Topology optimization of 2D structures

Bachelor's thesis

Degree Programme in Mechanical Engineering and Production Technology

Riihimäki, BEMPNU14A6

Valeriy Yakovlev

HÄMEEN AMMATTIKORKEAKOULU
HÄME UNIVERSITY OF APPLIED SCIENCES

ABSTRACT

Degree Programme in Mechanical Engineering and Production Technology
Riihimäki

Author	Valeriy Yakovlev	**Year** 2017
Subject	Topology optimization of 2D structures	
Supervisor(s)	Esa Murtola	

ABSTRACT

The objective of this thesis was to implement an algorithm for topology optimization of 2-dimensional elastic structures subjected to multiple cases of loading. For this purpose, a computer program in Python was written.

First, a brief description of structural optimization methods is given, which is then followed by more detailed description of topology optimization. After that, common problems that occur during the optimization process and their solutions are described. Finally, the validity of the program is checked using results available in literature and a comparison of structures obtained using single loading case and multiple loading case methods is presented.

After a comparison of structures obtained using different methods it was concluded in what circumstances each method should be used.

Keywords Topology optimization, compliance minimization, Python.

Pages 30 pages

LIST OF SYMBOLS

F	Global force vector
U	Global displacement vector
ρ_i	Density of element i
ρ_{min}	Minimum allowable density
v_i	Volume of element i
V	Volume of the structure
p	Penalization parameter
$\boldsymbol{t}$	Boundary tractions
Ω	Design domain
u	Displacement field at equilibrium position
v	Virtual displacement field
E_{ijkl}	Stiffness tensor
λ_1	Lagrange multiplier for volumetric constraint
λ_i^+	Vector of Lagrange multipliers for maximum density constraints
λ_i^-	Vector of Lagrange multipliers for minimum density constraints
r_{min}	Radius of filter of sensitivities

LIST OF FIGURES

CONTENTS

1 INTRODUCTION

Nowadays optimization of structures is particularly important. Not only does it reduce the material costs, but it also helps to improve mass, stiffness and other important structural characteristics. The main challenge in obtaining efficient structures is that their design is usually highly non-intuitive, especially if they are subjected to multiple loads or combinations of loads.

Topology optimization is a tool which can be used at the conceptual phase of a design process to obtain an optimal structure which can serve as a starting point for further development. The essence of the method is for a given design space to provide such a material distribution that optimizes some parameter, called an objective function, or parameters of a system subjected to a set of constraints. In this thesis, the objective function was compliance. Compliance defines how much a structure deforms under loading, whereas the constraint function is the final volume of the structure.

Many real-world structures might be subjected to multiple loads and all possible combinations of these loads. Single loading case method, if chosen for optimization of such structures, can be inadequate. This method assumes that all loads are acting simultaneously. To start with, it might be simply impossible to describe multiple loads assuming that all of them act at once. Some of these loads, for example, might act in different time and in different directions. If loads are equal and act on the same part of a body but are opposite to each other, applying these loads simultaneously will result in zero net force. But if different loads act on the different parts of a body and do not interfere with each other, such loading situation can be approximated by a single loading case.

The problem is that if we try to optimize such a structure assuming that all loads are applied simultaneously, the result will be optimal only for this particular combination of loads. The main disadvantage of this kind of structures is that they might have high sensitivity to the loading pattern. It means that removing one or several loads might result in a structure with inacceptable performance for this new combination of loads.

This is clear that if a structure is to be subjected to multiple loads which may act independently i.e. a multiple loading case, all combinations of loads must be considered.

A computer program in Python was written to implement the optimization procedure. The program consists of two modules: first module is used to find displacements within a structure by means of finite element analysis, the second module is responsible for the optimization procedure and changes parameters of the structure based on the information provided by the first module.

1.1 Structural optimization

Structural optimization is a set of methods and techniques which are used to find optimal layout of mechanical structures subjected to external and/or body forces. In this thesis, only cases with external loads are considered.

There are three main types of structural optimization: size, shape and topology optimization. Different methods of optimization have difference optimization variables. If the structure in Figure 1 is to be optimized using size optimization, the heights of each block would be used as optimization variables. In shape optimization, the boundary of the structure is the optimization variable. In topology optimization the structure is divided into elements and the so called artificial densities of each element are used as design variables.

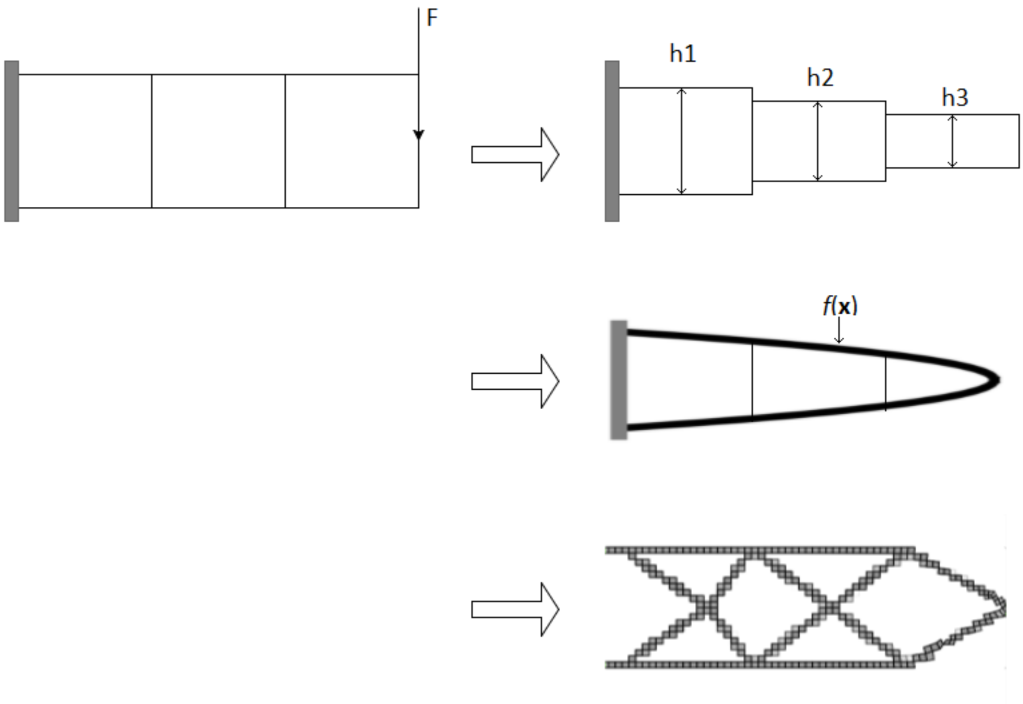

Figure 1 From top to bottom: size, shape and topology optimization

During size optimization, such properties of a system as: thickness, cross-sectional dimensions, spring stiffness, hole diameter etc., are allowed to change in order to solve the optimization problem. It is commonly used, for example, in truss design to find optimal distribution of member areas and in finding optimal stacking sequence for composites. The main disadvantage of this method is that it does not change the boundaries of the structure. An illustrative example here would be optimization of the cross-section of a beam, where starting with a hollow round cross-section you will still have the same kind of cross-section in the end of the optimization process but with, most likely, different dimensions. This leads to a result which is optimal only for the given initial layout of the structure.

Shape optimization on the other hand allows the change of boundary shapes, which gives more design freedom and allows to obtain more complex and optimized designs. Continuing the previous example, here the boundaries of the circular cross-section would be allowed to change, nonetheless this method does not permit merging and removal of boundaries. This means that the resulting structure will also depend on the initial layout.

To overcome the restrictions of size and shape optimization, topology optimization is used. Topology optimization is performed over a design domain subjected to a set of boundary conditions. During the optimization process, new boundaries can be created and old ones can be removed. Resulting structures usually have peculiar shape, which does not allow to use them directly. This method is primarily used in the beginning of a design process to obtain optimal layout of a structure, which then can be fine-tuned in order to comply with other possible constraints, such as manufacturing requirements etc.

1.2 Brief history of topology optimization

Pioneering work in topology optimization was made by Bendsøe and Kikuchi (1988). They used artificial composite material with microscopic voids. In order to obtain macroscopic properties of such material they used homogenization theory. The homogenization theory allows to obtain macroscopic properties of a composite material with microstructure consisting of periodic unit cells, based on the properties of these unit cell. Properties of one cell were calculated numerically for various sizes of voids and cell orientations and then interpolated to obtain continuous relation between density, orientation and mechanical properties of the cell. After that they formulated compliance minimization problem where design domain was discretized into finite elements. Material throughout each element had the same hole size and cell orientation, these properties were used as design variables in optimization process.

Shortly after that what today is called the SIMP (Solid Isotropic Material with Penalization) method was presented by Bendsøe (1989), where artificial density function which relates element's density and stiffness was introduced. The artificial density function makes contribution to stiffness from elements with intermediate values of density too costly in terms of their volume which promotes 0/1 distribution of densities among the elements. Artificial density was used as a design variable. The authors of Bendsøe (1989) noted that the method works very efficiently and results in 0/1 distribution of densities almost for all element if so called penalization parameter is chosen to be high enough (much greater than one), but the problem was that there was no physical meaning for the intermediate values of density.

Later Bendsøe and Sigmund (1999) showed that physical properties given by SIMP method can correspond to properties of a real composite material consisting of voids and some amount of material if penalization parameter is greater than some specified value which depends on the material's Poisson's ratio.

Diaz and Bendsøe (1992) extended the homogenization method to handle multiple loading cases. They showed that only minor changes need to be made to the original method in order to find optimal solution. The authors of that paper also noted that structures obtained by single loading case method are usually not stable if the material distribution or loading pattern within them are perturbed even slightly and that it is possible to avoid this problem using multiple loading case method.

A number of new optimization methods such as evolutionary structural optimization, soft kill option and level-set method were developed, but these methods are not considered here. A review of topology optimization methods can be found in Cazacu and Grama (2014).

1.3 Applications of topology optimization

Topology optimization was first presented as a tool for optimizing material layout for one specific purpose - make elastic structures subjected to static loadings as stiff as possible with some constraint on final volume. But in fact, the problem of distribution of a material in a design domain in order to obtain optimum solution which complies with some constraints can be extended far beyond this initial definition.

For example, in Bendsøe et al. (2005) the authors found optimal material distribution for an acoustics problem, where the objective was to maximize wave energy passing through a specified part of the design domain and Bendsøe and Sigmund (2003) showed numerous other applications including the design of material microstructure and optimizing fluid flow.

These are only a few examples of using topology optimization in different areas of physics and engineering. In this thesis, we were primarily interested in the optimization of elastic structures subjected to static loadings and volumetric constraints. To give some idea of what kind of optimization problems and constraints can be formulated for elastic structures, two different design problems with various objectives and various types of constraints are briefly presented.

1.3.1 Compliance minimization with volumetric constraints

This is the first and most common type of topology optimization problem. It was first presented in Bendsøe and Kikuchi (1988) and still remains very popular. The goal of this optimization problem can be stated as follows: having a design domain subjected to external static loadings (tractions) on some part of its boundaries Γ_t and subjected to some constraints on displacements on the other part of its boundaries Γ_u such that $\Gamma_t \cap \Gamma_u = \emptyset$ (or simply saying Γ_t and Γ_u do not intersect), find such a material distribution within the design domain so that the objective function (compliance) is minimized and the final volume of the structure equals to some specified fraction of the initial volume of the design domain.

An example of a design domain subjected to this type of constraints and optimized material distribution within this domain can be seen in Figure 2, where (a) is the initial design domain with tractions and displacement constraints, and (b) is the optimized material distribution within this initial domain. The volume of the optimized structure is 40% of the design domain volume.

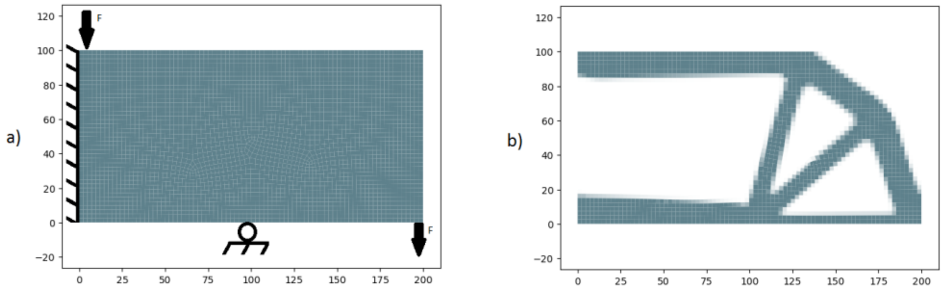

Figure 2 a) Design domain and boundary conditions b) Optimization result

For this case the optimization problem in a discrete form is given as

$$\min_{\rho,\mathbf{U}} \mathbf{F}^{\mathbf{T}}\mathbf{U}$$

$$\text{s.t} \quad \sum_{i=1}^{N} \rho_i \, v_i \leq V$$

$$\left(\sum_{i=1}^{N} \rho_i^p K_i\right) \mathbf{U} = \mathbf{F}$$

$$0 < \rho_{min} \leq \rho_i \leq 1 \quad i = 1, \dots, N$$

Since only this type of optimization problem and its slight variations are considered in this thesis, a more detailed description of them will be given later.

1.3.2 Mass minimization with stress constraints

This type of optimization problem can be described as finding the lightest structure such that some stress measure (typically von Mises stress) within this structure does not exceed a predefined maximum value (typically yield strength of the material).

This optimization problem turns out to be more complicated than the previous one. First, it is susceptible to the so-called singularity phenomenon, which is a presence of non-zero stresses in elements with densities approaching zero. This problem was discovered in truss optimization with stress constraints and reasons with analytical example of this problem for a three-member truss are given, for example, in Bruggi (2008). Turns out that this topology optimization problem is also susceptible to this phenomenon under certain conditions (Duysinx and Bendsøe (1998)). The problem can be solved in several ways. One way is to use ε-relaxed formulation of stress constraints where optimization process is successively carried out for smaller and smaller values of ε using the previous results as a starting point for each new iteration. Another way is to use different conditions mentioned above. Such a method was proposed in Bruggi (2008) and has the name of qr approach.

Another difficulty is the amount of constraints that have to be considered. Since stress is a local measure, stress in each element must be controlled. This requires lots of computation time and might make it difficult to find solution in adequate time limits. One approach to solving this problem is to introduce a global stress measure, namely a p-norm with p as large as possible. This method might not be particularly accurate as structures usually have elements with stresses ranging from zero to some large values which leads to over estimation of maximum stress within a structure.

To solve the problem of lack of accuracy of global stress measure, cluster technique is used. Here elements are sorted according to their stress values and divided into clusters to which slightly modified global stress measure is applied. Clusters are formed so that values of stress in a given cluster are as close to each other as possible. This approach improves accuracy of the global stress measure and allows for more precise control over stresses in a structure while providing only a small number of constraints, which is equal to the number of clusters.

2 THEORY

2.1 Formulation of optimization problem

The following notation can be used to describe a problem of finding such x that minimizes the objective function $f_0(x)$ and satisfies $f_i(x)$ and $h_i(x)$ which are inequality and equality constraints respectively. It is common to refer to x as to a design vector.

$$\begin{aligned} &\min_x f_0(x) \\ &\text{s.t.}\ \ f_i(x) \leq 0 \quad i = 1, \dots, m \\ &\qquad h_i(x) = 0 \quad i = 1, \dots, p \end{aligned}$$

Where m and p are the number of inequality and equality constraints respectively.
This notation will be used to state the compliance minimization problem in the following section.

2.2 Compliance minimization

Compliance defines how much a body deforms under applied load in the direction of the load. This can be extended to a number of forces and displacements associated with these forces.
Following the notation described above, we first need to define the objective function. The usual approach to this problem is to describe the objective function in terms of the work done by external forces on the structure we are trying to optimize. Since the forces are constant and do not depend on the design variables, the only parameter that can be changed in order to minimize the objective function is the displacement field. Thus, minimization of the objective function implies minimization of displacement of the points at which the loads are applied.

To describe the work done by body forces and boundary tractions on a body at its equilibrium position, the following equation is used

$$l(u) = \int_{\Gamma_T} \boldsymbol{t} \cdot \boldsymbol{u}\, d\Gamma_T + \int_{\Omega} \mathbf{f} \cdot \boldsymbol{u}\, d\Omega$$

Where $\boldsymbol{t}$ is the boundary traction, $\boldsymbol{u}$ is the displacement field at the equilibrium position, **f** is the body forces, Γ_T is the part of the body surface, where tractions are defined and Ω is the body volume. Only the first term of this equation will be used as we don't consider body forces in this thesis.

The next step in describing the minimization problem is to define the constraint functions. In the case of compliance minimization, there are two main constraints. First, the body should be in equilibrium. To include this constraint, the virtual work method is used. Second, the volume of the body should be less or equal to the specified value.

Internal virtual work of an elastic body at equilibrium position under virtual displacement can be expressed as

$$a(u,v) = \int_{\Omega} E_{ijkl}\, \epsilon(u)_{ij} \epsilon(v)_{kl}\, d\Omega$$

Where v is the virtual displacement, E_{ijkl} is the stiffness tensor and $\epsilon_{ij} = \frac{1}{2}\left(u_{i,j} + u_{j,i}\right)$.

According to the virtual work principle, a body is in equilibrium position if for any virtual displacement, internal virtual work equals to external virtual work. This allows to define the compliance minimization problem with volumetric constraints for an elastic body as

$$\begin{aligned} &\min_{u \in U, \rho}\ l(u) \\ &\text{s.t.}\quad a(u,v) = l(v) \\ &\qquad E_{ijkl} = \rho(x)^p E^0_{ijkl} \\ &\qquad \int_{\Omega} \rho(x)\, d\Omega \leq V \\ &\qquad 0 < \rho_{min} \leq \rho(x) \leq 1 \end{aligned}$$

Where U is the set of kinematically admissible displacement fields, which means that displacements do not violate the imposed boundary conditions, V is the final volume and ρ_{min} is the minimum value of density, typically it is 10^{-3}.

When solving the above problem by means of FEM, discrete formulation is used

$$\begin{aligned} &\min_{\rho, \mathbf{U}}\ \mathbf{F^T U} \\ &\text{s.t}\quad \sum_{i=1}^{N} \rho_i\, v_i \leq V \\ &\qquad \left(\sum_{i=1}^{N} \rho_i^p K_i \right) \mathbf{U} = \mathbf{F} \\ &\qquad 0 < \rho_{min} \leq \rho_i \leq 1 \quad i = 1, \dots, N \end{aligned}$$

Where $\mathbf{F}$ is the global loading vector, V is the final volume and $\mathbf{U}$ is the global displacement vector and also a solution of $\mathbf{KU} = \mathbf{F}$ where **K** is the global stiffness matrix.

2.3 Solution method

Solution of this problem using optimality criteria method is given in Bendsøe and Sigmund (2003) where updating scheme for elemental density is described as

$$\rho_{t+1} = \begin{cases} \max\big((1-\zeta)\rho_t, \rho_{min}\big) & \text{if } \rho_t B_t^{\eta} \le \max\big((1-\zeta)\rho_t, \rho_{min}\big) \\ \min\big((1+\zeta)\rho_t, 1\big) & \text{if } \min\big((1+\zeta)\rho_t, 1\big) \le \rho_t B_t^{\eta} \\ \rho_t B_t^{\eta} & \text{otherwise} \end{cases}$$

Where ρ_t is the element density at iteration t, ς is a move limit, η is a tuning parameter and B_t is given as

$$B_t = \frac{-\frac{\partial c}{\partial \rho_e}}{\lambda \frac{\partial V}{\partial \rho_e}}$$

Where λ is a Lagrange multiplier found by bisection method to satisfy the volumetric constraints and partial derivative of $c(\rho)$ with respect to elemental density, which is also called sensitivity, is given as

$$\frac{\partial c}{\partial \rho_e} = -p\rho^{p-1}\mathbf{u}_{\mathbf{e}}^{\mathbf{T}}\mathbf{k}_0\mathbf{u}_{\mathbf{e}}$$

Where $\boldsymbol{u_e}$ is the vector of nodal displacements of an element and $\boldsymbol{k_0}$ is the local stiffness matrix.

2.4 Optimality criteria

In order to solve the compliance minimization problem, optimality criteria method can be used. The method is based on using Lagrange multipliers technique and applying the KKT (Karush-Kuhn-Tucker) (Karush (1939)) (Kuhn & Tucker (1951)) conditions to find critical points of the problem.

The optimization problem to be solved is given as

$$\min_{\rho,\mathbf{U}} \mathbf{F}^{\mathbf{T}}\mathbf{U}$$

$$\text{s.t} \quad \sum_{i=1}^{N} \rho_i \, v_i \le V$$

$$\left(\sum_{i=1}^{N} \rho_i^p K_i\right)\mathbf{U} = \mathbf{F}$$

$$0 < \rho_{min} \le \rho_i \le 1 \quad i = 1, \dots, N$$

First, we note that the displacement filed **U** which is used as an input for the optimization problem, is the solution of **KU = F**, thus the second constraint is satisfied automatically. So, the problem is reduced to

$$\min_{\rho} c(\rho, \mathbf{U}(\rho)) = \mathbf{F^T U}$$

$$\text{s.t} \quad \sum_{i=1}^{N} \rho_i \, v_i \leq V$$

$$0 < \rho_{min} \leq \rho_i \leq 1 \quad i = 1, \dots, N$$

Now, we apply the method of Lagrange multipliers and construct the Lagrangian for this problem.

$$\mathcal{L}(\rho,\ \lambda_1,\ \lambda^-,\ \lambda^+) = c(\rho) + \lambda_1\left(\textstyle\sum_{i=1}^{N} \rho_i v_i - V\right) + \textstyle\sum_{i=1}^{N} \lambda_i^-(\rho_{min} - \rho_i) + \textstyle\sum_{i=1}^{N} \lambda_i^+(\rho_i - 1)$$

Then we apply KKT conditions which are given as

$$\nabla c(\rho_0) + \sum_{i=1}^{L} \lambda_i \, \nabla g_i(\rho_0) = 0$$

$$\lambda_i g_i(\rho_0) = 0 \quad i = 1, \dots, L$$

$$\lambda_i \geq 0 \quad i = 1, \dots, L$$

Where *N* is the number of elements, *L* is the number of constraints, g_i is the constraint function and ρ_0 is the critical point.

From the KKT conditions we get the following system of equations

$$\frac{\partial c(\rho)}{\partial \rho_i} + \lambda_1 v_i - \lambda_i^- + \lambda_i^+ = 0$$

$$\lambda_1 \left(\sum_{i=1}^{N} \rho_i v_i - V\right) = 0$$

$$\lambda_i^-(\rho_{min} - \rho_i) = 0 \quad i = 1, \dots, N$$

$$\lambda_i^+(\rho_i - 1) = 0 \qquad i = 1, \dots, N$$

For intermediate values of density i.e. $\rho_{min} < \rho_i < 1$ second and third constraints are not active, thus $\lambda^- = \lambda^+ = 0$ and we can write $\frac{\partial c(\rho)}{\partial \rho_i} + \lambda_1 v_i = 0$ from which we get the following relation

$$\frac{-\frac{\partial c(\rho)}{\partial \rho_i}}{\lambda_1 v_i} = 1$$

2.5 Material representation

During the topology optimization process, we are primarily interested in the material distribution in the design domain i.e. where we have solid material and where we have voids. This way of material representation is called discrete. The most common way to solve this kind of material distribution problem is to replace the discrete representation of material density with a continuous one and introduce some form of penalization parameter which forces the resulting solution to obtain 0/1 values of density and avoid intermediate ones.

One of the most popular methods used today to interpolate material density between 0 and 1 is the SIMP (Solid Isotropic Material with Penalization) method which was first presented in Bendsøe (1989). Using the SIMP model, stiffness tensor at a point is described as

$$E_{ijkl}(x) = \rho(x)^p E^0_{ijkl}$$
$$x \in \Omega$$
$$0 \leq \rho(x) \leq 1$$
$$p \gg 1$$

Where E^0_{ijkl} is the base material stiffness tensor, ρ is element's density, p is penalization parameter which is usually greater or equals to 3 (for material with Poison's ratio equals $1/3$) and Ω is the design domain. The reason why the penalization parameter should be greater than some value is discussed in Bendsøe (1999) where the problem of correspondence of physical properties of material with intermediate densities obtained by SIMP method to physical properties of real materials is investigated. The authors of that paper concluded that material with intermediate densities obtained by SIMP method can correspond to physical properties of some real composite materials with microscopic voids if the penalization parameter satisfies the following inequality

$$p \geq \max\left\{\frac{2}{1-\nu_0}, \frac{4}{1+\nu_0}\right\}$$

Where ν_0 is the Poisson's ratio of the base material.

It is shown in Figure 3 how this formulation penalizes intermediate values of density and forces the solution to obtain 0/1 form.

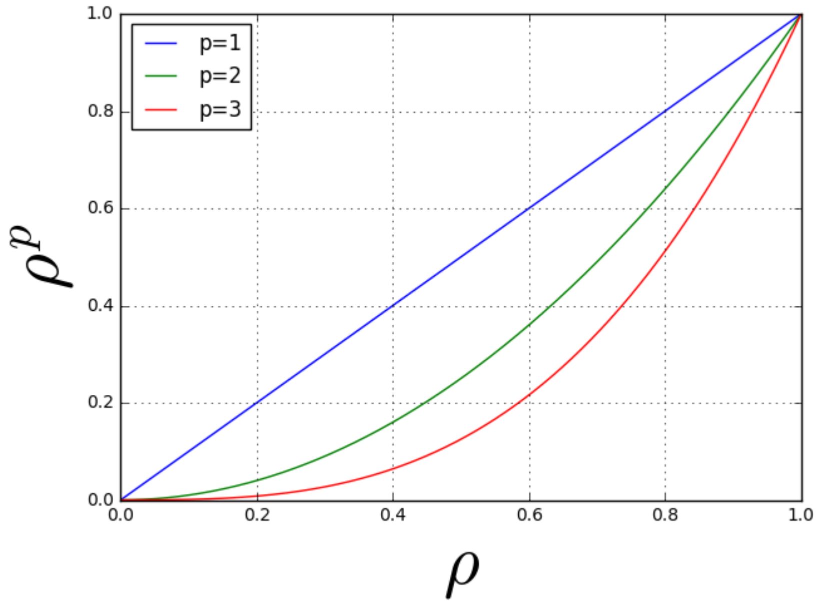

Figure 3 Effect of penalization parameter on density

2.6 Finite element method

FEM was chosen for the calculation of the displacement field. There are many different kinds of finite elements available, but among all of them the four node isoparametric quadrilateral elements were chosen as they offer a great combination of accuracy and computation speed, which compensates for the problems that arise from using this kind of elements. It is also sometimes important to depict the shape of a structure as accurately as possible. Even though the sides of the 4-node element are straight, many boundaries, including splines and circles, can be represented with sufficient accuracy if the number of the elements is large enough.

Figure 4 shows how such elements as circles and splines are approximated using finite elements and how mesh size affects approximation quality.

In order to discretize the optimization domain, Gmsh is used. Gmsh is an open-source software which allows, among other things, to generate a mesh of a geometry with high level of control over the mesh type and quality. After the mesh has been generated, a text file containing all the required information about the mesh is saved. The file is then read by our program to reconstruct the mesh and use it for finite element analysis.

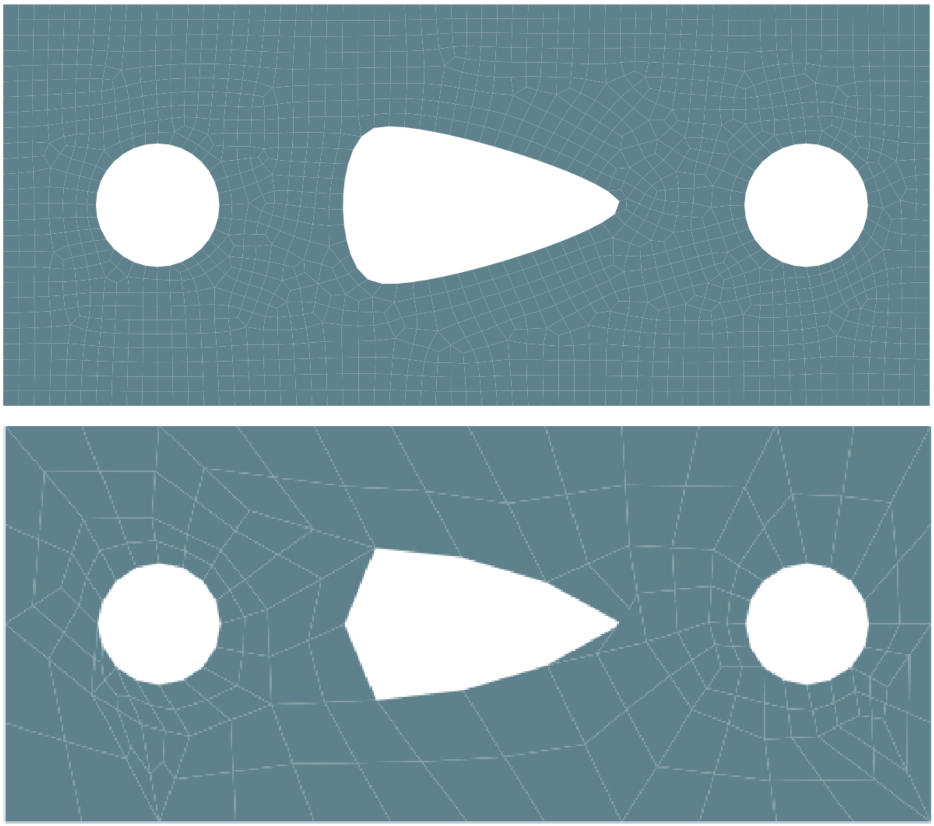

Figure 4 Different mesh quality. Top: fine mesh, bottom: coarse mesh

2.7 Mesh independency

The optimization problem as stated above in its continuous form does not have a solution. It is caused by the fact that the continuous form has infinitely many design options which leads to creation of microstructures with an optimized use of material. (Bendsøe and Sigmund (2003)) The problem is that these microstructures are not isotropic and cannot be represented by the initial problem statement for isotropic materials. In discretized space, where the number of design options is finite, this problem manifests itself as mesh dependency.

Due to mesh dependency one gets different optimal solutions to different mesh qualities with more and more fine-scale structure for increasing mesh quality instead of one unique solution but just more refined one.

One of the simplest and most efficient ways to avoid mesh dependency is filtering of sensitivities. This method was proposed by Sigmund (1994). It is based on changing the sensitivity of a given element based on the weighted average of the sensitivities of its neighbors within a specified radius and can be stated as:

$$\frac{\widehat{\partial c}}{\partial \rho_j} = \frac{1}{\rho_j \sum_{i=1}^{N} \widehat{H}_i} \sum_{i=1}^{N} \widehat{H}_i \rho_i \frac{\partial c}{\partial \rho_i}$$

Where N is the number of elements and $\widehat{H}_i$ is given as

$$\widehat{H_i} = r_{min} - dist(i,j), \{i \in N \mid dist(i,j) \ \leq \ r_{min}\}, j = 1, \dots, N$$

The $\widehat{H}_i$ operator turns to zero if $dist(i,j) > r_{min}$ and $dist(i,j)$ defines distance between the centers of elements i and j.

2.8 Checkerboard pattern

An example of the checkerboard pattern can be seen in Figure 5. This can be viewed as an alternating sequence of solid and void elements. According to Sigmund and Peterson (1998) it was believed that this kind of material distribution represents optimized microstructure, but in a paper published by Diaz and Sigmund (1995) the reasons why checkerboard pattern occurs during the optimization process were investigated. There the authors stated that this arrangement of material is undesirable as it is caused by numerical instability and does not correspond to real properties of such kind of a structure. The stiffness predicted by the finite element method is much larger than the real stiffness.

Investigation of four- and eight- node elements used with the SIMP interpolation scheme showed that for four node element values of penalization parameter larger than 1 result in a checkerboard pattern, the same kind of restriction is also valid for eight node elements, but maximum allowed penalization parameter in this case is larger and depends on Poisson’s ratio of the material.

As stated above, one of the disadvantages of using four node elements is that they are susceptible to the checkerboard pattern. There is a number of ways to fix this problem, many of which are described in Sigmund (1998).

Since we are already using the filter of sensitivities described above to ensure mesh independency, it was decided to employ this method against the checkerboard pattern too. In order to use the filter as a measure against checkerboards only and not set any mesh independency, it should be set in such a way that each element’s sensitivity depends on the sensitivities of 8 of its neighbors, but even with larger values of the filter radius it resolves the problem.

This filter has been used in numerous works and has recommended itself as a simple and reliable tool to avoid the checkerboard pattern.

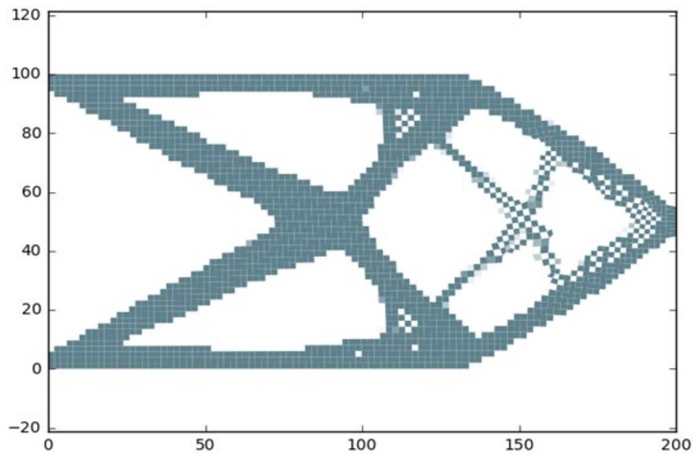

Figure 5 Checkerboard pattern

2.9 Member size control

Due to manufacturing limitations, there can be specific constraints and requirements to structures. One of these requirements is the minimum member size. Member size can be controlled using sensitivity filtering. By setting r_{min} to larger values, minimum member size correspondingly becomes larger. The results of the application of the member size control are presented in Figure 6. As can be seen, thin members in (a) are removed and replaced by thicker ones (b). All structures in Figure 6 have the same volume.

It is known that as the filter radius approaches infinity, it tends to distribute densities evenly throughout the domain. From experiments with different filter radii it was noted, that after the filter radius becomes large enough, resulting structures have many areas with intermediate densities (Figure 6 (c)).

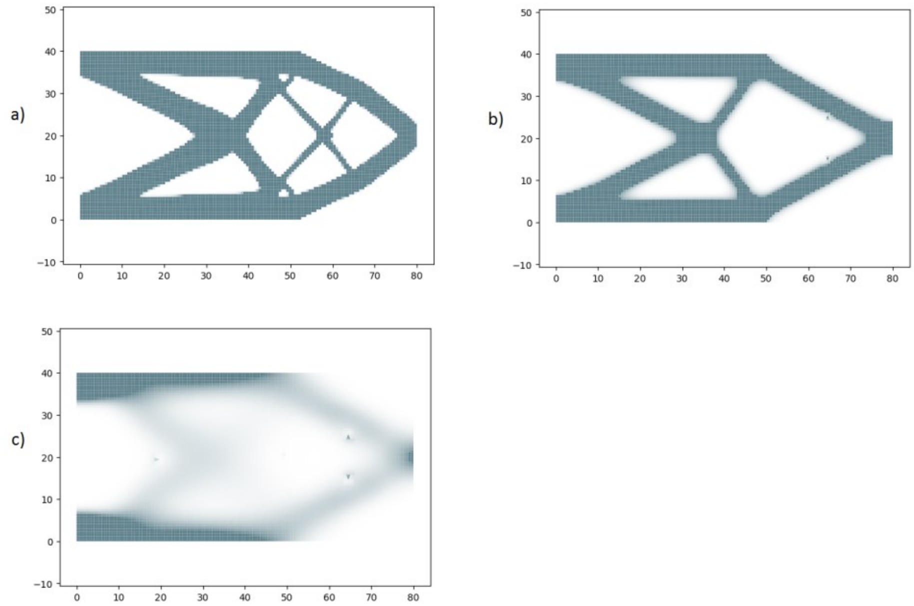

Figure 6 Effect of increasing filter radius. a) r_{min} = 0.96, b) r_{min} = 3.84, c) r_{min} = 7.68

Another example is presented in Figure 7. The structure in (a) was obtained with filter radius value such that it prevents checkerboard pattern only. This structure has many fine members which are difficult to manufacture and, since buckling of the structure is not considered, these fine members might be unstable. To fix this problem we simply increase the filter radius.

Structures (a), (b), (c) and (d) were obtained with filter radius 4, 6, 8, and 12 respectively. As can be seen, members become thicker as filter radius increases, this leads to more stable structure which is also easier to manufacture. The main drawback of having thicker members is that the structure becomes less optimal. It can be seen if compliances of these four structures are compared. Taking compliance of (a) as 1, compliances of (b), (c) and (d) are 1.056, 1.111 and 1.233 respectively.

More advanced techniques for member size control are available. These methods allow to control not only the minimum member size, but also things like minimum hole size, structural patterns, symmetry and even manufacturing process specific constraints. Usually these methods require some change of the original optimization problem statement and cannot be implemented as an add-on to the existing optimization code.

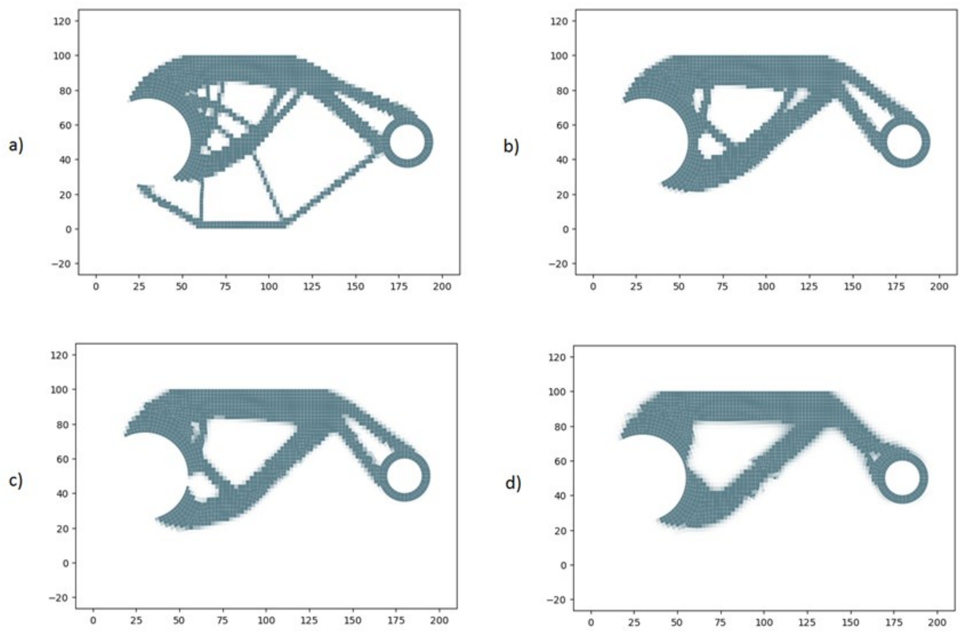

Figure 7 r_{min} increasing from (a) to (d).

2.10 **Multiple loading cases**

The optimization model for single loading case can be extended further to account for multiple loading cases. The importance of this can be seen in Figure 8, where a structure (a) is subjected to two concentrated forces which can act independently from each other. The Figure 8 (c) shows optimized material distribution for a single loading case, i.e. when all loads are applied simultaneously. As can be seen if the horizontal load is removed, the structure is subjected to high stresses at the root. This shows how ignoring different loading cases can lead to structures which are very sensitive to the loading pattern and can become highly non-optimal if the loading pattern for which optimization was performed is changed.

As proposed in Diaz and Bendsøe (1992) to avoid this problem, the previously described algorithm can be modified to account for multiple loading cases. Only a few minor changes need to be made. First, a separate FEM model is used for each loading case. The objective function becomes a weighted sum of objective functions of each of these models, as well as sensitivity. The models share the same material density field which is changed during the optimization process.

Updated objective function

$$c(\rho) = \sum_{i=1}^{N} w_i F_i^T U_i$$

Updated sensitivity

$$\left(\overline{\frac{\partial c}{\partial \rho_e}}\right)_i = \sum_{j=1}^{N} w_j \left(\frac{\partial c}{\partial \rho_e}\right)_{ij} \qquad i = 1, \dots, E$$

Where N is the number of loading cases and E is the number of elements.

The result of this improvement can be seen in Figure 8 (b). Now, comparing it with the result obtained using the single loading case, it can be seen that even if one of the loads is removed, the structure still works good and does not have points of high local stress (at the support). The drawback of the multiple loading case method is that the structure it produces is not as good as the structure produced by the single loading case method for the situation when all loads are applied at once. A more detailed comparison of the two methods is made in the next chapter.

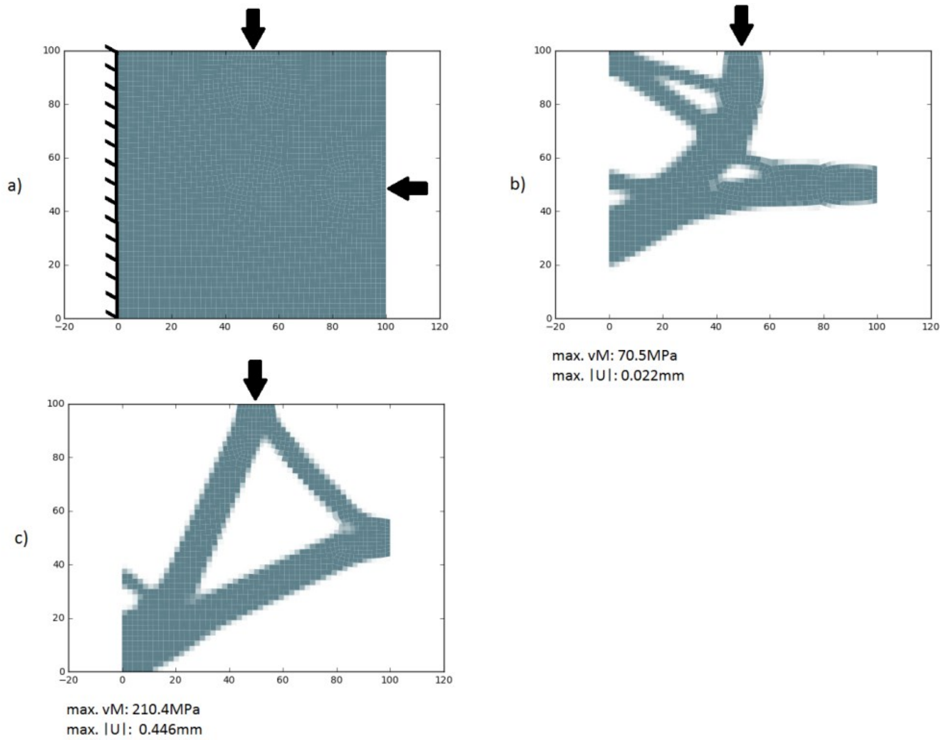

Figure 8 Comparison of single and multiple loading cases.

3 EXAMPLES

3.1 Program validation

In order to verify that the algorithm was implemented correctly, it was chosen to use standard cases which are available in literature. The results can be seen in Figure 9. Available published results, which are depicted in black, were found in Biyikli and To (2014) and were compared with the results obtained with the help of our program. Even though different optimization algorithms were used, the results are mostly similar. Based on this it can be concluded that the algorithm described in the previous paragraphs was implemented correctly.

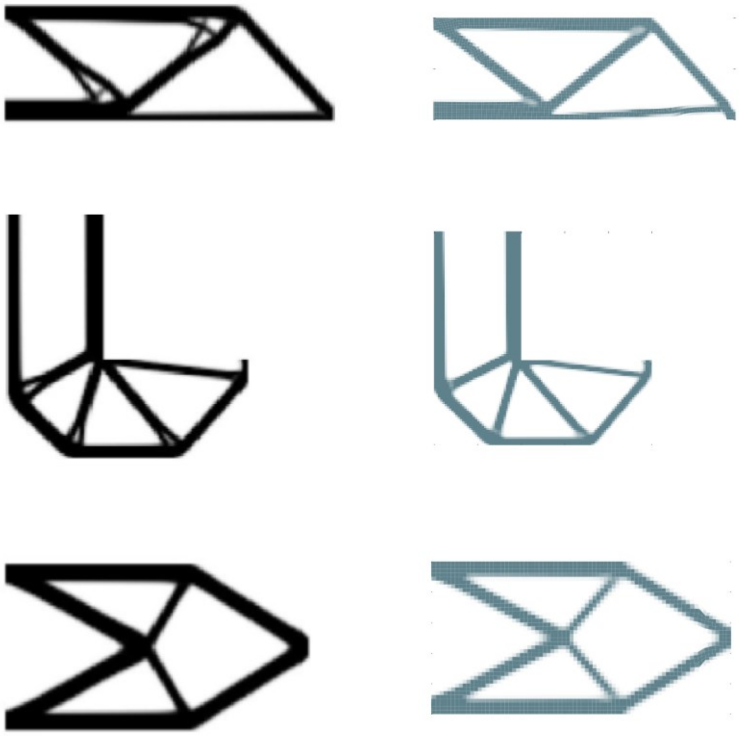

Figure 9 Comparison with results from Biyikli and To (2014)

3.2 Comparison of single and multiple loading cases

This chapter describes the difference between structures obtained using single and multiple loading case optimization method. One of the most important parameters of a structure under loading are deflections and stresses produced within the structure. Despite the fact that the structures presented here were optimized with respect to their compliance only, stresses are still considered as important comparison criterion.

3.2.1 Example 1

As the first example, we consider a simple block of solid material with height and width of 100 millimeters. The block is rigidly fixed on the left side and loaded by three concentrated forces of equal magnitude. The optimization results are shown in Figure 10.

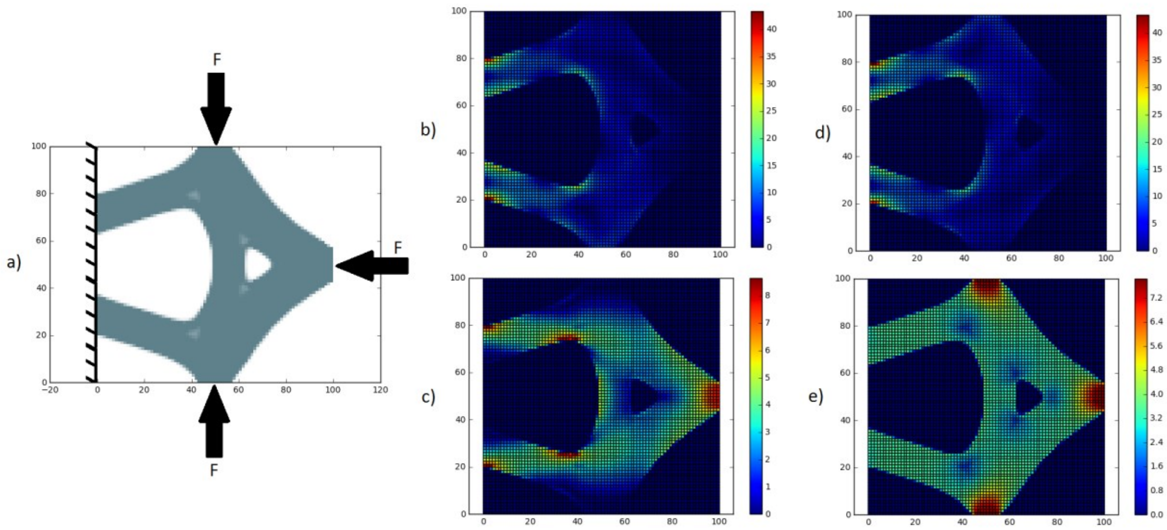

Figure 10 Optimization result for single loading case. a) optimization result and loads, b) stresses from the bottom load, c) stressed from the upper load, d) stresses from the right load, d) stresses from all loads applied simultaneously

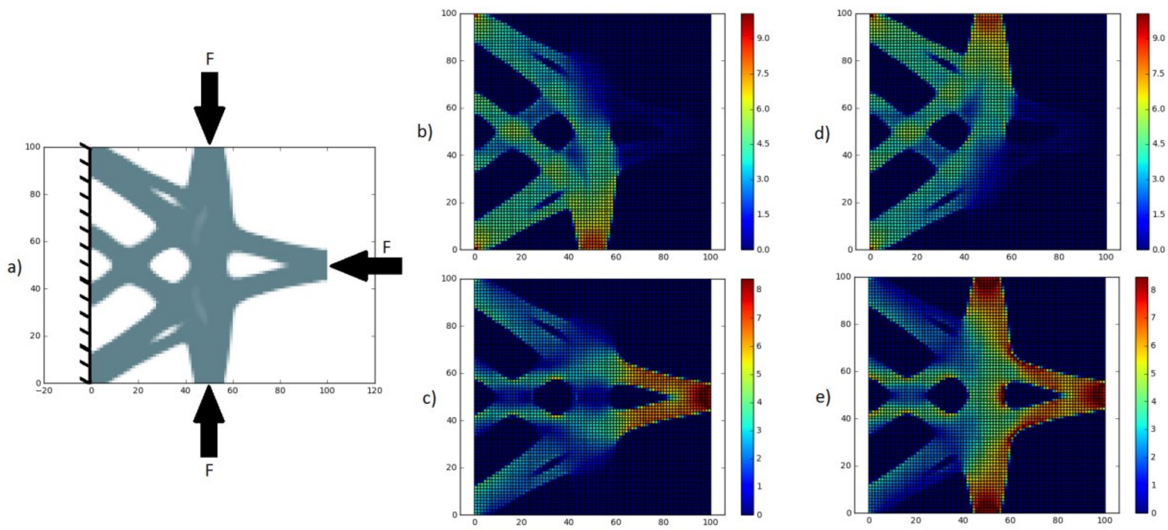

Figure 11 Optimization result of multiple loading case

The quantities used for comparison are summarized in Table 1.

Table 1.

	Single loading case		Multiple loading case	
Loading	Compliance [N-mm]	$\max(\sigma_{vM})$ [MPa]	Compliance [N-mm]	$\max(\sigma_{vM})$ [MPa]
All	0.332	7.5	0.468	9.3
Upper	1.367	45.0	0.296	9.7
Bottom	1.367	45.0	0.296	9.7
Right	0.251	8.5	0.232	8.2

Where $\boldsymbol{\sigma_{vM}}$ is the von Mises stress.

As can be seen in Table 1, in the case when all loads are applied simultaneously the first structure, which was optimized using the single loading case method, shows a better result than the second structure, which was optimized using the multiple loading case method. This is expected, since in the first case we solve one concrete problem and find the best solution for this problem, while in the second case we solve 3 different problems and find a structure which is optimal for the case when each of these 3 loads can be applied independently from each other, but not for the case when all the loads are applied at once.

From the optimization results it can be seen that the single loading case method is preferable when we can be sure that the structure which we want to optimize will be subjected only to one loading pattern for which the optimization process is performed. Otherwise, if the loading pattern is changed, high deflections and stresses can be expected. On the other hand, if it is known that the loading pattern is unsteady and can change, it is much more reasonable to use multiple loading case method. Even though this method produces structures which are not as good as those obtained using single loading case method, such structures are much less sensitive to changes in the loading pattern.

3.2.2 Example 2

The structure in Figure 12 was optimized using three different methods. The structure is subjected to two forces which can act independently from each other. The first method was the single loading case method, where all the loads are assumed to act simultaneously. The second method was the multiple loading case method, where the two loads are assumed to act separately. The third method was the multiple loading case method, but besides the two acting loads, another, third load was added. The third load was simply a combination of the two loads acting on the structure. Thus, we had three loading cases.

The design domain was a block 200 by 100 mm with two holes. The block was rigidly fixed to the left hole and loaded at the right hole. There was a ring of material around the hole on the right, elements within this ring were not changed during optimization.

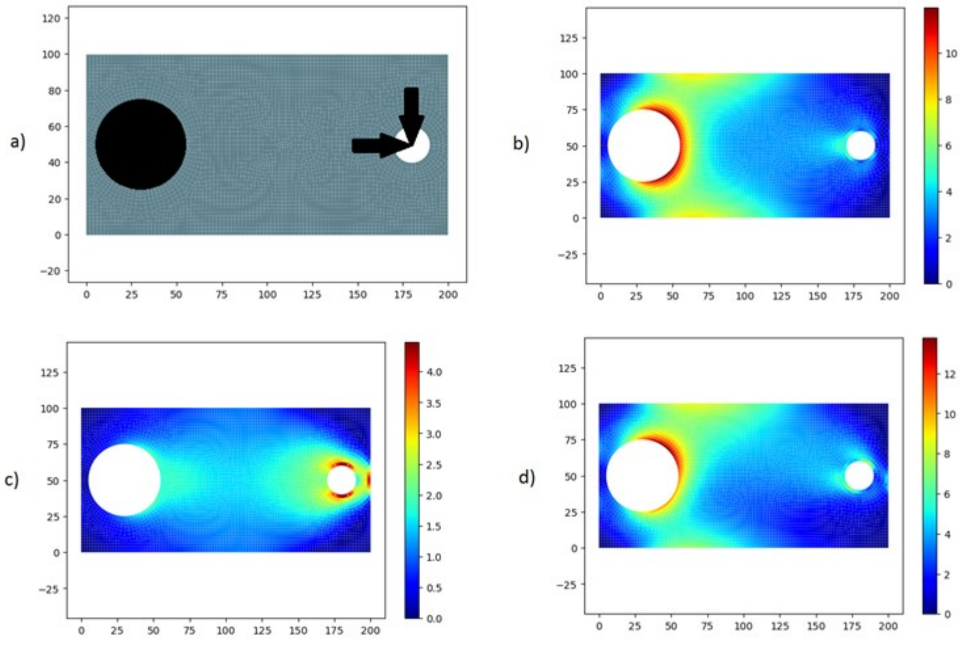

Figure 12 a) Design domain and boundary conditions b) von Mises stress from vertical load c) von Mises stress from horizontal load d) von Mises stress from both loads applied simultaneously

Now, we present results obtained by different optimization methods and discuss differences between them.

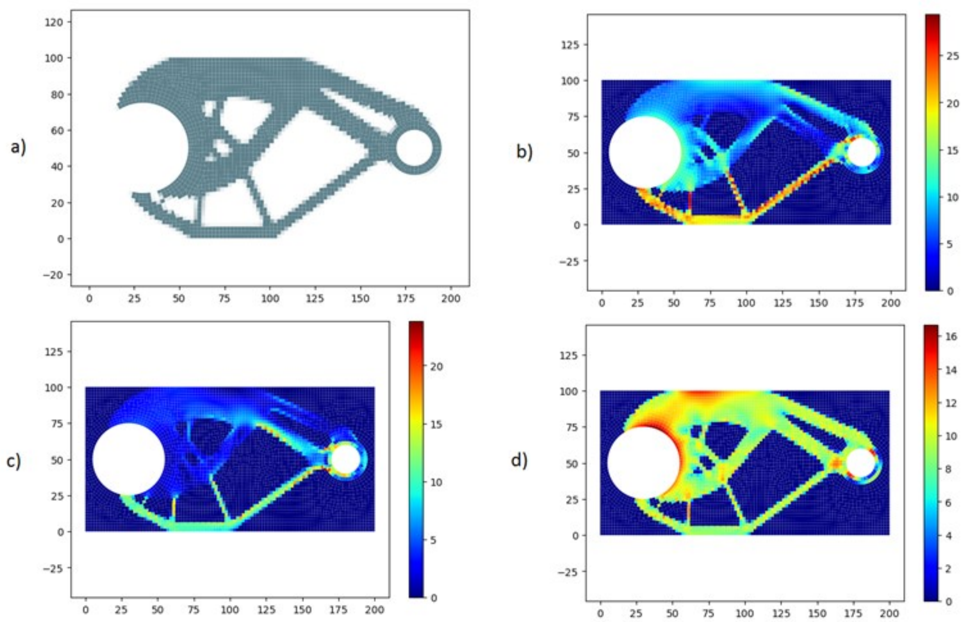

Figure 13 Single loading case optimization result

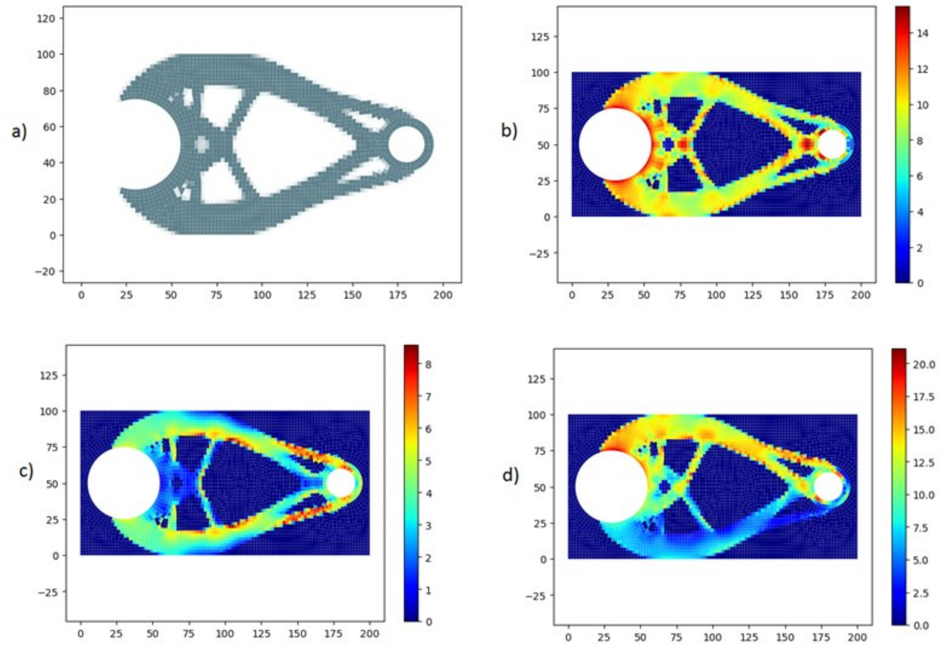

Figure 14 Multiple loading case (two loads) optimization result

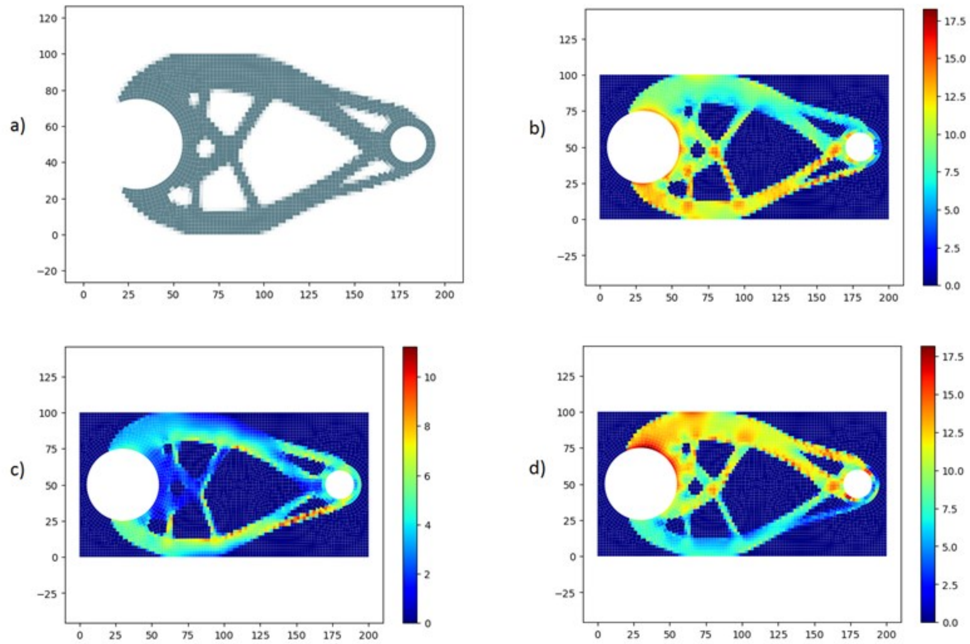

Figure 15 Multiple loading case (three loads) optimization result

The quantities used for comparison are summarized in Table 2.

Table 2.

	Single loading case		Multiple loading case (two loads)		Multiple loading case (three loads)	
Loading	Compliance [N-mm]	$\max(\sigma_{vM})$ [MPa]	Compliance [N-mm]	$\max(\sigma_{vM})$ [MPa]	Compliance [N-mm]	$\max(\sigma_{vM})$ [MPa]
Vertical	4.700	29.4	3.285	15.5	3.394	18.3
Horizontal	1.511	23.8	0.604	8.6	0.657	11.2
All	3.279	16.7	3.882	21.2	3.549	18.2

Comparison of the single loading case and the standard multiple loading case methods supports the conclusion made in the previous example. However, introduction of the third loading in the standard multiple loading case method noticeably influences the optimization result. The resulting structure, if compared against the standard multiple loading case method, has better performance in the case when all the loads act simultaneously, but worse performance when loads act separately.

From the observations made in this and the previous examples the following can be concluded

- Use single loading case method only if the loading pattern is known and is not expected to change. This method gives the best results for structures subjected to one loading pattern, but if the loading pattern is changed, performance of such structures Is usually inacceptable. Example: Figure 13.
- Use standard multiple loading case method if a structure is subjected to a number of loads and these loads are expected to act randomly and independently from each other. This method gives structures with much better performance for the cases when loads act separately from each other, but if the loads are applied simultaneously, such structures have worse performance than structures obtained using single loading case method. Example: Figure 14.
- There is a third possibility that a structure is subjected to a number of loads which are mostly expected to act simultaneously, but the loading pattern still can occasionally change. To protect the structure against such sudden changes and still not to lose too much performance for the main loading case, multiple loading case method with additional loading can be used. The additional loading is essentially a combination of all loads applied to the structure. Structures obtained using this method still have good performance when loads act separately from each other or the loading pattern is changed, and are considerably better than the structures obtained using standard multiple loading case method for the case when all loads applied simultaneously. Example: Figure 15.

3.2.3 Example 3

Sometimes structures are subjected to such a combination of loads, that it is impossible to even approximately describe this combination using the single loading case method, for this kind of situations the multiple loading case method is the only option that can be used for optimization. The single loading case method can't be used here since some forces from different loading pairs act on the same part of the body and are parallel to each other. Simultaneous application of such interfering loading pairs might lead to the loading pattern which doesn't represent the real loading situation.

In this chapter, we present a structure subjected to four different combinations of loads.

Initial geometry of the structure and all the combinations of the loads can be seen in Figure 16 and Figure 17 respectively. The structure is rigidly fixed to the two upper holes and is loaded by four pairs of loads acting on the two lower holes.

As the result of the optimization process, we get a structure which is essentially a frame which consists of triangular elements (Figure 17).

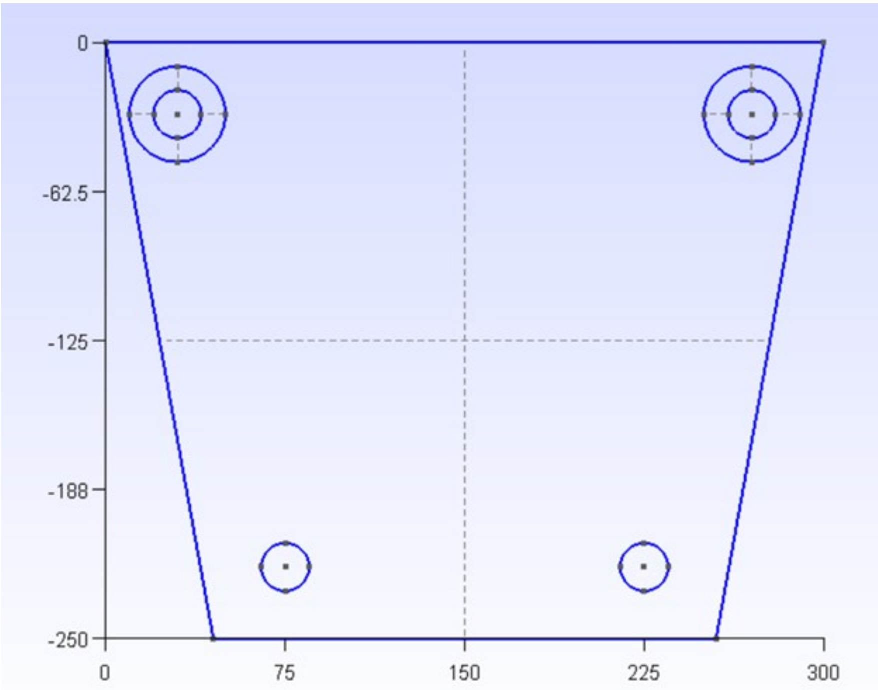

Figure 16 Dimensions of the design domain in Gmsh

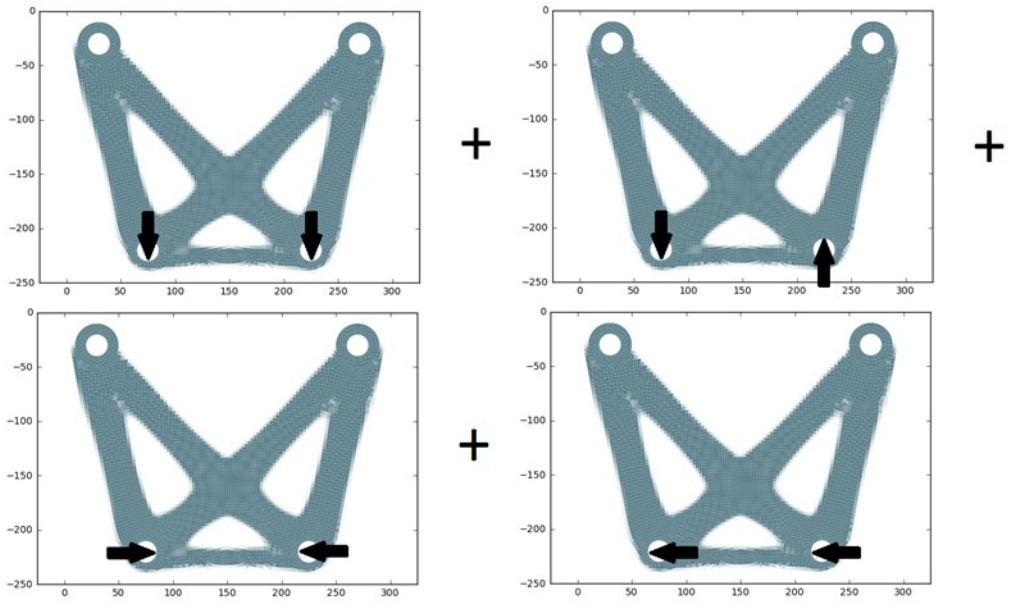

Figure 17 Optimized structure and loading cases

Using the three examples presented above the author attempted to demonstrate some of the strengths and weaknesses of different optimization methods and to make it more clear why and when one method can be more preferable than another.

4 PROGRAM DESCRIPTION

In this chapter, we discuss the structure of the program and the algorithm used. We also present an illustrative example to demonstrate the setting of an optimization case to show some capabilities of the program.

Workflow

1. Create the geometry and mesh in Gmsh
2. Save the mesh in a text file
3. Set the number of loading cases
4. Set the boundary conditions, external forces and passive regions
5. Set the filter radius
6. Set the final volume fraction
7. Select the number of iterations
8. Start the optimization
9. Save the result

The process begins in Gmsh where we create the required geometry and make its mesh which must consist of four node quadrilateral elements only. The mesh is then saved in a text file. Then we need to reconstruct the mesh and use it for FEM analysis. To do that, the saved text file is read by our program and all required information including nodal coordinates, element's nodes and boundaries is saved.

The next step is to set the number of loading cases, this defines the number of *models* used in the optimization process. If the number of loading cases is greater than one, LU decomposition of the global stiffness matrix is used. It allows to reduce the increase in calculation time due to introduction of new loading cases.

To define boundary conditions, we use entity id's which are provided by Gmsh and read from the saved text file. Each boundary has its unique entity id. Such boundary conditions as pin support, roller support and distributed loading are available.

Then, if required, we set the radius of the filter of sensitivities. Finally, volume fraction for the final structure is set and the number of iterations is defined. After that the optimization process starts. After the optimization is finished, images with the optimized structure and stresses in this structure are shown.

The sequence of actions during one iteration can be seen in the Figure 18.

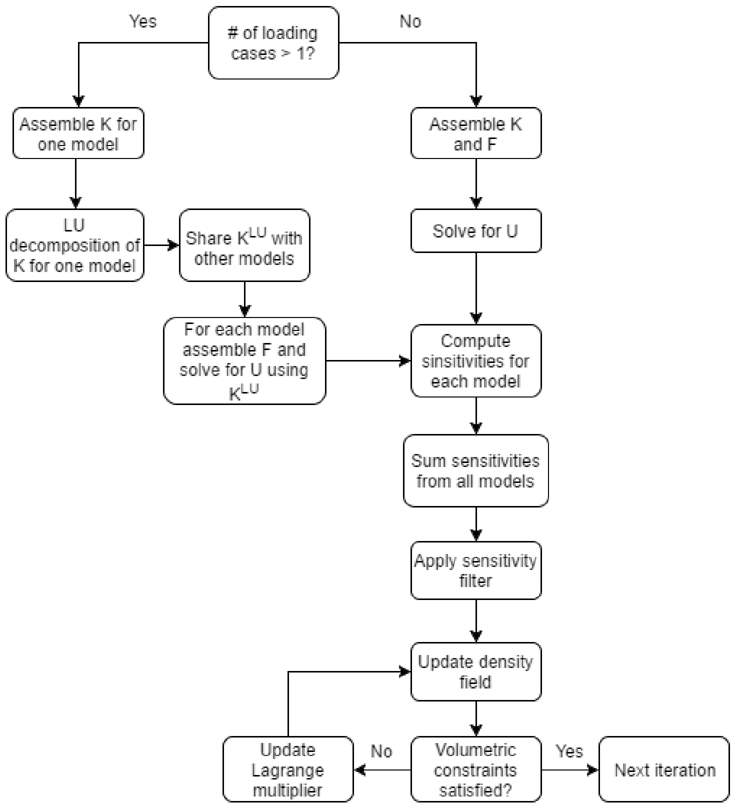

Figure 18 One iteration

4.1 Optimization example

Now we consider one optimization case and go through the whole process from meshing to the final result. The structure in Figure 19 is fixed at the top edge and has 3 pairs of loads, areas with blue and green mesh won't be changed during the optimization process, elements in these areas are called passive elements.

First, we create geometry and mesh in Gmsh Figure 19.

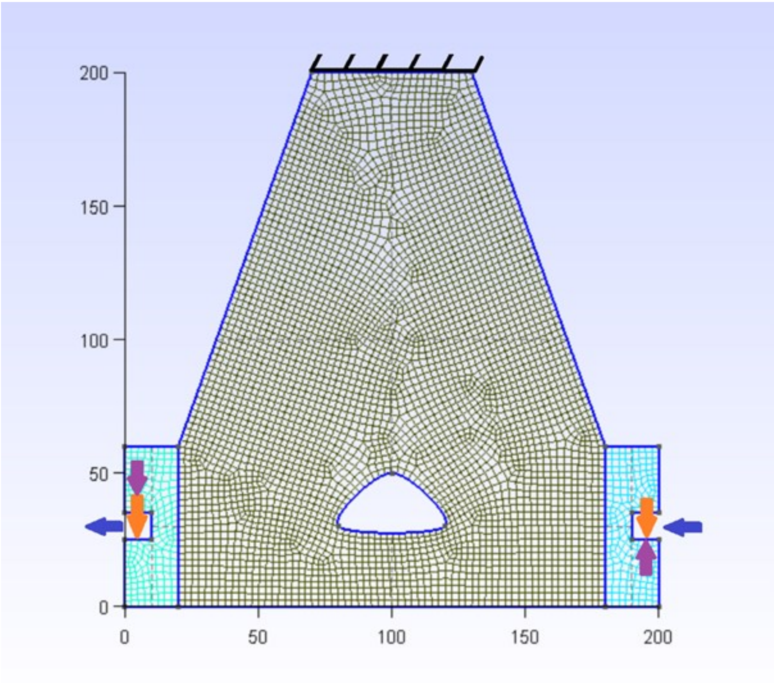

Figure 19 Mesh and loadings

After that the mesh is saved in a text file from which our program can read it using command "fm.mesh("mesh.msh")" where fm is the name of our FEM module. As can be seen in Figure 19, the structure has three different loading pairs, so this case will be treated as multiple loading case. To set the number of loading cases, we use the following command "models = [fm.FEM(m) for i in range(3)]" one model is used for each loading case. To set displacement boundary conditions, "setbc(bc_id, bc_type)" command is used, where bc_id is the entity id of the boundary and bc_type is either 1 (fixed), 2(fixed in x free in y) or 3(fixed in y free in x). The parts of the structure near the load application points are not allowed to change during the optimization process. To make these areas passive, we use the following command "setpasselems(plane_id)" where plane_id is the entity id of the plane which is will not be unaffected during optimization. To set loads "models[N].setload(bc_id, [load_x, load_y])" is used where N is the number of the model. The final steps would be to set radius of the sensitivity filter using "setfilter(r_min)" choose the final volume and number of iterations. All code required to set up this case is presented below.

```
m = fm.mesh("untitled.msh")  # read mesh
models = [fm.FEM(m) for i in range(3)]  # set number of loading cases
map(lambda x: x.setbc(12, 1), models)  # set boundary conditions to all models
models[0].setpasselems(23)  # set passive elements
models[0].setpasselems(25)

models[0].setload(17, [-10., 0])  # set loads
models[0].setload(6, [-10., 0])

models[1].setload(18, [0., 50])
models[1].setload(5, [0., 50])

models[2].setload(18, [0., -50])
models[2].setload(7, [0., 50])

models[0].setfilter(2*1.6)  # set filter radius
```

The final step of the process is to get the results. To do that, we print an image with optimized material distribution. As can be seen in Figure 20 (a), the initial selection of the filter radius results into creation of fine structural members which might be undesirable. To get rid of these members, we increase the filter radius and get slightly different solution (b). The drawback of removing fine members is decrease in optimality of the solution. In this case structure (a) is slightly stiffer than structure (b).

Due to the fact that the problem statement does not include buckling constraints, it requires some criticism in accessing the resulting structures as they might be unstable. In this case the optimized structure has a few long and slender members which might buckle during application of one or more loading pairs.

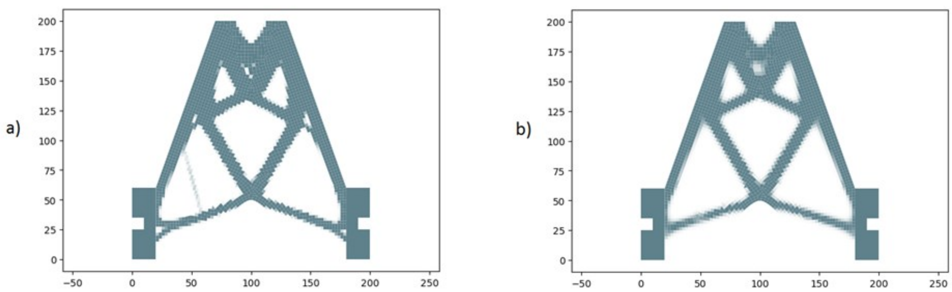

Figure 20   Optimization result

5 CONCLUSION

The objective of this thesis was to implement an algorithm for topology optimization of 2 dimensional elastic structures subjected to multiple cases of loading. The program was written in Python as this computer language offers great flexibility, simplicity and speed of development.

The program consists of two modules, the first module is responsible for the finite element analysis and plotting of the results. The module provides means for computation and visualization of displacements and stresses as well as for visualization of optimization results. The second module is responsible for the optimization procedure. There such parameters as the final volume, the filter radius, the number of iterations, convergence criteria are given. The second module also provides all required information such as the boundary conditions, the external forces and the mesh file to the first module.

Both modules use a number of external packages, numpy for array operations and array handling, scipy for operations on sparse matrices and matplotlib for plotting.

After a comparison of several optimization cases where structures were subjected to multiple loads i.e. multiple loading cases, differences between structures obtained by the single loading case, the multiple loading case and the multiple loading case with additional loading methods were discussed and it was concluded when it is advantageous to use one method over another.

The optimization algorithm implemented here is quite old and has been implemented almost in any CAD/analysis program. This leaves a lot of room for improvements of the program developed in this thesis. The optimality criteria method can be replaced by the method of moving asymptotes (MMA) (Svanberg (1987)) and other types of optimization problems can be implemented. Of particular interest is mass minimization with clustered stress constraints (Holmberg et al. (2013)) which provide better accuracy in accessing the maximum stress than the global stress measure and a lower computation time than local stress constraints. Further development of the program will be moving in the direction of implementing of these techniques.

REFERENCES

Bendsøe M. P., & Kikuchi N. (1988). Generating optimal topologies in structural design using a homogenization method. *Computer methods in applied mechanics and engineering 71*, 197-224.

Bendsøe M. P. (1989). Optimal shape design as a material distribution problem. *Structural Optimization 1*, 193-202.

Bendsøe M. P., & Sigmund O. (1999). Material interpolation schemes in topology optimization. *Archive of Applied Mechanics 69*, 635-654.

Bendsøe M. P., Lund E., Olhoff N., & Sigmund O. (2005). Topology optimization – broadening the areas of application. *Control and Cybernetics 34, No. 1.*

Bruggi M. (2008). On an alternative approach to stress constraints relaxation in topology optimization. *Struct. Multidisc. Optim., 36*. 125–141. DOI 10.1007/s00158-007-0203-6

Cazacu R., & Grama L. (2014). Overview of structural topology optimization methods for plane and solid structures. *ANNALS OF THE UNIVERSITY OF ORADEA Fascicle of Management and Technological Engineering. volume XXIII (XIII), 2014/3 issue 3*. DOI: 10.15660/AUOFMTE.2014-3.3043

Diaz A., & Bendsøe M. P. (1992). Shape optimization of structures for multiple loading conditions using a homogenization method. *Structural Optimization 4*, 17-22.

Diaz A., & Sigmund O. (1995). Checkerboard patterns in layout optimization. *Structural Optimization 10*, 40-45.

Duysinx P., & Bendsøe M. P. (1998). Topology optimization of continuum structures with local stress constraints. *Int. J. Numer. Meth. Engng. 43*, 1453–1478.

Geuzaine C. and Remacle J.-F (2009). Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities. *International Journal for Numerical Methods in Engineering 79(11)*, 1309-1331.

Holmberg E., Torstenfelt B., and Klarbring A. (2013). Stress constrained topology optimization. *Structural and Multidisciplinary Optimization*, 1–15. DOI 10.1007/s00158-012-0880-7

Karush W. (1939). "Minima of Functions of Several Variables with Inequalities as Side Constraints". M.Sc. Dissertation. Dept. of Mathematics, Univ. of Chicago, Chicago, Illinois.

Kuhn, H. W.; Tucker, A. W. (1951). "Nonlinear programming". Proceedings of 2nd Berkeley Symposium. Berkeley: University of California Press, 481–492. MR 47303.

Svanberg K. (1987). The method of moving asymptotes – a new method for structural optimization. *International Journal for Numerical Methods in Engineering, vol., 24*. 359-397.

Sigmund O. (1994). Design of material structures using topology optimization. Technical University of Denmark, Lyngby, Denmark.

Sigmund O., & Petersson J. (1998). Numerical instabilities in topology optimization: A survey on procedures dealing with checkerboards, mesh-dependencies and local minima. *Structural Optimization 16*, 68-75.

To. A. C., & Biyikli E. (2014). Proportional Topology Optimization: A new non-gradient method for solving stress constrained and minimum compliance problems and its implementation in MATLAB. *PLoS ONE 10(12)*. DOI: 10.1371/journal.pone.0145041