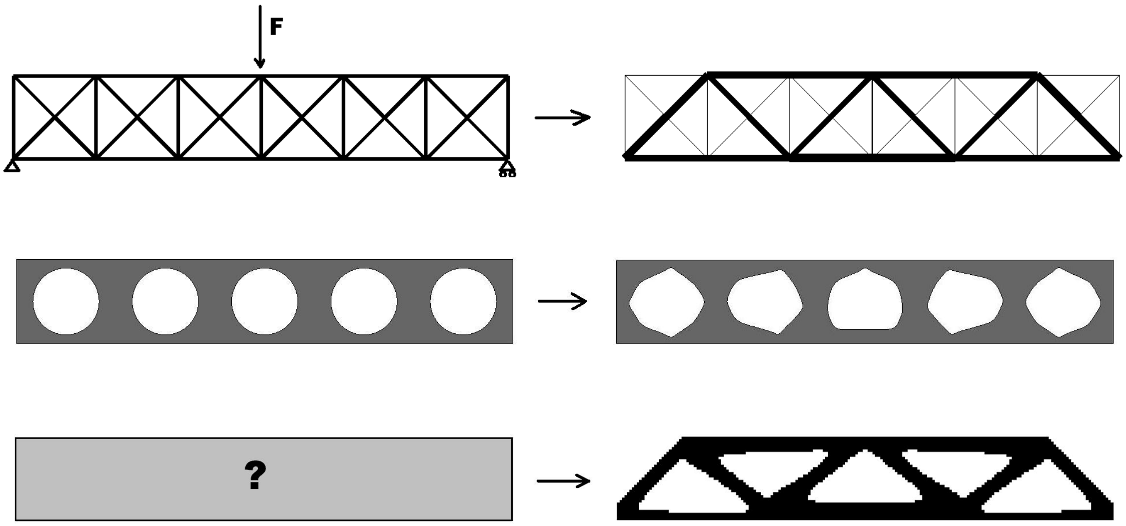
CHALMERS



Methodology for Topology and Shape Optimization in the Design Process

Master's Thesis in the Master's programme Solid and Fluid Mechanics

ANTON OLASON
DANIEL TIDMAN

Department of Applied Mechanics
Division of Dynamics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2010
Master's Thesis 2010:11

MASTER'S THESIS 2010:11

Methodology for Topology and Shape Optimization in the Design Process

Master's Thesis in Solid and Fluid Mechanics
ANTON OLASON
DANIEL TIDMAN

Department of Applied Mechanics
Division of Dynamics
CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2010

Methodology for Topology and Shape Optimization in the Design Process
ANTON OLASON
DANIEL TIDMAN

Master's Thesis 2010:11
ISSN 1652-8557
Department of Applied Mechanics
Division of Dynamics
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: + 46 (0)31-772 1000

Cover:
The cover image depicts the three different types of structural optimization; size, shape and topology optimization

Chalmers Reproservice
Göteborg, Sweden 2010

Methodology for Topology and Shape Optimization in the Design Process
Master's Thesis in Solid and Fluid Mechanics
ANTON OLASON
DANIEL TIDMAN
Department of Applied Mechanics
Division of Dynamics
Chalmers University of Technology

Abstract

In the last decade the use of commercial structural optimization software have increased rapidly. An especially interesting field is topology optimization where optimization methods are used to generate a design concept early in the design process. However, the possibilities and limitations of topology and shape optimization in the design process at Saab Microwave Systems have not been investigated previously.

The objective of this master's thesis is to investigate how and when structural optimization should be applied in the design process. The used tools are HyperMesh, Optistruct and HyperView which are parts of the software suite HyperWorks from Altair Engineering. Experience and knowledge in using structural optimization have been obtained by an initial literature study combined with evaluation of multiple trial cases of different nature. The trial cases have been performed as limited design projects where structures were improved or designed by using different types of optimization. The most common task has been to reduce mass with maintained mechanical properties as a constraint. This has been used to develop a sensible methodology together with guidelines for practical matters such as parameter values and recommended options.

It has been found that there are essentially in two stages of the design process that structural optimization can be applied. In the early stage of concept generation, topology optimization should be used to develop an efficient structure from the beginning. At this level an automatized variation of optimization parameters was proven useful to find the best feasible design possible. In the later stage, shape and size optimization should be used to fine-tune the structure realized from the topology optimization. Using optimization in this manner gives great possibilities to save time and mass as well as it may produce innovative designs no one would ever think about. Altogether, this makes structural optimization an outstanding tool in the design process.

The HyperWorks package was found to be a capable tool for performing the used types of structural optimization giving usable results. In most of the trial cases the problem could be modeled as desired in means of geometry, discretization and optimization setup.

Applying topology optimization in the concept stage requires a close cooperation between the designer and the analysis engineer to create the design domain. Further, a proper formulation of loads and boundary conditions is crucial for a usable solution. Interpreting the results from the topology optimization is a difficult task and requires experience and knowledge of other aspects such as manufacturability. Such matters have not been investigated in any greater detail.

Keywords: topology optimization, shape optimization, SIMP, Optistruct, methodology

Contents

Preface

This master's thesis is the final part of the master's programme Solid and Fluid Mechanics. It is also our last efforts as students at Chalmers University of Technology. The work was carried out from January to June 2010 at the office of Saab Microwave Systems in Lackarebäck.

We would like to thank everyone who has helped and contributed to this Master's thesis work. In particular we would like to thank our supervisor at Saab Microwave Systems, Ruoshan Luo, for invaluable help and guidance during the work. The feedback and know-how has helped us to continue our work in the right direction. Also we would like to thank our supervisor and examiner Håkan Johanssonat at Applied Mechanics, Chalmers, for guidance and expertise within the field of solid mechanics and structural optimization. His devotion to our work has inspired us to produce more interesting and usable results. We are also very thankful to Jan Lindahl, analysis engineer at Saab Microwave Systems, for reflections and tips contributing to the technical height. Further we would like to thank all the employees at Saab's department in Lackarebäck for their hospitality and cakes at the Friday coffee brakes.

In addition we want to address thanks to Altair Engineering who has generously provided software licenses and given very helpful and appreciated courses in how to use the software (HyperMesh, Optistruct and HyperView). Altair Engineering has also assisted with technical support in the best possible way.

This report is published in two slightly different versions, one public and one internal for Saab.

Göteborg June 2010
Anton Olason, Daniel Tidman

1 Introduction

This is a master's thesis project initiated by Saab Microwave System to investigate the possibility of using topology and shape optimization in the design process. In this chapter the background, purpose, objective and limitations are described together with the used method.

1.1 Background

At Saab Microwave Systems (SMW) a variety of different mechanical structures are developed. It is always desired to find the best possible design under some given circumstances. The current design method is often an iterative process where a design is proposed by the designer and then analyzed using the finite element method (FEM) by a structural engineer. The analysis will often also be complemented by mechanical tests on a prototype. Possible modifications to the design to satisfy unfulfilled requirements or to improve the design are then made by the designer. The new design is analyzed and the process is repeated. This iterative process may take a lot of time and result in a suboptimal design as multiple changes are added to each other with no overall view.

In the last decade the use of commercial optimization software for topology and shape optimization in the industry has increased rapidly and has shown to be applicable to many different types of problems. The idea is that the designer and the structural engineer both are involved in the initial stage when the first design proposal is conceived. Topology optimization is used to generate a good design concept. However, the possibilities and limitations of such a method in the design process at SMW have not yet been investigated.

A similar work has been performed by Fagerström and Jansson at Volvo Car Corporation [1]. The software that will be used are the preprocessor HyperMesh, the optimization tool Optistruct and the postprocessor HyperView, all of them products of Altair Engineering.

1.2 Purpose

The purpose of this work is to gain an understanding of what topology and shape optimization is, the theory behind and how it can be used to improve the design process at SMW.

1.3 Objective

The main task is to investigate how and when structural optimization can be used at SMW in the design process. This includes an evaluation of possibilities and limitations and to find areas where this technology is applicable. The end result is a sensible methodology of when and how to perform an optimization using a commercial software suite. Guidelines for different aspects of the process like values of various parameters and options will also be determined. All the required steps will be presented in a flowchart.

1.4 Scope and limitations

The focus of this work is to develop a practical and robust method when using topology optimization in the design process and this is where most of the effort will be put. The theory and mathematics behind the different optimization methods will not be investigated in any greater detail. Boundary conditions, loads and material parameters are considered to be known, the derivation of them are beyond the scope of this work.

1.5 Method

The work for the master thesis was carried out at SAAB Microwave Systems at the office in Lackarebäck where computers and software needed were provided. The semester started out with planning resulting in a planning report briefly describing the different steps to be performed and their respective required time.

The work was divided in three different phases: Study of literature and software, study of trial cases and development of methodology. The literature study was conducted to get a general understanding of optimization and topology optimization. This included reading a number of scientific articles as well as books on the subject. The understanding of the theory was later used to produce good results more efficiently and to analyze them. In addition, time was invested in learning about practical matters such as how the different software are used and how they work together (HyperMesh, Optistruct and HyperView). How the theory comes into practice in the software was also studied.

In the second phase the focus was shifted to get experience and knowledge of working with topology and shape optimization by performing several trial cases of different nature. The trial cases were carried out as short design projects, starting from specifications and demands of a structure resulting in a final design concept. The used approaches, results and learning were evaluated for each case. This more practical phase included study of robustness of solutions when varying loads and constraints and evaluation of different parameters and their influence on the solution. A lot of effort was spent on how to perform the different stages of the optimization to get as good results as possible.

Finally, a reasonable and effective methodology of how to effectively use optimization in the design process was produced. The methodology was based on gained understanding and experience of optimization as well as thoughts of designers and analysis engineers at SMW. The methodology was presented as a step-by-step guide book to make it as useful as possible at SMW after the finish of the thesis work.

2 Theory

Here the basics of optimization in general and topology optimization in particular will be described; for a more in-depth look on mathematical optimization please refer to Rao [2] and Ehrgott [3] and for structural optimization see Bendsøe [4] and Klarbring and Christensen [5].

2.1 Mathematical optimization

The basic principle of optimization is to find the best possible solution under given circumstances [2]. One example of optimization is finding the quickest route when using the public transportation system or, as in the case of structural optimization, finding the optimal distribution of material that satisfies some given requirements. This is most often done by decisions made by the passenger or the engineer from their own experience and knowledge about the subject.

The objective of the optimization problem is often some sort of maximization or minimization, for example minimization of required time or maximization of stiffness. To be able to find the optimum solution the 'goodness' of a solution depending on a particular set of design variables needs to be expressed with a numerical value. This is typically done with a function of the design variables known as the *cost function*.

Mathematically the general optimization problem is most often formulated as minimization of the cost function (which can easily be transformed to maximization by minimizing the negative function) subject to constraints, this can be expressed as [2]:

$$\begin{aligned} &\text{Find } \boldsymbol{x} = \begin{Bmatrix} x_1 \\ x_2 \\ \vdots \\ x_n \end{Bmatrix} \text{which minimizes } f(\boldsymbol{x}) \\ &\text{subject to } \begin{cases} g_i(\boldsymbol{x}) \leq 0, \ i = 1, 2, \dots, m \\ h_j(\boldsymbol{x}) = 0, \ j = 1, 2, \dots, n \end{cases} \end{aligned} \tag{2.1}$$

where $\boldsymbol{x}$ is the vector of design parameters and $f(\boldsymbol{x})$ is the cost function. The functions $g_i(\boldsymbol{x})$ and $h_j(\boldsymbol{x})$ are called the *inequality constraint function* and the *equality constraint function* respectively and they define the constraints of the problem. This is called a *constrained optimization problem*.

2.1.1 Multicriteria optimization

In many cases there are multiple objectives that need to be taken into account. One example used by Ehrgott [3] is when buying a car; it is for example desired to have a car that is powerful, cheap and fuel efficient. Obviously it is not possible to find a car that is the best in every aspect, a powerful car is normally neither cheap nor fuel efficient.

A concept often used in optimization with multiple objectives is Pareto optimality. A solution is said to be *Pareto optimal*[1] if there exists no other feasible solutions that would decrease any of the objective functions without causing an increase in any of the other objective functions [6]. The set of the Pareto optimal solutions is called the *Pareto front* [6], for the case of two objectives this can easily be visualized in a two-dimensional diagram. From the Pareto front interesting information about the trade-off between different objectives and how they affect each other can be obtained.

[1] Other notations such as *efficient* or *non-inferior* are also used in literature [3]

One method of solving the multicriteria optimization problem is by *scalarization*, i.e., by transforming the multiple objective functions into a scalar function of the design variables. The simplest scalarization method is the weighted sum method [3]:

$$\min_{\boldsymbol{x}} \sum_{k=1}^{p} w_k f_k(\boldsymbol{x}), \quad \text{where } f_1, \ldots, f_k \text{ are the objective functions} \tag{2.2}$$

By varying the weights w_k, different Pareto optimal solutions may be found.

Another approach is to just consider one of the objective functions and constraining the other, the ε-constraint method [3]:

$$\begin{aligned} &\min_{\boldsymbol{x}} f_j(\boldsymbol{x}) \\ \text{subject to } &f_k(\boldsymbol{x}) \leq \varepsilon_k, \ k = 1, \ldots, p, \ k \neq j \end{aligned} \tag{2.3}$$

The problem is then solved with different values on the constraints ε_k.

2.2 Structural optimization

Structural optimization is one application of optimization. Here the purpose is to find the optimal material distribution according to some given demands of a structure. Some common functions to minimize are the mass, displacement or the compliance (strain energy). This problem is most often subject to some constraints, for example constraints on the mass or on the size of the component.

This optimization is traditionally done manually using an *iterative-intuitive* process that roughly consists of the following steps [5]:

1. A design is suggested

2. The requirements of the design is evaluated, for example by a finite element analysis (FEA)

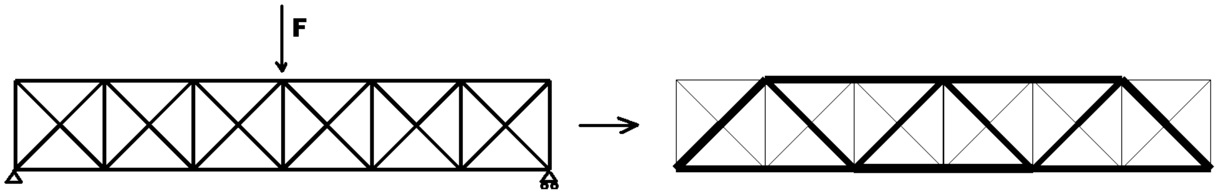
3. If the requirements are fulfilled, the optimization process is finished. Else, modifications are made, a new improved design is proposed and step 2–3 are repeated

The result depends heavily on the designer's knowledge, experience and intuitive understanding of the problem. Changes to the design are made in an intuitive way, often using trial and error. This process can be very time consuming and may result in a suboptimal design.

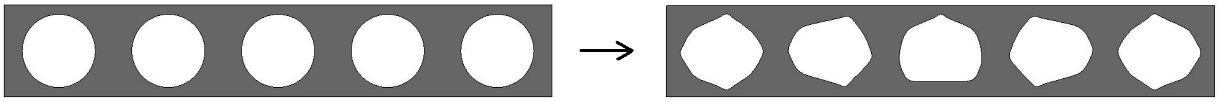
The problem of structural optimization can, according to Christensen and Klarbring [5], be separated in three different areas: *sizing optimization*, *shape optimization* and *topology optimization*, see Figure 2.1.

2.3 Sizing optimization

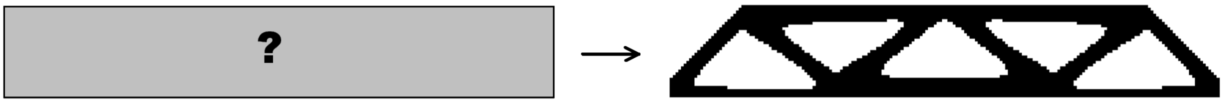
Sizing optimization is the simplest form of structural optimization. The shape of the structure is known and the objective is to optimize the structure by adjusting sizes of the components. Here the design variables are the sizes of the structural elements [5], for example the diameter of a rod or the thickness of a beam or a sheet metal. See Figure 2.1a for an example of size optimization where the diameter of the rods are the design variables.



(a) Sizing optimization



(b) Shape optimization



(c) Topology optimization

Figure 2.1: Different types of structural optimization

2.4 Shape optimization

As with sizing optimization the topology (number of holes, beams, etc.) of the structure is already known when using shape optimization, the shape optimization will not result in new holes or split bodies apart. In shape optimization the design variables can for example be thickness distribution along structural members, diameter of holes, radii of fillets or any other measure. See Figure 2.1b for an example of shape optimization. A fundamental difference between shape vs. topology and size optimization is that instead of having one or more design variable for each element the design variables in shape optimization each affect many elements.

2.4.1 Perturbation vector approach

One way of introducing shape changes to the discretized finite element model is with the perturbation vector approach [7]. First one or more shapes are defined as perturbations added to the vector of nodal coordinates ($\boldsymbol{r}_0$),

$$\boldsymbol{r} = \boldsymbol{r}_0 + \boldsymbol{p} \tag{2.4}$$

By doing a linear combination of the perturbations, the design variables for the optimization can then be defined as the weights of the perturbation vectors. One design variable per shape vector:

$$\boldsymbol{r} = \boldsymbol{r}_0 + \sum_{i=1}^{n} w_i \boldsymbol{p}_i, \quad n = \text{number of shapes/design variables}$$
$$\text{with limits on weights } w_i^{\min} \leq w_i \leq w_i^{\max} \quad i = 1, \ldots, n$$

The optimization problem is then to find the optimum set of shape weights.

2.5 Topology optimization

The most general form of structural optimization is topology[2] optimization. As with shape and size optimization the purpose is to find the optimum distribution of material. With topology optimization the resulting shape or topology is not known, the number of holes, bodies, etc., are not decided upon. See Figure 2.1c.

From a given design domain the purpose is to find the optimum distribution of material and voids. To solve this problem it is discretized by using the finite element method[3] (FEM) and dividing the design domain into discrete elements (mesh). The resulting problem is then solved using optimization methods to find which elements that are material and which are not. This result in a so called 0-1 problem, the elements either exists or not, which is an integer problem with two different states for each element, a so called ISE topology (**I**sotropic **S**olid or **E**mpty elements) [10].

The number of different combinations is 2^N, where N is the number of elements. As a normal FE-model easily results in hundreds of thousands of elements, this problem is out of reach to solve for any practical problem[4] [4].

The two main solution strategies for solving the optimization problem with an ISE topology are the density method and the homogenization method. Other methods, which will not be further studied, includes using genetic algorithms or heuristic methods such as evolutionary structural optimization[5] (ESO)[10]. Rozvany [11] points out that "ESO is presently fully heuristic, computationally rather inefficient, methodologically lacking rationality, occasionally unreliable, with highly chaotic convergence curves" and that "ESO is now therefore hardly ever used in industrial applications".

2.5.1 Density method

One way to get a problem that can be solved is to relax the problem by letting the material density take any value between zero and one, i.e., 0 % to 100 % density. By making this relaxation it is possible to use gradient based optimization methods to find a minimum of the objective function. The design variable of the optimization problem is the density which is a function varying over the design domain. In the FE discretization the density is most often approximated to be constant over each element, the resulting problem thus has one design variable, the density, per element.

In practice, this also makes it similar to sizing optimization; here the sizes are the densities of the elements. This relaxation does not have a simple physical explanation. When considering elements in 2D the density could be represented as a varying thickness of a plate. In 3D there is no similar counterpart; a solid with 50 % material is neither physically reasonable nor very intuitive.

Topology optimization using this problem formulation is called the *density method.* To get a result which is possible to manufacture, it is desired that the solution only consists of solid or empty elements, to make it behave more like an ISE topology. To approach this behavior intermediate densities are penalized, i.e., the cost of intermediate densities is higher compared to the relative stiffness. This will make intermediate densities unfavorable. Without penalty the relation stiffness–material cost is linear, $\boldsymbol{E} = \rho \boldsymbol{E}^0$ where $\boldsymbol{E}$ is the

[2] Topology is an area of mathematics that studies properties of geometric objects that depend on the shape, but not size, distances or angles; properties that are independent of any continuous deformation [8]. From a topological point of view a coffee mug and a donut can be said to be equivalent since they both have exactly one hole. A coffee mug can by smooth deformations be transformed into a donut, but not into a sphere (zero holes) or a tea pot (two holes).

[3] It is assumed that the reader is familiar with the finite element method. For an introduction to the topic please refer to Ottosen and Petersson [9]

[4] For example: a model with 10 000 elements would result in $2^{10000} \approx 10^{3010}$ number of combinations. If it was possible to evaluate one millions combinations per second, testing all of the combinations would take about 3.17×10^{2996} years.

[5] Evolutionary structural optimization use some criterion, e.g., von Mises stress, to eliminate elements in every iteration [11].

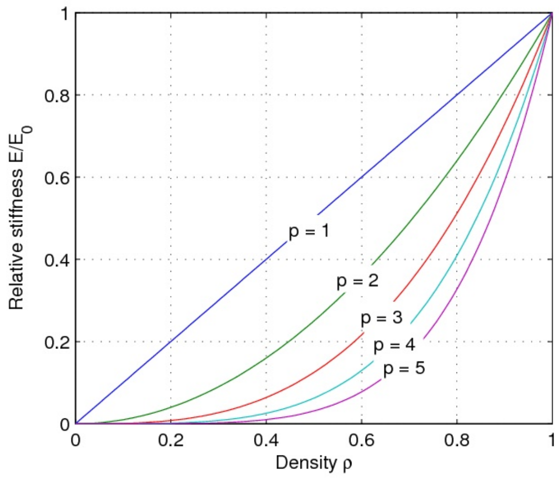


Figure 2.2: Relative stiffness as a function of density with different penalization factors [12]

elasticity tensor and ρ is the density, $0 \leq \rho \leq 1$. One popular method to achieve penalized intermediate densities is by letting the stiffness of the material be expressed as [4]:

$$\boldsymbol{E} = \rho^p \boldsymbol{E}^0, \quad \text{Mass} = \int_\Omega \rho \, \mathrm{d}\Omega, \quad p > 1 \tag{2.5}$$

When the densities are assumed constant over each element the density-stiffness relation can be implemented simply by scaling the element stiffness matrices before assembling them into the global stiffness matrix:

$$\boldsymbol{K}_e = \rho_e^p \boldsymbol{K}_e^0 \tag{2.6}$$

Where p is a penalization factor greater than zero, typically $2 - 5$. The resulting cost-stiffness relation can be seen in Figure 2.2. In literature the density method together with this penalization is often called the *SIMP method* (**S**olid **I**sotropic **M**icrostructures with **P**enalization) [10]. Unfortunately this penalization will make even the problem of minimizing compliance a nonconvex problem[6], thus finding the global optimum will be very difficult [10].

The unphysical aspect of this 'fictitious material' used in the density method led to that the adoption of this method was delayed by almost a decade [10]. This problem was later solved by Bendsøe and Sigmund [13] who managed to find a physical interpretation of intermediate densities by constructing microstructures from voids and material that realizes the material properties, with some limits on the penalization factor (for example, $p \geq 3$ when $\nu = 1/3$ due to the Hashin-Shtrikman bounds[7] [14]). See Figure 2.3.

The classical topology optimization problem of minimizing the compliance while constraining the mass can with the density method, assuming linear elasticity, can be formu-

[6] A convex problem have only one local minima, which coincides with the global minima
[7] A theoretical limit on the elasticity for composite materials

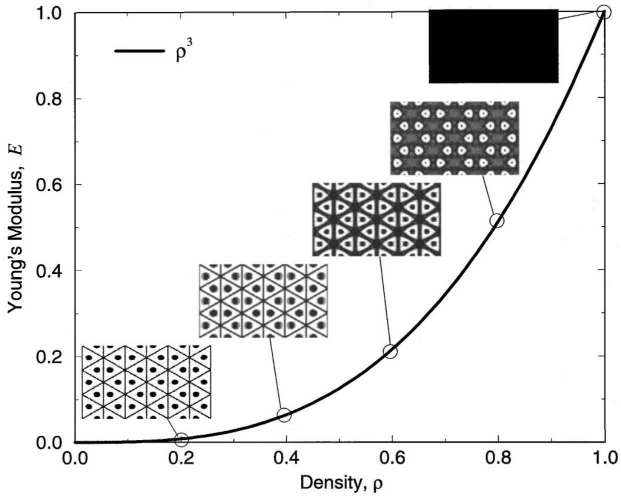


Figure 2.3: Microstructures realizing the material properties with $p = 3$ and $\nu = 1/3$ [13]

lated as[8]:

$$\begin{cases} \min_{\boldsymbol{\rho}} \; C(\boldsymbol{\rho}) = \boldsymbol{F}^T \boldsymbol{u}(\boldsymbol{\rho}) \\ \text{s.t.} \; \begin{cases} \boldsymbol{\rho}^T \boldsymbol{a} = V \\ \rho_{\min} \leq \rho_e \leq \rho_{\max}, \; e = 1, \ldots, n \end{cases} \end{cases} \tag{2.7}$$

Where $\boldsymbol{\rho} = [\rho_1, \ldots, \rho_n]^T$ is the design vector consisting of the element densities and $\boldsymbol{a} = [a_1, \ldots, a_n]^T$ is the vector of element areas. The displacements are easily found as $\boldsymbol{u}(\boldsymbol{\rho}) = \boldsymbol{K}^{-1}(\boldsymbol{\rho})\boldsymbol{F}$

Two of the main advantages of the density method are that it does not require much extra memory, only one free variable is needed per element (the density) and that any combination of design constraints can be used [10].

2.5.2 Homogenization method

The main idea of the homogenization method is that a material density is introduced by representing the material as a microstructure. The microstructure is a composite material with an infinite number of infinitely small voids [13]. This leads to a porous composite that can have a density varying between 0 % and 100 %. Some common types of microstructure are solids with square or rectangular holes or some sort of layered microstructure, see Figure 2.4. Since the macroscopic properties of the microstructure are not isotropic an orientation angle is also needed [15].

For a layered microstructure the elasticity can be found analytically, but for most other types of microstructures the elasticity needs to be calculated numerically by using the finite element method for different sizes and then interpolating between these values [15, 4]. The microstructures do by themselves provide some penalization on intermediate densities but this is most often not enough and some additional penalization needs to be introduced [10].

The optimization is then carried out similarly to the density method. The problem is discretized into finite elements with the design variables (hole sizes and rotation) assumed to be constant over each element.

[8]This is called a *nested formulation* [5] since the equilibrium constraint $\boldsymbol{K}(\boldsymbol{\rho})\boldsymbol{u} = \boldsymbol{F}$ is taken into account by letting $\boldsymbol{u}(\boldsymbol{\rho}) = \boldsymbol{K}^{-1}\boldsymbol{F}$. In a *simultaneous formulation* the equilibrium equation would be included as an equality constraint function

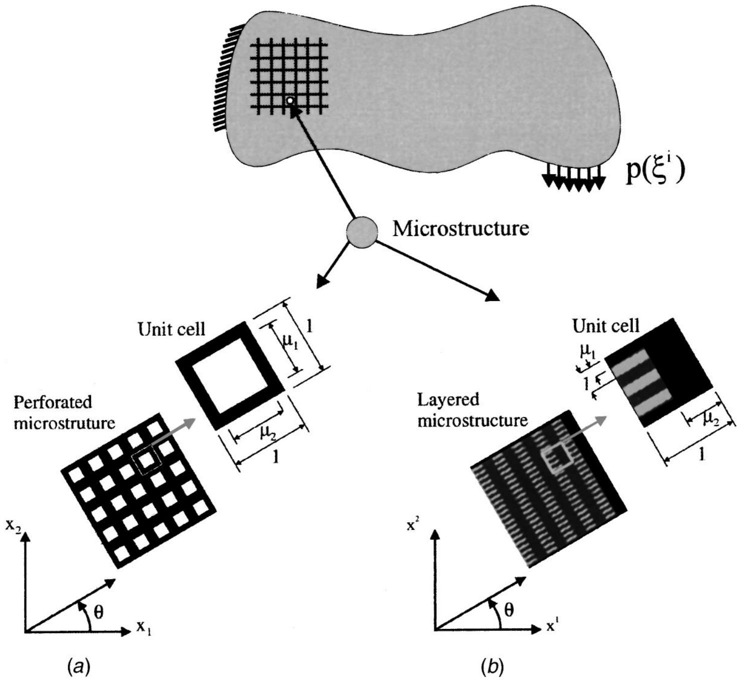


Figure 2.4: Examples of microstructures with rotation in 2D: a) *Microstructure with rectangular holes and* b) *Layered microstructure [12]*

One obvious disadvantage of the homogenization method is that more design variables per element are required than when using the density method. Also, and maybe even more serious is that currently the homogenization can only be used for optimization with the compliance as cost function or constraint [10].

2.5.3 Checkerboarding

One common problem that arises in topology optimization for both the density and the homogenization method is *checkerboarding*. Checkerboarding refers to the checkerboard pattern that is formed with alternating elements with density of 1 and elements with density of 0. The cause of this is that in the discretized finite element model the stiffness for elements just connected by the corners are greatly exaggerated [11]. This together with the common practice to use one design variable per element and using penalization of intermediate densities will lead to checkerboard patterns. This can for example not happen with shape optimization since shape optimization has global support, i.e, each design variable affect many elements.

There exists many different methods to suppress checkerboarding, this includes

- Higher-order finite element – e.g., in 2D, instead of using 4-noded rectangular elements 8-noded elements are used. One obvious drawback of this method is that more computational work is needed since the number of degrees of freedom (DOF) is greatly increased [16].

- Filter – a filter is used in each iteration that smoothes the density. The density of one element is averaged with respect to elements in the neighborhood

- Different discretization of FE-model and design variables – The mesh for the design variables are coarser than the finite elements, each design variable controls the density of more than one element [11]

- Density slope control[9] – The local gradient of the element densities is restricted to some value [17]

The filter and the density slope control method can also be modified to prevent mesh-dependency by enforcing a minimum member size.

[9]A variant of this method is implemented in Optistruct, first the optimization is performed with a constraint on the density slope, then the optimization constraint is relaxed to get clearly defined members, see Zhou [17] for details.

3 Overview of Used Tools

Here the most important software used in this thesis will be briefly described. The main program used for performing finite element analyses and optimizations is the *solver* Optistruct 10.0 from Altair Engineering [7]. There are also other available software for design optimization such as Tosca [18] and MSC.Nastran topology optimization [19]. In this thesis Optistruct is chosen as optimization software by request of Saab Microwave System. To be able to set up the problem and review the results HyperMesh and HyperView are also used. HyperMesh is the *preprocessor* which is used to discretize (mesh) a CAD model, set boundary conditions, properties and options and to set up the problem to be solved (optimization, static analysis, modal analysis etc.). From HyperMesh a file which completely describes the problem is exported and then processed using Optistruct. The results from Optistruct can then be evaluated using the *postprocessor* HyperView. A schematic overview of the workflow can be seen in Figure 3.1.

HyperMesh, Optistruct and HyperView are all part of the software suite Hyperworks 10.0, and as such they are designed to easily integrate with each other.

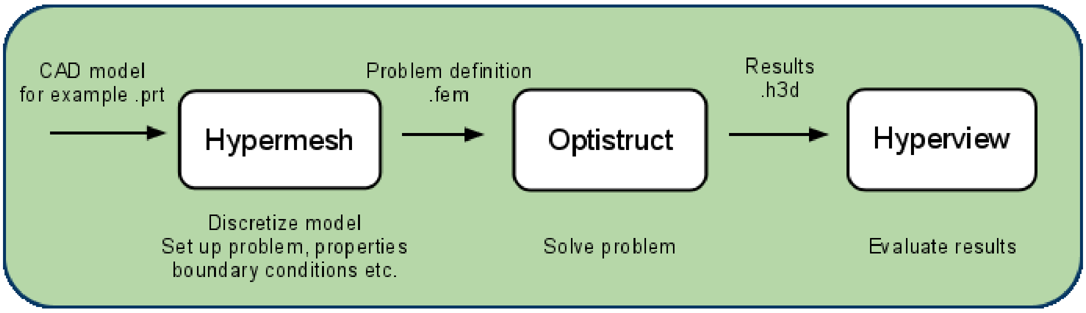


Figure 3.1: An overview of the workflow in Hyperworks

3.1 Optistruct

As mentioned before, Optistruct is the solver used for performing structural optimization. Optistruct started out as a research code at a university research lab in 1991 [20]. The only problem solved was the minimization of weighted compliance and/or eigenfrequencies using the homogenization method (the homogenization method is described in Section 2.5.2 above). In 1993 the first commercial version was marketed as Altair Optistruct 1.0. The current version of Optistruct is 10.0 from 2009.

Optistruct do not have any graphical interface, the full problem formulation is made using HyperMesh and any other options are supplied via the command line.

3.1.1 Features

Optistruct is capable of performing a range of different finite element analyses including static, modal, buckling and thermal analyses. Different types of loads such as point forces, pressure, gravitational loads, thermal loads, etc., can be applied [7].

When setting up the FE model many different types of elements are supported including: different types of three-dimensional solid elements, two-dimensional shell elements and other types of elements such as beams, bars, springs and point masses.

Optistruct is also the program that is used for performing the structural optimization in this work (see Section 2.2 for structural optimization). The supported types of optimization in Optistruct include:

- Topology optimization of 2D and 3D domains
- Free-size optimization, find the optimal thickness distribution of shell elements
- Shape optimization from user-defined shapes
- Free-shape optimization, selected boundary nodes may move without user-defined shapes
- Topography optimization, find the best reinforcement pattern of a shell structure
- Size optimization, design variable are for example a shell thickness or cross-section of beams or rods.

The method used for topology optimization in Optistruct is depending on the problem[10] either the density method or the homogenization method [20, 7]. See Section 2.5.1 and 2.5.2. Shape optimization is performed using the perturbation vector approach, see Section 2.4.1.

3.2 Usage

Here the procedure for setting up and performing an optimization using the Hyperworks software suite will be described. The types of optimization of interest are topology/free-size and shape optimization. The optimization problem in Optistruct is set up in roughly the same way for each type of optimization. The required steps are:

- Acquire/create a FE model
- Define design variables and constraints on the design variables
- Define responses that will be used as objective or constraints
- Formulate optimization objective
- If desired, set constraints on responses

Most of the engineer's work is done in HyperMesh and HyperView.

3.2.1 FE modeling

A discretized FE model will always be needed when performing an optimization with Optistruct. This is normally created by importing a CAD-model into HyperMesh where it is meshed using the available tools. The resulting FE model should strive to capture the behavior of the modeled component. This is done by representing the modeled component with different types of elements of varying sizes. There is always a trade-off between accuracy and computational cost. Details on how to perform the discretization is beyond the scope of this work. However, a few different meshes should be used to ensure sufficient accuracy.

[10] The method is chosen automatically by Optistruct depending on what responses are used

Table 3.1: Common responses used as objective or constraint

Responses from design	
Mass	can be calculated from the whole model or just from some properties
Volume	
Mass fraction	
Volume fraction	average density, only applicable for topology optimization
Responses from load cases	
Weighted compliance	is calculated from one or more static load cases
Weighted eigenfrequencies	
Static displacement	on one or more nodes
Static stress	can not be used with topology or free-size optimization[11]
Temperature	thermal responses can not be used with topology or free-size optimization
Buckling	can not be used with topology or free-size optimization

A key difference when performing a topology optimization versus a shape optimization is that for topology optimization the resulting FE model is a mesh of the whole *design domain* while for shape and size optimization the mesh is just of the existing component.

In the FE modeling phase boundary conditions, loads and load cases must also be set up. If the objective of the optimization is to, for example, maximize the eigenfrequencies, then a load step that calculates the desired eigenfrequencies must be defined.

3.2.2 Optimization objective – constraints

Both the objective and different constraints are defined in a similar way in Optistruct. First a 'response' needs to be created. A response is a numerical measure of some aspect on the design variables or an analysis on the model. The response can then be used either as an objective function or as a constraint. Optistruct supports many different responses; some of the more important can be seen in Table 3.1.

It is also possible to put geometric constraints on the design, for example:

- Extrusion constraint – forces the design to have the same cross-section in a specified direction, or along a curve
- Draw direction constraint – makes the design not have any cavities in one direction, to make it possible to manufacture the component by casting
- Symmetry constraint – symmetry in 1, 2 or 3 planes or cyclic symmetry (rotational)

3.2.3 Topology optimization

Here the steps required to perform a topology optimization will be described and a brief review of the most important options will be presented. This assumes that a FE-model of

[11] Stress constraints can be applied to topology optimization in a special way

the problem is available and different *properties*[12] are used for the design and non-design elements. All of the manual work with setting up the problem for topology optimization is done using HyperMesh.

The minimum requirements for performing a topology optimization are that a design variable has been set and that an objective function has been defined. The order of setting up the problem is not important; the following steps can be performed in any order.

Design variables A natural first step is to start with creating the design variable for the optimization. The design variable tells Optistruct which elements which are subject for optimization. Also, the design variable may have some options connected to it such as manufacturing constraints, stress constraints and minimum/maximum member size.

The design variable is created by using the corresponding dialog in HyperMesh, the design elements are chosen by selecting one or more properties in HyperMesh. Thus the design elements need to be distinguished from non-design elements by having different properties. Only solid or shell elements can be used as design elements.

Constraints on design variables Apart from the constraints on the optimization (mass constraint etc.) it is possible to set constraints directly on the design variables. These are defined in same dialog as where the design variables are created. The different types of design constraints are different manufacturing constraints such as setting a draw direction (casting constraint), extrusion constraint or symmetry constraints.

This is also how stress constraints are applied to a topology optimization problem. Additionally, minimum or maximum member size control can be applied. Minimum member control is recommended since it suppresses checkerboard patterns, see Section 2.5.3. The Optistruct help files [7] recommend a minimum member size of at least three times the element size.

Objective function and constraints Exactly one objective must be defined to perform the optimization. This objective may be to either maximize or minimize a previously defined response, see Section 3.2.2, Table 3.1 for some different responses that can be used.

One or more constraints on the responses may also be defined by setting an upper and/or lower bound.

3.2.4 Shape optimization

Shape optimization is performed similarly as the topology optimization. The main difference is in how the design variables are defined.

Design variables There are two different ways of setting up the shape optimization for Optistruct, free-shape and normal shape optimization. For a normal shape optimization a design variable is defined by first creating a *shape*, or deformation, using a module in HyperMesh called Hypermorph. With Hypermorph it is possible to create shape changes by using a wide range of methods. See Section 2.4.1 for a brief explanation of the perturbation vector approach used in Optistruct.

When a shape has been defined with Hypermorph a design variable is easily created from the shape, together with bounds on maximum or minimum magnitude of the shape change.

[12]Properties are defined using HyperMesh and describes an element by type, material, etc.

The second method is the free-shape approach. With free-shape optimization it is sufficient to choose a set of nodes on the boundary. Optistruct will then use a proprietary method [7] to automatically define shape changes to alter the boundary. Constraints can also be set on how the nodes are allowed to move, for example on a plane or along a line etc. It is also possible to use the manufacturing constraints mentioned in the previous section about topology optimization.

Objective function and constraints The method of setting up the objective function and any constraints on the responses is the same as for topology optimization, see Section 3.2.3. But with shape optimization it is possible to use some additional responses such as buckling factor or temperature as constraint or objective.

3.2.5 Size optimization

Size optimization is performed similar to topology and shape optimization. The design variables are set by choosing some numerical value to be changed such as thickness of a shell, various properties of beam, etc. These design variables can also be set to take discrete values or choose between a set of predefined values.

4 Trial Cases

To achieve knowledge and understanding of how optimization can be used in the design process a couple of different trial cases have been evaluated. The work was performed as pilot cases starting from specifications and limitations and ending up with proposals for final products. In some of the cases, specifications and limitations were only partly known. The steps to reach a reasonable resulting design using the tools of HyperWorks were to be found out during the work process. An iterative process using a method of trial-and-error was used. Many of the different approaches led to unsatisfactory results which, although they gave an increased understanding of the problem, will not be presented here.

Here the work is described step by step from the definition of the problems to the analysis of the resulting design and an evaluation of the used method.

4.1 Trial case: Fixture

The fixture is to be designed to hold a test object of weight 1 kg during vibration testing from 1 Hz to 2000 Hz. Since the fixture should not affect the test results it is desired that the first resonance frequency of the fixture with mounted test object is at least 2500 Hz. It should also be possible to fasten the fixture to the shake board, which is done with M10 bolts placed in a square pattern with 70 mm c/c. Bolt holes for the test object are also required and they need to be accessible from both sides to enable assembly of the test object. Also, the material can not be too thick at these locations since the bolts have a certain maximum length. The test object must be fastened on a plane with an angle of 45° relative to the shake board.

The goal of the optimization is to find a design of the fixture with the lowest eigenfrequency at least 2500 Hz and to make the structure as lightweight as possible. A low weight is desired to not impair the shaking properties of the shake board and 2500 Hz is chosen to have a margin of safety.

The optimization of the fixture was approximately made using the following steps:

1. Formulate requirements and objective
2. Set up design domain
3. Perform topology optimization
4. Realize concept
5. Analyze realized design

4.1.1 Design domain

The first, most simple, design domain was made to satisfy just one limitation, the sloping plane to put the test object (TO) on. A solid, five sided aluminum structure was built for this purpose. After an initial optimization the resulting structure was found to make it impossible to mount the test object, instead the domain was redesigned to meet such demands. Two different parts of the domain were then identified: The base plate (BP) which connects the structure to the shake board, and the mid part (MP) which holds the test object. It was then assumed that the BP could be taken out of the domain and considered rigid during optimization of the MP. The plan was to place the optimized structure of the MP on the BP in a later stage. Voids of width 20 mm were constructed behind the bolts points to make the bolts accessible from behind, see Figure 4.1. It was important to define these voids at an early stage since optimization without voids would result in structures in which the bolts are not accessible. The thickness of the material between the bolt points and the voids was chosen to be 10 mm.

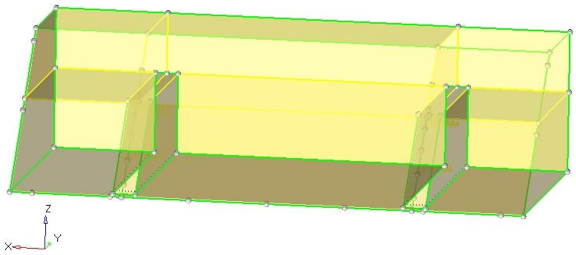


Figure 4.1: The design domain of the mid part seen from behind with the two voids for assembling bolts from behind

4.1.2 Topology optimization

The bottom of the MP was constrained in all DOFs to simulate that it was fixed to the BP. Four point masses simulated the test object.

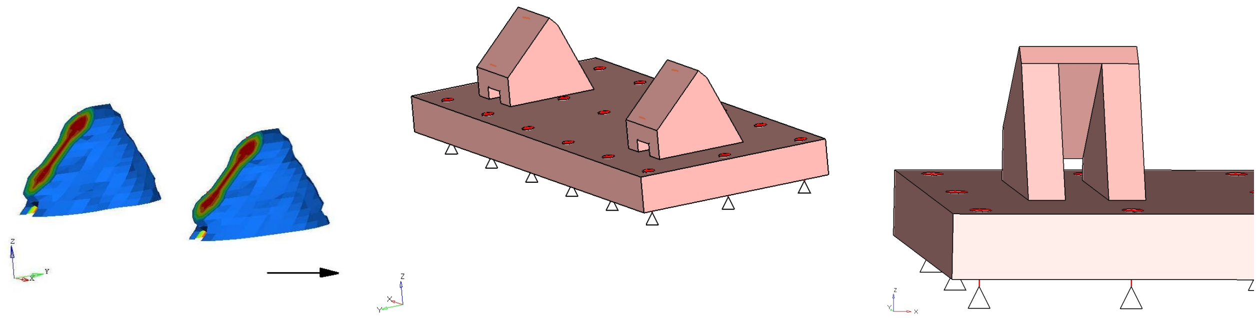
Constraining eigenfrequencies When optimization is performed to increase the frequency of the first mode the frequency of higher modes will not be taken into account by Optistruct. While the mass is decreased and the first frequency held constant during a number of optimization iterations, modes of higher order will have their frequencies decreased. This continues until the frequency of a higher mode decreases below the first mode, resulting in a new first mode. Optistruct will then only take the new first mode into account while continuing the optimization process leading to a new switch of frequencies etc. The oscillating objective function or constraint makes the objective non-differentiable which leads to a diverging solution [21].

During the work it has been found out that this phenomenon often occurs while constraining or maximizing the first eigenfrequency in this manner. However, there is a workaround of the problem using constraints on the mean value of a couple of the lowest eigenfrequencies. The constraint is then set up as

$$C_{\text{freq}} \geq \sum_{i=1}^{n} \frac{w_i}{\lambda_i} \tag{4.1}$$

where n is the number of eigenfrequencies to be accounted for, the maximum number possible in Optistruct is $n = 6$. The choice of n depends on the geometry and is found by trial and error. A too small number gives an oscillating objective while a too large number gives a longer solution time. w_i are weight factors where the default value of $w_i = 1$ is a good choice since the lower modes already are prioritized by the characteristics of the constraint formulation. Note that the reciprocal eigenvalues are used instead of frequencies and the constraining constant C_{freq} is chosen as an upper limit instead of a lower as in the case of constraining the first frequency discussed above. The value of C_{freq} is found by hand calculations but often has to be changed iteratively until the optimization gives a set of eigenfrequencies in the required range.

The penalization factor (`DISCRETE`) for intermediate element densities is chosen to 3 to get a clearly defined result and the minimum member size is chosen to 15 mm in order to get a mesh independent solution.



(a) (b) (c)

Figure 4.2: Results from topology optimization and the following realization of concept.

The optimization problem is then set up as follows:

- Design domain: mid part
- Minimize mass
- Constraints on eigenfrequencies: $\sum_{i=1}^{n} \frac{w_i}{\lambda_i} \leq 2 \cdot 10^{-9}$, $n = 6$

The computational time for this problem was 24 minutes on an 8 GiB, 3 GHz dual core computer. The resulting topology can be seen in Figure 4.2a.

The resulting first eigenfrequency was $f_1 = 2816$ Hz and the mass of the supports was about 0.4 kg. Since the BP was considered as rigid at this stage the value of f_1 had to be higher than 2500 Hz as it would decrease when the BP was introduced to the model. Many other optimizations were formulated and performed, e.g. with the test object modeled as one point mass on a rigid element, connected to the MP by bolts. The results differed in details but all led to the same conceptual design. This indicates that the result used for the next step was not a local but a global optimum.

4.1.3 Realization of concept

From the results of the topology optimization a concept of the design could be realized, see Figure 4.2. According to the results the supports should be formed as two equal units that are symmetrical and consisting of two walls of width 17 mm and a ‘roof’ of predefined thickness 10 mm, leaning with an angle of 45°. All dimensions for the concept design were translated from the topology optimization. A simple CAD model which also included the BP with bolt holes could now be made in ProEngineer. The thickness of the BP was set to 30 mm as an initial guess.

Analysis of realized concept The CAD model was then imported to HyperMesh where it was meshed, constrained by bolts and point masses were added, the mass of the aluminum structure was now 5.6 kg. An eigenvalue analysis was then performed which resulted in $f_1 = 2451$ Hz which, as expected, was a bit lower than for the MP only. However, this verifies that the realization is reasonable.

4.1.4 Conclusions

The topology optimization resulted in a design concept which, when realized, ended up in a design with the first resonance frequency in the desired range. This rough design could be further improved by performing a shape optimization. The results indicate that this method of using topology optimization with constraints on the eigenfrequencies is capable of producing a good design concept.

4.2 Trial case: Clip

The considered component is a clip used to hold a circuit board in place. The clip is fastened with double bolts. See Figure 4.3 for the original design of the clip. The material is assumed

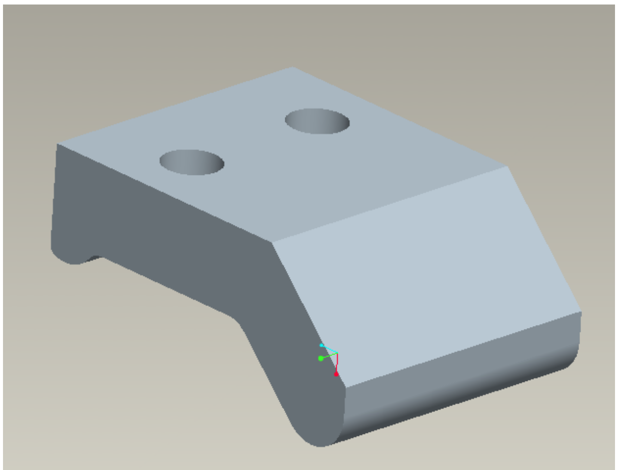


Figure 4.3: Original design of clip

to be aluminum with the following properties: $E = 70\,\text{GPa}$, $\nu = 0.33$, $\sigma_y = 245\,\text{MPa}$.

From now on only half of the clip will be considered due to symmetry. This saves both computational time and manual work required to set boundary conditions, properties, etc. The results from the FE-analysis will be the same.

4.2.1 Problem formulation

Loading The clip is fastened with two reduced M3 bolts. The cross sectional area of the reduced bolts are $3.14\,\text{mm}^2$ which can be compared to normal M3 bolts with a cross sectional area of $5.03\,\text{mm}^2$. The forces from the bolt is assumed to be same as the pre-load, which is often set to be 60 % of the maximum force the bolt can carry before yielding. This gives the load:

$$F_f = 0.6 \cdot A_b \sigma_y = 0.6 \cdot 3.14 \cdot 600 \approx 1.1\,\text{kN}$$

Boundary conditions The supports seen in Figure 4.4 are locked in the x-direction (the up direction); the rear support is also locked in the z-direction (front). At the symmetry plane, symmetry boundary conditions are applied: translational DOFs are locked in y-direction and rotational DOFs are locked along the x- and z-axis.

Objective The objective is to minimize the mass while keeping the stresses within safe levels. Safe levels are for simplicity assumed to be 80 % of the yield stress, which in this case is

$$\sigma_{\max} = 0.8 \cdot \sigma_y = 196\,\text{MPa}$$

There are also constraints on the design due to the fact that the component should be feasible to manufacture. The component is created by extrusion. Therefore the topology has to be the same throughout the material in the y-direction, with the exception of the

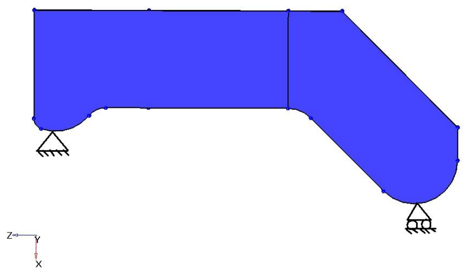
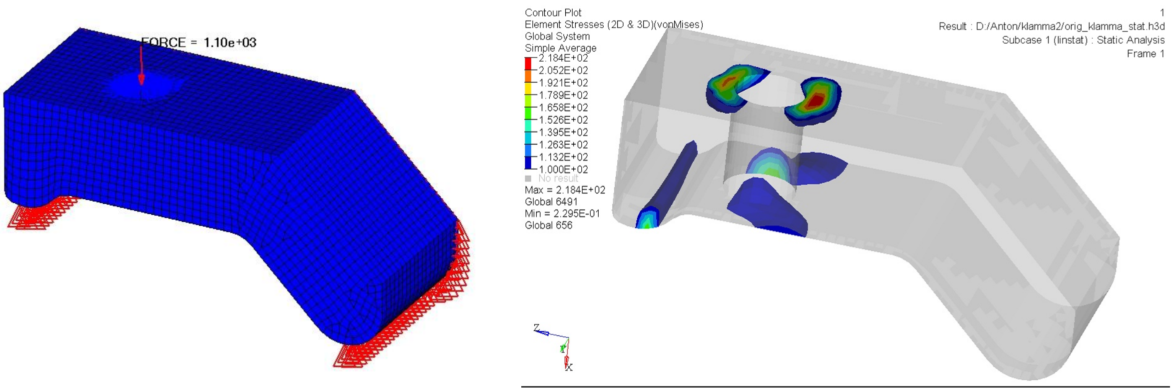


Figure 4.4: Drawing of boundary conditions



(a) *(b)*

Figure 4.5: (a) FE model of the original clip. (b) Stresses exceeding 100 MPa*. The maximum stress is about* 160 MPa*, ignoring stress concentrations at the fixed nodes and from the applied load*

holes for the bolts which are added after the extrusion. Additional holes may also be possible.

Static analysis of original design As a first step a static analysis of the original component is performed. The FE-model, complete with boundary conditions and external forces can be seen in Figure 4.5a. This also calculates the mass of the (half) clip to be 1.88 g. The force is applied at the nodes closest to the hole using rigid links and this will result in artificial high stress concentrations on those elements. Solving for the static load case results in stresses according to Figure 4.5b.

4.2.2 Topology optimization

The optimization problem was formulated in Optistruct as:

- Objective: minimize mass
- Stress constrained to a maximum of 196 MPa
- Extrusion constraint in the y-direction
- Minimum member size: 2 mm
- Intermediate densities penalty (DISCRETE) = 3

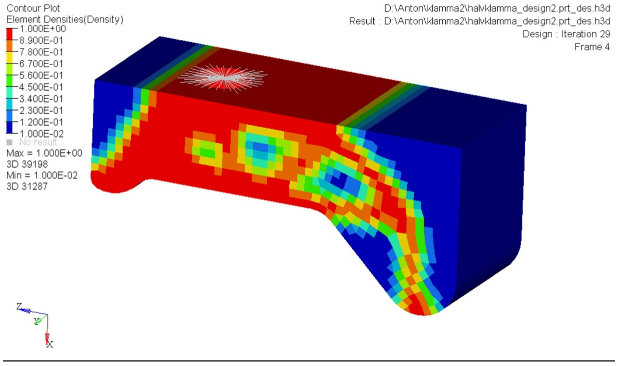
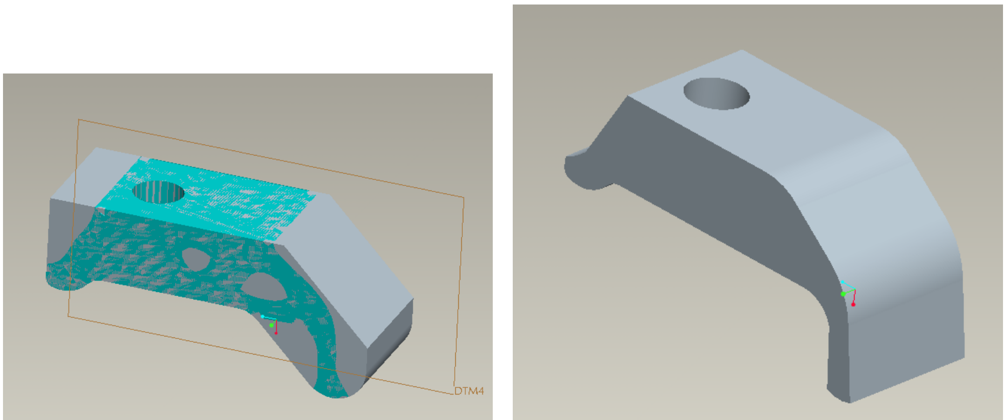


Figure 4.6: Results from topology optimization, red areas have high density and blue areas low



(a) (b)

Figure 4.7: Realization of concept. (a) The CAD-model of the clip imported along with the design concept from the topology optimization. (b) The realized design concept, created by adjusting the original clip

The design domain is chosen to be slightly larger than the original clip, although this turned out to be unnecessary as the resulting topology did not exceed the boundary of the original clip. Results from the topology optimization can be seen in Figure 4.6

4.2.3 Realization of concept

The post-processing capabilities of Hyperworks (OSSmooth) are used to produce a CAD-model from the results from the topology optimization that can be used as a guideline when realizing the concept. OSSmooth is run with a density tolerance of > 0.7, ignoring elements with a lower density. The resulting CAD-model is imported along with the model of the original design, see Figure 4.7a. The original clip model is then adjusted to the design concept from the optimization. Only the outer boundary is considered — cavities are ignored, see Figure 4.7b. The resulting clip has a mass of 1.32 g, which is 70 % of the mass of the original design.

Static analysis of realized model A static analysis is performed on the realized model, maximum stress is about 180 MPa. This is slightly higher than for the original clip.

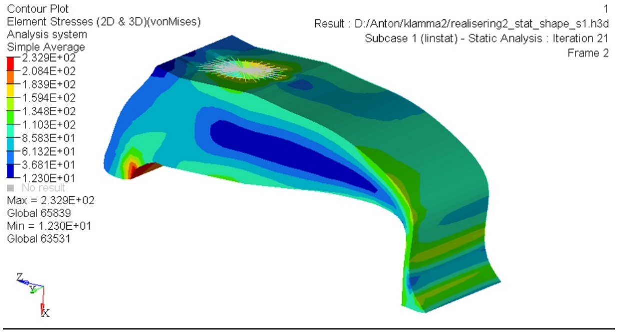


Figure 4.8: Stresses of the shape optimized clip

4.2.4 Shape optimization

To improve the design a shape optimization is performed on the realized model. This is done by using free-shape optimization in Optistruct. With free-shape optimization it is sufficient to choose which nodes on the boundary that should be able to move, it is not needed to define the shape perturbations. The objective is the same as before; minimize the mass with constraints on the maximum stress.

The resulting shape and stresses can be seen in Figure 4.8. The weight of the shape optimized clip is reduced to 1.15 g, 61 % of the original clip, the maximum stress is about 181 MPa which is lower than before the shape optimization. Note that the front end is very thin and may be at risk for buckling. A buckling analysis gives that the buckling factor is just above unity and thus the front end is at risk for buckling and should be reinforced.

4.2.5 Conclusions

This procedure of optimizing the clip was straightforward and resulted in a design that was lighter and at the same time fulfilled the requirements. No buckling requirements were defined prior to the work and the somewhat small buckling factor may be a problem. This shows that properties that are not optimized for are out of control and hence may be arbitrary bad.

If the requirements admit, an additional topology optimization could be performed with a different design domain. It would be interesting to see if an increase in the thickness of the design domain could result in a lighter or stronger design. The shape optimization that would follow should then contain buckling requirements.

4.3 Trial case: TRU

The TRU (Transceiver Receiver Unit) consists of supporting structure made of a base and sheet metals and different components. The parts are put together with bolts of varying dimensions.

The objective is to improve the TRU by optimizing the supporting structure with the objective of increasing the eigenfrequencies and decreasing the mass.

4.3.1 FE-modeling

The TRU is a complicated structure and consists of a lot of different parts held together by a large amount of bolts. Structural parts, with the exception of the base, are modeled as 2-dimensional shell elements, the base is modeled with solid elements and the electronic components are modeled as point masses with rigid links. The bolts are modeled as beam elements.

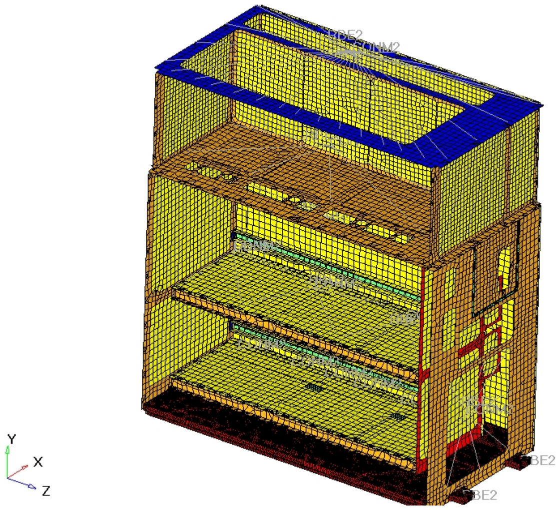


Figure 4.9: FE model of TRU unit

The weight of the complete meshed structure was 273 kg, which can be compared to the actual TRU with a weight of 274 kg. The complete meshed TRU unit can be seen in Figure 4.9.

4.3.2 Aim and loading conditions

The aim is to make the TRU lighter under the mechanical requirement that it must withstand different types of dynamical loads without any yielding of the material. For this purpose an equivalent quasi-static load of 30 g, which will cover the dynamic response of the mechanical requirements, is applied in the X-, Y- and Z-directions. The structure should also have the first eigenfrequency as high as possible, preferably over 35 Hz.

To limit the required work only the center wall is considered for optimization.

4.3.3 Topology optimization

Static and modal analyses were performed on the existing TRU. The first eigenfrequency was found to be 35.3 Hz. The mass of the original center wall is 3.5 kg.

The center wall consists of 15 mm aluminum sheet metal which has been milled to 2 mm in the middle to reduce the weight. It is therefore natural to do a topology or free size optimization where the objective is to find the optimal thickness distribution of the shell elements.

The simplest case of optimization is the minimization of compliance with a constraint on the mass. In this case the mass is constrained to the mass of the original wall and the objective is to minimize the compliance when applying a gravity load in the x- and y-direction.

The results from the optimization can be seen in Figure 4.10. The resulting compliances are $1.786\,437 \times 10^5$ J and $7.399\,670 \times 10^4$ J respectively for the x- and y-direction. This is slightly *worse* than for the original wall.

4.3.4 Conclusions

Several different ways of formulating the optimization problem of the wall were tested, among others:

- Minimization of compliance, with loading in one or several directions

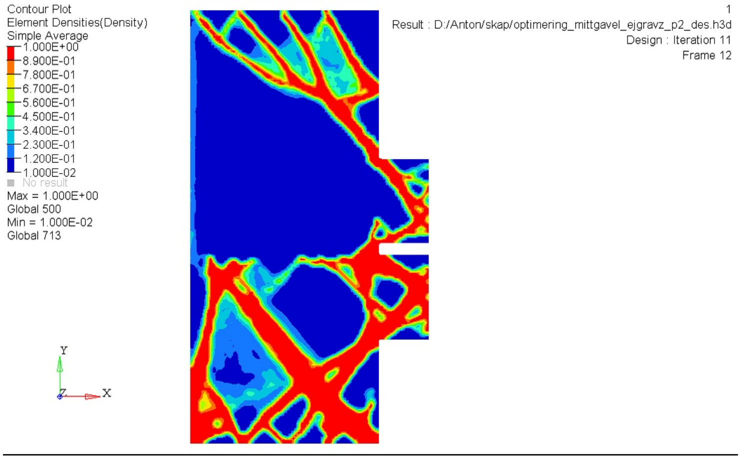


Figure 4.10: Density distribution from topology optimization of centre wall

- Maximization of the first eigenfrequency or a maximization of a weighted average of eigenfrequencies
- Minimization of mass with different constraints on eigenfrequencies, stresses, etc.
- Optimization with different definitions on the design and non-design areas
- Optimization with many different combinations of parameters

In summary, none resulted in any design that was a noticeable improvement of the original design. This was due to a number of causes. To begin with the center wall did not have as big effect on the characteristics of the complete system as initially thought. The loads that the center wall had to withstand were not very big. Also, the weight of the center wall was only 3.5 kg, which can be compared to the weight of the whole system which is 273 kg, so any substantial weight savings could not be found by optimizing the wall.

Due to the many design constraints on the wall, mostly from prescribed bolt holes (45 holes), there were not very much that could be optimized. There was a minimum thickness of 2 mm to get a shielding wall, and a thickness 15 mm to be able to put the bolt holes. When the material for the base thickness and extra material near bolt holes have been used there is not very much left that can be changed.

Finally due to the complexity of the system the resulting optimization problem was highly non-linear and difficult to solve. For example, when some material is removed the wall will be less stiff and this results in that the forces of the wall decreases since the other parts instead are taking care of those forces. Often the optimization resulted in results that were worse than for the original wall (as in the example above).

4.4 Trial case: Strut for airborne radar

The struts for an airborne radar are to be studied and optimized for increased performance and minimized weight. The work is divided into three parts: conceptual optimization, detailed optimization, and finally, an analysis of the resulting structure.

4.4.1 Geometry

The strut is an extruded aluminum profile with a steel tube along the center connected to one circular beam of high strength steel at each end. In the small gap between the aluminum profile and the steel tube a sticky filling is used. The beams are fastened to

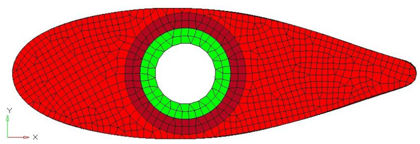


Figure 4.11: (a) Strut with steel beam seen from the side. (b) Cross section of the design domain, extruded holes of the original design are filled. The steel tube consists of the green elements. Red elements are made from aluminum, of which the light red elements are elements subjected to topology optimization.

Table 4.1: Load cases for whole strut, margin of safety is included in the loads

Load	**Description**
Air load	Distributed along the front edge
Symmetric bird load	Acting at the middle of front edge
Asymmetric bird load	Acting at middle of front edge and side
Compressive force	Acting at the two rod ends

the radar unit and aircraft body by rod ends. To decrease computational work symmetry is used at the middle of the strut on a plane perpendicular to it. The geometry of the problem is seen in Figures 4.11a and 4.11b.

The outer boundary of the original structure consisting of the aerodynamic profile and the inner radius of the steel tube is used when defining the design domain for topology optimization, see Figure 4.11b. Discretized, the domain consists of 72 242 elements and 75 447 nodes giving 225 441 degrees of freedom.

4.4.2 Load cases

There are four load cases that may be critical to the strut while in service and therefore have to be taken into account during optimization. A summary of the load cases is found in Table 4.1

The bird loads used here are higher than those designing the original strut. The bird will split by the cutting edge of the original strut meanwhile the profile considered here will result in a more energy consuming deformation generating higher loads.

Bird model The loads are based on a strike of a cylindrical shaped bird of length $L = 2D = 213\,\text{mm}$ and a mass of 1.8 kg. The velocity of the bird relative to the aircraft is 128 m/s. The bird strike generates dynamic loads on the strut and the equivalent static loads have been calculated prior to this thesis work.

Symmetric bird strike The strut must be able to withstand the loads created when a bird strikes the middle (at equal distance from the radar and the aircraft hull) of the strut at the same time as air loads are present. For a symmetric hit, the scenario when the bird hits the middle of the strut is supposed to be the most critical and is therefore designed for. The positioning of the air load and bird load are seen in Figure 4.12 and 4.13.

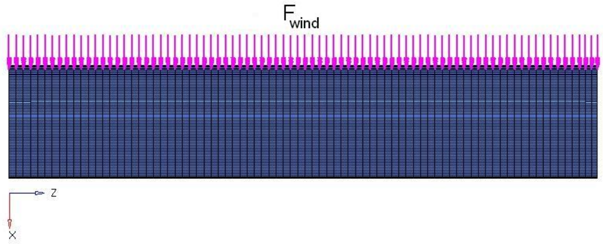


Figure 4.12: The air load applied on the front are seen from the side. Constraints are not visible.

Asymmetric bird strike A bird may just as likely hit the strut a bit off-center to the right or the left on the front. As in the case of a symmetric strike, a hit on the middle of the length of the strut is considered the most critical scenario.

Linear buckling The critical buckling load is specified to be more than $P_c > 108.8\,\text{kN}$. Buckling analysis however, can not be performed during topology optimization in Optistruct and therefore a measure of the bending stiffness of the strut in the direction of the first buckling mode is introduced. In this way the area moment of inertia can be designed. Based on Euler's first buckling case and Euler-Bernoulli beam theory the largest affordable displacement at the symmetry plane of the strut due to a force at the same location is calculated, see Figure 4.13b. Using $F = 1\,\text{kN}$ in (4.2) and (4.3) gives $u_{y,max} = 7.40\,\text{mm}$. This method of dimensioning the strut for buckling has been proven to work out accurately in a reference analysis. Euler's first buckling for half of the strut is seen in equation (4.2):

$$P_c = \frac{\pi^2 EI}{4L^2} \rightarrow I = \frac{4P_c L^2}{\pi^2 E} \tag{4.2}$$

Insertion in the governing Euler-Bernoulli beam equation using given values results in (4.3):

$$u = \frac{FL^3}{3EI} = \frac{FL\pi^2}{12P_c} = 7.40\,\text{mm} \tag{4.3}$$

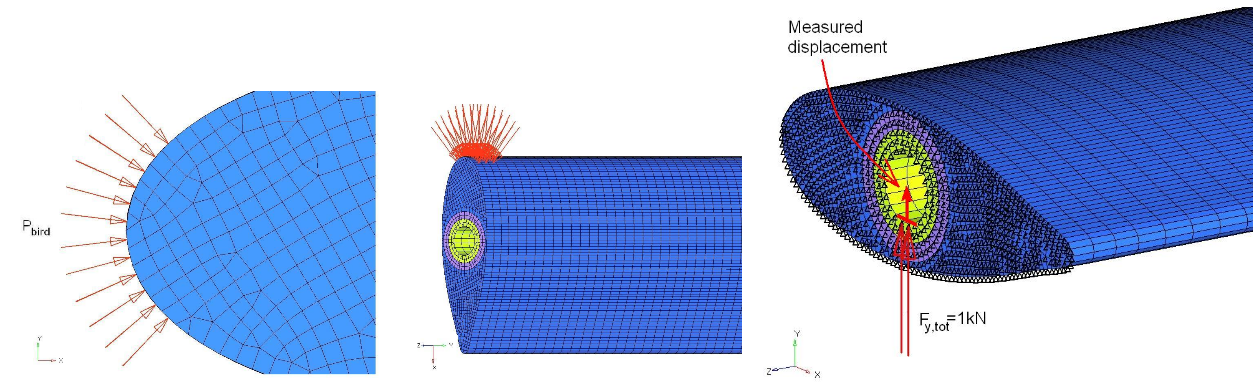
4.4.3 Aim and Limitations

The strut, as it turns out, is constructed stable enough to withstand the load cases above without stresses exceeding the yield stress of $\sigma_s = 225\,\text{MPa}$. What is interesting is to find out if it is possible to find a structure that is lighter and still fulfills the criteria of buckling load and maximum stress. A safety margin of 1.5 is already added to the given loads.

The structure is limited by the aerodynamic outer profile and the inner steel tube. It is also defined that there must be a minimum thickness of aluminum around the steel tube. The fastening layer between the aluminum and steel is considered as ideal. Further the structure must be able to be constructed by extrusion and it must be symmetric in the direction of travel to avoid lateral forces.

4.4.4 Conceptual design

This section describes how a concept has been formed.



(a) (b)

Figure 4.13: (a) The symmetric static pressure generated by bird. Together with the air load this describes the first load case. (b) The force and measurement position for dimensioning of buckling properties.

Optimization Topology optimization in Optistruct is used to find the basic topology of a structure so that a concept can be refined. The cross section of the design domain can be seen in Figure 4.11b. When running topology optimization with extrusion constraint, it is not possible to define pattern grouping constraints such as symmetry at the same time.The only way to still get a symmetric structure is to define load cases of both signs and optimize with respect to both of them. The two optimization approaches minimization of volume with constrained stresses as well as minimization of compliance with constrained volume fraction were tested. It was found out that the first method gave a topology which seemed unrealistic and had a large proportion of element densities between 0.2 and 0.6. Therefore compliance minimization and volume fraction constraint have been used for the final and satisfactory topology optimization, set up as follows:

$$\begin{cases} \min\limits_{\boldsymbol{x}} f(\boldsymbol{x}) \\ \text{s.t.} \begin{cases} \text{volume fraction} \leq 0.4 \\ u_{y,max} \leq 7.8\,\text{mm} \end{cases} \end{cases} \tag{4.4}$$

where $\boldsymbol{x}$ is the vector of design variables, $f(\boldsymbol{x})$ is the weighted compliance of the load cases symmetric bird strike, asymmetric bird strike left and asymmetric bird strike right with weights 1, 0.5 and 0.5 respectively. The constraint of u_{y-max} is active just for the loads designing for buckling, which are defined as a symmetric pair. The resulting topology is seen in Figure 4.14. Minimum member size is set to 10 mm which is three times the typical length scale of the elements and the penalty parameter `DISCRETE` for intermediate element densities is set to 3 in order to get a clear result.

As seen in Figure 4.14 the result suggests that material shall be put all the way around the outer boundaries with a large reinforcement at the back tip. A concept is refined from these observations and realized in a CAD model using ProEngineer.

4.4.5 Detailed design

In counterpart to the conceptual design the exact mass and properties of the strut are now interesting. The concept acts as a starting point for the detailed optimization where shape and size of members are to be fine-tuned. The load cases are the same except for the case dimensioning for buckling since it was found out that the structure is safe in that

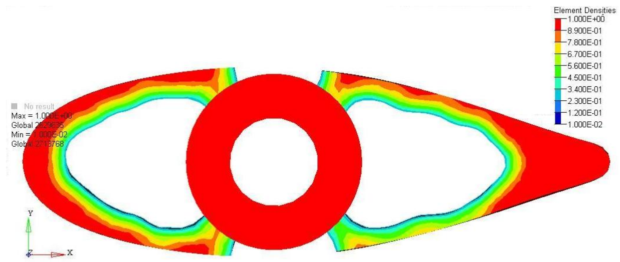


Figure 4.14: The resulting topology from problem 4.4. Even though the problem was stated to give a symmetric result it is not entirely symmetric. This is an unwanted consequence when using extrusion constraint, constraint of minimum member size and a low volume fraction resulting in not enough material to fulfill the minimum member constraint everywhere without changing topology.

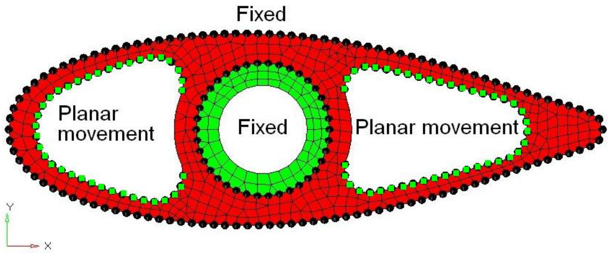


Figure 4.15: Here an almost uniform material thickness has been used around the boundaries except at the back tip were a reinforcement is situated. Material is also put around the steel tube to hold it fixed. The designable and fixed grid points are highlighted. The cross section is constant over the length of the strut (into the paper).

regard and the designing load case therefore could be removed. Design variables are now the grid points at the inner surface of the streut except on surfaces around the steel tube, see Figure 4.15. The outer boundary and the boundary of the steel tube are also fixed. The optimization is constrained to move the designable nodes in a plane perpendicular to the length axis. Demands on symmetry and possibility to extrusion are also present. The optimization problem is set up according to equation (4.5).

$$\begin{cases} \text{minimize mass} \\ \text{s.t.} \quad \max \text{vonMises} < 225\,\text{MPa} \end{cases} \tag{4.5}$$

The optimization did not converge due to element distortion. Therefore three concepts with small differences were designed in ProEngineer and the best was the last one which managed to do 7 iterations before the elements got too distorted, the result is seen in Figure 4.16. This shape was also considered to be the most optimal one under present circumstances.

A simple study of the mesh dependence is performed by making a coarser mesh of the concept and run the free shape optimization on that. It is assumed that a coarser mesh is more robust against element distortion due to change in shape of the boundary. However this turned out to be a false hypothesis in this application as the optimization ended prematurely after just 5 iterations. The resulting shape also differs from the one with a finer mesh which may be a result of the change in step size which is performed automatically by Optistruct, partly based on mesh size.

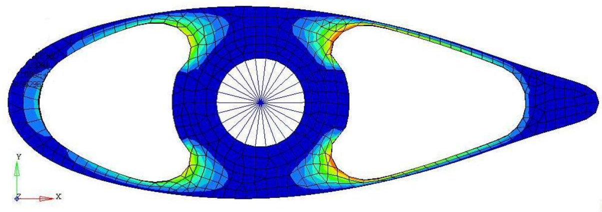


Figure 4.16: The shape change of optimized strut cross section. Shape change is proportional to the color.

Table 4.2: Properties for the resulting structures

Response	**Original**	**Fully optimized**	**Shape optimized only**
Mass	14.7 kg	13.8 kg	13.7 kg
Max stress, sym. birdload	258 MPa	420 MPa	258 MPa
Max stress, asym. birdload	292 MPa	560 MPa	309 MPa
Max disp. sym. birdload	9.7 mm	10.7 mm	11.5 mm
Max disp. asym. birdload	25.5 mm	24.5 mm	26.4 mm
Critical buckling load	183 kN	214 kN	180 kN
First resonance	27.3 Hz	30 Hz	28.1 Hz

Optimization of original design Free shape optimization is used to find out whether there is room for improvement of the original strut or not. The same load cases and optimization problem formulation as above is used. The shape change was found to be rather small indicating that the design was well dimensioned from the beginning. The results in absolute numbers are seen in Table 4.2.

4.4.6 Analysis

A static analysis of the properties of the resulting structures is performed to conclude if any improvement has been attained. The measured responses are mass, maximum stress and maximum displacement for each load case, critical buckling load and first resonance frequency. The results are presented in Table 4.2.The cause of the large maximum stresses is that they are found in the steel tube which has much higher yield limit than the surrounding aluminum.

The properties are very much alike until it comes to stresses and critical buckling load. The fully optimized structure has very high stresses in the steel tube caused by bird loads and it has very good buckling properties. The shape optimization of the original structure has made it a little bit lighter while the mechanical properties remain almost unchanged. The aerodynamic aluminum sheet is included in the masses of the original and the shape optimized only structures. The weight of the aluminum sheet is 0.5 kg.

As seen in Figure 4.16 the walls at the back sides of the strut are made very thin with risk of giving the structure local eigenmodes with low eigenfrequencies. However, the first local mode turns out to be at 950 Hz which is considered to be far above the range of danger.

4.4.7 Conclusions

Minimization of mass combined with stress constraints when performing topology optimization is an approach which may lead to infeasible designs causing error in the optimization process in Optistruct.

One problem which needs to be kept in mind while performing free-shape optimization is the risk of getting distorted elements; this is not handled in a nice way in Optistruct which simply crashes.

The only way to produce a convergent solution on a problem like this is to first study the result of the topology optimization very carefully while designing the concept to limit the shape change in the following detailed optimization. In addition a mesh of good quality and robustness must be produced.

Free shape optimization is a relative time consuming method when compared to the corresponding topology or shape optimization, about 8 times longer per iteration when the number of elements is almost the same.

4.5 Trial case: Plate for thermal conduction

The possibility to optimize a structure with respect to both thermal and mechanical demands is studied next. The geometry is found from a baseplate of aluminum for a number of electronic devices and antennas. The plate is supposed to transfer heat from the electronic components to the bottom where it is attached to a cool surface. The difference in temperature between the cool surface and the components must not be larger than 30 K, and the first resonance frequency must be at a reasonable level. The objective is to minimize the weight.

Optimization with respect to heat transfer is a completely new feature in Optistruct and currently the only available methods are shape and size optimization. This limitation makes it impossible to perform a complete conceptual design through topology or free size optimization as done in the other trial cases. Therefore a different approach is to be used by first using free size optimization for mechanical properties and then freeze the obtained mechanical structure and perform a size optimization on the plate.

However, the results from the thermal optimization turned out to be difficult to interpret and also resulted in designs that were hard to find trustworthy or reasonable. Test optimizations using other simpler geometries also resulted in confusing outcomes. Very unintuitive designs were suggested by Optistruct with violated constraints and poor results in terms of heat conduction properties. Therefore the trial case was closed with hope for future improvement of the software by Altair.

4.6 Trial case: Mounting base for rotating radar

The mounting base for a rotating radar used in service for many years is taken into consideration for a redesign. The mounting base connects the antenna to the revolving board and is seen in Figure 4.17. The mounting base, which was designed by 2D CAD a few decades ago, is made of cast aluminum and has a weight of 22 kg and the mass of the antenna is 337 kg.

The present design has a few problems with accessibility of some of the bolt holes and there may be potential for decreasing the weight. The objective is to come up with a conceptual design that is improved at these points under the limitations that contact surfaces must be kept, the inside must be protected from electro magnetic radiation and three service holes as seen in the figure must be present. The structure must also be possible to construct mainly by casting. The only mechanical specification is that the mounting base with the antenna mounted on it must be able to withstand an acceleration of 30 g in any direction without getting stresses near the yield stress of $\sigma_s = 214\,\text{MPa}$.

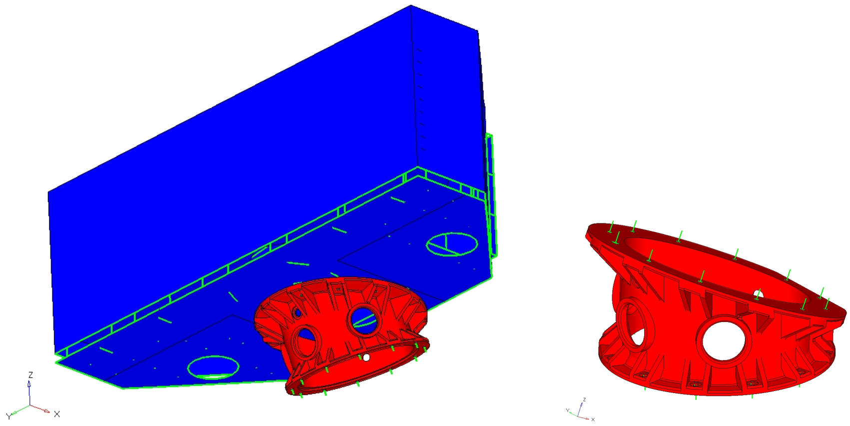
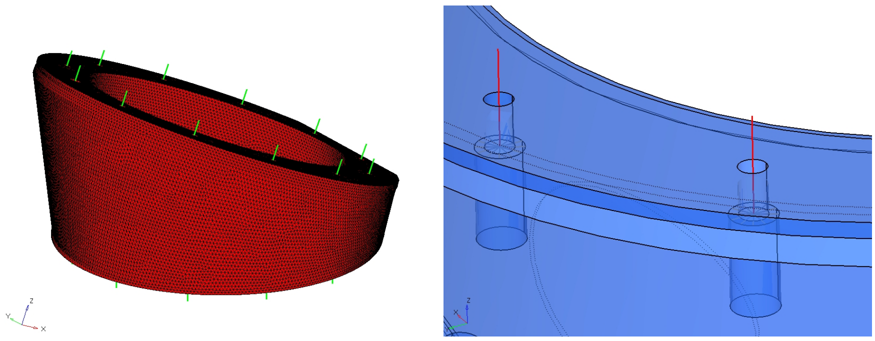


Figure 4.17: The present outline of the mounting base and the antenna on top of it.



(a) (b)

Figure 4.18: (a) The solid design domain without service holes. (b) Zoom at two bolt positions and their respective bolt holes and accessibility voids.

4.6.1 Approach

During the work with this trial case a more automatized approach will be used compared to some of the other trial cases. A large number of optimizations with slightly differing setups will be performed followed by a selection of the most effective design to be used as concept. The concept will then be realized and optimized further.

4.6.2 Design domain

The design domain is chosen as the upper and lower contact surfaces of the mounting base and the tube connecting them, see Figure 4.18. The three service holes are not introduced at this step to make it possible to investigate if there might be better positions for them. Bolt holes and voids to be able to put the bolts in position are introduced. Inner surfaces and contact surfaces will not be included in the design domain and a symmetry constraint is introduced to keep the one plane symmetry of the structure. In order to exclude mesh dependency, two different meshes were used in the beginning and the finer one was used for the continued optimizations.

4.6.3 Loadcases

Accelerations of 30 g in different directions are introduced to the optimization to produce stiffness in any direction. After a study of robustness, see Section 5.5, where different load cases were weighted differently in a number of optimization runs it was found that four different directed accelerations accounted for in the optimization gave good results for all directions. Those were accelerations in the X-,Y- and Z-directions complemented with an acceleration in the XY-plane directed 20° from the X-axis. Using more load cases would not improve the structure but only increase the computation time.

4.6.4 Concept generation

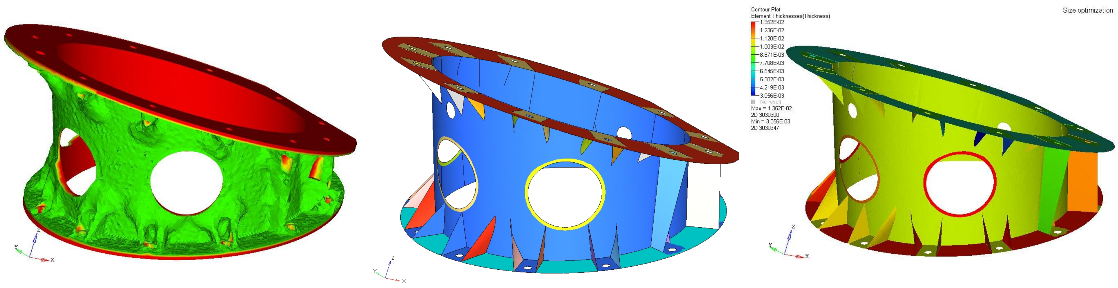
The topology optimization problem was set up as follows:

$$\min_{\boldsymbol{x}}(f)\ , \quad f = \sum_{i=1}^{4} c_i(\boldsymbol{x})$$
$$\text{s.t.} \begin{cases} \text{volume fraction: } V \leq \texttt{volfrac} \\ \text{minimum member size: } d_{max} \geq \texttt{mindim} \end{cases}$$

where `volfrac` and `mindim` are used in a number of different combinations, c_i are the compliances for the individual load cases. The relation between f, `volfrac` and `mindim` were then studied to find the most optimal combination. The total compliance, f, was plotted against `volfrac` and `mindim` leading tho the knowledge that using `volfrac` $= 0.175$ and `mindim` $= 1.75$ cm gives a structure that is little improved by adding mass or decreasing minimum member size and yet is relatively easy to interpret. For an example of such a plot, see Figure 5.1. Positions for the service holes could then be defined at the front and sides of the structure since no reinforcements were formed there. However, the optimized structure contained a number of cavities making it impossible to construct by casting and a redefinition of the design domain had to be done to deal with this problem. Service holes were added, and the design domain was divided into 11 separate design domains were each domain was constrained to be able to produce by casting. Each domain got its own draw direction and a new automatized parameter study was performed, this time also studying the impact of the parameter `maxdim`. This parameter is used not to get too large material formations which can not be made by casting. With a maximum member size of 3 cm, a minimum member size of 1.5 cm and `volfrac` of 10 % a simple and well performing structure was formed, see Figure 4.19a. The resulting topology was then interpreted and realized in a shell model, see Figure 4.19b.

4.6.5 Detailed optimization

The concept was then objected to size optimization where 25 surfaces of the structure had variable thicknesses. The different regions are seen marked with different colors in Figure 4.19b. In the detailed design process the structure is meant to be fine tuned to the exact requirements which in this example means that no stresses may exceed the yield stress of $\sigma_s = 214$ MPa with a safety factor of 1.5. To optimize for this the maximum allowed stress in the structure except for elements close to bolt holes and corners was set to be exactly the yield stress. The optimization was set up to minimize mass with constrained stress and all design variables were allowed to vary between 0 and 1.5 cm. The optimization result is seen in Figure 4.19c.



(a) (b) (c)

Figure 4.19: (a) Optimization result with manufacturability constraints and service holes. (b) Realized concept divided into separate properties. A shell model is used to save time and make detailed optimization simpler. (c) The size optimized concept, the color of each surface indicate the material thicknesses from blue at 3 mm *to red at* 13 mm.

Table 4.3: Properties for the resulting structure

Response	**Original**	**Optimized**
Mass	21.9 kg	16.2 kg
Max stress, X-acc.	45 MPa	110 MPa
Max stress, Y-acc.	60 MPa	120 MPa
Max stress, Z-acc.	40 MPa	93 MPa
Sum of compliance	305 Nm	400 Nm

4.6.6 Analysis

In Table 4.3 the properties of the shape optimized and the original structure can be seen. When using the stress constraint described above the non local stresses became less than $\sigma = \sigma_s/1.5 = 142$ MPa as desired.

4.6.7 Conclusions

Automated study of the impact of different parameters is a powerful tool in the beginning of the optimization process. Instead of using trial and error to find the best optimization setup an automated generation of optimizations with parameters in a specified range makes it possible to choose the best setup with less effort and better result. In this trial case it is seen that the original design might be over sized. A lightening of 5.7 kg gave rise to perhaps too high stresses but a product lighter than the original still meeting tough safety demands can most likely be produced. Matters of robustness of this structure are discussed in Section 5.

4.7 Trial case: Strut fitting

The considered strut fitting is the link between the fuselage and two supporting struts similar to the strut investigated in Section 4.4.

4.7.1 Problem formulation

The current strut fitting is made of steel and weighs approximately 4.73 kg. As this component is mounted on an aircraft it is crucial that the weight is kept to a minimum. The objective is to minimize the mass while still fulfilling the requirements on allowed stresses. The loading conditions are given as 8 different load cases, 4 for different bird strikes and 4 load cases due to different steering maneuvers. For simplicity it is assumed that the stresses are allowed to have about the same magnitude as for the original strut fitting. The stresses of the original strut are about 200 MPa for the maneuvers. For the bird strikes the material is allowed to have some plastic deformations, the largest resulting stresses for a linear static analysis of the original fitting is about 800 MPa.

Table 4.4: Load cases for the strut fitting from maneuvers, in the axial direction

Load	**Description**
LC13	Maximum tensile force in the front strut
LC14	Maximum compressive force in the front strut
LC3	Maximum tensile force in the diagonal strut
LC4	Maximum compressive force in the diagonal strut

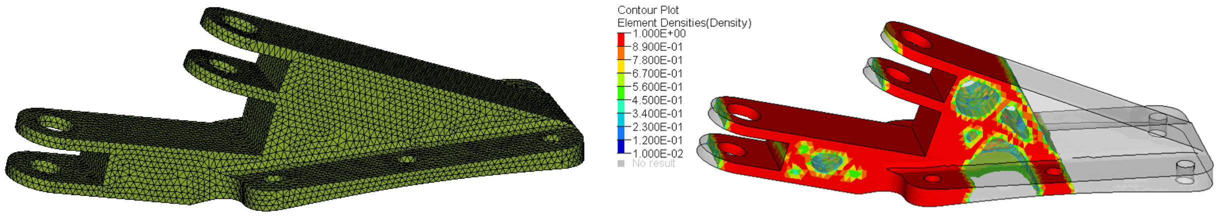
Table 4.5: Load cases for the strut fitting from bird strike

Strut	**Load case**
Front strut	Symmetric impact
Front strut	Asymmetric impact
Diagonal strut	Symmetric impact
Diagonal strut	Asymmetric impact

4.7.2 Topology optimization

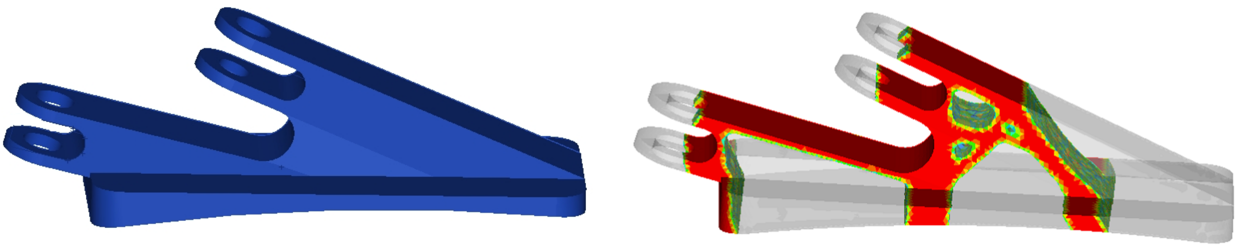
The objective in the optimization problem is to minimize compliance while the material volume fraction is constrained. This optimization is performed for a couple of different volume fractions to find a structure with the stresses in the specified range. To find a 'nice' topology that is feasible from a manufacturing point of view an extrusion constraint is applied and the minimum member size is set to 12 mm. The design domain is chosen to be a slightly modified version of the original strut fittings, see Figure 4.20a.

A volume fraction of about 50 % results in acceptable stresses from the load cases. The resulting topology can be seen in Figure 4.20b. One obvious difference from the original design is that the two rear bolts are not included. Note that the strength of the bolts is not included in the model and it is therefore needed that an additional analysis of the bolts is performed. The mass of the design concept is 3.33 kg.



(a) (b)

Figure 4.20: (a) The design domain and (b) the result from the optimization



(a) (b)

Figure 4.21: (a) New design domain and (b) the result from the optimization with extrusion constraint

4.7.3 New design domain

From the initial topology optimization it is easily seen that it would be much more effective if the base plate could be moved forward. A new design domain is proposed with the base plate extended in the front. See Figure 4.21a. Topology optimization with the objective to minimize the compliance with the volume fraction constrained is performed. To prevent internal cavities some sort of constraint is also needed on the design variables. Two different approaches are tested: using a draw direction constraint and by using an extrusion constraint. The resulting design concept with applied extrusion constraint can be seen in Figure 4.21b The draw direction constraint results in a topology that is highly unsymmetrical but inner cavities are effectively suppressed, see Figure 4.22. The resulting stresses are acceptable.

The mass is 2.3 kg for the concept with extrusion constraint and 1.5 kg for the concept with draw direction constraint. Additional material is needed to make the base plate, fillets and other details needed but it is evident that a resulting finished design will be lighter than the original design which have the mass 4.73 kg.

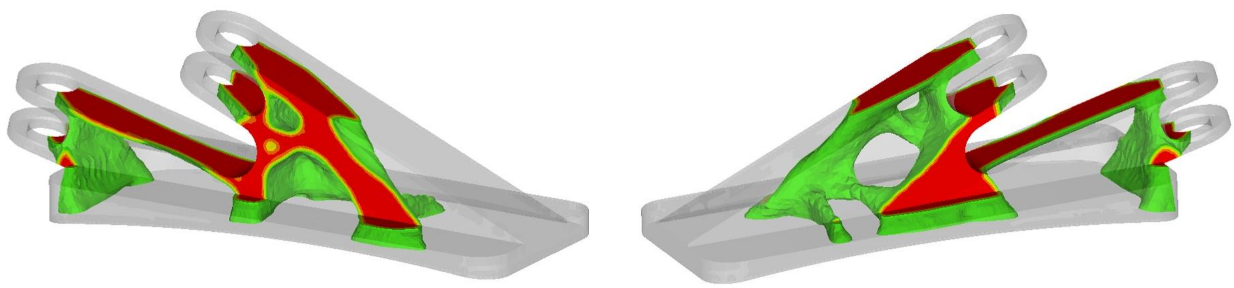


Figure 4.22: Resulting topology seen from the left and from the right. Draw direction constraint applied

4.7.4 Conclusions

From the initial topology optimization important knowledge regarding the structure were found. It is seen that the base plate possibly is placed in a suboptimal position and that the structure may get lighter and stronger if the plate were to be moved. Just by looking at the problem this insight is not easily realized. The mass of the design concept cannot be directly translated to the mass of the finished product; additional mass might be needed for fillets or for manufacturability reasons. But it is clear that a re-design have the potential to decrease the mass considerably.

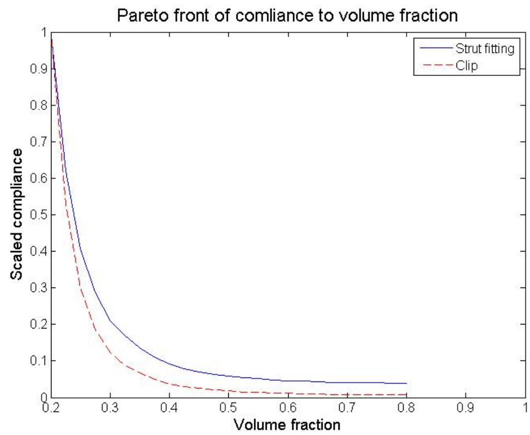


Figure 5.1: The relation between compliance and `volfrac` *for the trial cases strut fitting and the clip. Values are scaled such that the maximum is unity. Plots like these are very valuable when deciding the final weight of the structure contra its stiffness.*

5 Parameter Study

When performing a topology optimization using Optistruct there are many different parameters and options that affect the end result. To get an increased understanding of the impact of these parameters a parameter study was conducted. The problems that seemed suitable for a parametric study are some of the trial cases, see Chapter 4. Each parameter of interest is treated in separate subsections.

Automated processing To automate the process of testing different combinations of parameter values a simple script was created using the programming language Python. This script can be used to test many different parameter combinations without the need of any human interaction after the initial set up. This makes it possible to perform lengthy calculations during off-work hours and also reduces the risk of simple typing errors and mix-up of different files. The script source code and a brief explanation can be found in Appendix A.

5.1 Volume fraction

The parameter `volfrac` controls what volume fraction of the initial design domain that shall be used in the optimized structure. The parameter can be introduced to the optimization setup either as the objective or as a constraint where the highest and/or lowest values are specified. When used as constraint it is always satisfied prior to mechanical constraints such as eigenfrequencies and deformations and it therefore has a very big impact on the final solution.

The connection between `volfrac` and compliance of different structures has been studied resulting in the Pareto fronts in Figure 5.1. The shape of the relation curves have been found to be much similar for entirely different structures but with a slightly more linear behavior for structures where the design is predefined to a larger extent.

This kind of study can be interesting in the early stage of topology optimization since it gives a measure of the possibilities of the continued optimization work. It is also interesting to study the relation of mass and, in this particular case, stiffness; for example to decide if an increase in stiffness is worth the required increase in mass.

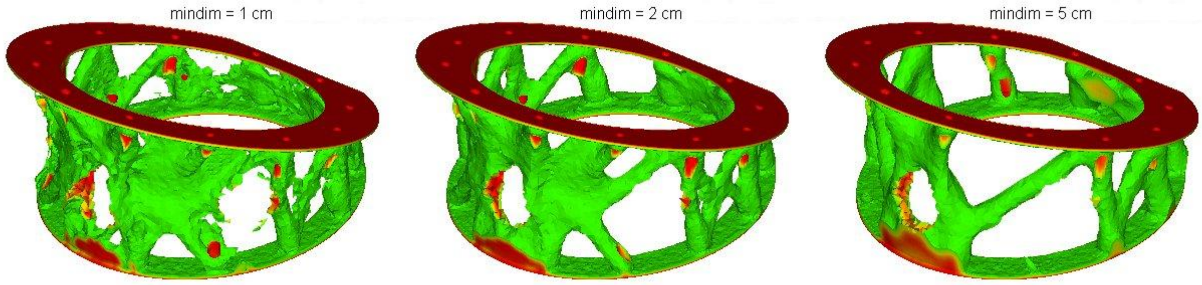
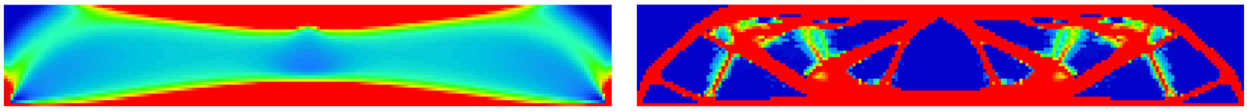


Figure 5.2: The effect of the minimum member parameter, **mindim***, applied on the mounting base trial case, see Section 4.6. All other parameters are fixed. Increasing minimum member size leads to a simpler but worse performing topology.*



(a) DISCRETE *= 0* *(b)* DISCRETE *= 2*

Figure 5.3: An increased DISCRETE *leads to a clearer distinction between solid and void elements*

5.2 Minimum member size

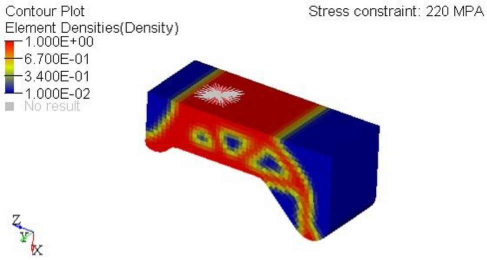
The minimum diameter any member of the structure is allowed to have is constrained with the parameter mindim in Optistruct. The benefits of using a sufficient large mindim are mesh independent solutions, checkerboarding is suppressed and more clear results that are easier to realize are obtained. On the other hand, the optimization will need more iterations to reach convergence, the solution will lose optimality and structural members such as thin surfaces and walls may not be formed. Therefore a minor parameter study should be done even when the desired minimum member size of the structure is known due to product specifications.

An example of the impact of mindim on the result in topology optimization can be seen in Figure 5.2.

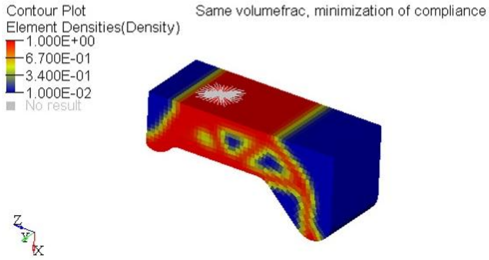
5.3 Penalization of intermediate densities – DISCRETE

An important parameter for topology optimization is the penalty factor that penalizes intermediate densities. The definition of the penalty factor can be found in Section 2.5.1. In Optistruct the setting corresponding to the penalty factor is known as DISCRETE and is related to the penalty factor as DISCRETE $= (p - 1)$ [7].

The DISCRETE parameter is employed in order to achieve a more clear structure. A low value will result in a structure with a large proportion of elements with intermediate densities, meanwhile a higher value will give a structure which may be easier to realize due to a slightly more discrete density distribution, see Figure 5.3. The value may be varied between zero, which is the default, and up to about four. Larger values may lead to unintuitive structures with bad properties. Since a higher DISCRETE also leads to checkerboarding some checkerboarding suppression method should be used.



(a)



(b)

Figure 5.4: (a) Minimization of mass with stress constraint. (b) Minimization of compliance with the same mass as in (a)

5.4 Stress constraint in topology optimization

In Optistruct it is possible to put a constraint on the maximum stress in a structure. This could be very useful since it is often the case that the problem is to minimize the mass while not exceeding some requirements on the stresses. This constraint uses a method to filter out high stress concentrations around point loads, boundary conditions and stress concentrations due to the geometry [7].

Different optimizations with the objective of minimizing the mass with a stress constraint have been performed. The results indicate that Optistruct performs an optimization with the objective of minimizing the compliance and then tries to find a volume fraction that gives acceptable stresses, i.e., if the stresses are too large more material is added by increasing the volume fraction. See Figure 5.4 for an example of a comparison of optimization with stress constraint and a normal optimization for minimizing the compliance.

5.5 Robustness

A solution which is stable to changes in designing conditions or performs well for conditions which it is not designed for is said to be a robust solution. Such conditions can be material properties, loads, failure of some part of the structure or failure of an adjacent structure. Note that a structure that is optimal in one particular case cannot be optimal if the conditions are changed, i.e., a structure that is more robust is also less optimal for the design case.

In this thesis work robustness due to failure of an adjacent structure and differing loads have been studied. The result of the studies are individual for the exact problem but gives an understanding of the method to study robustness.

5.5.1 Designing load cases

Which load cases that are active when optimizing the mounting base, see Section 4.6, were varied to study differences in the resulting topologies. A number of optimizations were performed and the resulting compliances of each load case were plotted for each optimization setup. The change in compliance and topology with respect to the optimization set up is a measure of the robustness. In Figure 5.5 the variation of compliances with respect to acceleration direction is seen. Several optimizations were performed with variation of which load cases that it is designed for. It is seen that the characteristics are similar when the structure is optimized for a load in the 45°-direction only as when optimized for loads in X,Y, Z, 20°, 45°,70° directions. This indicates that the optimized design will withstand loads in any direction of the xy-plane when only designed for a few of them which can save

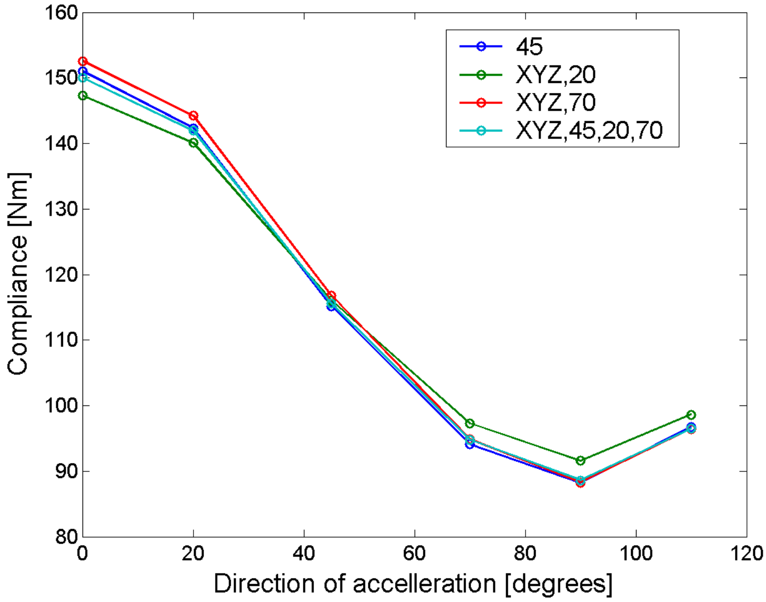


Figure 5.5: The relation between compliance and direction of acceleration, 0 degrees is in the x-direction and 90 degrees in the y-direction. Each curve describes the compliance for a structure optimized for stiffness in different directions specified in the label. The legend should read like this: The first optimization is optimized for loads in the directions $\hat{x}$, $\hat{y}$, $\hat{z}$, and 20 degrees from the x-axis in the xy-plane. Note the cosines-looking behavior of the compliance.

Table 5.1: Robustness to bolts failure

Response	**Original**	**Optimized**
Max stress, X-acc.	48 MPa, 6.3% increase	120 MPa, 9.1% increase
Max stress, Y-acc.	62 MPa, 3.3% increase	166 MPa, 38.3% increase
Max stress, Z-acc.	42 MPa, 5.0% increase	142 MPa, 52.7% increase
Sum of compliance	316 J, 3.6% increase	423 J, 5.8% increase

time.

The values for the unrealized topologies indicate a rather robust behavior due to the small changes in compliances. For example, designing with respect to loads in the 45°-direction only gives almost identical properties as when designing with respect to loads in six different directions. The symmetry constraint of the mounting base is one reason to this.

5.5.2 Robustness against failure

The change in stress distribution and compliance due to failure of two bolts at the bottom of the mounting base was studied for the optimized structure and for the original one. The relative change is considered to be a measure of the robustness to such a failure. The results are seen in Table 5.1. The stresses are nonlocal.

The results indicate that the original design is very robust in this case meanwhile the optimized structure gets stresses above the safety limit and much higher relative changes.

6 Development of Methodology

The objective of this thesis work is to create a methodology of how to use optimization in the design process at SMW. The methodology will be presented as a flowchart with recommendations for how to perform the design-process. The flowchart is meant to be used as a complement to the already existing and more general flowchart developed by SMW.

In order to produce a robust and usable methodology it is developed based on experience and conclusions from the trial cases and parameter study together with thoughts of designers and engineers at SMW. Methodologies similar to the one presented below are described by Shin et al. [22] and Krog [23] as well as in internal reports at SMW. According to those and own conclusions, topology, shape and size optimization can be applied mainly in two areas of the design process: topology optimization in the concept design and shape and size optimization in the detailed design. The main steps of the proposed methodology can be seen in the flowchart in Figure 6.1. How to perform the optimization with Optistruct is briefly described in Chapter 3. Here an overview of the methodology is presented, the more detailed and practical methodology can be found in Appendix B.

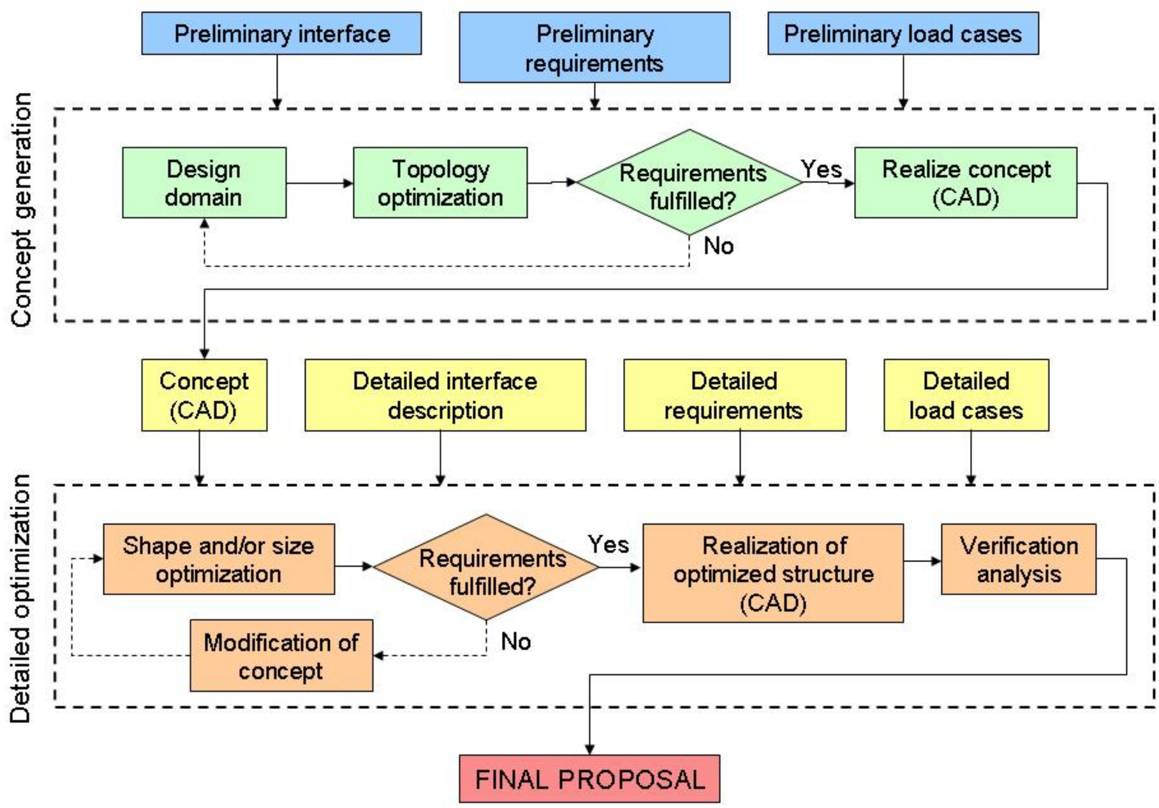


Figure 6.1: The basic steps of the methodology presented in a flowchart

6.1 Concept generation

During this initial phase preliminary specifications are used to set up topology optimizations that result in different topologies which are to be refined into one or more concepts.

A concept in this application is characterized by a structure were positions, connections, relative sizes and the like are defined but not confirmed for the structural units. A successful result of this phase is a structure that fulfills all the requirements and that is possible to realize in a manufacturable concept.

The process starts with a topology optimization based on preliminary requirements of geometrical interface, mechanical responses, load cases and manufacturability constraints. All requirements, with exception for the load cases and mechanical responses, are first defined loosely to get an understanding of the problem and an idea of what the final structure may look like and what can be expected from the optimization. By not applying all requirements in this early stage important knowledge of the problem may be found. For example which parts that are important or maybe one hole can be moved to improve the structure?

In an iterative way the requirements and design domain are then gradually defined more strictly based on resulting topologies and specifications of the final product until a design which fulfills all specifications and can be realized in a concept is reached. By this way of iteratively changing the design domain the mesh will be changed leading to knowledge of the mesh dependence of the problem, which is always important. This approach was used with good results especially in the trial case of the fixture, Section 4.1.

During this process the authors recommends performing multiple optimizations using different values of optimization parameters to come up with a set of different topologies making it possible to chose the best one to use for a concept. This can also be done automatically, see Chapter 5 for more details and Section 4.6 where this approach was used on a trial case.

6.1.1 Realization

The concept is to be realized as a CAD model based on the different topologies. This is a very tricky part which demands knowledge of constructional principles and the present method for manufacturing. The main approach is to set a certain threshold for the element densities and export the topology consisting of element densities above that value using the tool *OSSmooth* in HyperMesh. The topology can then be imported directly to a CAD program for a more or less direct translation as discussed in the trial case about the clip, see Section 4.2.

Fagerström and Jansson [1] argue for a standardized method for the realization. According to them the results should always be presented in the same way so the designer and the structural engineer could improve their skills to interpret the topologies into a concept. This should be done by always using the same threshold value for the densities when showing the parts of the structure and exporting it. A threshold value of 0.3 is suggested by Srinivas [24] when designing 2D structures but higher values have shown to be preferable for most 3D structures. Which threshold value to use in a standardized method has not been found out. Further, the topologies and their resulting concepts should always be presented with figures and mechanical properties listed below to make subsequent work more efficient.

6.2 Detailed optimization

The concept interpreted from the topology optimization is most often not an optimal structure, it only has the optimal shape but structural units are most likely badly dimensioned. In the detailed design phase the dimensions of the concept is fine tuned by size and shape optimization to produce an optimal structure on all levels. The two methods can be performed separately after one another or simultaneously if the sensitivities of the objective function related to the methods are of the same order [22]. This phase may also need to be performed iteratively if the concept turns out to be far from optimal and hence not possible to modify completely to the optimum in just one optimization. By experience it is known that Optistruct may be sensitive of getting distorted element during shape optimization, see Section 4.4.

Definitive requirements are now used to end up with a structure that in absolute numbers fulfills all specifications. The optimized structure is realized into a final CAD which is verified in a FE-analysis to ensure that all specifications are still met. The realization includes adding fillets, etc., to make it manufacturable, such changes may also change the mechanical properties and the mass.

7 Synthesis

In this chapter different aspects of the work with developing a methodology will be discussed. Also, the used software is evaluated and some recommendations for further work is given.

7.1 Lessons learned

Many different trial cases, more than initially planned, have been evaluated. This method of performing optimization on many different types of problems has resulted in insight and knowledge in how to implement topology and shape optimization in the design process. An alternative approach could have been to have just one single case and then gone through the complete design process to the finished product, instead of many different trial cases in different stages of the design process.

It was found that it is necessary to have a well defined problem before performing the optimization, the loads and boundary conditions should be at least approximately known and specified. The optimal structure will normally violate requirements that are not specified since every new design requirement leads to a reduction of the design space, unless the requirement is already satisfied. In the trial case of the clip, Section 4.2 a structure sensitive to buckling is proposed and in the trial case of the strut, Section 4.4 a very thin-walled structure is generated. Producing thin and unstable structures is a frequently repeated characteristic for optimizations without enough requirements.

When using the eigenvalue as objective or constraint it was found that it was often necessary to study several eigenmodes at once. If just one eigenmode is considered and there is an change in the order of the eigenmodes the optimization will suddenly consider a different eigenmode; this leads to an oscillating problem that may not be possible to solve. See Section 4.1.

One of the most difficult parts turned out to be interpreting and realizing the resulting topologies. The result consists of lots of elements with intermediate densities and some estimate has to be done on which parts and features that are important. It is also difficult to estimate properties such as strength and mass for the finished product from the design concept.

Topology optimization should be seen as a tool in the design process that is useful to generate an efficient design concept that can be used in the early stages of the design process. The result from the topology optimization is far from a finished product. A considerable amount of work is required to transform the concept into a finished product.

7.2 Evaluation of software

In this thesis work the HyperWorks software package was used for all FE-calculations and optimizations giving a lot of time to test end reflect over its capabilities. During the work with trial cases and study of parameters the software were found out to cover a wide range of problem solving possibilities in the field of study. However, some specific shortcomings sometimes seriously affecting the work were also found. Conclusions of the software are listed below.

- Possibility to choose among a large variety of objective functions and optimization constraints most often makes it possible to set up the optimization in the desired way

- In general very good prospects to define the problem exactly as wished in means of geometry and mesh, loads, design domains and manufacturability

- In HyperMesh it is possible to directly apply geometrical changes due to all types of optimization results for further analysis
- The post processor HyperView makes it possible to study and present results in a good and usable way
- The interface of HyperMesh appears rather old and the learning period is quite long
- The solution process when applying shape optimization to some cases acts chaotic and does not converge
- Free shape optimization may distort elements leading to interrupted optimization before convergence is reached
- There are problems when optimizing for heat conduction; topology optimization is not applicable and size optimization of shells gives questionable results. However, possibility to optimize for heat properties is a completely new feature in Optistruct.
- Stress constraints for topology optimization has been found out to be a rather poor feature which may be misleading to the user.
- Using buckling factor as a constraint or objective is not possible for topology and free-size optimization

The overall opinion is that the software fulfills its purpose as an optimization and analysis tool and is a good support for any mechanical engineer who uses it.

7.3 Recommendations for further work

As time and effort is limited there are many aspects of this work that can be improved upon and extended. Some of the more interesting paths to take are

- Implement the developed methodology in the design of a new component and evaluate and draw conclusions. Complete the whole design process to a finished product.
- Further study how to interpret/realize the resulting design concept from the topology optimization and how to estimate the properties (stress, mass, etc.) of the finished product from the design concept.
- Evaluate the resulting solutions with regard to the optimality of the solution. How to assure that a global optimum is found and what affects this.
- An additional study of shape and size optimization in analogy with the parameter study of topology optimization. Use it to improve the detailed optimization part of the methodology.

References

[1] Martin Fagerström and Magnus Jansson. Topology optimization in the design process. Master's thesis, Chalmers University of Technology, 2002.

[2] Singiresu S. Rao. *Engineering Optimization — Theory and Practice*. John Wiley & Sons, third edition, 1996.

[3] Matthias Ehrgott. *Multicriteria Optimization*. Springer Berlin Heidelberg, second edition, 2005.

[4] Martin Philip Bendsøe. *Optimization of Structural Topology, Shape and Material*. Springer-Verlag Berlin Heidelberg, 1995.

[5] Anders Klarbring and Peter W. Christensen. *An Introduction to Structural Optimization*. Solid Mechanics and its Applications. Springer Science + Business Media B.V., 2009.

[6] Carlos A. Coello Coello, Gary B. Lamont, and David A. Van Veldhuizen. *Evolutionary Algorithms for Solving Multi-Objective Problems*. Genetic and Evolutionary Computation. Springer US, second edition, 2007.

[7] Altair Engineering Inc. Optistruct 10.0 help files, 2009.

[8] Nationalencyklopedin. Topologi. `http://www.ne.se.proxy.lib.chalmers.se/lang/topologi`. [Online; accessed 2-March-2010].

[9] Niels Ottosen and Hans Petersson. *Introduction to the Finite Element Method*. Prentice Hall, 1992.

[10] G. I. N. Rozvany. Aims, scope, methods, history and unified terminology of computer-aided topology optimization in structural mechanics. *Struct. Multidisc. Optim.*, 21:90–108, 2000.

[11] G. I. N. Rozvany. A critical review of established methods of structural topology optimization. *Struct. Multidisc. Optim.*, 37:217–237, 2009.

[12] Hans A. Eschenauer and Niels Olhoff. Topology optimization of continuum structures: A review. *Appl. Mech. Rev.*, 54(4):331–390, 2001.

[13] Martin Philip Bendsøe and O. Sigmund. Material interpolation schemes in topology optimization. *Archive of Applied Mechanics*, 69:635–654, 1999.

[14] Z. Hashin and S. Shtrikman. A variational approach to the theory of the elastic behavious of multiphase materials. *J. Mech. Phys. Solidss*, 11:127–140, 1963.

[15] Martin Philip Bendsøe and Noboru Kikuchi. Generating optimal topologies in structural design using a homogenization method. *Computer Methods in Applied Mechanics and Engineering*, 71:197–224, 1988.

[16] O. Sigmund and J. Petersson. Numerical instabilities in topology optimization: A survery on procedures dealing with checkerboards, mesh-dependencies and local minima. *Structural Optimization*, 16:68–75, 1998.

[17] M. Zhou, Y. K. Shyy, and H. L. Thomas. Checkerboard and minimum member size control in topology optimization. *Struct. Multidisc. Optim.*, 21:152–158, 2001.

[18] FE Design. Fe-design gmbh - cae-dienstleistungen und software zur strukturoptimierung: .topology. `http://www.fe-design.de/en/toscatopology.html`, 2007.

[19] MSC Software. Design optimization. `http://www.mscsoftware.com/Solutions/Applications/Design-Optimization.aspx`.

[20] H. Thomas, M. Zhou, and U. Schramm. Issues of commercial optimization software development. *Struct. Multidisc. Optim.*, 23:97–110, 2002.

[21] Luo Z. Yang J. and Chen L. A new procedure for aerodynamic missile designs using topological optimization approach of continuum structures. *Aerospace Science and Technology*, 10:364–373, 2006.

[22] J.K. Shin, K.H. Lee, S.I. Song, and G.J. Park. Automotive door design with the ulsab concept using structural optimization. *Struct Multidisc Optim*, 23:320–327, 2002.

[23] L. Krog, A. Tucker, and G. Rollema. Application of topology, sizing and shape optimization methods to optimal design of aircraft components. Airbus UK Ltd, Altair Engineering Ltd, 2002.

[24] A.R. Srinivas. Robust design and realization of spacecraft payload elements using optistruct as an optimization tool. Space Application Centre ISRO, 2008.

A Script

The program is written in Python and consists of two parts, `parameter_2.py` and `run_param.py`. Settings and paths are set in `run_param.py` which in turn executes the main program `parameter_2.py`. `run_param.py` is modified for each parameter study and resides in the same directory as the `.fem`-file. Python is a general-purpose high-level programming language; the interpreter required to run the program and more information can be found at `http://www.python.org/`.

run_param.py

```
# -*- coding: cp1252 -*-
test_run = True # If true .fem-files will be created but Optistruct is not run
noninteractive = False # Run without any user interaction

optistruct = 'C:/Altair/hw10.0/hwsolvers/bin/win32/optistruct.bat'
param2 = 'C:\Documents and Settings\U000232\Desktop\My Dropbox\exjobb\script\parameter_2.
    py'
options = ' -cpu 2 -core in' # flags for Optistruct

runtype = 'sets' # sets, files or combinations
# sets --   one set of values at a time, the number of values must be the same
#           for everykey
# combinations -- every combination of parameter values
# files -- just run optistruct for each file in the list
orig_filename = 'klamma' #.fem, a list of files if runtype = 'files'

# param_sets = [[key, field, list of values, name]], ignored if runtype is files
param_sets = [['DCONSTR         1', 5, [0.3,0.5,0.7], 'volfrac'],
              ['+       MEMBSIZ', 3, [12,15,20,25], 'mindim']]
# Exempel
#   1    |    2    |    3    |    4    |    5    | <- field
#                            |  lower  |  upper  |
#DCONSTR |        1|        3|         |0.3      |
#DOPTPRM |DISCRETE|3.0       |
delete_files = ['.hgdata','.hist','.HM.comp.cmf','.HM.ent.cmf','.html','.mvw',\
                '.oss','.res','.sh','.stat','_frames.html','_hist.mvw',\
                '_menu.html']
execfile(param2)
```

parameter_2.py

This is the main program:

```
# -*- coding: cp1252 -*-
print '###########################################'
print '                Parameter'
print '###########################################'

from subprocess import call
from datetime import datetime
import os
import sys

def printres(line): # print a string to the screen and later save it to out-file
    if len(line) > 0 and line[-1] == '\n' :
        results.append(line)
    else:
        results.append(line + '\n')
    print line
try:
    noninteractive
except NameError:
    noninteractive = 0
try:
    test_run
except NameError:
    test_run = 0
```

```python
try:
    replace_with_string
except NameError:
    replace_with_string = 0
# wait for user input
def wait():
    if not noninteractive:
        raw_input('Press Enter ... ')
# find and modify a line
def change_param(lines,key, field, value):
    found = 0
    for line in lines: # Go through every line of the .fem-file
        if key in line:
            found += 1
            # swap the value
            if replace_with_string:
                new_line = line[0:((field-1)*8)] + '%8s' % value[0:8] + line[((field)*8):]
            else:
                new_line = line[0:((field-1)*8)] + '%8.3g' % value + line[((field)*8):]
            lines[lines.index(line)] = new_line
            print 'modified line to: ' + new_line[:-1]
    if found != 1:
        printres('Warning, found %d instances of %s' % (found, key))
        if test_run:
            wait()
def run_optistruct():
    printres('name: ' + new_filename)
    printres(' ')
    if not test_run:
        try:
            os.remove(new_filename + '.out')
        except WindowsError as (errno, strerror):
            pass
        code = call(optistruct + ' ' + new_filename + '.fem' + options, shell=True)
        printres('exitcode: ' + str(code))
    else: # is test run
        try:
            original = open(new_filename + '.fem').readlines()
        except IOError as (errno, strerror):
            printres('Error: ' + strerror + ' (' + new_filename + '.fem)')
    # Delete files
    for ext in delete_files:
        try:
            os.remove(new_filename + ext)
        except WindowsError as (errno, strerror):
            pass
    # Read some data from .out-file
    try:
        outdata = open(new_filename +'.out').readlines()
    except IOError as (errno, strerror):
        print 'Error: ' + strerror + ' (' + new_filename + '.out)'
        outdata = []
    error = True
    for line in reversed(outdata):
        if 'ITERATION ' in line:
            printres(line)
            break
        elif 'ITERATIONS' in line:
            printres(line)
            break
        elif 'Objective Function' in line:
            printres(line)
        elif 'ELAPSED TIME' in line:
            printres(line)
        elif 'Maximum Constraint' in line:
            printres(line)
        elif error and '*** ERROR' in line:
            error = False
            for line in outdata[outdata.index(line):]:
                printres(line[:-1])
        elif 'Sum of Weight*Compliance' in line:
            radnr = outdata.index(line)
            for line in outdata[radnr-4:radnr+1]:
                printres(line)
# run_optistruct() ends here
###        Main program starts here        ###
```

```
results = []
starttime = datetime.now()
exitcodes = []
done = False
if 'combinations' in runtype:
    current_set = [0]*len(param_sets)
elif 'sets' in runtype:
    current_set = 0
elif 'files' in runtype:
    if type(orig_filename) != type(list()):
        printres('Error: list of filenames required')
        wait()
        sys.exit()
else:
    printres('Error: runtype=' + runtype + '?')
    wait()
    sys.exit()
if 'files' in runtype:
    for filename in orig_filename:
        printres('----------------------------------------')
        new_filename = filename
        run_optistruct()
else:
    # Load .fem-file into a list
    try:
        original = open(orig_filename + '.fem').readlines()
    except IOError as (errno, strerror):
        print 'Error: ' + strerror
        wait()
        sys.exit()
    while not done: # Do for every parameter combination
        new_filename = orig_filename
        mod = original[:] # copy list
        printres('----------------------------------------')
        for param in param_sets:
            if 'sets' in runtype:
                current = current_set
            else:
                current = current_set[param_sets.index(param)]
            value = param[2][current]
            field = param[1]
            key = param[0]
            name = param[3]
            printres(name + ' ' + str(value))
            change_param(mod, key, field, value)
            if replace_with_string:
                new_filename += '_%s-%3s' % (name, value[-3:])
            else:
                new_filename += '_%s-%.3g' % (name, value)
        f = open(new_filename + '.fem', 'w')
        f.writelines(mod)
        f.close()
        run_optistruct()
        # next set/combination
        if 'combinations' in runtype:
            i = 0
            while True:
                current_set[i] += 1
                if current_set[i] == len(param_sets[i][2]):
                    current_set[i] = 0
                    i += 1
                    if i == len(param_sets):
                        done = True
                        break
                else:
                    break
        elif 'sets' in runtype:
            i = 0
            current_set += 1
            if current_set == len(param_sets[0][2]):
                done = True
        print
runtime = datetime.now() - starttime

results.insert(0,'##############################################\n')
results.insert(1,'                Results\n')
```

```
results.insert(2,'##############################################\n')
results.insert(3,'\n')
results.insert(4,'Finished at ' + str(datetime.now()) + '\n')
results.insert(5,'Run time: ' + str(runtime) + '\n\n')
# Write results to file
f = open('parameter_output_' + datetime.now().strftime('%b%d-%H%M') + '.txt', 'w')
f.writelines(results)
f.close()
# Print results to screen
for line in results:
    print line,
sys.stdout.flush()
wait()
```

B Methodology

Here the more detailed methodology that is to be used as an aid when using structural optimization in the design process is presented.

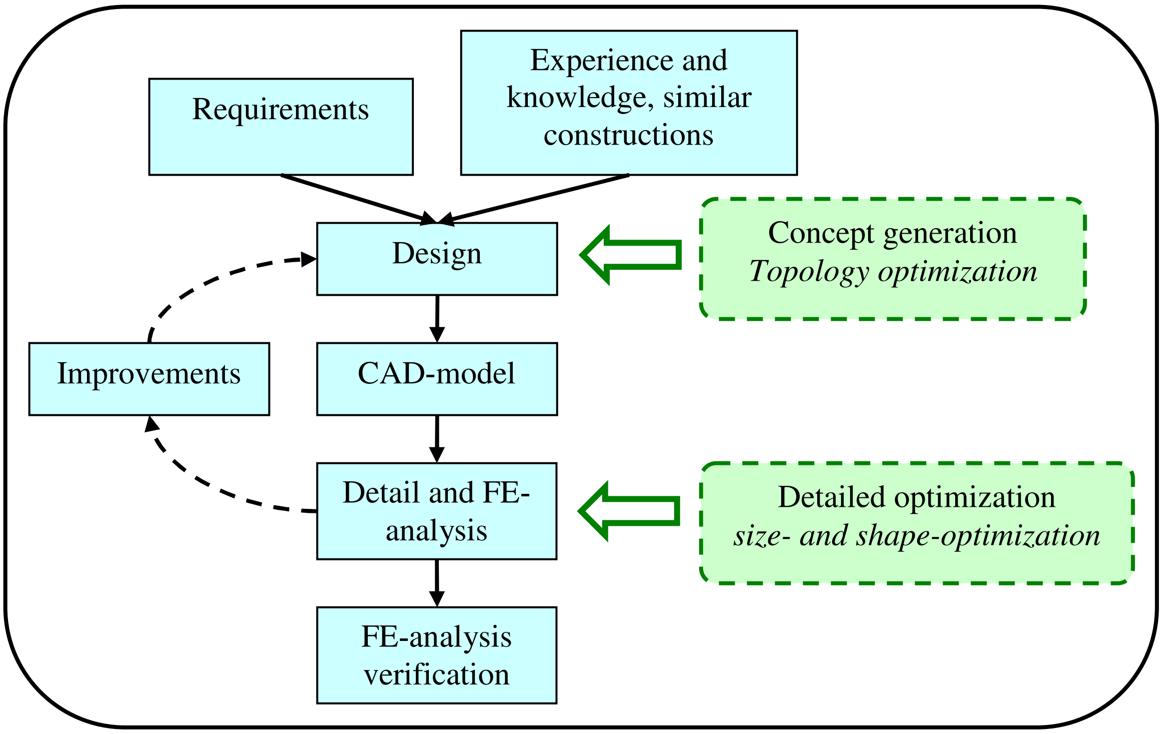
Methodology

Here is a simplified overview of the traditional design process and where the different types of structural optimizations can be applied. There are basically two different areas where structural optimization should be performed; in the early design phase where topology optimization is used to generate a good concept and in the detailed design phase where size- and shape-optimization is used to further improve the structure.

The optimization methodology can be applied to a variety of mechanical problems including solid structures, shell structures and truss structures. Problems were it should be used with caution due to small outlooks to improvement and good results are:

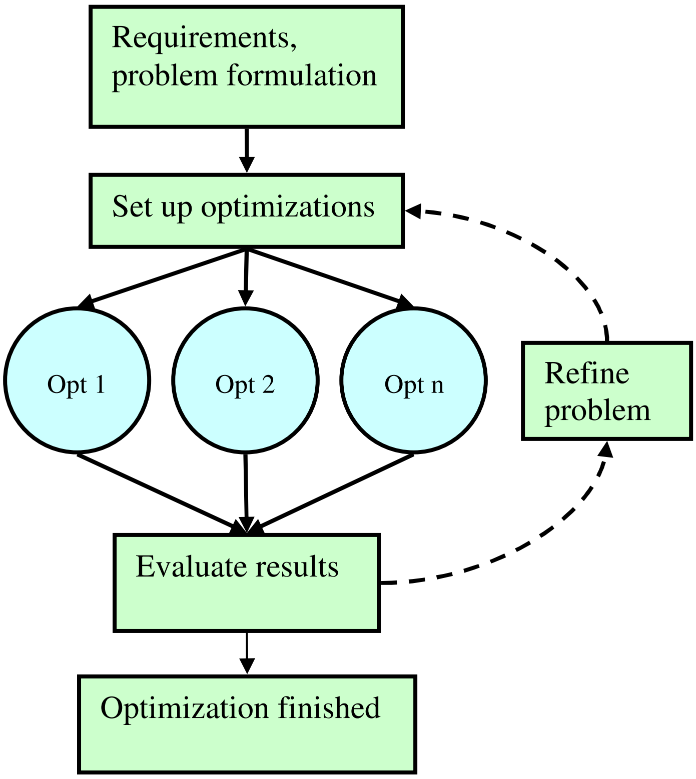
- For substructures that in means of mass and stiffness have small impact on the whole structure. The changes in properties due to chances of the substructure must be measurable in order to get a working optimization.
- For structures with obscure load cases and requirements. Optimization designs the structure for the specified problem only which means that specifications must be known to some extent.

More details on the *Concept generation* and the *Detailed optimization* are presented in subsequent sections.



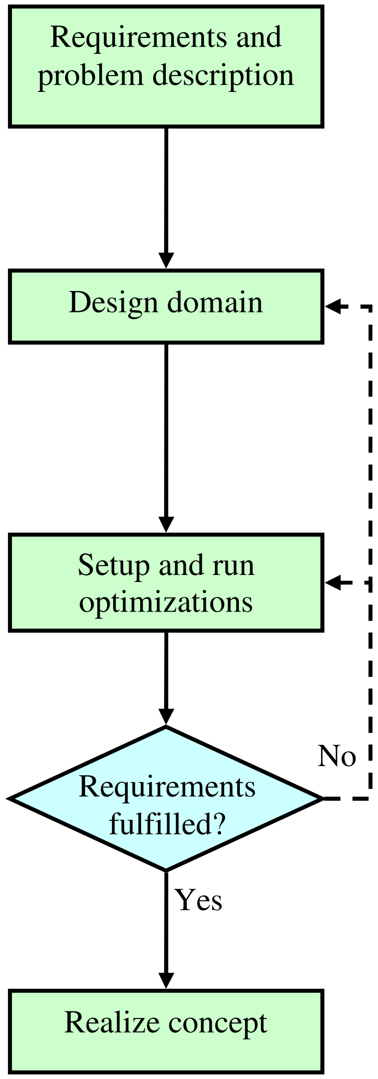
One important principle that applies to both concept generation and detailed optimization is that many different optimizations should be performed. It is not possible give guidelines that will yield a good result in just one optimization. Instead multiple optimizations must be performed and then evaluated to possibly refine the problem in an iterative way to get the best result.

In the beginning the requirements and specifications should be as relaxed as possible to not constrain the optimization too much and risk missing possible easy design improvements.



Concept generation

A flow chart for the concept generation is presented followed by a deeper description of each step. Chose of parameters and other details are discussed in the respective sections. The process of concept generation should be seen as an iterative process where the problem formulation and design domain is incrementally improved until the best possible solution is found.



Requirements on geometrical interface, loads, mechanical responses, manufacturability etc. are specified.

A design domain is generated from geometrical requirements. This step may have to be redone.

The design is discretized. Loads and constraints are specified. Design domain, optimization parameters, objective function and optimization constraints are specified. Topology and/ or free size optimization is run.

The topologies are analyzed to see if it fulfills the requirements and if it is possible to realize in a concept.

A topology is chosen and a concept is interpreted from it.

Requirements and problem description

In this introducing step all requirements for the final product are presented and translated to numerical requirements that can be introduced in the optimization work. Load cases, constraints and modeling aspects are also defined. The objective of the work is presented, e.g. minimization of mass or maximization of eigenfrequencies.

Geometrical requirements

The geometrical interface is specified in terms of

Largest allowed design volume	Define the largest allowed domain (volume or area) in which the final structure must be formed.
Predefined parts	Define position and shape of parts that must not be changed by the optimization. E.g. contact surfaces, connections to other structures etc.
Parts that must be present	Define what parts that must be present in the final structure but can be placed according to the optimization. E.g. Screw holes, holes for cables, connections to other structures etc.
Symmetry or the like	Is there any type of symmetry or pattern repetition that is desired?
Accessibility for mounting	What points of the structure must be accessible for mounting or other requirements. They have to be defined so that voids can be placed at those positions in the design domain.

Load cases and constraints

Define loads and constraints for the structure.

Static loads	Define magnitude, direction and position for forces as well as the corresponding constraints. The right relation between the loads is more important than the absolute magnitudes. The structure may be arbitrarily bad for load cases that are not introduced.
Acceleration loads	Gravitational loads can simulate acceleration or shake tests.

If the structure that is to be optimized is a part of a larger structure and forces may take other ways than through the structure being optimized, the results will most probably be rather poor. Instead insert static forces on the structure alone.

Mechanical requirements

Define requirements on masses and mechanical responses.

Mass	Define the largest/smallest allowed mass of the final structure.
Stress	Define the largest allowed stresses of the final structure, any safety limit?
Displacement	Define the largest/smallest allowed displacement at different parts of the structure if necessary.

Manufacturability requirements

What method will be used for manufacturing? What are the thinnest/thickest allowed material thicknesses?

Extrusion	What direction?
Casting	Single or split? Directions?
Material thickness	Minimum and maximum member size?

Objective, criterion of success

Define the objective of the conceptual design and what type of result that shall be considered as a success.

Design domain

The design domain is created, preferably by a designer and an analysis engineer in cooperation. The idea is to start with a simple domain leaving much for the optimization to do and successively refine it after optimizations. In this way the chances to reach a global optimum are increased and the concept will be more optimal. The design domain is preferably made as a CAD-model.

Set up the design domain using the following tips

First design domain	Construct a design domain using the geometrical requirements. Design the predefined parts, fill up the largest allowed design volume and introduce voids for mounting etc at predefined positions if any.
Refined domain	After initial optimizations parts that must be present in the final structure but don't have predefined positions are introduced. Use conclusions from the optimizations to choose the best positions. Construct additional voids for mounting. If some part of the design domain obviously is unnecessary it could be removed to make computations faster.
Partition the design domain	The design domain may be partitioned and parts can be left out of the calculations. E.g. if a structure is to be placed on a plate which is to be fastened in something; the plate can be excluded and the structure supposed to be fixed in the former contact surface to the plate. See Section 4.1 about the fixture. The plate is then introduced later when the other parts of the structure are ready. This method is used when it is hard to fulfill manufacturability constraints.

Set up and run optimizations

Discretize the model and define load cases

Mesh the model with an element size of a third of the desired minimum member size. This will ensure a mesh independent solution later on. Complete the model and connect it to possible adjacent structures.

Define the loads, constraints and load cases the structure shall be subjected to.

When the design domain has been refined and shall be optimized again the mesh size can be slightly shifted to study if the analysis is mesh independent.

When the model is finished a test run is performed to study if

- there is a possibility to reach a successful result with the optimization; that the requirements are not unreasonable
- the model is built up properly; that elements are connected correctly etc., a modal analysis is good to spot errors

Topology or free size optimization?

Topology optimization is used for solid structures and for shell structures where a truss like result is wanted. Free-size optimization is used for shell structures where the thickness is to be optimized and gives a more smooth distribution of material than topology optimization. In general free-size optimization gives more optimal results than topology optimization but the realization of the structure may be more difficult.

Design domain and design variables

The definition of the design domain, design variables and manufacturing constraints are described in the following table.

Properties	Create *properties* and assign elements of the model to *properties* that define design domain and non-design domain. The designable area could be partitioned into a number of separate design domains possibly giving more control in the means of material distribution, manufacturability etc.
Design variables	Create design variables by choosing property and then defining manufacturability constraints.
Extrusion (topology)	The optimization will act by a constant cross section. Direction is chosen.
Casting (topology)	By choosing draw direction the final structure theoretically will be possible to construct by casting. No internal cavities will be formed. If the product is to be casted from many directions the design domain is divided into a number of sub domains, each one with its own draw direction.
Minimum member size	The smallest allowed member size is specified. The element size should be roughly a third of this size.

Other parameters	There are other parameters available when creating the design variables; maximum member size, max stress and fatigue. These should be used with caution.
Symmetry and pattern repetition	When defining the design variables, demands on different kinds of symmetry and pattern repetitions can be activated. Use this in accordance with the geometrical requirements.
Limitations	Manufacturability constraints and symmetry/pattern repetition constraints cannot be use simultaneously.

Parameters

A number of parameters that influence the optimization result can be activated and varied to achieve better results. The parameters and their effects are described below.

MINDIM	Controls the minimum member size, very effective at suppressing checkerboard patterns and enforce a discrete solution. Should be at least 3 times the element size.
DISCRETE	Controls the penalization of intermediate densities, a value of 2 is recommended for 2D-elements and 3 for 3D-elements
CHECKER	Suppresses checkerboard patterns, results in a lot of intermediate densities, consider the use of MINDIM instead
MATINIT	The initial material distribution
OBJTOL	The convergence criteria. Default is 0.005, lower leads to more iterations. Is most often OK with the default value.
DESMAX	Maximum number of iterations. Default is 30 (80 when MINDIM is used). Default is most often sufficient; if max number of iterations is reached and the solution appears to converge, DESMAX should be increased.
SCREEN	A control card that should be used to output information about the iterations to the command window

Responses

A number of different responses that can be used both as objective function and constraint are available and can be seen below

Compliance	Typical as objective function, inverse of the stiffness of the structure
Weighted compliance	Weighted sum of compliances from different load cases
Volume	
Mass	
Eigenfrequency	The frequency of one particular eigenmode, note that if the eigenmode number changes order with another eigenmode this will result in an oscillating optimization problem and a bad result. If so weighted eigenfequencies

	must be used.
Weighted eigenfrequencies	Weighted sum of the reciprocal eigenfrequencies, at most 6 modes. Trial and error might be needed to set weights and find a value which produces eigenvalues in the desired range
Static displacement	Displacement in one node, in one direction or the total distance. If the acting force is applied in the measured node, minimization of displacement is very similar to minimization of compliance.
User defined function	

Responses that are not available for topology optimization includes buckling factor and temperature.

Objective

The objective function is defined by choosing a response to either maximize or minimize. Exactly one objective function must be created.

Constraints

Constrains are defined by setting upper and/or lower bounds on a response. Multiple constraints can be created.

Instead of using the available stress constraint it is recommended to perform multiple optimizations with the objective of minimizing the compliance together with different upper bounds on the volume fraction. From this the stresses for each volume fraction is studied manually and a volume fraction that gives reasonable stresses is chosen.

Automated study

To easily test different combinations of optimization parameters and constraint it is very suitable to perform an automated study where several optimizations with different parameters are performed. By automating this study time can be saved by doing multiple optimizations in off-work hours and by eliminating the required work to change parameters. The solutions can then be analyzed and the best one is chosen for further work.

MINDIM	Vary in a range around the specified minimum member size for the final structure.
DISCRETE	Vary the value to study if a result which is more clear and easy to realize is possible to reach.
MATINIT	Vary to study if the optimization converges to the same structure. This gives information of whether a global optimum is reached or not.
volume fraction, mass fraction, volume or mass	If one of the responses volume fraction, mass fraction, volume or mass is used as a constraint: vary the value in a wide range. Plot the resulting values of the objective function to the constraint value. This gives information of the relation between decreased mass contra loss of goodness of the structure and vice versa.
Load cases	If weighted compliance is used as objective or optimization constraint the individual weights are varied to study the robustness of the solution. Direction and position of loads and constraints can be varied. The topology can be subjected to load cases which it is not optimized for and the robustness studied.

Criterion of success

The properties of the resulting topologies are hard to measure in absolute numbers. The mass and other properties of the final structure will most probably differ from those of the structure at this stage. Therefore a limit of safety should be used on eigenfrequencies, displacements, etc. Topology optimization will most likely give a structure with high stresses; however, such matters are designed for in the detailed design phase. A structure that is considered as a candidate to continue with should fulfill the following requirements:

- Any demands on eigenfrequencies or displacement etc. are fulfilled with a limit of safety
- The topology should be able to realize in a concept without any drastic changes in topology
- The topology has been proven to be a global optimum
- The topology has shown to be robust by using variation of load cases
- The overall behavior and shape of the topology seems reasonable

Realize concept

One of the most difficult tasks when performing the topology optimization is to interpret the results. By setting up the optimization problem in a good way this step can be made easier but it is still far from unambiguous.

The designer and the analysis engineer should at this stage work together so that aspects such as manufacturing are taken into account and sound simplifications are made.

The results are post processed in HyperView. One good way to visualize the result is by only showing the elements that have a density above a certain threshold. The threshold is varied to get a good understanding of the result.

The post processing capabilities of HyperMesh, OSSmooth, can be used to generate an IGES-model from the results than can be opened in a CAD-software to be used as a guideline. OSSmooth also requires a threshold density when producing the model.

Detailed optimization

In this stage a CAD-model of the structure is available that is similar and have the same topology as the end result. The objective in this stage is to fine-tune, or refine, the structure to make it as good as possible.

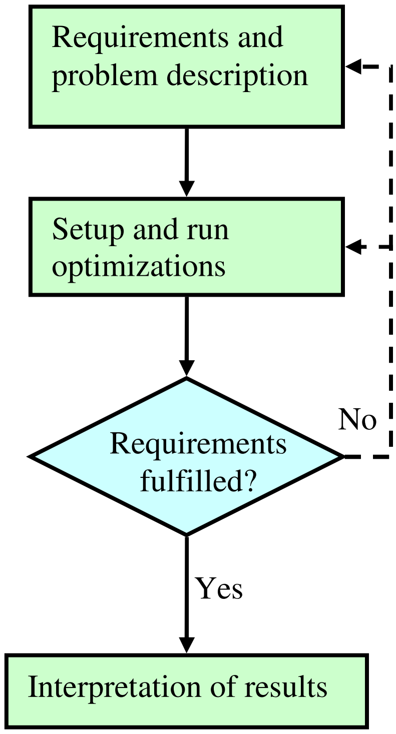
There are basically two different types of detailed optimization: size optimization and shape optimization. Shape optimization can also be performed by free-shape optimization. This should be an iterative process where several subsequent optimizations are performed to get the best possible result.

Requirements on geometrical interface, mechanical responses, manufacturability, etc., are specified.

The optimization problem is set up; can be a combination of size, shape and free-shape optimization.

Evaluate if the requirements are fulfilled and ensure that a global optimum has been found

The results are interpreted and implemented in the real structure



Requirements and problem description

The requirements and objective on the design is specified by creating an objective function and setting constraints by using a combination of available responses.

Multiple optimizations with varying parameters and options should be performed to get a feeling of the problem and to find the best possible solution. Constraints and requirements can in the beginning be relaxed to not limit the design space too much. The specifications are then refined to get the desired end result.

Available responses

A table of the available responses that can be used as objective function or as constraints in size or shape optimization can be seen below. It can be noted that in this detailed optimization buckling factor and stress can also be used as responses.

Compliance	Typical as objective function, inverse of the stiffness of the structure
Weighted compliance	Weighted sum of compliances from different load cases
Volume	
Mass	
Eigenfrequency	The frequency of one particular eigenmode, note that if the eigenmode number change place with another eigenmode, i.e. an order change, this will result in an oscillating optimization problem and a bad result. If so weighted eigenfequencies must be used.
Weighted eigenfrequencies	Weighted sum of the reciprocal eigenfrequencies, at most 6 modes. Trial and error might be needed to set weights and find a value which makes the eigenvalues in the desired range.
Von Mises stress	
Static displacement	Displacement in one node, in one direction or the total distance
Temperature	In the evaluated trial case optimization with temperature as a response did not yield a good result
Buckling factor	The buckling factor of a given mode
User defined function	

Setup and run optimization

In the detailed optimization there are three types of structural optimization available when using Optistruct; size, shape and free-shape optimization.

Size optimization

In size optimization the design variables are first created. Then almost any type of parameter that can be specified by a numerical value such as shell thickness, beam width/height, etc., can be set to depend on the created design variables. It is also possible to let some property depend on a linear combination of design variables.

Requirements on design variables

Boundaries	If necessary, specify lower and upper bounds on the design variables
Discrete	The design variable can be specified to take values from a given set of discrete values. For example if it is desired to find the best steel sheet thickness only a couple of thicknesses are available
Start value	The start value of the variables can be chosen and a number of optimizations with varied start values are highly recommended since different results may occur

Shape optimization

Shape optimization is performed by first creating different *shapes* using the tool HyperMorph in HyperMesh. Then design variables are created from these shape changes together with limits on how big the allowed shape changes are (as a multiplication factor of the shape change).

Shape changes can be created in many different ways. It is possible to create shape changes that controls length, radius of fillets, diameters of holes, etc., of parts of the FE-model. The range in which the change may vary can be specified to for example restrict the change in an area due to space limitations.

Free-shape optimization

Free-shape optimization is performed by specifying nodes on the boundary that are allowed to be moved to fine-tune the structure. Nodes can also be specified to be fixed if the node must not move due to geometric requirements.

It has been noted that Optistruct may be sensitive of getting distorted elements when performing free-shape optimization, see Section 4.4: *Trial case: strut for airborne radar*. If this happens the program will simply crash. One way to prevent this is by making the finite element mesh more similar to the final shape. This can be done by looking at the shape change of the last iteration step before the program crashes and making a new mesh that is more similar to the end result.

Design constraints

Manufacturing requirements	If necessary set an extrusion or draw direction constraint on the design. Both cannot be used simultaneously.
Geometric requirements	Set nodes that must not be moved to be fixed. Define if the boundary is allowed to grow and/or shrink and by how much.
Nodal movement	Decide how the boundary nodes should be allowed to move; free, along a vector or on a plane

Note that when defining the design variables the choice of *smoothening method* should be at default, "*optimized for speed*". Otherwise an error in the program may occur, the producers of Optistruct are aware of the problem.

Interpretation of results

Interpreting the result from the shape or size optimization is a simple task compared to the interpretation in the concept generation case. The boundaries are well defined and numerical values on aspects such as mass, stresses, displacements, etc., are readily available. However, one has to be aware of that angles and dimensions may have changed so that geometrical requirements are not exactly fulfilled in the detail optimized structure.