Post-Processing of Topology Optimized Designs

Case Study of a Drone Arm

Kjetil Skorpen Mathiesen
Jens Ness
Oscar Nicolai Løbak Sæther
Erlend Vatsvåg

Faculty for engineering science (IV)
Norwegian University of Science and Technology - NTNU
Trondheim, Norway

FAKULTET FOR INGENIØRVITENSKAP
Institutt for maskinteknikk og produksjon
7491 Trondheim

Besøksadresse:
R.Birkelands vei, 2B, Trondheim

RAPPORT BACHELOROPPGAVEN

Tittel på engelsk:

Post-Processing of Topology Optimized Designs
Case Study of a Drone Beam

Tittel på norsk:

Etterbehandling av Topologioptimalisert Design
Case Studie av en Drone Arm

Prosjektnr
MTP-K-2019-02

Forfattere

Kjetil Skorpen Mathiesen
Jens Ness
Oscar Nicolai Løbak Sæther
Erlend Vatsvåg

Oppdragsgiver(e) eksternt

Sevendof

Dato levert	Antall vedlegg:	Totalt antall sider:
20.05.19	4	144

Veileder(e) intern

Evangelos Tyflopoulos

Rapporten er ÅPEN

Kort sammendrag
Topologioptimalisering er en matematisk prosess som bidrar til å optimalisere design ved å fjerne unødvendig masse. Det finnes flere matematiske algoritmer for topologioptimalisering som blir brukt i kombinasjon med dataassistert konstruksjon. Tre av disse metodene ble undersøkt, testet og sammenliknet.

Topologioptimalisete deler har ofte en tendens til å ende opp med problematisk geometri, noe som gjør det vanskelig å produsere dem uten etterbehandling. På bakgrunn av dette har gruppen undersøkt og utarbeidet ulike metoder, for hvordan å gå fra et topologioptimalisert resultat, til et ferdig design.

For å bruke den tillærte kunnskapen og de forskjellige metodene, ble et samarbeid med Sevendof igangsatt. En dronearm ble redesignet, topologioptimalisert, etterbehandlet og validert ved bruk av dataassistert konstruksjon.

Stikkord fra prosjektet
Topology Optimization, SIMP, BESO, LSM, Lattice Optimization, Post-Processing, Design Space, CAD, FEA, Drone Arm, Case Study, Sevendof, Optimization, Software Testing, Solidworks, Abaqus, Ansys, ParetoWorks, Altair Inspire.

Preface

This thesis is written as the conclusion of a three year education in mechanical engineering at Norwegian university of Science and Technology, NTNU. The focus of this thesis is post-processing of the Topology Optimization process and the application of the method. Tools such as computer aided design and finite element analysis were also essential to the completion of this thesis.

During the spring term of 2019 this thesis was produced, with the subject first presented to us by Evangelos Tyflopoulos in November 2018. We also cooperated with Sevendof, a Trondheim based start-up developing industry service drones.

The group responsible for writing this thesis wanted to work on something new and exiting to us, where the different background of the members could be utilized. Working on this thesis has been gratifying and a learning experience where we gained a lot of experience. Both with regards to the thesis matter, writing, group dynamics and other small experiences, we will have great benefit from this experience in years to come.

We want to take this opportunity to thank Evangelos Tyflopoulos for presenting this thesis to us, guiding us through this process and generally being available to answer all our questions.

We also want thank Sevendof and Przemysław Dominik Gacia for providing us with a case study and general counseling during the many hours spent writing this thesis.

Lastly we also want to thank our friends and families that have helped support us through not only this thesis, but also through this three year education.

Kjetil Skorpen Mathiesen

Jens Ness

Oscar Nicolai Løbak Sæther

Erlend Vatsvåg

Trondheim 20.05.2019

Abstract

Topology optimization is a mathematical process that alters the shape and material distribution of a design. It is implemented in combination with computer-aided design and finite element analysis. Topology optimization contributes to solving basic engineering problems: reducing weight and maximizing stiffness. The topology optimized results will often be rough, and therefore have to be post-processed to obtain good result. This process consists of evaluating the results, re-designing the part and then validating the new optimized design.

In this assignment a literature research about some of the currently available topology optimization algorithms has been conducted. In addition, topology optimization using lattice structures has been researched. Different programs utilizing these algorithms were then tested. These programs were tested using a simple 3D model and then compared against each-other using the same parameters. The programs were ranked based on the results, solver time, general overview and ease of use and then compared.

Several methods for post-processing of the topology optimized results were researched and devised. These are explained in this thesis. Research on how the design space of a model affects the topology optimization results were also conducted. The research conducted was then utilized on a case study. The case study was provided by the company Sevendof, who develops industrial service drones. To increase flight time, these drones should be as light as possible. The purpose of the case study is to utilize topology optimization to save as much weight a possible on the drone arm while meeting the design and structural requirements. Sevendof supplied a set of load cases. Load case 1 is based on the maximum thrust of the motors and is used to optimize the drone arm. Load case 2 describes a critical failure. Studies will be conducted to provide data on this scenario. A rough design outline of the drone arm was provided as a starting point. The arm was re-designed, topology optimized, post-processed and then validated to meet the conditions set by Sevendof.

Sammendrag

Topologioptimalisering er en matematisk prosess som bidrar til å optimalisere design ved å fjerne unødvendig masse. Det finnes flere matematiske algoritmer for topologioptimalisering som blir brukt i kombinasjon med dataassistert konstruksjon. Tre av disse metodene har blitt undersøkt, testet og sammenliknet ved hjelp av en simpel 3D-figur.

Topologioptimalisete deler har en tendens til å ende opp med problematisk geometri, noe som gjør det vanskelig å produsere dem uten etterbehandling. På bakgrunn av dette, har denne oppgaven tatt for seg ulike metoder for etterbehandling. Gruppen har også undersøkt og utarbeidet ulike metoder, for hvordan å kunne gå fra et topologioptimalisert resultat, til et ferdig design. Disse metodene er forklart i detalj i denne bacheloroppgaven. I tillegg ble det undersøkt hvordan størrelsen på det tilgjengelige designområdet kan påvirke topologioptimaliseringsresultater.

For å bruke den tillærte kunnskapen og de forskjellige metodene, ble et samarbeid med Sevendof igangsatt. Sevendof er et firma som produserer industrielle droner, som trenger å være så lette som mulig. Oppgaven gruppen ble tildelt var å bruke topologioptimalisering for å gjøre en dronearm lettere. Sevendof ga gruppen noen kriterier som dronearmen måtte oppfylle. Dronearmen ble redesignet, topologioptimalisert, etterbehandlet og validert ved bruk av dataassistert konstruksjon.

Sevendof presenterte to ulike scenarioer for dronearmen. Load case 1 og load case 2. Det førstnevnte scenarioet representerer kreftene utøvd av motoren, ved full effekt, på armen. Dronearmen ble designet ut ifra disse påkjenningene. Load case 2 representerer en kritisk situasjon som ble undersøkt.

Contents

List of abbreviations

NTNU	Norwegian University of Science and Technology
TO	Topology Optimization
LO	Lattice Optimization
MPa	Megapascal
FEM	Finite Element Method
FEA	Finite Element Analysis
CAD	Computer Aided Design
SIMP	Solid Isotropic Material with Penalization
ESO	Evolutionary Structural Optimization
BESO	Bidirectional Evolutionary Structural
UI	User Interface
LSM	Level Set Method
PP	Post-Processing
CNC	Computer Numerical Control
AM	Additive Manufacturing
RP	Rapid Prototyping
WBS	Work Breakdown Structure
RAMP	Rational Approximation of Material Properties
OMP	Optimal Microstructure with Penalization
NOM	Non-Optimal Microstructure
DDP	Dual Discrete Programming
AESO	Additive Evolutionary Structural Optimization
FOS	Factor Of Safety
xFEM	Extended Finite Element Analysis
DSC	Deformable Simplicial Complex
Al	Aluminum
Ti	Titanium
V	Vanadium

RPM	Revolutions Per Minute
RPS	Revolutions Per Second
DIN	Deutsches Institut für Normung
ANSI	American National Standards Institute
RMIT	Royal Melbourne Institute of Technology

List of Figures

List of Tables

1 Introduction

This section will explain the subject and the groups motivation for choosing it. It will also present the problem and the restrictions put upon it.

1.1 Subject and Motivation

Topology Optimization (TO) is a mathematical process that optimizes the material layout within a fixed set of constraints. It is used as a tool in the basic engineering problem of reducing weight while maintaining strength and stiffness. While TO gives the engineer new solutions, it is not perfect. The results from a TO study often has to be interpreted, re-designed and refined into a finalized design that can be brought to production.

TO is an exciting field that intrigued our group, which provides a good basis for new learning. It also allows the different strengths of the group to be leveraged. The group gets to utilize already acquired skills in the topic of Computer Aided Design (CAD) and Finite Element Analysis (FEA), among others, while further developing expertise with new tools, software and theory.

1.2 Problem

Since TO is an evolving method, this thesis will research, test and analyze different TO software, scripts and plugins aimed at commercial programs, and open source software. It will also go into depth on the method of post-processing the TO results. This consists of the interpretation of TO results, the re-design process, and validation of the new structure.

In addition a partnership with the company Sevendof has allowed us access to a case study. Sevendof has supplied the group with a design for an aerial drone arm and a set of load cases. Load case 1 is based on the maximum thrust of the motors and is used to optimize the drone arm. Load case 2 describes a critical failure. Studies will be conducted to provide data on this scenario.

The problem statement for this thesis is presented in **Appendix A**. The case study provided by Sevendof is presented in **Appendix D**.

1.3 Restrictions

Due to the depth of the given subject, the scope of this thesis has to be limited to ensure the quality.

1.3.1 Topology Optimization Methods

In this thesis the amount of theory on different TO methods will be limited to the methods that are used. The depth will also be restricted to give a basic understanding of the methods, due to the complexity of the algorithms. This is done in order to maintain the schedule that has been set for the thesis.

1.3.2 Topology Optimization Post-Processing

Optimizing the design from a TO study can be difficult and requires experience, knowledge and the right tools for the job. The group will limit the scope of the topology optimization post-processing to simple guidelines and examples with the tools and technology available at this time. In the future, more sophisticated methods might be developed that makes the methods laid forth in this thesis obsolete.

1.3.3 Result Validation

The validation of the TO study results are restricted to computer simulations. This is done because of the limited value of producing a prototype for testing and the time it would take to do such a validation.

2 Group Dynamic

The purpose of this chapter is to give an insight in to how the group structured their work. It will describe the pre-project, the work process of the group, how the project was broken down and the time management.

2.1 Pre-Project

The project work started with a pre-project, orchestrated by NTNU, with the purpose of creating a framework for this thesis. The pre-project contains a rough time schedule, risk management matrix, goals and a work breakdown structure (WBS). From this framework a WBS, **Figure 2.1**, and Gantt-form, **Figure 2.3**, were developed. These will be explained in greater detail later.

For a project to be successful it requires good communications. Therefore, the group early wrote and signed a "Joint Venture Agreement" to establish a healthy work relationship. The agreement laid some ground rules for meetings, attendance and absence. Another agreement was also signed, a "Standard Agreement", to ensure the rights of the involved parties. It was signed by the students, NTNU and Sevendof.

During the pre-project phase, construction of a prototype was considered. A prototype could be useful by means of practical testing. However, it was determined at a later stage that the time and effort invested in the making of a prototype would exceed the benefits.

To ensure that the thesis were within boundaries of the assignment, meetings with the supervisor were had whenever needed. The drafted risk matrix contains some of the challenges the group could come up against. **Table 2.1** shows the matrix in question. A risk management plan, with actions to take to minimize the risk was also drafted up, see **Table 2.2**.

Probabillity		Consequence	
1	Low	1	Little
2	Moderate	2	Moderate
3	High	3	High
4	Very high	4	Very High

1-2	Acceptable
3-6	Tolerable
7-16	Unacceptable

Factors	Probability	Consequence	Risk level (P*C)
Not enough time	2	4	8
Absence due to illness	2	1	2
Loss of data/research	1	4	4
Conflicts	2	1	2

Table 2.1: Risk evaluation

Factors	Risk level	Consequence	Action
Not enough time	8	Thesis not completed on time	Not an option, thesis must be deliverd on time
Absence due to illness	2	Group member not completing their part	Discuss in group, divide work between remaning members
Loss of data/research	4	Significant setback or failure to deliver on time	Make backups, at end of each working day
Conflicts	2	Reduced efficiency	Discuss in group, if escalated talk to supervisor.

Table 2.2: Risk management plan

2.2 Work Process and Progress

To break down the thesis into manageable tasks, a WBS was used, see **Figure 2.1**. This structure was used to simplify the project and give clear goals. It also provided a good representation of what processes are dependent on one another.

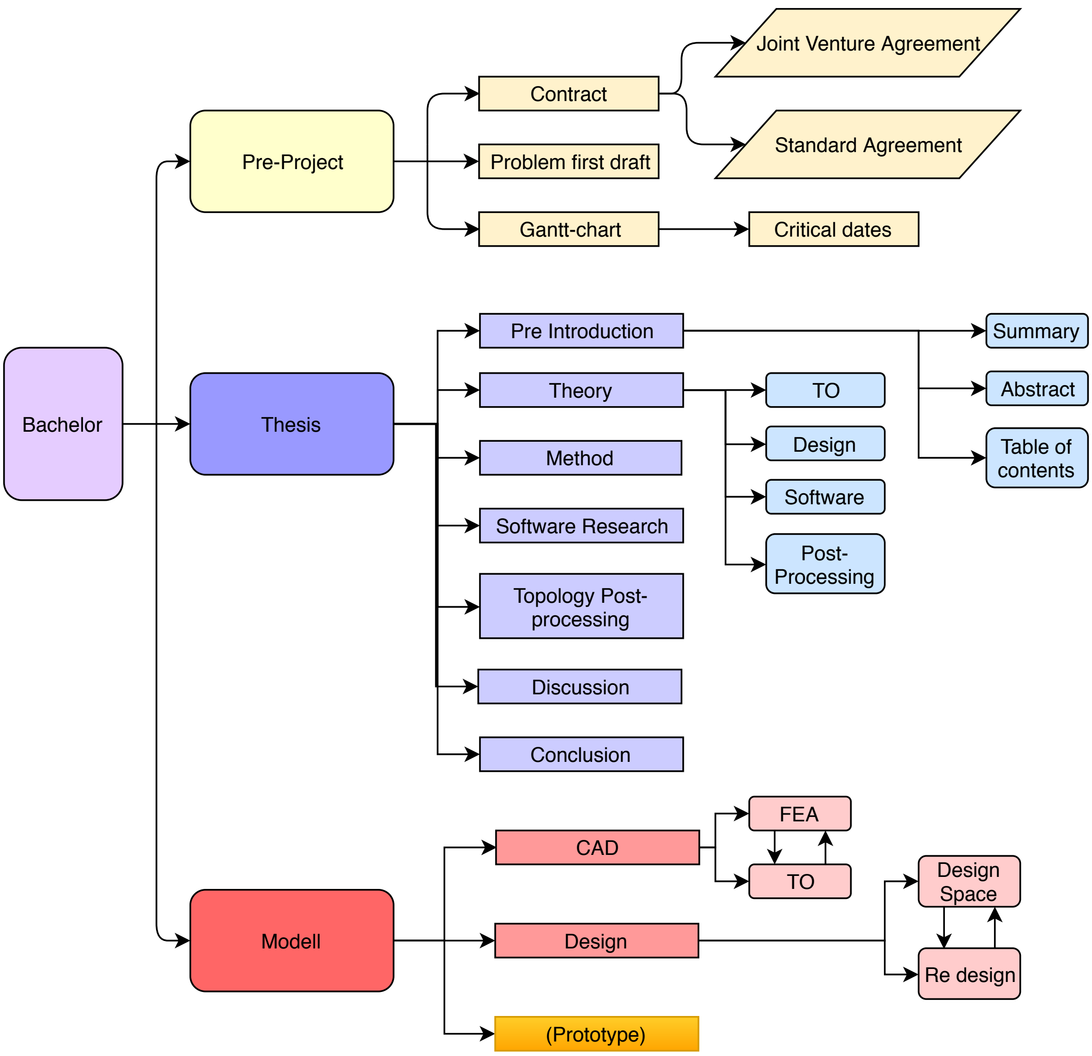
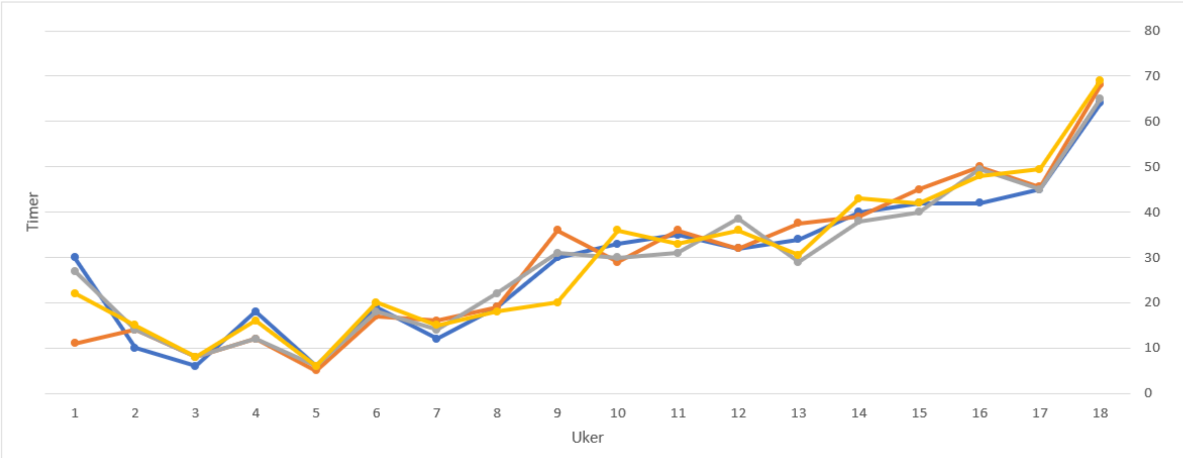


Figure 2.1: Work Breakdown Structure of the thesis

Early on, the group started recording their working hours to get an overview of the project, and how it was going compared to the planned schedule. As seen in the graph in **Figure 2.2**, the number of hours is rather low in the start of the process, which was caused by a secondary project running parallel to the thesis. In week five the project finished and the thesis hours increased. To keep track of critical dates, planned duration and overall progress, a Gantt-chart was used **Figure 2.3**. The Gantt-chart gives a clear view of the overall progress, but also the progress and duration of subsections.



	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	Sum
Erlend	30	10	6	18	6	19	12	19	30	33	35	32	34	40	42	42	45	64	517
Jens	11	14	8	12	5	17	16	19	36	29	36	32	37,5	39	45	50	45,5	68	520
Oscar	27	14	8	12	6	18	14	22	31	30	31	38,5	29	38	40	49,5	45	65	518
Kjetil	22	15	8	16	6	20	15	18	20	36	33	36	30,5	43	42	48	49,5	69	527

Figure 2.2: Graph of working hours

ACTIVITY	PLANED START	PLANED DURATION	ACTUAL START	ACTUAL DURATION	% COMPLETE
Pre-Project	1	5	1	5	100 %
Litterature Research	2	7	2	7	100 %
Software Research	3	7	3	7	100 %
Testing of software	9	5	5	10	100 %
Post-processing research	8	5	8	7	100 %
Drone Arm Design	12	5	11	7	100 %
Completion of thesis	16	3	16	3	100 %

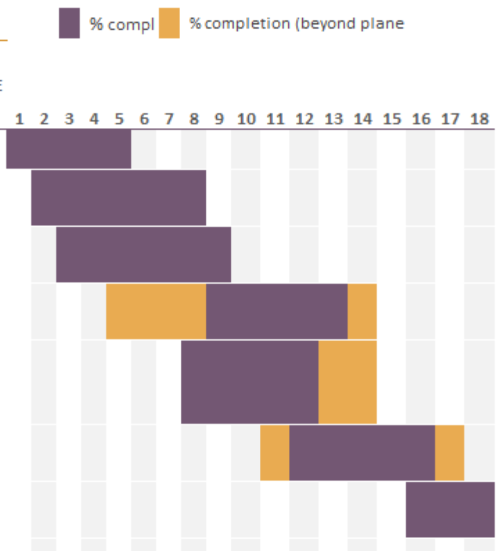


Figure 2.3: Gantt diagram

3 Theory

This chapter contains the theory that the thesis is built upon. It also encompass background information and concepts the group considers relevant to the understanding of this thesis, such as mechanics, finite element analysis and more. It will present relevant material and explain subjects used in conjunction with TO.

3.1 Mechanics

Mechanics is the area of science devoted to understanding how physical objects behave and react when exposed to forces or displacement. A simple example with a beam is used to showcase some basic principles.

When a beam, fixed to a wall, is exposed to a force at the opposite end, it will experience displacement. The premise is shown in **Figure 3.1**. Under the figure, the formula of displacement for this exact beam is shown [1].

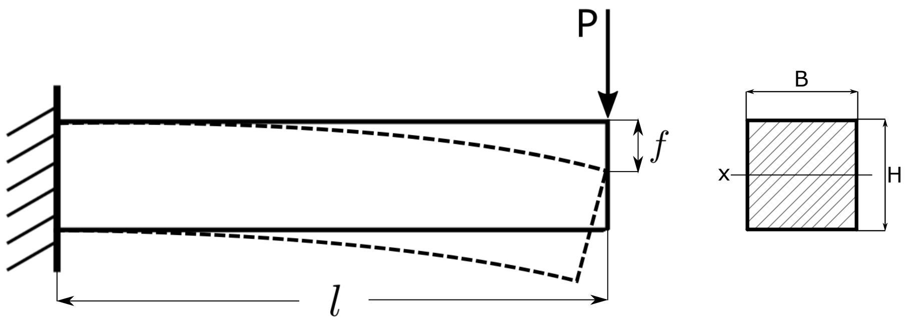


Figure 3.1: Simple beam with a force at the end

$$f = \frac{Pl^3}{3EI} \tag{1}$$

$f =$ Displacement, $P =$ Force, $l =$ Length, $E =$ Elastic modulus, $I =$ Second moment of area

To minimize the displacement for this beam, there are four variables that can be altered. First of all, one can reduce the force (P) or the length of the beam (l). By doing so, the output of the formula will be less than before. Other than that, one can increase the value of the elastic modulus (E) or the second moment of area (I). The elastic modulus can be adjusted by swapping out the material used, i.e. from aluminum to steel. By increasing the size of the cross section, the second moment of area will be increased. All of these variables can be altered alone or together to get the desired amount of displacement. In some cases, the variables are restricted, and can therefore not be changed. The reduction of displacement is then decided by the variables that can be changed.

$$I = \frac{1}{12}BH^3 \tag{2}$$

$I =$ Second moment of area, $B =$ Base, $H =$ Height

The formula for the second moment of area shown in **Equation 2**, applies to rectangular cross sections. If increasing the height (H) or the base (B), the second moment of area (I) will increase. Other cross sections are calculated differently. In general, if the area increases, the second moment of area will also increase.

3.2 Materials

To achieve a strong and robust part the choice of material is an important factor, and can sometimes be challenging. There are often several factors that restrict the materials that are available. This can be factors such as:

- Tensile Strength
- Yield Strength
- Density and weight
- Other material properties
- Other environmental factors

Different materials and metals can only support a given stress before being permanently deformed or break under the load. These two different points are known as the yield point and the breaking point. The value is given as the yield strength and the tensile strength, normally given in megapascal (MPa). If the stress does not exceed the yield point, it is inside what is known as the elastic deformation area. Here the stresses will not cause permanent deformation to the material and the part will regain its original shape. This behavior can be shown in a stress-strain curve. These curves can vary greatly from material to material, but can generally be divided into brittle and ductile materials. Brittle materials break quickly with little change in deformation, but in general ductile materials will deform before reaching the yield and then breaking point as shown in **Figure 3.2** [2].

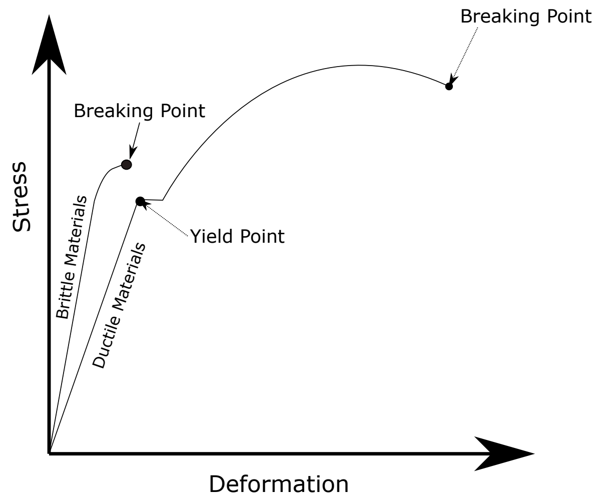


Figure 3.2: General strain-stress curve for metals

Different materials have different densities. When designing, weight is often a limiting factor and density therefore becomes an important factor to consider. This means that not all materials are always a viable option. Aluminum has roughly $\frac{1}{3}$ the density of steel and titanium has roughly $\frac{1}{2}$ the density of steel [2].

Density is defined as:

$$\rho = \frac{m}{V} \tag{3}$$

$$\rho = \text{Density}, m = \text{Mass}, V = \text{Volume}$$

To alter a metals properties, it is possible to add small amounts of other elements to it. This is called alloying and is a common practice. By adding in small amounts of other elements you can radically change material properties [2].

3.2.1 Aluminum

Aluminum (Al) is classified as light metal and is used in a variety of applications. With a low density and adequate strength, it is perfect for saving weight. Pure Al has a yield strength of 103-132 MPa, but certain alloys can reach quite high, such as the alloy 7070-T6 with a yield point of 570 MPa.

Al 99.5%	
Yield strength	103-132 MPa
Density	2.71 g/cm^3
E-module	68600 MPa

Table 3.1: Properties of 99.5% Al

Aluminum is divided into two categories, wrought and cast. The wrought aluminum comes in eight different series, each with difference alloy elements and uses. Each alloy is designated with four numbers. The first digit indicates the major alloy element and thus the series. The second digit is a variation of the alloy and the third and fourth being specific for that alloy. The cast alloys have two standards one primarily used in America and one used in Europe, ANSI and DIN.

Each alloy also gets a temper designation which comes after alloy designation. This designation indicates how the alloy has been treated. The temper designation consists of a letter and or a number [3].

In the aerospace industry there are several alloys used. They are preferred for their lightness and their strength, with the 7000 series having the highest yield strength [4].

- 2024-T3 alloy
- 6061 alloy
- 7050 alloy
- 7075 alloy

Aluminum	2024-T3
Tensile strength	485 MPa
Yield strength	345 MPa
Density	2780 kg/m^3
E-module	72400 MPa

Table 3.2: Properties of 2024-T3 Aluminum

3.2.2 Titanium

Titanium (Ti) is a classified as a light metal with the strength of steel, but only half the weight. This makes it perfect for industrial use but also the aerospace industry.

Titanium is divided into 38 different grades, this includes pure Ti but also the different alloys. In aerospace grade 5 is the most widely used, but there are also variations here [5].

Grade 5 Ti	Ti 6%Al 4%V
Tensile strength	900-950 MPa
Yield strength	880-920 MPa
Density	4.4 g/cm^3
E-module	104000-113000 MPa

Table 3.3: Properties of Grade 5 Ti

3.2.3 Other Relevant Materials

Some other materials utilized when producing drones is fiberglass polyester and carbon-fiber. These are lightweight and strong materials that offer rigidity and strength close to their metal counterparts. These materials are considered composite materials because they utilize fibers for strength, and a thermoset-polymer to bind it all together. The fibers can be woven together, randomly arranged or flattened into a mat. The directions of the fibers are important with regards to the properties of the materials. The different structures and the somewhat random element of the fibers means, that the internal structure is not uniform.

When utilizing TO, the complex structures mean that a non-uniform material will not be able to support the geometry that is supposed to give the part strength. The material will not be as strong in all directions, and can lead to a critical flaw in a part. On this basis, the use of composite materials will not be explored further in this thesis.

3.3 Finite Element Analysis

The finite element analysis (FEA) is a numerical approach for analyzing structures and models. It uses partial differential equations and integral equation to find an approximate solution to the given problem. The method is heavily used in structural mechanics, as it originated as a method of stress analysis in solid structures. Now it has been implemented in analysis for heat transfer, electric and fluid flow among others. The tool has received a lot of attention in engineering, due to its diversity and flexibility. The reason behind its success can also be related to the improvements in computer hardware, as the mathematical equations used are complex and needs a lot of computing power. The method has also improved the engineering design process significantly. It provides a way of testing the design early in the design process, without having to make a prototype for testing. This is illustrated in **Figure 3.3** [6] [7].

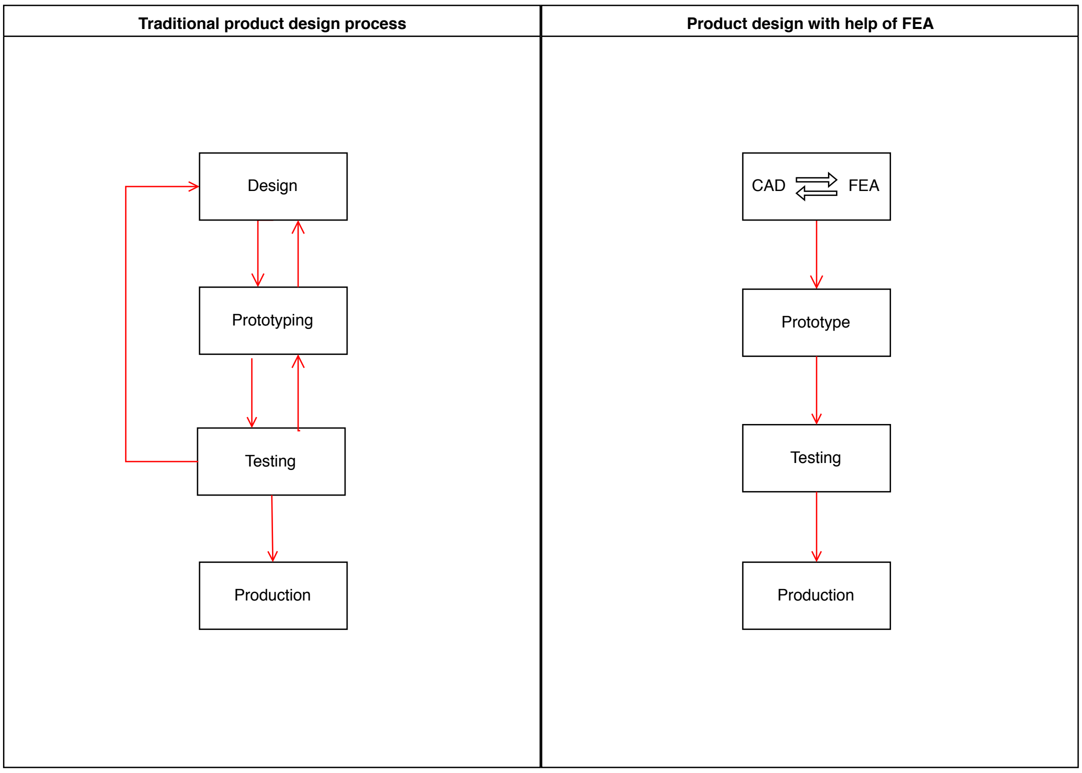


Figure 3.3: Design process with and without FEA

In a given engineering problem there are some unknowns. In solid mechanics, these problems are displacements within the structure. As a model is a continuum, or a set of more, the unknowns are infinite. To reduce the unknowns, the finite element method divides the model into a finite set of elements. These elements contain assumed approximate functions that expresses the unknown variables. The functions conveys the element properties, which in structural mechanics is the stiffness characteristics. The elements are defined with nodal points, and can vary to make the optimal shape of the elements. It is up to the user to define how many nodes each element should contain, and by that expressing the shape of the elements [6].

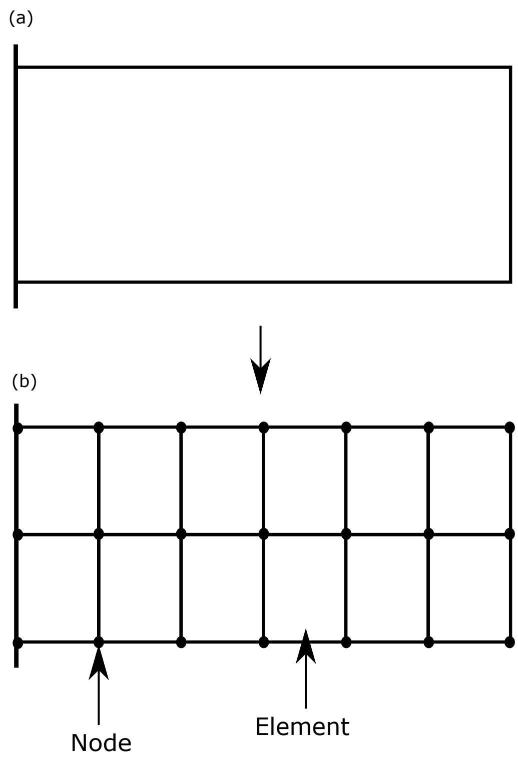


Figure 3.4: Example of elements and nodes on a simple 2D model of a beam

In **Figure 3.4a** a simple 2D model of a beam is shown. The figure is then divided into 12 elements, into a mesh, as shown in **Figure 3.4b**. This is a very simple mesh where each element is defined by four nodes. This mesh could now, in theory, be used in an FEA. When it comes to 3D-structures, elements are often polyhedrons.

3.3.1 Mesh and Element Size

The standard mesh shape in SolidWorks is a triangle. The triangles can be altered using different variables such as global element size and the tolerance, see **Figure 3.5**. When the mesh is generated the software will try to fill faces first with equilateral triangles close to the specified element size. Edges are done last where the tolerance is used to bond the mesh into one coherent piece.

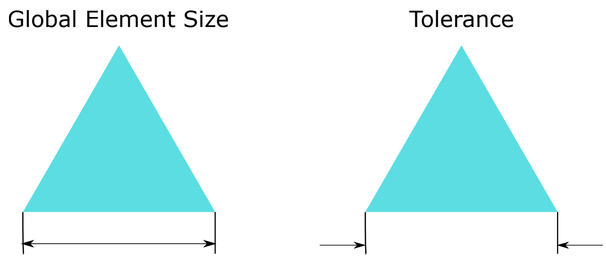


Figure 3.5: Global Mesh size and Tolerance illustrated

Another option is the curvature-based mesh, it is generated a bit differently than the standard mesh. Here the mesh element size is determined by how many triangles can fit into a circle, see **Figure 3.6**. The user can specify maximum and minimum values for size and the minimum number of elements in the circle. Curvature-based mesh is often used when meshing complex geometries [8].

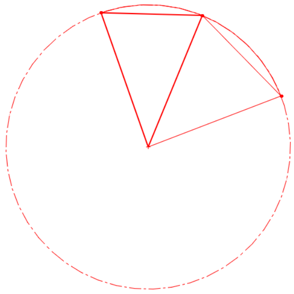


Figure 3.6: Curvature-based mesh size

It is important to note that other programs use other mesh styles. There are other mesh shapes such as squares and different polyhedrons such as a hexahedron, see **Figure 3.7**. Other programs such as Ansys also allows the use of different mesh shapes on the same part, this is called multizone mesh.

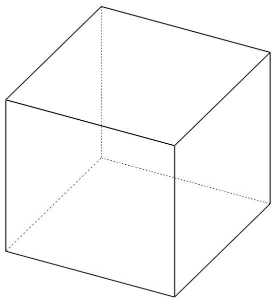


Figure 3.7: An example of a hexahedron mesh

By using a small number of elements, the chances of getting misleading results increase. A finer mesh provides a more accurate result at the cost of more computing power.

Figure 3.9 shows the distribution of von Mises stress in a bottle opener. The stress is graded from red to blue, where red is highest and blue is lowest. The bottle opener is fixed in two points (green) and has an applied load (purple). It is clear that the different element sizes gives different results. The first mesh, with the largest element size, indicates more stress than the two finer meshes. As the finer mesh uses more computing power, a favorable approach could be to start with a coarser mesh, and make it finer until the results converges.

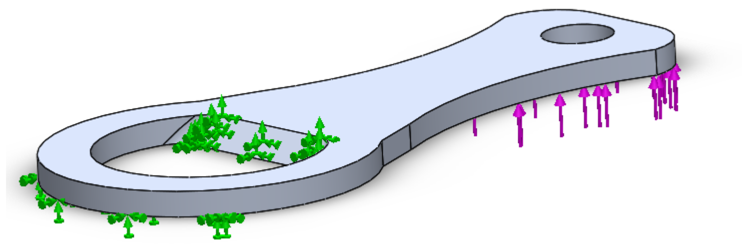


Figure 3.8: Constraints and loads for bottle opener
Model from SolidWorks

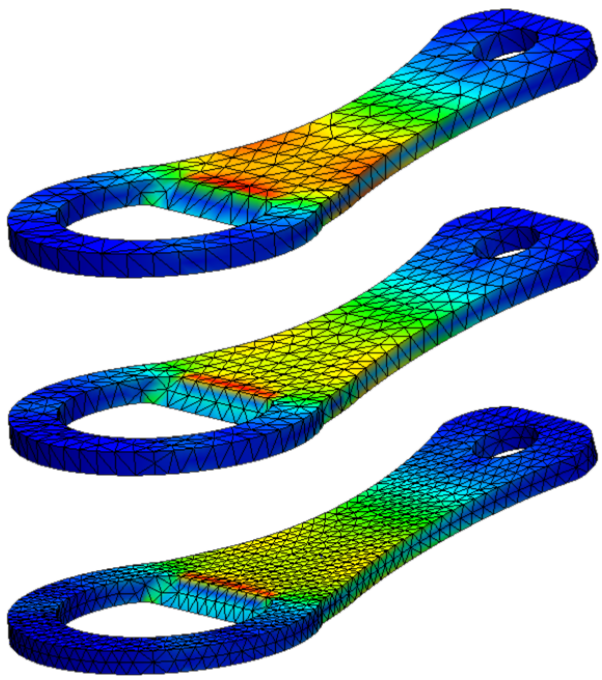


Figure 3.9: FEA of a bottle opener with different element sizes
Model from SolidWorks

3.3.2 Mesh Control and Singularities

When performing a FEA, you might have regions in the model that are more exposed to stress than others. A good way to increase the accuracy of the analysis in these specific areas, is to use a tool known as mesh control. Mesh control provides a finer mesh localized in the chosen area. Applying mesh control in a region, can yield a significant difference in the results [9].

In some models, sharp edges can cause problems for the simulation. The sharp edges can cause a singularity in the mathematical model that the program uses to complete its analysis. A singularity is, in theory of relativity, defined as a point in space and time where its properties are infinite. In FEA, a singularity can cause an unrealistic high value of maximum stress, and therefore make the analysis misleading. In **Figure 3.10**, this problem is showcased. As seen in (a), the highest stresses are concentrated to the small area where the bodies of the part meet in a sharp corner [10].

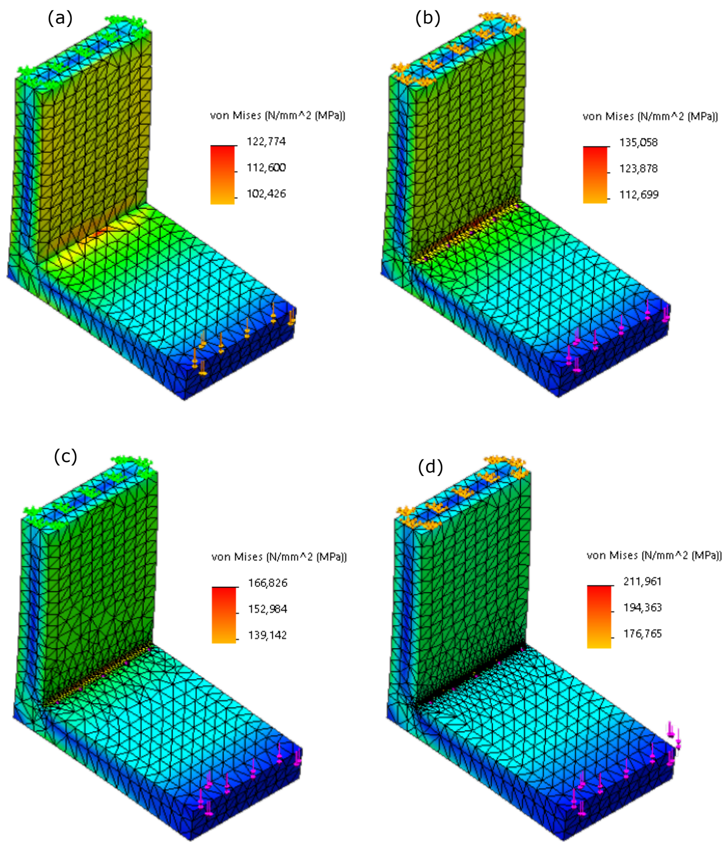


Figure 3.10: Singularity in a bracket
Model from SolidWorks

Figure 3.10 shows an analysis without mesh control in (a), and then mesh control of different sizes in (b), (c) and (d). All the mesh controls has a ratio of 1.5, but the element size varies. In (b), there is an element size of **1.5 mm**, **1 mm** in (c) and **0.5 mm** in (d). As the results shows, the von Mises stress increases for each analysis and does not show any sign of converging. The finer mesh does nothing to improve the accuracy of the model, but rather just increases the stresses, this is a typical sign that a singularity is present. To rectify this, one method is to alter the model to avoid the stress concentrations in the first place.

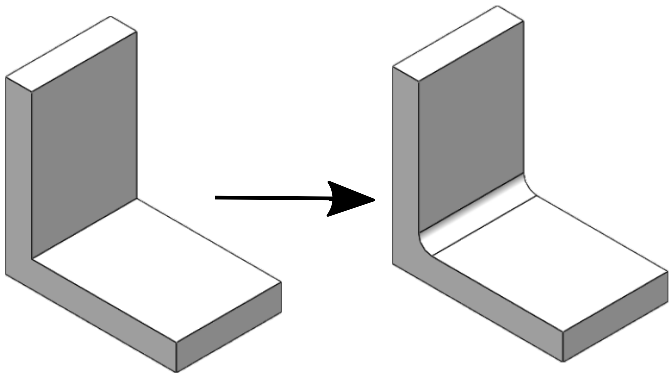


Figure 3.11: Remodel
Model from SolidWorks

Figure 3.11 shows the altered design of the bracket. To avoid the sharp corner where the stresses where previously concentrated, a fillet was added. A fillet is the practise of rounding corners. After remodeling, the analysis is run again to determine if the altered design has improved the stress concentrations.

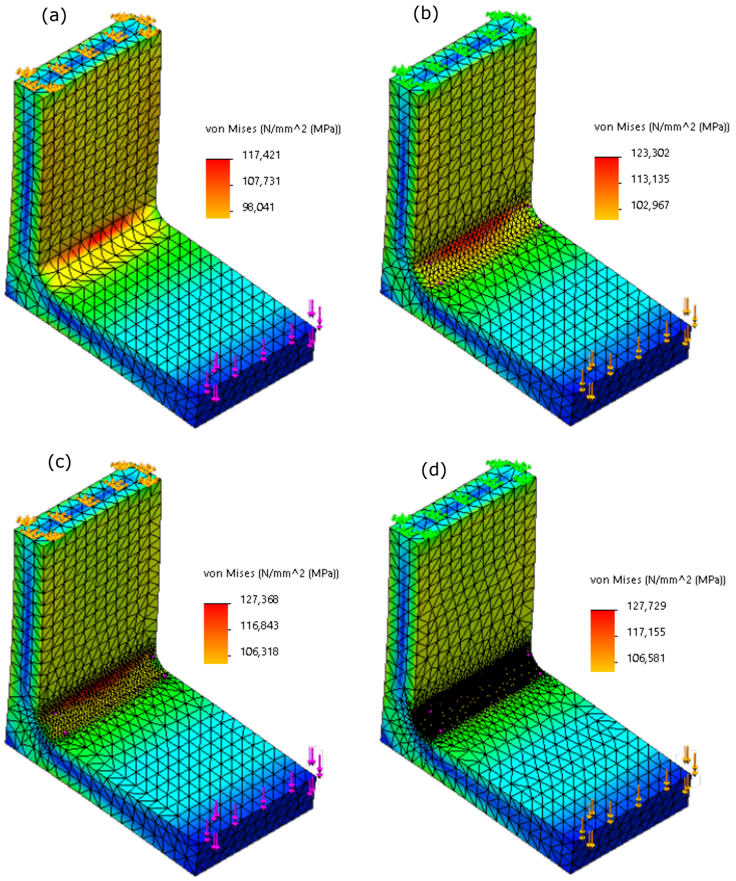


Figure 3.12: Stress concentration in a remodeled bracket
Model from SolidWorks

As shown in **Figure 3.12**, the finer mesh does not translate into higher and higher stresses. The stresses converge on a result. This means that the singularity is gone and the redesigned part gives a more realistic result. The tests show the importance of model design, but also the importance of experience when utilizing FEA in the design process.

3.3.3 Finite Element Analysis Studies

The simulation tab in SolidWorks offers several types of studies, the studies utilized in this thesis will be explained further.

Simulation types	
Static	A static simulation with linear materials and geometry
Frequency	Used to find the natural frequency with or without loads and fixtures
Thermal	Used to simulate effects of thermal loads on designs
Buckling	Used to test the buckling strength of a part, with or without loads
Fatigue	Used to test cyclic loads on parts
Non-linear	A static simulation with non-linear material and geometry
Linear-dynamic	A dynamic simulation with linear materials and geometry
Non-linear dynamic	A dynamic simulation with non-linear materials and geometry

Table 3.4: Simulation studies available in SolidWorks

A **static simulation** is a simulation where the forces applied are static. A static force is a constant force applied to a stationary object.

The assumption in a static study is that the forces are applied slowly over time, allowing for a simplified simulation to be conducted saving time and computational power when compared to a dynamic study. Another assumption is that the material behaves in a linear fashion as well. A linear material means a material that has a linear curve up to the yield point, see **Figure 3.2** in **Chapter 3.2**. It also has to have relatively little deformation, or little enough displacement that the material properties are not affected [11].

The **frequency study** is used to identify the natural frequencies of a design and can be done with and without load and fixtures. It is important to know of these frequencies as they may cause resonances that can damage or destroy components.

A **non-linear static study** is a static study with materials that have a non-linear curve up to the yield point and/or large displacements. A rough estimate to evaluate this is if the displacement is close or bigger than the thickness of the part. It can also be used when two parts are interacting in a way that leads to significant deformations. [12].

The difference between a static and a dynamic study has to to with time, and time as a variable. Consider a bridge, if a train is stopped and is stationary on the bridge, it can be considered a static study because the sum of the forces equals zero. If however the train is moving across the bridge, there are several variables that are time dependant in the study. Variables such as acceleration and velocity are present and then the forces will not be equal to zero. The dynamic studies can therefore provide more realistic simulations since time is used as a variable.

The **linear-dynamic study** is a lot more demanding when it comes to computational power, compared to static studies. In a dynamic study the forces are generally applied inside a given time frame, allowing for forces to be dynamically applied to a part. As with a static study, a linear-dynamic study uses materials with linear properties [12].

The **non-linear dynamic study** is used when the materials utilized has non-linear properties or behaves in a non-linear fashion. Such a study should also be considered when there are large displacements. As with the linear-dynamic study the computational strain is significant especially with more complex parts. This study is probably the most accurate when it comes providing realistic results [11].

Part of choosing which of these studies to use is deciding what assumptions and simplifications you can make, and which ones apply to a given scenario.

3.4 Topology

Firstly, topology is known in mathematics as the study of shapes. It is focused on the continuous deformation of an object, such as stretching, bending, twisting etc. It can be explained using the Euler characteristic of a shape. The Euler characteristic is defined as:

$$X = V - E + F \tag{4}$$

X = Eulers characteristic, V = Vertices(Corners), E = Edges, F = Faces

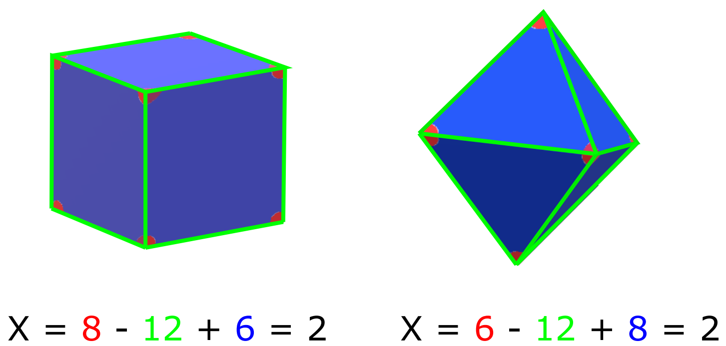


Figure 3.13: Eulers characteristic of a cube and octahedron

In **Figure 3.13** the Eulers characteristic of the cube and octahedron is calculated and compared, they both have a value of **two.** This means they are equal, you can deform both figures indefinitely, but with regards to topology they are the same. This also applies to other shapes. A tetrahedron (or three sided pyramid) also has an Euler characteristic of **two.** A 3D objects with no holes, has an Euler characteristic of **two.**

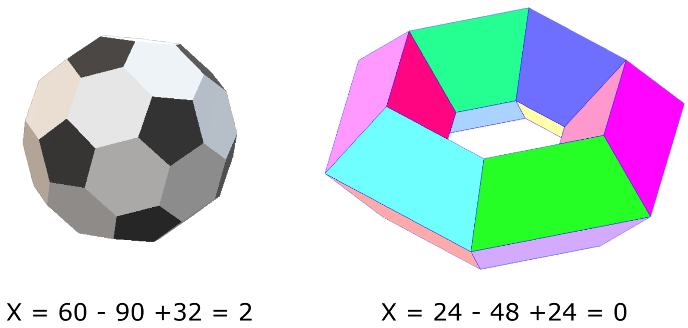


Figure 3.14: Euler characteristic of a football and a Hexagonal torus [13]

In **Figure 3.14** we can see a football, it has a value of **two** as well. If we now punch a hole in it, it suddenly has a value of **zero.** This is true for all objects vaguely torus shaped. This means that in terms of topology, a donut and a coffee cup, is the same [14][15].

3.5 Topology Optimization

Topology optimization (TO) uses complex mathematical methods to manipulate the topology of a part, but also drastically change it until it meets the set boundary conditions. By changing the topology, the material layout can be altered to get the most optimal results. By applying TO to a design, it can be optimized with regards to stiffness, weight, displacement, etc. The process works towards an optimum, matching the given load and boundary restrictions. TO can also optimize with regards to other properties, such as thermal, fluid flow and vibration. This thesis will focus on TO used for stiffness, weight and displacement. TO is only one of three different kinds of optimization used in the design process:

- Size Optimization
- Shape Optimization
- Topology Optimization

What separates these three methods are how they work and when they are applied. Size and shape optimization are primarily used to increase strength and finding the best compromise between several design parameters. TO can also be used to increase strength, but mostly is used to reduce weight and maintain the strength of the non optimized design. TO also has the benefit of being able to add holes, trusses and voids into the design. This means it has to depend less on the design parameters than the other two methods and will produce more optimized design when fewer restrictions are applied. **Figure 3.15** shows the differences between size, shape and topology optimization. This figure is inspired by *Topology Optimization: Theory, methods and applications* by Bendsøe and Sigmund. [16]

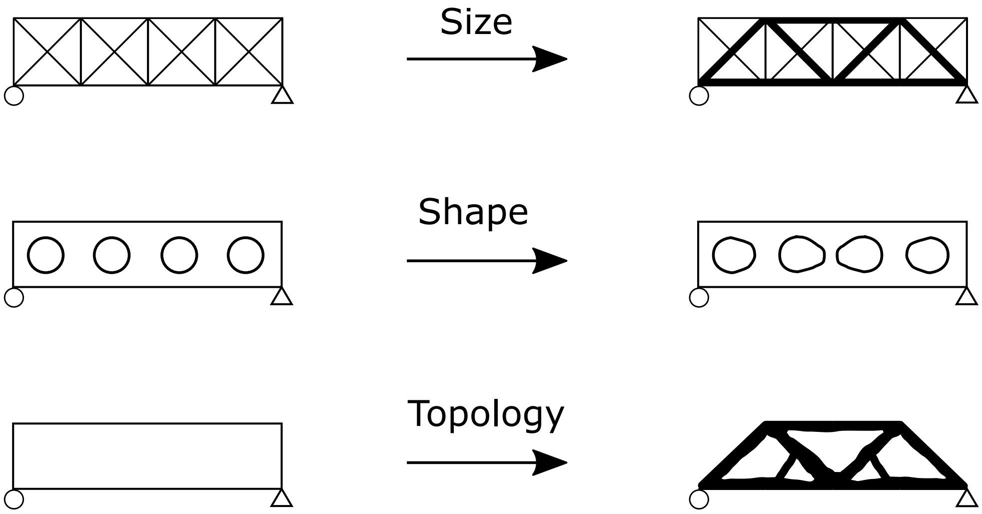


Figure 3.15: Differences between size, shape and topology optimization

Topology optimization is a complex mathematical method that itself has different branches. In **Figure 3.16** we can see that TO is divided into three main categories, element based, discrete and combined. These also have sub levels that divide them even further [17].

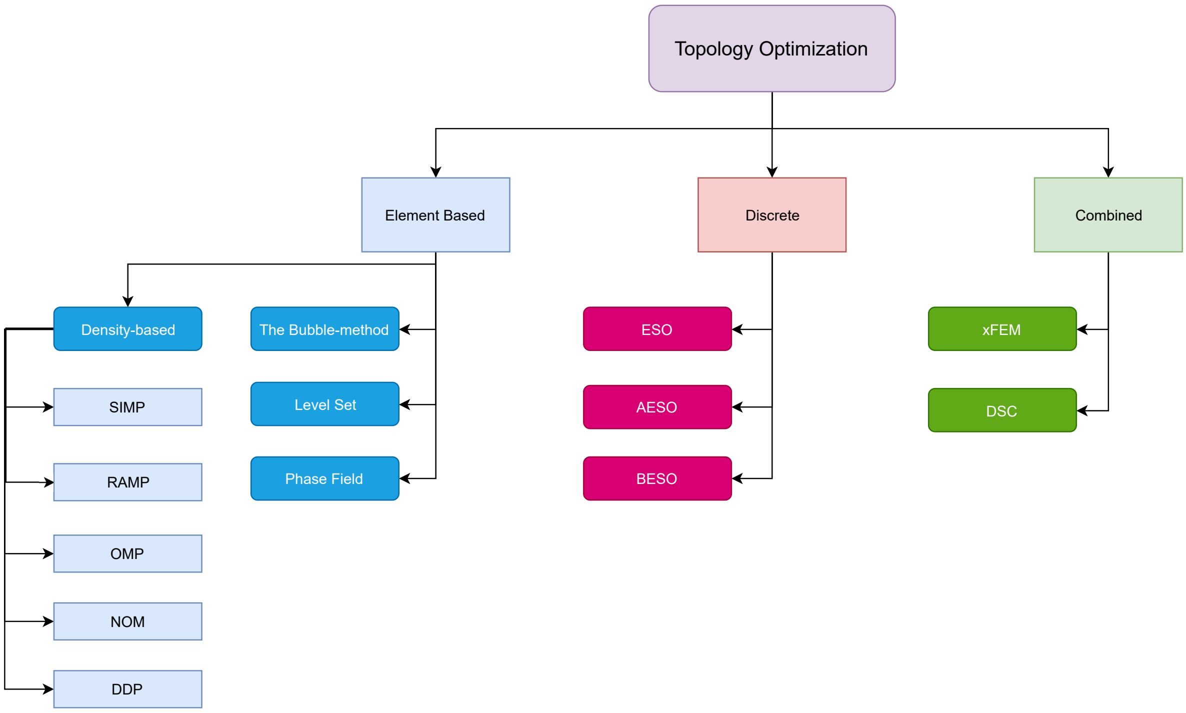


Figure 3.16: Topology optimization diagram

3.5.1 Solid Isotropic Microstructure with Penalization

Solid Isotropic Microstructure with Penalization (SIMP) is one of the most common mathematical methods to solve a TO problem. SIMP is an element based process, and is categorized as an Isotropic-Solid/Empty topology study. The basic idea of SIMP was proposed by Bendsøe in 1989. [18]

In **Figure 3.17** there are two simple examples of how Isotropic-Solid/Empty topology in 2D works. The figure is inspired in part by "Aims, scope, methods, history and unified terminology of computer-aided topology optimization in structural mechanics" by Rozvany

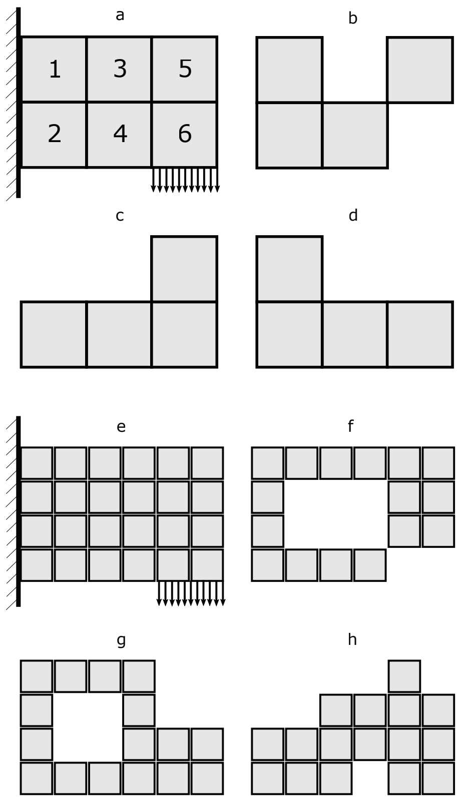


Figure 3.17: ISE Topology Optimization

In the first example (a-d) there is a part consisting of six elements/blocks. The main goal is to optimize this part with the given constrains, boundary conditions and loads. In this case, the objective is to remove 1/3 of the weight of this part. This means two out of six elements need to be removed.

(a) Shows the problem. Element 1 and 2 is fixed to a surface and element 6 is affected by a force, pointing downwards.

(b) Is showing a not feasible solution because the force is supposed to act directly on element six, which is removed.

(c) Is a showing a possible solution, but not optimal.

(d) Is showing the optimal solution.

In the next example (e-h), the constraints are the exact same, but the elements have been split into four equal sized elements. This gives a total of 26 elements, instead of six. Here there are lot more solutions, than in the first example. When simulating this in a CAD program and a fine mesh is used (more elements), the simulation will take much longer to complete. This is because the program has many more variations to go through [19] [17].

(e) Shows the problem

(f) Not feasible solution

(g) Possible solution, maybe optimal

(h) Possible solution, but not optimal

ISE topology optimization usually have a large number of ground elements. Utilizing this method directly would be expensive, and a use of a continuous variable formulation is therefore advised. The continuous variable formulation can introduce density of elements, resulting in a large amount of gray elements. To maintain the black and white elements needed for a topology optimization, penalization of grey elements is introduced. The penalty factor diminishes gray elements to either black ($\rho = 1$) or white ($\rho = \rho_{min}$). Various functions can be used for this. One of the simplest is called the power law, and was introduced by Bendsøe in [18]. [20] [21]

One way of describing the formula mathematically is described in "Overview of structural topology optimization methods for plane and solid structures" by Cazacu and Grama. [21] The problem is formulated as:

$$SE(P) = \sum_{e=1}^{N} (\rho_e)^p [\, u_e]^{\,T} [\, k_e | u_e] \tag{5}$$

This formula is subjected to the constraints:

$$V^* - \sum_{e=1}^{N} V_e \rho_e = 0 \tag{6}$$

$$0 < \rho_{min} < \rho_e \leq 1 \tag{7}$$

$$SE = \text{strain energy}, P = \text{optimization parameters}, e = \text{element}, N = \text{number of elements},$$

$$\rho = \text{density}, p = \text{penalty factor}, u_e = \text{nodal displacement vector}, k_e = \text{stifness matrix},$$

$$V^* = \text{target volume}$$

3.5.2 Evolutionary Structural Optimization

Evolutionary Structural Optimization is a relatively simple concept. It is based on the idea that a structure evolves towards an optimum by removing low stress elements bit for bit. This process can be seen in natural structures such as shells, bones and trees. This is also called the hard-kill method. Where elements have one of two colours that correlate to a value. Black for solids and white for voids. ESO begins with an oversized part and then slowly moves towards a structural optimum. ESO only removes material and is considered computationally inefficient [22].

The original ESO algorithms relied on finding the von Mises stresses in each element (σ_e^{vm}) and then comparing those with a predetermined or maximum von Mises stress for the structure (σ_{max}^{vm}). A rejection ratio (c_{rr}) is defined and any element that does not meet this threshold is eliminated.

$$\sigma_e^{vm} < c_{rr} * \sigma_{max}^{vm} \tag{8}$$

This is process is repeated with the same rejection ratio until the structure reaches an equilibrium, meaning there are no more elements that qualify for elimination. The rejection ratio can then be altered according to a defined evolutionary rate c_{er}.

$$c_{rr}^{new} = c_{rr}^{old} + c_{er} \tag{9}$$

The elimination process can now start again and will run until equilibrium is reached or the required optimum is reached [23].

Bidirectional Evolutionary Structural Optimization is a subsection of ESO. This method allows new elements to be added to the structure where they are needed. Mostly in close proximity to high stress elements. In **Figure 3.18** the steps in a typical BESO process is shown. This instance is a simple 2D design, fixed on the left hand side and the load towards the right hand side. Through steps a-h the different iterations of this design can be observed.

(a) Shows the design space, fixture points and load position

(b) The FEA of the part, showing loads in colour. Red means more stress, blue means less or no stress

(c) The first iteration of the BESO process. There is some material being removed on the left hand side of the part

(d) Second iteration of the part, even more material has been removed, now seeing material being removed on the right side as well.

(e) Some iterations were skipped

(f) Around halfway through the iterations

(g) A finished design is beginning to emerge

(h) The final iteration, this process needed 42 iterations to arrive at a optimum.

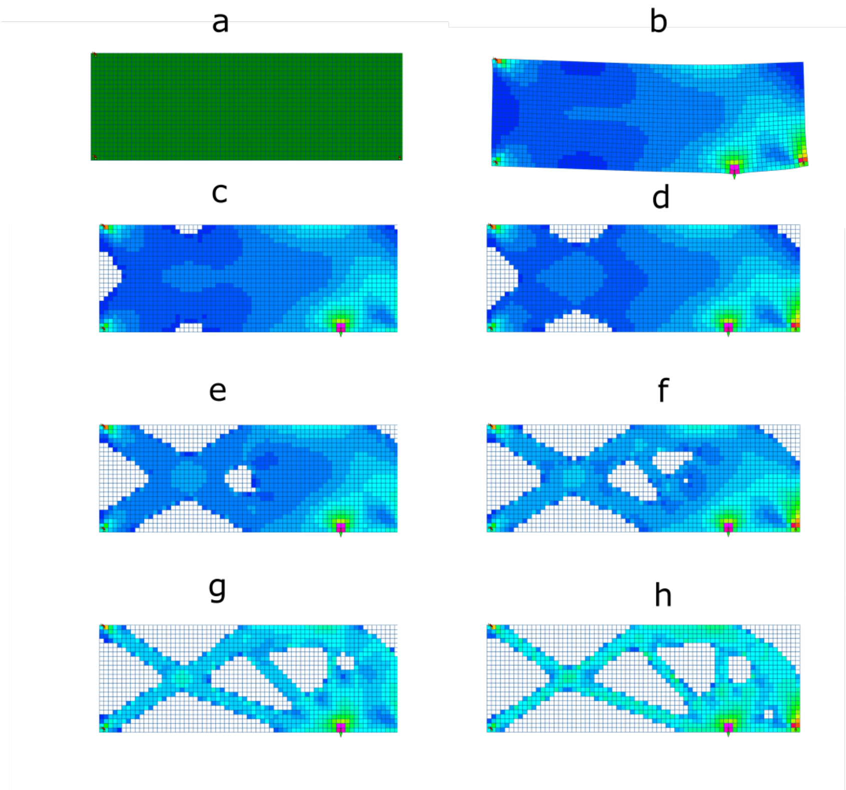


Figure 3.18: Steps of a 2D BESO process

3.5.3 The Level Set Method

The level set method (LSM) is a numerical method used for tracking interfaces and shapes and was invented by Osher and Sethian in 1988. It is based on the Hamilton-Jacobi equation, a mathematical equation used in physics, which thrives when it comes to identifying conserved quantities for mechanical systems. LSM is currently being used in a plethora of different disciplines, such as image processing, computer graphics, computational geometry optimization and computational fluid dynamics.

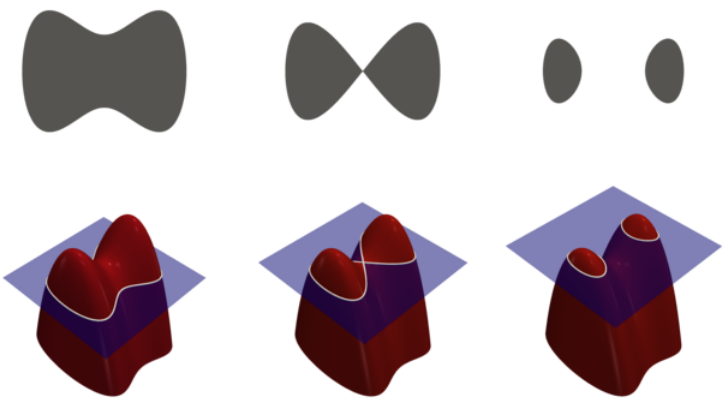


Figure 3.19: Illustration of the level set method[24]

When it comes to TO the LSM works by moving two scalar fields, v(x,y) and g(x,y), through a design area Ω . The v-field determines the geometric restrictions of the process and the g-field determines nucleation of new holes based on set parameters. The parameters set for g(x,t) are defined by the w-variable in the equation, if this is set to zero the level-set function φ will move outwards from Ω with the boundary v(x,y) and the equation will work more as a shape optimization tool. If LSM is to be used for TO, a value w needs to be introduced for a hole nucleation to occur.[25]

$$\frac{\delta\varphi}{\delta t} = v|\nabla\varphi| - wg \tag{10}$$

$\varphi =$ Level-set function, $t =$ Time, $v =$ Geometric scalar field, $g =$ Nucleation scalar field, $w =$ constant

3.6 Lattice Optimization

The state of technology in recent years have opened the window for the possibility of Lattice Optimization (LO). LO alters the geometry by replacing the solid infill of the part with a lattice infill. Due to the excellent properties it has such as; the strength and stiffness to weight ratio, energy absorption rate and thermal isolation, it is currently being researched for use in several different industries such as aerospace, medical, automotive and naval. **Figure 3.20** displays the use of LO on a hip prosthetic

Figure 3.20: Example of LO used in hip prosthetic [26]

Although there are a lot of advantages to using lattice structure there are still some problems with the technology required to produce it. The only reasonable way to manufacture a product with lattice structure is by utilizing RP and AM. Even with these two methods, it is still not optimal as the AM production method might trap material inside the lattice structure, if the design is completely closed. LO is also quite demanding when it comes to computational power, and even with a powerful computer the time increase is significant.

3.7 Challenges with Topology Optimization

There are some challenges with using TO when it comes to producing parts. Another issue is that the design usually is rough and contains geometry that is not easy to manufacture. This means another simplification process is required before the design can be put into production. With the traditional production methods such as machining, casting and extruding, there is a limit to the geometry that can be easily manufactured. There are certain constraints that have to be taken into account, such as tool access, draft angels and other clearances.

Something to consider is the problem of the local optimum. By applying the constraints and other hard restrictions, the results of the TO study might not be what is truly optimal. When the design is restricted too much by the constraints, such as bolt location or a limited design space, the TO results will be limited. To find the true optimum the boundary conditions also have to be optimized, which is not always possible. [17]

There are also some mathematical instabilities that can occur when using TO. The most visually striking is the checkerboard effect, see **Figure 3.21**

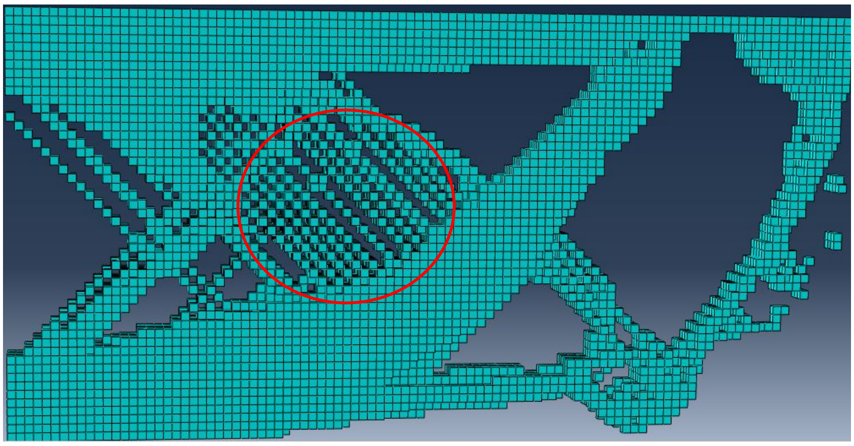


Figure 3.21: The checkerboard effect showcased
Model from Abaqus

The checkerboard effect is defined as *"a periodic pattern of high and low values of pseudo-densities,arranged in a checkerboard layout"* by Shukla, Misra, and Kumar in “Checkerboard problem in finite element based topology optimization”. Checkerboarding occurs because of numerical instabilities in the code and produces an unrealistic stiffness in the design. The pattern is also difficult to produce. There are several methods to help with the checkerboard effect, and the easiest one to implement, is just to increase the elements in the FEA. This requires additional computational power and as such increases the simulation time [27].

In some cases it is possible to get TO results that are non viable in their current form, see **Figure 3.22**. These results show a part of the design not connected at all to the rest. This result highlights the importance of setting up the study correctly. With this specific example there are two fixed points, the top surface and the through going hole. Then a force is applied to the surface on the right. **Figure 3.22** shows a TO result with a **50%** mass reduction and although the solver has fulfilled the requirement, the design is rendered completely useless.

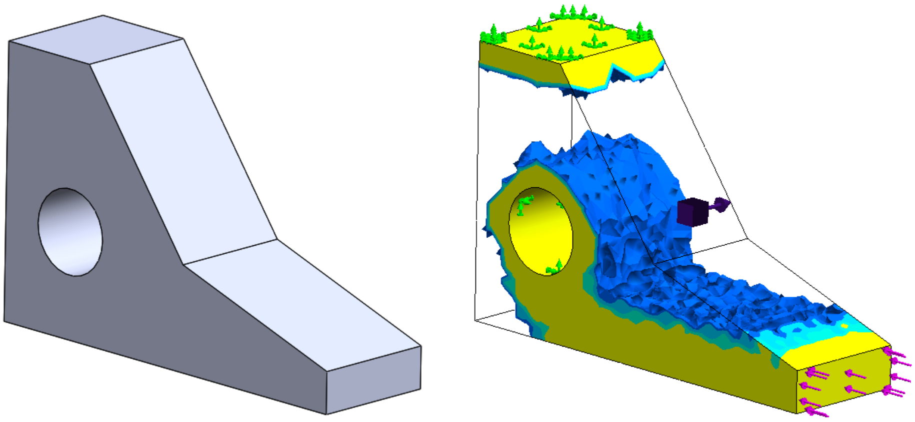


Figure 3.22: A Non-Viable Result
Models from SolidWorks

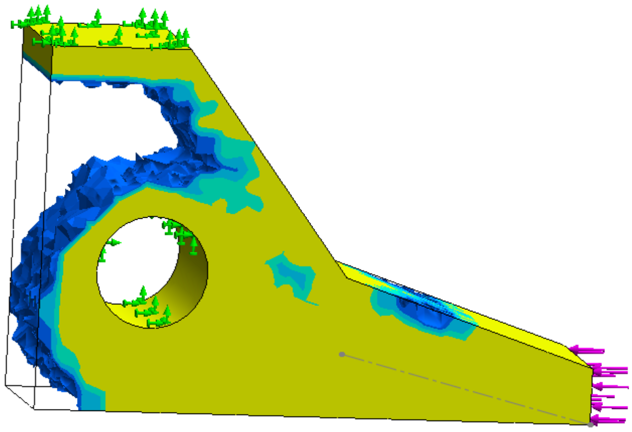


Figure 3.23: The same design but with 30% mass reduction
Model from SolidWorks

Figure 3.23 shows a new TO study, this time with a mass reduction off 30% instead of the 50% shown in **Figure 3.22**. This has resulted in a design that is heavier than the previous one, but at least it is one coherent design.

3.8 Design Study Tool in SolidWorks

The design study tool is a simulation that runs through several variations of a design specified by the engineer to find the optimal design. The input parameters can be dimensions, forces or other variables. The study will run a FEA with the dimensions given, then alter the variables before running another FEA. The simulation will run through all the combinations of variables. Information on the different designs is available, and the engineer can see what design combinations meet the design criteria.

It is also important to specify the end goal correctly with such a study. The end goal can be what ever is needed, such as mass reduction, a set volume, minimum or maximum displacement, etc. The design study does take a lot of computational power if the amount of variables are many, or the steps are small. It offers an optimized design early in the design process which allows the engineer to base further work on calculated numbers, rather than only intuition and experience.

3.9 Production Methods

Producing the TO design can prove to be a challenge, often the complex geometry of the design can be difficult to produce with traditional production methods such as casting and extrusion.

3.9.1 Casting

Casting can be described as pouring molten metal into a mold cavity, where the metal solidifies and takes the shape of the mold. There are many different casting methods, but they are mainly divided into three categories, see **Table 3.5**.

Expandable Molds	The mold is broken to remove the casting
Permanent Mold	Used for repeatable castings, the casting is removed without destroying the mold
Composite Molds	Made of two or more materials, they have a permanent and a expendable portion

Table 3.5: The major casting categories

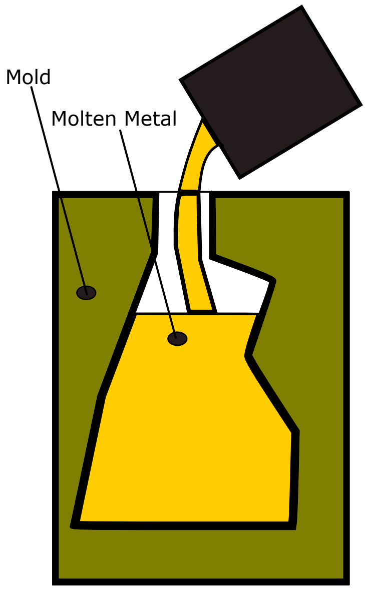


Figure 3.24: The basic casting process

Figure 3.24 shows the basic casting process. In theory it is a simple method, but there are several limitations. Casting complex parts with lots of details is often not feasible, as the metal cools down before flowing into all the details. When using permanent mold the draft angle also has to be considered, draft angle is the angle between the casting and the mold, see **Figure 3.25**. The draft angle is there to help remove the casting from the mold. If the angle is to steep, it will be difficult to remove the part from the mold, therefore a more shallow angle is preferable. Draft angles often are between 0.5-2 degrees. When it comes to TO, casting has limitations and will not always work, it is largely dependant on the geometry of the design [28].

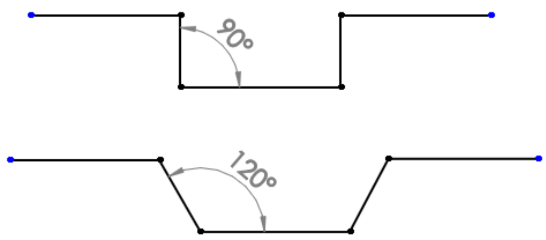


Figure 3.25: Draft Angle in effect

3.9.2 Extrusion

Extrusion is a production method for producing continues cross sections in large or small batches. The process works by pressing a material through a static die, this deforms the material into the required shape. Imagine squeezing frosting through a tube onto a cake, that is the basic extruding process. The process can be seen in **Figure 3.26**. One requirement for the extrusion process is a constant cross section. Typical materials used for this are copper, aluminum, steel and magnesium [28].

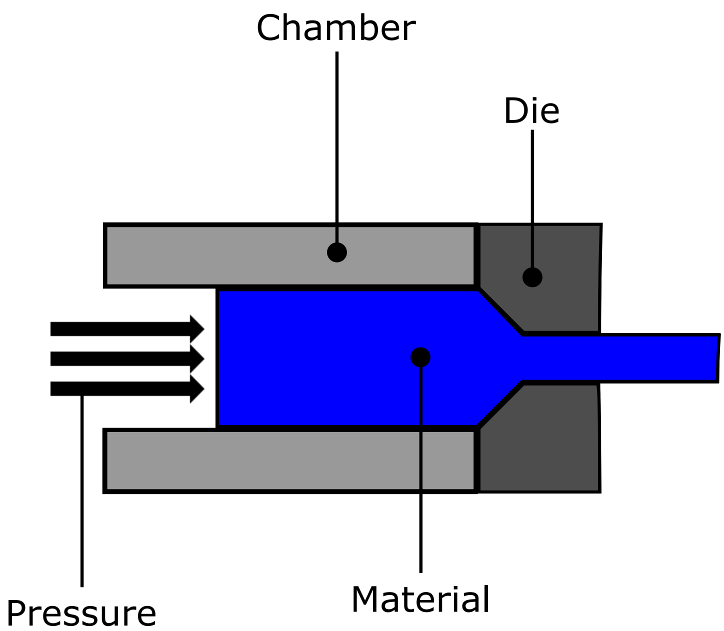


Figure 3.26: The basic extrusion process

3.9.3 Rapid Prototyping and Additive Manufacturing

A useful tool when working with CAD models is the technology of rapid prototyping (RP), also known as 3D printing. RP is used to rapidly produce prototypes. This can be very beneficial when working with TO and in general design work. It allows one to quickly print a prototype and see the model in real life, helping with the design process. RP works by slicing a CAD model into thin layers, and stacking them on top of each other. These layers are printed one by one to produce the 3D geometry. There are certain things that need to be taken into consideration, even with such a new technology. If a part sticks out over itself and remains "hanging" in thin air, it is referred to as overhang. If the overhang becomes too big, it requires additional support material to successfully print. **Figure 3.27** shows the overhang concept and the support material.

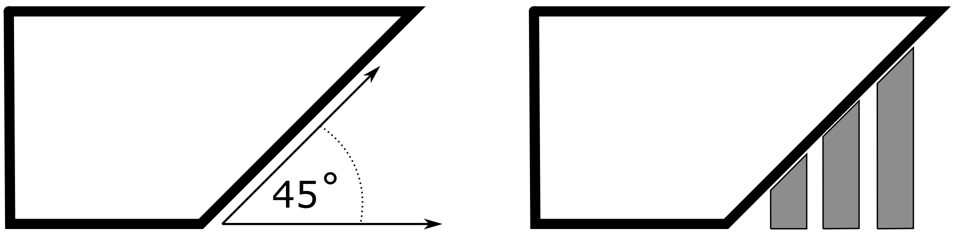


Figure 3.27: Overhang with and without support

Another production method, known as additive manufacturing (AM) is more suited to producing the TO results. AM is a more developed version of RP. AM works on the same principles as RP, slicing up the CAD model into thin layers and producing them on top of each other. Where RP produces prototypes, AM aims to manufacture usable parts rather than prototypes. Today, there are machines that can produce parts in a range of different polymers and metals [28][29].

4 Methodology

In this chapter the general methods used in this thesis will be explained and outlined. It will contain relevant information on the preferred software, how it operates and some of the relevant tools and options available. Other software used will be explored and explained in their respective chapters.

4.1 General Method

This thesis is focused on subjects previously unknown to the group. Due to the limited information on application of TO, the group had to apply an approach to the research. This consisted of intuition, ingenuity, trial and error, and creativity. Failure in this context is not always bad, but can give valuable insights that can be exploited and applied to other scenarios and designs. Other tools, such as brainstorming and discussion, were used to great effect, to generate ideas, designs and methods.

4.2 Literature Study

A big part of this thesis depends on gaining reliable information about the different subjects of study. To do so, the group has used different resources. Students at NTNU has access to an online library know as Oria, which contains the gathered resources of the library, such as books, articles, journals, etc. In addition there are also several physical libraries available. Another tool also utilized, was the Google Scholar site. Google Scholar is an online library of scholarly literature and is the largest academic search engine in the world.

Both supervisors, at NTNU and at Sevendof, has shared articles and other information on the subjects to help further improve the quality of the thesis. To further this, different sources on the same subject has been cross-examined. There has also been frequent contact between the student and the supervisor from NTNU, to keep the group on the right track regarding the main goals of the thesis.

4.3 General Finite Element Analyses Method

The main software used in this thesis is SolidWorks created by Dassault Systèmes. It was chosen because of the groups experience with this software, but also because it offers a integrated CAD, FEA and TO package.

The FEA lays the ground work for the TO study and is a vital part to get right. If the FEA is not defined correctly the results from the analysis will not be correct and then the TO study will be based on flawed results, undermining the entire design.

There are broadly three steps to defining the FEA:

1. Defining the fixture case
2. Defining the load case
3. Specifying the mesh

Other factors to consider are material, connections set and what type of study to run. To give a basic understanding of the FEA method, this chapter will only focus on the static study.

Defining the fixture case consists of coming up with a realistic method for fixturing the part or finding the most accurate way to represent a real life fixture in the FEA environment. This can can be challenging as there are often many ways to define a problem. SolidWorks offers a few options when it comes to fixturing, **Table 4.1** shows the different options.

Fixture	
Fixed geometry	Completely fixes the part, can be applied to faces, features and edges
Roller/Slider	Allows movement in a plane, but prevents movement normal to the plane
Fixed hinge	Allows rotation around an axis
Elastic support	Applies a elastic foundation between selected faces
Bearing fixture	Applies a bearing connection between a shaft and a housing
Foundation bolt	Applies a bolted connection to a virtual foundation
Advanced fixtures	Allows for more advanced fixtures

Table 4.1: Fixture options in SolidWorks

To showcase the method, a simple example is used. In **Figure 4.1** a fixed geometry is applied to the inside face of the holes, simulating a bolted connection that is restraining this part.

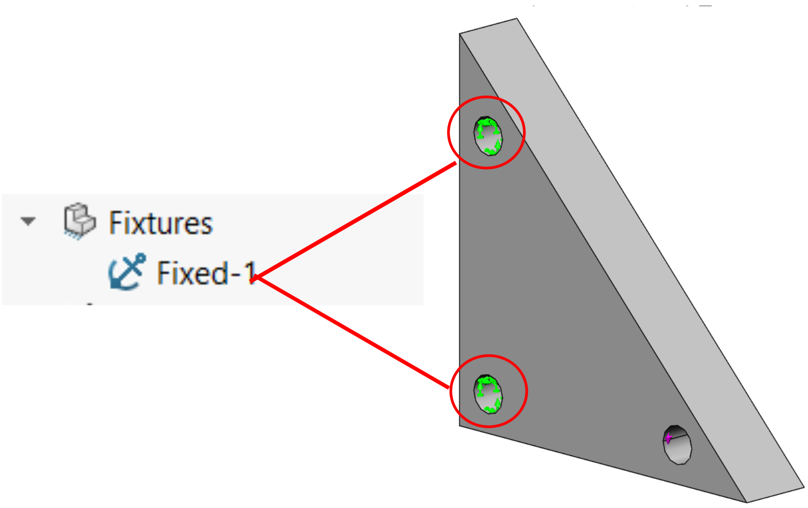


Figure 4.1: Fixed geometry applied on a design
Model from SolidWorks

Defining the load case is a similar process to defining the fixture case. A load case must be applied in a realistic manner to the part. To define this properly can be a challenge. SolidWorks has several options when it comes to defining the load case. **Table 4.2** shows the alternatives.

Load	
Force	A force applied to a face, feature or edge
Pressure	Applies a pressure on face
Gravity	Applies gravity on the part
Centrifugal Force	Applies a centrifugal load on a axis, face or cylindrical face
Bearing load	Applies a bearing load on a cylindrical face
Prescribed Displacement	Applies a displacement to a face, feature or edge
Remote load	Applies loads with a load source not directly working on the part
Distributed mass	Applies a load distributed equally on a face

Table 4.2: Load options in SolidWorks

In **Figure 4.2** a force is applied to the lower half of a cylindrical face, this is to simulate a pin with a downwards force applied to it.

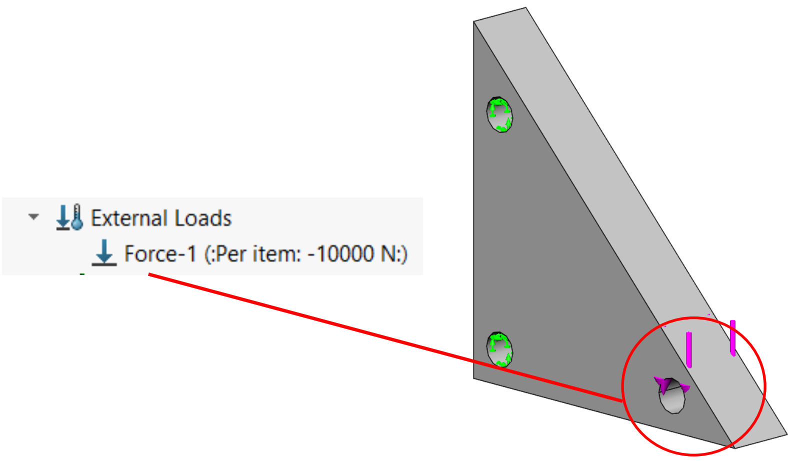


Figure 4.2: Force applied to a design
Model from SolidWorks

Specifying the mesh is all about finding the optimum between accurate results and simulation time. Sometimes a rougher mesh is better when just running quick simulations. SolidWorks gives a large amount of control over the mesh to the user. **Figure 4.3** and **Table 4.3** shows the different options for the various mesh types.

Mesh types	
Standard mesh	Uses standard mesh triangles, the global size and the tolerance can be set
Curvature-based mesh	Uses mesh triangles that can fit into a circle. Max and min element size can be specified, also how many elements fit into a circle and the growth size ratio
Blended curvature-based mesh	The same options as the curvature-based mesh but uses another algorithm for calculating the mesh

Table 4.3: Mesh options in SolidWorks

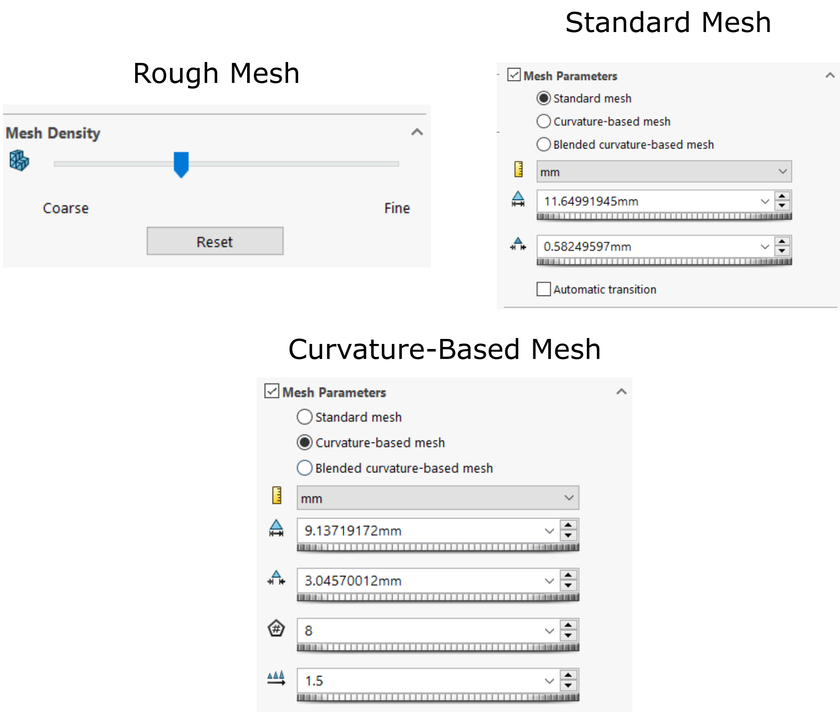


Figure 4.3: Different mesh options

The approach during this thesis and the different examples presented in it, has been to use a rough mesh to speed up simulation time and ensure that a good pace was kept. When studies were conducted on the parts supplied by Sevendof, a finer mesh was used to ensure more accurate results in the studies.

4.4 Design Study

After doing a simulation, i.e. static analysis, one can use these results to optimize the design with the design study tool. By defining a goal, such as lightest possible design with a displacement less than 2mm, the simulation will run through the possible alternatives by altering the variables accordingly. In this next example, an I-beam is used to highlight how the design study works.

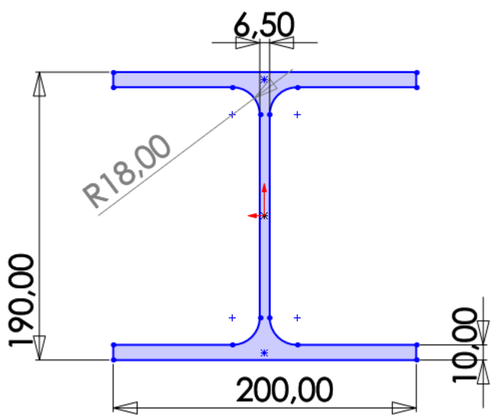


Figure 4.4: Dimensions of I-beam

The dimensions of the beam is displayed in **Figure 4.4**. This is a standard beam, EN 10 025, a HE 200A beam. [1]. The length of the beam is **8 m**. First, a static study is conducted, with fixtures in both ends and a force applied in the middle. The results from the study will form the basis of the design study.

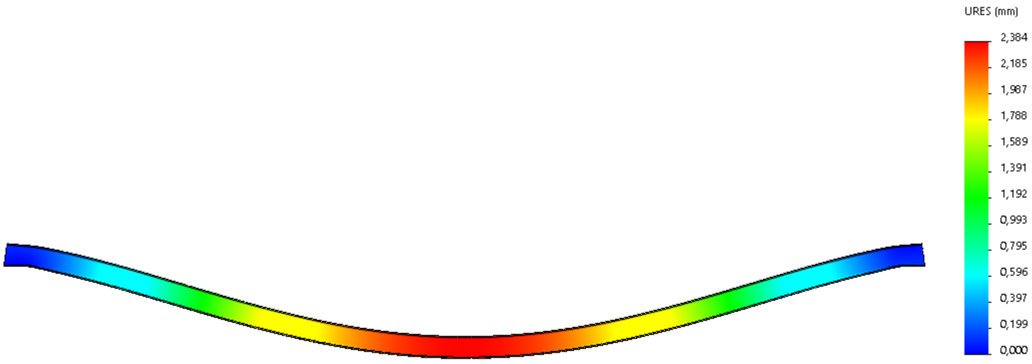


Figure 4.5: Displacement results from static study
Model from SolidWorks

Figure 4.5 displays the displacement plot from the static study. It is wanted to decrease this to under **2 mm**, while keeping the beam as light as possible.

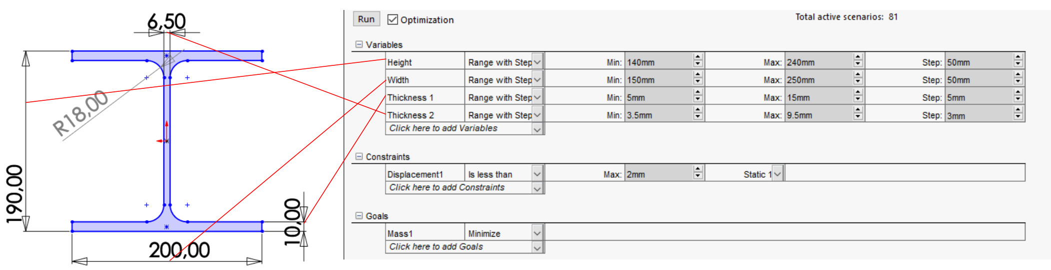


Figure 4.6: Design study input

Figure 4.6 shows the different variables the beam contains. The height of the initial design is **190 mm**. This variable is set between **140 mm** and **240 mm** with a **50 mm** interval step. This means the design study will test three different heights. As for the width, the study will test designs for three different widths, $\pm$ **50 mm** from the initial design. The same goes for the two thicknesses, with $\pm$ **5 mm** and $\pm$ **3 mm**. The study will then check all the combinations of the different variables, resulting in **81** different designs.

Under **Constraints**, the displacement plot from the static study is used. The study wants to find a design that has less than **2 mm** displacement. The constraint is what the solver sees as most important, the number one goal. The scenarios that does not fulfill this demand will be "failed" designs. Under goals, different goals of the design study is put. These goals are less important than the constraints, and is only a secondary goal for the study. In this case, the secondary goal of the study is to minimize mass. The study will now find a design that will give less displacement than **2 mm** and chose the one with the least amount of mass as the optimal solution.

		Current	Initial	Optimal (39)	Scenario 22	Scenario 23	Scenario 24	Scenario 25
Height		190mm	190mm	240mm	140mm	190mm	240mm	140mm
Width		200mm	200mm	150mm	200mm	200mm	200mm	250mm
Thickness 1		10mm	10mm	10mm	15mm	15mm	15mm	15mm
Thickness 2		6.5mm	6.5mm	6.5mm	3.5mm	3.5mm	3.5mm	3.5mm
Displacement1	< 2mm	2.38204mm	2.38204mm	1.97243mm	3.66676mm	2.33871mm	1.68887mm	3.18828mm
Mass1	Minimize	43064.99184 g	43064.99184 g	37664.99184 g	53304.99184 g	54704.99184 g	56104.99184 g	65304.99184 g

Figure 4.7: Design study results

Figure 4.7 shows some of the scenarios from the design study. The optimal design was iteration 39, in green. The reason this is the optimal scenario is that it has the least amount of mass, while having displacement less than **2 mm**. The scenarios marked in red does not fulfill the constraint, and thus is a failed design. Marked in white are the scenarios that fulfill the constraint, but are not the optimal design due to more mass.

As the results show, the new optimal design has less mass and less displacement than the initial design. The difference in the initial design and the optimal design is shown in **Figure 4.8**.

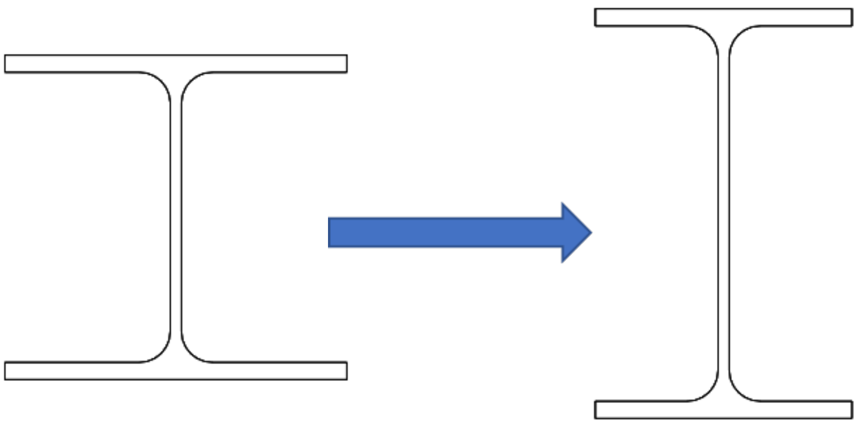


Figure 4.8: Old design compared to new design

This exact example should be tested with less variable interval steps, but due to having so many variables the number of scenarios will increase heavily. Having one extra step in each variable will result in **256** different scenarios. For studies of this calibre, an idea is to do one rough study for the initial study and then add more design studies with less interval space for the variable based on the best result. This may result in an even more optimized structure.

4.5 General Topology Optimization Method

As previously mentioned, SolidWorks by Dassault Systèmes has been chosen as the main CAD package. The TO in SolidWorks relies on SIMP as its main TO solver, but there are other independent plugins available.

The TO study is dependent on the CAD and FEA functions. They work together to produce designs that are optimized within the given boundaries. The process can be broken down into the six steps shown in **Figure 4.9**. The figure is in part inspired by "State of the art of generative design and topology optimization and potential research needs" by Tyflopoulos et al. [17]

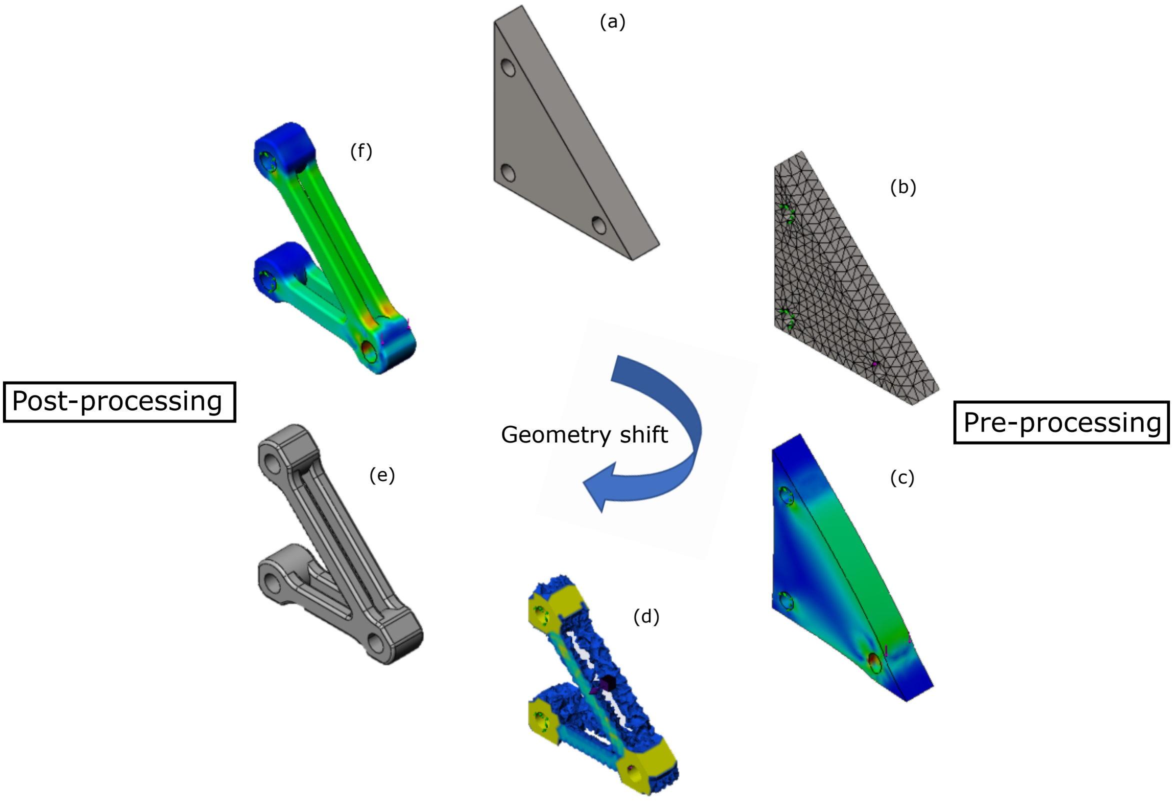


Figure 4.9: The geometry shift model of a three hole bracket with SolidWorks

(a) Cad geometry	(d) TO results
(b) FEA model	(e) TO remodel
(c) FEA results	(f) FEA of TO remodel

The process used for topology optimization in Solidworks is shown in **Figure 4.9 (a - f)**. The process starts off with a CAD geometry. This geometry is then turned into a FEA model, by defining fixtures, loads, material and mesh. When this is done, the first simulation of the model is initialized. The result is then used as base for next step, the TO. The TO result is displayed in (d), and is used as basis for the remodel. The different colours indicates where material can be removed. The plot is shown in **Figure 4.10**. By using a redesign-method, an altered model is created, shown in (e). As a last step, the re-design is tested with FEA to validate that this model can withstand the loads it was designed for.

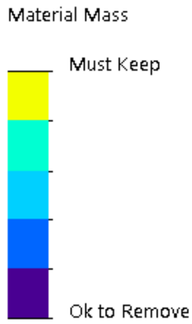


Figure 4.10: TO chart showing material removal

It is also important to define the problem correctly. As with FEA, if the problem is defined and set up incorrectly, the results will reflect this. Using the tools available will help with defining the problem to ensure better results.

4.5.1 Topology Optimization Tools

There are several tools available to help refine and improve the results of the TO study. The most critical tools are the ones in the **Goals and Constraints** tab. In this tab you can specify what kind of optimization is preferred. SolidWorks offers these alternatives:

- Best stiffness to Weight ratio
- Minimized maximum displacement
- Minimized mass with displacement constraint

The **Best stiffness to Weight ratio** will try to optimize the part in a way that gets the best stiffness to weight ratio. The **Minimized maximum displacement** option will optimize the part to achieve the least displacement possible. The **Minimized mass with displacement constraint** will minimize the mass primarily but with a maximum displacement input. All of these have their uses and scenarios.

The next important tab is the **Manufacturing controls**. In here are the tools to help ensure that the TO results are usable and not too unrealistic. SolidWorks offers four tools here

- Preserved region
- Thickness control
- De-mold direction
- Symmetry plane

The **Preserved region** tools is used to designate some surfaces, faces and points completely off limits to the TO process. This means they will not be altered in any way. This is very useful when there are some critical parts to a design that can not be allowed to change. The depth of the protected surface can also be specified.

Thickness control is a tool used to ensure a certain thickness in the part. Here either a maximum thickness or a minimum thickness can be specified.

De-mold direction is directly connected to casting as the production method. As parts have to be cast, they also have to be removed from the mold, that means they often need geometry that lends itself to casting as a production method. This tools allows one to specify the de-mold direction, either mid-plane (both directions), pull direction only and stamping (pull direction only)

The final tool is the **Symmetry plane**, it is used to maintain symmetry in a part. This can be very useful with regards to design, because the results may not distribute the mass symmetrical if the algorithm is left to itself. With this tool three kinds of symmetry can be specified:

- Half symmetry - the part is split in half around a plane
- Quarter symmetry - the part is split into four identical bodies around two planes
- One-eighth symmetry - the part is split into eight identical bodies around three planes.

4.5.2 Use of Other Topology Optimization Software

In the following chapter, software utilizing different TO methods will be compared against each other. The approach for each individual program will be explained in detail and the results will be compared. This is put into its own chapter, to give a more structured overview, as the scope of this comparison is fairly deep.

5 Software testing

To compare different methods for topology optimization, a few different programs have been tested. The procedure used in these programs will also be discussed in detail. The goal of this comparison is to get data on the different methods. Solver time, user interface (UI) and results are compared. The three different TO methods utilized were SIMP, BESO and LSM. TO will also be pitted against LO to see how they compare.

A simple model was created to compare the TO, and simulated in each software. The model is a simple fixed beam with a pressure load distributed at the end of the beam. The dimensions of the beam is **200 mm*100 mm*10 mm** with the load of 4000 N distributed over **400 mm^2**. The material used for all the different tests are alloy steel with a yield strength of 540 MPa.

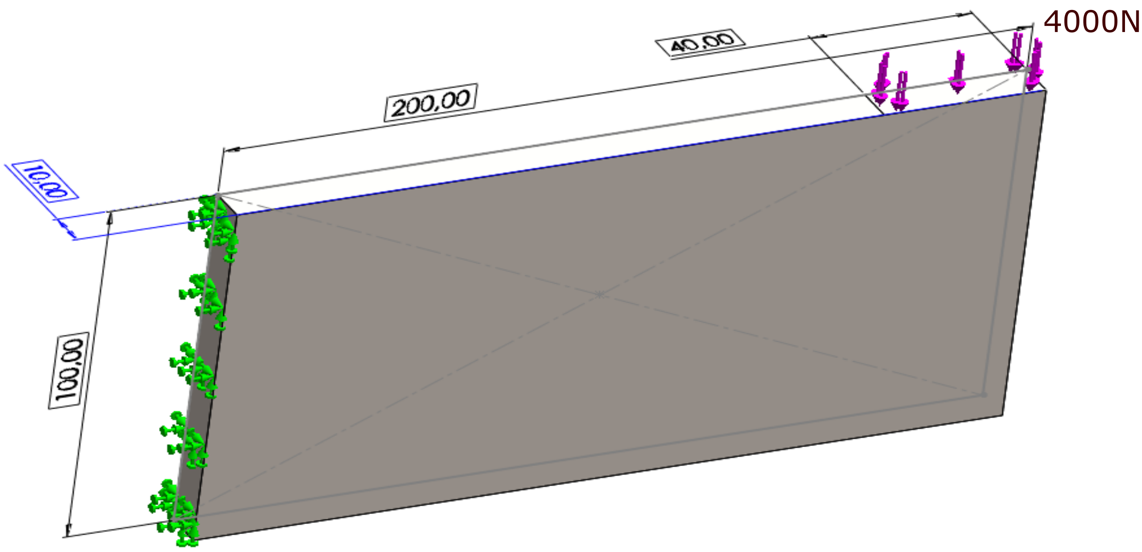


Figure 5.1: Test-model
Model from SolidWorks

5.1 Solid Isotropic Microstructure with Penalization

For SIMP, the integrated TO tool in SolidWorks was used. It is recommended to run a study, before a TO study. This is done to ensure that the loads applied, do not result in stresses beyond the yield strength of the material used.

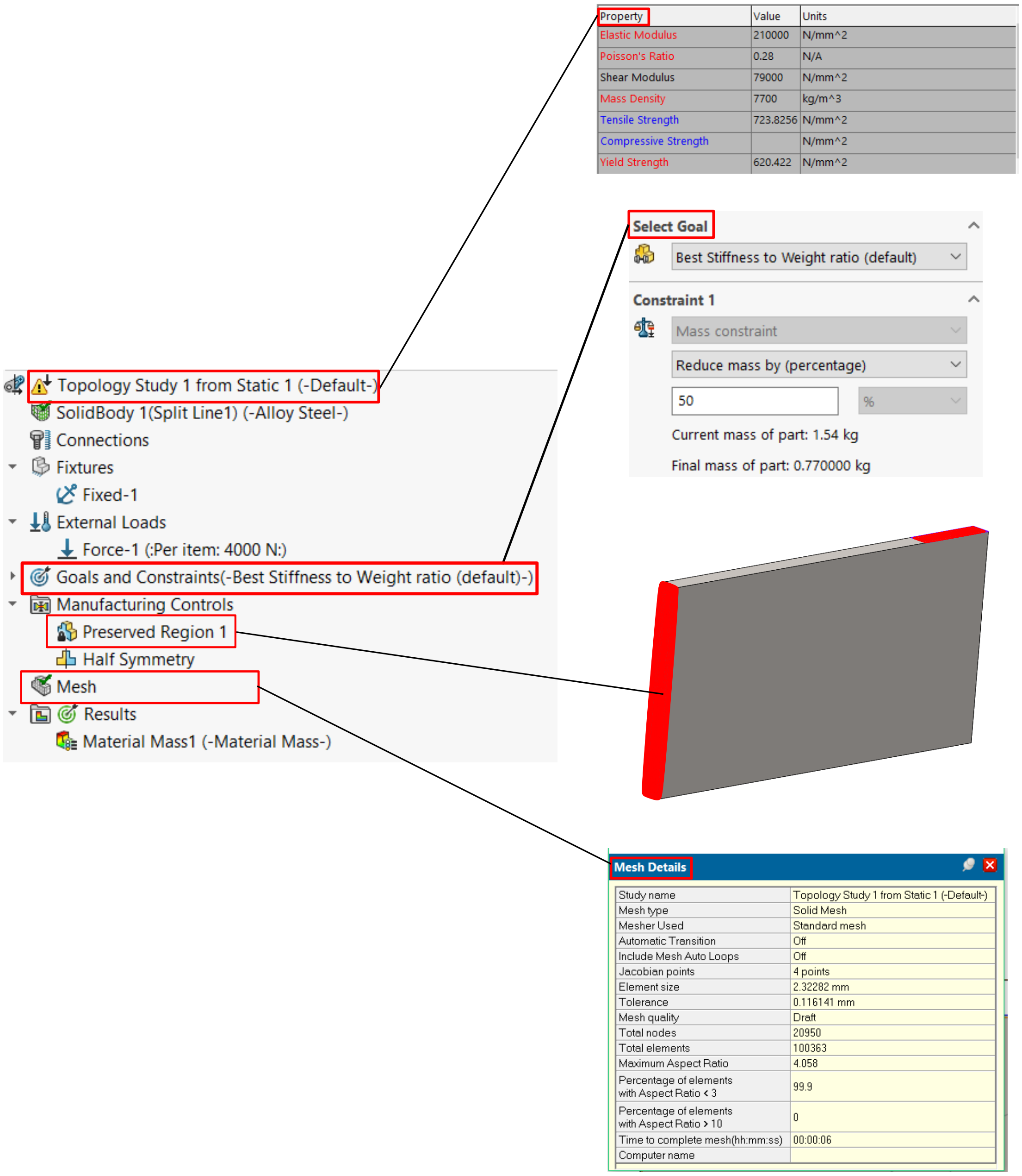


Figure 5.2: TO study settings

As shown in **Figure 5.2** the setup for a TO study is really similar to a static study in SolidWorks in regards to the material selection, mesh generation, fixtures and applied load. The **Goals and Constraints** setting chosen was **Best Stiffness to Weight ratio**. Two preserved regions were also added, where the part is fixed and where the load is applied. In addition a symmetry plane was added to get a better result from the study. The other settings were taken directly from the static study.

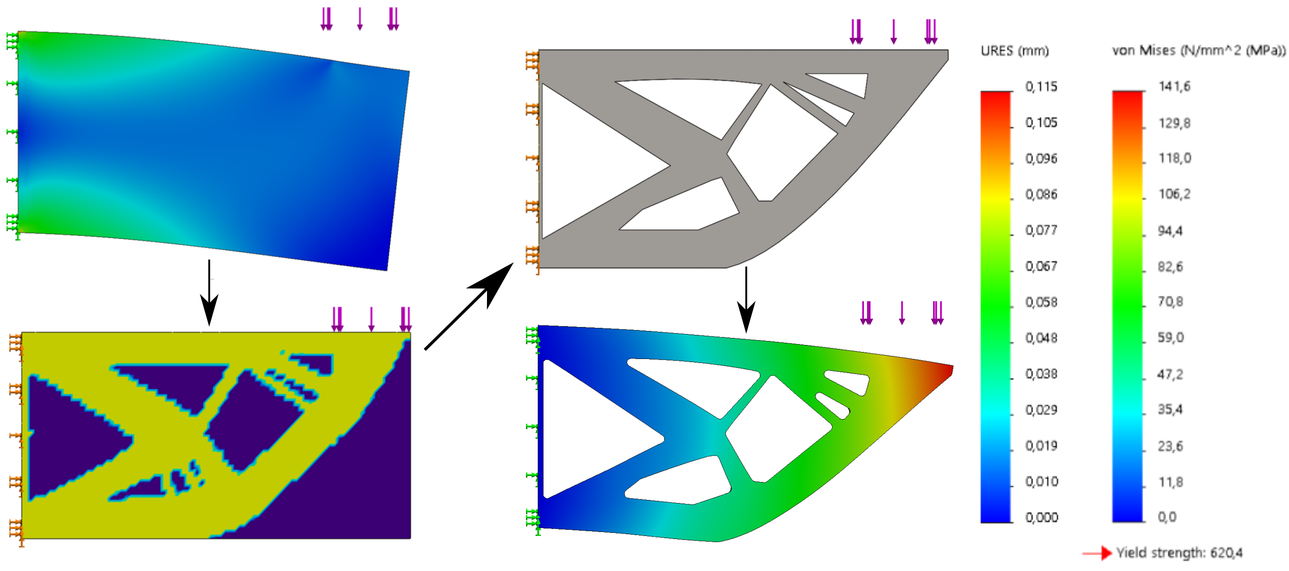


Figure 5.3: Process

Study name	Topology Study 1 from Static 1
No.of Nodes	20950
No. of elements	100363
Total Solution time	00:17:42.869000

Figure 5.4: TO solver time

The results of the SIMP TO study is shown in **Figure 5.3** and **Figure 5.4**. It took the solver 17 minutes and 43 seconds to solve this study. The solver time mainly depends on computing power and mesh size, but is recorded for the comparison.

5.2 Bidirectional Evolutionary Structural Optimization

The BESO method was tested with the 2D program **Beso2D**, the plugin **Ameba** for the CAD program **Rhinoceros** and a python script for the CAD program **Abaqus**. Some of the programs were problematic to use and for this reason, several programs had to be tested.

5.2.1 Beso2D

The first program tested was the 2D program, Beso2D. This program was the starting point for testing with the BESO method. The 2D program was utilized to establish a base before moving on to 3D programs. The program itself was first presented by Huang and Xie in 2010 and can be found on the sites of *Royal Melbourne Institute of Technology* (RMIT) [30][31].

The program itself has a rather simple UI design and is intuitive to use. It has some options to create simple 2D geometry and generate the mesh. With only two fixture options and two load options this program has limited use, but is useful to visualize and simulate simple problems. **Figure 5.5** shows the UI layout and options available.

The program functions similarly to other 3D CAD programs with regards to setup and problem defining. Fixtures, loads and material properties have to be defined as usual. Then a FEA study is conducted to provide a base for the BESO study, but it is not required. The available plots from the FEA are **von Mises stresses** and a geometric representation of the displacement, but not an actual plot of the displacement.

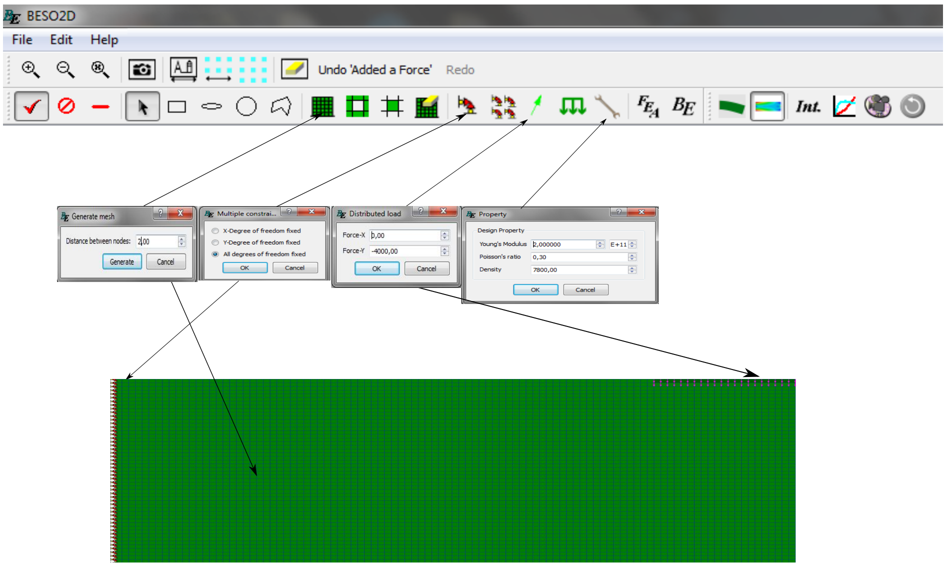


Figure 5.5: Beso2D setup

Figure 5.6 shows the BESO parameters that can be altered in the study. The first two are evolutionary parameters which affect the result from iteration to iteration. The three at the bottom are factors that alter when the algorithm will converge on a solution.

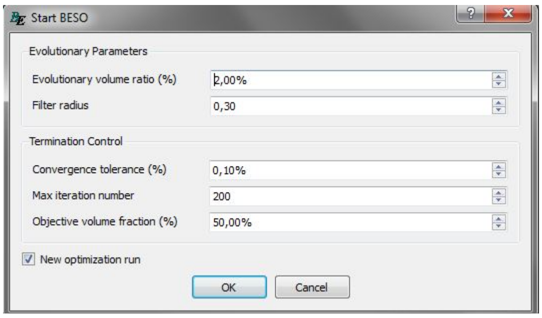


Figure 5.6: Beso2D TO-Options

Figure 5.7 shows the FEA results of the beam before the BESO algorithm was applies. **Figure 5.8** shows the beam after the TO process was completed.

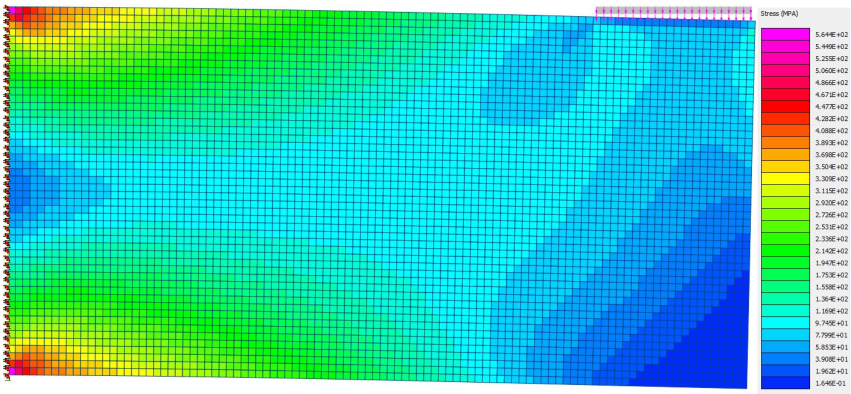


Figure 5.7: FEA static study result from Beso2D

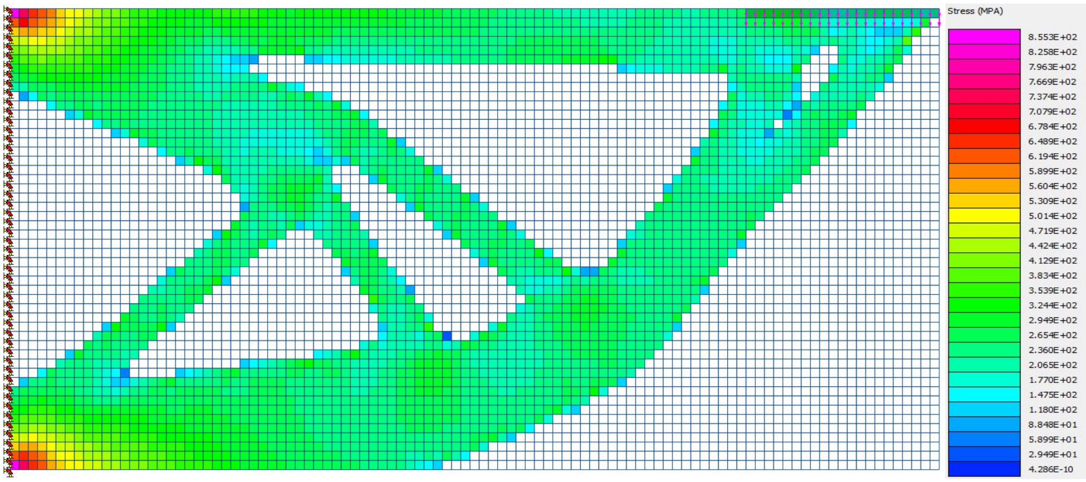


Figure 5.8: BESO result from Beso2D

The Beso2D program was useful as a soft start and provided valuable insight into the BESO optimization, but a more complex program was required for the 3D simulations.

5.2.2 Rhinoceros/Ameba

The first 3D BESO program tested was a plugin for the CAD program Rhinoceros 3D called Ameba, which operates using a visual programming language knows as Grasshopper. The group had no previous experience with any sort of programming, but with access to online guides and other resources it became possible to test Ameba [32][33].

The company that develops Ameba also provides various documentation, including a user manual and several example files. Unfortunately there was a problem with the software. When generating a mesh for the design geometry the software would only display an error, see **Figure 5.9**. During the troubleshooting process several of the example files were tested and resulted in the same error. Other user complaints with the same error were also found. The developer was contacted and provided several possible solutions, but non worked. Due to these problems, the program was abandoned.

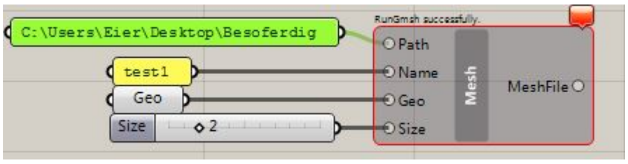


Figure 5.9: Error message in Ameba "input string was not in a correct format"

5.2.3 Python Script for Abaqus

The next 3D BESO program that was tested was a script for the CAD program Abaqus [34]. The script was presented by Zhi Hao Zuo in an article in 2015. It is also available freely at the RMIT's site [31][35].

The script uses the CAD model to set all the parameters. This means that there is no need to alter the script itself, as the problem could be defined within Abaqus.

However, when running the script, another issue was encountered, see **Figure 5.10**. The error turned out the be related to the location of the work directory in Abaqus. The work directory needs to be set to the folder were the model is saved in order to function.

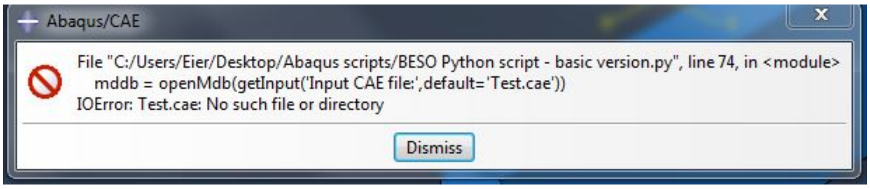


Figure 5.10: Error message when trying to run script when the file is not in the work directory

With a fully defined model and the work directory properly configured the script functions as intended. One important thing to note is that the input file needs to be in .CAE format, contain a model named "Model-1" with a dependant part called "Part-1" and a static step called "Step-1".

The BESO paramaeters that can be changed are the volume fraction, filter radius and evolutionary rate. The next step is to select the input file previously mentioned.

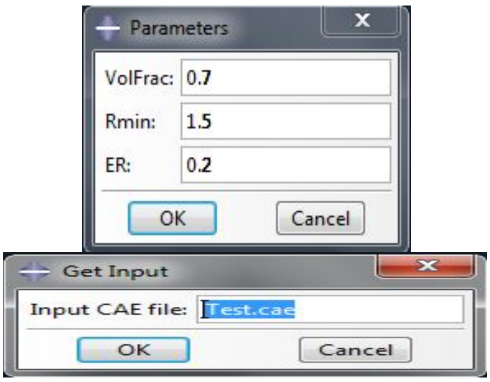


Figure 5.11: Input parameters script

After this, the script will run the optimization process until it reaches a convergence. How many iterations it will take for program to reach a convergence depends on the input variables. When it reaches convergence it will create a new file called "final design.cae". A visual representation of this design does not automatically show up in Abaqus, so another script has be utilized. This script only requires one parameter, the input file, see **Figure 5.12**

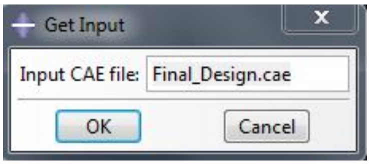


Figure 5.12: Input parameters visualization-script

The test model was tested using the input parameters specified in **Table 5.1**. The mesh utilized had 30000 elements, as it was restricted by computational power. The script ran through 42 iterations and resulted in the result shown in **Figure 5.13**

Volume Factor	0.5
Filter Radius	4mm
Evolutionary rate	0.02

Table 5.1: BESO Input Parameters

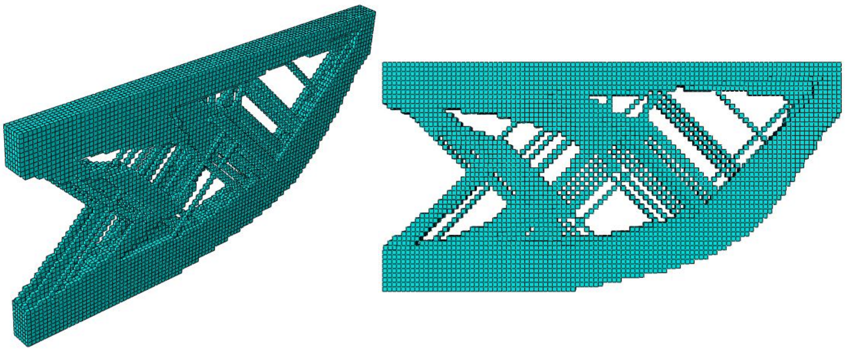


Figure 5.13: BESO Result
Model from Abaqus

The script produces a geometric mesh, but does not produce any result plots. The solution implemented was to remake the result in a known CAD program to run a static study on it as this would yield displacement and stress plots. **Figure 5.14** shows the remade model in SolidWorks.

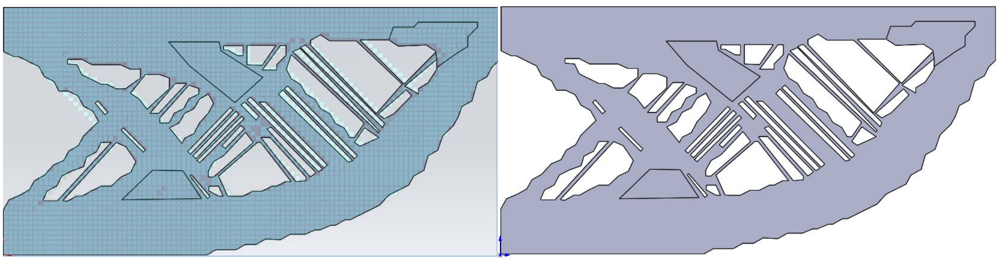


Figure 5.14: Model in Solidworks of BESOmesh

The study yielded a max displacement of **0.095 mm** and a max von Mises stress of 178 MPa as shown in **Figure 5.13**.

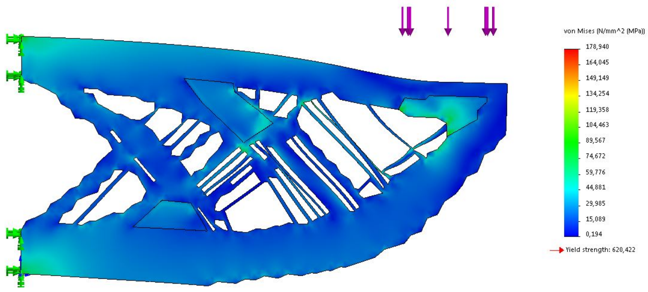


Figure 5.15: Result plot BESO3D
Model from Solidworks

5.2.4 Challenges with Abaqus

When testing Abaqus, several obstacles had to be overcome. It turned out Abaqus operated without specifying units. It relies on unit relations instead of fixed outputs. If the length of a part is defined in millimeters and the force is in Newton, the output will display mass in tonnes and stresses in MPa. Likewise, if the millimeters swapped with meters the mass will be outputted in kilogram (Kg) and stresses in pascal (Pa). See **Table 5.2** for the units used. Abaqus also requires material properties to be defined manually since there is no material library.

Quantity	SI	SI(mm)	SI	US Unit(ft)	US Unit(inch)
Length	m	mm	m	ft	in
Force	N	N	kN	lbf	lbf
Mass	kg	$tonne$ $(10^3 kg)$	$tonne$	$slug$	$lbf\ s^2/in$
Time	s	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	kPa	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3}J)	KJ	$ftlbf$	$inlbf$
Density	kg/m^3	$tonne/mm^3$	$tonne/m^3$	$slug/ft^3$	lbf s^2/in^4

Table 5.2: Units in Abaqus

When running the script, a hard limit on mesh elements was found. It is unclear if this limit it caused by a lack of computational power or some kind of script error. In the testing 5000 elements was the maximum amount of elements, any more and the script refused to initialize. Even with 5000 elements the study took two hours and 25 minutes to initialize and run. A significant increase when compared to other TO methods. The source article mentioned a solution which would increase the computational efficiency [35]. This required editing of the script code, and implemented **"Numpy scripting"**. After this the script could run 30.000 elements, but it did however take the same amount of time as previous runs.

As previously mentioned, the script does not produce any deformation or stress plots. The solution consisted of rebuilding the optimized result in a known CAD program. There are significant issues with this method as it is a difficult and time consuming process. All attempts to export a model directly from the results ended in non-viable files that could not be used. The mass of the rebuilt design did not meet the criteria set for the TO study as the redesigning process proved difficult. Mass properties of the rebuilt design can be seen in **Figure 5.16**

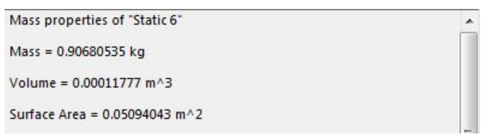


Figure 5.16: Mass-properties of BESO model in Solidworks

5.3 Level Set Method

Finding software that utilized the LSM also turned out to be a challenge. In the search for usable programs, a Matlab script was found. The script however only worked for 2D structures. It is called "levelset88" and was published by Otomori et al. in 2015. The other program used was "ParetoWorks", a 3D plugin created by SciArt and works with SolidWorks.

5.3.1 Levelset88

The code was presented in the paper “Matlab code for a level set-based topology optimization method using a reaction diffusion equation” by Otomori et al. It is important to note that the code is for educational use only.

The code consists of 88 lines and uses 200 iterations to solve a problem.

```
levelset88 (1).m
% Matlab code for topology optimization using a reaction diffusion equation
function [str,phi] = levelset88(nelx,nely,Vmax,tau)
%% Parameter definition
E0 = 1;
Emin = 1e-4;
nu = 0.3;
nvol = 100;
dt = 0.1;
d = -0.02;
p = 4;
phi = ones((nely+1)*(nelx+1),1);
str = ones(nely,nelx);
volInit = sum(str(:))/(nelx*nely);
```

Figure 5.17: Matlab code preview

Figure 5.17 shows a preview of a small section of the code. To use the code, four variables have to be altered to fit whatever scenario is presented, see **Table 5.3**.

nelx	elements in the x-direction
nely	elements in the y-direction
Vmax	maximum volume
tau	regulation parameter

Table 5.3: Matlab code variables

The code has a standard problem integrated, the problem is as rectangle, fixed on the left-hand side with a force applied downwards in the center of the right side, see **Figure 5.18**

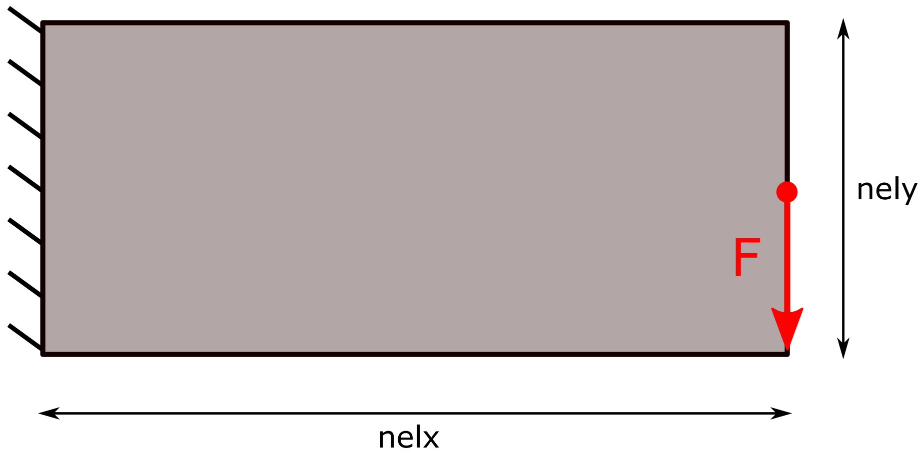


Figure 5.18: Standard problem

With lack of coding knowledge, it was difficult to change the parameters to the desired values. Therefore the standard problem integrated in the code had to be utilized despite it not matching the test beam. The results did not yield any stress or displacement plots and so is mostly irrelevant. **Figure 5.19** shows four different results, each with a different *"tau"* value. Because of the difficulties with changing the parameters and the lack of plots it was decided to move to the next program.

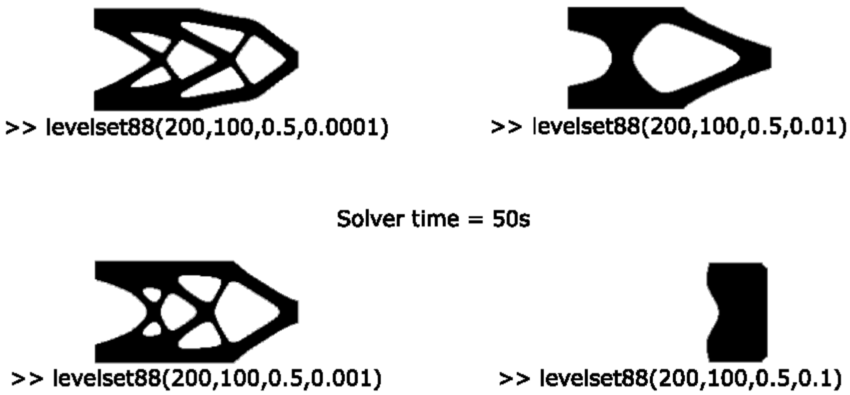


Figure 5.19: Different values for the regulation parameter

5.3.2 ParetoWorks

ParetoWorks is a 3D TO plugin for SolidWorks developed by SciArt Software,Inc. It was developed at the *University of Wisconsin Madison Engineering Representations and Simulation Lab* by the team at SciArt.

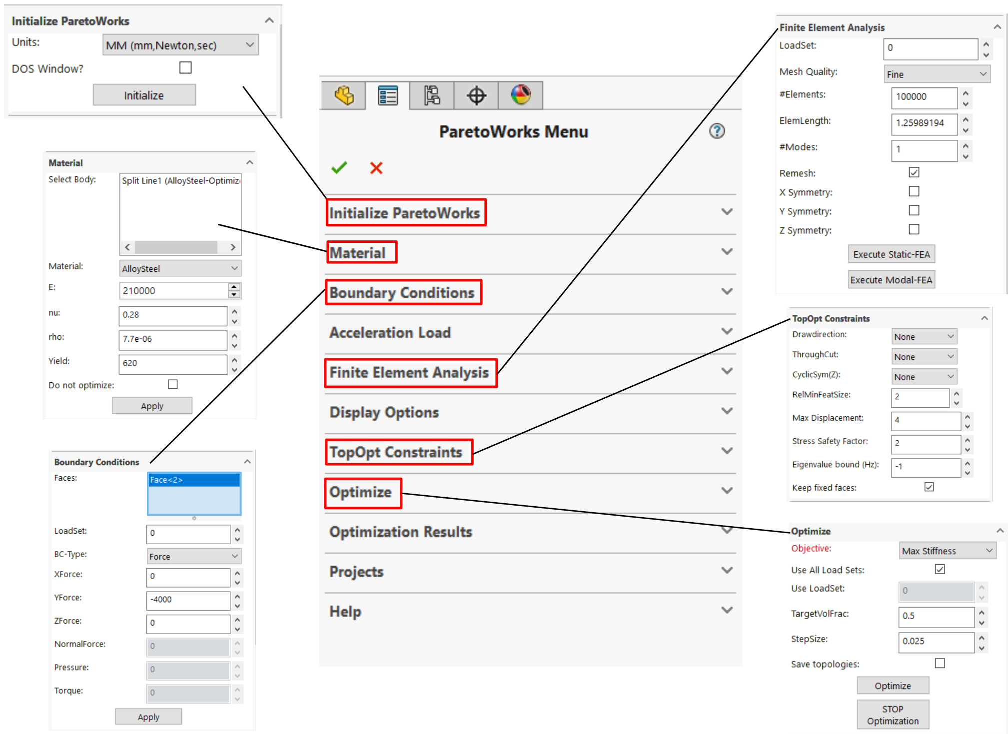


Figure 5.20: ParetoWorks setup

Figure 5.20 shows the settings available in ParetoWorks. The layout is similar to the built-in topology optimization function in SolidWorks. The following lists mention some of the different parameters available.

Optimize

- Min Volume
- Max Stiffness (Default)
- Max Strength
- Max 1st Eigenmode

TopOpt Constraints

- Drawdirection - If a part is going to be manufactured by casting or machining.
- ThroughCut - Cuts away material all the way through the part, in a selected direction
- CyclicSym(Z) - Creates cyclic symmetry
- Keep fixed faces

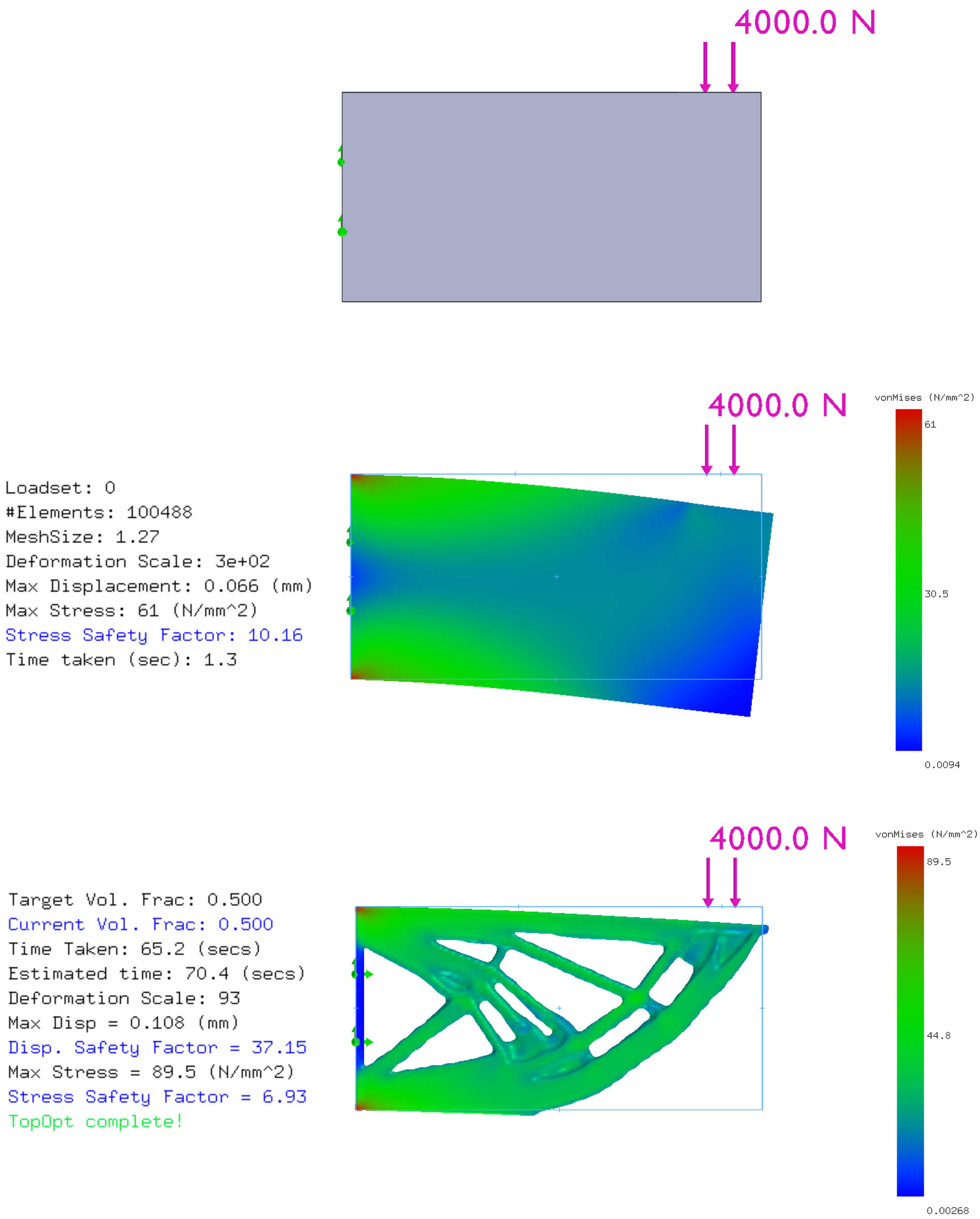


Figure 5.21: ParetoWorks results

ParetoWorks uses its own solver based on LSM and offers quick and good results. The plugin also runs a new FEA on the optimized result, offering plots for displacement, von Mises stresses, FOS etc. **Figure 5.21** shows the steps in the ParetoWorks process and then the optimized design. The solver is surprisingly quick and efficient as shown with the optimization only taking 65.2 seconds.

5.4 Topology Optimization Software Comparison

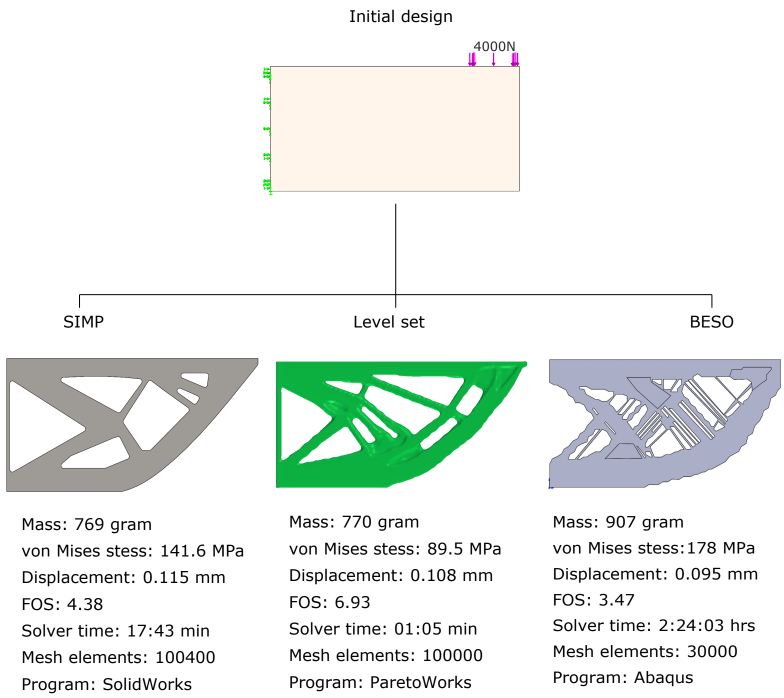


Figure 5.22: Comparison results

Figure 5.22 shows the results from the different methods used. The most notable about the results, was the solver time. ParetoWorks solved the problem in only about 1 minute, with the SIMP method taking over 17 minutes and the BESO script taking two and a half hours to complete. Visually, the SIMP and LSM models are clearly superior to the BESO result.

As for the other results, the displacement is pretty close for all the programs. The deviating results besides the solver time is the Mass and the von Mises stress. In regards to von Mises stress, the LSM is greatly ahead of the two others. Comparing the mass, SIMP and LSM are equal, but the BESO method falls significantly behind. This can be attributed to the remodeling step that had to taken.

Comparing the von Mises stresses in the different models, there are some surprising discrepancies. The validation of the SIMP method results in **141.6 MPa**, and LSM in **89.5 MPa**. This is a huge difference in two relatively similar models, which is suspicious. The validation process in ParetoWorks is automatic, and may have a flawed solver.

Figure 5.23 shows the rankings of the programs given by the group. It is based on experiences using the different software and rated accordingly. We acknowledge that the results of this is biased and only based on a limited experience working with them. First impressions matter when using such software, and the other criteria (Solver time and results) are based on numbers.

Program	Usability	Overview	Solver time	Results
SolidWorks	Excellent	Great	Medium	Good
ParetoWorks	Great	Good	Excellent	Great
Abaqus	Medium	Medium	Bad	Medium

5	Excellent
4	Great
3	Good
2	Medium
1	Bad

Figure 5.23: Comparison of the different software

SolidWorks

SolidWorks is overall a good program for TO. It is easy to use, the overview looks clean and there are many different online resources available. The solver time is not the greatest, and it is slow if the mesh is too fine.

ParetoWorks

ParetoWorks went above all expectations, mainly because of the quick solver time. The usability is great, but it took some time to get used to, because of limited information available about the program. Another advantage is that ParetoWorks automatically runs a static study after completing the TO study.

Abaqus

The BESO script for Abaqus would not be recommended for TO, unless you specifically want to use the BESO method. The TO process takes time to get comfortable with, and the literature behind it focuses primarily on the math behind the method and how the script is written, not how to use it. The script is also fairly computationally demanding, and the solver took about two and a half hours to finish with a descent mesh. There is also limited information on the static results behind the TO process and post processing is difficult. Lastly it is worth noting that this is a script made for research purposes and not a commercial program like the other tested programs.

5.5 Lattice Optimization

In some cases there are strict limitations when it comes to changing the shape of a part. In these cases it might be more advantages to use LO instead of TO. To compare the results from the LO, the TO test model was utilized. There are currently limited software that allows LO. The two programs tested was Ansys and Altair Inspire.

5.5.1 Ansys

Ansys for LO was recommended by the supervisor from NTNU, who also provided a guide [38]. The guide was used, but when it came to the validation process some problems occurred. **Figure 5.24** shows the project schematic for the LO setup. Three applications were used within Ansys to conduct the LO:

- Spaceclaim: A 3D modeling application
- Mechanical: A simulation application
- Material Designer: Used to create new materials

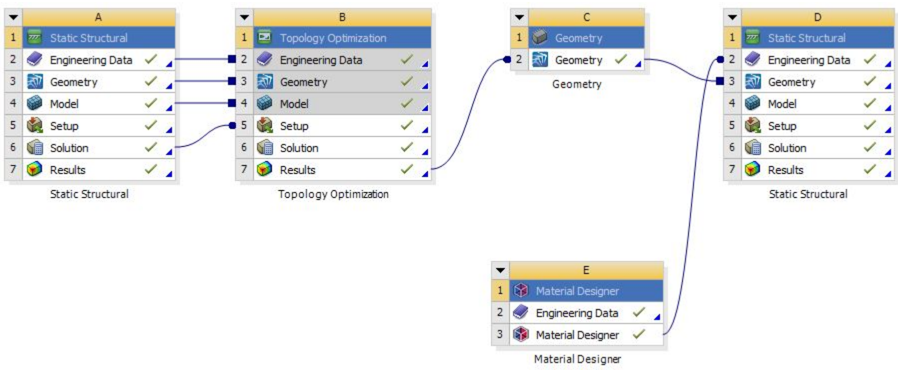


Figure 5.24: Project schematic of LO in Ansys

After running a static study of the model the results were transferred into a TO study where the optimization type was changed from TO to LO. The parameter for this study is pictured in **Figure 5.25**. These parameters are used to limit the computational strain of the optimization process.

Details of "Optimization Region (Lattice)"	
Design Region	
Scoping Method	Geometry Selection
Geometry	All Bodies
Exclusion Region	
Define By	Boundary Condition
Boundary Condition	All Boundary Conditions
Optimization Option	
Optimization Type	Lattice Optimization
Lattice Type	Cubic
Minimum Density	0,2
Maximum Density	0,6
Lattice Cell Size	10, mm

Figure 5.25: Parameters for LO-study in Ansys

The optimization process calculates the possible density of each lattice cell within the set density range and gives a density plot shown in **Figure 5.26**. For structural reasons the loaded and fixed surfaces is excluded from this optimization process and will have a density of one.

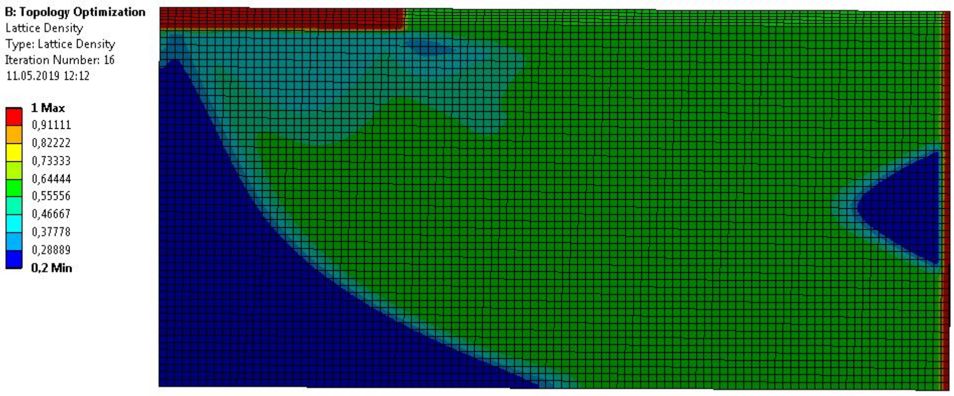


Figure 5.26: Lattice density created in LO-study
Model from Ansys

In order to apply lattice structure to the model, it needs to be transferred into Spaceclaim. The solid body is then selected from the configuration tree and the shell function is utilized. This is were the parameters of the lattice infill is determined. To use the density results from the LO study the **"use density attributes"** box needs to be selected. In **Figure 5.27** the parameters for the lattice infill is shown. The shape of the infill will auto select based on lattice type selected in **Figure 5.25**. It is important to not select any other shape as this is the shape the model is optimized for.

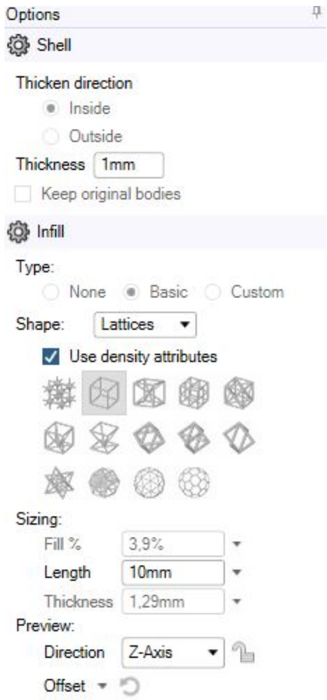


Figure 5.27: Lattice infill options in Spaceclaim

When researching the software an online resource recommended using the material designer to create a new homogenized material to save some time[39]. **Figure 5.28** shows the lattice structure of the material created with the material designer. This material is based on structural steel and shares the same young's module, Poisson ratio and shear module.

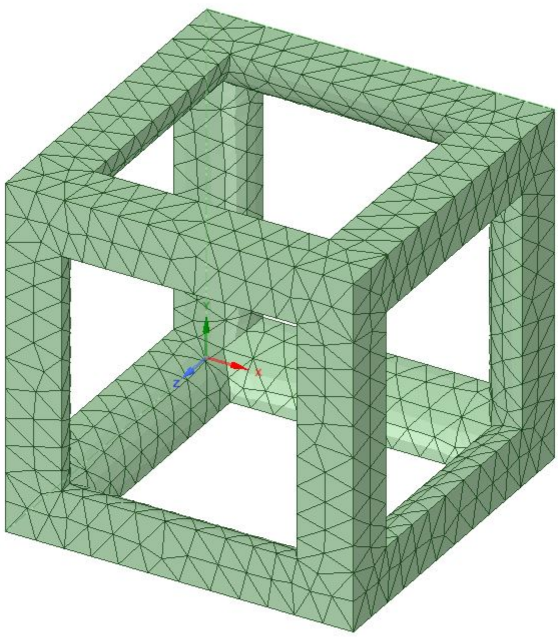


Figure 5.28: Lattice structure of new material from material designer

The first thing one needs to do in the validation study is to check if the creation of lattice structure in Spaceclaim was successful. Using the section plane tool it is possible to create a cross section of the model to check the internal structure. **Figure 5.29** shows a section view of the model.

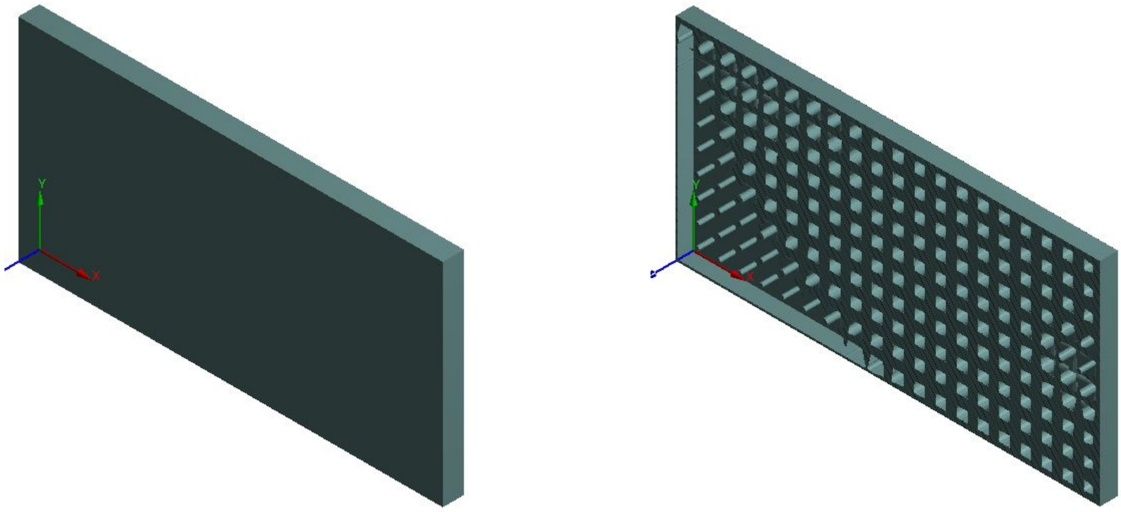


Figure 5.29: Section view of lattice structure in Ansys

After confirming a successful lattice infill the validation process is set up similar to a normal static structural study. There are however some parameters to take into consideration due to the complex structure of the model. With the model use of cartesian meshing was specified, but when researching it was found that tetrahedronic and patch independent mesh could be used. With the cartesian method it is important to make sure the mesh is fine enough to cover the entire model. The mesh used had an element size of 1mm.

In the validation study it is not possible to select faces or edges to apply fixtures or loads. The solution to this is to create named selection of nodes, which can be time-consuming on larger models. There is an option to lasso pick or box select nodes, but in this case it ended up selecting the internal nodes as well. In **Figure 5.30** the setup of the model can be seen. The fixtures used was a normal fixed support and the load was a Direct FE nodal force.

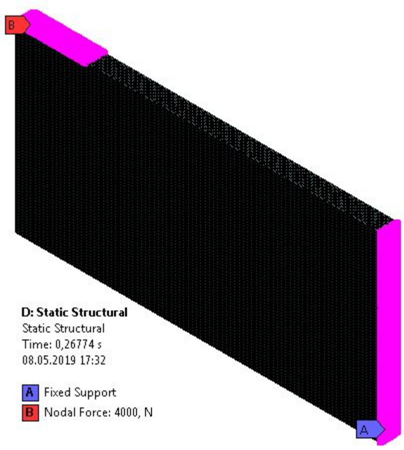


Figure 5.30: Setup of validation study showing loads and fixtures
Model from Ansys

Figure 5.31 displays the total deformation and von Mises stresses of the validation study. A maximum displacement of **0,17mm** and maximum von Mises stress of **335 MPa** was achieved. The high stress on some of these nodes might be a result of the method used for fixtures. If the nodes with the high stresses shown in **Figure 5.32** are excluded, the results are comparable with the results from the TO-studies. **Figure 5.33** shows the weight oft he model after LO.

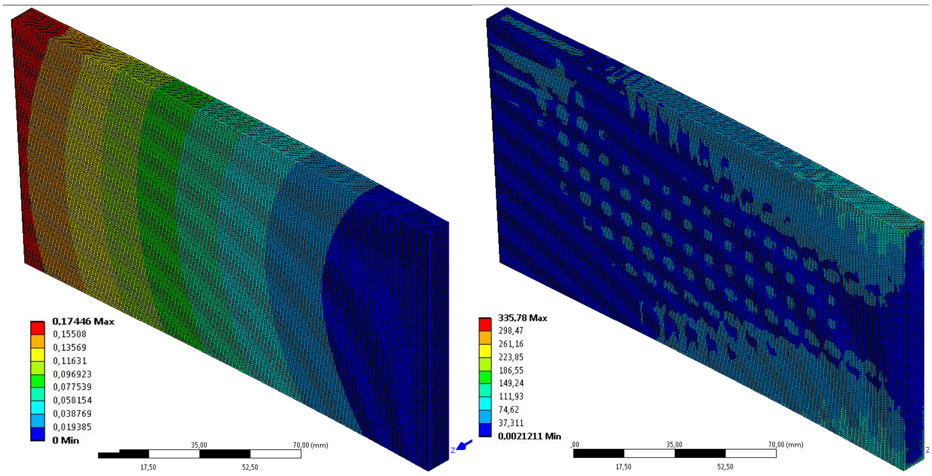


Figure 5.31: Von Mises stress and deformation from lattice validation study
Model from Ansys

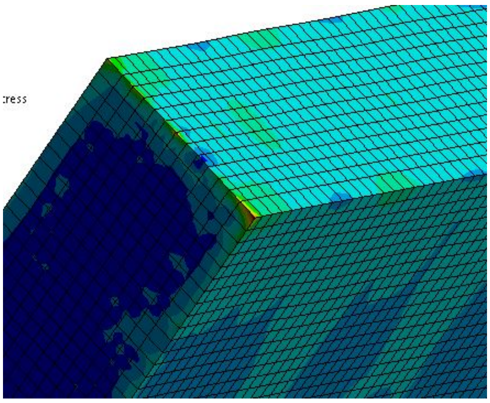
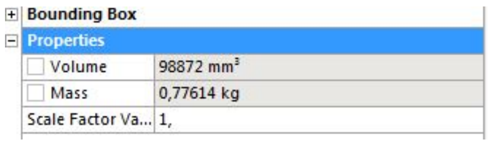


Figure 5.32: High-stress nodes
Model from Ansys



Bounding Box	
Properties	
Volume	98872 mm³
Mass	0,77614 kg
Scale Factor Va...	1,

Figure 5.33: Weight of model after LO in Ansys

Using Ansys for LO turned out to be time-consuming and dependent on a lot of computational power. A possible inaccuracy can come from the selection of nodes for the fixtures and loads. In the model, the fixture selection contains almost 5000 nodes and the load has about 2000. Seeing as there are so many nodes, it is possible that some were missed in the selection phase.

5.5.2 Altair Inspire

The way LO is implemented in this software makes it faster and easier to use than Ansys. After designing a geometry and applying boundary conditions, one can go directly into the LO. The LO study applies the lattice structure and validates the results in the same process, so there is no need for an additional validation study.

The program automatically creates an open lattice structure, where Ansys created an internal lattice structure. To optimize with an internal lattice structure, a partition of all external surfaces need to be created. The program does not allow for changes to be made to the shape of the lattice structure, so the standard diagonal structure had to be used.

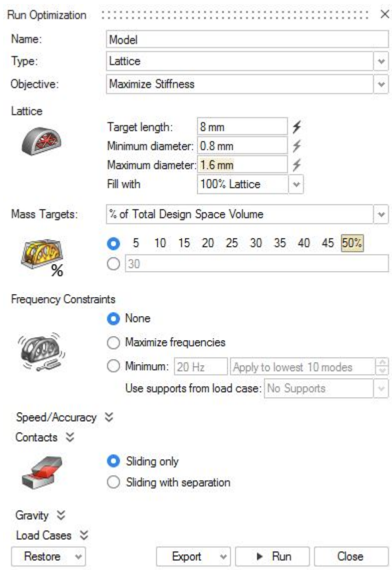


Figure 5.34: LO options for Altair Inspire

Figure 5.34 displays the parameters that were used for this study. These parameters yielded the lattice structure shown in **Figure 5.35**, which is notably finer than in Ansys. The process of creating and validating this structure took far less time than in Ansys even though the lattice cell size is smaller.

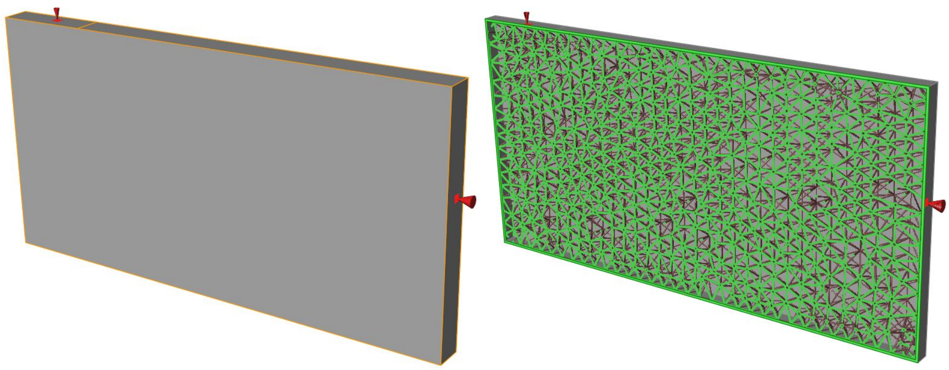


Figure 5.35: Section view of lattice structure in Altair Inspire

Figure 5.36 displays the results from the validation of the TO-study. A maximum von Mises stress of **177 MPa** and a maximum displacement of **0,287 mm** was achieved.

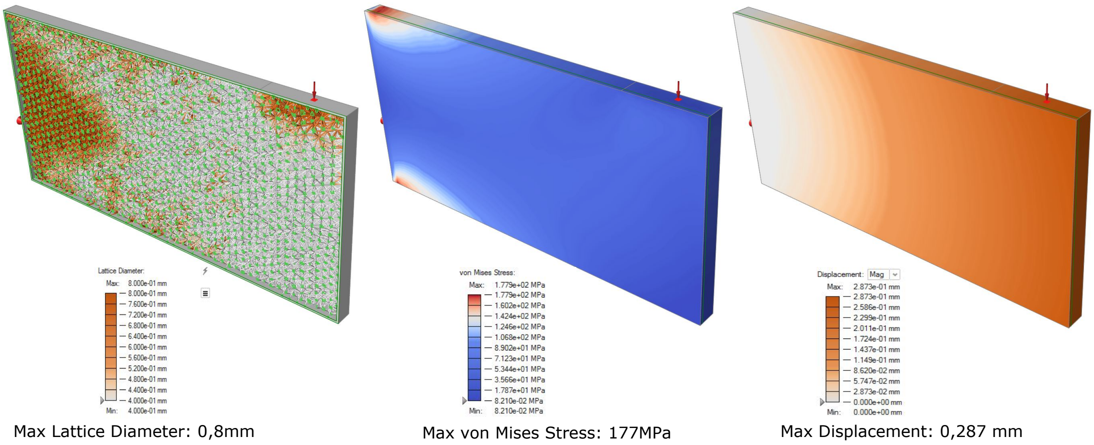


Figure 5.36: Validation result of LO in Altair Inspire

As pictured in **Figure 5.34** the mass target was set to 50%, but when looking at mass properties for the optimized design displayed in **Figure 5.37**, it resulted in 5% of the initial mass. After discussing this with the groups counselor, who also got the same error, the decision was made to not investigate this further.

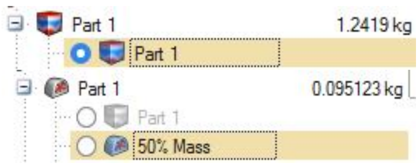


Figure 5.37: Mass discrepancy with LO in Altair Inspire

5.6 Lattice Optimization Compared to Topology Optimization

Ansys presented a lesser result on von Mises stresses and displacement compared to the tested TO methods, but the results might have been penalized due to computational limitations. For instance, the only mesh option which could successfully be applied was a uniform Cartesian mesh. Using a multizone mesh might yield a different result. With a lattice cell size of **10 mm*10 mm*10 mm** in a design space of **200 mm*100 mm*10 mm** there is only possible to create 200 lattice cells, which probably is less than the optimum. Using a different geometry for the lattice cells might also yield a different result.

The results from Altair Inspire are questionable as the output weight did not meet the input criteria, and will thus not be taken in to consideration further on in the thesis. However, if the results are to be trusted they are astonishing. It yielded a result with deformation and von Mises stresses similar to the TO methods, but with 10 times less mass. If this is correct, this program greatly outperformed the others.

The software testing provided a useful insight on the use of different algorithms for the TO-process, but the raw output from the TO study often produces a complex geometry. The next part of the thesis will focus on improving the raw geometry into a design more suited for manufacturing.

6 Post-Processing

The post-processing (PP) of TO is normally one of the last steps before the design can be finalized. PP consists of simplifying and redesigning the TO results, then validation that redesign. **Figure 6.1 (e)(f)** illustrates these steps.

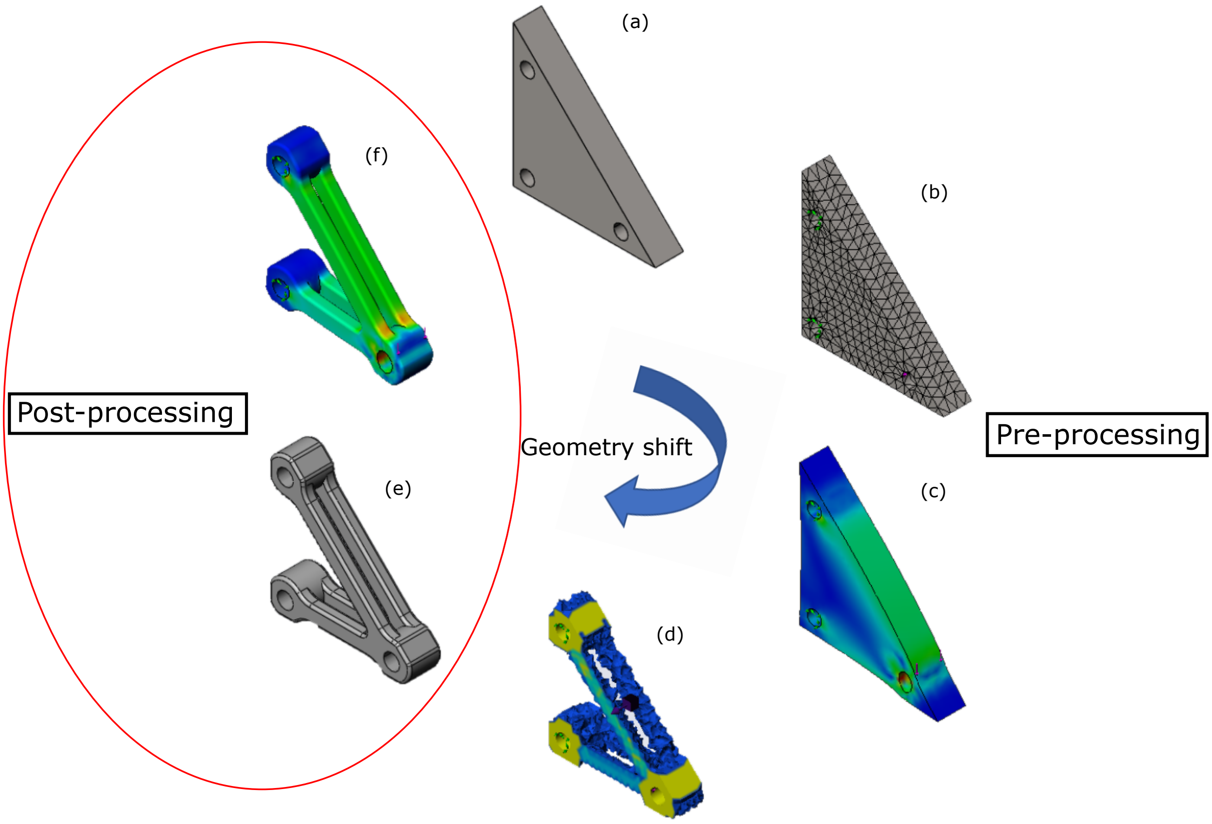


Figure 6.1: Current step in the TO process

To showcase some different approaches to PP, several models were chosen as examples. Ranging from relatively simple to more complex. It is important to note that the TO results are up for interpretation. There are certain things to take into consideration, such as manufacturing. Sometimes the results from the TO are not viable due to the increased cost of producing such complex geometries. In the end it is up to the engineer to analyze and interpreter the results so that they fit the requirements.

6.1 Methods

Some different TO PP methods are showcased and explained in detail to show the range of TO results and work strategies. There are many ways to do PP, these examples are just some of the methods discovered. All models in this chapter are taken from SolidWorks, unless otherwise specified.

6.1.1 Converting Mesh to Solid

To showcase some available options, a simple bracket is utilized. It is fixed in the top hole and a remote load is applied to the bottom three holes, see **Figure 6.2**. The load is 100,000 N and the material used is alloy steel.

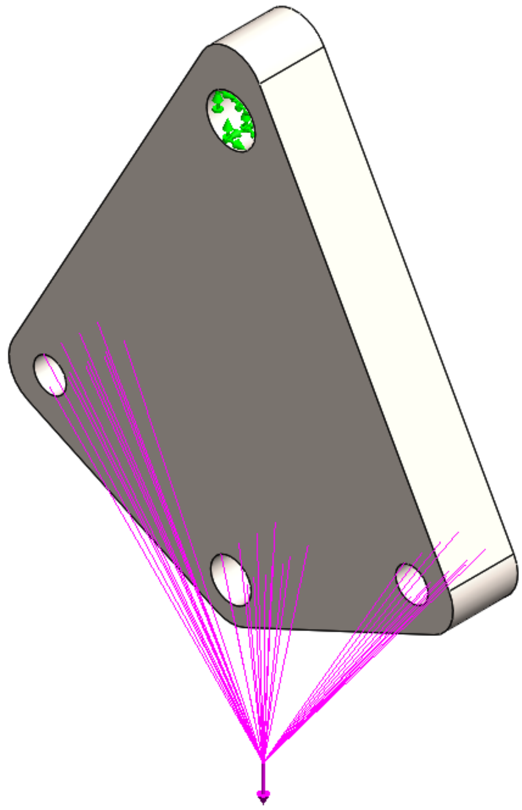


Figure 6.2: Simple bracket model with fixtures and loads

The TO study provides several rough plots of stresses and displacements in the optimized design. These plots are only rough due to the nature of the optimized part, as it can now contain elements of reduced density that are not realistic. That means the plots will inherently not be precise at this stage, but they do however give a rough estimate of the stresses, stress concentrations and the displacement. **Figure 6.3** shows the plots from the TO study.

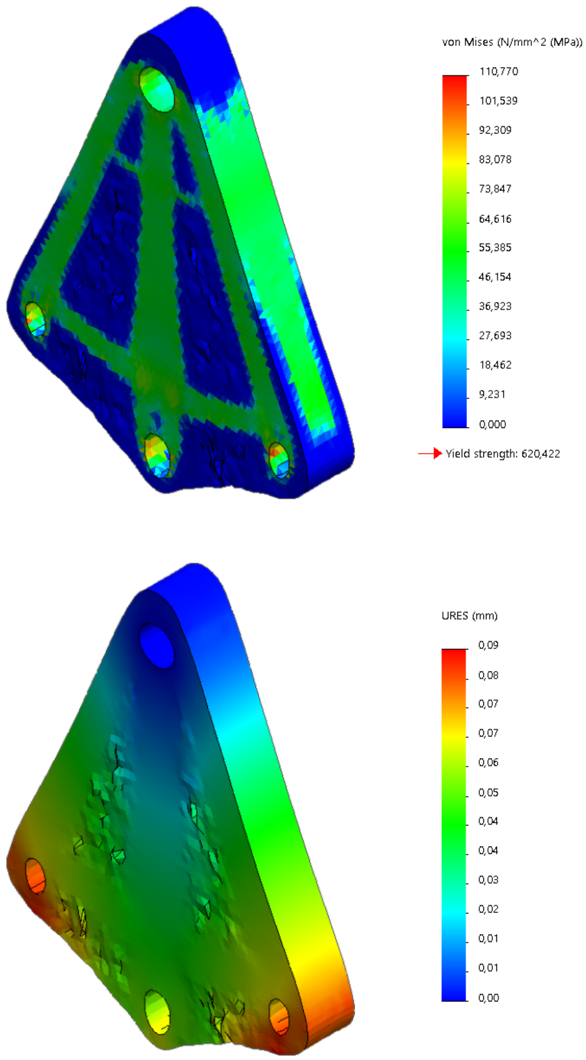


Figure 6.3: Rough plots from the TO study

Another useful tool is the smooth mesh process. This smooth mesh can be exported as a solid body, surface body or as a graphic body. The graphic body option is a less computational heavy mesh that is very useful when modifying designs.

If exported as a solid body, it will be treated by the software as a normal part, and can also be manipulated as normal. The exported body can sometimes be rough and contain geometry that traditional production methods can not produce. If however RP or AM is used, and the optimized design meets all the requirements, it can be directly sent to manufacturing.

The process of validating the smooth mesh body is currently difficult, as the part now consists of many small triangles that make FEA studies hard to complete, see **Figure 6.5**. **Figure 6.4** shows the original model up against the optimized results and then the exported smooth mesh.

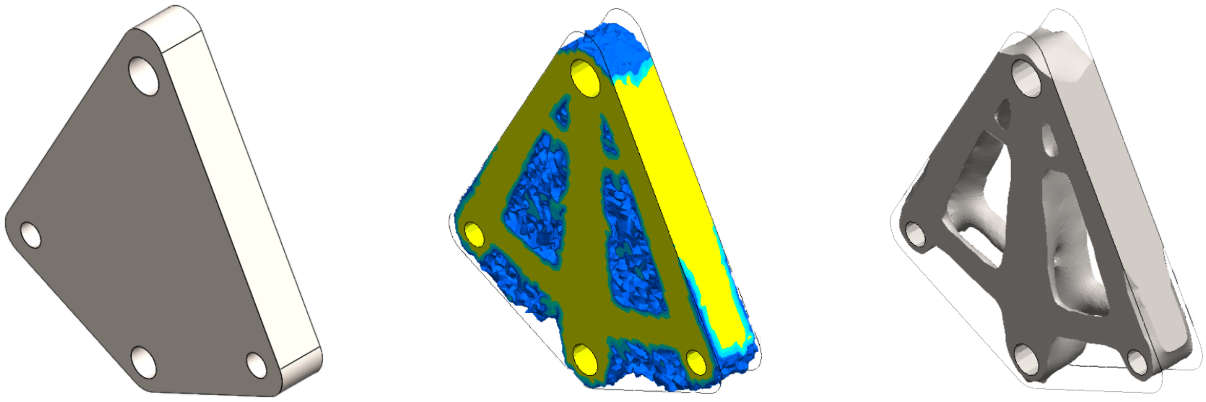


Figure 6.4: Original design, TO results and smooth mesh

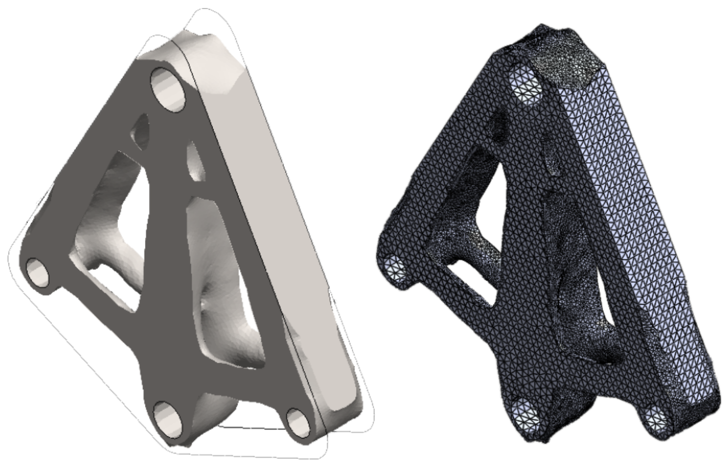


Figure 6.5: Rendered smooth mesh and exported smooth mesh

As exporting and validating the smooth mesh in SolidWorks was difficult, Ansys Workbench was also tested. The process of exporting a smooth mesh was simple and Ansys provides several tools to help prepare the part for validation.

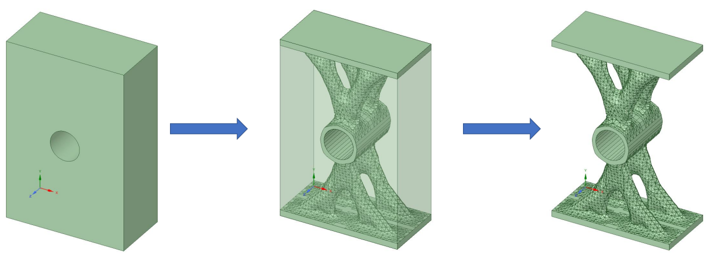


Figure 6.6: Ansys smooth mesh converted to solid

Figure 6.6 shows the conversion of a TO study result where 90 % of the mass has removed and converted to a new solid body. In **Figure 6.7** the setup for the validation study of the solid body is shown.

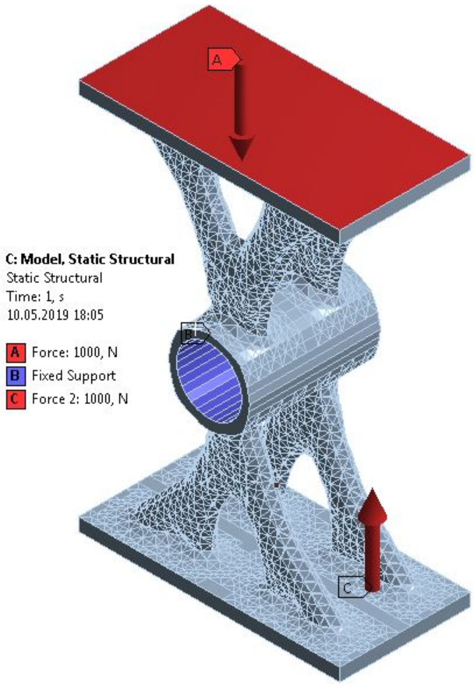


Figure 6.7: Validation setup for the smooth mesh converted to solid
Model from Ansys

The result are shown in **Figure 6.8**. The stresses are higher after the TO process, but still has a FOS of about three. This validation method can be very useful in many scenarios. Instead of manually redesigning the part, the automatic exporting can save a lot of time. When several different iterations of TO results are tested, this method can increase the efficiency of the validation process.

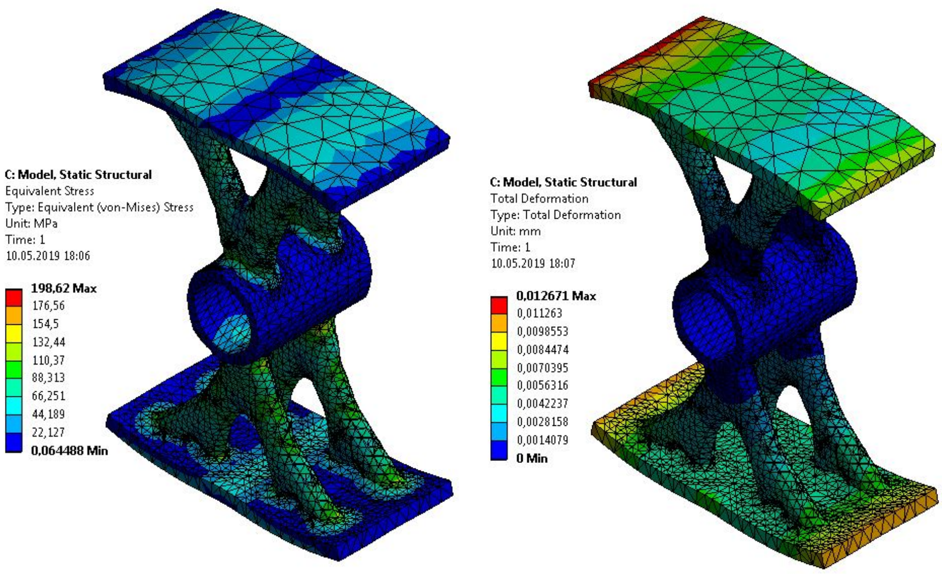


Figure 6.8: Result of the smooth mesh converted to solid
Model from Ansys

6.1.2 Extruded Cut

The next method is very easy and straightforward. When a TO study results in holes, those holes can simply be drawn on the original design and then cut away. It can be implemented on thinner objects, such as plates. This method was found using online resources.[40]

As **Figure 6.9** shows, the example used for this method is a thin structure with fixed point in one hole and a remote load in the other hole. The figure has gone through the pre-processing part of the procedure, and is ready for the post-processing.

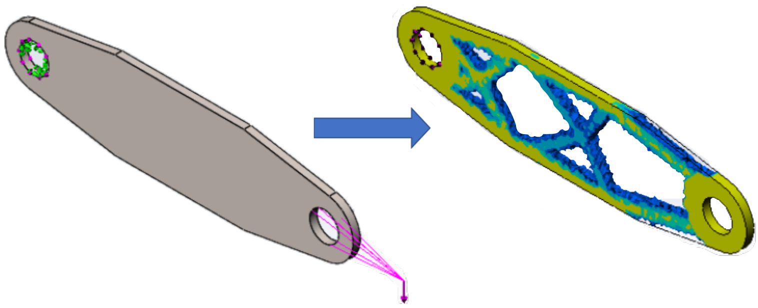


Figure 6.9: Pre-processing of simple example

As this is a thin plate, all that is needed is to overlay the simulation results on the original model. SolidWorks has its own tool for this, called simulation display. This allows the user too easily alter the original model with the context of the simulation results available

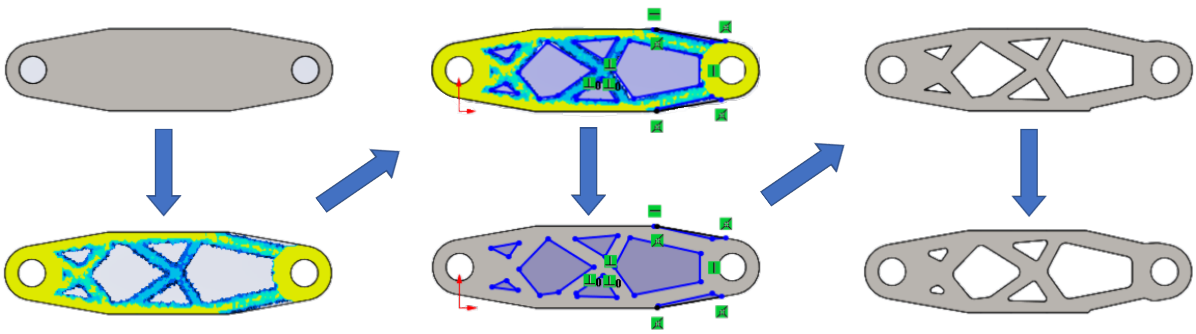


Figure 6.10: Post-processing of simple example

Figure 6.10 shows how the model changes from each step. The first step shows the original design. The simulation results are then overlaid using the simulation display tool. Then, in step three, the sketching process begins. Here, the overlaid holes are sketched directly onto the original design. Step four shows the sketches on the design with the simulation display turned off, this can be done to give a better overview of where the cuts are supposed to go and if any are missing. In the next step, the sketches are cut out. The last step is to add some fillets and chamfers if they were not added during the sketching process. This is to avoid stress concentrations and to ease the production process. When the model is finished, it is ready for the validation step where the optimized results and the original can be compared

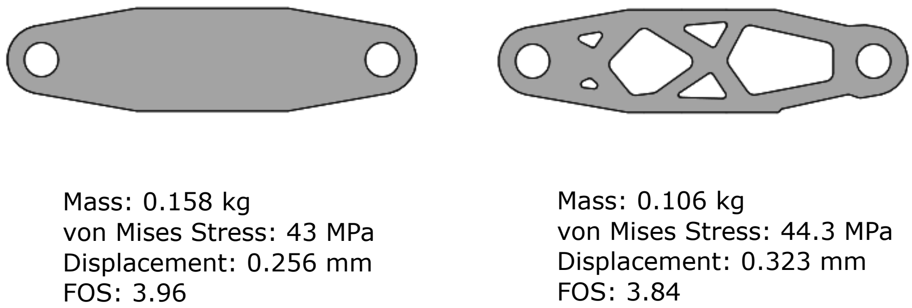


Figure 6.11: Comparison of the initial design and the updated design

As **Figure 6.11** shows, one can see the improvement in weight. The new design is 33% lighter. The displacement has increased some, but the FOS is about equal.

6.1.3 Lofted Cut

The next example is a variant of the method explained in the previous **Chapter 6.1.2**. The example model is a simple beam, it is fixed in one end, and has a force directed down on the other edge. During the pre-processing phase, the external surfaces of the beam was set to be protected, see **Figure 6.12**.

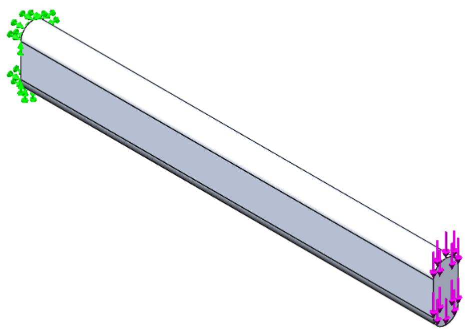


Figure 6.12: Fixing point and force on beam

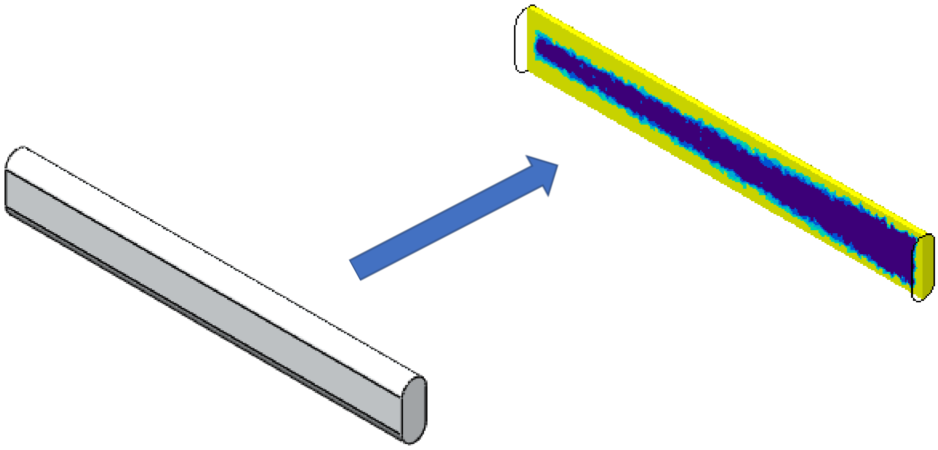


Figure 6.13: Pre-processing of beam with protected surfaces.

Figure 6.13 shows the original beam, and the results from the TO study. The TO result is clipped to see the interior of the beam, as there are no changes to its exterior. Now, to post-process this result, the original beam can still be used and modified where it is needed. In this case, a perfect way to do so is by using the tool called, **Lofted cut**.

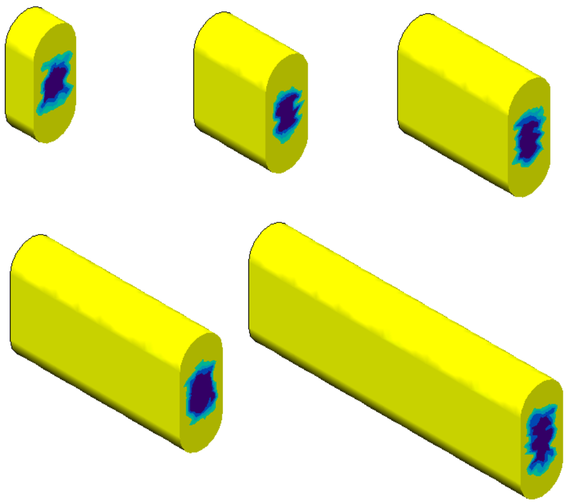


Figure 6.14: Different section clippings of beam

As **Figure 6.14** shows, the hole from the study is not uniform. If the exact results from the TO study were to be used, it would be a hard task to produce this type of beam. An idea is to simplify the shape of the cut in the PP, to get a finer finish. By creating new planes parallel to the section clippings, the simulation display can be used to sketch out the new cuts.

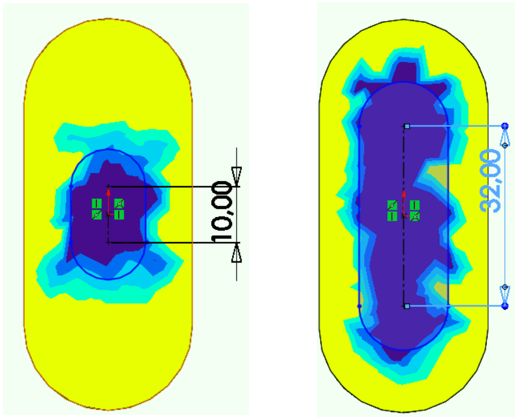


Figure 6.15: Sketches for lofted cut

Using this method some of the yellow material, labeled as **must keep** will be removed. This can be crucial, but is up the the engineer to decide. In this case some of the material labeled **ok to remove** has been retained to make up for it.

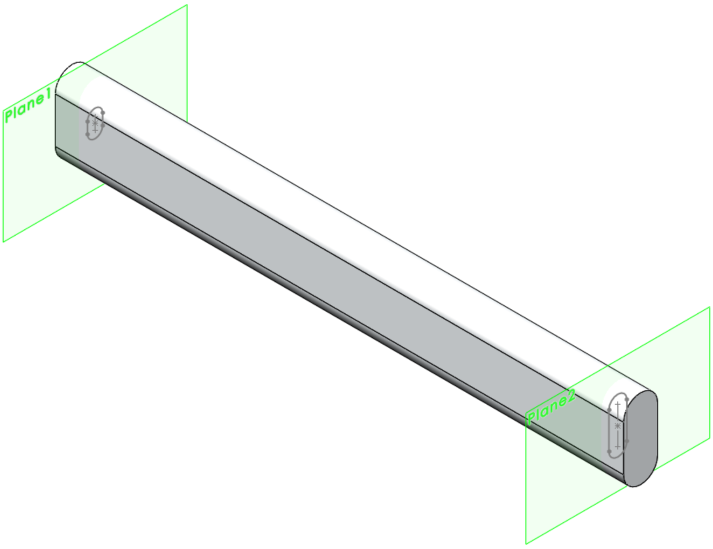


Figure 6.16: Overview over the planes and sketches for lofted cut

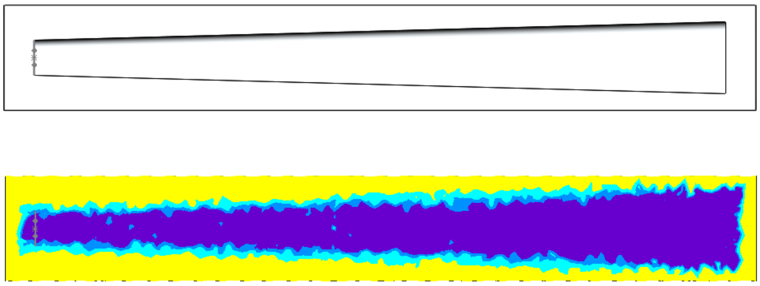


Figure 6.17: Section view of finished model and TO results

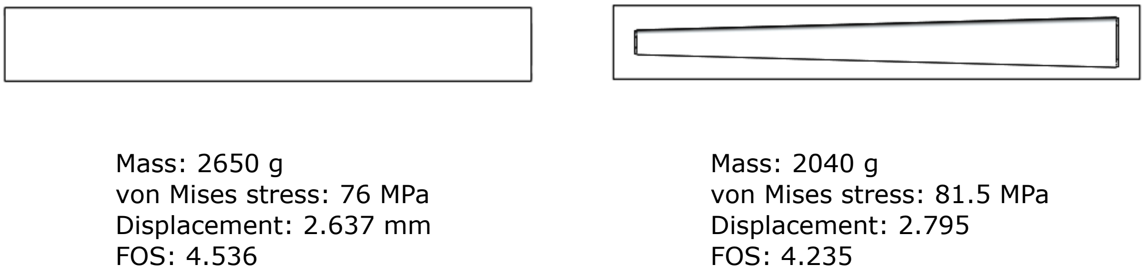


Figure 6.18: Comparison of old and new model

The goal for this TO study, was to reduce the mass by 30%. The final product had an reduction of 23%. The results from the TO study was only used as an indication, and not directly transferred. This can explain the deviation of the mass reduction.

6.1.4 Rapid Validation

In some instances, several TO studies are executed to achieve the desired result. If the design allows it, a method for rapidly checking through different options is to use the design study tool. For this example, the case from **Chapter 6.1.3** is continued. The beam is now set to have displacement less **3 mm**, and should be as light as possible. Instead of going through many different TO studies, an idea is to utilize the design study tool. Then, different values for the lofted cut can be set to test many iterations of the beam.

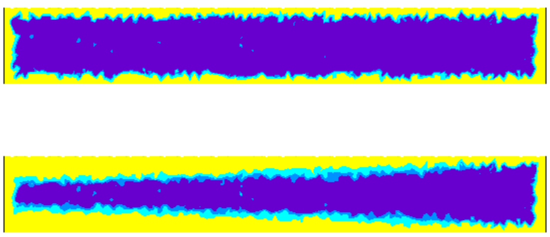


Figure 6.19: Difference between two TO results

As shown in figure **Figure 6.19** another TO study was executed to discover where additional mass could be removed. The new TO result indicates where the design can be altered, which gives the requirements for the design study.

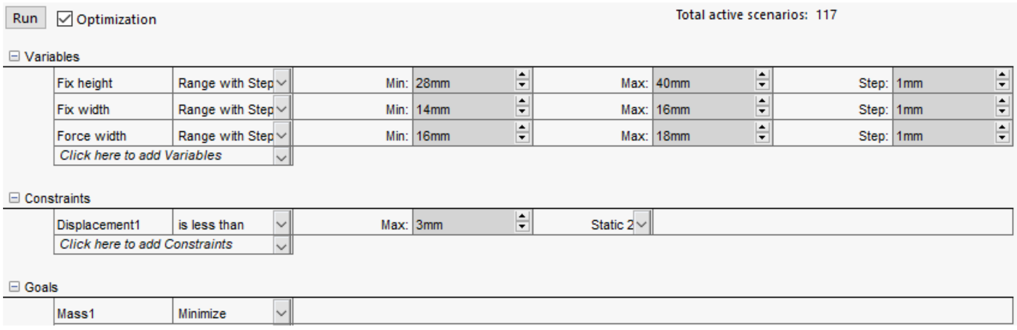
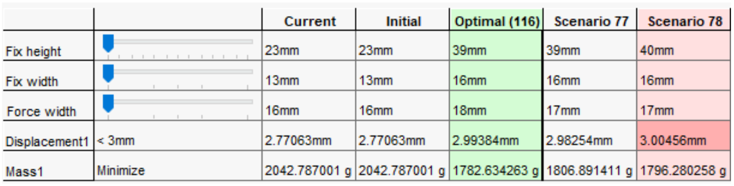


Figure 6.20: Design study input to rapidly check alterations

The input variables were the width and the height of the fixed end, and the width of the opposite end. By going through the **117** different scenarios, all variations of the lofted cut are tested. The best cut for the set restraints are then chosen.



		Current	Initial	Optimal (116)	Scenario 77	Scenario 78
Fix height		23mm	23mm	39mm	39mm	40mm
Fix width		13mm	13mm	16mm	16mm	16mm
Force width		16mm	16mm	18mm	17mm	17mm
Displacement1	< 3mm	2.77063mm	2.77063mm	2.99384mm	2.98254mm	3.00456mm
Mass1	Minimize	2042.787001 g	2042.787001 g	1782.634263 g	1806.891411 g	1796.280258 g

Figure 6.21: Design study input to rapidly check alterations

After the optimal beam is found, it is compared to the initial design. An additional **257.4 g** of mass is removed, with a low increase in stress and displacement.

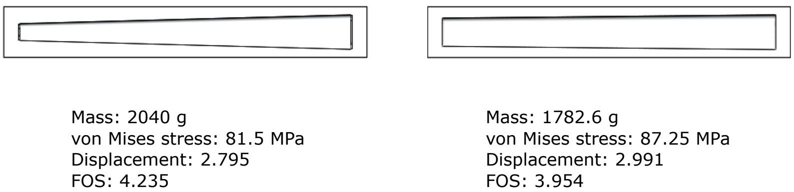


Figure 6.22: Difference in results of lofted cut studies

6.1.5 Rebuild for Manufacturing

When presented with more complex parts the TO PP becomes more complex as well. In this example, **Figure 6.23**, a more advanced approach is utilized.

The four holes in the base plate are the fixture points. A remote load is applied to the two holes in the perpendicular plates. The load is centered between the two plates and **100 mm** from the bottom of the base plate

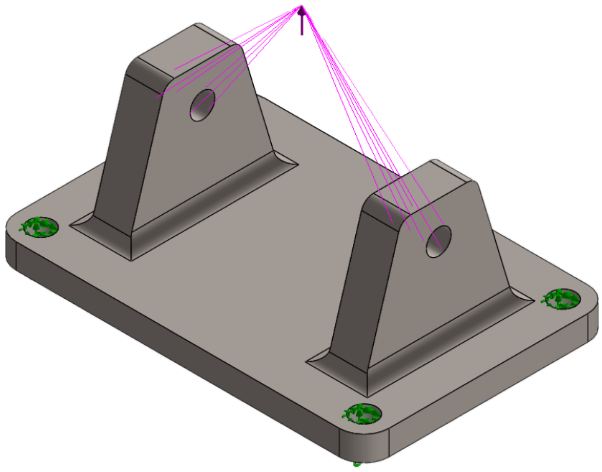


Figure 6.23: The fixtures and load applied to the bracket

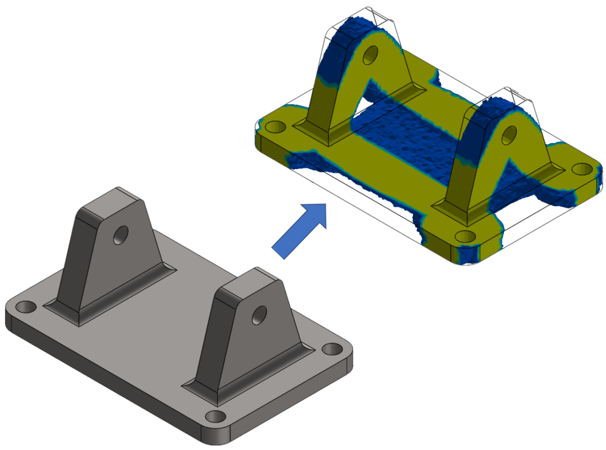


Figure 6.24: Pre-processing of bracket

On this part a combination of 2D cuts, overlaid simulation display and doing a complete redesign can be favourable. To begin with the part is put through FEA and TO, the result is then overlaid onto the original design and roughly cut out to give an idea on how the new design should be. By starting completely over at this point, the engineer can take the optimized design into consideration from the beginning of the design rather then modifying an existing design. Starting from scratch also gives the engineer an opportunity to ensure that the design is symmetrical and otherwise designed with the production process in mind. It is important to take the production method into consideration when designing and optimizing.

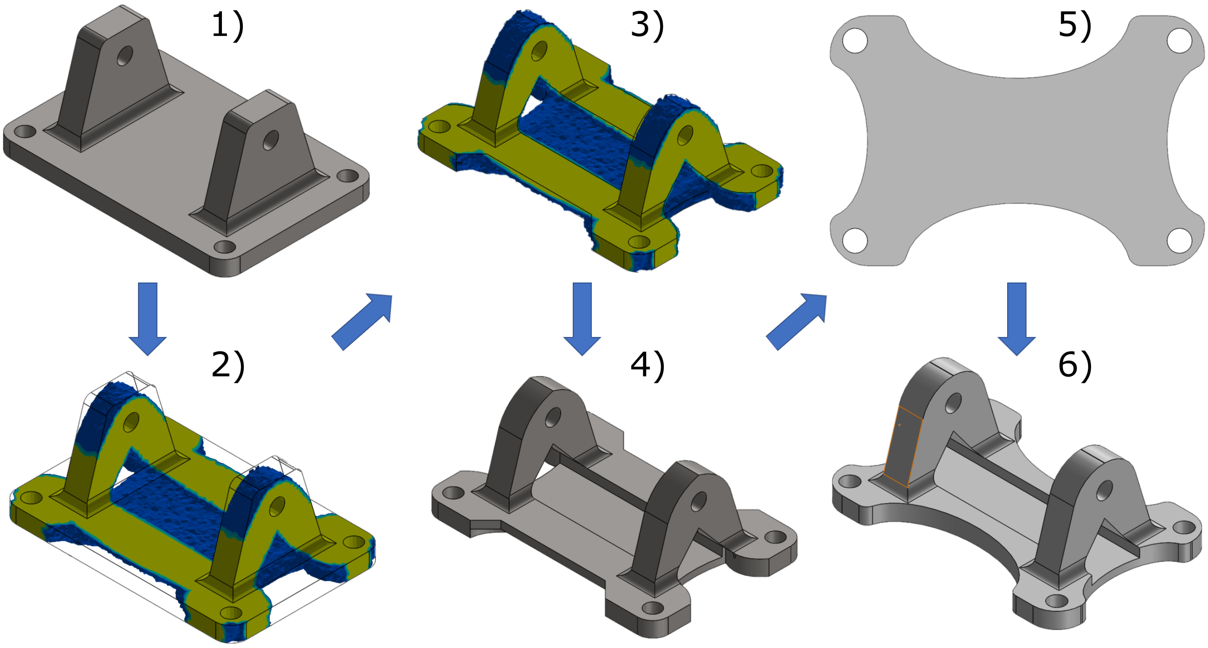


Figure 6.25: Post-processing of a bracket

Figure 6.25 shows the steps in the process. The first two steps are the original design and the results from the TO study. The results are then overlaid onto the original design and several 2D cuts are made to remove the material. Step four shows the rough cut out design. Using this rough design as a basis to refine the design from is optional. With the new design, production methods can be taken into account. Step five and six show how the new design differs from the rough one in step four. The differences are easy to see in **Figure 6.26**. The cutaway sections are more rounded, there are fewer sharp corners and in general more aimed towards ease of manufacturing

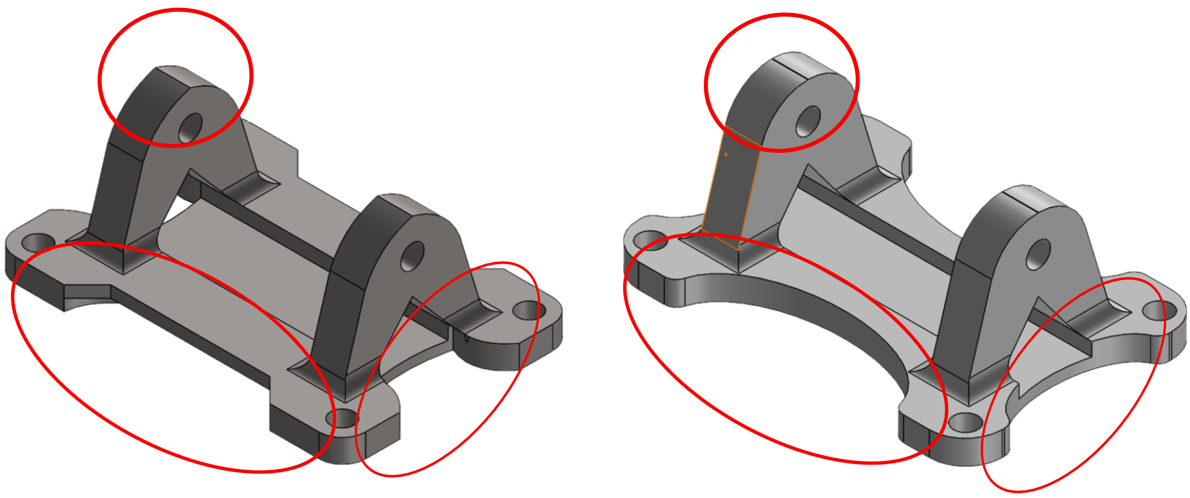


Figure 6.26: Highlighting the differences in the rough design and the rebuilt design

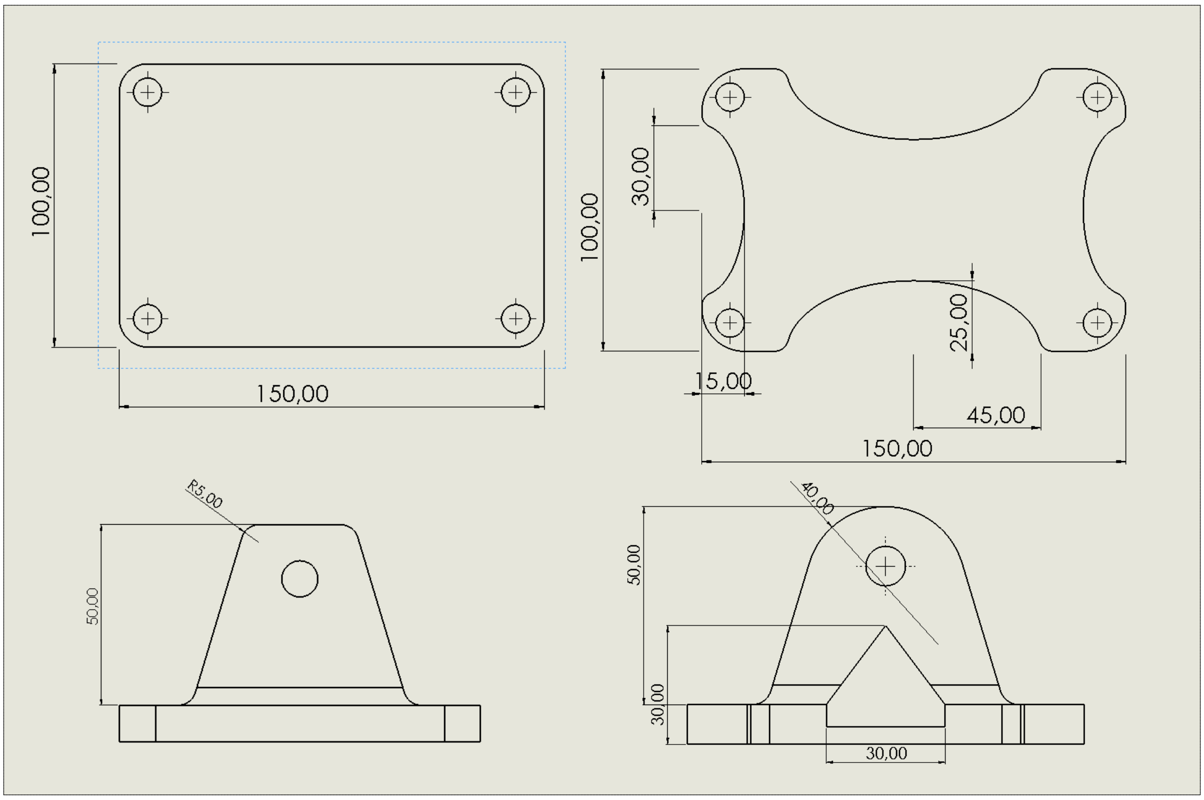
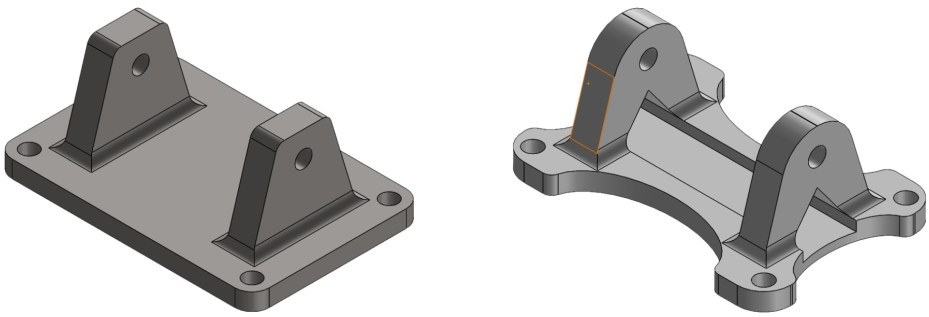


Figure 6.27: Comparing the original design and the new improved design



Mass: 1.636 Kg
von Mises Stress:150 MPa
Displacement: 0.026 mm
FOS: 4.14

Mass:1.040 Kg
von Mises Stress: 166 MPa
Displacement: 0.04mm
FOS: 3.73

Figure 6.28: Comparison of the two brackets

Figure 6.27 shows the differences in design between the old and the revised version. **Figure 6.28** shows that the new design is 36% lighter, the stresses increased with 16 MPa, the displacement increased with 0.014 mm and the FOS decreased with 0.41. The new design is about equal to the old design in all but weight.

6.1.6 Layer by Layer

This next method is based on redesigning a part, layer by layer, by using a graphic body from the TO results as reference. This method is best suited for more complex geometries, and is quite time-consuming. The method can be used to provide a detailed replication of the TO result. The example showcasing this method contains a table made out of 6061 alloy aluminum with a yield strength of 55 MPa. **Figure 6.29** shows the fixtures in each bottom corner, and a force of 1000 N applied on the top.

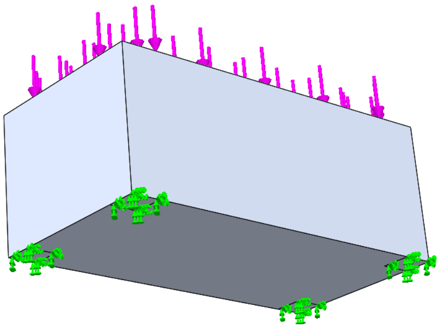


Figure 6.29: Constraints

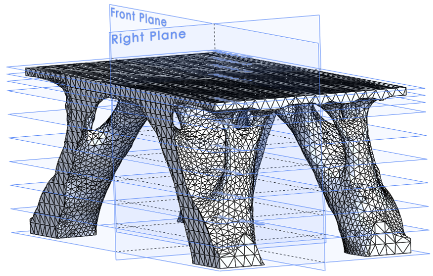


Figure 6.30: Smoothed mesh from TO-results

This part is symmetric along the front plane and the right plane, which makes it possible to design $\frac{1}{4}$ of the part, and then mirror it to produce the design. **Figure 6.30** shows the mirror-planes.

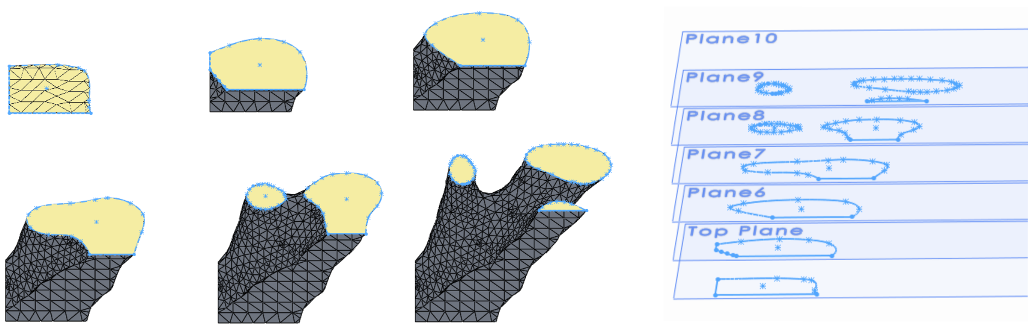


Figure 6.31: Sketches in different planes

After exporting the smoothed mesh to a graphic body, one has to create as many reference planes as is necessary. These will be used to draw the different cross sections which gradually changes from plane to plane. A function called **Section View** is used throughout the whole process, and allows the design to be "cut open", revealing the cross section of the part in whatever plane is needed. **Figure 6.31** shows the first six sketches and where they are located in the different planes. When all the sketches are done, a function called **Boundary Boss/Base** is used. This adds material between the profiles, in order to create a solid body. **Figure 6.32** shows how this works. After all the profiles have been joined together, locating and removing sharp edges that can cause stress concentrations is done, either with fillets or chamfers.

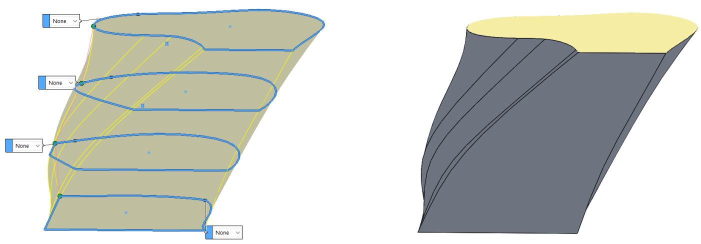


Figure 6.32: Boundary Boss/Base

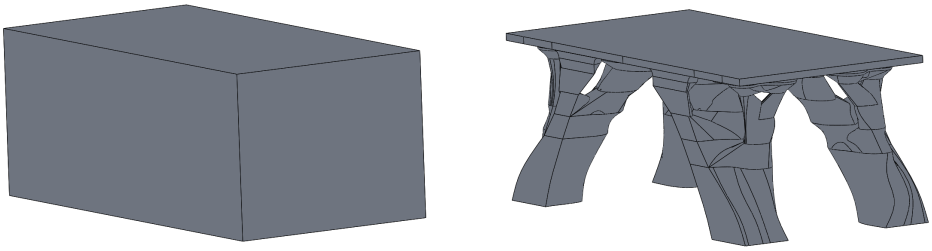


Figure 6.33: Comparison of the two tables

Figure 6.33 Shows the new design compared to the first design. The new design is 76% lighter and the von Mises stress went from 1.6 MPa to 2.1 MPa.

6.2 Selecting the Correct Method

As mentioned, it is up to the engineer to evaluate the TO result and choosing a design method. This chapter has given an overview of different methods, and how they are implicated. These can be used alone or in combination with each other to gain the best possible final design. It is also important that restrictions in regards to production and implementation is adhered to. Next, the effect of design space in TO is discussed.

7 Design Space

The design space is the available area where the part is supposed to fit, this makes the design space vary wildly from part to part. At times there is a lot of freedom, making room for a more optimized design, but sometimes the available space is restricted.

Altering the design space of a model may hugely vary the results of the TO results. It is therefore important to consider if the design space itself is optimized. The models chosen for this section have two different design spaces to highlight the radical differences it can cause. The purpose of these examples is to explore the effects of different design spaces and how that can be utilized for more optimized designs. The models showcased here are all from SolidWorks.

7.1 Example One

In this example, a metal plate is fixed by a hole on the left-hand side, and affected by a force of 1000 N pointing downwards inside the hole on the right side. The material used, is alloy steel with a yield strength of 620 MPa.

Figure 7.1 shows the process of going from a given problem and design space, to a finished model. To design the optimized parts, the procedure first mentioned on **Figure 6.10** in **Chapter 6.1.2** was used. The next step was to do a validation study, and test how the optimized parts compare.

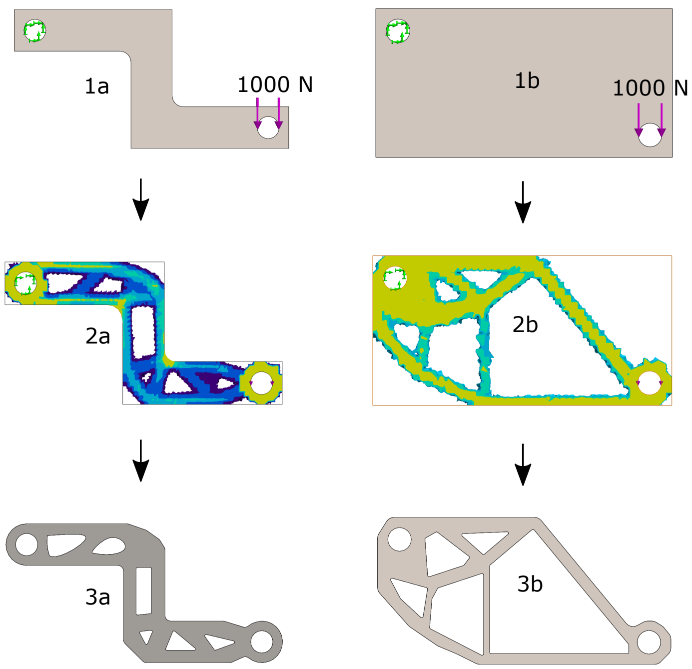


Figure 7.1: Differences in design space

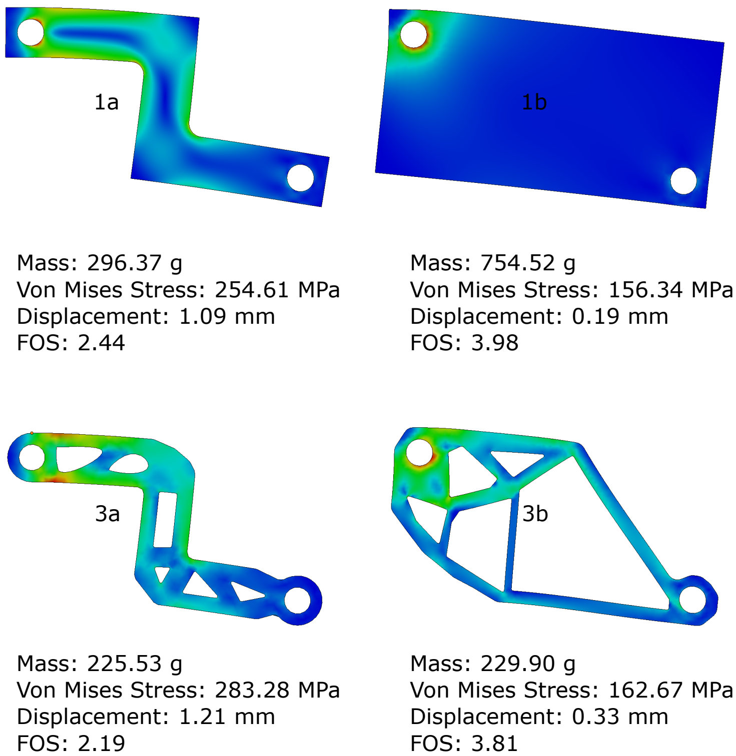


Figure 7.2: Comparing results of the different models

Figure 7.2 shows the von Mises stresses. In this case, the results show that in regards to displacement, stress and weight, the increased design space made it possible to make a stronger part. Both of the optimized parts **(3a and 3b)** are almost the exact the same weight. Even though the weight is close to equal, the difference in distribution of mass results in better FOS and less displacement.

7.2 Example Two

In this example, the same TO study from **Figure 6.29** will be used as it is a good example to showcase this method.

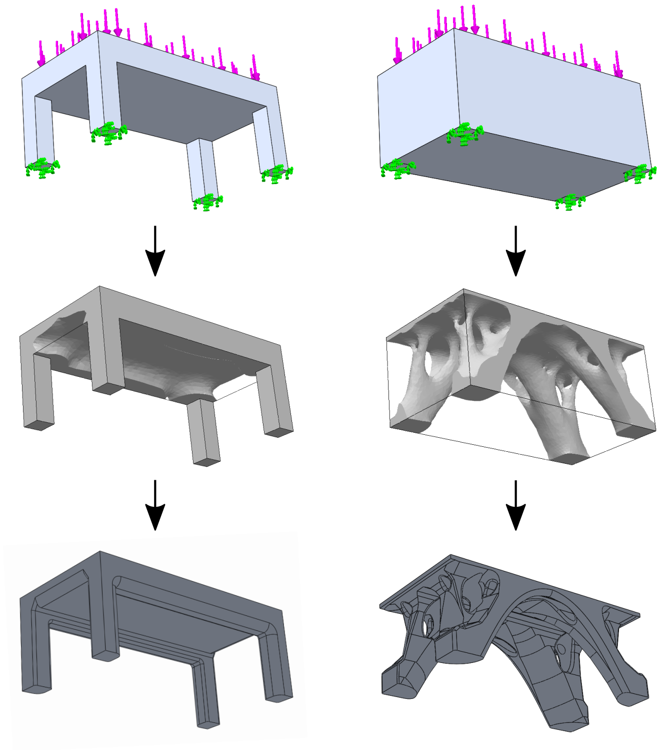


Figure 7.3: Comparing results of the different models

The part on the left-hand side has a smaller design space, which means the program has limited areas to remove material from. In this case, the optimized part with the biggest design space came out with a complex geometry, and will be harder to manufacture than the other part. On the other hand, this part will distribute the forces more even, which results in less stress concentrations. **Figure 7.3** shows the different geometry achieved by only changing the design space.

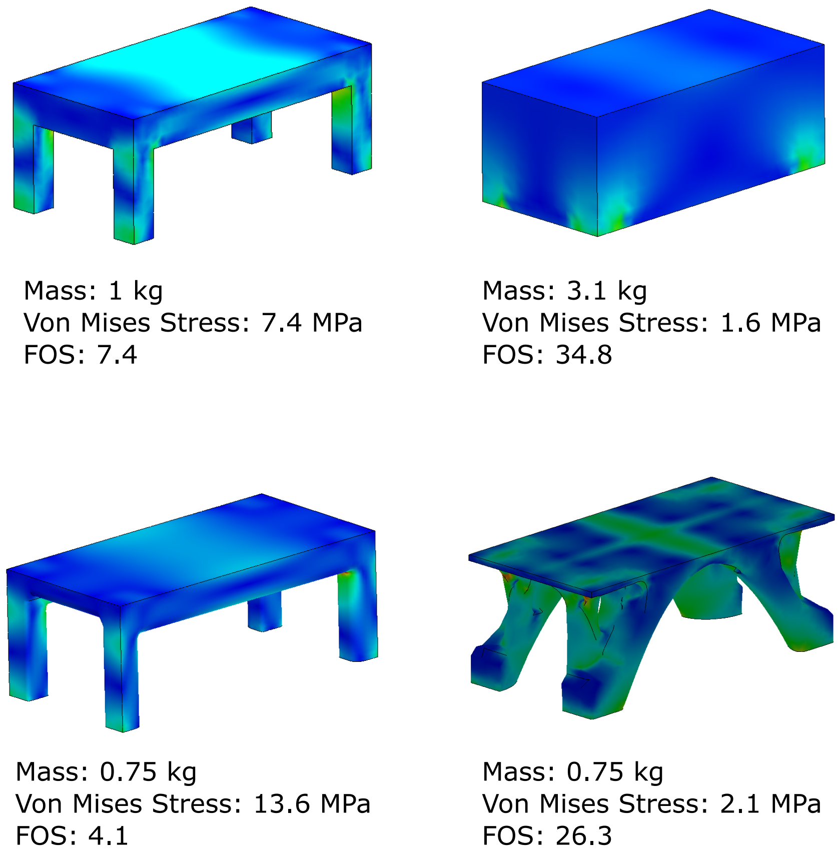


Figure 7.4: Comparing results of the different models

Figure 7.4 displays the reduction of stress, when the maximum design space is used for TO. This indicates how important it is to know the size of the available design space, before running a TO study. It is also important to have knowledge of which production methods are available. If the design space is to be increased, one should to consider possible conflicts. There might not be room for a bigger design space without causing collisions with other assemblies. This is essential in many scenarios, also in the following case study.

8 Case study

To use TO on a realistic case, a collaboration with Sevendof was initialized. They are designing a new drone and want to keep it as light as possible to increase the air-time. The group was instructed to optimize the drone arm. To solve this problem, the group implemented their research on TO. **Figure 8.1** shows the components of the drone arm from an angular perspective . It is important to note that the arm will have two motors, one on each side of the head. This is illustrated in **Figure 8.3**. All models in this chapter and all subsequent chapters are captured from SolidWorks unless otherwise specified.

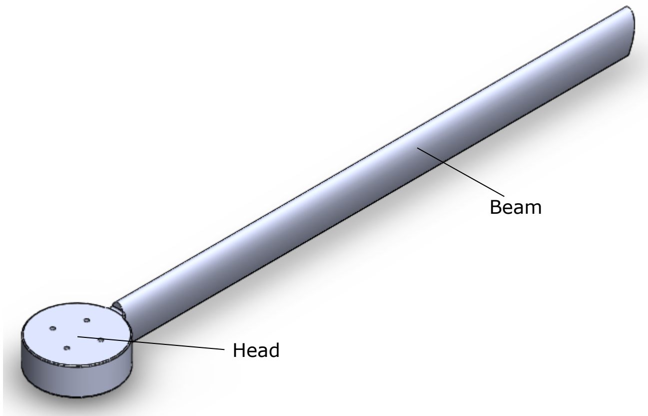


Figure 8.1: Arm design from Sevendof

Sevendof provided the group with two different load cases. Load case 1 is based on the maximum thrust of the motors and was used to optimize the drone arm. Load case 2 describes a critical failure, hence it will not be taken in to consideration when designing the part. Studies on load case 2 were done in order to see how the drone arm fared in a critical failure situation. The cases are described in **Appendix D**.

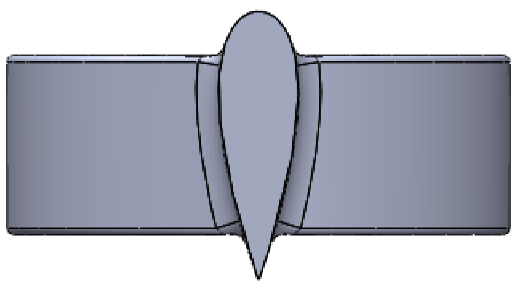


Figure 8.2: Arbitrary beam profile design from Sevendof

When redesigning the arm for Sevendof, a combination of different methods and theory had to be used. The group was given a design from Sevendof that was to be used as basis for the case study. The design of the beam profile was not the final design, but rather arbitrary. In the following subsection, the restrictions are noted.

8.1 Restrictions

There are several restrictions for the beam: force, displacement, height, length, material and protected surfaces. These will be explained in detail throughout this section.

8.1.1 Force and Displacement

The maximum thrust results in a load of **600 N**. This upward force is generated by two set of rotors, one on the upper hand side of the beam and one on the lower hand side. The load case is illustrated in **Figure 8.3**.

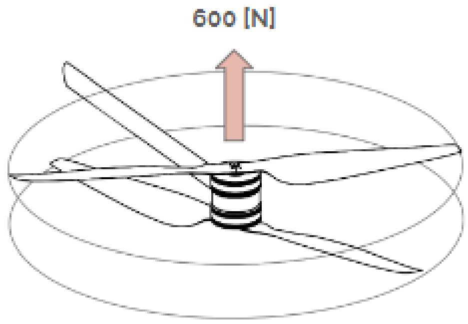


Figure 8.3: Illustration of the load case with maximum thrust

The force will cause displacement and stress in the beam. To keep the propellers from colliding with the beam and breaking, the maximum displacement can not exceed **10 mm**.

8.1.2 Height and Length

For the same reason the maximum deflection is given, the height of the beam is also restricted. The set height of the beam is **45 mm**. Also, the length of the beam should not be changed, and is set to a total length of **642,5 mm**.

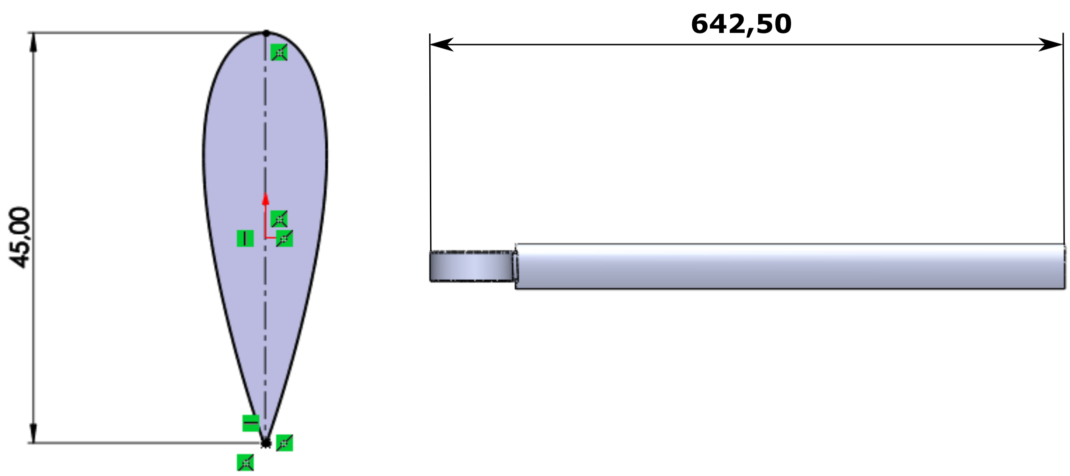


Figure 8.4: The set height and length of the beam

8.1.3 Protected Areas and Surfaces

To keep the drone as aerodynamic as possible, the surface of the beam is preserved. This means that only the inside of the beam will be altered. The fixing points for the motor will also be protected. As there are motors on both sides of the beam, they need to be connected as one unit, and have to withstand the forces.

8.1.4 Material

As this drone arm is supposed to be as light as possible, it excludes the use of high density materials, e.g. alloy steel. Rather, lighter materials should be used like aluminum or titanium. These materials can keep moderate to high yield strength, while having a density and weight as low as $\frac{1}{3}$ as alloy steel. As mentioned in **Chapter 3.2**, carbon fiber and fiberglass polyester are excluded from this case study.

8.2 Assumptions

There are several assumptions made while designing this arm. It is assumed that the beam is treated as a full body and is seamlessly fixed to the main body. As there are two motors on each arm, it is assumed that they both apply the same amount of force, **300 N** on each side. To simulate these forces, a remote load is set **30 mm** from the surface of the mount to both sets of holes, seen in **Figure 8.5**. The remote load is **45 mm** from the center. This height is calculated from the torque and force from load case 2, illustrated in **Appendix 10**.

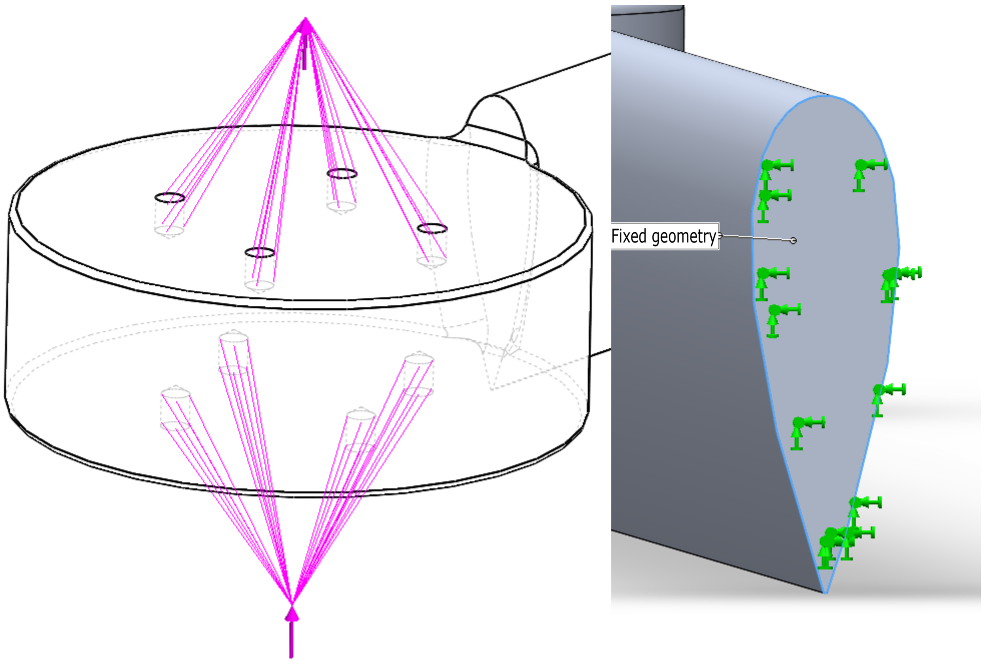


Figure 8.5: Remote load to simulate motors, and fixing point for the drone arm

As the beam is quite long and narrow, a non-linear static study will increase the accuracy of the simulation. However, a non-linear study can not be used as basis for a topology study. The FEA studies will therefore be static studies, but will be verified with non-linear studies when the TO is concluded.

8.3 Initial Study

There are several steps in this study that has to be taken into account. Firstly, the design must be checked, to see if it can withstand the forces and meet the requirements.

8.3.1 Analysis of Aluminum Beam

The first step in the TO study is to make sure the initial design can withstand the forces while still staying within the given restrictions. The design should not come too close to the maximum deflection. To start up the design phase, the general TO method from **Chapter 4.5** is used. In this first test, aluminum 2024-T3 was chosen as the material for the beam. This material is used in the aerospace industry and has a relatively high yield strength.

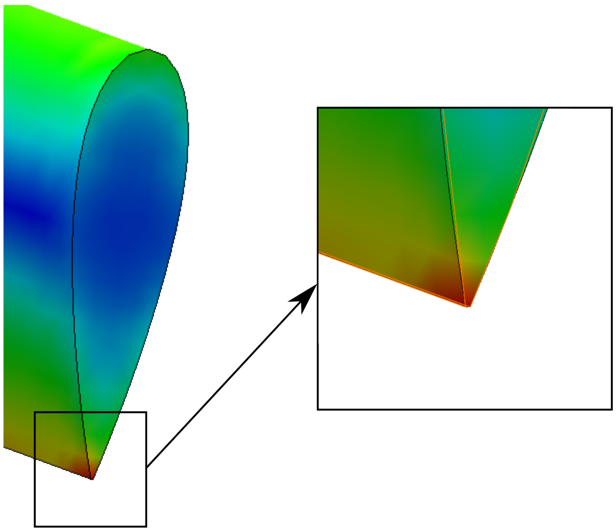


Figure 8.6: Possible stress concentration in initial beam

After running the first FEA, a possible stress concentration is spotted. This is tested by utilizing the mesh control tool. The results from the different mesh control sizes is shown in **Table 8.1**. Since the von Mises stress increases at every instance, without converging, it can be concluded that there is a singularity. The singularity does not affect the displacement plot.

Mesh control size	**von Mises Stress**
No mesh control	239,66 MPa
3 mm	239,64 MPa
2 mm	314,96 MPa
1 mm	390,38 MPa
0,5 mm	519,26 MPa

Table 8.1: von Mises tension for different mesh control sizes

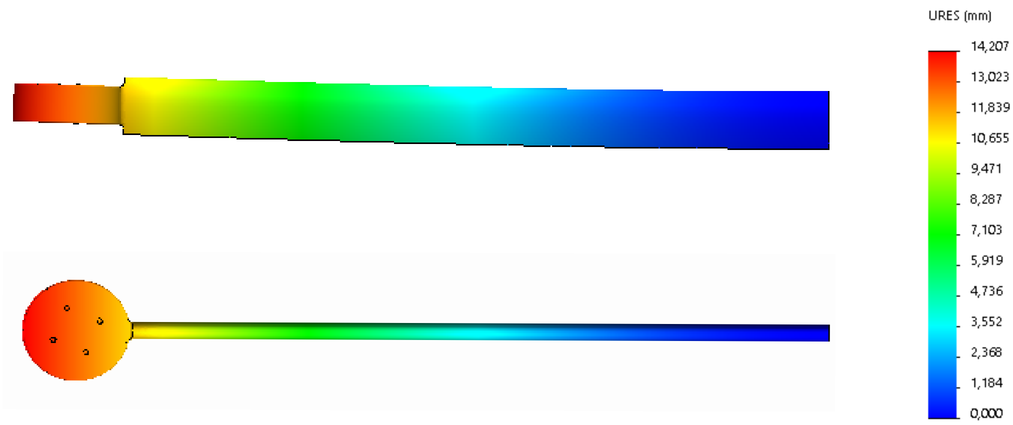


Figure 8.7: Displacement of initial beam with aluminum

As **Figure 8.7** shows, the displacement in the beam is quite high, at **14.2 mm**. This displacement is **4.2 mm** more than the maximum allowed value. The displacement in the model has to be reduced. In order to do so, the formula of displacement in a beam is used. It is described in **Chapter 3.1** and is as follows:

$$f = \frac{Pl^3}{3EI} \tag{11}$$

f = Displacement, P = Force, l = Length, E = Elastic modulus, I = Second moment of area

As neither the force or length of the beam can be changed, there are only two ways of decreasing displacement. By increasing the elastic modulus, or increasing the second moment of area. Changing the elastic modulus is done by changing the material, hence titanium will be tested next.

8.3.2 Deflection in Titanium Beam

The titanium used, is an alloy called Ti-6Al-4V. This material has an elastic modulus that is approximately **22 000 N/mm**^{2} higher than the aluminum alloy previously used. This will help reduce the displacement.

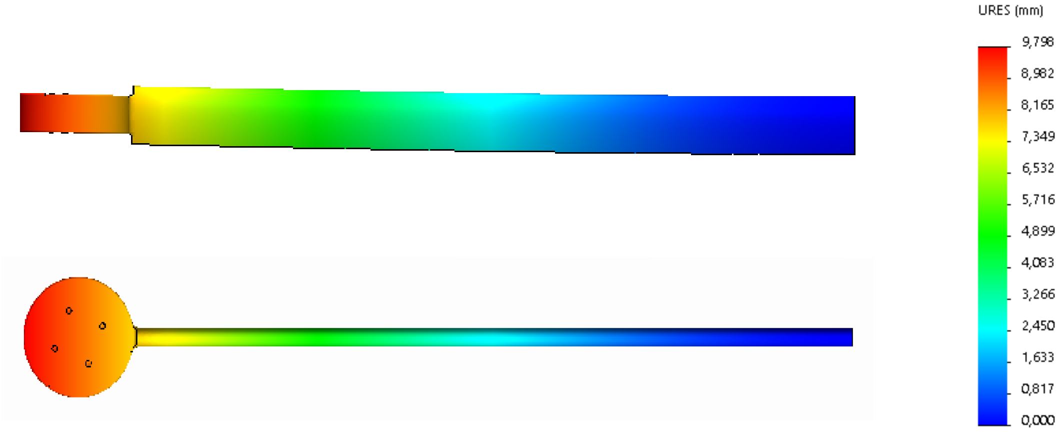


Figure 8.8: Displacement of initial beam with titanium

As **Figure 8.8** shows, the deflection in the beam is still quite high. It has definitely improved from the aluminum beam, but is still too high. The displacement is **9.8 mm**, which does not give a lot of room for improvement. The beam can only endure **0.2 mm** more displacement.

8.3.3 Thoughts on the Initial Design

Based on these results, the initial beam design from Sevendof is only suitable if titanium is used. Both the studies suggests too high displacement for further optimization, since removing material will result in an increased displacement. The only way to reduce the displacement, is to increase the second moment of area. This can be done by either using other dimensions for the design Sevendof provided, or try other designs. As the design from Sevendof was arbitrary, the group decided to try other designs and shapes. The argument is that it will be simplified for further research. In the opinion of the group, it will also result in a more optimal design.

For simplicity purposes, the material used for testing from this point on will be aluminum 2024-T3. This will result in a design that can be used for both aluminum and titanium. The further testing of only aluminum should not result in crucial differences in design.

8.4 Redesign Concepts

To gain the best possible design, different options should be tested. By utilizing research, intuition and brainstorming, three possible profile designs are suggested. The designs are shown in **Figure 8.9** below.

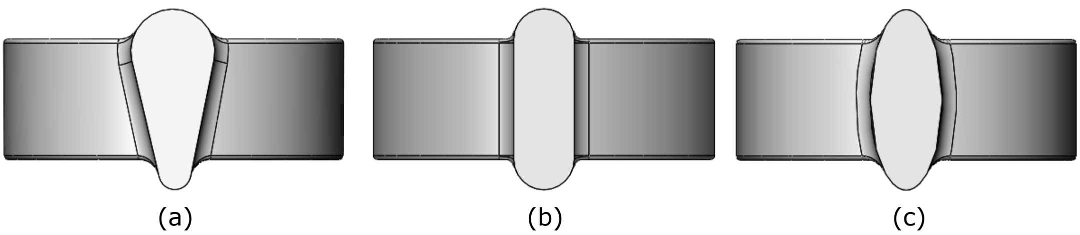


Figure 8.9: Redesign suggestions to beam profile

Beam (a) is a design inspired by the original beam from Sevendof. Beam (b) is a rectangular beam rounded at the top and bottom. Beam (c) has an oval shape. All of these designs are first draft suggestions to the beam profile design. The width and different diameters of the design are not final. To get the final design, the beams must be tested accordingly. After testing the beams, they will be compared to find the most optimal design.

8.5 Design Study

To figure out which of the three suggested designs is the optimal to use, they will all be tested. First they will go through a static study, which will then be used for a design study. The design study will enable changes to given dimensions to find the part which gives the lowest weight for a given displacement. This will be done before initiating the TO study. As the topology study will cut away mass, the displacement of the beam will also increase after the topology study. The maximum displacement for the beam has to be below **10 mm**. When utilizing TO in SolidWorks, an increased displacement of more than **50%** can give inaccurate results. For that reason, the displacement in the design study will be kept below **7 mm**, as it will provide room for TO while maintaining accurate results.

8.5.1 Beam A

The first design to be tested is beam (a). First, a static study is initiated as a base for a design study. The displacement plot is show in **Figure 8.10**, below.

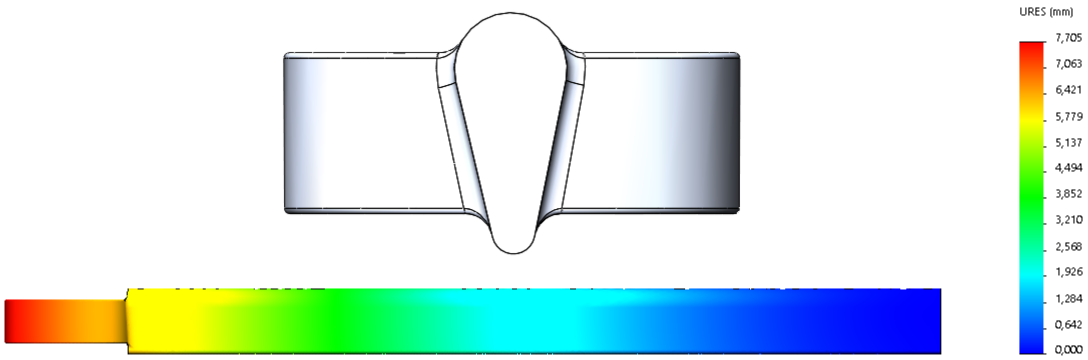


Figure 8.10: Deflection in beam A

In **Figure 8.11** the initial dimensions of the beam is shown. The beam has a set height of **45 mm**. The diameters of the upper and lower circle can be altered. With a design study, the shape can be changed to increase the strength of the design.

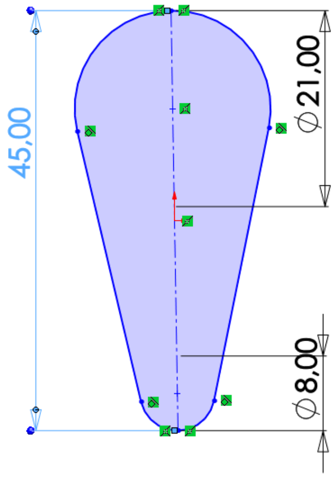


Figure 8.11: Dimensions of Beam A

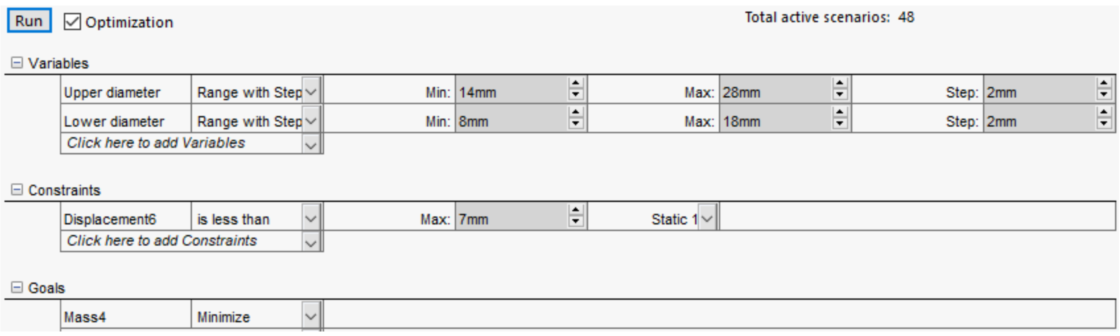


Figure 8.12: Input in design study for Beam A

To get the best possible design result, a wide selections of scenarios was used. The upper circle will range from **14 mm** to **28 mm**, with a **2 mm** step. The lower circle will range from **8 mm** to **18 mm**, also with a **2 mm** step. The input is illustrated in **Figure 8.12**. When the design study finishes it provides an optimized design. This design is then put through another design study with smaller step increments. The first study is used to get an idea of what the optimal diameters of the circles are. The new study will then be concentrated around the optimal diameters from the first study, to further improve the design.

		Initial	Optimal (34)	Scenario 8	Scenario 13	Scenario 14	Scenario 20	Scenario 27
Upper diameter		21mm	16mm	28mm	22mm	24mm	20mm	18mm
Lower diameter		8mm	16mm	8mm	10mm	10mm	12mm	14mm
Displacement6	< 7mm	7.70513mm	6.80693mm	6.44597mm	7.01262mm	6.69803mm	6.90553mm	6.83313mm
Mass4	Minimize	1479.933845 g	1502.130843 g	1793.689875 g	1561.577549 g	1644.020289 g	1528.535297 g	1508.732222 g

Figure 8.13: Selection of different results from design study

The optimal scenario from the design study has a **16 mm** upper and lower circle. These dimensions results in a **6.8 mm** displacement, with a weight of **1502 g**. This is a vast improvement from the initial beam. With only **22 g** more mass, the beam has reduced its displacement by **0.9 mm**. This is a **11%** reduction in displacement with a

1.5% increase in mass. There are several other plausible scenarios that can be used, i.e. scenario 27, but scenario 34 is still more optimal. As displacement is set to go beneath **7 mm**, there may be an even lighter model closer to the maximum displacement. With the result from the first design study in mind, the best result might be with around **15.5 mm** equal sized circles.

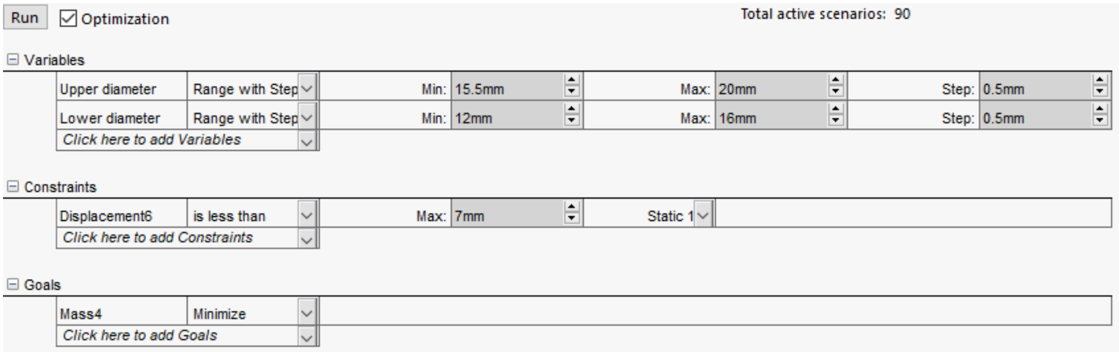


Figure 8.14: Input in second design study for beam A

To further improve the design, the new study will concentrate on these numbers, with smaller increments. This will allow the study to find the optimal design. As the first study had **2 mm** steps, the next study will only use **0.5 mm** steps. The upper circle will range from **15.5 mm** to **20 mm**. These are the diameters which yielded the best result from the first study. The lower circle will go from **12 mm** to **16 mm** for the same reason. The input is shown in **Figure 8.14**

		Initial	Optimal (71)	Scenario 26	Scenario 27	Scenario 53	Scenario 62	Scenario 72
Upper diameter		20mm	15.5mm	18mm	18.5mm	16.5mm	16mm	16mm
Lower diameter		16mm	15.5mm	13mm	13mm	14.5mm	15mm	15.5mm
Displacement6	< 7mm	6.20483mm	6.98969mm	7.03181mm	6.9462mm	6.99668mm	6.99216mm	6.89751mm
Mass4	Minimize	1626.042435 g	1472.484882 g	1482.665084 g	1499.72574 g	1474.11579 g	1472.892149 g	1487.430844 g

Figure 8.15: Selection of different results from second design study

The results from the second study, in **Figure 8.15**, shows that the original assumption was correct. The most optimal design is with equal circles, with a **15.5 mm** diameter. This provides the lightest design, which still holds the **7 mm** upper boundary of displacement. Again, the results shows that there are several possible combinations, but the beam with equal diameters gives the best result. As this design study was for beam A, the shape has completely changed into beam B. For this reason, beam B is a more optimal shape than beam A. There is no reason to test beam B further, as the different combinations of diameter dimensions have already been tested.

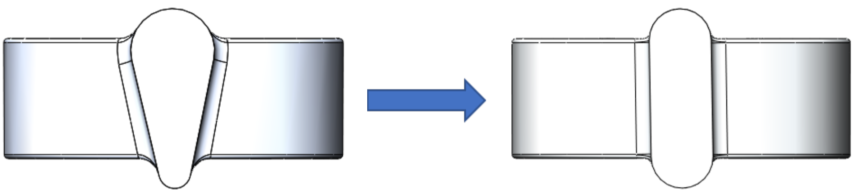


Figure 8.16: Change from beam A to beam B with design study

8.5.2 Beam C

As beam B already has been optimized, beam C is the next starting point. The design study starts with a static study, as showcased in **Figure 8.17**.

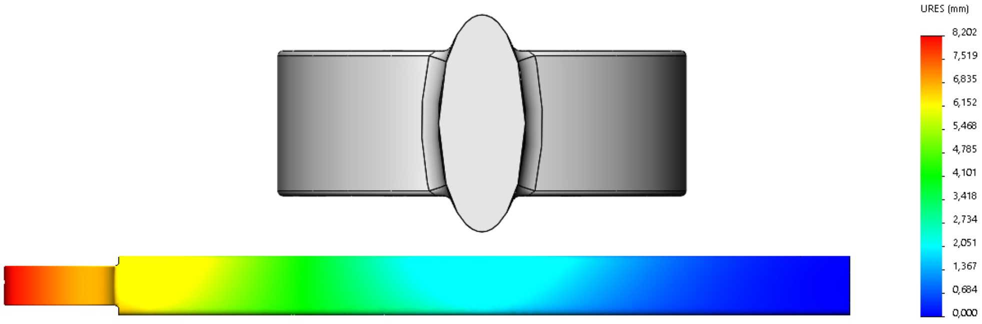


Figure 8.17: Deflection of Beam C

The shape of this beam is oval, which itself provides restrictions to how much change the design can undergo. The only variable to change is the width, which starts off at **18 mm**. Since it is controlled by the half of that radius, the steps will be reduced to **0.25 mm**, and have it test for the interval between **9 mm** and **12 mm**. This is showcased in **Figure 8.19**. This only results in 13 different scenarios.

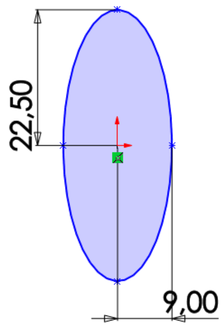


Figure 8.18: Dimensions of Beam C

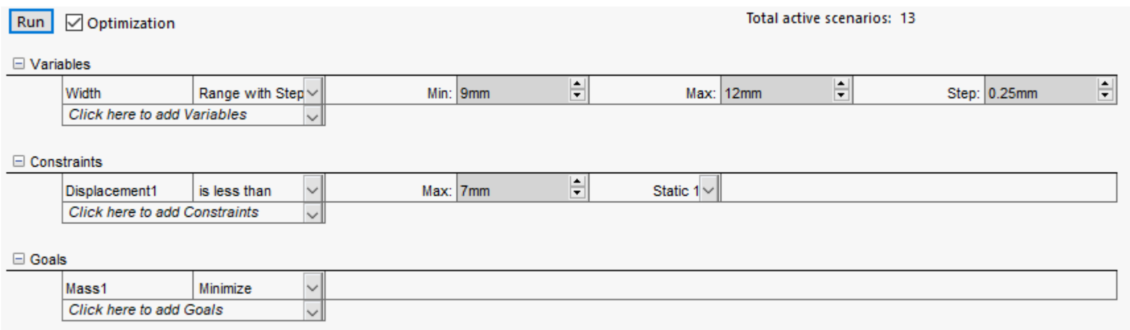


Figure 8.19: Input beam C

The study results in a design with a total widht of **21.5 mm**. This width will result in a **6.87 mm** displacement with a mass of **1649 g**.

		Initial	Optimal (8)	Scenario 6	Scenario 7	Scenario 9	Scenario 10
Width		9mm	10.75mm	10.25mm	10.5mm	11mm	11.25mm
Displacement1	< 7mm	8.20194mm	6.86515mm	7.20031mm	7.02822mm	6.70868mm	6.5597mm
Mass1	Minimize	1457.501183 g	1649.323385 g	1594.467248 g	1621.676221 g	1676.703229 g	1703.948942 g

Figure 8.20: Selection of different results from design study beam C

The result could be further optimized with even lesser steps, but by comparing the other results it is clearly not required. Scenario 7 is the closest to a **7 mm** displacement, surpassing by only **0.03 mm**. The weight however is only reduced to **1621.7 g**. By comparing to the other results, it is not comparable beam B.

8.5.3 Comparison of Beams After Design Study

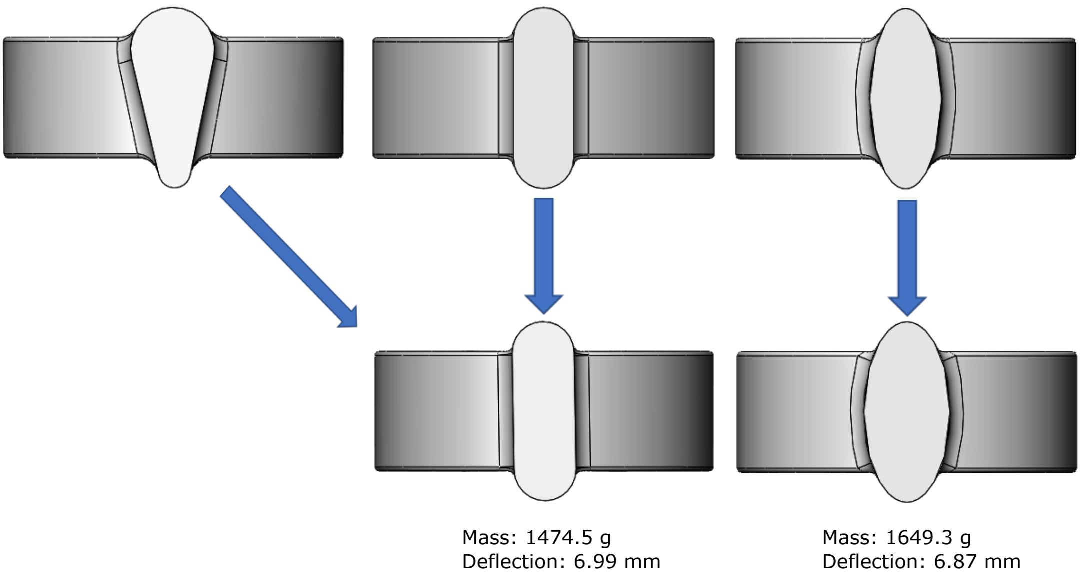


Figure 8.21: Results of redesigns after design study

After running the design study, the three design suggestions has turned into two. By comparing the two, it is possible to conclude that a rectangular beam with curved edges is the better of the two designs. It is close to **175 g** lighter than the oval design, with little difference in deflection. As beam B is the best design, it will be used for further optimization with TO.

8.6 Topology Optimization of Beam

To begin the TO process, a regular static study of the intended beam is performed. For this part of the process, the mesh is fine, as these results needs to be as accurate as possible.

8.6.1 Manufacturing Controls and Mesh

After the FEA study is finished, it is copied into a TO study. Here, several manufacturing controls are added. Firstly, as described in assumptions, the surface of the beam is protected. There will only be reduction in mass from the inside of the beam. Secondly, there will not be any changes to the head of the beam. Lastly, a symmetry plane going along the length of the beam is applied. This is implemented to make the TO results as symmetrical as possible. The symmetry is illustrated in **Figure 8.22**

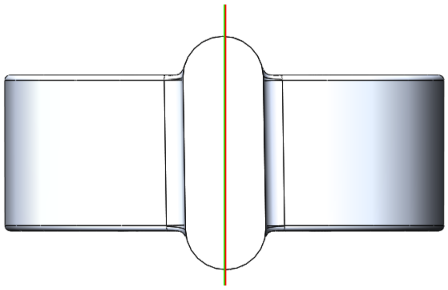


Figure 8.22: Symmetry plane for TO study

In addition to adding the manufacturing controls, a fine mesh is also generated for the study. As a fine mesh was already utilized on the initial FEA study, the TO will now have an even finer mesh. This is done to provide the most accurate results possible. This will however drastically increase the solver time.

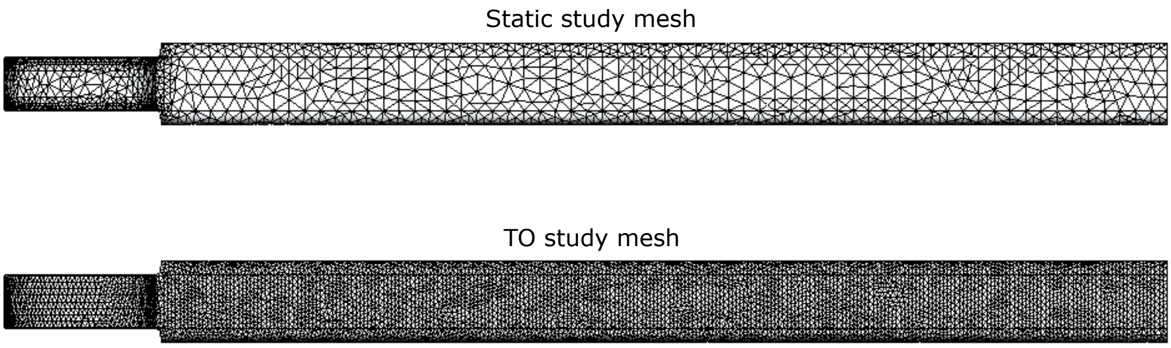


Figure 8.23: Mesh difference from static FEA study to TO study

8.6.2 Goals and Constraints

After creating a mesh and applying the manufacturing controls, the goals and constrains have to be set. As described in **Chapter 4.5**, there are three different settings for goals and constraints in SolidWorks. Out of these three, only two of them are relevant for this study. Several studies with different input will be done to get the best possible results.

The first study used is called **Best Stiffness to Weight ratio**. In this study, the mass can be reduced by a percentage or by a specified mass value. Reducing the mass by a percentage was used as there is no specific value to reach. The starting percentage will range from **30-55%**, with a **5%** jump between the studies. This will result in six different studies.

The second relevant study is called **Minimize Mass with Displacement constraint**. This study has two different inputs, either a specified value of maximum displacement or a specified factor. The specified factor will be based on the static study. The factor, **1.4**, is multiplied by the maximum displacement from the static study. This will result in a constraint of around **9.8 mm**. For safety, the study was also tested with specified factor. A value of **9.5 mm** was used. By using these different inputs, the chance of error is reduced. Both of the maximum displacement values are also set to a bit lower than **10 mm**, for safety. **Figure 8.24** shows the different goals and constraints used.

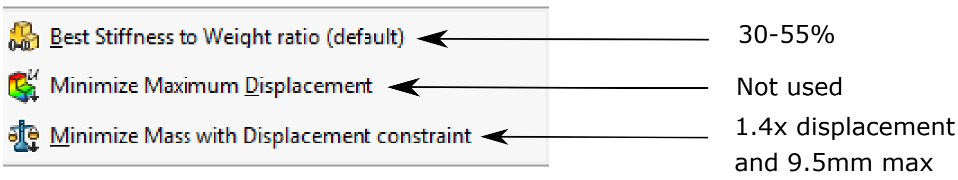


Figure 8.24: Goals and constraints in SolidWorks

8.6.3 Results From Topology Optimization Studies

In **Figure 8.25**, the results from the study **Best Stiffness to Weight ratio** is displayed. **Figure 8.26** displays the results from the study **Minimize Mass with Displacement constraints**. All the results are cut to show the inside of the beam, where mass can be removed. Each of the different inputs showcases one view from the side and one from the top. Looking at the results, it is obvious how the TO wants to optimize this beam. The end where the beam is fixed, has the most mass. The longer towards the beam head you move, the more mass is removed. In all the studies, the thickness of the cut stays almost the same at every point of the beam.

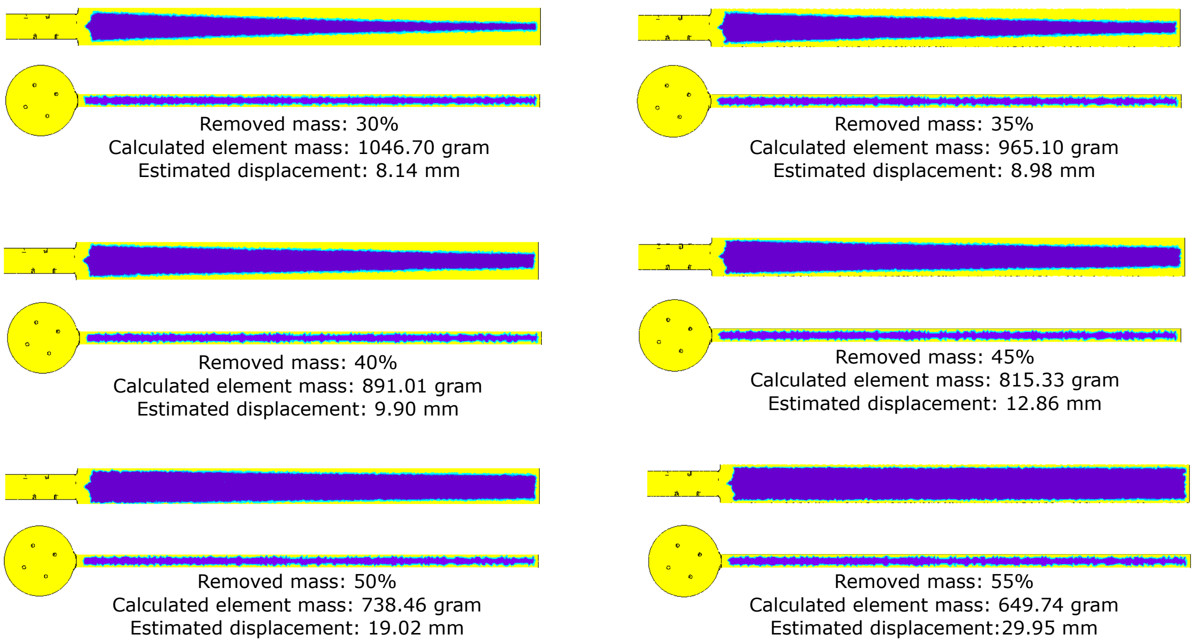


Figure 8.25: TO with best stiffness to weight ratio (30-55%)

By removing **30%**, the mass removed near the fixing point is lowest. Closing in on **50%** and **55%**, the mass removed is close to the same in the entire beam. These results have the look of a beam that easily could be extruded.

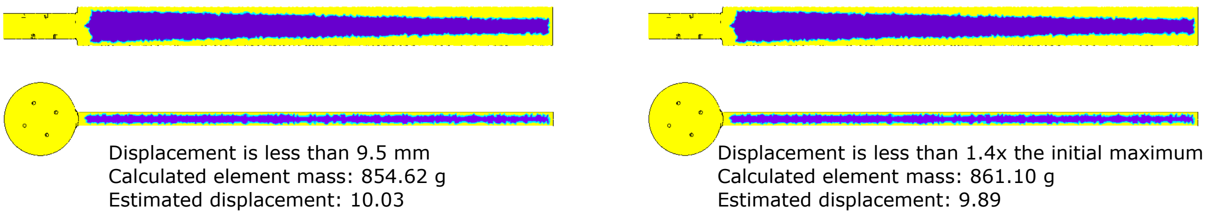


Figure 8.26: TO with different displacement constraints

Figure 8.26 shows the results with the displacement constraints. These results are close to equal. The results from the **9.5 mm** maximum displacement has the lowest calculated mass. This seems counter intuitive, as the other result should have a larger displacement constraint, at around **9.8 mm**. However, as the estimate displacement suggests, the **9.5 mm** results in a higher displacement. Both the results falls straight in between the **40%** and **45%** results from **Figure 8.25**. It will result in about **42.5%** mass removal. As the results from the displacement constraint are the only ones taking displacement in consideration, this will be used for further testing. The next step in the method is post-processing.

8.6.4 Post-Processing of Beam

For the post processing, the lofted cut method described in **Chapter 6.1.3** is the most suitable method. First, the sketch planes are made at both ends of where material can be removed. This is illustrated in **Figure 8.27**

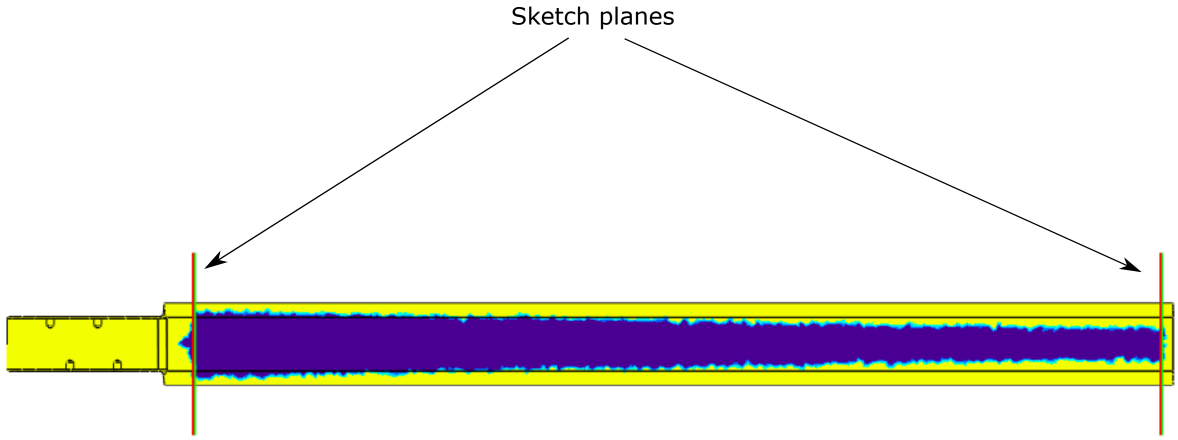


Figure 8.27: Sketch planes for lofted cut

After the sketch planes are created, they are used to do a section clipping of the TO results. This way, the sketches for the lofted cut can easily be designed. **Figure 8.28** visualizes this process. The sketches are not exactly as the TO results suggests, but a close approximation.

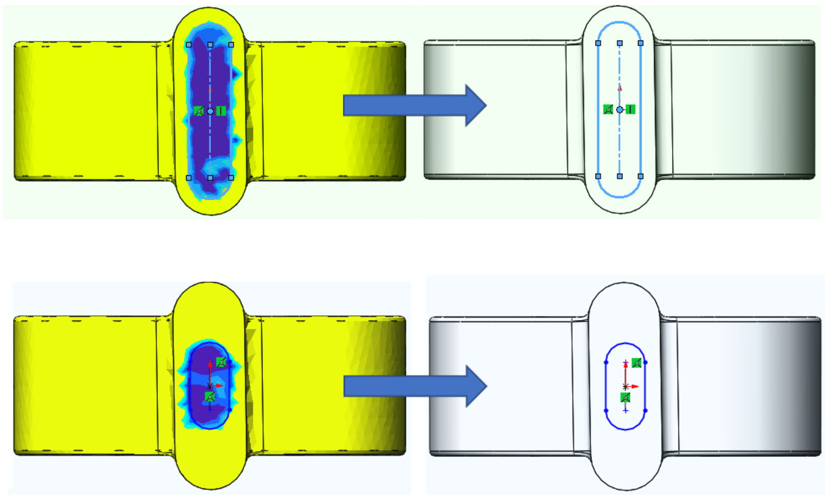


Figure 8.28: Sketches for lofted cut

The sketches are then used for a lofted cut, and creates the new model. The section clipped is shown in **Figure 8.29**. Surprisingly, this loft cut only resulted in a weight of **1121.53 g**, which is approximately **260 g** more than the calculated element mass suggested. The displacement also has a lower value than expected. **Figure 8.30** displays the FEA study conducted after the lofted cut was executed. It resulted in a displacement of **7.821 mm**. This suggests that there are some errors in the calculations of mass and displacement in the TO study.

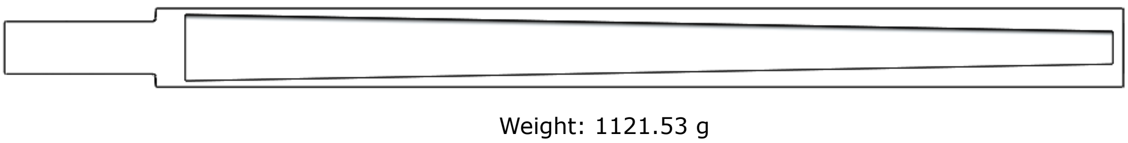


Figure 8.29: Section clipping of lofted cut

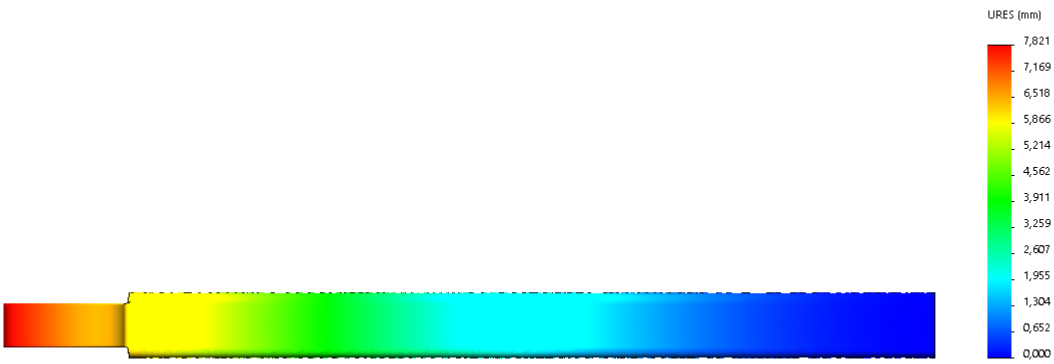


Figure 8.30: Displacement in topology optimized beam

As the mass removed resulted in a higher weight than expected and a better displacement than excepted, further testing is needed. To do so, the same post-processing procedure is executed for the other results. The testing was done for the **45-55%** removed mass, from the results in **Figure 8.25**.

TO mass reduction	Mass after post-processing	Displacement after post-processing
45%	1029.59 g	8.483 mm
50%	967.18 g	9.958 mm
55%	932.13 g	11.515 mm

Table 8.2: Mass and displacement in Topology optimized beams

In **Table 8.2** the different masses and displacements for the post-processed TO results are displayed. The calculated mass and displacement from **Figure 8.25** suggested different results. In the validation studies, there is only a relatively small difference in mass between the **45%** and the **50%** beams, a **62.4 g** difference. This does however result in a large displacement difference, over **1.475 mm** larger. Using the **55%** resulted in an additional **35 g** of removed mass. However, this results has a far higher displacement, at **11.515 mm**. When more mass is removed, it will have to be removed from the right side of the beam, where it is fixed. Some of the mass is also removed along the lofted cut. This mass seems to be crucial for the displacement.

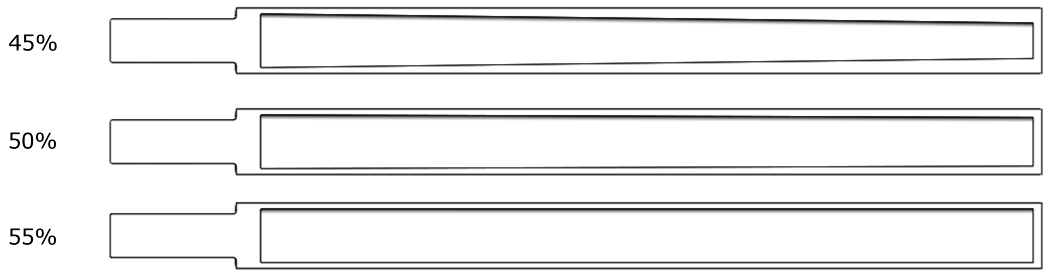


Figure 8.31: Inside view for the different post-processed beams

The small difference in mass results in large difference in displacement, due to the location of the mass removed. The **55%** beam has the same dimensions on both edges of the lofted cut, and is thus an extrusion. This weakens the beam significantly. To make the strongest possible beam, the lofted cut with less material at the fixing point is clearly the best option. The **45%** beam is suggested as the design. It is light while still having an acceptable safety margin when it comes to displacement, with **1.5 mm** lower than the maximum

8.6.5 Design for Production

The beam has been tested as one part, that does however make it difficult to realize. To produce this design as one piece is not feasible with the current production methods. If Additive Manufacturing is used, the inside hollow will be filled with production material and since the beam is completely closed, the material will remain inside the beam. If this design is going to be produced, changes have to be made. Discussing the possibilities for production, the group suggests the best way to solve the issue is to divide the beam into two parts. The head of the beam as one solid, and the beam itself as the other.

The design is displayed in **Figure 8.32** and shows how the parts are divided to give a possibility to produce the beam open in one end. This also provides a basis for joining the two components.

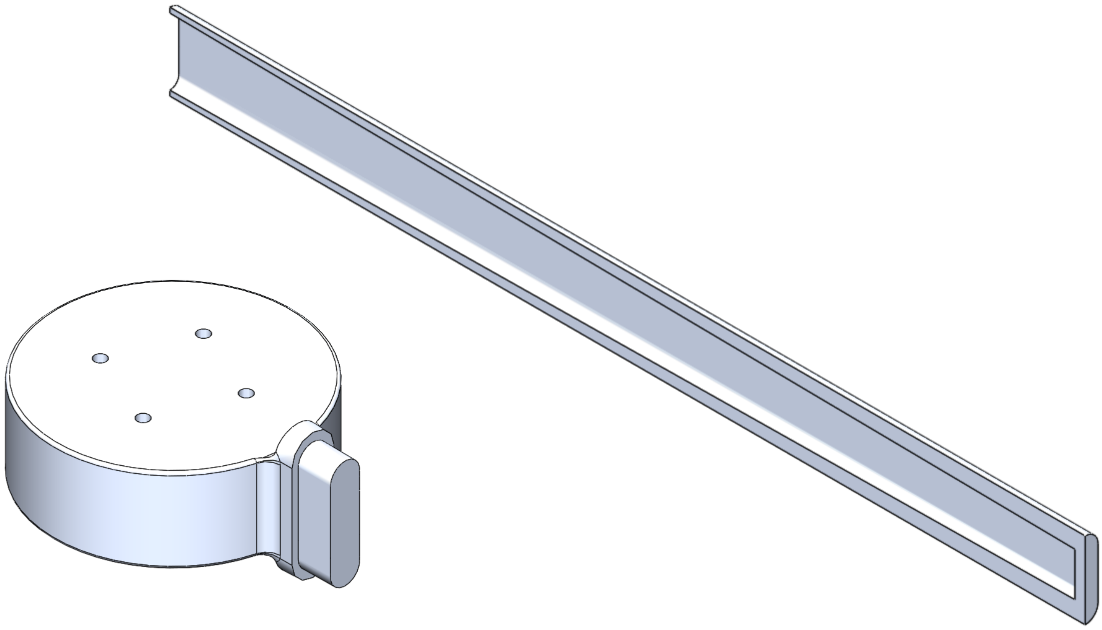


Figure 8.32: Design for production

The joined assembly is displayed in **Figure 8.33**. There are several possibilities to assemble these components, including permanent and non-permanent methods. Welding, chemical bonding, bolting and shrink fitting are all possible options. However, some small design alterations may have to be made to facilitate the chosen method.

Figure 8.33: Assembly of production design

The beam head can be produced using a variety of methods, e.g. casting, machining or a combination of the two. For the beam, there are fewer reasonable production methods available. The most likely methods are AM or casting. If casting is used, expendable mold casting seems like the ideal method. The beam is long and narrow, which could result in difficulties regarding draft angles using other casting methods. As **Figure 8.34** shows, there are no draft angles on the sides. This could result in difficulties when extracting the part from a permanent mold.

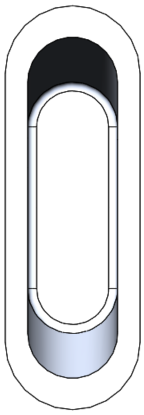


Figure 8.34: Visualization of draft angle issues

8.7 Final Design Comparison

The design has gone through a lot of changes during the design phase. The cross section of the beam has been altered, tested and TO has been implemented. Below, there is a comparison between the new design and the initial design from Sevendof.

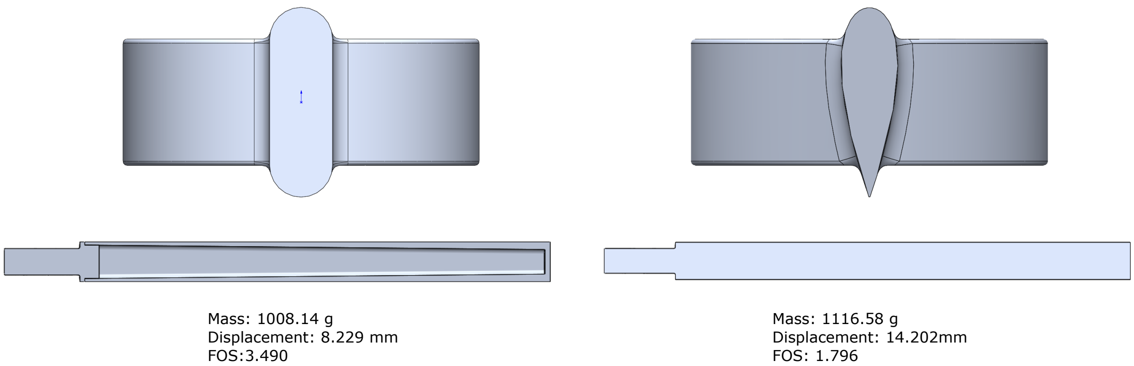


Figure 8.35: Comparison of initial beam design and optimized assembly

As displayed in **Figure 8.35**, the design changes has significantly improved the beam. Firstly, the mass has been reduced by **108.44 g**. While making it lighter, the beam has also become stronger and stiffer. It has a displacement of **8.229 mm**, which is **5.973 mm** less than the initial design. Lastly, the FOS has been increased by **1.694**, resulting in a FOS of **3.49**.

8.8 Other Possibilities

Even though the final design was improved, there are still some other variations of the design, production and material that are viable. Different options are explored next.

8.8.1 Design

There are many variations of the design that can be used. One way of finding different variations is with the design study tool. Since the TO result gives insight to where mass can be removed, setting up an design study based on this can be advantageous. Instead of setting up new TO studies with different widths of the beam, post-processing and then validating them, the method explained in **Chapter 6.1.4** can be utilized. This can save a lot of time.

All the TO studies conducted on the beam, suggests the same height of the cut in the force end of the beam, while the fixed end has a variation of heights. Since the overall shape of the lofted cut stays the same, the dimensions that can be altered are known. Combining these variables with the width of the beam itself, can result in an even better optimized design. The alterable dimensions are displayed in **Figure 8.36**, as an example.

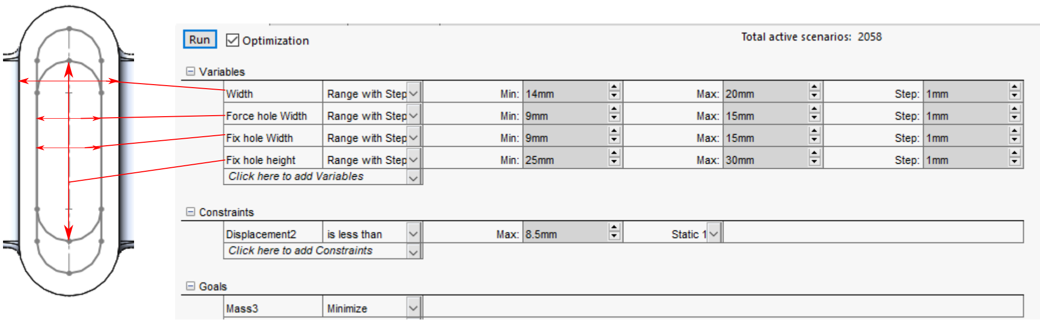


Figure 8.36: Input in rapid validation study of beam

Running this exact study will result in 2058 scenarios. This requires a lot of computing power and storage. More steps will result in an increase of scenarios, but could result in an even better optimized design.

		Optimal (340)	Scenario 80	Scenario 285	Scenario 536	Scenario 1257	Scenario 2058
Width		17mm	16mm	18mm	17mm	17mm	20mm
Force hole Width		15mm	13mm	14mm	15mm	13mm	15mm
Fix hole Width		15mm	10mm	14mm	12mm	13mm	15mm
Fix hole height		25mm	25mm	25mm	26mm	28mm	30mm
Displacement2	< 8.5mm	8.4244mm	8.37635mm	7.78763mm	8.31493mm	8.6086mm	7.90584mm
Mass3	Minimize	932.753779 g	1003.035739 g	1028.265806 g	974.7140507 g	979.3405122 g	1049.433163 g

Figure 8.37: Selection of results from rapid validation study of beam

Figure 8.37 shows a selection of the results from the rapid validation. As one can see, there are several different scenarios which works perfectly fine for the beam. Some are even lighter with less displacement. Something to be aware of, is that some of the results may not be optimal regarding production. The most optimal design from the study, has a width of **17 mm**, while the lofted cut has a width of **15 mm**. This results in a thickness of **1 mm** on each side, which may not be suitable for manufacturing

Even though this study could help further improve the beam, it was not proceeded with, as the method was discovered at the very end of the thesis.

8.8.2 Lattice Optimization

LO might be a viable option for the optimization of this part. As described in **Chapter 5.5**, LO is suitable for structures with protected external surfaces. This can be applied to the drone beam. However, manufacturing a LO result is problematic with the current state of technology. Due to limited information on the subject, LO proved difficult to learn. By the time this was sorted out the group had chosen to move on with TO as the main optimization process for the drone beam.

8.8.3 Production

Some other production methods include extruding profile of the beam and then cutting it to size, as explained in **Chapter 3.9.2**. To do this the cross section of the beam has to be constant, that means the **55%** option is utilized. However, the **55%** option does not meet the displacement requirement. To meet the requirement the amount removed by the extrusion has to be reduced, meaning a heavier beam, but one that can be produced relatively cheaply and efficiently.

8.8.4 Material

There are also possibilities to use a different material, i.e. titanium or carbon fiber. These materials holds an overall higher stiffness and strength. If the material is changed, some alterations could be made to get an even lighter beam.

8.9 Load Case 2

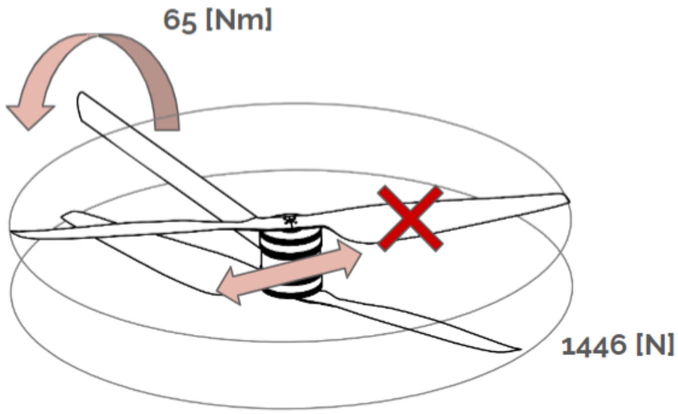


Figure 8.38: Load case 2
Provided by: Sevendof

Load case 2 describes a critical failure. If one of the propellers breaks on one of the motors, a centrifugal force of **1446 N** and momentum of **65 Nm** will start to affect the drone arm, see **Figure 8.38**. In order to land safely, the drone must last ten seconds with these loads. To simplify this problem, these following restrictions were set:

- The longitudinal force is ignored
- The force is oscillating and decreasing towards zero, show in **Figure 8.39**
- There is no delay from when the propeller breaks to the motor turns off.
- The lift provided by the motor with the fully functional propeller, and the broken propeller will not be included in the simulation. This force is shown on **Figure 8.40**, colored in blue.

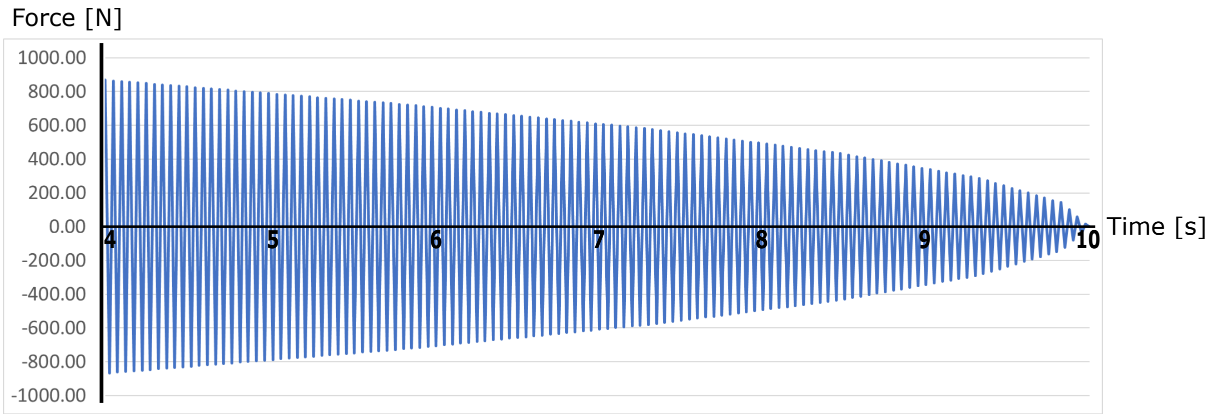


Figure 8.39: Force and Time Graph from 4 - 10 seconds

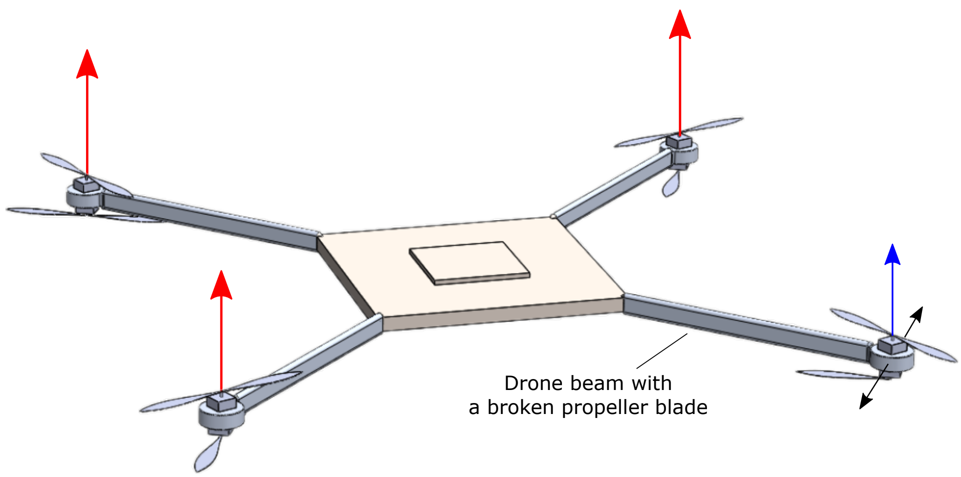


Figure 8.40: How the forces work, in the exact moment one propeller blade breaks

Figure 8.41 shows the simplified and predicted movement of the beam, it also shows one of the natural frequencies of the beam. However, resonance does not pose any great dangers as it is only is experienced momentarily as in this case. Thus the problem of resonance will not be explored further in this thesis.

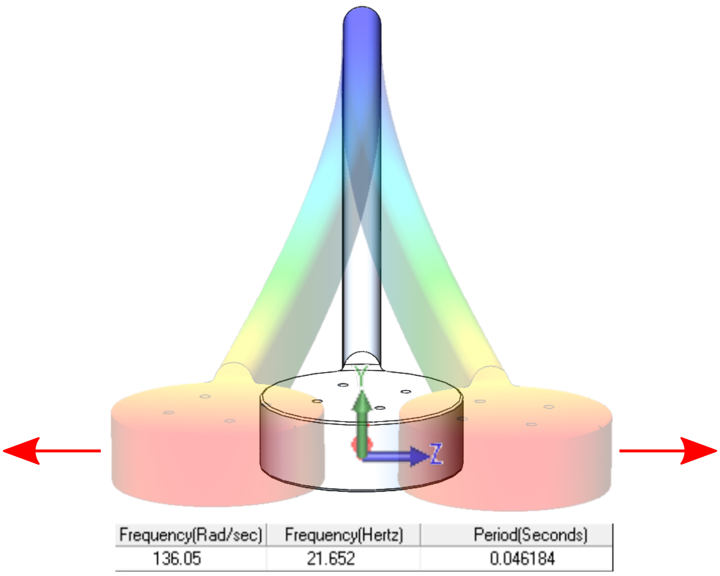


Figure 8.41: Predicted movement of beam and natural frequencies

8.9.1 Breaking Down the Problem

Because the force varies with time, the study should be solved as a dynamic simulation. In order to find all the plots needed, one first need to find out how many full rotations the propeller does in ten seconds, before a complete stop. The propeller rotates at $4000RPM \approx 66.7RPS$. **Figure 8.42** shows RPS and time, the hatched area represents the total rotations in 10 seconds. This can easily be calculated; $\frac{66.7RPS*10s}{2} \approx 333.3Rounds$. For each time the propeller rotates one full rotation, the forces shown on **Figure 8.41** will peak two times, in the Z-direction. This means that the number of total alternations will be: $2 * 333.3 \approx 667$.

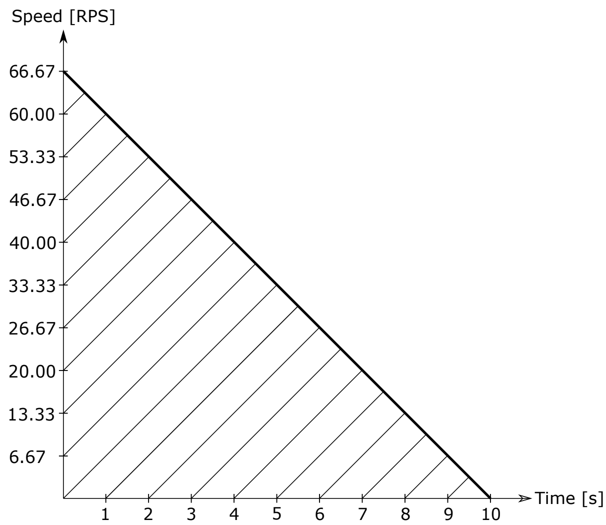


Figure 8.42: RPS and seconds graph

When the rotational speed decreases, the number of alternations will also decrease. The total time was split into 10 equal intervals (0-1, 1-2 .. etc.), to make the time increment constant in each step. This is done in order to make the time increment hit each force peak, and make the simulation easier. The matrix on **Table 8.3** shows the data used to plot each curve for the different intervals.

Time [s]	RPS	Force [N]		Time Interval [s]	Average Full Rotations	Alternations
0.00	66.67	1446.00		0-1	63.33	127
1.00	60.00	1301.40		1-2	56.67	113
2.00	53.33	1156.80		2-3	50.00	100
3.00	46.67	1012.20		3-4	43.33	87
4.00	40.00	867.60		4-5	36.67	73
5.00	33.33	723.00		5-6	30.00	60
6.00	26.67	578.40		6-7	23.33	47
7.00	20.00	433.80		7-8	16.67	33
8.00	13.33	289.20		8-9	10.00	20
9.00	6.67	144.60		9-10	3.33	7
10.00	0.00	0.00			Total:	667

Table 8.3: Matrix for predicted rotation

8.9.2 Simulation

The beam used for this simulation is the same as the assembly shown on **Figure 8.32**. The material is Aluminum 2024-T3 with a yield strength of 345 MPa. For the first simulation, linear dynamic (modal time history analysis) was used.

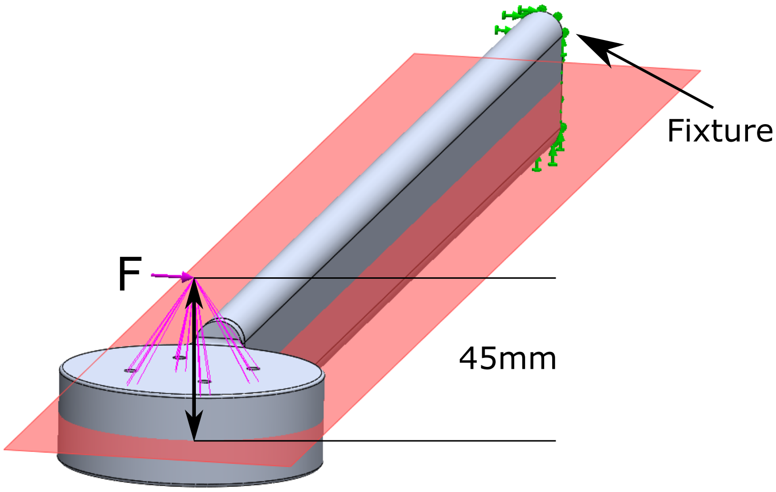


Figure 8.43: Constraints on beam

The force, F, is located 45mm above the plane shown on **Figure 8.43**. This will include the momentum of; $1446N * 45 * 10^{-3}m \approx 65Nm$, given in load case 2.

Interval [s]	Max. Von Mises Stress [MPa]	Max. Resultant Displacement [mm]
0-1	360.6	77.1
1-2	354.1	75.5
2-3	336.9	73.4
3-4	319.8	70.9
4-5	299.0	67.1
5-6	278.9	62.0
6-7	243.0	54.5
7-8	196.3	44.6
8-9	137.2	31.8
9-10	57.4	14.2

Table 8.4: Maximum displacement and maximum von Mises stresses - Linear dynamic study

Table 8.4 shows the results from each of the intervals. The displacement and stresses are decreasing with the time, because the applied force is also decreasing over time.

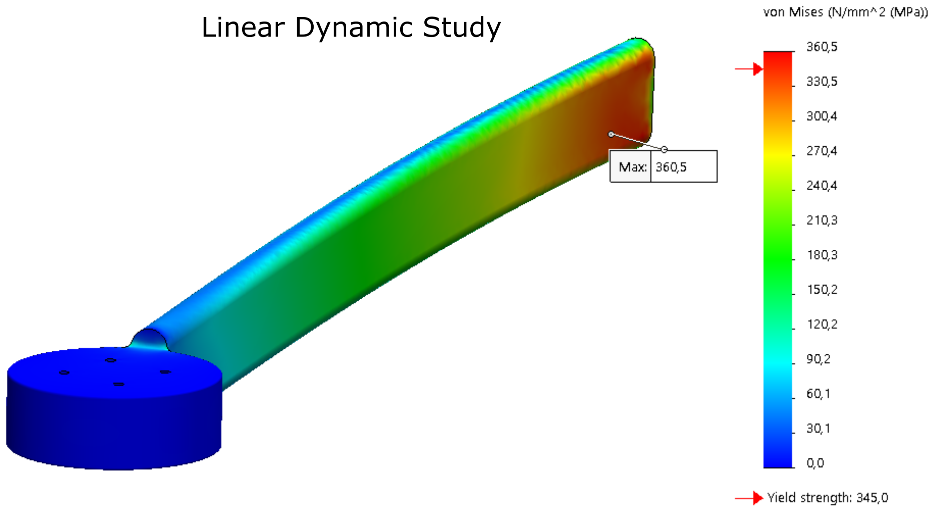


Figure 8.44: Result showing maximum von Mises stress (0-1 seconds)

Figure 8.44 shows that the maximum stresses is above the yield strength. Based on these results, the arm will be permanently deformed. Since the results indicates a large displacement, a nonlinear dynamic study should be conducted in order to ensure more accurate results.

Nonlinear dynamic studies requires a lot of computing power and are time-consuming. Due to the complexity of the nonlinear dynamic study, it was only run in the time interval with the highest stresses, 0-0.4 seconds. To prevent the program from crashing, the study had to be split into two individual studies.

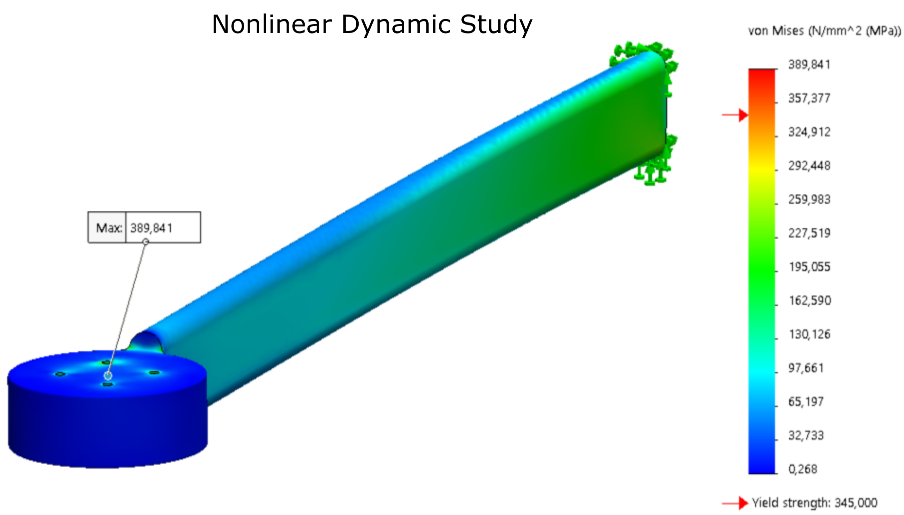


Figure 8.45: Result showing maximum von Mises stress (0-0.2 seconds)

Time	Max. Von Mises Stress [MPa]	Max. Resultant Displacement [mm]	Solver Time
0-0.2	389.84	42.57	01:55:15
0.2-0.4	301.19	40.58	01:06:02

Table 8.5: Nonlinear study results

Figure 8.45 shows the maximum stresses acting inside of the holes on the head. The reason for this is unclear and might be caused by incorrect inputs. The nonlinear study resulted in increased stresses, but in less displacement, see **Table 8.5**. The stresses are still above the yield point, as with the linear dynamic study. With lack of knowledge about these types of studies and the huge time requirement, the group decided to conclude the study.

The results from the studies conducted in load case 2 may be unreliable. The case will be further discussed in next chapter, along with the rest of the thesis.

9 Discussion

This chapter contains discussions about the different software, post-processing, design space and the case study. The points made and the suggested improvements in this chapter are based on the results gathered in this thesis, as well as the groups subjective experiences using TO.

9.1 Software

Most commercial CAD programs today mostly utilize the SIMP method for their TO algorithms. This is reflected in the available software as most other options are from scientific papers as research material or proofs of concepts. The only notable exception found to this was the ParetoWorks plugin. As mentioned in **Chapter 5.3.2**, it utilizes the LSM algorithm for TO. In the comparison done in **Chapter 5.4** it presented the best results. The main advantage is the solver time, as it is far superior to the other two tested methods. The results also presented a significantly lower stress, which may indicate flaws to the FEA solver within ParetoWorks. However, the speed of the TO is amazing and it produced visually pleasing results. At the time of writing this thesis, the LSM is not a commercially used method for TO. However, based on the results presented by ParetoWorks there might be a shift in the preferred method for TO in the future.

Based on result of the research done in this thesis, the BESO method would not be recommended for TO. A long solver time combined with a questionable geometry makes it not suitable for TO compared to the other methods tested in this thesis. However, there was a limited selection of programs to test this method with and it seemed likely the one chosen was intended for research purposes. This might give BESO a disadvantage in the comparison. On the other side, seeing as there is such a limitation to programs using BESO, it might not be the best algorithm for TO.

Another method to consider is the LO. This method is comparable to TO as it significantly reduces weight while maintaining stiffness. It is also very useful if the design can not be altered much, but weight needs to be reduced. Some of the issues for LO has been associated with the manufacturing of the results and the software. As presented in this thesis, the software currently used for LO proved either difficult to use, or produced a non-viable result. The computational power required to create a viable LO result on a larger scale has also been a limiting factor. As production methods become more advanced and computers become more powerful, it becomes easier to produce and can be more readily utilized.

A combination of LO and TO might prove to be beneficial when designing strong and lightweight parts. However, further research on the matter is required.

9.2 Non-Viable Results

It is possible when using TO that the results end up being non-viable, as mentioned in **Chapter 3.7**. When this occurs it is important to realize that a failed simulation is not always wasted. It might contain information important to the case. In **Figure 3.23**, reducing the removed mass fixed the issue, but that is not always an option. A bad result might be indicative of a problem that is not correctly defined or one that has been simplified to much or a range of other issues. It would be preferable if in later versions of TO software such errors as disjointed geometry would be made clear to the user earlier, who could then make a decision based on this new information. A simple error message stating something along the lines of: *"Error: the current Topology optimization study will result in disjointed geometry"*. It could be beneficial as it would save time and the usage of computational power.

9.3 Post-Processing

As presented in this thesis, there are a variety of different post-production methods. The methods are separated to make it easier to explain and showcase the differences, but it is important to note that the different methods overlap and can be used in conjunction with each other. How they are utilized depends on the engineer and their work-process. Either way, the most important part of post-processing is that it results in a design that meets all the requirements and can be produced.

9.4 Validation of Topology Optimization Results

Validation of TO results is extremely important. If validation of the TO results is built in or the software contains a way to convert the results into a solid body, a lot of time could be saved. This could make it easier to validate which TO results is the best to use.

As for now, the **import smoothed mesh** tool in SolidWorks is problematic when it comes to simulation of the new part. However, it is possible to use rough estimate plots for displacement and stress within the TO study. These values seems to be rough, as they were completely different from the validation study of the post-processed part. This was mentioned early on and showcased in the case study of the drone beam. In addition, the calculated element mass was also far off from the actual value of the post-processed parts. There could be several reasons for these inaccuracies. Firstly, the calculations are based on the entirety of the model, which can contain porous elements. This can cause inaccuracies, and is also warned against by SolidWorks. Secondly, the remodeled designs is not exactly the same as the TO study suggested. It is merely a close representation of the TO results. This will cause some differences in the stresses and displacement.

The group did discover that it was possible to convert the TO results to a solid body while also validating the results in Ansys. This is not advised by the program, as the rebuild may result in failed reconstruction. However, if the reconstruction is successful, the following FEA should give more accurate results, as it does not contain porous elements. This offers a more reliable validation of the TO results, which can be very useful for the engineer.

If there are more iterations of a design from TO studies, new imported bodies could give indication of the stresses and displacements in the design. This could be very useful to quickly find the best design. Instead of post-processing and testing each iteration yourself, the program can do it for you. Then, only the best design will be needed to be post-processed and validated, before manufacturing the part.

9.5 Displacement Constraints as Topology Optimization Study Input

In the case study, displacement constraints was one of the inputs used for the TO study. Two different studies were done. The first was set to keep the displacement at **9.5 mm** and the other **1.4x** the maximum displacement from the FEA. These gave close to equal results. However, the resultant displacement was lower than the value set. Using these results as indication for a redesign, the validation study resulted in a **7.821 mm** displacement. This does display some weaknesses with the current state of TO software. This may be due to the porous elements used in SIMP.

9.6 Design Space

As demonstrated in **Chapter 7.2**, the utilization of design space plays an important in conjunction with TO. If the design space is not optimized, the TO results may not reach its true potential. Another relevant tool is the design study, which can be used throughout the design process. It has the potential to help further optimize the design. The question of when to utilize the design study depends largely on what is being designed. In the case of the drone beam, the design study was employed early in the process to find the best shape within the allocated design space. There is also a case to be made for using the design study after the TO, but applying it on a complex geometry might prove to be difficult and time-consuming.

9.7 Load Case 1

In load case 1, the design was altered a lot. The group figured there were better design possibilities regarding the stiffness and strength of the beam, than the design from Sevendof. The only restriction in redesigning the cross section was that it was no aerodynamic disaster. For that reason, beam designs were somewhat restricted. After finding the design that had the best mechanical properties, TO was introduced to make the beam as light as possible. After going through several iterations, the final design was created. The arm was split into two solid parts, which was necessary to make it producible. The final design ended up being lighter, stiffer and stronger than the initial design.

The lack of specifications in the case study made it difficult to find one optimal solution. As mentioned in **Chapter 8.8**, there are other possibilities regarding material and design. If another material was specified, the beam should go through optimization specified for the set material.

9.8 Load Case 2

It proved challenging to set up a simulation for load case 2. The linear dynamic study resulted in a maximum displacement of **77.10 mm**, and the nonlinear dynamic study of **42.57 mm**. Large displacements requires nonlinear studies, and is likely to yield a more realistic result. However, the group lacks expertise with these kind of studies, and will thus not draw any conclusions based on them.

Based on the applied loads, it is possible make some assumptions of how the drone will behave in the air. The horizontal forces that occur may result in rotational movement of the entire drone. This is illustrated on **Figure 9.1**. This could lead to the drone possibly spinning out of control. There is also the matter of the broken propeller blade. It could induce vibrations in the entire drone, which might lead to a crash. More complex studies on the entirety of the drone should be conducted.

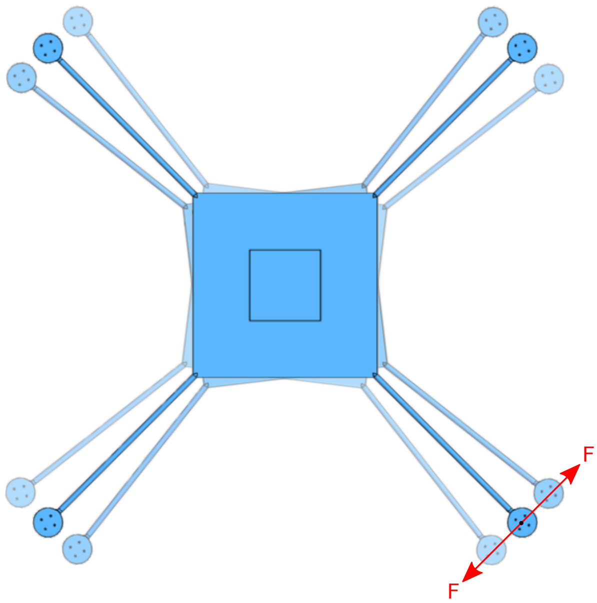


Figure 9.1: Possible movement of the drone with a broken propeller blade

One option to minimize the sideways displacement is to add supports to each beam on the drone. Adding supports will make the whole drone body stiffer, and the displacement will be lowered. **Figure 9.2** shows a suggestion on where the supports can be. This will have to be researched further. The disadvantage of adding supports, is that it adds extra weight to the drone.

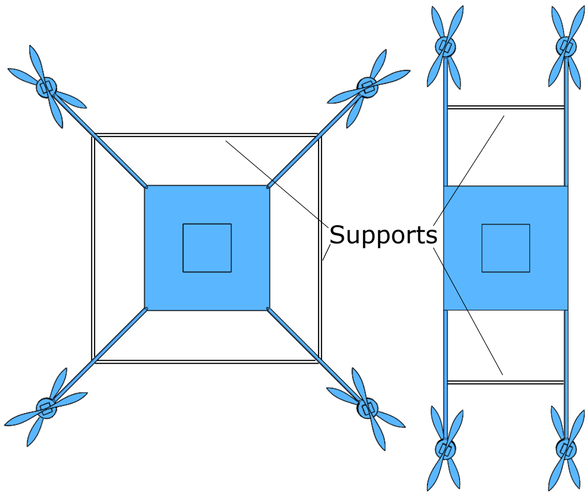


Figure 9.2: Supports for drone with different design

10 Conclusion and Further Work

This thesis has given an overview on the use of topology optimization for structural simulations. There has been conducted an apprehensive software comparison to pit three different TO algorithms against each other, to see which algorithm proved the most promising by the state of TO today. Traditional TO has also been compared to lattice optimization. Furthermore, research on post-processing techniques was conducted to find different ways of turning the raw TO results in to a usable model. The research on PP methods and design space were combined and applied to optimize a drone arm in collaboration with Sevendof. The finished model of the drone beam is **9.7%** lighter, and the displacement is **42.1%** less than the initial design.

In conclusion TO is a tool that has many different uses and advantages. By applying TO early in the design process, the time usage can be reduced and the quality of the design can be improved. TO offers a relatively quick analysis, which often results in great reduction of material mass in the design. A reduction in material means reduced cost which might lead to cheaper parts. It is also important to mention that in some industries weight directly affects energy usage, like in the aerospace and maritime industries. TO allows the engineer to depend more on facts and data rather than basing the design only on previous experience and guess work. This can lead to more optimized designs that better utilize the material without being over engineered. Based on our research, TO displays a lot of promise. However, there is still room for software based improvements.

Further Work:

Further work on the case study should be carried out in the following areas.

- Investigate if utilizing the design study method mentioned earlier in **Chapter 8.8.1** could lead to better and more optimized designs. Production methods should also be considered during this investigation as the designs might be better, but can prove harder to manufacture. Also, LO can be further explored in conjunction with the drone beam.
- Running simulations with the full drone to give more realistic results.
- Running a full non-linear dynamic study on load case 2. As the group was limited by computational power, a full simulation should be done to get the most accurate results possible.
- Consider the possible gains and challenges with producing a prototype and testing the two different load cases with it. This could prove or disprove the results from the simulations conducted in this thesis.

References

[1] Jarle Johannessen. *Tekniske tabeller*. Utg. nr 2. Oslo: Cappelen, 2002. ISBN: 978-82-02-16822-3.

[2] Harald Falck-Ytter. *Materialteknologi Del 1 Grunnlag*. Gyldendal Norsk Forlag AS, 2009.

[3] ESAB. *Understanding the Aluminum Alloy Designation System*. https://www.esabna.com/us/en/education/blog/understanding-the-aluminum-alloy-designation-system.cfm. (Accessed on 04/16/2019). 2019.

[4] Roger Lumley. *Fundamentals of aluminium metallurgy: production, processing and applications*. Elsevier, 2010.

[5] Christoph Leyens and Manfred Peters. *Titanium and titanium alloys: fundamentals and applications*. John Wiley & Sons, 2003.

[6] S.S. Bhavikatti. *Finite Element Analysis*. Daryaganj, INDIA: New Age International, 2004. ISBN: 978-81-224-2524-6. URL: http://ebookcentral.proquest.com/lib/ntnu/detail.action?docID=358028 (visited on 02/07/2019).

[7] Henry S. Valberg. *Applied Metal Forming: Including FEM Analysis*. New York, UNITED STATES: Cambridge University Press, 2010. ISBN: 978-0-511-72613-2. URL: http://ebookcentral.proquest.com/lib/ntnu/detail.action?docID=803211 (visited on 02/07/2019).

[8] HAWK RIDGE SYSTEMS ENGINEERING TEAM. *Curvature Based Mesh Advantages in SolidWorks Simulation*. https://hawkridgesys.com/blog/curvature-based-mesh-advantages. (Accessed on 04/10/2019). Nov. 2013.

[9] *2017 SOLIDWORKS Help - Mesh Control Parameters*. URL: http://help.solidworks.com/2017/english/solidworks/cworks/c_mesh_control_parameters.htm (visited on 02/22/2019).

[10] P. Kurowski. *Engineering Analysis with SOLIDWORKS Simulation 2017*. SDC Publications, 2017. ISBN: 9781630570767. URL: https://books.google.no/books?id=ONgBDgAAQBAJ.

[11] Irfan Zardadkhan. *When to use Nonlinear Analysis in SolidWorks Simulation?* https://www.javelin-tech.com/blog/2014/05/nonlinear-analysis/. (Accessed on 04/16/2019). May 2014.

[12] SOLID SOLUTIONS MANAGEMENT LTD. *SOLIDWORKS Simulation Capabilities | Solid Solutions*. https://www.solidsolutions.co.uk/solidworks/Simulation/Features/default.aspx. (Accessed on 04/16/2019). 2019.

[13] Tom, Ruen. *Hexagonal torus*. [Online; accessed April 2, 2019]. 2013. URL: https://upload.wikimedia.org/wikipedia/commons/4/4b/Hexagonal_torus.png.

[14] David S Richeson. *Euler's gem: the polyhedron formula and the birth of topology*. Vol. 6. Springer, 2008.

[15] TEDx. *The Shape of Things to Come | Jeff Murugan | TEDxTableMountain - YouTube*. https://www.youtube.com/watch?v=Ovwl-DoHHwA. (Accessed on 04/02/2019). June 2016.

[16] M.P Bendsøe and O Sigmund. *Topology Optimization: Theory, methods and applications*. Springer, 2003.

[17] Evangelos Tyflopoulos et al. "State of the art of generative design and topology optimization and potential research needs". In: *DS 91: Proceedings of NordDesign 2018, Linköping, Sweden, 14th-17th August 2018* (2018).

[18] M. P. Bendsøe. "Optimal shape design as a material distribution problem". In: *Structural optimization* 1.4 (Dec. 1989), pp. 193–202. ISSN: 1615-1488. DOI: 10.1007/BF01650949. URL: https://doi.org/10.1007/BF01650949.

[19] GIN Rozvany. "Aims, scope, methods, history and unified terminology of computer-aided topology optimization in structural mechanics". In: *Structural and Multidisciplinary optimization* 21.2 (2001), pp. 90–108.

[20] George IN Rozvany. "A critical review of established methods of structural topology optimization". In: *Structural and multidisciplinary optimization* 37.3 (2009), pp. 217–237.

[21] Razvan Cazacu and Lucian Grama. "Overview of structural topology optimization methods for plane and solid structures". In: *Annals of the University of Oradea, Fascicle of Management and Technological Engineering* (2014), pp. 1583–0691.

[22] Yi M Xie and Grant P Steven. "A simple evolutionary procedure for structural optimization". In: *Computers & structures* 49.5 (1993), pp. 885–896.

[23] Liang Xia et al. "Bi-directional Evolutionary Structural Optimization on Advanced Structures and Materials: A Comprehensive Review". In: *Archives of Computational Methods in Engineering* 25.2 (Apr. 2018), pp. 437–478. ISSN: 1886-1784. DOI: 10.1007/s11831-016-9203-2. URL: https://doi.org/10.1007/s11831-016-9203-2.

[24] Wikipedia, the free encyclopedia. *Level set method.* [Online; accessed March 21, 2019]. 2018. URL: https://upload.wikimedia.org/wikipedia/commons/7/78/Level_set_method.png.

[25] David Herrero Pérez. "Level Set Method Applied to Topology Optimization". In: (2012).

[26] Altair. *figure-15.png (659×242).* https://insider.altairhyperworks.com/wp-content/uploads/2018/05/figure-15.png?fbclid=IwAR1RKou1Ijd3jhf3r6k8WE97XzZN9CQOg8VgVGNAa8SQakRgmiQ-71_pto0. (Accessed on 05/20/2019). May 2018.

[27] Avinash Shukla, Anadi Misra, and Sunil Kumar. "Checkerboard problem in finite element based topology optimization". In: *International Journal of Advances in Engineering & Technology* 6.4 (2013), p. 1769.

[28] Serope Kalpakjian, Steven R. Schmid, and Vijay Sekar K. S. *Manufacturing engineering and technology / Serope Kalpakjian, Stephen R. Schmid*. Seventh. Pearson Education South Asia, 2014.

[29] D Brackett, I Ashcroft, and R Hague. "Topology optimization for additive manufacturing". In: *Proceedings of the solid freeform fabrication symposium, Austin, TX*. Vol. 1. S. 2011, pp. 348–362.

[30] Xiaodong Huang and Mike Xie. *Evolutionary topology optimization of continuum structures: methods and applications*. John Wiley & Sons, 2010.

[31] Xiaodong Huang and Mike Xie. *Software - RMIT University*. https://www.rmit.edu.au/research/research-institutes-centres-and-groups/research-centres/centre-for-innovative-structures-and-materials/software. (Accessed on 04/30/2019). 2010.

[32] XIE Technologies. *Home - Ameba*. https://ameba.xieym.com/. (Accessed on 04/30/2019). 2019.

[33] Associates Robert McNeel. *Rhino 6 for Windows*. https://www.rhino3d.com/. (Accessed on 04/30/2019). 2019.

[34] Dassault Systèmes. *Abaqus Unified FEA - SIMULIA™ by Dassault Systèmes®*. https://www.3ds.com/products-services/simulia/products/abaqus/. (Accessed on 04/30/2019). 2002-2019.

[35] Yi Min Xie Zhi Hao Zuo. "A simple and compact Python code for complex 3D topology optimizationn". In: *Advances in Engineering Software* (2015).

[36] Masaki Otomori et al. "Matlab code for a level set-based topology optimization method using a reaction diffusion equation". In: *Structural and Multidisciplinary Optimization* 51.5 (2015), pp. 1159–1172.

[37] SciArt Software,Inc. *SciArt Software, Inc | Rethinking Design*. https://www.sciartsoft.com/. (Accessed on 05/01/2019). 2018.

[38] Ted Harris. *How to Use Lattice Optimization in ANSYS Mechanical and ANSYS SpaceClaim 19.2*. https://www.padtinc.com/blog/the-focus/how-to-use-lattice-optimization-in-ansys-mechanical-and-ansys-spaceclaim-19-2. (Accessed on 05/01/2019). Dec. 2018.

[39] SandeepMedikonda. *Question about lattice optimization*. https://studentcommunity.ansys.com/thread/question-about-lattice-optimization. (Accessed on 05/05/2019). Mar. 2019.

[40] GoEngineer. *SOLIDWORKS Simulation - Topology Optimization - YouTube*. https://www.youtube.com/watch?v=iV2iMt-FB5s&fbclid=IwAR2eUbv8XgWZifD1_4qONq-YKO006-ZRP27sex3g635ftk5Nw46C4-ywr68. (Accessed on 05/18/2019). Aug. 2018.

A Problem Statement

Post-Processing of Topology Optimized Designs
Case Study of a Drone Arm

Topology optimization contributes in solving the basic engineering problem by finding the limited used material. Structural optimization reduces the material usage, shortens the design cycle and enhances the product quality. SO can be implemented according to size, shape, and topology. Topology optimization is usually referred to as general shape optimization (Bendsøe 1989). Most of the techniques optimize either the topology or both the size and the shape. On figure.

If TO is integrated into the traditional finite element analysis, the procedure can be divided to 8 steps as it is shown in Figure 1. This figure illustrates the geometry shift of a structure from its original geometry to topology geometry. In the beginning, FEA is implemented. It is possible to be used geometric modifications in order to simplify the initial problem. This stage is challenging to be computerized because it involves applying experience and judgement in a qualitative manner. However, the most crucial step at FEA is the definition of the problem statement and its equivalent mathematical model with all the required parameters (material properties, loads and restraints). The optimum results occur through the discretization (meshing) of the model and with a repetitive convergence method. The topology optimization method offers a new optimized design geometry with a notable mass reduction (or increment) which can be used as a new starting point for the FEA. Finally, the new FEA results validate or evaluate the success of the TO approach (Tyflopoulos *et al.* 2018).

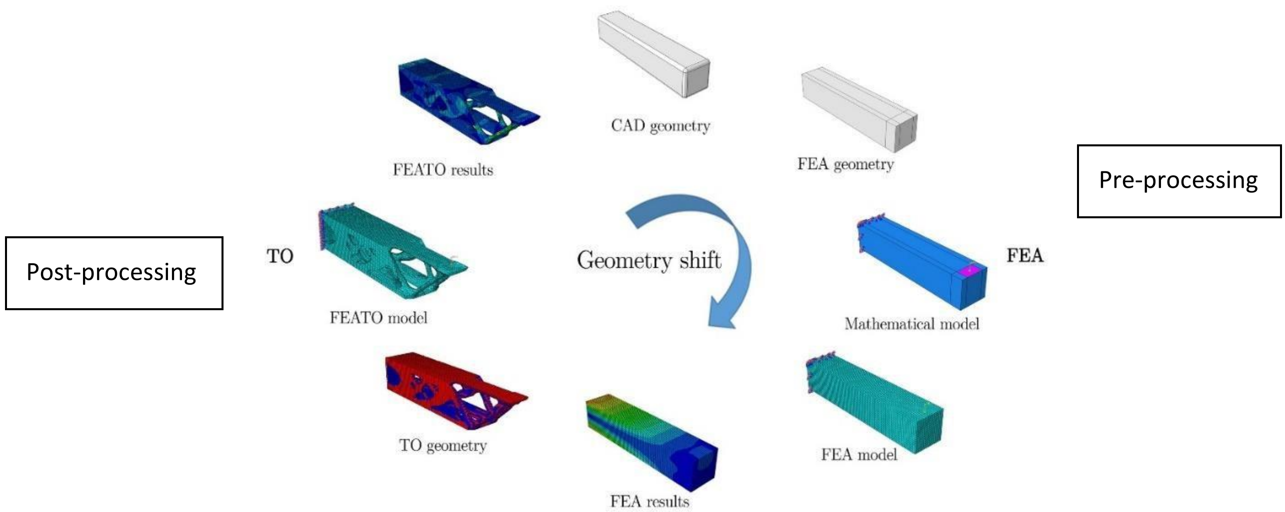


Figure 1: The geometry shift model of a cantilever beam with Abaqus (Tyflopoulos et al. 2018).

This bachelor thesis is focused on the post-processing of topology optimization methodology. The post-processing, as it is illustrated on Figure 1, consists of the interpretation of the topological optimized results, the re-design and validation of the structure with FEA (TO geometry-FEATO results). A case study of a drone arm will be used in order to support the theory and tie the academic text to a realistic application of topology optimization.

Tasks

- Make a literature research about the current state of the art of topology optimization and the implemented approaches.
- Make a comparison between software that uses different methods of topology optimization. Try to identify similarities and differences between them.
- Test the programs with models.
- Research different types of post-processing methods.
- Conduct research about using topology optimization on a drone arm and try to identify potential gains and challenges.

Comments

- The bachelor thesis will be written in English
- It is possible your data to be used for research purposes
- It is possible to be co-authors in a scientific paper if you want (after you will have handed in your bachelor thesis)

References

Bendsøe, M.P. (1989) 'Optimal shape design as a material distribution problem', *Structural optimization*, 1(4), 193-202, available: http://dx.doi.org/10.1007/BF01650949.

Tyflopoulos, E., Flem, D.T., Steinert, M. and Olsen, A. (2018) 'State of the art of generative design and topology optimization and potential research needs' in *DS 91: Proceedings of NordDesign 2018, Linköping, Sweden, 14th - 17th August 2018 DESIGN IN THE ERA OF DIGITALIZATION* The Design Society, 15.

B Popular Sience Article

Using Topology Optimization on a Drone Arm

Erlend Vatsvåg, Jens Ness, Kjetil Skorpen Mathiesen and Oscar Nicolai Løbak Sæther

[1] *Norwegian University of Science and Technology, Trondheim, Norway*

Today there are several different computer simulation programs that help with reducing weight and maintaining the strength of a design. Some of these programs utilize topology optimization. Research on these programs, the differences between them and how they work, have been conducted. This research was then put to use on a real life part, supplied by the Trondheim based start-up company Sevendof. They specialize in industrial service drones, which needs to be as light as possible. The part to be redesigned, was the arm of the drone, where the motor and propeller are mounted. The arm was redesigned, topology optimized, post-processed and then validated to meet the conditions set by Sevendof. In the end, the beam was roughly 10% lighter and bent 40% less than the original design.

1 Topology Optimization

Topology optimization is an up-and-coming and exciting technology that can be used in all sorts of applications. In general, the technology is used to reduce weight of structures, while maintaining acceptable strength and stiffness. There are many different methods and programs used for topology optimization. Some of these were tested and compared, to get an overview of the advantages of the different methods. Eight different programs were tested in total, some of which worked fine and others not so much.

The main focus was on what to do with the results generated by the program, post-processing, as the results normally come out rather rough. These results can contain sharp edges and shapes that can not easily be produced, and are in need of some work in order to be useful. In this work, several different methods to post-process the results was devised.

As the group wanted to test the methods, a collaboration with the start-up company Sevendof was established. They are developing industrial service drones and which needs to be as light as possible. They provided a starting design for the drone arm with all the information needed to use topology optimization on it. The project was chosen because of the technology, the possible uses and because it was exciting to the group.

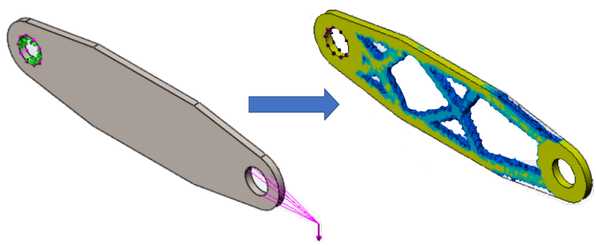


Figure 1: *A simple example of Topology Optimization*

In this example, shown in Figure 1, there is a small plate with holes in it. One hole is a fixing point, the plate can not move from this point. The other hole has a force working on it. The program then simulates how the plate reacts, and then calculates how much material it can remove without the plate breaking due to the forces. This is the basic topology optimization process.

2 Post-Processing the Results

As mentioned, post-processing the results is a significant part of the job. This was the main focus, where most of the research was done. The group devised several different ways to go about it, and while the methods are different they do tend to overlap when being used. Automatic post-processing methods were also discovered, but the results from these tools were not always perfect and may not work for all production methods.

3 Different Software

The programs tested ranged from open source scripts to commercial programs. Some were simple 2D versions, that allowed quick and easy testing of the different algorithms. Others were more complex 3D programs that could generate more elaborate designs. These were the main focus of the testing.

4 Drone Arm

The drone arm had a couple of requirements, such as how much it was allowed to bend, how tall it could be and general restraints on the design. It could not bend more than 10mm or be taller than 45mm. Several different profiles were tested to see which ones fit all the requirements. To reduce weight, material was removed from the inside of the arm.

Figure 2: *The arm cut in half*

5 A Promising Design Tool

While the topic has been researched for a long time, the technology allowing it to be practically used is rather new. It is becoming more and more popular as an engineering tool, and might become an important tool in the design process. As good as it is, it still has some issues. It is completely possible to get results that are not connected, leaving one part floating in space. In conclusion, it is a promising technology, but there is still room for improvements.

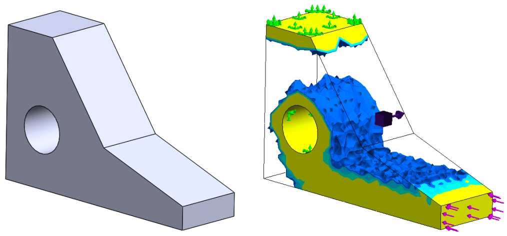
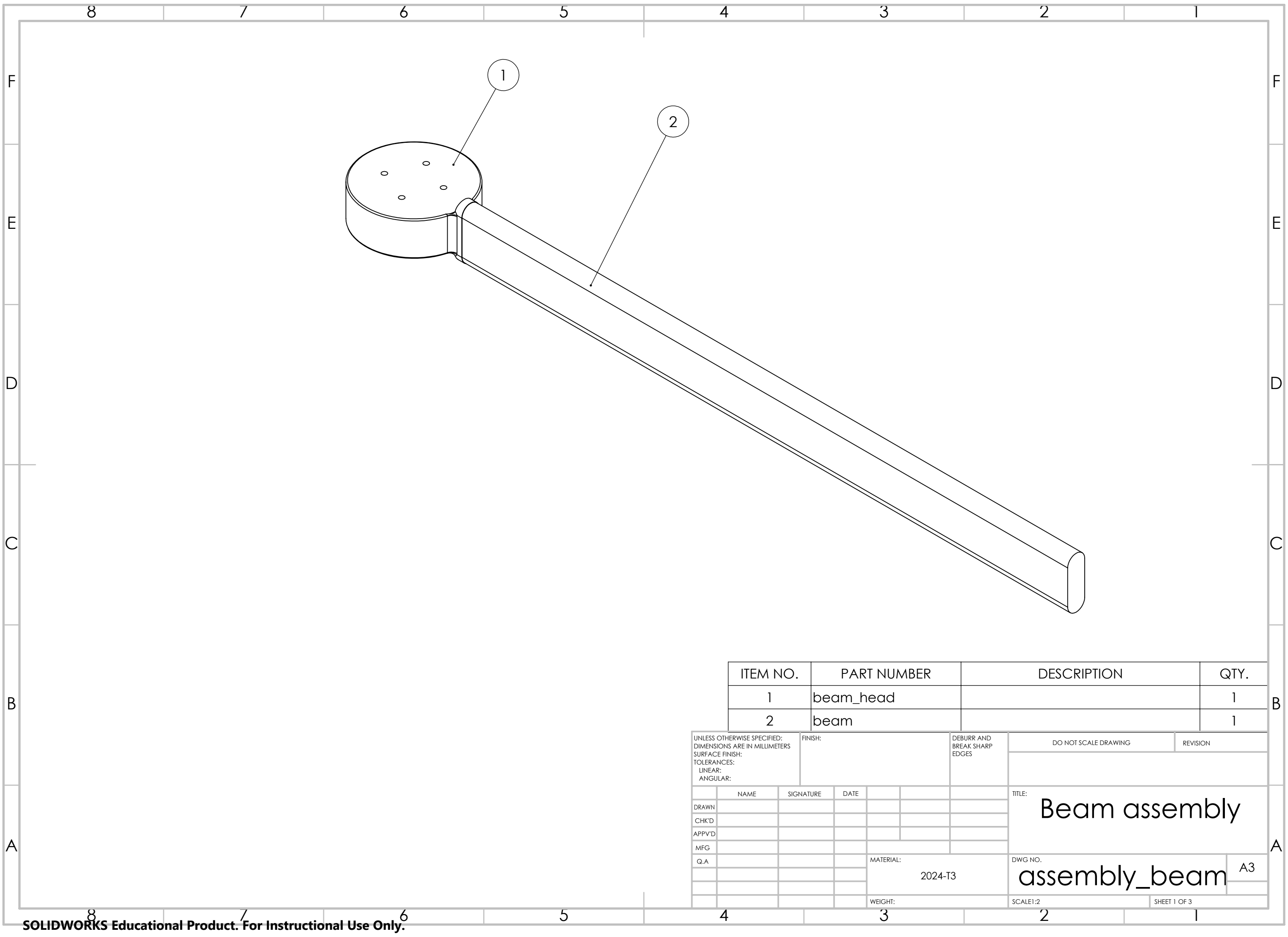
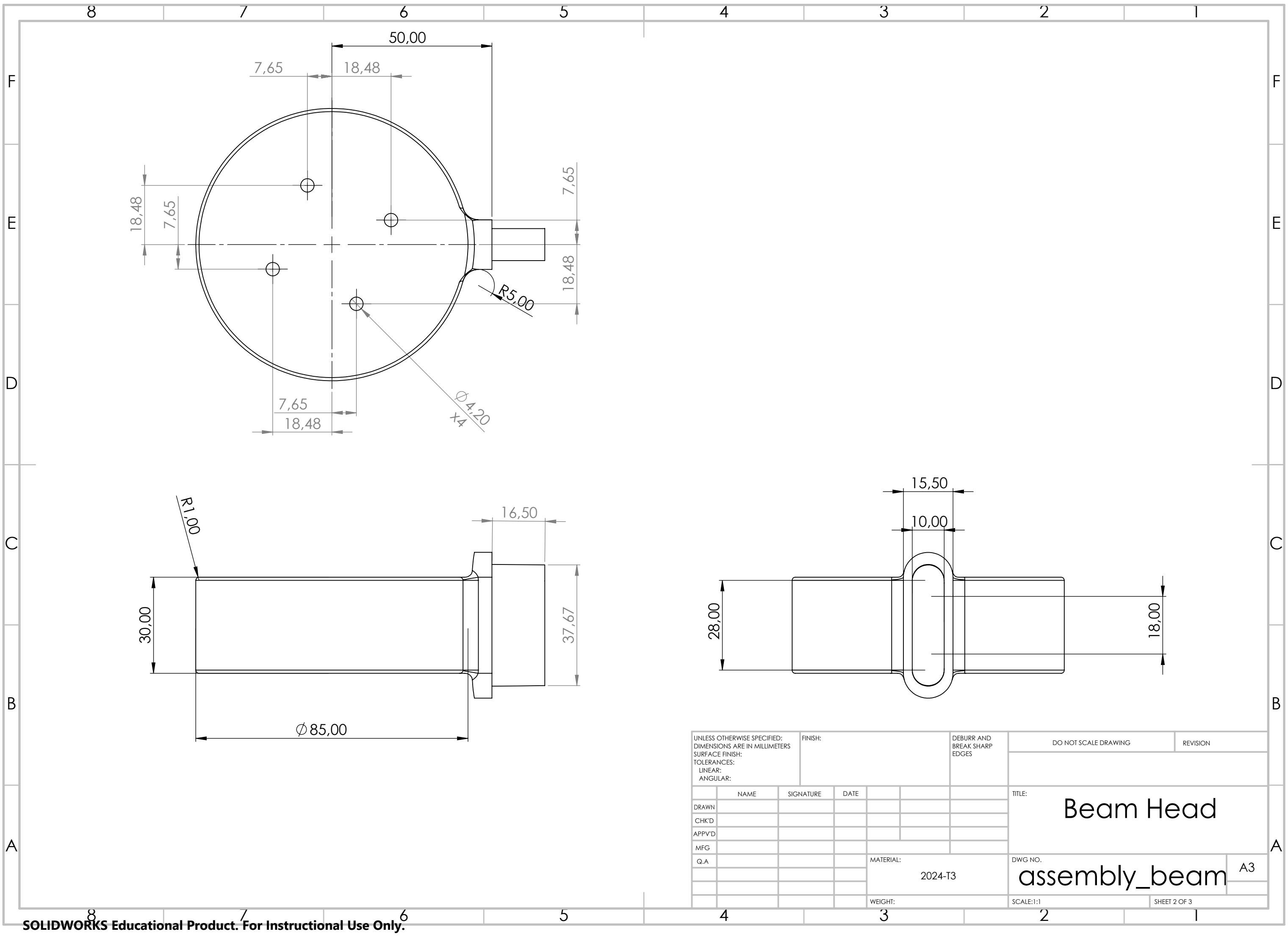
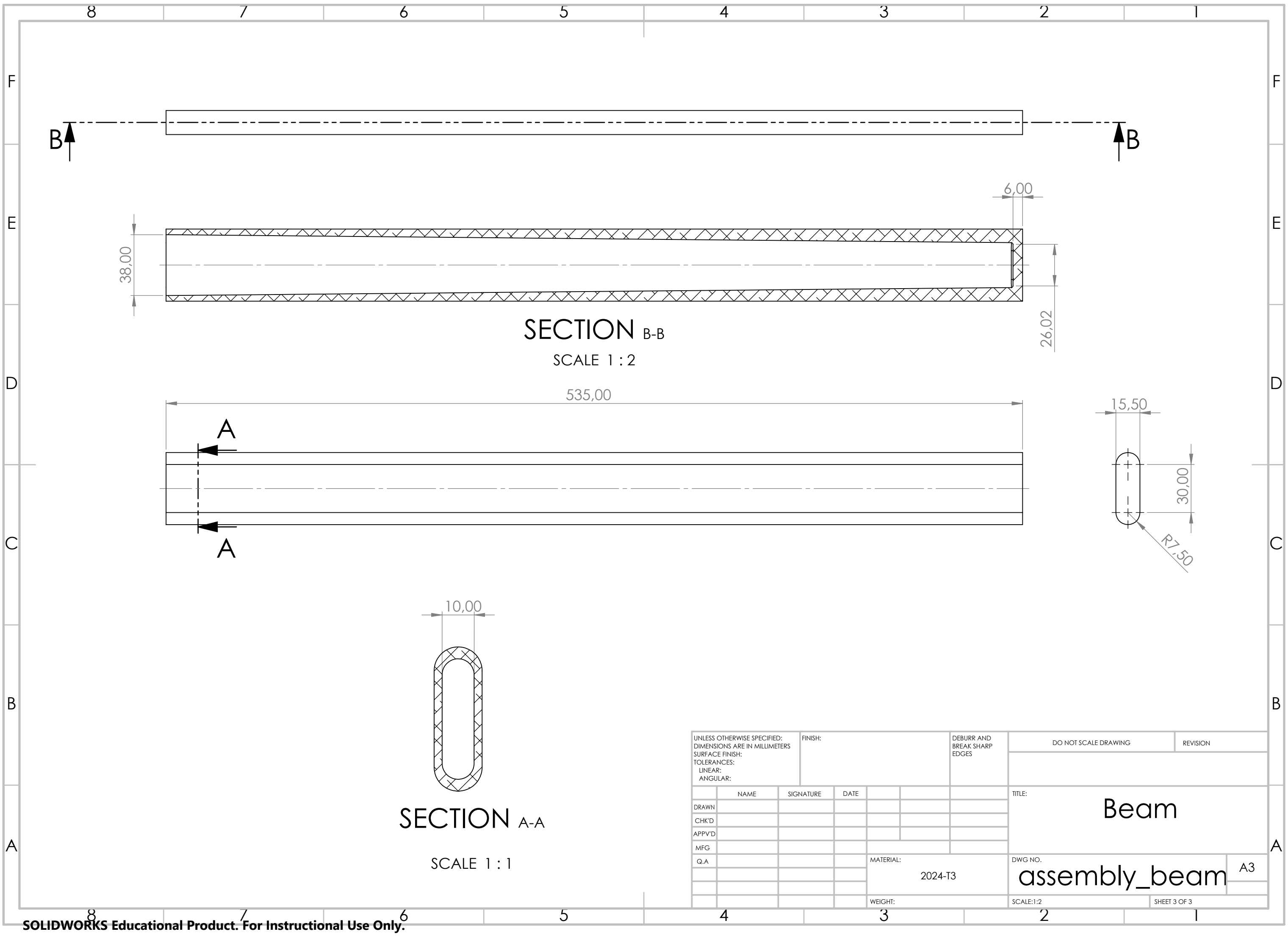


Figure 3: *A non-connected result*

C Technical Drawings




50,00
7,65
18,48
7,65
18,48
7,65
18,48
R5,00
7,65
18,48
∅4,20
x4
R1,00
16,50
30,00
37,67
∅85,00
15,50
10,00
28,00
18,00
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:
FINISH:
DEBURR AND BREAK SHARP EDGES
DO NOT SCALE DRAWING
REVISION
NAME
SIGNATURE
DATE
DRAWN
CHK'D
APPV'D
MFG
Q.A
TITLE:
Beam Head
MATERIAL:
2024-T3
DWG NO.
assembly_beam
A3
WEIGHT:
SCALE:1:1
SHEET 2 OF 3
SOLIDWORKS Educational Product. For Instructional Use Only.


B
B
6,00
38,00
26,02
SECTION B-B
SCALE 1 : 2
535,00
A
A
15,50
30,00
R7,50
10,00
SECTION A-A
SCALE 1 : 1
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:
FINISH:
DEBURR AND BREAK SHARP EDGES
DO NOT SCALE DRAWING
REVISION
NAME
SIGNATURE
DATE
DRAWN
CHK'D
APPV'D
MFG
Q.A
TITLE:
Beam
MATERIAL:
2024-T3
DWG NO.
assembly_beam
A3
WEIGHT:
SCALE:1:2
SHEET 3 OF 3
SOLIDWORKS Educational Product. For Instructional Use Only.

D Load case from Sevendof

“Topological optimisation of drone arm.”

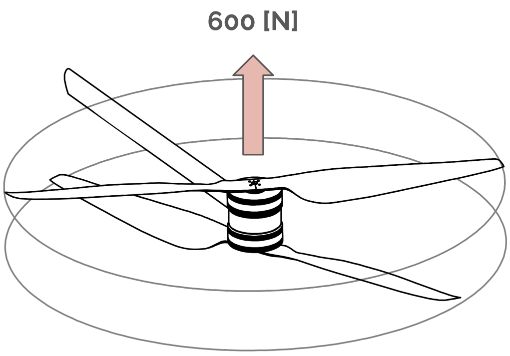
1st meeting for specifying problem and cooperation framework
Trondheim 2019-01-16

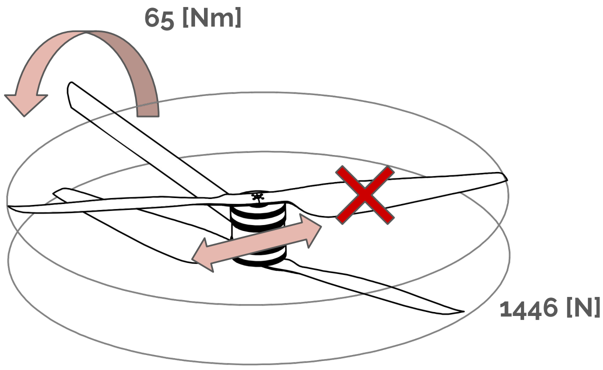
sevendof NTNU

Problem Specification

Load Case I:

- Load from MAX thrust
- Optimisation driven by Stiffness
  - Assumption - MAX deflection **10 [mm]**
  - *Risk of conflict between pusher propeller and the beam*

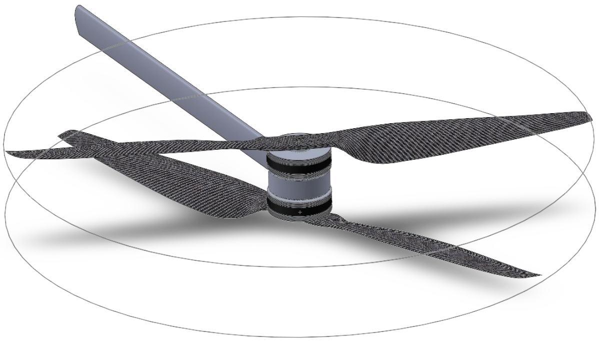


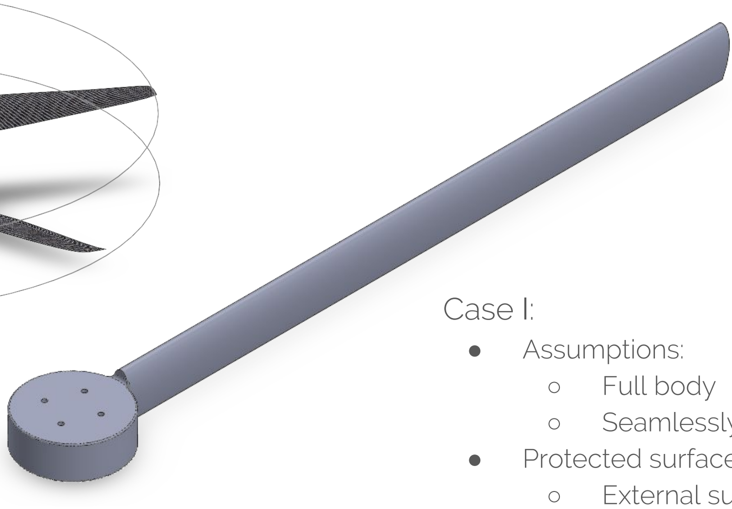


Load Case II:

- Load from broken propeller blade
- Introduce momentum from reaction forces (at 4000 RPM)
  - Beam Twist momentum **65 [Nm]**
  - Beam Side force **1446 [N]**

Initial Geometry - Case 1





Case I:

- Assumptions:
  - Full body
  - Seamlessly fixed to main body
- Protected surfaces:
  - External surface of the beam
  - Motor fixing points