UNIVERSITY OF GAZİANTEP GRADUATE SCHOOL OF NATURAL & APPLIED SCIENCES

APPLYING TOPOLOGY OPTIMIZATION TO DESIGN OF PLANER MACHINE PARTS

M.Sc. THESIS
IN
MECHANICAL ENGINEERING

BY
İBRAHİM GÖV

JAN 2009

Applying Topology Optimization to Design of Planer Machine Parts

M.Sc. Thesis

in

Mechanical Engineering

University of Gaziantep

Supervisor

Assist. Prof. Dr. M. Akif KÜTÜK

By

İbrahim Göv

January 2009

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

Name of the thesis: Applying Topology Optimization to Design of Planer Machine Parts
Name of the student: İbrahim GÖV
Exam date: 15.01.2009

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. Sedat BAYSEÇ)
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. İbrahim H. GÜZELBEY	____________________
Assist. Prof. Dr. M. Akif KÜTÜK	____________________
Assist. Prof. Dr. Abdulkadir ÇEVİK	____________________
Assist. Prof. Dr. Ahmet ERKLİĞ	____________________

ABSTRACT

Applying Topology Optimization to Design of Planer Machine Parts

GÖV, İbrahim
M.Sc. in Mechanical Eng.
Supervisor: Asst. Prof. Dr. M. Akif KÜTÜK
January 2009, 70 pages

During the last two decades a new field of applied mathematics has been used very commonly in industrial and scientific applications, namely the topology optimization techniques. The technique takes very long solution times on Finite Element (FE) programs. Special topology optimization programs exist but also not preferred due to high cost. There are number of numerical Finite Element Method (FEM) based package programs to analyze structures. Developing a macro to use with a commercial FE program is considered to be very useful to optimize topology of machine parts during design stage.

Some methods for topology optimization have been proposed in the literature. These methods are investigated in this study. The main idea of topology optimization is removal of low stressed material from a structure in an iterative process. Element Removal Method (ERM) depends on this idea. In this 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 %.

Key Words: topology optimization, element removal, FEM, structural optimization.

ÖZET

Topoloji Optimizasyonunun Düzlemsel Makine Parçaları Tasarımına Uygulanması

GÖV İbrahim
Yüksek Lisans Tezi, Mak. Müh. Bölümü
Tez Yöneticisi: Yrd. Doç. Dr. M. Akif KÜTÜK
Ocak 2009, 70 sayfa

Topoloji optimizasyon yöntemi olarak adlandırılan, uygulamalı matematiğin yeni bir alanı son yirmi yıldır endüstriel ve bilimsel uygulamalarda yaygın olarak kullanılmaktadır. Yöntem, Sonlu Eleman (SE) programlarında uzun çözüm süresi gerektirmektedir. Özel topoloji optimizasyon programları mevcut olmasına rağmen yüksek fiyatları nedeni ile tercih edilmemektedirler. Yapıların analizi için birçok nümerik Sonlu Eleman Metoduna (SEM) dayalı paket programları mevcuttur. Tasarım esnasında makine parçalarının topolojisinin optimize edilebilmesi için, ticari bir SE programı ile kullanılmak üzere bir makro geliştirmenin çok yararlı olacağı düşünülmüştür.

Literatürde topoloji optimizasyonu için bazı yöntemler ileri sürülmüştür. Bu çalışmada bu yöntemler araştırılmıştır. Topoloji optimizasyonun ana fikri tekrarlanan bir işlem içerisinde düşük gerilime sahip malzemenin yapıdan çıkarılmasıdır. Eleman Silme Metodu (ESM) da bu fikre dayanmaktadır. Bu çalışmada ESM'nun uygulanması için ticari bir SE programı olan Ansys'de algoritma geliştirilmiştir.

Geliştirilen bu algoritmanın sonuçları yayınlanan bazı sonuçlarla karşılaştırılmıştır. Ayrıca bu algoritmanın sonuçlarıyla Ansys'in topoloji optimizasyon aracının sonuçları da karşılaştırılmıştır. Tüm karşılaştırmalar tatmin edici sonuçlar vermiştir. Geliştirilen algoritma topology optimizasyonunun uzun çözüm süresi dezavantıjını geride bırakmıştır. Yüksek eleman sayılı parçaların optimizasyonu için gerekli zaman %90'a kadar düşürülmüştür.

Anahtar kelimeler: topoloji optimizasyonu, eleman silme, SEM, yapısal optim.

ACKNOWLEDGEMENTS

I would like to thanks my supervisor Assist. Prof. Dr. M.Akif KÜTÜK, and also special thanks Prof Dr. İ. Halil GÜZELBEY, Assoc. Prof. Dr. Bahattin Kanber, Assist. Dr. Ahmet ERKLİĞ, Ö. Yavuz BOZKURT and Ali KILIÇ for their valuable contributions.

Also I would like to thank to my family for their supports.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS

A	=	Cross-sectional area
b	=	Design variable
D	=	Mean diameter
D	=	Structural domain
e	=	Element
E	=	Young's modulus
f	=	Element force
H	=	Heaviside function
I	=	Moment of inertia
k	=	Element stiffness matrices
K	=	Stiffness matrix
M	=	Bending moment
P	=	Load vector
T	=	Geometric transformation matrix
t	=	Thickness
U	=	Displacement
u	=	Nodal displacement
V	=	Volume
W	=	Work
y	=	Transverse location
γ	=	Penalization parameter
Γ	=	Traction surface
δ	=	Dirac function
ε	=	Strain

ν	=	Poisson's ratio
ρ	=	Material density
$\overline{\sigma}$	=	Allowable stress
σ	=	Stress
τ	=	Traction load
Φ	=	Level set function
Ω	=	Computational domain

CHAPTER 1

INTRODUCTION

The application of topology optimization is not used very commonly in industrial problems [6]. The main reason is that, we must use large number of element and iteration to find optimum structure but this causes very long solution time for optimization process and also increases hardware requirements.

Many methods are applied to topology optimization, for decreasing solution time such as parallel programming, fuzzy logic, genetic algorithm, neural network. These methods are used to obtain the best solution at shortest time. But the designer needs mathematical programs (i.e. Matlab, Matematica) for using methods. The mathematical programs are not useful for an industrialist for modelling and Finite Element Analysis (FEA) solutions because they are not customized for these applications. Use of these programs requires very long source codes for modelling and also FEA solutions. This is very difficult and useless. Also, topology optimization methods can be applied with programming languages (i.e. FORTRAN, C++), but they have same problems with mathematical programs.

For those difficulties, using a customized FEA program (Ansys) is decided. Modelling of the problem and FEA solutions can be realised very easily using this program. The program have topology optimization tool, but it is not generally preferred due to very long solution times especially with high element numbers and iteration loops.

The goal of the present study is to search efficient methods for structural topology optimization from the basic mathematical theory to up-to-date numerical algorithms and practical applications; and to apply the most efficient method to two dimensional machine parts. So, some methods for topology optimization have been investigated.

The simple idea of the topology optimization is the removal of less efficient material from a structure in an iterative process. An algorithm is developed to apply this simple idea on planer machine parts. For applications, Element Removal Method (ERM) is adapted into Ansys.

To show validity of this algorithm results of this algorithm are compared with some published results. The results of the developed algorithm are also compared with that of Ansys topology optimization tool.

CHAPTER 2

LITERATURE SURVEY

2.1. Introduction

Topology optimization is now being applied for optimal design in such diverse areas as stiffness, natural frequency, electro-magnetism, fluids. After Bendsoe and Kikuchi's pioneering works, topology optimization method has become popular and has been successfully applied into industrial design. Then Bendsoe and Sigmund have systematically developed new theories, methods and applications for topology optimization.

Early topology optimization techniques worked on finding optimal layout and geometry for trusses. Most topology optimization research in the next two decades was concentrated in the attainment of optimal configuration of discrete structures. Many methods are developed to optimize these structures.

Many approaches implemented for solving topology optimization problems in the last decades such as homogenization method, material distribution method, the evolutionary structural optimization, level set based optimization and material cloud method. In this chapter, these optimization methods are discussed which are available in the literature.

Prior to introducing details of topology optimization methods, some related material is discussed in following sections.

2.2. Mechanical Engineering Design

Mechanical design means the design of things and systems of a mechanical nature-machines, products, structures, devices, and instruments. For the most part, mechanical design utilizes mathematics, the materials sciences, and the engineering mechanics sciences. One of the most difficult problems that designers have to overcome is the balancing of ‘trade-offs’. In the case of a structural component in an aircraft there might be a trade-off between weight and strength. To maximise load carrying capacity, the unloaded weight of the aircraft must be as small as possible, and one way of achieving this is to minimise the amount of material in each component. However, there comes a point where a particular component cannot be made any lighter without reducing its load carrying capacity to a point where structural failure is an unacceptable risk [2].

A design engineer working in the field of research and development has to often design completely new structures. The loading and support conditions of a particular design problem are usually known in advance, but the designer is unsure of what the actual structure should look like. The weight is known to be one of the main factors of the final cost of load carrying structures, and for that reason, the weight reduction is often set as the main objective of the design task.

Computer aided design (CAD) is a powerful tool for designers. In all industrial application, the design procedure must be very sensitive and fast to supply the new and/or improved product features. For this reason, the designers mostly use CAD programs especially when designing a new product. The new product is designed on the computer then structural analysis are used to the product can endure load and boundary conditions. After the structural analysis, the designed product can be improved easily. So, the solution times can be decreased.

Many methods are developed to help the designer for CAD processes. Structural optimization methods, which commonly used, are some of them. These methods are developed for help to obtain the optimum design. By using these methods, the designer can form an opinion about the rough shape of the design part. So the structural optimization methods can be decrease the solution time of the part design.

2.3. Structural Optimization

Of the engineering disciplines, structural design has probably seen the most widespread development and application of numerical optimization techniques. Linear programming was used to design structures based on the plastic design theory as early as 1956 (Heyman); the application of nonlinear programming techniques to the design of elastic structures was first introduced by Schmitt in 1960. Schmitt coined the phrase "structural synthesis", which is today the common term used to identify numerical optimization as a general design strategy. While the application of these techniques is not yet commonplace in design industry, the methodology for structural optimization is well developed for large classes of problems [3].

Although for special cases unique analysis techniques may be employed, most linearly elastic structures today are analyzed using the popular finite-element technique, using the displacement method of analysis. With this analysis tool, we have the unique advantage of being able to calculate gradients of the weight of the structure and most common constraint with little computational effort beyond that required for a single analysis. For this reason, structural synthesis can efficiently deal with relatively large numbers of design variables and constraints.

In structural optimization, new methods, such as topology optimization, have been used for effective designs in last decades. Topology optimization is more powerful method to obtain initial design geometry. Topology optimization can be used for new product design and also can be used to improve mechanical strength of a used product. In the following sections, some common methods of topology optimization are investigated which are available in the literature.

2.3.1. Material Distribution Method (MDM)

Dulyachot Cholaseu (2006) presented a numerical method for minimum-weight shape design of a mechanical part with a stress constraint. The design method aims to produce a fully-stressed design by distributing material according to the local von Mises stress. Material distribution is represented as element thickness instead of density or elastic modulus in a two-dimensional finite element model, which makes the optimum design practical. The method is demonstrated with a short cantilever beam design problem.

The advantages of this design method over other mathematical based topology design methods are its simplicity and manufacturability of the optimum design. The method can be applied to any finite element source codes that are available. The method can also be extended to multiple load cases by employing virtual stress distributions (union product of von Mises stress distribution from all load cases). The limitation of the method is that it is currently applicable only to in-plane load cases. Stresses in the optimum design are based on the plane stress assumption, so, fine tuning is required upon constructing a three-dimensional model of the optimum design. Further study is required to extend the method to cover fully three-dimensional load cases.

Arash Mahdavi, et. al. (2005) studied a parallel processing algorithm for the compliance topology optimization problem, where the FEA and sensitivity calculations have been parallelized. Without assembly of the global stiffness matrix, the memory requirement as well as computation time has been reduced. To reduce the computation time of such problems, parallel computing in combination with domain decomposition is used. The power law approach has been used as the material distribution method and for locating the optimum solution; an optimality criteria-based optimizer is used. The results of the study show that the parallel computing technique is a useful tool for solving computationally intensive topology optimization problems.

2.3.2. Level Set Approach (LSA)

Jian Hua Rong and Qing Quan Liang (2007) presented a level set based optimization method for topology optimization of continuum structures with bounded design domains and traction interface constraints. To overcome the limitations of current level set methods and the stopping issue of structural boundary movements, the study presents a set of new level set based optimization formulae for the optimal design of continuum structures with bounded design domains. Secondly, in order to overcome the difficulty in nucleating holes in the design domain in the level set based optimization method, the paper introduces a new optimization strategy with a small possibility random topology mutations and crossovers. A mixed topology optimization algorithm is implemented and presented for the compliance minimization problems of continuum structures with material volume constraints.

Chungang Zhuang, et. al. (2007) presented the level set method for the multiple materials design. Since the level set method can naturally handle topological changes during the interface propagation, the level set model based material implicit representation has been applied to structural topology optimization. However, the level set method cannot generate new holes during the optimization process and the final result depends on the initial topological guess. In their study, the new holes are generated according to the distribution of von Mises stress in material region to suppress the dependence of the initialization. They established the replacement strategy of multiple materials to make the objective function minimum. And then, the shape optimization is implemented by the propagating using the level set method with the descent gradient direction. The replacement of multiple materials and new topology are generated according to the distribution of von Mises stress, which suppresses the dependence of initial topological guess. The computational cost is high for vector level set model based structural topology optimization.

Mei Yulin and Wang Xiaoming (2004) proposed and studied the level set method for structural topology optimization, which is basically a steepest descent method by combining the shape sensitivity analysis with the Hamilton–Jacobi equation for moving the level-set function. They considers the multi-material and multi-constraint problems, and investigates the topology optimization algorithm by using the different material representation models, and broadens its application from stiff structure designs and compliant mechanism designs to material designs by a number of benchmark examples. Meanwhile in order to further improve computational efficiency and overcome the difficulty that the level set method cannot generate new material interfaces during the optimization process; the multi-material topological derivative analysis is incorporated into the level set method for topological optimization. Several numerical techniques are employed to enhance the ability of the level set method for structural topology optimization.

ChunGang Zhuang, et. al. (2006) examined level set method for topology optimization of heat conduction problem. They presented a numerical approach of topology optimization under multiple load cases for heat conduction problem. This framework is based on the theories of topological derivative and shape derivative for elliptic system. It is employed level set model to implicitly represent geometric boundary of thermal conductive material. Introducing topological derivative will generate new topology in the design domain, which suppresses the dependence of initial topology guess to some extent. They develop an effective numerical technique to implement the optimal design with multiple load cases for heat conduction problem. Numerical examples demonstrate that their proposed approach is effective and strong for topology optimization of heat conduction problem.

2.3.3. Element Connectivity Parameterization (ECP)

Gil Ho Yoon, et. al. (2006) studied ECP. The topology optimization of three-dimensional geometrically non-linear structures was carried out by two versions of the ECP methods, the external ECP (E-ECP) method and the internal ECP (I-ECP) method. Both methods yield numerically stable results because the solid finite elements used to predict the structural response of a three-dimensional body remained solid throughout the optimization of the ECP methods.

Gil Ho Yoon and Yoon Young Kim (2005) developed an ECP formulation for the topology optimization of multi-physics problems in order to avoid the numerical difficulties and yield improved results. In the proposed ECP formulation, finite elements discretizing a given design domain are not connected directly, but through sets of one-dimensional zero-length links simulating elastic springs, electric or thermal conductors. The discretizing finite elements remain solid during the whole analysis, and the optimal layout is determined by an optimal distribution of the inter-element connectivity degrees that are controlled by the stiffness values of the links. The detailed procedure for this new formulation for multi-physics problems is presented. Using one-dimensional heat transfer models, the problem of the element-density-based method is explained and the advantage of the ECP method is addressed.

Gil Ho Yoon and Yoon Young Kim (2006) studied topology optimization of material-nonlinear continuum structures by the element connectivity parameterization. Instead of the element density formulation, the ECP formulation is developed for the topology optimization of isotropic-hardening elastoplastic or hyper-elastic continua by using commercial software. ECP varies the stiffness of zero-length linear elastic links that connect design domain-discretizing finite elements. Unloading was not considered. But the advantages of ECP in material-nonlinear problems were demonstrated. Multilinear elastoplastic problems and the Mooney–Rivlin hyper-elastic problems were solved to demonstrate the effectiveness of the ECP method.

2.3.4. Evolutionary Method

Gwun Jang and Byung Man Kwak (2006) proposed new strategies and implementations strongly improving the performance of the design space optimization. In the conventional topology optimization, the design domain is fixed. It is, however, desirable to make the design domain evolve into a better one during optimization process by increasing or decreasing the number of design pixels or variables, which is called space optimization. A breakthrough in obtaining sensitivities when design space expands has been made recently with necessary mathematical background, but due to coupling effect and others, sensitivity results have not been satisfactory.

Grant Steven, Osvaldo Querin, and Mike Xie (1999) studied ESO for combined topology and size optimisation of discrete structures. They demonstrated how the ESO concept can be applied to both pin-jointed and rigid-jointed 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 too much on this.

Pasi Tanskanen (2006) examined ESO method. On the basis of the previous studies, ESO was known to minimize the compliance-volume product of an element model. Consequently, ESO was seen to be analogous with a sequential linear programming (SLP) based optimization method, called the approximate optimization method. It can be stated that ESO is basically a standard mathematical programming algorithm, which just minimizes a particular objective function.

Pasi Tanskanen (2002) examined the theoretical background of the ESO method. The design domain is modelled by using equally sized elements, which are removed completely from the design domain using the strain energy based rejection criterion. Additionally, the element stiffness matrices and element volumes should be linearly dependent on the design variables, and linearly elastic material is assumed. In most ESO applications, only material removal is allowed. An element once removed should be allowed to re-enter the design domain. It can be stated that ESO is basically a standard mathematical programming algorithm, which just minimizes a particular objective function. The ESO optimization is very easy to apply. On the other hand, different constraints cannot be added into the problem.

2.3.5. Material Cloud Method (MCM)

Su-Young Chang et al. (2004) proposed MCM and showed the results of MCM, compared with those of the traditional density-based approach for several 2D linear static design problems. In MCM, the modification of design domain can be naturally and efficiently accomplished only depending on the change of values of design variables. In MCM, the design variables are central positions and sizes of material clouds, which are independent material patches.

Su-Young Chang and Sung-Kie Youn (2005) examined the MCM, and its mathematical investigation and numerical application for 3D engineering design. They summarized the concept of MCM and prove the existence of optimal solution(s) in the formulation of MCM to show the mathematical difficulties of this new method. In the MCM, the computational cost can be significantly saved due to the reduced number of active elements in design domain. But they observe that the optimal solution may be better or worse than that of the traditional density-based approach depending on the problem, because two methods search different local optimal solutions.

2.3.6. Genetic Algorithm (GA)

Shi-Yuan Wang et. al. (2004) implemented a bit-array representation method for structural topology optimization using the Genetic Algorithm (GA). The importance of structural connectivity in a design is further emphasized by considering the total number of connected objects of each individual explicitly in an equality constraint function. A violation penalty method is proposed to drive the GA search towards the topologies with higher structural performance, less unusable material and fewer separate objects in the design domain. An identical initialization method is also proposed to improve the GA performance in dealing with problems with long narrow design domains.

2.3.7. Homogenization Method (HM)

James K. Guest and Jean H. Prevost (2006) extend recent advances in the topology optimization of fluid flows to the design of periodic, porous material microstructures. Operating in a characteristic base cell of the material, the goal is to determine the layout of solid and fluid phases that will yield maximum permeability and prescribed flow symmetries in the bulk material. Darcy's law governs flow through the macroscopic material while Stokes equations govern flow through the microscopic channels. Permeability is computed via numerical homogenization of the base cell using finite elements. Using topology optimization, an inverse homogenization problem have been formulated and solved for designing porous material structures with maximum permeability.

James K. Guest and Jean H. Prevost (2006) studied on design of microstructures for maximized stiffness and fluid permeability. Topology optimization is used to systematically design periodic materials that are optimized for multiple properties and prescribed symmetries. 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. These modules are then combined and solved to maximize bulk modulus and permeability in periodic materials with cubic elastic and isotropic flow symmetries.

2.3.8. Element Removal Method (ERM)

Tyler E. Bruns and Daniel A. Tortorelli (2003) 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 this 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.

2.4. Conclusions

In this chapter, topology optimization methods are reviewed that are available in the literature. After reviewing of these articles, some problems about application of those methods are noticed. Such as, some applications are difficult to apply due to heavy mathematic involved. Different constraints cannot be added for some studies or holes must be added to design domain prior to solution of problem. Solution times are generally very long for almost all studies. Considering these facts, a useful and practical algorithm for topology optimization is tried to develop in further studies.

Writing a macro for optimization process in Ansys is considered to ease the application of method. During construction of the macro, criteria's to be observed due to literature must be: Method must be free of heavy mathematics for use of an engineer; any kind of constraint and force application must be possible; solution time must be low compared to available methods and tools. Considering these criteria's modelling and FEA solutions are done by Ansys and optimization macro is applied to stress analyzes results of Ansys. To decide on the method for topology optimization, available methods are discussed in next chapter.

CHAPTER 3

BASIC METHODS OF TOPOLOGICAL OPTIMIZATION

3.1. Introduction

Topology optimization generates the optimal shape of a mechanical structure. The structural shape is generated within a pre-defined design space. In addition, the user provides structural supports and loads. Without any further decisions and guidance of the user, the method will form the structural shape thus providing a first idea of an efficient geometry. Therefore, topology optimization is a much more flexible design tool than 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 for this reduced mass. Usually maximized properties are stiffness or lowest eigenfrequency. Another use topology optimization may also be minimizing the amount of structural mass (weight) subject to a given behavior measure.

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.

One of the design criteria's of most machinery is to meet the functional requirements with minimum weight, hence minimum material consumption and cost. Parallel to this requirement, the efforts will be focused on more integral parts that combine characteristics of high strength at low weight.

Topology optimization has become popular and has been successfully applied into industrial design since 1988, when Bendsoe and Kikuchi introduced the microstructure/homogenization approach for topology optimization. Bendsoe and Sigmund have systematically investigated new theories, methods and applications for topology optimization. Many methods have developed to facilitate and make useful the topology optimization in last decades. In this section, some mostly preferred methods are discussed:

- Material Distribution Method (density method)
- Level Set Approach (LSA)
- Evolutionary Structural Optimization (ESO)
- Material Cloud Method (MCM)
- Homogenization Method (HM)
- Optimality Criteria Method (OCM)

3.2. Material Distribution Method (MDM)

In the *Material Distribution Method*, fixed finite element grid model (checkerboard) used for design domain. Material density or elastic modulus or element thickness are parameterized related to physical properties (such as stiffness, thermal conduction, magnetic permeability, porosity, etc.) [22].

The material distribution method steps (figure 3.1) can be like this: First, a finite element model is formed for the initial shape. The finite element solver then solves for strain and stress distribution. The local von Mises stress distribution is used as information to adjust the elements' thickness. After the adjustment, a new finite element model is created and the process repeated. The optional trimming process (where elements thinner than t_{min} are removed) may be performed at specified iterations. The optimum design is obtained when a fully-stressed condition is achieved [5].

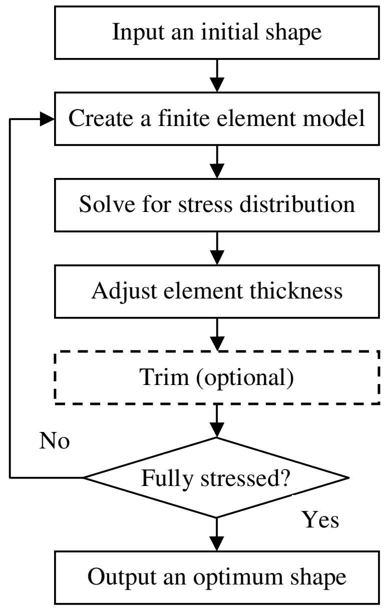


Figure 3.1 Topology optimization procedure [5].

From beam theory, major contribution to the element's von Mises stress comes from bending stress, although transverse shear stress exists. If other companents of von Mises stress are neglected, von Mises stress depends on the bending stress. The bending stress at any element *i*, shown in figure 3.2, is a function of bending moment, *M*, the transverse location, *y*, and the beam's sectional moment of inertia *I* as shown in equation (3-1a):

$$\sigma_i = -\frac{My}{I} \tag{3-1a}$$

Substitution expression of *I* into equation (a) to obtain equation (3-1b).

$$\sigma_i = -\frac{12My}{th^3} \tag{3-1b}$$

Above, *t* is the beam's thickness and *h* is the beam's height.

The bending moment at the position x of element i can be computed for a given load distribution and is considered constant in space. Position y of element i is constant. If h is kept constant, then the thickness of element i can be written in terms of bending stress as:

$$t_i = \frac{k}{\sigma_i} \tag{3-2}$$

where k is a constant.

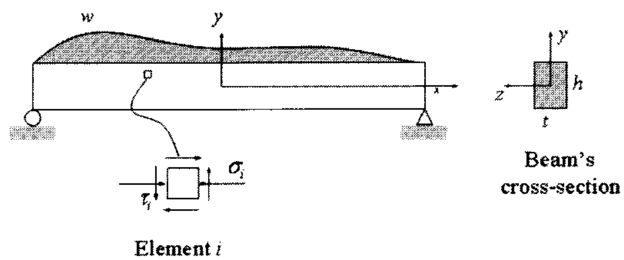


Figure 3.2 A simply supported beam under distributed load [5].

If the load is applied in the x direction, the relationship in equation (3-2) is still valid. In order to form a fully-stressed beam, one can vary the thickness of the beam's rectangular cross-section along the x-axis inversely proportional to the local maximum stress at a corresponding x-position [5].

Although there is no proof of validity of equation (3-2), when the thickness is also varied along the y-axis, the natural trend of biological growth supports the equation. More material should be distributed to the elements with high stress, which is inagreement with the trend of the equation. Hence, the relationship in equation (3-2) is assumed as a rough approximation to the sensitivity information of local thickness to local stress. As any numerical approximation, the solution process requires some form of relaxation. Thus, replace σ_i, in equation (3-2) with local von Mises stress, apply relaxation factor and the scheme takes the following form [5].

$$t_i^{j*} = t_i^{j*}\left(\frac{\sigma_i}{\sigma_{set}}\right)$$
$$t_i^{j+1} = \min\left[\left(1-r\right)t_i^j + rt_i^{j*}, t_{\max}\right] \quad (3\text{-}3)$$

where t_i^j is the original thickness of element i, t_i^{j+1} is the adjusted thickness of element i for the next iteration, $t_{\max}$ is the maximum allowable thickness, σ_i is the von Mises stress of element i, σ_{set} is the maximum allowable stress and r is a relaxation factor, $0 < r \leq 1$. The relaxation factor, r, controls the pace of the design iteration. If r is set to one, the scheme fully follows equation (3-2), which may result in an overshoot (element stresses exceed σ_{set}) [5].

A problem formulation of topology design for maximum stiffness of statically loaded linearly elastic structures:

The FEM format of the minimum compliance problem for a structure with given loading and prescribed volume V is shown below [22].

$$\min_{\rho} c(\rho)$$
$$s.t.: \sum_{e=1}^{N} v_e \rho_e \leq V, \qquad 0 < \rho \leq \rho_e \leq 1, \qquad e = 1, ..., N \quad (3\text{-}4)$$

The equilibrium equation is considered as part of a function-call that gives the value of the objective function $c(\rho)$ (assuming linear behaviour):

$$\text{c}(\boldsymbol{\rho}) = \mathbf{f}^T \mathbf{u}, \text{ where } \mathbf{u} \text{ solves: } \mathbf{K}(\boldsymbol{\rho})\mathbf{u} = \mathbf{f} \quad (3\text{-}5)$$

where **u** and **f** are the displacement and load vectors, respectively. The stiffness matrix **K** depends on the vector **ρ** of the element-wise constant *material densities* in the elements, numbered as $e = 1, . . . ,N$, in such a way that we can write **K** in the form

$$K(\rho) = \sum_{e=1}^{N} \rho_e^p K_e \tag{3-6}$$

where $\mathbf{K}_e$ is the (global level) element stiffness matrix for element e.

For the mathematical programming approach, gradients are typically required by the optimization algorithm employed to solve (3-4) and these can be derived directly or by use of the well-known *adjoint* method. In the formulation above, *fixed* FEM mesh that describes the design domain (the reference domain) and the structure is defined as a raster image by the densities ρ_e. Also, the load **f** is in this basic formulation design independent and is given in relation to the fixed mesh [22].

3.3. Level Set Approach (LSA)

In the *Level Set Approach*, Osher and Sethian (1988) have devised a level set method for numerically tracking fronts and free boundaries, which is used in many applications as motion by mean curvature. Sethian and Wiegmann (2000) use the material removal and addition scheme. The removal rate determines the closed stress contours along the new holes and the velocity of the boundary motion [7].

Ω is used to denote the computational domain and the designed structure described as the domain *D* occupies a part of the domain Ω. If the designed structure is composed of one material, computational domain is considered to be occupied by two materials, one is a solid material, and another is void. In this case the structure *D* can be represented implicitly by a real valued function Φ(x); which is defined in the domain Ω as follows

$$\begin{aligned} \phi(x) &> 0 \qquad \forall x \in D \setminus \partial D \\ \phi(x) &= 0 \qquad \forall x \in \partial D \\ \phi(x) &< 0 \qquad \forall x \in \Omega \setminus D \end{aligned} \tag{3-7}$$

As illustrated in figure 3.3, the structure or solid material is represented by {x| Φ(x) > 0; x ∈ Ω} the void is represented by {x| Φ(x) < 0; x ∈ Ω}, and the boundary ∂D of the structure can be described by the level set {x| Φ(x) = 0; x ∈ Ω} of the function Φ(x). Therefore one level set function Φ(x) can divide a domain into two parts or represent two different materials, while the level set is the material interface.

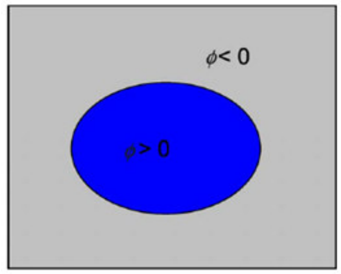


Figure 3.3 Level set representation of single material [7]

The level set algorithm: Osher and Sethian have developed a highly robust and accurate numerical method to solve the level set Eq. (3-8) [7],

$$\frac{\partial \phi_i}{\partial t} = V_n^i(x) | \nabla \phi_i \qquad \frac{\partial \phi_i}{\partial n} = 0, \quad i = 1, 2, ..., m \tag{3-8}$$

and the first order difference scheme in the two dimensions can be written as

$$\phi_{ijk}^{n+1} = \phi_{ijk}^{n} - \Delta t \left\{ \max((V_n^i)_{jk}, 0)\nabla^+ + \min((V_n^i)_{jk}, 0)\nabla^- \right\} \tag{3-9}$$

where ϕ_{ijk}^n denotes the value of ϕ_i at the time $n\Delta t$ and the grid note (x_j,y_k), and ∇^+, ∇^- are given as

$$\begin{aligned} \nabla^+ &= \sqrt{\max(D_{jk}^{-x}, 0)^2 + \min(D_{jk}^{+x}, 0)^2 + \max(D_{jk}^{-y}, 0)^2 + \min(D_{jk}^{+y}, 0)^2} \\ \nabla^- &= \sqrt{\max(D_{jk}^{+x}, 0)^2 + \min(D_{jk}^{-x}, 0)^2 + \max(D_{jk}^{+y}, 0)^2 + \min(D_{jk}^{-y}, 0)^2} \end{aligned} \tag{3-10}$$

and $D_{jk}^{\pm x}, D_{jk}^{\pm y}$ are shorthand notations of the forward and backward approximation to the first derivative with respect to x or y, respectively. Meanwhile, the highly accurate difference scheme can be constructed by using the essentially non-oscillatory interpolation and TVD Runge-Kutta method [7].

In order to obtain highly accurate numerical results and to prevent ϕ_i from deviating away from the signed distance function, a re-initialization step of the level sets is needed in iterations,

$$\frac{\partial \phi_i}{\partial t} = S(\phi_i^0)(1 - |\nabla \phi_i|) \tag{3-11}$$

where $S(\cdot)$ stands for the sign function, ϕ_i^0 denotes the iterative initial value. This approach allows us to avoid finding the material interfaces explicitly.

In the level set formulation, the normal velocity V^i_{n} is needed in a neighborhood of the material interfaces, but the shape sensitivity is only defined on the zero level set. The most natural way to extend V^i_{n} off the zero level sets is to let the velocity V^i_{n} be constant along the normal to the zero level, which leads to the following hyperbolic partial differential equation [7]

$$\frac{\partial V_n^i}{\partial t} + S(\phi_i)\frac{\nabla \phi_i}{|\nabla \phi_i|}\nabla V_n^i = 0 \tag{3-12}$$

3.4. Evolutionary Structural Optimization (ESO)

In 1993, Y.M. Xie and G.P. Steven introduced an approach called evolutionary structural optimization. A population used for the design domain. The population is evolved in randomization and a selection mechanism, until the optimum solution is reached.

This method employs a design domain constructed by the FE method, and furthermore, external loads and support conditions are applied to the element model. ESO is based on the simple idea that the optimal structure (maximum stiffness, minimum weight) can be produced by gradually removing the ineffectively used material (elements) from the design domain. The material removal can be carried out by assigning the corresponding elements a relatively small elastic modulus or thickness value. The element removal is typically based on the element von Mises stresses. The element strain energy based criterion has also been utilized. This iterative ESO procedure is to be repeated until the rejection criterion values of all the elements are within a given range [16].

Considering the engineering aspects, ESO seems to have some attractive features: the ESO method is very simple to program via the FEA packages and requires a relatively small amount of FEA time. The ESO method minimizes the compliance-volume (CV) product of a finite element model.

It is often proposed that if topology optimization is utilized to solve a design problem, the optimization should be done in two separate stages: in the first stage, the overall structure is outlined by applying a topology optimization method, and in the second stage, the sizing optimization can be employed. The standard ESO lacks generality, i.e. no specific stress or displacement constraint can be added into the minimization problem. For the reasons discussed, the optimal solution may still be reached by ESO. Besides, if the twostage procedure is employed, the sizing of the structural components will not take place until the second stage. Consequently, in that case, there is no actual need to enforce the stress and displacement constraints yet. ESO is obviously well suited to solve the first-stage optimization problems [16].

The second-stage sizing optimization can be performed independently, regardless of the first-stage optimization approach. Additionally, the objective of the minimization can be changed, if necessary. However, if some of the design constraints are not considered until the second stage, the ESO topology may fail in the sizing optimization [16].

Formulation of the ESO problem: Generally, the minimization problems can be expressed in the following standard form:

$$\begin{aligned}
&\min f(\{x\}), \\
&g_j(\{x\}) \le 0 \qquad j = 1,\ldots,k, \\
&h_j(\{x\}) = 0 \qquad j = 1,\ldots,l, \\
&x_j^{l} \le x_j \le x_j^{u} \qquad j = 1,\ldots,m,
\end{aligned} \tag{3-13}$$

where $f(\{x\})$ is the objective function and {x} is the vector of the design variables. $g_j(\{x\})$ and $h_j(\{x\})$ refer to the inequality and the equality constraints, respectively. k and l are the numbers of the inequality and equality constraints. x^u_j is the upper and x^l_j the lower bound of the design variable x_j. m denotes the total number of design variables [17].

In the numerical examples, the design spaces are modelled by using the FE plane stress elements. The design variables of the problems are not specified. Since in ESO some elements are removed from the design domain, i.e. the corresponding stiffness matrices are zeroed out, the values of the design variables have to affect the element stiffness matrices. Consequently, element thicknesses or element elastic moduli can be chosen as the design variables [17].

3.5. Material Cloud Method (MCM)

In the *Material Cloud Method*, an optimal design is to be extracted from the distribution of material clouds and the design variables are the central positions and sizes of material clouds. The material cloud is a finite material patch with a constant relative density of material as in figure 3.4. It is assumed that the shape of a material cloud is a square for 2D problem or a cube for 3D problem. The basic idea of MCM is to free the material from the computational mesh, so that the material patches can be moved freely and independently with one another [4].

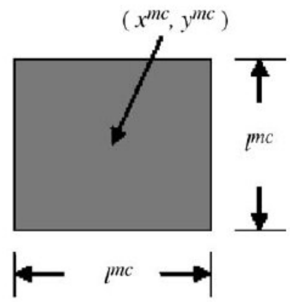


Figure 3.4 Material cloud (for 2D problem) [4]

There are three different ways of applying MCM for topology optimization. One is to optimize only central positions of material clouds, named MCMP. Another is to optimize only sizes of material clouds, named MCMS. The third is to optimize central positions of material clouds and subsequently to optimize sizes of material clouds, named MCMPS [4].

3.5.1. MCMP

In MCMP, material clouds can move around independently crossing the element-boundary and overlap one another under one condition that the sum of equivalent areas of material clouds contained in each element should not exceed the area of the element, which is an upper bound in physical sense [4].

An active element is defined as an element in which more than one material cloud is contained and an active node is defined as a node which constitutes the active element and an active DOF is defined as a DOF related to the active node. Figure 3.7 depicts the optimization procedure of MCMP [4].

The procedure of MCMP is as follows [4]:

(1) Step 1: Define a design domain and a sub-domain where material clouds are initially distributed (Figure 3.5(a)).

(2) Step 2: Allocate material clouds in all elements of the sub-domain (Figure 3.5(a)).

(3) Step 3: Modify positions of material clouds until a convergence condition is satisfied (Figure 3.5(b)–(d)).

(4) Step 4: Extract an optimal design form a converged distribution of material clouds (Figure 3.5(e)).

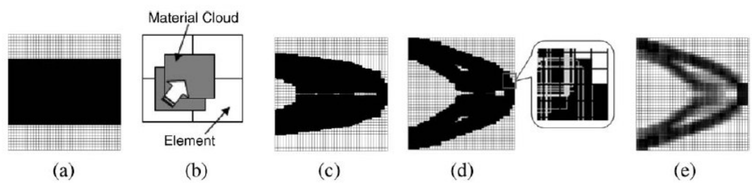


Figure 3.5 Optimization procedure of MCMP: (a) initial distribution of material clouds, (b) modification of positions of material clouds, (c) distribution of material clouds during iteration, (d) converged distribution of material clouds, and (e) distribution of equivalent densities of elements corresponding to (d) [4]

3.5.2. MCMS

In MCMS, a material cloud can grow and shrink only in the element where it is initially allocated, but cannot be vanished. Therefore, material clouds cannot be overlapped each other and the active design domain, which consists of all active elements, cannot be changed from initially defined whole design domain. Relative densities of material clouds are specified as unity and then initial areas of material clouds are determined similarly to the case of MCMP. Figure 3.6 depicts the optimization procedure of MCMS [4].

The procedure of MCMS is as follows [4]:

(1) Step 1: Define a design domain (Figure 3.6(a)).

(2) Step 2: Allocate material clouds in all elements of the design domain (Figure 3.6(a)).

(3) Step 3: Modify sizes of material clouds until a convergence condition is satisfied (Figure 3.6(b)–(d)).

(4) Step 4: Extract an optimal design from a converged distribution of material clouds (Figure 3.6(d)).

At Step 2, one material cloud is allocated in each element and the central position of the material cloud is identical to the center of the element. In figure 3.6(c), sizes of material clouds are varied in a range from a very small value to the size of an element. By MCMS, a clear optimal design configuration is obtained as shown in figure 3.6(d) [4].

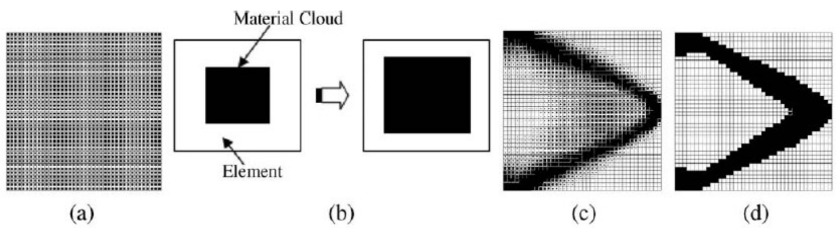


Figure 3.6 Optimization procedure of MCMS: (a) initial distribution of material clouds, (b) modification of sizes of material clouds, (c) distribution of material clouds during iterations, (d) converged distribution of material clouds [4]

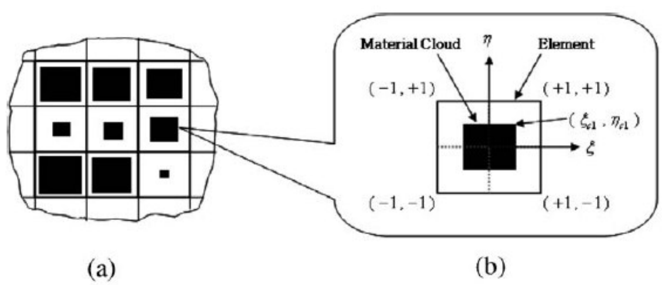


Figure 3.7 Material cloud domain and integration domain in MCMS: (a) material cloud domain and (b) integration domain for one material cloud [4]

Ω^{MCD} is the material cloud domain, which is the domain occupied by all material clouds (black region in figure 3.7(a)). It is assumed that the traction surface, Γ_t, the traction, $\boldsymbol{t}$ and the body force, $\mathbf{r}$ are independent on the design.

The optimization procedure of MCMS may seem to be similar with that of the traditional density approach like SIMP (solid isotropic material with penalization; Bendsoe and Sigmund, 2003). But due to the difference of optimization concepts, even though the same optimization algorithm is applied, the convergence and final results are quite different from those of SIMP [4].

3.5.3. MCMPS

In MCMPS, MCMP and MCMS are sequentially applied. Fig. 8 depicts the optimization procedure of MCMPS.

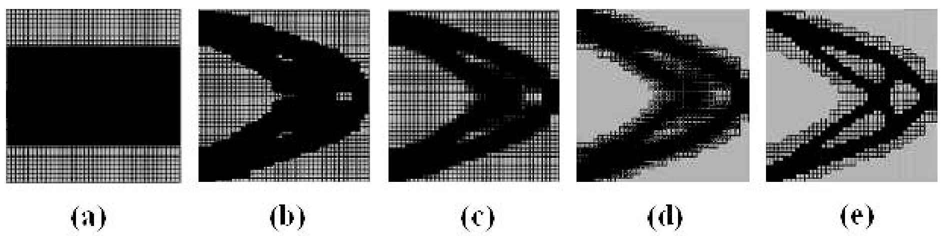


Figure 3.8 Optimization stages of MCMP [4]

In MCMPS, the design domain can be naturally expanded and reduced through movements of material clouds by MCMP and a clear resulting topology can be obtained with fast convergence by subsequent MCMS (Chang and Youn) [4].

3.6. Homogenization Method

The method is based on the modeling of a perforated material constructed from a basic unit cell consisting at a microscopic level of material and void and on the description of the structure by using a continuously varying distribution of the material density computed by invoking the formulas of the homogenization theory [20].

3.6.1. The homogenization equations

The goal of elastic homogenization is to determine the effective stiffness tensor C^H of a material, where the constitutive equation for linear elastic behavior is assumed:

$$\sigma = C^H \varepsilon \tag{3-14}$$

where σ and ε are the stress and strains fields, respectively [20].

As developed in Bensoussan et al. (1978) and Sanchez-Palencia (1980), the effective elasticity tensor of a periodic material can be expressed in energy form in the following manner:

$$C^H_{pqrs}\varepsilon^{o(kl)}_{pq}\varepsilon^{o(ij)}_{rs} = \frac{1}{|Y|}\int_Y C_{pqrs}(\varepsilon^{o(kl)}_{pq} - \varepsilon^*_{pq}(x^{kl}))(\varepsilon^{o(ij)}_{rs} - \varepsilon^*_{rs}(x^{ij}))dY \tag{3-15}$$

where C_{pqrs} is the elasticity tensor of the solid (matrix) material, $\varepsilon^{o(kl)}_{pq}$ are the test strain fields, $\varepsilon^*_{pq}(x^{kl})$ are the fluctuation strains caused by the inhomogeneous base cell defined through the strain–displacement relations

$$\varepsilon^*_{pq}(x^{kl}) = \frac{1}{2}\left(\frac{\partial x^{kl}_p}{\partial y_q} + \frac{\partial x^{kl}_q}{\partial y_p}\right) \tag{3-16}$$

and the displacement fields x^{kl} is found through solution to the following base cell problem:

$$\int_Y C_{ijpq}\frac{\partial x^{kl}_p}{\partial y_q}\frac{\partial \upsilon_i}{\partial y_i}dY = \int_Y C_{ijpq}\varepsilon^{o(kl)}_{pq}\frac{\partial \upsilon_i}{\partial y_i}dY \qquad \forall \upsilon \in \tilde{V} \qquad \tilde{V} = \{\upsilon : \upsilon \text{ is } Y\text{-periodic}\} \tag{3-17}$$

The test strain fields $\varepsilon^{o(kl)}_{pq}$ are chosen as unit vectors and symmetry is used to reduce the number of test strain fields in three-dimensional elasticity from nine to six (three normal and three shear strain cases) [20].

The homogenization is performed numerically using finite element analysis. The base cell is discretized and, after integration over each element and substituting the definitions of the unit strain tensors into the left-hand side of Eq. (3-15), the homogenized elasticity tensor is written using standard finite element notation as

$$C_{ijkl}^{H} = \frac{1}{|Y|}\sum_{e\in Y}(\mathrm{d}_o^{e(ij)} - \mathrm{d}^{e(ij)})^T \mathrm{k}^e(\rho^e)(\mathrm{d}_o^{e(kl)} - \mathrm{d}^{e(kl)}) \tag{3-18}$$

where $\mathrm{k}^e(\rho^e)$ is the stiffness matrix of element e expressed as a function of the element volume fraction ρ^e, $\mathrm{d}_o^{e(ij)}$ is the vector of nodal displacements for element e corresponding to the unit test strain field $\varepsilon^{o(ij)}$, and $\mathrm{d}^{e(ij)}$ is the vector of nodal displacements for element e related to the strain field $\varepsilon^*(\mathrm{x}^{ij})$. The displacements $\mathrm{d}^{(ij)}$ are unknown and are found by solving the matrix problem

$$\mathrm{K}(\rho^e)\mathrm{d}^{(ij)} = \mathrm{f}^{(ij)} \qquad \mathrm{d}^{(ij)} \text{ is } Y\text{-periodic} \tag{3-19}$$

where $\mathrm{K}(\rho^e)$ is the global stiffness matrix assembled ($\underset{e}{A}$) from the element stiffness matrices and the nodal forces $\mathrm{f}^{(ij)}$ result from the unit test strain field (ij) and are computed by

$$\mathrm{f}^{(ij)} = \underset{e}{A}\,\mathrm{k}^e(\rho^e)\mathrm{d}_o^{e(ij)} \tag{3-20}$$

Note that a uniform distribution of material yields a zero nodal force vector and consequently a zero nodal displacement vector $\mathrm{d}^{(ij)}$. The effective stiffness then equals the stiffness of the matrix material [20].

Boundary conditions are applied to the base cell boundaries to prevent rigid body motion and impose the unit strain field. In the two-dimensional case, for example, displacements in the direction normal to the boundary are constrained for the normal test strain fields, and in the direction parallel to the boundary for the shear test strain field. [20]

3.6.2. Inverse homogenization problem formulation

Homogenization theory allows us to compute effective properties of the bulk material given the topology of a base cell. Formulate and solve an inverse homogenization problem: find the optimal base cell topology that yields desired effective properties. The characteristic base cell is the design domain and a material distribution problem must be solved. As in traditional structural optimization, each element in the base cell possesses a volume fraction ρ^e, or relative density, that indicates what phase is present in the elemental domain, where $\rho^e = 1$ represents material present (*e* is a solid element) and $\rho^e = 0$ represents fluid present (*e* is a void element). Connectivity of the solid elements defines topology, which in turn dictates stiffness and fluid velocities. Therefore, ρ^e is traditionally the design variable (Bendsøe and Sigmund, 2003) [20].

The multi-objective approach allows the designer to assign different weights, or measures of importance, to the stiffness and transport terms in the objective function according to the materials intended future use. These weights then influence the final design. The topology optimization problem is formulated with constraints including prescribed elastic and fluid flow symmetries, the homogenization equations, lower (VL) and upper (VU) bounds on the available volume of material, and bounds on the element volume fraction ρ^e.

3.7. Optimality Criteria Method (OCM)

Optimality criteria method used in structural optimization, the simple structure of the continuum, single load problem can be utilized to generate extremely efficient computational update schemes for solving the problems. Iterative methods which, for a previously computed design and its associated displacements, update the design variables at each point independently from the updates at other points, based on the necessary conditions of optimality [25].

The objective of optimization process is to determine the stiffest possible structure for a given domain, amount of material, load distribution and support conditions by minimizing the compliance of the system [26].

The problem can be expressed as [25-26]:

$$\min_x : c(x) = U^T K U = \sum_{e=1}^{N} (x_e)^p u_e^T k_0 u_e = \sum_{e=1}^{N} c_e(x) \tag{3.21}$$

Subject to [25-26]:

$$\begin{aligned} &\frac{V(x)}{V_0} = f \\ &KU = F \\ &0 < x_{\min} \le x \le 1 \end{aligned} \tag{3.22}$$

where U and F are global displacement and force vectors, respectively, K is the global stiffness matrix, u_e and k_0 are the element displacement vector and stiffness matrix, respectively; x is the vector of design variables, and x_{min} is a vector of minimum relative densities (nonzero to avoid singularity). N is the number of elements, p is the penalization power, while $V(x)$ and V_0 are material volume and design domain volume respectively, and f is the prescribed volume fraction [26].

Optimality criteria method and heuristic design-updating scheme can be written like this [25-26]:

$$x_e^{new} = \begin{cases} \max(x_{\min}, x_e - move) & x_e B_e^{\eta} \le \max(x_{\min}, x_e - move) \\ x_e B_e^{\eta} & \max(x_{\min}, x_e - move) \le x_e B_e^{\eta} < \min(1, x_e + move) \\ \min(1, x_e + move & \min(1, x_e + move) \le x_e B_e^{\eta} \end{cases} \tag{3.23}$$

where *move* is a positive move limit and $\eta = 0.5$ is a damping co-efficient. B_e is found from the optimality condition [25-26]:

$$B_e = \frac{-\dfrac{\partial c}{\partial x_e}}{\lambda \dfrac{\partial V}{\partial x_e}} \tag{3.24}$$

The Lagrange multiplier λ is obtained using the bisection iterative method. The numerator is the sensitivity of the objective function with respect to the design variable, and is found as [25-26]:

$$\frac{\partial c}{\partial x_e} = -p(x_e)^{p-1} u_e^T k_0 u_e \tag{3.25}$$

Filtering technique works by modifying the element sensitivities as follows [25-26]:

$$\left(\frac{\partial c}{\partial x_e}\right)_{filtered} = \frac{1}{x_e \sum_{f=1}^{N} (r_{\min} - dist(e,f))} \sum_{f=1}^{N} (r_{\min} - dist(e,f)) x_f \left(\frac{\partial c}{\partial x_f}\right) \tag{3.26}$$

The original sensitivities are thus averaged over a circular area with centre located at the centre of the corresponding element (*e*) and radius r_{min}. These filtered sensitivities are used in the optimality criteria updating process [26].

In this chapter, some mostly used topology optimization methods, which are reviewed in chapter 2, are explained. The ERM is not explained in this chapter, because the method is developed and used for topology optimization in this study. Hence details of the method are preferred to be explained in a separate chapter.

CHAPTER 4

ELEMENT REMOVAL METHOD

4.1. Introduction

The application of topology optimization to industrial problems is not yet widespread. The main reason is that, as an iterative process, topology optimization of large real world structures is computationally intensive. Thus, there is a need to find computationally efficient ways to perform the topology optimization of complex structures with large number of degrees of freedom. One way to address this problem is using efficient discretization techniques like boundary element method and meshless techniques. DeRose Jr. and Diaz developed a meshless fictitious domain method based on wavelet basis and Galerkin scheme to solve computationally intensive 3D topology optimization problems. Maar and Schulz used multi-grid interior point approach for solving large topology optimization problems. All of the above approaches are based on reducing the number of floating point operations needed for solving the topology optimization problem.

Another approach is to use a faster finite element solver, either by using different element formulations, like p-version finite elements or by using a faster equation solver, like different iterative solvers. Reducing the total number of analyses through heuristic techniques like re-analysis can reduce the solution time as well.

Another approach is to increase the computational power of the system through parallel computing. Borrvall and Petersson considered topology optimization of 3D domains using parallel processing. In their work, they used the so-called regularized intermediate density control method to ensure the existence and uniqueness of the solution and to obtain black-and-white final layouts. In their method in order to enforce a black-and-white solution a penalty parameter should be calculated for each problem, which needs some numerical experiment adding to the complexity of the method. Their study showed the effectiveness of the preconditioned conjugate gradient solver for solving parallel topology optimization problems. Arash Mahdavi, Raghavan Balaji, Mary Frecker, Eric M. Mockensturm (2005) studied a simple and efficient parallel computing technique for solving large-scale topology optimization problems. They used a much simpler approach based on power law technique. An optimality criteria method has been employed for updating the design variables.

The methods to make the topology optimization faster are generally valid for experts and they need development of new methods or softwares. From point of designer use of topology optimization for complicated parts is not possible due to difficulty of programming and time necessary to solve the real-case problems.

To find accurate solution of topology optimization designer must increase number of iteration and element. When number of iteration and element are increased solution time also increased extremely. For this reason, in the optimization processes, passive elements (which are found from FE analyses) are eliminated after every iteration loop. In the next iteration loop, unnecessary elements will not be used during solution. Hence at each progressive solution loop the number of elements are decreased.

In figures below, the shamatic view of the element removal operation can be observed: Figure 4.1 shows design domain for the given loading and boundary conditions.

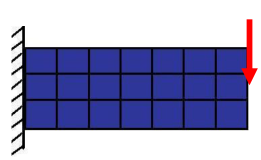


Figure 4.1 Design Domain

Application of topology optimization to the design domain yields optimized shape of figure 4.2. Then, inefficient elements are deleted after the first loop and new design domain can be obtained as given in figure 4.3.

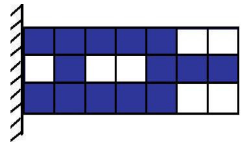


Figure 4.2 Optimized Shape

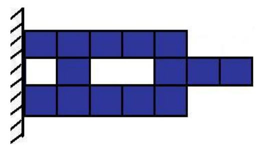


Figure 4.3 Optimized Shape after Element Removal

A topology optimization tool exists in Ansys. In Ansys, optimality criteria algorithm is used for topology optimization. Optimality criteria depend on minimization of compliance. Elements' density values are changed systematically from 0.001 to 1 in every iteration loop. If an element's density goes to 0.001 that means the element, which do not carry load, is unnecessary and can be removed from the design domain. If an element's density goes to 1 that means the element, which carry load, will be included in optimized domain. For the load and boundary condition given in figure 4.4 application of topology optimization tool yield the optimized domain of figure 4.5 where 50 % of the initial material is used. In figure 4.5 blue zones are indicating material to be removed while still elements are used at these zones.

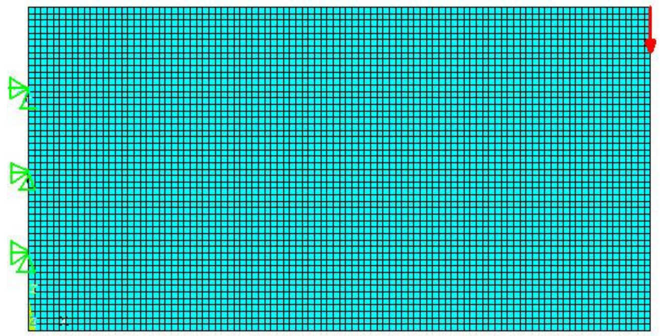


Figure 4.4 Design domain with load and boundary conditions

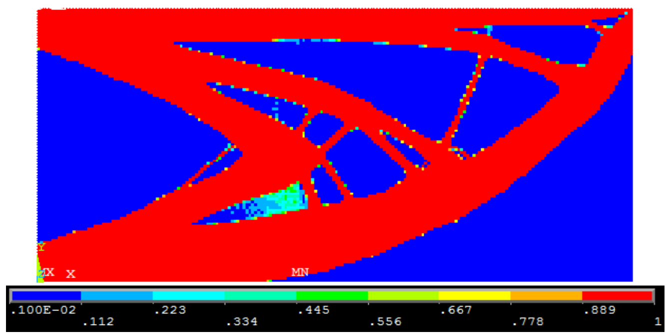


Figure 4.5 Optimized domain after topology optimization with Ansys

4.2. Comparison of Methods

Application of the topology optimization tool, defined above, to real world, requires high solution times. For example a machine part discritized into 100000 elements will require about 5 or 6 hours on a today's high performance computer. Due to this fact application of topology optimization is limited huge companies or institutes. High computation cost and requirement of high knowledge level to apply the method even on software prevents wide spreading of the topology optimization in industry. Decreasing necessary solution time and easing the use of method will make the use of method even for technicians in industry. Some methods are compared for the necessary solution time and tools are applied using Ansys FE solution. Basic tool to compare the results is Ansys topology optimization tool defined in previous section.

Ansys topology optimization tool is investigated to decrease the time consumption: A macro is developed that removes the elements with low stress value, which are selected form the optimized design domain at preceding iteration loop. This operation is applied periodically in optimization process. The outcome of the study is given in figure 4.6 which is completely same as original method's result. During optimization process after every element removal operation, element number is decreased. Therefore, the solution time is expected to be decreased. Solution times of original and modified methods are measured for a model of 20000 elements. Original method solves the problem in 22 minutes while the modified method needs 21 minutes to reach the final optimized domain. Hence it is observed that application of element removal to Ansys topology optimization tool is not as effective way as expected to decrease the optimization time.

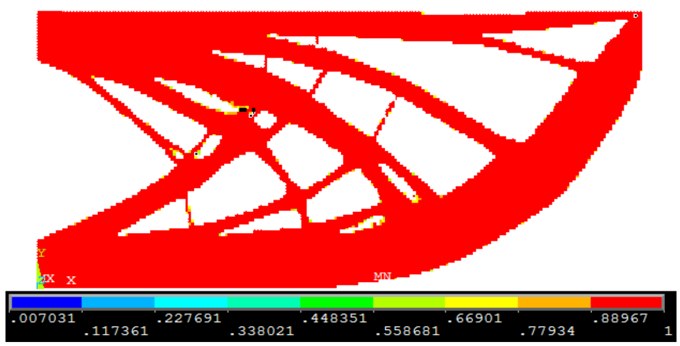


Figure 4.6 Optimized domain after topology optimization with element deletion

Application of different topology optimization algorithms are done afterwards. User Programmable Feature tool (UPFt) of Ansys is used to customize topology optimization process. With the use of UPFt programming is possible in Ansys. A macro is developed to apply the optimality criteria algorithm of which basics are defined by Sigmund 2001. At this step all optimization process is done using the developed macro. Application of the method to same 20000 element model yielded high optimization time. Result of the method is given in figure 4.7 for a small number of elements. As well as time problem, separation of the design domain was another problem of the method.

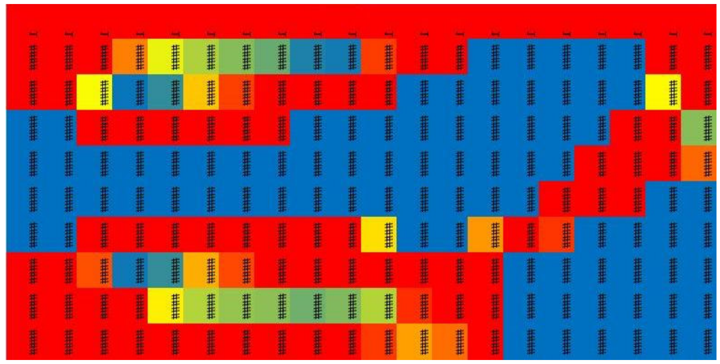


Figure 4.7 Optimized domain after customized Ansys

Level set method is another method which is applied using UPFt, method includes heavy mathematics. Application of the method yielded solution time more than twice of optimality criteria as expected due to equations involved.

Optimization problems are generally dealt with by constructing the mathematical model of the objective function. The number of the independent parameters used in mathematical model shows the degree of the mathematical complexity of the objective function. The model could be linear or non-linear depending on the nature of the problem with the probability of being constrained or unconstrained. As explained in chapter 3, methods given above are generally includes these mathematical models. Application of element removal method to FE solution is another algorithm for topology optimization. For this method Ansys is used for modelling and FEA of design domain. In this method a mathematical model is not optimized but the material is cropped due to applied criteria.

4.3. Element Removal Method

The main idea of the topology optimization is removing of inefficient (comparatively small stressed) elements from the design domain. Idea is directly applied for optimization. For selection of the elements to be removed, stress values are considered to be impressive factor. Stress values are calculated from FEA. FEA is applied on the design domain and after every FEA operation, elements, which have lowest stress values, and their nodes are deleted. Flow chart of the procedure is given in figure 4.8. Question during this operation was “How many elements will be deleted at each loop?”, that is decided from the total iteration number and the total volume reduction ratio? Amount of elements to be deleted is about 2% of available domain. After each element removal operation, the rest of elements and nodes are renamed to enumerate. Then these elements form the new design domain and a new optimization loop starts. These optimization cycles is stopped when one of the criteria’s is true: volume reduction ratio or pre-defined working stress value is reached. The optimization process requires less time for every next cycle because inefficient elements aren’t used in following FE solutions. Therefore, total optimization time is expected to decrease rapidly.

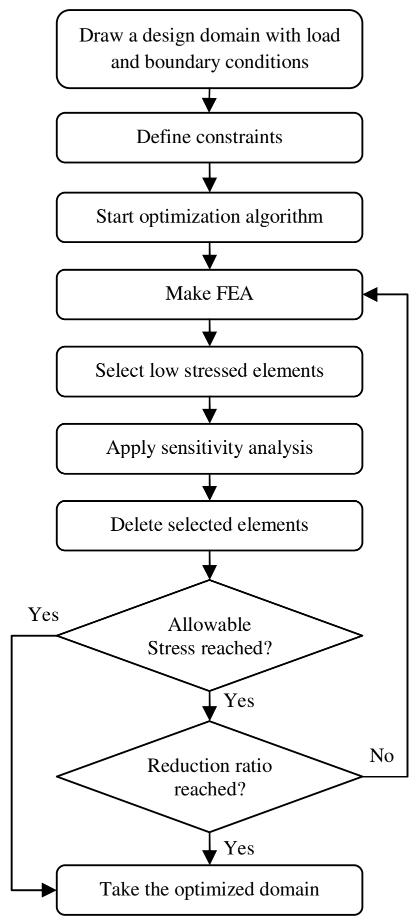


Figure 4.8 Element removal method procedure

Algorithm of Element Removal Method:

1. Draw design domain with load and boundary conditions

2. Define iteration number (*In*), volumetric reduction ratio (*Vr*), and constraints
   a) ***Iteration number (In)***: shows that volumetric reduction is reached in how many optimization loops, (if necessary)
   b) ***Volumetric reduction ratio (Vr)***: shows that how much material will be removed from design domain, (if necessary)
   c) ***Constraints***: may be stress and/or deformation. They limit the material reduction
   d) ***Element numbers***: initial element number ($elem_i$), last element number ($elem_l$)

3. Start optimization loop

3.1. Make FEA
- Calculate stress values of each element.
- Calculate selection number (*Sn*) and maximum selection stress value.
  Sn = Vr/In
- Sort elements depend on stress values.
- Find maximum selection stress value ($\sigma_{max,\ Sn}$) from sorted elements.

3.2. Start selection

3.3. Make sensitivity analysis
- Calculate selection coefficient (*k=0*) for each element

	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 $k_i = k_i + 1$

- Unselect elements depend on selection coefficient *(k)*

3.4. Remove selected elements

3.5. Check for constraint and reduction ratio
- If $\sigma_{max} > \sigma_{all}$ then stop optimization loop and go to 4
  Else continue optimization from 3

4. Take optimized model.

In figure 4.9, a design domain of 40x20 elements with loading and supporting conditions are given.

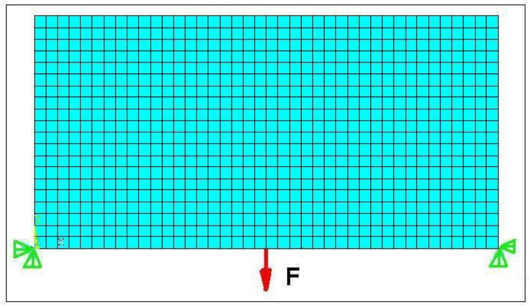
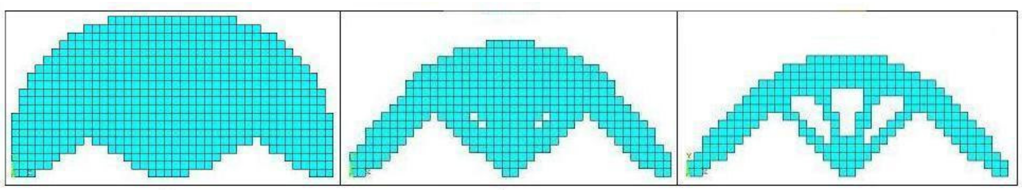


Figure 4.9 Design domain

Applying the element removal method, the design domain gets the shapes of figure 4.10 after first, 15th and last steps respectively. Relatively low stressed elements are removed from design domain periodically. And relatively high stressed elements are formed the last optimized domain.



a b c

Figure 4.10 Optimization steps

Below, some applications of the method are presented for simple problems.

For first example a design domain is modelled which is divided to 24000 elements and two bottom corners are fixed. A downward force of 1N is applied to middle of bottom edge. Model is given in figure 4.11 for which 65% volume reduction is selected.

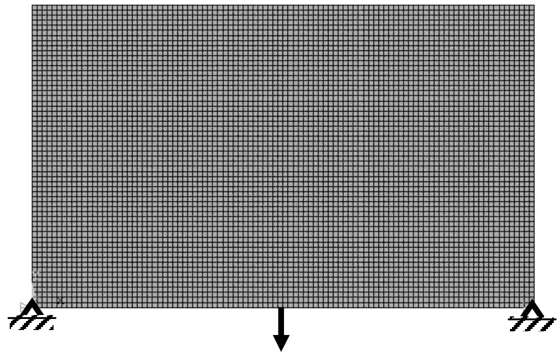


Figure 4.11 Design domain with 24000 Element

Applying the optimization process, optimized domain is obtained as given in figure 4.12. In figure 4.13, result of Ansys topology optimization tool is given for comparison. Same design criteria's are used for the Ansys optimization.

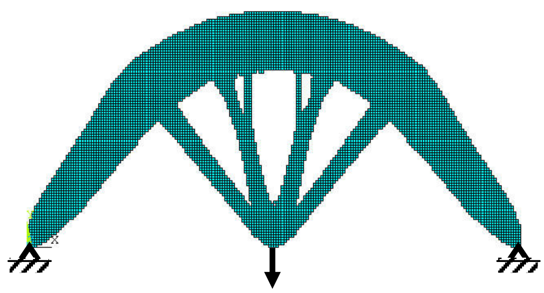


Figure 4.12 Optimized Domain with Element Removal Method

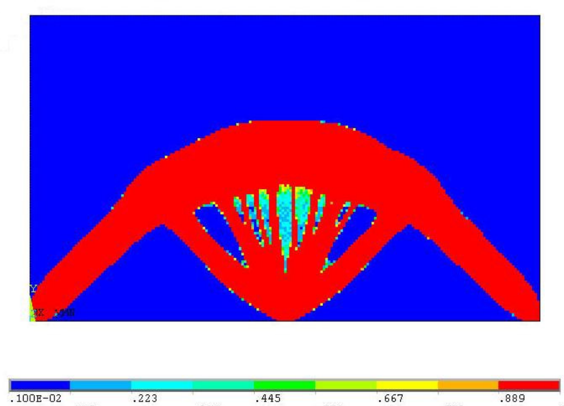


Figure 4.13 Optimized Domain with Ansys Topology Optimization Tool

Comparing the figures 4.12 and 4.13 similar outcomes are obtained. Developed algorithm is neater; less thin material zones are observed.

Simply supported cantilever plate is used as next sample design domain. Solution times are also compared for three alternatives available: element removal method, Ansys optimization, and Ansys optimization with element removal method [30].

Definition of problem: to remove 50% of materials from given design space safely, find the optimum topology at given boundary and loading conditions in allowed stress and deformation values.

A cantilever plate as shown in figure 4.14 is modelled. Size of design domain is 0.2*0.1*0.01 m^3 (w*h*t) and load is 1000 N. Ansys topology optimization parameters are as follows. Load case number is 1 as default value, percent volume reduction is 50%, solution approach is optimality criteria, convergence tolerance is 0.0001, and number of iteration is 40. Different element numbers are used such as 20000, 30000, 60000 and 100000.

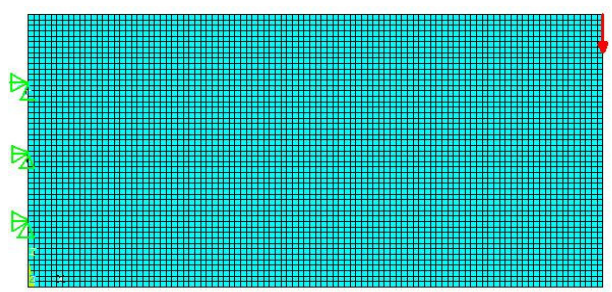


Figure 4.14 Design domain with loading and boundary conditions

Ansys optimization tool's result can be seen in the figure 4.15. In this figure, blue coloured elements are inefficient elements. Red coloured elements carry more loads from others. Elements are rated from 0.001 (blue) to 1 (red) which means an element rated as 0.001 carries lower load than other elements while elements rated as 1 carries higher load.

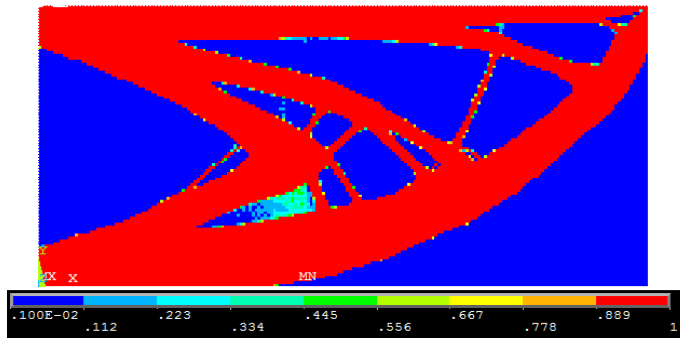


Figure 4.15 Optimized model with Ansys optimization

Result of Ansys optimization tool with element removal method is given in Figure 4.16. Blue coloured elements cannot be seen in the figure because they are removed from the design domain to decrease solution time. Hence inefficient elements are not used in following optimization loop. Aim of the procedure was to decrease solution time.

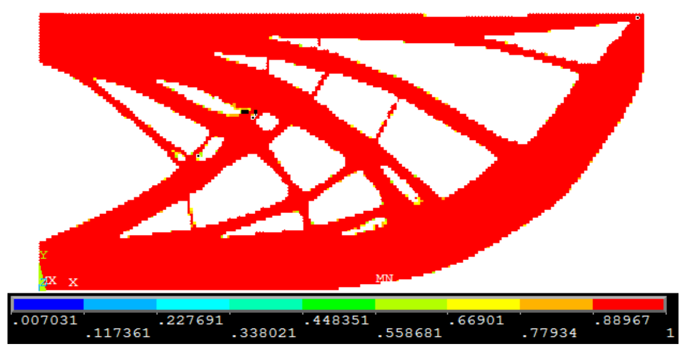


Figure 4.16 Optimized model Ansys optimization with element removal method

Result of element removal method is shown in Figure 4.17. Inefficient elements are removed from the design domain to decrease solution time.

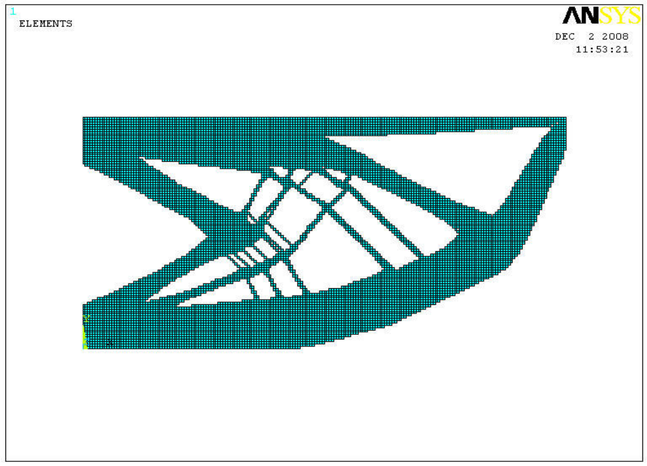


Figure 4.17 Optimized model with element removal method

Similar results are obtained for three methods which were element removal method, Ansys optimization tool and Ansys optimization tool with element removal method. Solution times for the three methods are plotted in figure 4.18 for different element numbers.

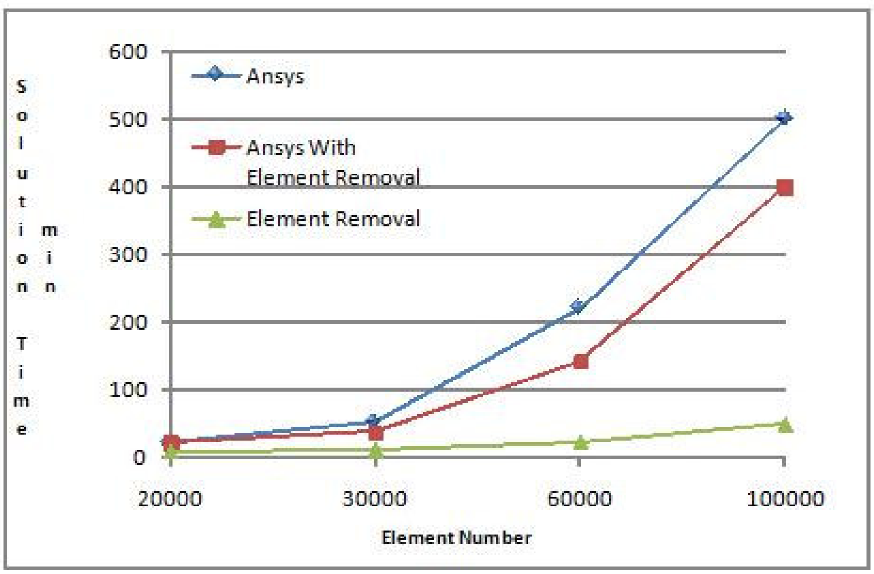


Figure 4.18 Solution times depend on solution method and element numbers

The solution times of the methods are also tabulated in table 4.1 for comparison. Tabulated time values are for a computer of 2.13 Ghz Intel Core2 Duo processor, 1GB RAM, and Windows XP SP3 operating system. Time reduction column of the table contains time reduction percent obtained by element removal method compared to Ansys optimization tool.

Table 4.1 List of solution times depend on solution method and element numbers

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%

Investigating the table, time cost of the topology optimization seems to be overcome. If 20000 elements are used time reduction is about 60 % while for increased number of elements time reduction reaches to 90 %. Time reduction ratio indicates effectiveness of the developed element removal method.

4.4. Results

In this chapter, element removal method is explained which is a method for topology optimization. The aim of the element removal method is initially demonstrated. Then the results and time requirement of this method are compared with Ansys topology optimization results.

Ansys FEA program is preferred for this study, because it is used very commonly in industrial applications. But the solution of topology optimization of this program takes very long time. To eliminate this difficulty an element removal algorithm is developed to use with the Ansys topology optimization tool. Although, the method yielded better result but was not adequate. Hence development of a macro is decided with the use of element removal method.

During development of the new macro some problems are encountered. These problems were mainly material discontinuity that is breaking of the domain, and forming of purity structure. A sensitivity analysis is developed to solve these problems.

Application of the element removal method yielded 60 to 90% decreased solution times depending on element numbers. When the outcomes of Ansys topology optimization and element removal method were compared generally similar results are obtained.

It is very important, for theoretical work, to support the output with experiments but this is not possible for now, since preparing an experimental set-up is very expensive and time consuming. The outputs of the developed macro, however, are compared with the literature in the next chapter.

CHAPTER 5

CASE STUDIES

Case studies are gathered under two headings. In the first part, developed macro's outcomes are compared with available literature. Aim of this section is to discuss validity of the macro.

In the second part some application examples are given from industry. In this part effectiveness of the topology optimization is underlined.

5.1. Comparison of Methods

In this section, element removal method is compared with other methods which are used in the literature. Three problems are studied for comparison. For first case, design domain is T-shaped with the vertical branch as shown in figure 5.1.

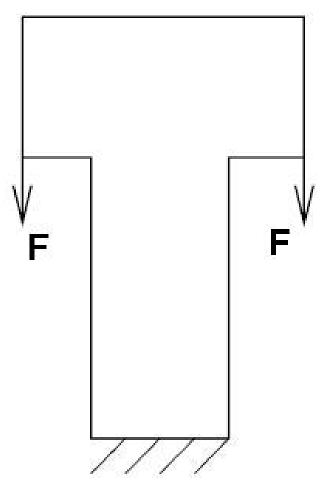


Figure 5.1 Design domain [27]

Element removal method's result of the design domain can be seen in figure 5.2. The domain is studied in literature and solution is obtained by application of Level Set Method of which result is shown in figure 5.3 [27].

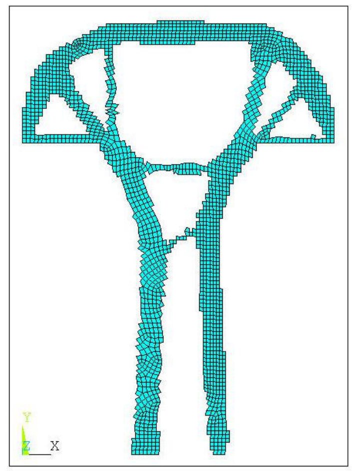


Figure 5.2 Optimization result of element removal method

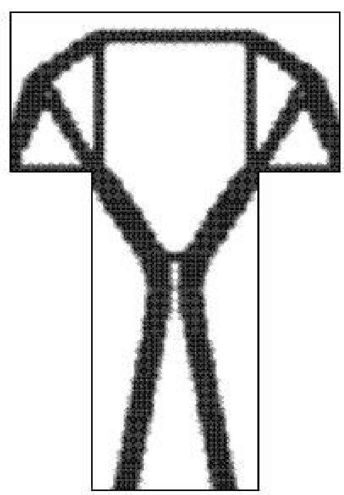


Figure 5.3 Optimization result of level set method [27]

Comparing figures 5.2 and 5.3, little difference is observed in the figures about the element distribution. Main reason of the difference is that the result at the figure 5.3 is optimized with shape optimization after topology optimization.

Second design domain is vertically loaded cantilever plate with both ends are fixed as shown in the figure 5.4. Element removal method's result of the design domain can be seen in the figure 5.5. Optimality Criteria Method is used to solve the problem by researchers and result of reference is shown in figure 5.6 [28].

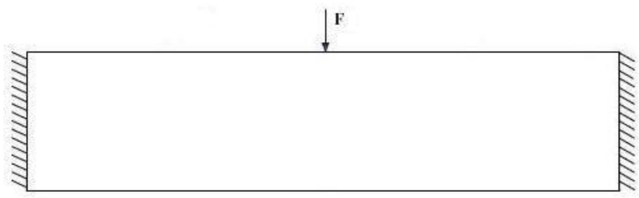


Figure 5.4 Design domain [28]

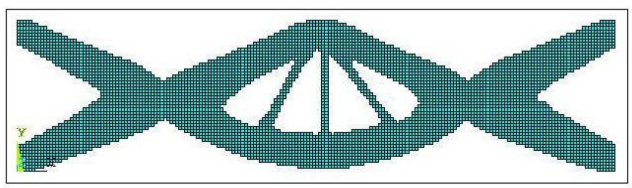


Figure 5.5 Optimization result of element removal method

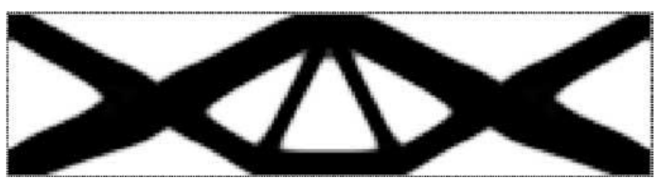


Figure 5.6 Optimization result of OCM [28]

Investigating the results again similar results are obtained for both methods.

The last design domain is also vertically loaded plate with both bottom corners are fixed as shown in the figure 5.7. Element removal method's result of the design domain can be seen in the figure 5.8, and Level Set Method's result is shown in the figure 5.9 [29].

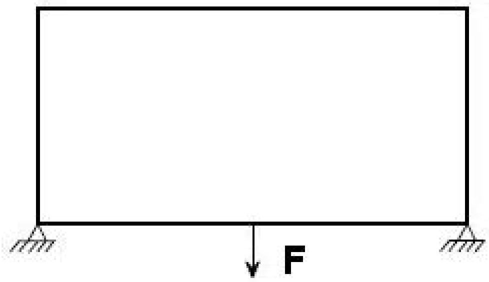


Figure 5.7 Design domain [29]

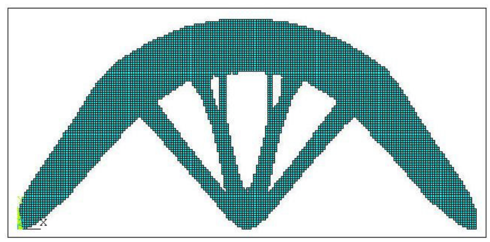


Figure 5.8 Optimization result of element removal method

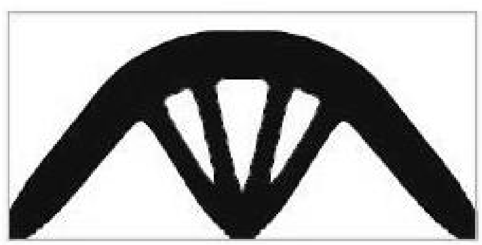


Figure 5.9 Optimization result of level set method [29]

In the above studies, the results of element removal method are compared with literature studies. The compared results are nearly same with the referenced results. Solution times of the referenced results are not given so the solution times cannot be compared.

5.2. Arm of a Wheel Loader

In this study, arm of a wheel loader (Figure 5.10) is optimized which is working recently. For this case Ansys's optimization results and developed macro's results are compared [31].

Figure 5.10 Arm of wheel loader with bucket

Original arm is modelled and joint loads are applied to the model. Then, the stress and deformation results are calculated using Finite Element Analysis (FEA). Ansys program is used for FEA.

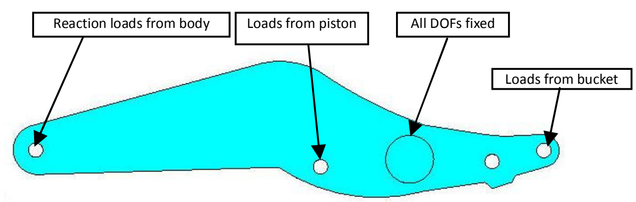


Figure 5.11 Original model

Loads are applied for two different position of arm: first one is bucket excavate ground and second one is excavating top position of the arm. Stress analysis results for these two loading conditions are shown in figure 5.12 and 5.13.

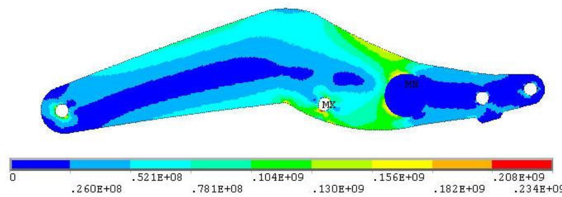


Figure 5.12 Von-Misses stress distribution in first loading conditions

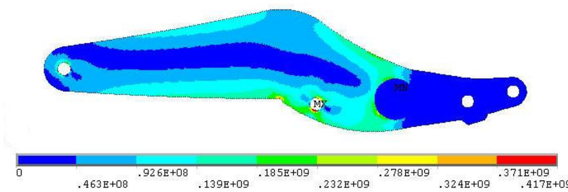


Figure 5.13 Von-Misses stress distribution in second loading conditions

After determining stresses and deformations of arm, a new model (Figure 5.14) is drawn roughly. All loads are applied on new model same as original model. This new model is used as design domain of topology optimization. Element removal methods are applied on the design domain.

Definition of problem: to remove maximum of materials from given design space safely, find the optimum topology at given boundary and loading conditions without pass the allowed stress and deformation values.

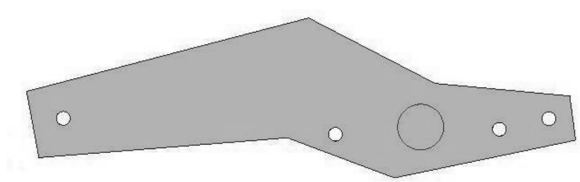


Figure 5.14 Design domain

Optimization algorithm is applied on two position of the design domain. Then, these two optimized model (Figure 5.15 and 5.16) are combined to form the optimized domain (Figure 5.17).

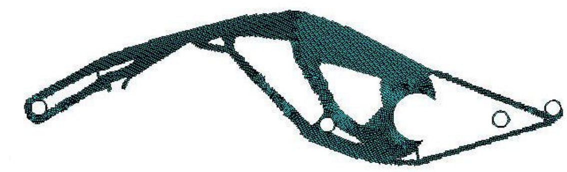


Figure 5.15 Optimized domain with first loading conditions

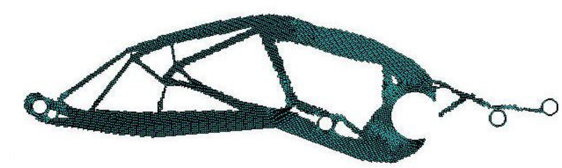


Figure 5.16 Optimized domain with second loading conditions

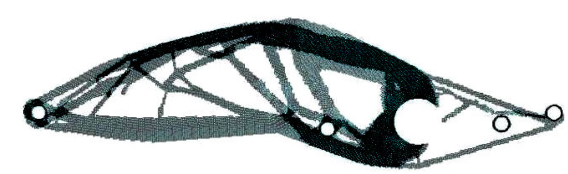


Figure 5.17 Combination of two optimized domains

But this model (Figure 5.17) is not useable as shown; it gives an idea to designer about the optimal shape. Designer must apply some improvements on this model to obtain producible one. After these improvements last model maybe as shown in Figure 5.18.

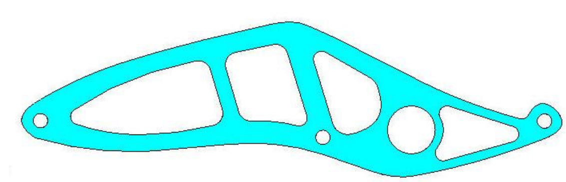


Figure 5.18 Last model after improvements

After the optimization, original model (Figure 5.11) and optimized model (Figure 5.18) are compared. Therefore, boundary and loading conditions of original model are applied on the optimized model. Then, stress and deformation results are compared (table 5.1).

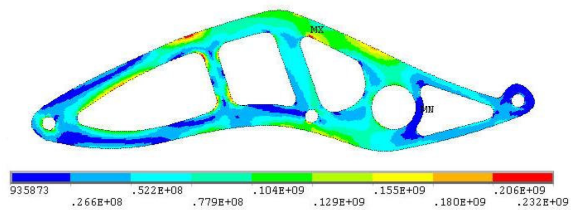


Figure 5.19 Von-Misses stress distribution in first loading conditions

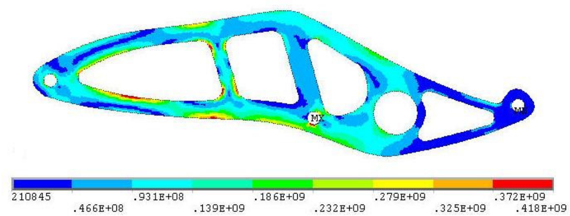


Figure 5.20 Von-Misses stress distribution in second loading conditions

Table 5.1 Comparison of original and optimized models

	Max. Stress	**Max. Deformation**	**Element No**	**Element Reduction %**
Original model	417 MPa	0.010 m	17191	
Optimized model	418 MPa	0.011 m	10832	37

In table 5.1, we can see results of original and optimized model. After the optimization process 33% reduction is shown on element number. In all models, same dimensions are used for elements, so we can say that 37% volume reduction is obtained. Maximum deformation value is increased 0.1% as shown on the table while that is 0.24% for the stress value for 37 % volume reduction.

5.3. CNC Router Plate

A plate is designed to carry cutting system of CNC router. It will be used for new wood carving workbench design. So there isn't any original model for comparison. Firstly, an initial rough model (Figure 5.21) is drawn with connection places and restricting geometry. Then FEA is applied to this initial model for stress and deformation results.

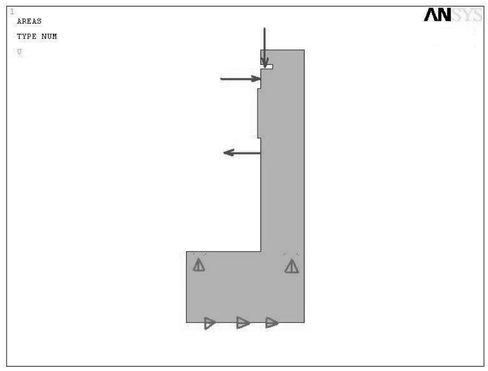


Figure 5.21 Design domain

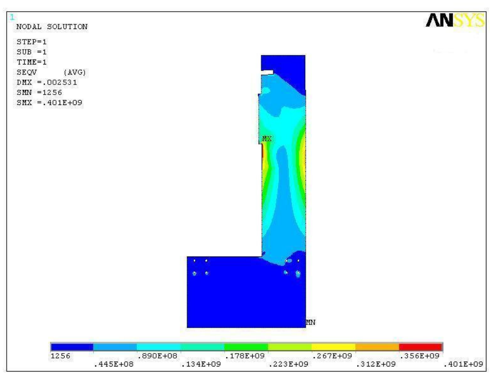


Figure 5.22 Von-Misses stress distribution in first loading conditions

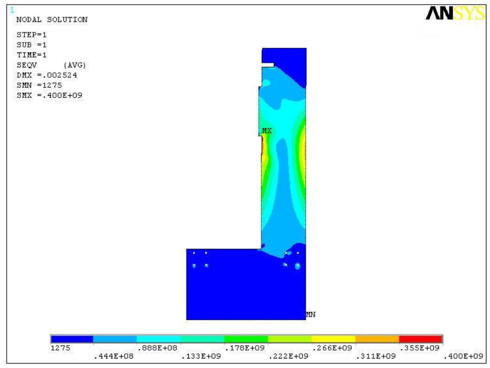


Figure 5.23 Von-Misses stress distribution in second loading conditions

Then Element removal method is applied to obtain the optimum topology. In the below figures optimization results are represented (Figure 5.24 and 5.25).

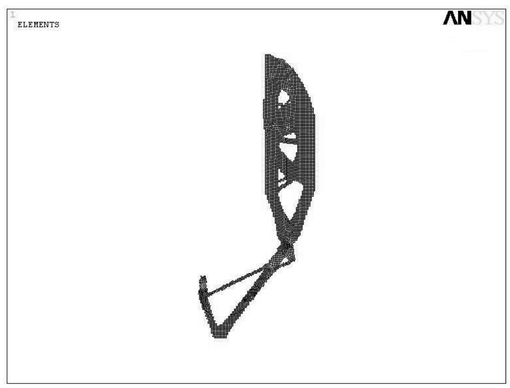


Figure 5.24 Optimized domain with first loading conditions

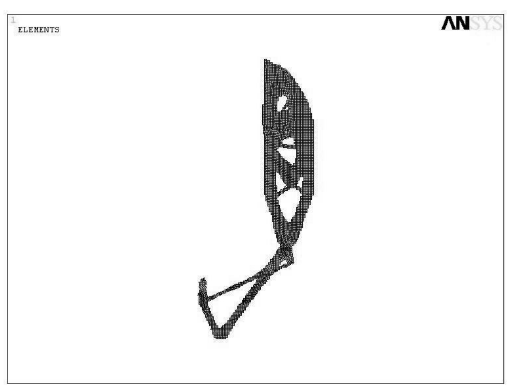


Figure 5.25 Optimized domain with second loading conditions

Arranging of element removal result yields the final model (Figure 5.26). In table 5.2, results of original and optimized model are included. The element number is reduced nearly 54% after optimization process. In all models, same element dimensions are used so we can say that 54% volume reduction is occurred. Almost halving the material, maximum deformation value is increased only 0.01% and stress value is increased 2.5%.

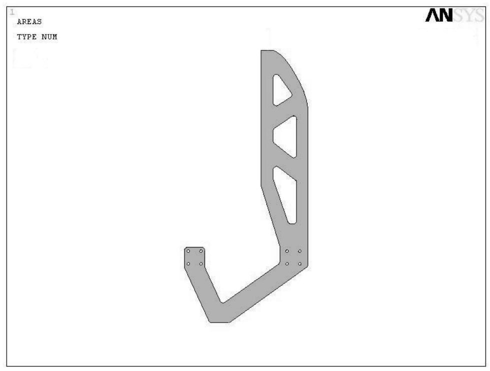


Figure 5.26 Last model after improvements

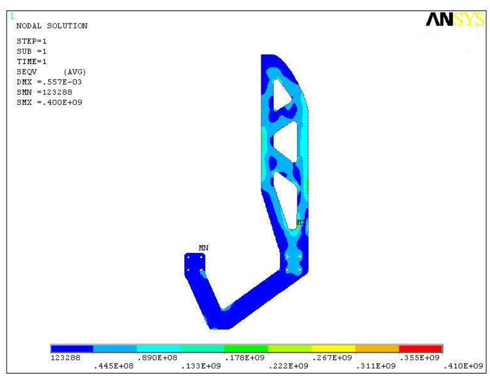


Figure 5.27 Von-Misses stress distribution in first loading conditions

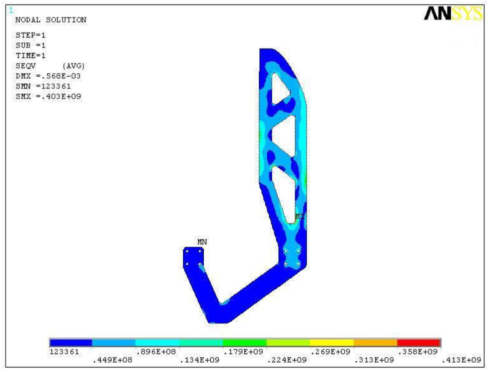


Figure 5.28 Von-Misses stress distribution in second loading conditions

Table 5.2 Comparison of original and optimized models

	Max. Stress	**Max. Deformation**	**Element No**	**Element Reduction %**
Original model	401 MPa	0.0025 m	9910	-
Optimized model	410 MPa	0.0026 m	4537	54

5.4. Fixation Plates

Advantages of topology optimization to fixation plates are indicated in this part [32]. Topology optimization is applied to three types of plates which are used in upper tibial osteotomy. Osteotomy was fixed by three types of plates and with their four types of combination which were supported by wedges with the height of 10 mm (TR-2002 02021Y-Hipokrat Corp., Izmir/Turkey) and six holed anatomical “T” plates without wedge (Hipokrat Corp., Izmir/Turkey) as shown in Figure 5.29.

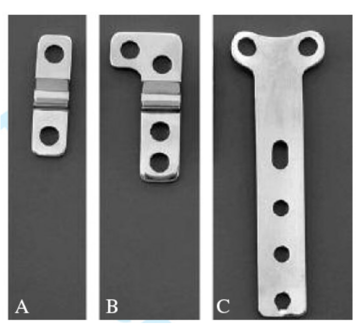


Figure 5.29 Appearance of the internal surface of different plates

A: Rectangular shaped two-holed plate with wedge.

B: Reversed "L" shaped four-holed plate with wedge.

C: Anatomical "T" plate with 6 holes.

The plates are modelled as shown in figure 5.30. Axial compressive load in the knee of an adult during single limb stance of fast walking was stated to be about 5600 N. In present study, load value for design and optimization of plates is taken as 8000 N. During stress analyses of initial forms of plates, the ones with wedge are fixed from the wedges and 8000 N load is distributed to holes in compressive direction, simulating the body load via screws. For the anatomical T plate, loads are applied from the two holes at the top and two holes at the bottom of the plate while two mid-holes are used to fix the plate.

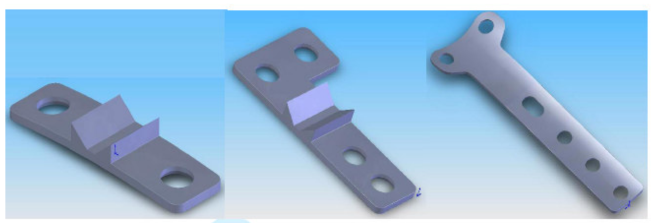


Figure 5.30 Models of plates

Results of analyses are given in table 5.3. In the table, maximum values of von-Misses stresses and deformation are given for each plate for the given load value.

Table 5.3 Analyses results for the plates

Plate	**Max. Stress**	**Max. Deformation**
A	1.7 MPa	0.05 μm
B	2.7 MPa	0.23 μm
C	2.1 MPa	0.27 μm

Analyses indicate that all the plates are very stiff and amount of deformation under even a high load value is very small. Also the von-Misses stress values are very low.

The initial forms of plates, given in figure 5.30, are defined as the design domain at the beginning of optimization process. Some design parameters; like position, number and diameters of holes, size and shapes of wedges, are considered as functional requirements of plates and no change is done on these features during the optimization loops. Plates A and B contains wedges, considering the wedges as design requirement this region is not included to optimization process, remaining parts of plates can be considered as plane stress problem with constant thickness. The design domain and FE model are given in figure 5.31 for the plates. In the figure, the plates are plotted after mesh. Elements used in the models are 8-noded structural and quadrilateral plane elements. Number of elements and nodes for the models A, B and C are given in table 5.4.

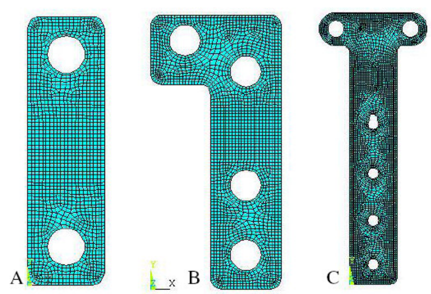


Figure 5.31 Meshed models of plates

Table 5.4 Number of elements and nodes used to design plates

Plate	No. of Nodes	No. of Elements
A	17410	5641
B	26165	8468
C	55799	18206

Topology Optimization is applied to plates with the criteria of 50 % Ω. Each plate is limited to allowable stress value of 5 MPa. After optimization, the plates are re-shaped as shown in figure 5.32.

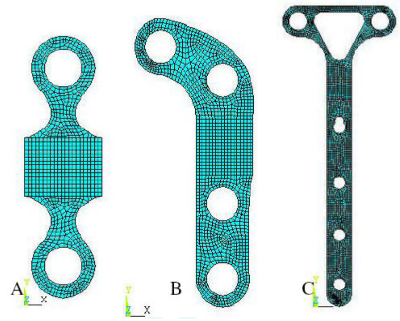


Figure 5.32 Plates after topology optimization

Effect of topology optimization is observed as reduced mass. Number of elements used for meshing of models after optimization and element number reduction ratio are given in table 5.5. Comparatively to initial shape, increased deformation and stress values are other effects to observe. New values for stress and deformation are given in table 5.6 for all plates. Highest mass reduction ratio is observed at plate C, T-plate, with a value of 50 %. Even using half of the material of initial shape, still stress and deformation values are low as seen in table 5.6.

Table 5.5 Number of elements of plates after optimization

Plate	**No. of Elements**	**Element Reduction %**
A	3227	42
B	5435	36
C	9044	50

Table 5.6 Stresses and deformations of plates after optimization

Plate	**Max. Stress**	**Max. Deformation**
A	3.7 MPa	0.2 µm
B	3.8 MPa	0.5 µm
C	4.1 MPa	0.7 µm

Stress values are increased on the plates as expected. But the new stress values are still low and hence plates are safe for operation. Stress distribution on the optimized plates is given in figure 5.26 for the three plates.

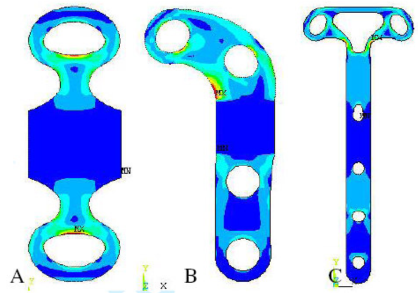


Figure 5.33 Von-Misses stress distribution on the plates after topology optimization

CHAPTER 6

CONCLUSIONS

Topology optimization is very useful tool for designer at design stage. Although application of this tool is difficult and solution times are very long. Therefore, the application of topology optimization for industrial problems is not widespread. Element removal method, which is explained in this study, is developed to eliminate these difficulties. A macro is developed to use with Ansys package for topology optimization of planer parts.

Several macros with different algorithms are developed during the study. Main criteria was time cost of each method. Firstly, an element removal algorithm is adopted to Ansys topology optimization tool. With this algorithm solution times are decreased approximately 10 to 20%. Resultant domains are compared with Ansys topology optimization tool results. They are observed to be almost same. After that initial step, different optimization methods are tried for less time consumption. UPFt of Ansys is used for applying other methods into Ansys. Optimality criteria update is added into UPFt. After application of this method some problems of optimality criteria is faced. Use of this criteira yields very long solution times. Hence element removal method is decided to work on. The developed macro has no relation with the geometric model. It can work on any model irrespective of kind of loading or type of constraints. Also the developed macro can create holes on design domain if necessary. The main advantage is solution time that is decreased approximately 90%. The resultant domains of developed macro are compared with some results available in literature. It is observed that the outcomes are almost same which indicates effectiveness of the method (keeping the solution time in mind).

The method is developed in Ansys, which is used commonly in industry for design and analyses processes. In addition, the developed algorithm can easily be adapted to other FEA programs. For these reasons, the method can be used very commonly.

The applications, which are presented in chapter 4 and 5, show that the method is very useful. Solution times can be decreased by use of the proposed method.

FUTURE WORKS

Present study is a pioneering work about topology optimization in Mechanical Engineering Department of the University. Efficiently use of the method is realized by present study.

To construct a complate topology optimization macro some topics must also be considered:

1. After the optimization process some thin links are arised in the resultant domain. These links are useless and gotten difficult to production of domain. In the next stage of this study, improvement of the continuity analyse is considered to eliminate these thinner links from the resultant domain.

2. In this study, optimized model is produced from optimized domain by designer. Developing an algorithm to automize the structural optimization of resultant domain is another goal.

3. The algorithm, which is described in this study, is produced for two dimensional problems under static loads. In the next stage, the algorithm is considered to be adopted for three dimensional problems. Also fatigue loading will be included for completeness of the topic.

4. Another planned work is to add a graphical user interface to developed optimization algorithm to be able to make a useful and widespread macro.

REFERENCES

[1] Joseph Edward Shigley (1986). *Mechanical engineering design*. (1^{st} ed.), McGraw-Hill Book Co.

[2] David G. Ullman (2003). *The mechanical design process*. McGraw-Hill Book Co.

[3] Garret N. Vanderplaats (1984). *Numerical optimization techniques for engineering design*. McGraw-Hill Book Co.

[4] Su-Young Chang, Sung-Kie Youn (2006). "Material cloud method - its mathematical investigation and numerical application for 3D engineering design". International Journal of Solids and Structures, 43, 5337–5354.

[5] Dulyachot Cholaseuk (2006). "A stress-based material distribution method for optimum shape design of mechanical parts". Thammasat Int. J. Sc.Tech., 11, No. 3.

[6] A. Mahdavi, R. Balaji, M. Frecker, E. M. Mockensturm (2005). "Parallel optimality criteria-based topology optimization design". 6th World Congresses of Structural and Multidisciplinary Optimization Rio de Janeiro.

[7] Mei Yulin, Wang Xiaoming (2004). "A level set method for structural topology optimization and its applications". Advances in Engineering Software, 35, 415–441.

[8] Jian Hua Rong, Qing Quan Liang (2008). "A level set method for topology optimization of continuum structures with bounded design domains". Comput. Methods Appl. Mech. Engrg., 197, 1447–1465.

[9] Chungang Zhuang, et al. (2007). "Von Mises stress and level set method based structural topology optimization with multi-phase materials". IEEE Conference on Automation Science and Engineering.

[10] Chungang Zhuang, ZhenHua Xiong, Han Ding (2007). "A level set method for topology optimization of heat conduction problem under multiple load cases". Comput. Methods Appl. Mech. Engrg., 196, 1074–1084.

[11] Gil Ho Yoon, Young Soo Joung, and Yoon Young Kim (2007). "Optimal layout design of three-dimensional geometrically non-linear structures using the element connectivity parameterization method". International Journal for Numerical Methods in Engineering, 69, 1278–1304.

[12] Gil Ho Yoon and Yoon Young Kim (2005). "The element connectivity parameterization formulation for the topology design optimization of multiphysics systems". International Journal for Numerical Methods in Engineering, 64, 1649–1677.

[13] Gil Ho Yoon and Yoon Young Kim (2007). "Topology optimization of material-nonlinear continuum structures by the element connectivity parameterization", International Journal for Numerical Methods in Engineering, 69, 2196–2218.

[14] In Gwun Jang and Byung Man Kwak (2006). "Evolutionary topology optimization using design space adjustment based on fixed grid". International Journal for Numerical Methods in Engineering, 66, 1817–1840.

[15] Grant Steven, Osvaldo Querin, Mike Xie (2000). “Evolutionary structural optimisation (ESO) for combined topology and size optimisation of discrete structures”. Comput. Methods Appl. Mech. Engrg., 188, 743-754.

[16] Pasi Tanskanen (2006). “A multiobjective and fixed elements based modification of the evolutionary structural optimization method”. Comput. Methods Appl. Mech. Engrg., 196, 76–90.

[17] Pasi Tanskanen (2002). “The evolutionary structural optimization method: theoretical aspects”. Comput. Methods Appl. Mech. Engrg., 191, 5485–5498.

[18] S.Y. Wang, K. Tai (2005). “Structural topology design optimization using Genetic Algorithms with a bit-array representation”. Comput. Methods Appl. Mech. Engrg., 194, 3749–3770.

[19] James K. Guest, Jean H. Prevost (2007). “Design of maximum permeability material structures”. Comput. Methods Appl. Mech. Engrg., 196, 1006–1017.

[20] James K. Guest, Jean H. Prevost (2006). “Optimizing multifunctional materials: Design of microstructures for maximized stiffness and fluid permeability”. International Journal of Solids and Structures, 43, 7028-7047.

[21] Tyler E. Bruns and Daniel A. Tortorelli (2003). “An element removal and reintroduction strategy for the topology optimization of structures and compliant mechanisms”. Int. J. Numer. Meth. Engng, 57, 1413–1430.

[22] Martin P. Bendsoe, et al. (2005). “Topology optimization - broadening the areas of application”. Control and Cybernetics, 34, 7–35.

[23] Prof. Olivier de Weck (2005). Structural design optimization lecture notes

[24] Chyi-Yeu Lin, Shin-Hong Lin (2005). “Artificial neural network based hole image interpretation techniques for integrated topology and shape optimization”. Comput. Methods Appl. Mech. Engrg., 194, 3817–3837.

[25] Martin P. Bendsoe and O. Sigmund (2002). *Topology optimization theory, methods, and applications*. Springer press.

[26] O. Sigmund (2001). “A 99 line topology optimization code written in Matlab”. Struct Multidisc Optim. 21, 120–127.

[27] Samuel Amstutz, Heiko Andra (2006). “A new algorithm for topology optimization using a level-set method”. Journal of Computational Physics, 216, 573–588.

[28] Zhen Luo et. al. (2006). “Fuzzy tolerance multilevel approach for structural topology optimization”. Computers and Structures, 84, 127–140.

[29] S.Y. Wang et. Al. (2006). “An extended level set method for shape and topology optimization”. Journal of Computational Physics. In press.

[30] İbrahim Göv, M. Akif Kütük (2007). “Topoloji optimizasyonunda eleman silme metodunun uygulanması”. 15. Ulusal Mekanik Kongresi. 437-446.

[31] M. Akif Kütük, İbrahim Göv (2008). “Topoloji optimizasyonunun tekerli yükleyici koluna uygulanması”. 4. Otomotiv Teknolojileri Kongresi.

[32] M. Akif Kütük, İbrahim Göv. “Application of topology optimization to fixation plates”. Journal of Applied Biomechanics. Submitted.

Gaziantep Üniversitesi Makine Mühendisliği Bölümü 27310 Şehitkamil/GAZİANTEP

Telefon 03423172524
E-posta igov@gantep.edu.tr

İbrahim GÖV

Kişisel Bilgi	▪ Medeni durum: Evli ▪ Milliyet: T.C. ▪ Yaş: 27 ▪ Doğum Yeri: Nizip/GAZİANTEP
Eğitimi	1999 Gaziantep Üniversitesi Şehitkamil/GAZİANTEP **Lisans**
Bildiği diller	İngilizce
Bilgi alınabilecek kişiler	Prof. Dr. İbrahim Halil GÜZELBEY Yrd. Doç. Dr. M. Akif KÜTÜK
Sahip olduğu patent ve yayınlar	İbrahim Göv, M. Akif Kütük (2007). "Topoloji optimizasyonunda eleman Silme metodunun uygulanması". 15. Ulusal Mekanik Kongresi. 437-446 M. Akif Kütük, İbrahim Göv (2008). "Topoloji optimizasyonunun tekerli yükleyici koluna uygulanması". 4. Otomotiv Teknolojileri Kongresi. M. Akif Kütük, İbrahim Göv. "Application of topology optimization to fixation plates". Journal of Applied Biomechanics.(submitted)