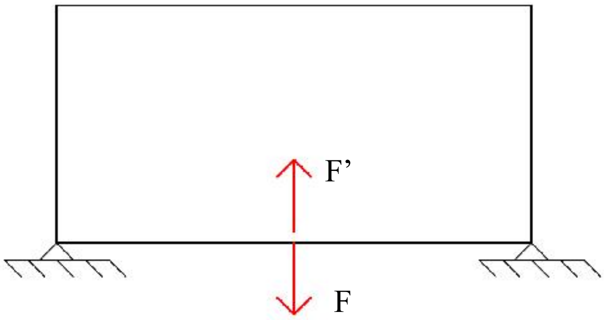


Figure 4.4 Bottom – centre loaded design domain under fatigue loading

Figure 4.5 Optimization result under fatigue loading for bottom – centre loaded design

At the second example, load is applied to the two end fixed beam from its top–centre as shown in Figure 4.6. The result of the designed domain according to Optimality Criteria Method (OCM) is shown in Figure 4.7 and the result of the designed domain according to ERM is shown in Figure 4.8.

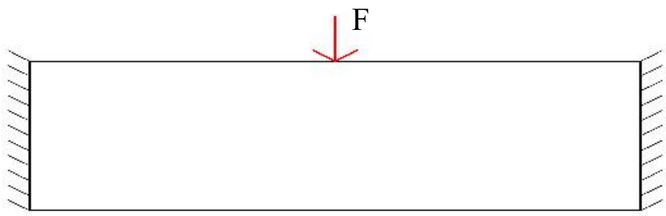


Figure 4.6 Top – centre loaded design domain under static loading [1, 24]

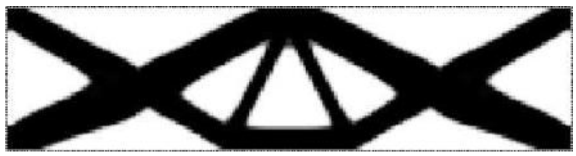


Figure 4.7 Optimization result of OCM [24]

Figure 4.8 Optimization result of top – centre loaded design by ERM under static loading [1]

The same beam is modelled for fatigue loading, and the repeated load is applied from the same point as shown in Figure 4.9. The optimization result under fatigue loading that is shown in Figure 4.10 is slightly different from the result of static loadings.

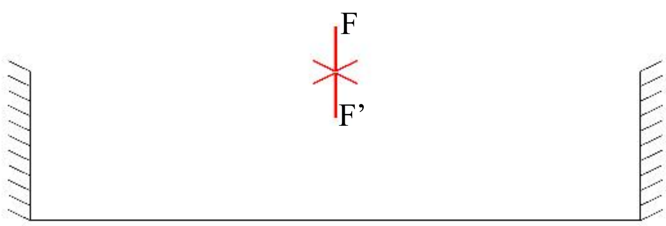


Figure 4.9 Top – centre loaded design domain under fatigue loading

Figure 4.10 Optimization result of top – centre loaded design under fatigue loading

The load is applied to the T - shaped beam from two ends as shown in Figure 4.11. The result of the designed domain according to the level set method is shown in Figure 4.12 and the result of the designed domain according to ERM is shown in Figure 4.13.

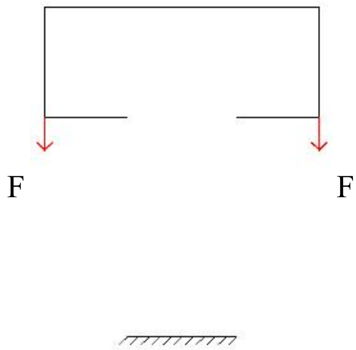


Figure 4.11 T – shaped design domain under static loading [1, 25]

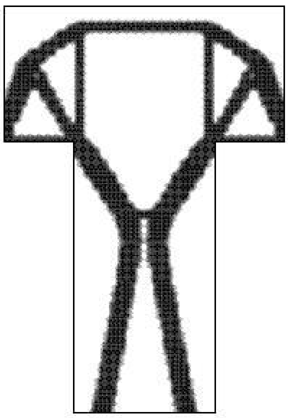


Figure 4.12 T – shaped optimization result of level set method [25]

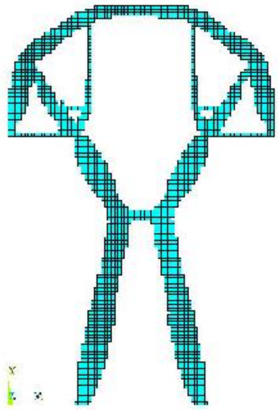


Figure 4.13 Optimization result of T–shaped design by ERM under static loading [1]

The same beam is modelled for fatigue loading, and the repeated load is applied from the same point as shown in Figure 4.14. The optimization result under fatigue loading shown in Figure 4.15. It is almost the same with the result of static loadings.

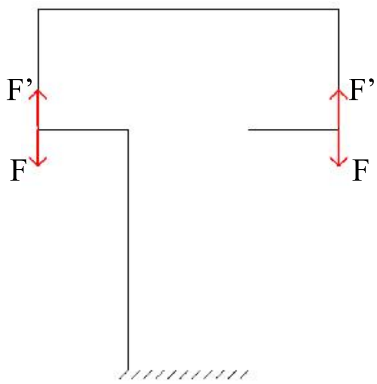


Figure 4.14 T – shaped design domain under fatigue loading

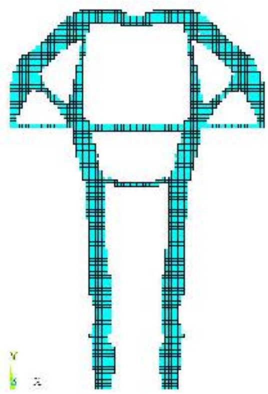


Figure 4.15 Optimization result of T – shaped design under fatigue loading

4.3. Numerical Examples

Examples of the topology optimization under fatigue loading are compared with the static loading results in this section.

The cantilever beam loaded from its centre as shown in Figure 4.16 is modelled for static loading. The size of the beam is 1m x 2m and the applied load is 1000 N to downward. The number of element is 5000 and the desired volume reduction is 60%.

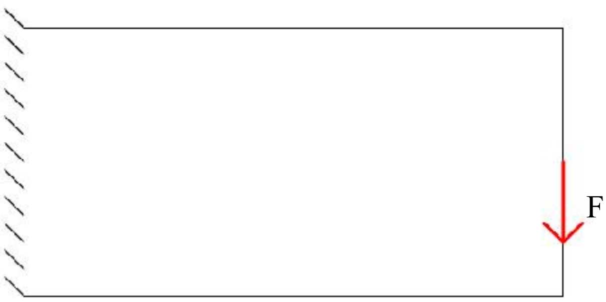


Figure 4.16 The cantilever beam under static loading

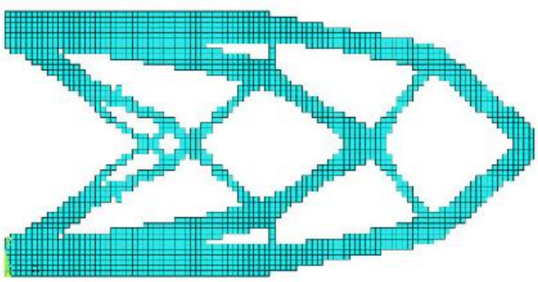


Figure 4.17 The cantilever beam optimization result under static loading

In the first design stage the centre loaded cantilever beam is fixed from its one end and the downward load is applied from its other end. 60% volume reduction is limiting value for optimization. After defining the limiting value, the finite element solution and the loop are started to find the comparatively low stressed value elements. For preventing porous structure or material islands, sensitivity algorithm is applied. Firstly %5 of low stressed value elements are removed from the design domain and the design domain is reduced %2 in each loop. Total number of iterations are 32 and number of remaining elements are 1962 in the new domain as shown in Figure 4.17. When this new domain is subjected to 1000 N downward load, the maximum stress occurred on the element is 242.49 kPa and maximum deformation of the model is $4.37 x 10^{-6}$ m.

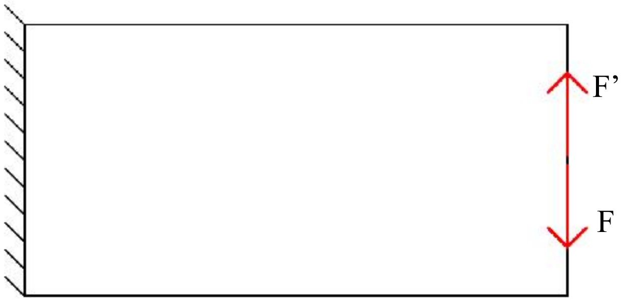


Figure 4.18 The cantilever beam under fatigue loading

The same cantilever beam (5000 elements with the dimension of 1mx2m) is modelled under the varying load between 1000 N downward direction and 1000 N upward direction as shown in Figure 4.18. In this optimization process, the limiting value is 60% volume reduction again while safety factor is taken as 1 to obtain a solution *according to volume reduction*. In this method, the aim is to find the high safety factor elements. After finding the high safety factor elements, the sensitivity analysis applied to find better element distribution. Firstly 5% of high safety factor elements are removed from the design domain again. The volume is reduced by %2. After 9 iterations the loop is ended and the new domain which has 1980 number of elements is obtained as shown in Figure 4.19. When this new domain is subjected to 1000 N downward load, the maximum stress occurred on the element is 258.15 kPa and maximum deformation of the model is $3.56 x 10^{-6}$ m.

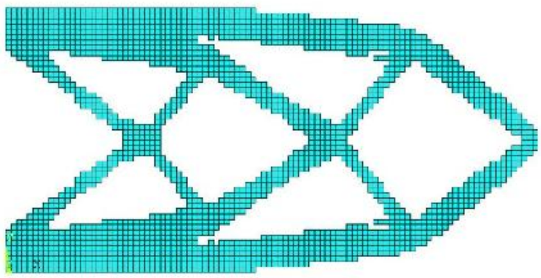


Figure 4.19 The cantilever beam optimization result under fatigue loading

The limiting value of cantilever beam which has 5000 numbers of elements is changed to safety factor and the minimum safety factor is assumed as n=2.20 to obtain a solution *according to safety factor*. The same 60% volume reduction is another limiting value. The varying load is increased to F=530 kN to obtain comparable stress values with yield and ultimate stress values of the material. Load is applied from its end and the high safety factor elements are selected. Firstly 5% of its volume is removed and in each loop the volume is reduced 2%. By the same time, the factor of safety are calculated and checked according to the given value. Before the algorithm reached the maximum volume reduction, minimum value of safety factor is reached. When the optimization is finished, total number of iteration was 4. The maximum stress occurred on the elements is 66.98 MPa when 530 kN load is applied and maximum deflection is $0.98x10^{-3}$ m. In the new design domain, 3824 elements exist. The optimization result is shown in Figure 4.20.

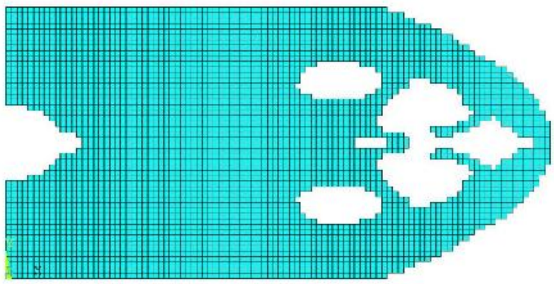


Figure 4.20 The cantilever beam optimization result under fatigue loading according to the limiting value of safety factor

The two end fixed beam loaded from its top – centre as shown in Figure 4.21 is modelled for static loading. The size of the beam is 1m x 3m and the applied load is downward 1000 N. The number of the element is 7500 and the desired volume reduction is 70%.

In the first design stage the beam is fixed from its both ends and the downward load is applied from its top–centre. Volume reduction 70% is limiting value. After defining the limiting value, the finite element solution and the loop are started to find the comparatively low stressed elements. Sensitivity algorithm is applied. Total number of iterations are 37 and number of remaining elements are 2232 in the solution as shown in Figure 4.22. When this new domain is subjected to

1000 N downward load, the maximum stress occurred on the element is 84.03 kPa and maximum deformation on the model is 0.86×10^{-6} m.

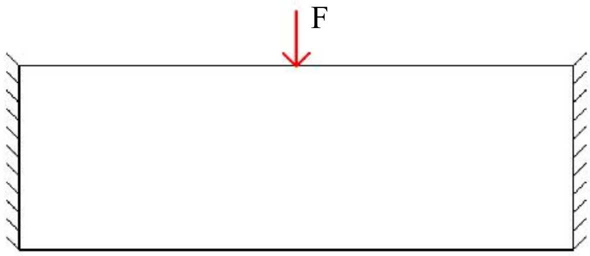


Figure 4.21 The two end fixed beam under static loading

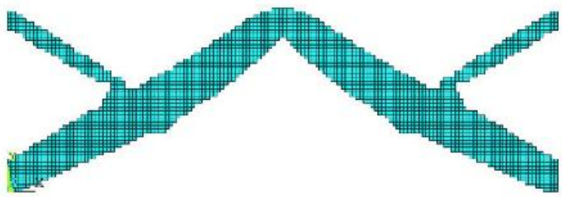


Figure 4.22 The two end fixed beam optimization result under static loading

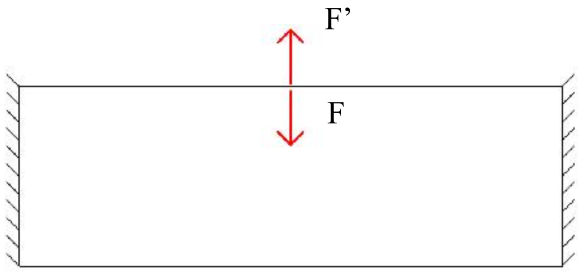


Figure 4.23 The two end fixed beam under fatigue loading

The two end fixed beam (7500 elements with the dimension of 1m x 3m) is modelled under reversed load between 1000 N downward direction and 1000 N upward direction as shown in Figure 4.23. In this optimization process, the limiting value is 70% volume reduction again. In this method, the aim is to find the high safety factor elements. After finding the high safety factor elements, the sensitivity analysis applied. Firstly 5% of high safety factor elements are removed from the design domain again. The volume is reduced by 2% in subsequent iterations. After 13 iterations the loop is ended and the new domain which has 2260 number of elements is obtained as shown in Figure 4.24. When this new domain is subjected to 1000 N downward load, the maximum stress occurred on the element is 121.92 kPa and maximum deformation the model is $0.7x10^{-6}$ m.

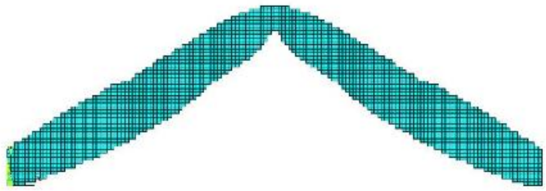


Figure 4.24 Two end fixed beam optimization result under fatigue loading

The limiting value of two end fixed beam which has 7500 numbers of elements is changed to safety factor and the minimum safety factor is required as n=2.20. Similarly, the varying load of 400 kN is applied from its centre and the high safety factor elements are selected. At the same time, the safety factor is calculated and checked according to the given value. Before reaching the maximum volume reduction of 70%, it is reached to the other limiting value of safety factor n=2.20. When the optimization is finished, total number of iteration is 6. The maximum stress occurred on the elements is 18.72 MPa and maximum deflection is $0.17x10^{-3}$m when the load of 400 kN is applied. The optimization result is shown in Figure 4.25.

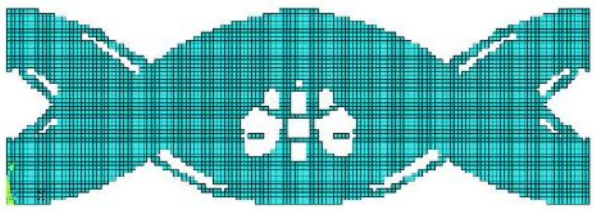


Figure 4.25 Two end fixed beam optimization result under fatigue loading according to the limiting value of safety factor

The two end fixed beam loaded from its centre as shown in Figure 4.26 is modelled for static loading. The size of the beam is 1m x 2m and the applied load is 1000 N downward. The number of the elements is 5000 and the desired volume reduction is 80%.

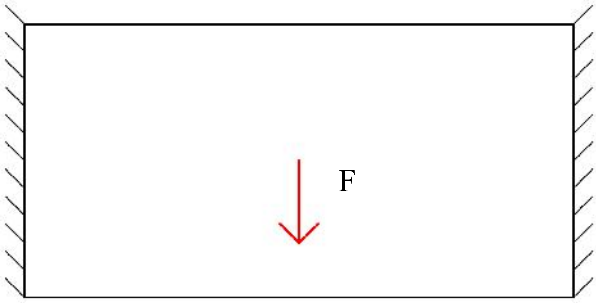


Figure 4.26 Centre loaded design domain under static loading

In the first design stage the beam is fixed from its both ends and the downward load is applied from its centre. 80% volume reduction is limiting value. After defining the limiting value, the finite element solution and the loop are started to find the comparatively low stressed elements. For preventing porous structure, sensitivity algorithm is applied. Total number of iterations are 39 and number of remaining elements are 1264 in the new domain as shown in Figure 4.27. When this new domain is subjected to 1000 N downward load, the maximum stress occurred on the element is 65.81 kPa and the maximum deformation of the model is $0.62x10^{-6}$ m.

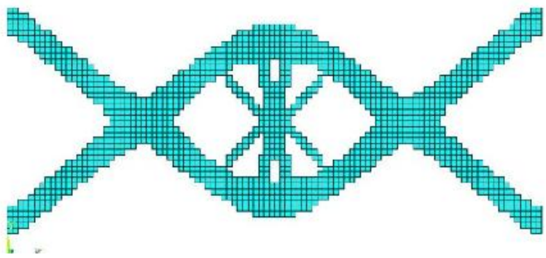


Figure 4.27 Centre loaded optimization result under static loading

The same beam (5000 elements with the dimension of 1m x 2m) is modelled under the reversed load between 1000 N downward direction and 1000 N upward direction as shown in Figure 4.28. In this optimization process, the limiting value is 80% volume reduction again. In this method, the aim is to find the high safety factor elements also. After 14 iterations the loop is ended and the new domain which has 1096 number of elements is obtained as shown in Figure 4.29. When this new domain is subjected to 1000 N downward load, the maximum stress occurred on the element is 64.08 kPa and maximum deformation the model is $0.58x10^{-6}$ m.

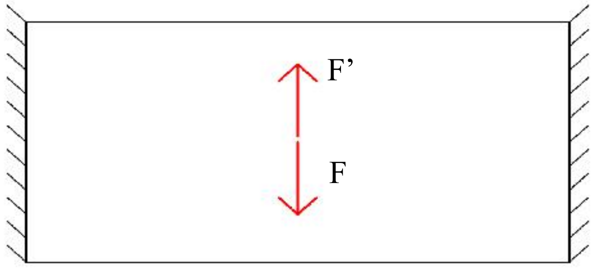


Figure 4.28 Centre loaded design domain under fatigue loading

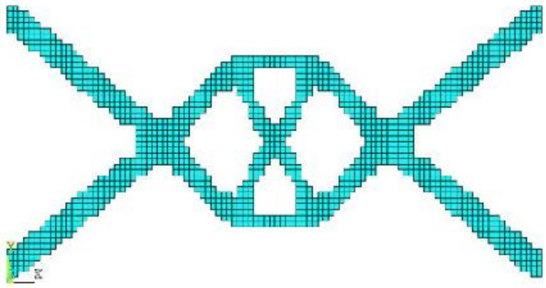


Figure 4.29 Centre loaded optimization result under fatigue loading

Load on the two end fixed beam which has 5000 numbers of elements is changed to 700 kN of reversed load and the minimum safety factor is assumed as n=2.20. Firstly 5% of high safety factor elements are removed from the design domain again. The volume is reduced by 2% in subsequent iterations. By the same time, the factor of safety are calculated and checked according to the given value. When the optimization is finished, total number of iteration is 9. When load of 700 kN downward load is applied, the maximum stress occurred on the elements is 27.84 MPa and maximum deflection is 0.19×10^{-3} m. The optimization result is shown in Figure 4.30.

The right – top loaded cantilever beam loaded from its end as shown in Figure 4.31 is modelled for static loading. The size of the beam is 1m x 2m and the applied load is 1000 N downward. The number of the elements is 5000 and the desired volume reduction is 60%.

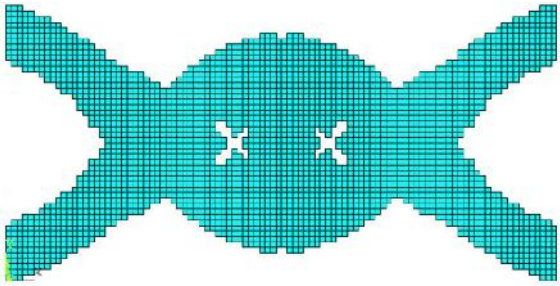


Figure 4.30 Optimization result of the two end fixed beam loaded from its centre under fatigue loading according to the limiting value of safety factor

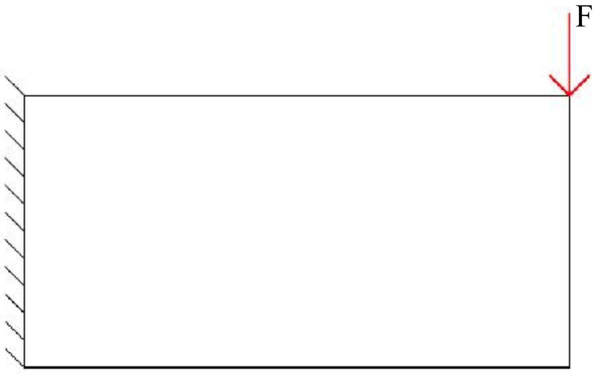


Figure 4.31 Right – top loaded cantilever beam under static loading

In the first design stage the right – top loaded cantilever beam is fixed from its one end and the downward load is applied from its other end. 60% volume reduction is limiting value. After defining the limiting value, the finite element solution and the loop are started. Total number of iterations are 27 and number of remaining elements are 2476 in the new domain as shown in Figure 4.32. When this new domain is subjected to 1000 N downward load, the maximum stress occurred on the element is 236.57 kPa and the maximum deformation on the model is $4x10^{-6}$ m.

The right – top loaded cantilever beam (5.000 elements with the dimension of 1m x 2m) is modelled under the reversed load between 1000 N downward direction and 1000 N upward direction as shown in Figure 4.33. In this optimization process, the limiting value is 60% volume reduction again. In this method, the aim is to find the high safety factor elements also. After 11 iterations the loop is ended and the new domain which has 2243 number of elements is obtained as shown in Figure 4.34. When this new domain is subjected to 1000 N downward load, the maximum stress occurred on the element is 264.07 kPa and maximum deformation of the model is $4x10^{-6}$ m.

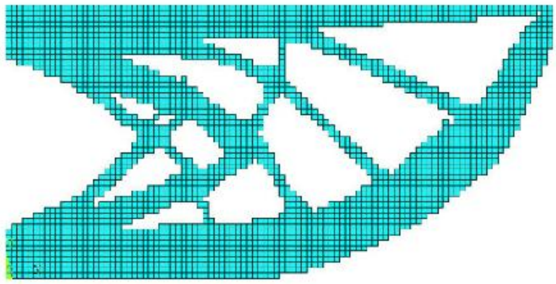


Figure 4.32 Right – top loaded cantilever beam optimization result under static loading

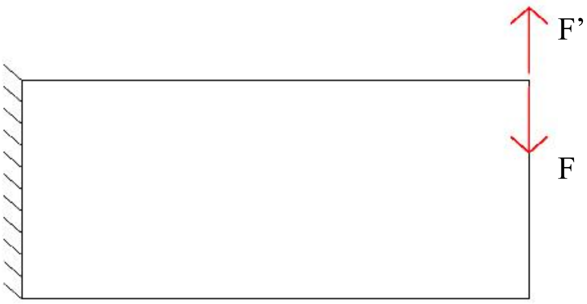


Figure 4.33 Right – top loaded cantilever beam under fatigue loading

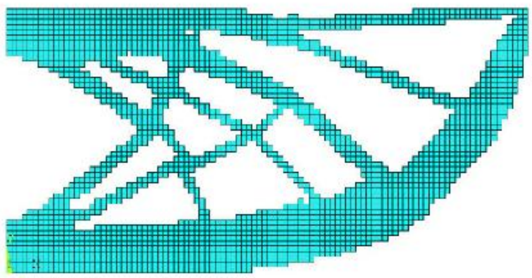


Figure 4.34 Right – top loaded cantilever beam optimization result under fatigue loading

Load on the two end fixed beam which has 5000 numbers of elements is changed to 200 kN of reversed load and the minimum safety factor is assumed as n=2.20. Similarly, the reversed load of 200 kN is applied from its end and the high safety factor elements are selected. When the optimization is finished, total number of iteration was 5. If downward direction 200 kN load is applied to the model, the maximum stress occurred on the model is 33.77 MPa and maximum deflection is $0.58x10^{-3}$ m. The optimization result is shown in Figure 4.35.

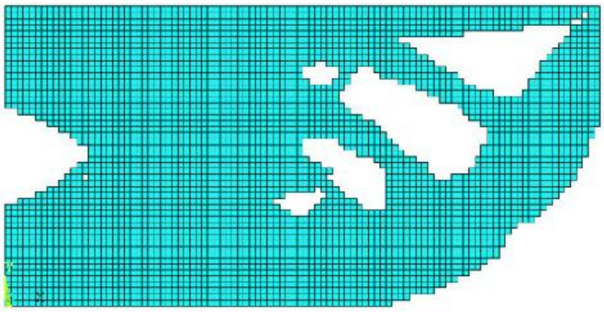
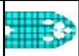
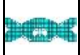
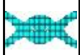


Figure 4.35 Optimization result of the right – top loaded cantilever beam under fatigue loading according to the limiting value of safety factor

The numerical values of static loading and fatigue loading cases are gathered in Table 4.1.

Table 4.1 Comparison of solutions for static and fatigue loading

Initial Domain		**Design for static loading**			**Design for fatigue loading**				**Design for safety factor n = 2.20**			
Model (Figure No)	Element Number	**Model**	Element Number	Deformation ($mx10^{-6}$)	**Model (Figure No)**	**Model**	Element Number	Deformation ($mx10^{-6}$)	**Model (Figure No)**	**Model**	Element Number	Deformation ($mx10^{-3}$)
4.17	5000		1962	4.37	4.19		1980	3.56	4.20		3824	0.98
4.22	7500		2232	0.86	4.24		2260	0.70	4.25		5434	0.17
4.27	5000		1264	0.62	4.29		1096	0.58	4.30		2784	0.19
4.32	5000		2476	4.00	4.34		2243	4.00	4.35		3902	0.58

Investigating the values in Table 4.1, it is seen clearly the optimization results of the static loading conditions and the fatigue loading conditions are similar. The deformations of the fatigue loading conditions are slightly smaller than the deformations of the static loading conditions. In the third cases of the examples, the desired volume reduction is same with static and fatigue conditions. But the main limiting value is safety factor. The optimization is finished before it is reached to the desired volume reduction due to reach of minimum value of safety factor. Therefore the optimization result contains more material compared to the others. The element numbers of these models are very big and they are safer than the other models so the deformations are smaller.

4.4. Discussions

This chapter is grouped under two main parts. In the first part, the comparison with literature was given. Three cases were examined geometrically. In almost all examples, similar results were obtained.

In the second part, the topology optimization algorithm under fatigue loading developed in this thesis are compared with the literature [1] which was for static loading cases. The results were compared according to the maximum deformations and the stress values. During these cases, optimization is performed for volume reduction ratio to obtain same conditions of optimization. Difference between the two algorithms is that the previous algorithm considers the von Misses stress value while safety factor is considered in this recent study. Four cases are examined for comparison. Visually the similar optimization results are obtained in these cases. The maximum deformation values of fatigue optimization are slightly better than that of optimization for static condition. Another point in optimization according fatigue is that, thin connections observed in static case are disappearing.

As a whole it can be concluded that optimization results according to fatigue and static loadings result similar solutions for 2D parts. Validity of the design according to safety factor constraint is checked with hand calculations and it is approved that the safety factor values of optimization are correct values.

During the calculations and optimization operations factors affecting the endurance strength are considered to be unity as the main objective of the study is to start to fatigue concept under topology optimization heading.

As a conclusion; it is seen that the developed algorithm yields reliable results under simplified conditions for design of parts under fatigue and static loadings.

CHAPTER 5

CONCLUSIONS

5.1. Discussions and Conclusions

Topology optimization is a powerful method for the use of design engineers which gives an idea of optimum material distribution in the design space. General drawback of the topology optimization is time required for the solution. For example optimization of simple planer cantilever beam example may require up to 3 or 4 hours. In the concept of previously completed study [1], developed ERM method yielded up to 90% of time reduction. The developed algorithm was proved to be applicable for 2D planer parts under static loadings. Applicability of the developed algorithm to real case problems requires some improvements like inclusion of 3rd dimension and fatigue cases.

Main idea in the present thesis was improvement of the ERM method with the inclusion of fatigue loading. In the applications there are some types of fatigue loading. During the present study the optimization procedure is developed for reversed type of loading. Optimization according to minimum value of safety factor is added to algorithm.

Infinite life is considered in this thesis for the optimized parts. During the calculation of endurance limit (S_e) of a part, the k factors must be determined. As the main objective in this thesis is to simply add the fatigue design to ERM, these k factors are all considered to be unity. Among the failure theories, Soderberg approach is preferred as that is the most conservative one. Using the theory, safety factor for each element in the design domain is calculated and elements to be removed are selected among these high safety factor elements.

Application of the developed algorithm is shown on a cantilever beam. Verification of the algorithm is done by two ways:

i. Before optimization VR value is set to a required value while n (safety factor) is set to 1. Hence algorithm is making optimization according to VR as it can never reach to n=1 as the load value is not so high. For this case it is observed that a comparable but slightly better result is obtained compared to static loading solution.

ii. For the second case VR value is set to a high value (70%) while n is set to its required value 2.20. For this case algorithm resulted a solution with more material. Outcomes of the solution are checked with hand calculation and it is seen that the results are valid for safety factor.

Results of developed algorithm are compared with that of available literature in Chapter 4. The optimization outcomes are compared according to geometrical results and they are observed to be similar. After this comparison, the numerical problem cases are examined. The comparison is done according to the maximum deformation occurred on models. Similar geometrical results are obtained for static and dynamic loading cases. Generally outcomes of the optimization according to the fatigue cases yielded less deformation value compared to the static cases under the same static load values.

Finally, investigating the optimization results and deformation values of Table 4.1, it is concluded at the end of thesis that, the developed algorithm is working well under static and reversed loadings.

Determination of safety factor for each element and removal of high safety factor elements are contribution of the thesis to ERM algorithm.

5.2. Future Works

Especially in industrial applications for getting better results the algoritm can be developed in the future as:

1. The algorithm is developed for 2D plane stress applications in this study. In real cases, all of the machine parts can not be considered as 2D. So the study can be expanded for 3D parts.

2. The optimization can be adapted to different types of materials, such as composite, functionally graded materails. Since usage areas (especially in aerospace industry and automotive industry) of these materials are increasing rapidly.

3. There are two ways to verify the theoritical studies: one of them is to do experimental study and the other one is comparing with similar studies which are existing in literature. Present studies outcomes are compared with that of references as there is not any experimental setup in the laboratory. Performing experimental studies will completely verify the accuracy of the new developed algorithm.

4. In this thesis, k factors are accepted as unity. In real cases, the values of k factors must be different from unity. Also, after each iteration stress concentration is changed. Algortihm may be developed to use realistic k values.

REFERENCES

[1] Göv, İbrahim (2009). Applying topology optimization to design of planer machine parts.*M.Sc Thesis in Mechanical Engineering, University of Gaziantep*.

[2] Grant Steven, Osvaldo Querin, Mike Xie (1999). Evolutionary structural optimisation (ESO) for combined topology and size optimisation of discrete structures.*Computer Methods in Applied Mechanics and Engineering.*,**188**, 743 – 754.

[3] T.E. Bruns, D.A. Tortorelli (2003). An element removal and reintroduction strategy for the topology optimization of structures and compliant mechanisms.*International Journal for Numerical Methods in Engineering*, **57**, 1413 – 1430.

[4] James K. Guest, Jean H. Prévost (2006). Optimizing multifunctional materials: Design of microstructures for maximized stiffness and fluid permeability.*International Journal of Solids and Structures*, **43**, 7028 – 7047.

[5] Zhan Kang, Xiaoming Wang, Rui Wang (2008). Topology optimization of space vehicle structures considering attitude control effect. *Finite Elements in Analysis and Design*, **45**, 431 – 438.

[6] Bayrakçeken, H., Taşgetiren, S., Yavuz, İ. (2006). Two cases of failure in the power transmission systems on vehicles: A universal joint yoke and a drive shaft. *Engineering Failure Analysis*, **14**, 716 – 724.

[7] Seung K. Koh (2008).Fatigue analysis of an automotive steering link.*Engineering Failure Analysis*, **16**, 914 – 922.

[8] D. Colombo, M. Gobbi, G. Mastinu, M. Pennati (2008). Analysis of an unusual McPherson suspension failure.*Engineering Failure Analysis*, **16**, 1000 – 1010.

[9] E.S. Palma, C.L. Petracconi, S.E. Ferreira (2009). Fatigue behaviour analysis of a rear tow hook pin of a passenger vehicle.*Engineering Failure Analysis*, **16**, 2408–2416.

[10] Gruen Florian, Eichlseder Wilfried, Puchner Klaus (2003). Shape and topology optimization regarding fatigue analysis.*Cumulative Fatigue Damage*.

[11] P. Häuβler, A. Albers (2005). Shape optimization of structural parts in dynamic mechanical systems based on fatigue calculations. *Structural Multidisc Optimization*, **29**, 361 – 373.

[12] Albert Albers, Pascal Häuβler (2005). Topology optimization of dynamic loaded parts using multibody simulation and durability analysis.*NAFEMS Seminar: Optimization in Structural Mechanics*.

[13] Mirosław Mrzygłód, Andrzej P. Zieliński (2006). Numerical implementation of multiaxial high – cycle fatigue criterion to structural optimization.*Journal of Theoretical and Applied Mechanics*, **44**, 691 – 712.

[14] M. Mrzygłód, A. P. Zieliński (2006). Parametric structural optimization with respect to the multiaxial high – cycle fatigue criterion.*Structural Multidisc Optimization*, **33**, 161 – 171.

[15] M. Mrzygłód, A. P. Zieliński (2007). Multiaxial high – cycle fatigue constraints in structural optimization.*International Journal of Fatigue*, **29**, 1920 – 1926.

[16] Boris Desmorat, Rodrigue Desmorat (2008). Topology optimization in damage governed low cycle fatigue. *Comptes Rendus Mecanique*, **336**, 448 – 453.

[17] Kaya, N., Karen, İ., ÖztürkF. (2010). Re-design of a failed clutch fork using topology and shape optimisation by the response surface method. *Materials and Design*, **31**, 3008 – 3014.

[18] Dulyachot Cholaseuk (2006). A stressed based material distribution method for optimum shape design of mechanical parts.*Thammasat International Journal of Science and Technology*, **11**, No. 3.

[19] http://cat.inist.fr/?aModele=afficheN&cpsidt=20175452

[20] Grégoire Allaire, Zakaria Belhachmi, François Jouve (1996). The homogenization method for topology and shape optimization – single and multiple loads case.*European Journal of Finite Elements*, **5**, 649 - 672.

[21] Joseph Edward Shigley (2006). *Mechanical engineering design*.(8th ed.), McGraw-Hill Book Co.

[22] Joseph Edward Shigley (1983). *Mechanical engineering design*. (4th ed.), McGraw-Hill Book Co..

[23] S.Y. Wang et. Al. (2006). "An extended level set method for shape andtopology optimization". Journal of Computational Physics.In press.

[24] Zhen Luo et. al. (2006)."Fuzzy tolerance multilevel approach for structuraltopology optimization". Computers and Structures, **84**, 127–140.

[25] Samuel Amstutz, Heiko Andra (2006). "A new algorithm for topology optimizationusing a level – set method".Journal of Computational Physics,**216**, 573–588.