

Topology optimization of a mounting bracket

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<p>Sammandrag:</p> <p>Detta examensarbete presenterar den topologiska optimeringsprocessen av ett fästelement som optimerades med solidThinking Inspire och SolidWorks. Topologisk optimering kan tillämpas då man vill förbättra en existerande geometrisk form eller producera en ny form som bör vara inom en förhandsbestämd storlek, ha en bestämd massa och klara av krafter som den utsätts för. Optimeringen fungerar genom att forma om ett arbetsstycke samtidigt som vikten minimeras eller maximeras. Målet var att minska fästelementets massa med 70 % med en Factor of safety på 1.2. Det slutliga resultatet visade en lyckad optimering med båda programmen, där massan med Inspire reducerades med 66 % och 67 % med SolidWorks. Den maximala mekaniska påfrestningen ökade från 145 MPa till 422 MPa och Factor of Safety hölls mellan 1 och 6. På grund av error i SolidWorks lyckades inte det optimerade fästelementet analyseras efteråt för en jämförelse. Alla tre fästelement 3D Printades med Arcadas MakerBot Replicator 5 för visuell demonstration.</p>	
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<p>Abstract:</p> <p>This thesis presents the topology optimization process of a mounting bracket using solidThinking Inspire and SolidWorks. Topology optimization can be used whenever you need to improve an existing design or design a brand-new part which needs to fit in a specific space, be lightweight and withstand certain forces. It works by taking a block of material and shaping it by removing material, minimizing or maximizing part mass, displacement or compliance to fulfill set limits such as maximum stiffness or amount of deformation allowed. The reduced weight tied to cost benefits provided by topology optimization can be seen in various industries.</p> <p>The goal was to reduce 70 % of the original weight. After optimization the achieved reduction in weight was 66 % using Inspire and 67 % using SolidWorks. The maximum stress was increased from 145 MPa to 422 MPa using Inspire. Factor of safety was between 1 and 6. Due to frequent errors in SolidWorks no post FEA- analysis could be performed for comparison.</p> <p>Each part was 3D Printed using Arcadas MakerBot Replicator 5 for visual demonstration.</p>	
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ABBREVIATIONS

CAD	Computer Aided Design
FEA	Finite Element Analysis
TO	Topology Optimization
SIMP	Solid Isotropic Microstructure with Penalization
BESO	Bi- Directional Evolutionary Structural Optimization
CNC	Computerized Numerical Control
RMB	Right Mouse Button

FOREWORD

This thesis presents the topology optimization process. It starts with an introduction presenting areas of use, where industries are using it and how the method works. This is later followed by a practical example in which a mounting bracket undergoes the topology optimization process using two different software.

1 INTRODUCTION

1.1 Background

Topology optimization is a mathematical approach for optimizing parts and structures. Introduced in the late 1980s it is today used in the automotive-, aerospace-, marine- and biomedicine industry. It works by optimizing the material distribution in a given design space with a given set of loads, boundary conditions and constraints with the final goal of maximizing the part strength while minimizing weight. (Gould, 2015)

It has gained popularity in the past years due to increased availability in Computer Aided Design- software being both easy to use and affordable. The topology optimized designs have efficient material use and can be optimized for manufacturing techniques such as casting and 3D Printing.

The resulting designs often have biological-like shapes resembling shapes found in nature such as skeletal, honeycomb and porous shapes containing curvy surfaces with bends, twists and voids. (Gould, 2015)

The topology optimization process is linked to and follows the creation of a part which is done using a Computer Aided Design (CAD) -process.

1.2 Objectives

- Demonstrate the topology optimization- process
- Topology optimize a mounting bracket using solidThinking Inspire and Solid-Works
- Compare the weight reduction and difference in stress to the original mounting bracket
- Compare the geometrical differences between each bracket
- 3D Print both optimized brackets for visualization

1.3 Relevance to industry

The traditional design process of a mechanical part involves designing a part using computer aided design- software and proposing it to a structural engineer who analyses it using finite element analysis (FEA). A first prototype is produced and mechanically tested. The design is modified or changed completely based on the test results. The new/modified design is tested again and if it does not meet requirements the process is repeated. This type of process may be time consuming and the end result may not be the best possible due to the many modifications not working together optimally. (Olason & Tidman, 2010)

What makes topology optimization efficient is the automated process generating conceptual designs that replace the traditional trial-and-error design process.

It is applicable to many different design problems because of algorithms dictated by different restraints taken from solid mechanics, fluid dynamics and thermal dynamics. The algorithms help engineers think outside of the traditional design ideas and generate completely new ideas and designs, even for well- engineered products. (Liu & Yongshen, 2016)

1.4 Relationship to existing knowledge

The thesis brings theory of topology optimization from relevant sources of information including master-, and bachelor theses. Theory used in connection to the practical part is gathered from online tutorials and examples using the solidThinking Inspire optimization tool and the topology study- tool found in SolidWorks.

2 LITERATURE REVIEW

In this part the basic process of topology optimization will be explained together with solid modelling and surface modelling. The different software used in the optimization process is presented and explained.

2.1 Computer Aided Design (CAD)

2.1.1 Solid Modelling

A CAD- model, also known as a Solid Model, acts as a starting point for the topology optimization process.

A solid model is created by 2D drawings/ sketches of different views of an object. These different views representing the top, sides and bottom work to describe the 3- Dimensional object. Solid modelling works by transforming these 2- Dimensional views into an electronic 3D model. The 3D model has a volume, boundary surface, holes, and hidden voids. Solid models can be rotated for inspection and even intersected to view their inner structures. (Pechter, n.d.)

Every operation done in the CAD- model will be shown in a feature tree including the sub- components and their relations which make up a single feature. The feature tree can be interpreted as a tree data structure where the individual parameters used to determine geometries are small leaves while the geometries are larger branches making the object. The feature tree gives an easy access to features and their components allowing easy modifications of the CAD model such as removing/ adding features, modifying parameters and optimizing the model. However, when inserting new sub-branches into the feature tree the inter-structural relations need to be considered in order to leave a stable, error free CAD model.

Solid modelling creates solid objects meaning the model is “watertight” with the internal details included. Individual parts can be combined into a whole creating an assembly. (Deshmukh, 2016)

Solid models are used to record manufacturing data such as weight, volume and cost of production. (Hudspeth, 2016)

2.1.2 Surface Modelling

Surface modelling lets the user create more complex shapes. It focuses on the external of the object and unlike solid modelling leaves a “non-watertight” hollow shell. A surface is stretched over a 3D curve created by the user according to the surface boundaries. Surface modelling allows for more free- flow shapes that can be both open and closed. Making changes to the design in surface modelling can be difficult since the changes are not defined by parameters, making a change in one area not updating the whole design.

Surface modelling represents the skin of an object. These skins have no thickness or material type. (Deshmukh, 2016)

The process of surface modelling generally starts by defining profiles of the part. The profiles may consist of several lines, arcs, splines and fillets which are made with tangent, contact or curvature continuity. The profiles created with the 3D sketch tool are drawn on surfaces intersecting the model. The surfaces are created using surface tools and defining the surface using the profiles as guides and boundaries. (Dvorak, 2003)

2.1.3 CAD software

In general, there are two kinds of 3D Design software, CAD and 3D Modeling. A Computer Aided Design (CAD)- software is usually used to create industrial and mechanical objects. On the other hand, 3D modeling software gives more freedom to design artistic models which are not restricted in form or function by design parameters. In the past 3D

modelling software have been used in the film and game industries to produce various animations and designs. However, they can also be used to create 3D Printable objects. (Carlota, 2019)

CAD software can be used for specified outcomes. The software can be tailored to a certain goal. It is a technical tool which has functions in industrial design, mechanical design, architecture and aerospace industries. A CAD- model contains vital information of the part regarding material, dimensions, tolerances and manufacturing process specific information. In addition, many CAD- software are constantly improved and updated adding features such as advanced rendering and animation capabilities for better product visualization. As an example, when designing a part in CAD you can quickly prepare it for 3D Printing by saving it to STL- file format, something which has become a standard in all major CAD - software. (Carlota, 2019)

Listed are some of the leading software which are used in various tech- industries. The basic CAD modelling can be performed with all of them including parametric modeling, finite element analysis, technical visualization and illustration.

Table 1. Comparing different CAD software

Software	Developer	Characteristic features	License
Creo	Parametric Technology Company	Very capable, sets a broad range of features. Seen as an industry standard. [1]	Free 30-day trial, Free 1- year student license, 1- year standard license 2390 \$ [1]
FUSION 360°	Autodesk	Cloud based platform, stores all files on the cloud. Users can access their files almost anywhere. Organic geometry, surface modeling, user friendly . Lacks more advanced simulation features such as manufacturability. [5]	Free 3-year student license, 1- year standard license 495 \$, Free 1-year license for Startups [6]
SolidWorks	Dassault Systèmes	Robust simulation tools, manufacturing tools. User friendly, short learning curve. Accessible. [3]	Free 1-year student license, Permanent license 3995 \$, Annual subscription 1295 \$/ year [4]
AutoCAD Inventor	Autodesk	Provides professional grade 3D mechanical design and manufacturing tools. Geared towards professionals.[12]	Free 3-year student license, 1- year subscription license 1985 \$ [11]
CATIA	Dassault Systèmes	Seamless cooperation between different departments, used in automotive, aerospace industry to simulate products and manufacturing processes. Concept design to Capable to simulate the products entire manufacturing process. Complex for beginners, steep learning curve. [7]	Yearly lease 4500 \$ / one time purchase 11 200 \$ / annual maintenance of 2000 \$ [8]
Rhinoceros	Robert McNeel & Associates	Used in industrial design, architecture, product design and 3D Printing. Efficient at editing surfaces, create organic shapes and surface modeling. Hard to re- modify models after completion. [9]	Educational 195€, Full license single user 995€ [10]

[1] (Carlota, 2019), [2] (Novedge, 2019), [3] (Cali, 2019), [4] (Copmuter Aided Technology, n.d.), [5] (Ostrem, n.d.) [6] ((Autodesk, 2019), [7] (Aaltonen, 2013) [8] (Giarratana, 2017), [9] (William, 2013), [10] (McNeel & Associates, 2019) [11] (Autodesk, 2019), [12] (Ooin, n.d.)

2.2 3D Printing

3D Printing works by slicing a digital model in layers and adding material layer by layer thus building it up into a physical part. Because of the additive technique, 3D Printing is also known as Additive Manufacturing. The 3D Printing technique differs from traditional manufacturing techniques such as machining which removes material from a work-piece. During 3D Printing no special tools are required, instead the part is directly manufactured on the build platform. (3D HUBS, 2018)

3D Printing always starts with a 3D model which acts as the blueprint. The model is sliced by the program into many thin 2-dimensional intersections/layers. Each layer is assigned a specific G-code of x-, y- and z-directions which act as coordinates for the printer to follow. Although the basic principle of building up the model layer by layer is the same in all 3D printers, what differs the different techniques is the way they layer the material. For example, printers using the Fused Deposition Modelling- technique (FDM) melt plastic filaments through a metal nozzle and places it on the build platform. Another method called Selective Laser Sintering (SLS) uses a laser which melts the powder- form material, reapplying a new layer of material prior to sintering it. Plastics are the most common materials however metals can also be used. Printed objects can have a variety of properties. They can be a combination of many different plastic materials with different properties such as rubber or clear plastic. Printed objects often require post processing such as sanding, painting or hardening to reach desired functionality and/or surface quality however this prolongs the total production time. (3D HUBS, 2018)

3D Printing has established a specific role in the area of manufacturing. Previously it used to be a niche market targeted for professional use however today it is used by both professionals and hobbyists. Every year new 3D Printer- models are released to the market.

The benefit of 3D Printing is its ability to produce complex, natural shapes which are not achievable by other manufacturing methods. More complex shapes do not cost more to print compared to simple parts. Since a 3D Printer is the tool and does not require additional investments one can relatively quickly setup production and produce complex parts

for a fraction of the cost it would take to reach the equivalent result using another manufacturing method. 3D Printing is also a very flexible production method since all parts can be customized according to specific needs without having to change tools/machines reducing the production costs.

Since it is not intended for mass manufacturing it is today mostly used for prototyping purposes. In this area 3D Printing is the preferred method since parts printed are usually ready overnight. A project idea to physical product lasts today between 8-10 weeks when it previously used to take over 8 months.

Printed parts can today have specific properties such as high heat resistance, biocompatibility, high stiffness and strength. (3D HUBS, 2018)

3D Printed parts are in general weaker than equivalent parts produced with traditional manufacturing methods.

According to 3D HUBS, “Generally, 3D printed parts have physical properties that are not as good as the bulk material: since they are built layer-by-layer, they are weaker and more brittle in one direction by approximately 10% to 50%”. However, parts with excellent mechanical properties can be printed using Direct Metal Laser Sintering (DMLS) and Selective Laser Melting (SLM) which have found applications in demanding industries such as the aerospace industry. (3D HUBS, 2018)

The surface quality of the printed part depends on the accuracy and calibration of the machine. Generally, FDM machines have the lowest accuracy and print with +/- 0.5 mm tolerances. This means printing a hole which is 20 mm in diameter in reality is between 20.5 mm and 19.5 mm in diameter. Other printing techniques have higher accuracy, as an example Stereolithography 3D- printers have an accuracy between +/-0.01 mm and PolyJet machines have accuracies which can be measured in micrometres. Metal parts requiring specific accuracy are often machined after printing to achieve desired tolerance. (3D HUBS, 2018)

The primary 3D Printing techniques are as follows:

- FDM (Fused Deposition Modelling), material in filament form is dispensed through a heated nozzle.

- SLA (Stereolithography) Liquid polymer material is cured using UV- light
- SLS (Selective Laser Sintering) Polymer in powder form is heated up using a laser just below melting point. A thin layer of new material is placed on top and the laser sinters the selected area. This process is repeated.
- DMLS (Direct Metal Laser Sintering) Powder metal particles are fused together using a high energy heat source such as a laser. (3D HUBS, 2018)

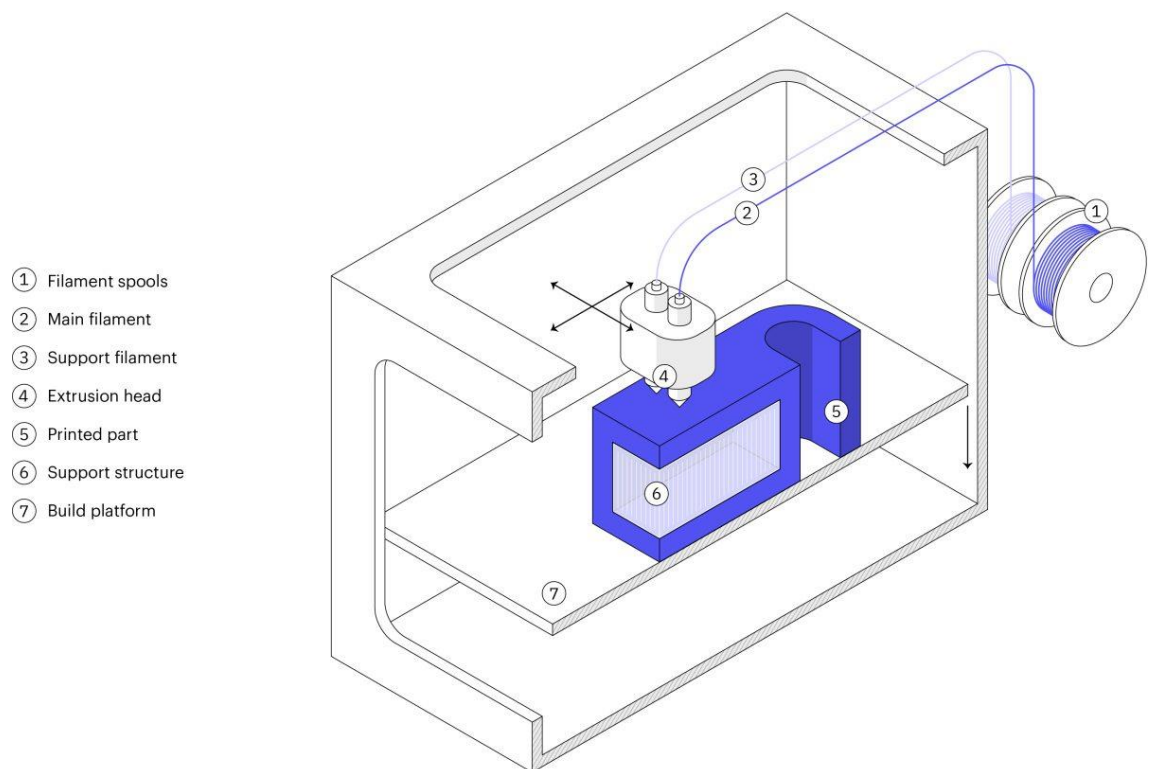


Figure 1. FDM- Printing technique (3D HUBS, 2018)

2.2.1 Direct Metal Laser Sintering (DMLS)

Engineers in the aerospace- industries use 3D Printing of metals to produce high performance parts. In these fields the capabilities provided by 3D Printing is beneficial when producing topology optimized parts with good strength to weight ratio.

DMLS is used to print metal parts with complex shapes which are not achievable using traditional manufacturing methods. Parts produced using DMLS have excellent physical properties, often even stronger than the equivalent traditionally produced part. Metal alloys which are difficult to process can be used in DMLS. However, parts produced with

DMLS generally have very high prices and can cost between 4500 € to 23 000 €. Therefore, DMLS should be used on parts which cannot otherwise be produced. (3D HUBS, 2018)

Topology optimized parts, if 3D Printed would most likely be produced using the DMLS-technique to retain similar material and part properties to the original part.

2.3 Topology Optimization Description

Generally, parts are designed and improved based on older designs or concepts. Therefore, the design is already pre- determined and has defined parameters. In case there is no design you would usually start with a couple of conceptual designs, define their parameters according to the environment and apply existing product development methods. Another method to work on a completely new design is to start with a material block and let the optimizer shape and size the block of material. This method of optimization is known as topology optimization. Topology optimization can be used whenever you need to design a brand-new part or improve an existing one which needs to fit in a specific space, be lightweight and withstand certain forces.

Topology optimization works by taking a block of material and shaping it by removing material, minimizing/maximizing its mass, displacement or compliance to fulfill constraints such as displacement, minimum element size, depending on what the user wants. (Hale, 2017)



Figure 2. Standard part on the left, topology optimized version on the right (bct-technology, 2017)

2.4 Discretization

Topology optimization is a form of structural optimization. Its purpose is to seek the optimal distribution of material. The design space is discretized into a grid (mesh) of finite elements called isotropic solid microstructures, using finite element methods (FEM). (Dassault Systemes, n.d.)

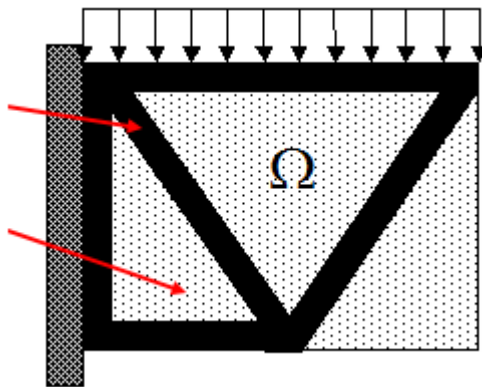


Figure 3. Solid (black) and empty (void) elements (Dassault Systèmes, 2019)

This discretization is performed to make the functions, variables, models and equations suitable for numerical evaluation and implementation in computers. Optimization methods are used to solve the problem, showing which elements are voids and which are solids in the mesh. This is an integer problem with each element having two possible stages, either 0 or 1. The number of elements makes the problem impractical to solve since the

number of possible combinations is 2^N where N = number of elements. The two main solution methods to solve this problem include the density method (SIMP) and the homogenization method. This thesis will focus on the SIMP method (Olason & Tidman, 2010)

2.5 Density method - Solid Isotropic Microstructure with Penalization method (SIMP)

The SIMP method predicts the optimal material distribution in a given design space, for given loads, boundary cases, manufacturing limits and requirements in performance. According to Dassault Systèmes, “The introduction of a continuous relative density distribution function avoids the binary, on-off nature of the problem. For each element, the assigned relative density can vary between a minimum value ρ_{\min} and 1, which allows the assignment of intermediate densities for elements (characterized as porous elements): ρ_{\min} is the minimum allowable relative density value for empty elements that are greater than zero. “

Any densities which fall between it and 0 are automatically made into voids, $p = 0$. This density value insures less errors in the finite element analysis.

Because the relative material density, p_e , continuously changes for each element means the Young’s Modulus of the material at each element can also continuously change. Each element, e , has a relation between the relative material density factor, p_e , and the Youngs modulus of the assigned isotropic material, E_0 according to:

$$E(p_e) = p_e^p E_0$$

Where:

$E(p_e)$ = stiffness tensor / local stress strain relation

p_e = material relative density factor

p = penalty factor value

$E_0 =$ Youngs modulus of assigned isotropic material

(Dassault Systemes, n.d.)

Relative stiffness as a function of density using different penalization factors

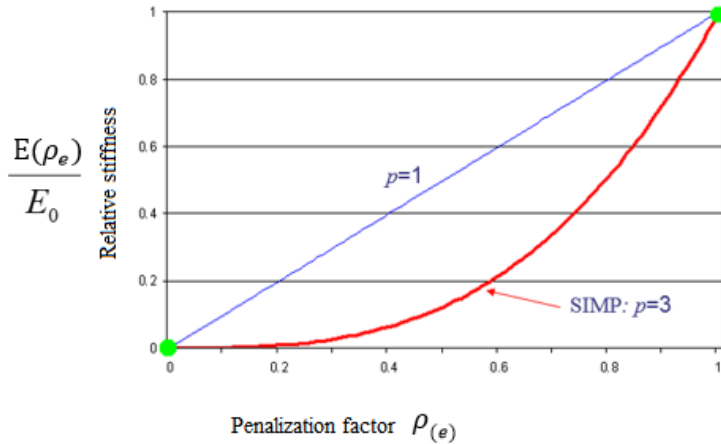


Figure 4. The relative stiffness changes according to the used penalization factor (Dassault Systèmes, 2019)

The use of the penalty factor, p , decreases the effect elements with intermediate densities have on the total stiffness. The penalty factor optimizes the solution to concern elements which are either solid ($p_e = 1$) or voids ($p_e = 0$). For materials with poisson ratio $\nu = 0.3$. it is recommended to use a penalty factor of $p = 3$. (Dassault Systemes, n.d.)

SIMP- method is the standard in topology optimization. As an example, the finite element design- tool TOSCA uses the SIMP method. It Is widely used in the industry mainly because it integrates manufacturing restrictions which allows for the creation of an optimized model suitable for manufacturing. (Fiebig, et al., 2015)

2.6 Checkerboarding problem

In an optimal solution, elements are either 100 % or 0 % filled. However, a problem of the SIMP method is the resulting checkerboard problem in the optimized model. The checkerboard problem consists of a periodic pattern of 0 and 1 density elements in a checkerboard pattern which does not conform to the optimal distribution of material. This distribution of filled and void elements can be classified as a third intermediate material which has not been penalized. This problem is caused by errors in the finite element formulation where the stiffness of elements connected to each other in the corners is exaggerated. The checkerboard pattern was first believed to be optimal microstructures but later proven to be numerical noise. (Zhou, et al., 2014)

Since then, filtering techniques have been developed designed to average the element densities with respect to surrounding elements. These filters make smoother density transitions between elements, leaving more well-defined edges. (Pedersen, et al., n.d.)



Figure 5. Topology optimization with and without checkerboard filter (Wang, n.d.)

2.7 SIMP versus Homogenization method

The homogenization method was one of the earliest methods of topology optimization. The homogenization method step-by-step faded away and was surpassed by the more efficient and simpler SIMP method. The SIMP method is simpler because it uses fictitious isotropic material representations while the homogenization method relies on the true composite material which can be anisotropic. When penalization of intermediate densities is put into action there is no need to use the true composite materials, which have more complicated properties and require more modeling and computational effort than the simpler SIMP-method. Therefore, SIMP is the most common method used in commercial topology optimization programs. (Allaire, et al., 2018)

2.8 BESO, Bi- directional Evolutionary Structural Optimization method

The BESO- method in single material distribution problems uses something called a “hardkill” method to remove voids from the structure. (Ghabraie, n.d.)

The Young's modulus of each element in the finite element model is assigned close to zero while still retaining part performance. The elements showing no stress/load bearing are eliminated. This iterative process is continued until only elements with stress are left. (Larsen & Jensen, 2009)

2.9 Finite element modelling- setup

Topology optimization requires a finite element model which consists of triangular faceted surfaces. These are referred to as tessellation objects and approximate the boundary of the optimal part. The current standard is to have the designer create a parametric CAD model which approximates the tessellation and works as the boundary volume for the optimized part. (Larsen & Jensen, 2009)

The mesh in the created finite element model may have inconsistencies which can be gaps in the model surface or overlapping of surfaces. These may be fixed by running a simplification tool which resolves these errors. (Cuillière & Francois, 2014)

This boundary or volume of space occupied by the CAD model/design domain is important to the creation of an optimal design since the optimizing algorithm may only change the density of elements which are present from the start. Therefore, it is critical to maximise the design domain to increase the number of possible configurations. Applying topology optimization on the parametric CAD model requires defining design and non-design space. When optimizing a part there are always surfaces/features in the part which are in relation with other components, such as joint, mounting holes and fixtures and thus the interface cannot be modified.

The material in these areas are referred to as non-design space. These areas can be volumes of the part which are critical to the operation and need to be a certain shape. Interfaces which need to be compatible with other surfaces and volume of space where other parts may operate while in use. (Cuillière & Francois, 2014)

Consequently, the area/volume which is intended to be affected by the optimization algorithm is called design space. (McLennan, 2015)

2.10 Topology optimization setup

After creating a CAD model and having specified design- and non-design space, the topology optimization is defined. An algorithm is chosen, being SIMP, BESO or something else, if not already predefined by the program. The type of optimization and boundary conditions are defined according to part specific needs. For example, assigning part material, a maximum/minimum percentage of weight or volume reduction, strain energy limits. Adding method of manufacturing to make the end result more suitable for manufacturing.

2.11 Post processing

After running the optimization an optimized parameter file is created. An iterative process of smoothing is to be done to reduce sharp edges caused by removing individual elements from the design space. (McLennan, 2015)

2.12 Examples of topology optimized parts

Topology optimization in industry is focusing on improving existing results. The main target is to reduce weight while retaining or improving part strength. The optimized part in combination with a reduced weight results in lower manufacturing costs by either:

- Reducing the amount of assembled parts, requiring less assembly work
- Cutting down the amount of material needed thus lowering material costs
- A combination of both

Another important aspect is saving time in the development process. Below are some examples of industry optimized parts which support the above statements. (Fiebig, et al., 2015)

2.12.1 Door support for Fairchild Dornier 728-100 aircraft

This door support optimized by Eurocopter was structurally optimized to reduce weight while retaining stiffness. The part weight was reduced by 20 % from 9.15 kg to 7.5 kg while the design time was cut from 3 months to 3 weeks.

(Altair University, 2019)

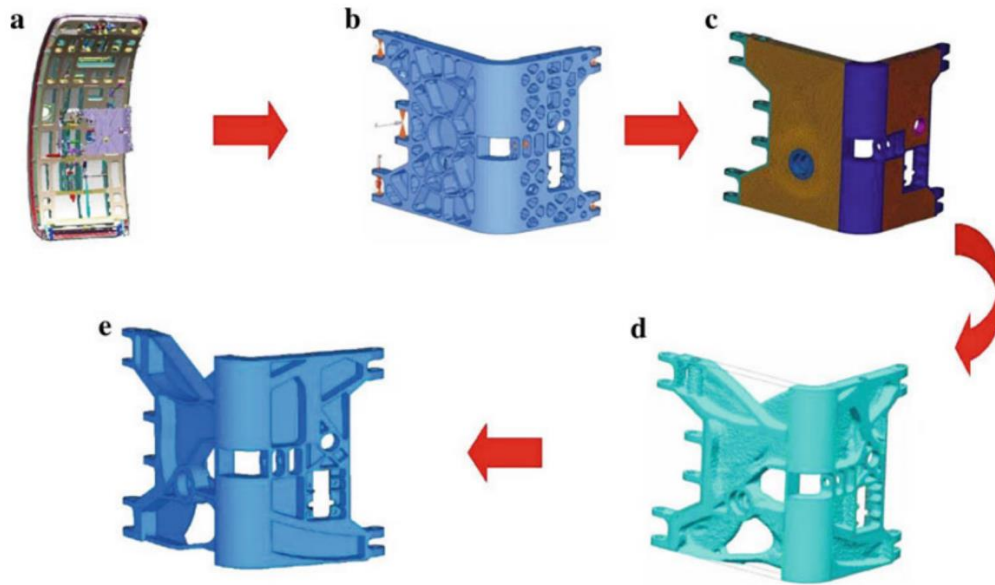


Figure 6. Topology optimized door support (Cheng, n.d.)

2.12.2 Aircraft engine pylon

SOGECALAIR Aerospace, a large supplier in the aerospace industry, developed this concept for an engine pylon, a component which attaches the engine to the wing. The part weight was reduced by 23 % from 450 kg to 350 kg while the number of parts was cut from 654 to just 14. (Abdelkader, 2015)

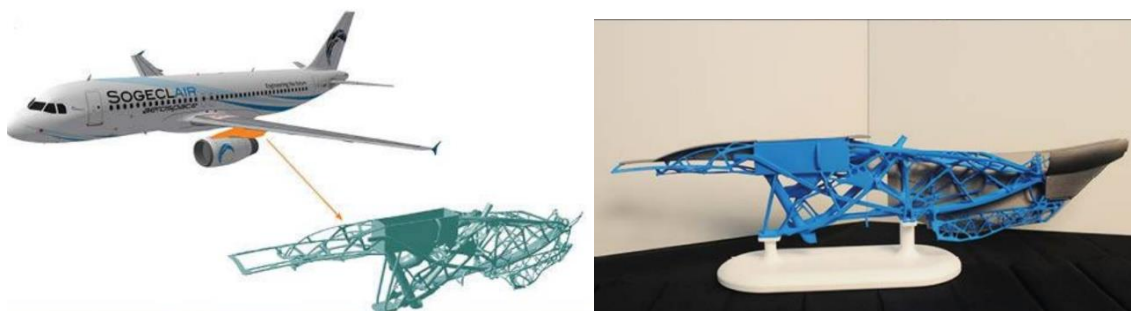


Figure 7. Topology optimized engine pylon together with a 3D Printed model (Clapaud, A, 2015)

The impact these optimized parts have on cost savings in the commercial airline industry is significant in the lifespan of a single aircraft since fuel consumption increases with

aircraft weight. Added weight requires added thrust and thrust corresponds to fuel consumption. For short distance flights a larger part of fuel is burnt during take-off and climb. The heavier the aircraft the more fuel is burnt during climb. By reducing weight by 100 kg equals fuel savings of 8 – 17 kg. (Steinegger, 2017)

2.12.3 Suspension component for compact disk harrow used in agriculture

The multiple welded part assembly optimized using solidThinking Inspire was reduced to only one cast part. The part weight was reduced by 8 % from 245 kg to 225 kg. Manufacturing costs were reduced by one third due to the reusable casting mold. The 16.5-meter welding seams were no longer required while the durability was increased by factor of 2.5. (Castings sa, n.d.)

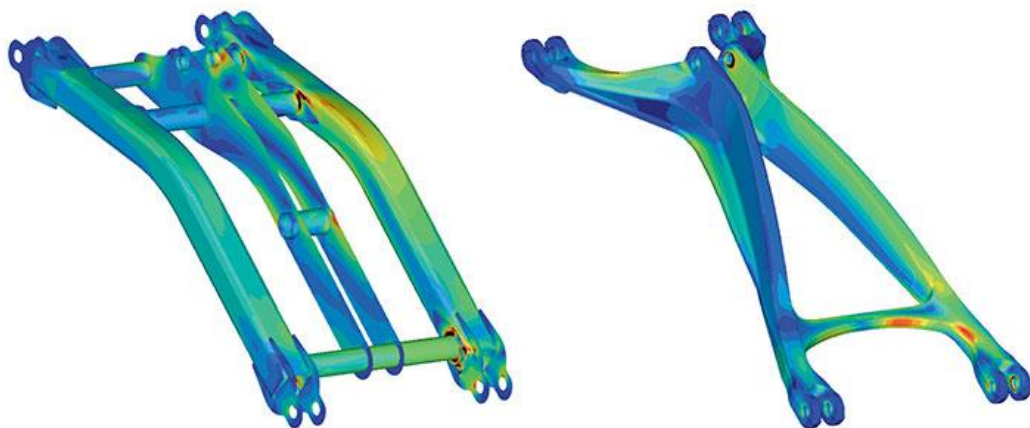


Figure 8. The cast part showed fewer stiffness points (red) due to smoother transitions (Castings s.a, n.d)

2.13 Altair solidThinking Inspire 2019

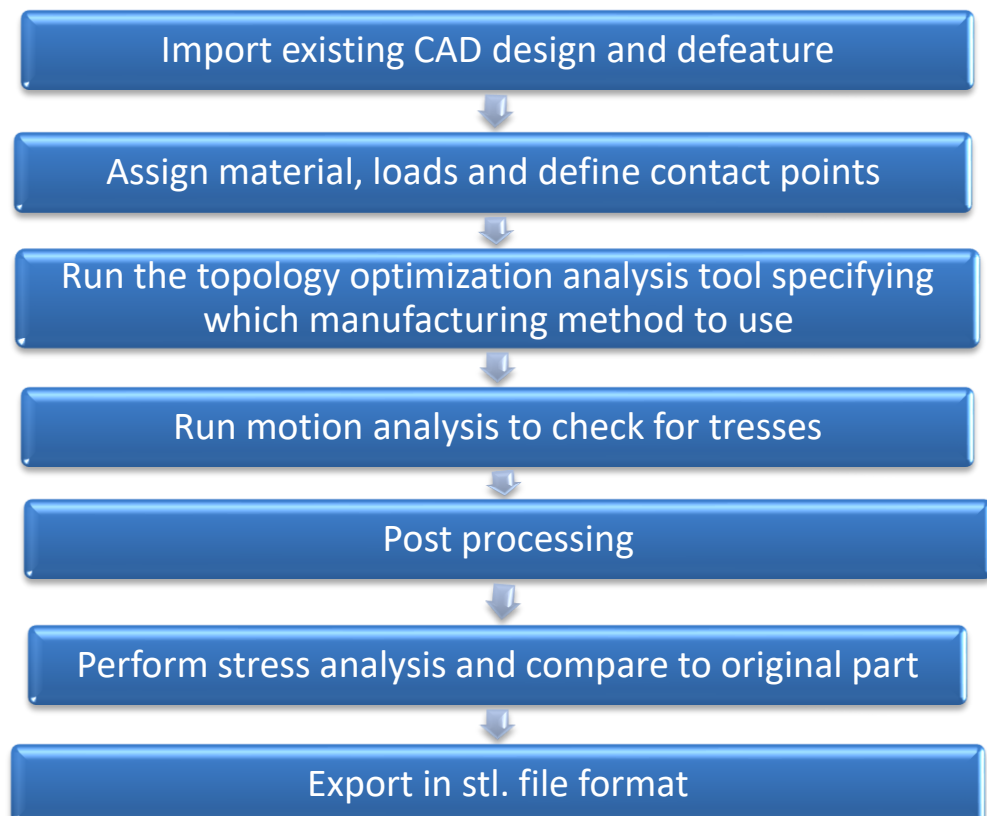
The Altair solidThinking Inspire software is a simulation driven design software allowing designers and engineers to both create designs and perform simulations. Some of the key features of Inspire include:

- Assembly and simulation of mechanical systems and determining the forces on parts for optimized analysis
- Create weight saving design alternatives tailored to either traditional- or additive manufacturing with the built-in topology optimization tool
- Simulating parts experiencing buckling and loads
- Importing files from CAD software and modifying them

(Altair, n.d.)

Altair Inspire enables the user to do things which used to be limited to professionals in the field of finite element analysis (FEA). It is also suitable for students since it does not require a lot of time learning FEA and gives the same accurate results. By taking an initial design and running an analysis Inspire can based on part performance suggest improvements using the design proposal tool. It also helps creating optimized designs suitable of manufacturing. A new feature in the 2019 edition is the added Fit- tool which at the end of a topology optimization- run enables the user to automatically smoothen the surface of the optimized part.

The basic workflow of Inspire is the following:



(Altair University, 2019)

2.14 SolidWorks Topology Study

A recent feature in SolidWorks is the added topology optimization tool found in the simulation add-in. It uses the Tosca optimization engine. It works by changing the design space into a mesh and running a simulation including user defined loads, constraints and boundary conditions. The program analyses how the original part will deform under load and then compares these to a measure a deviation. It looks at the stiffness of each element and cuts out the ones not structurally benefiting the part. The parts to be cut can be modified by the user. Manufacturing constraints can also be set such as thickness, mould separation direction, symmetric geometry around axis etc.

When starting the program, the user chooses one goal and one constraint. These include:

- Best stiffness to weight ratio (Get the most stiffness while removing x% of mass, (where mass is the constraint)
- Minimize mass with displacement constraint
- Minimize maximum displacement

Constraints include:

- Required mechanical copies
- Percentage of mass removed
- Manufacturing process
- Maximum deflection

(Wasserman, 2017)

The basic workflow of SolidWorks topology study consists of:

- Import existing design and defeature
- Assign material
- Add fixtures
- Apply loads
- Select goal of topology analysis and constraint type
- In manufacturing constraints assign preserved regions (non-design space)
- Assign thickness, symmetry and de-mold constraints

- Run the topology optimization analysis tool by right clicking mesh and choosing mesh and run
 - Modify percentage of mass retained/removed of optimized part with slider
 - Create smooth mesh
 - Run motion analysis to check for tresses
 - Perform stress analysis and compare to original part
 - Export smoothed mesh as new configuration or new part file.
 - The new part file can be saved as STL. file
- (Bower, 2018)

3 METHOD

A standard mounting bracket from Parker Hannifin was chosen to demonstrate the topology optimization process. The software used were Solidthinknig Inspire 2019 and SolidWorks 2019.

The bracket connected to a rod- eye using a pin. The part number was 1455310000 and had a nominal force limit of 12.5 kN. The specified material was AISI 316, also known as stainless steel and the reported part weight was 1.3 kg without the pivot pin and locking plate + screws.

In this case, the topology optimization process was performed using a ready made 3D CAD model (.ipt- format) provided by Parker Hannifin. The part- drawings provided by Parker Hannifin lacked certain details and dimensions required to 3D model an accurate model of the monting bracket. Since accuracy of the part was to be maintained it was decided to use the ready made model. A step- by- step process showcasing the 3D CAD modeling of the mounting bracket can be found in the appendices.

3.1 Topology optimizing Mounting bracket using Inspire 2019

The bracket was saved in Parasolid-format. This was necessary since solidThinking Inspire did not support SolidWorks Educational-version files. Parasolid- format retained all necessary surface geometry features and model data.

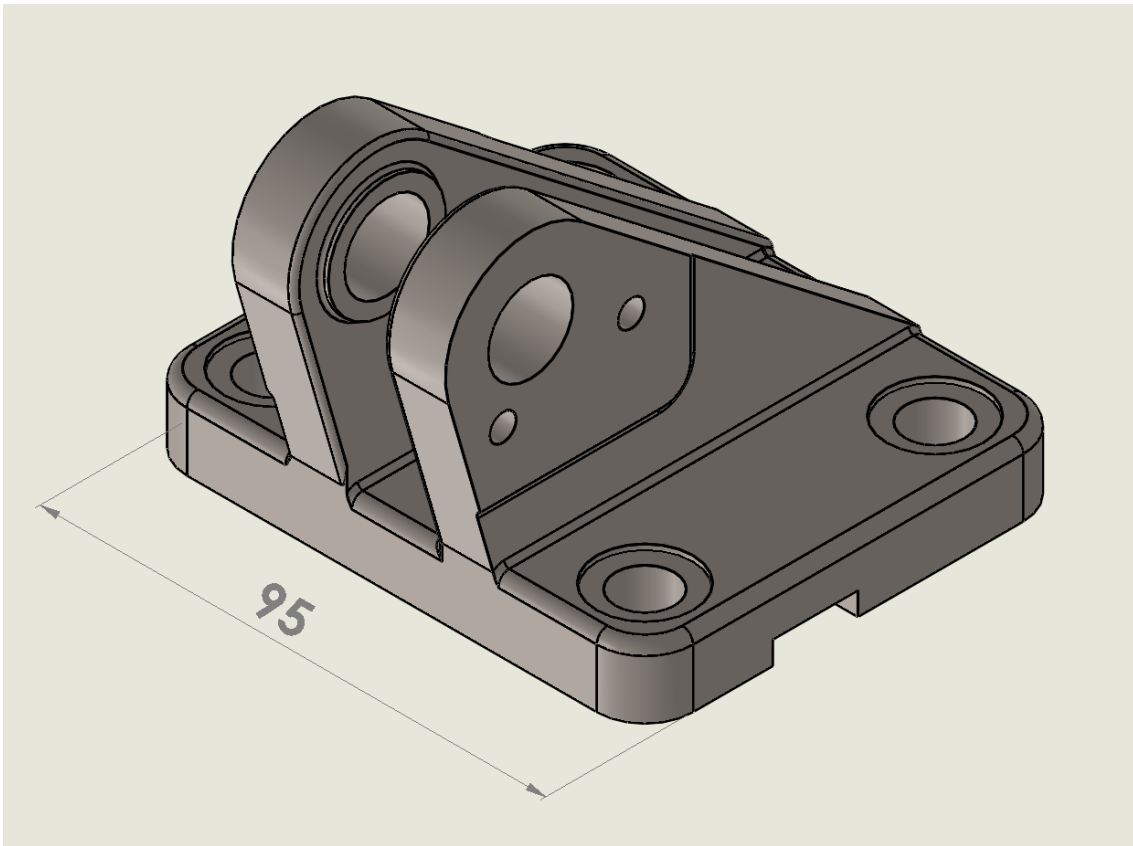


Figure 9. Mounting Bracket by Parker Hannifin (Oliver Folger, 2019)

3.2 Creating Design- and Non-design spaces

The part had to be prepared for the topology optimization process. This included defining design- and non- design space. Solid Thinking Inspire would not recognize the four mounting holes as well as the two large holes for the pivot pin without defining the design space. To define the design space circles had to be sketched around each hole, extruded to cylinders to cover each hole entirely. Then, using the Boolean tool each cylinder was set to intersect the holes yet keeping them at the same time. This split the areas inside each cylinder to become separate bodies while remaining attached to the rest of the bracket. Each hole which was now a separate body was assigned non- design space and the rest of the part automatically became design space.

3.3 Applying forces and supports

The next step was to define forces and loads. According to Parker Hannifin the bracket had a nominal force limit of 12.5 kN. Therefore, a force of 6.250 kN per pivot pin hole was applied. Supports were assigned to the for mounting holes at the base of the part.

To apply forces, the two pivot pin holes were selected, and the direction of the force applied in relation to the x-, y- and z-axes. In this case the force to be aligned with the top edge of the mounting bracket being 45°, therefore $X = -1.00$, $Y = 0.00$ and $Z = 1.00$. These directions depended on what default orientation the part had. Many different load configurations could be saved and run on the same model. In this case all forces and supports used on the bracket were called Load1. The orientation of the force served as an example and in reality altering directions could have been assigned.

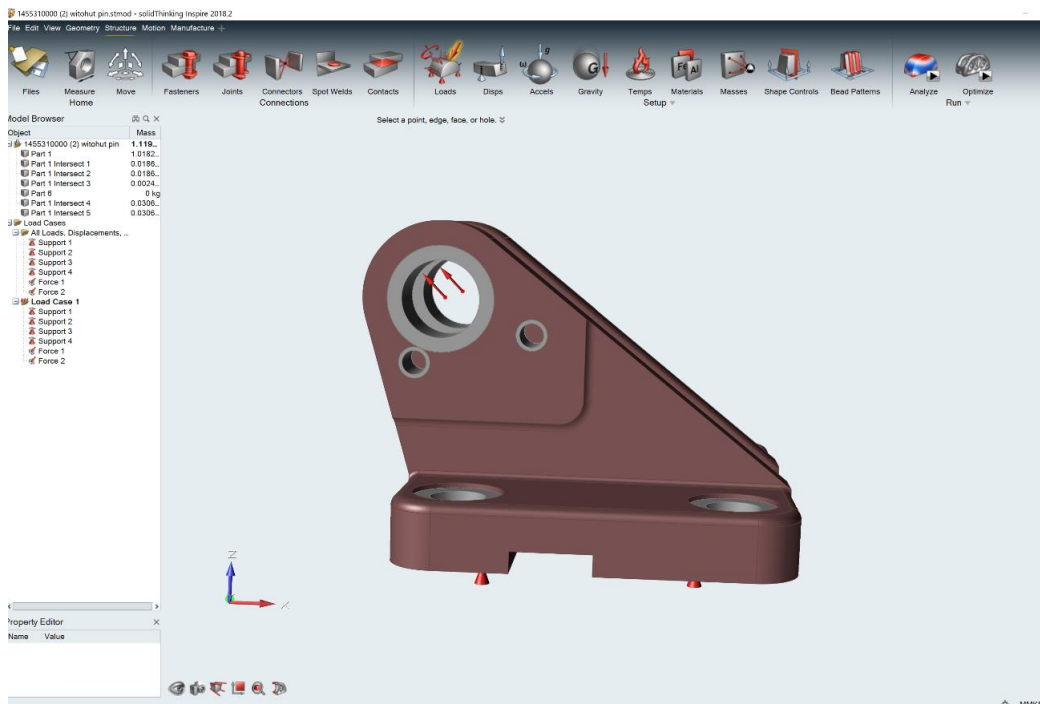


Figure 10. Applied forces and supports (Load 1) with design space in red, non-design space in grey (Oliver Folger, 2019)

Running the FEA analysis showed the part could easily withstand the 12.5 kN force it was designed for. Minimum Factor of Safety, 1 appeared on one of the back-mounting holes.

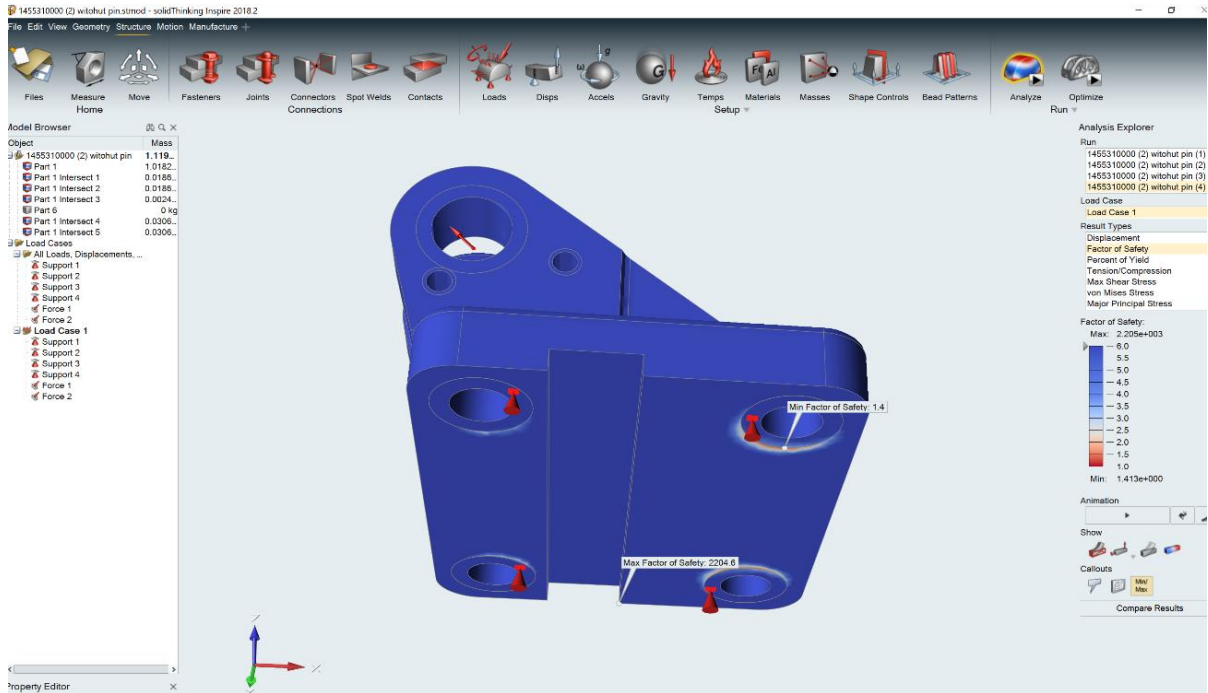


Figure 11. FEA analysis (Oliver Folger, 2019)

3.4 Topology Optimization

The bracket was now ready to be optimized. The settings for optimization were found under the Optimize- tool. In there the type of optimization, percentage of mass to be reduced while retaining part stiffness, minimum and maximum element sizes was assigned. The settings used for optimization included:

- Type: Topology
- Objective: Maximize Stiffness
- Mass Targets: 30% of Total Design Space Volume
- Thickness constraints: Minimum 1.8375 mm
- Factor of safety 1.2
- Load Case 1

It is to be noted the small thickness constraint chosen gave a more accurate result however this resulted in a processing time of about 4 hours. The new part mass was reported to be 400 g.

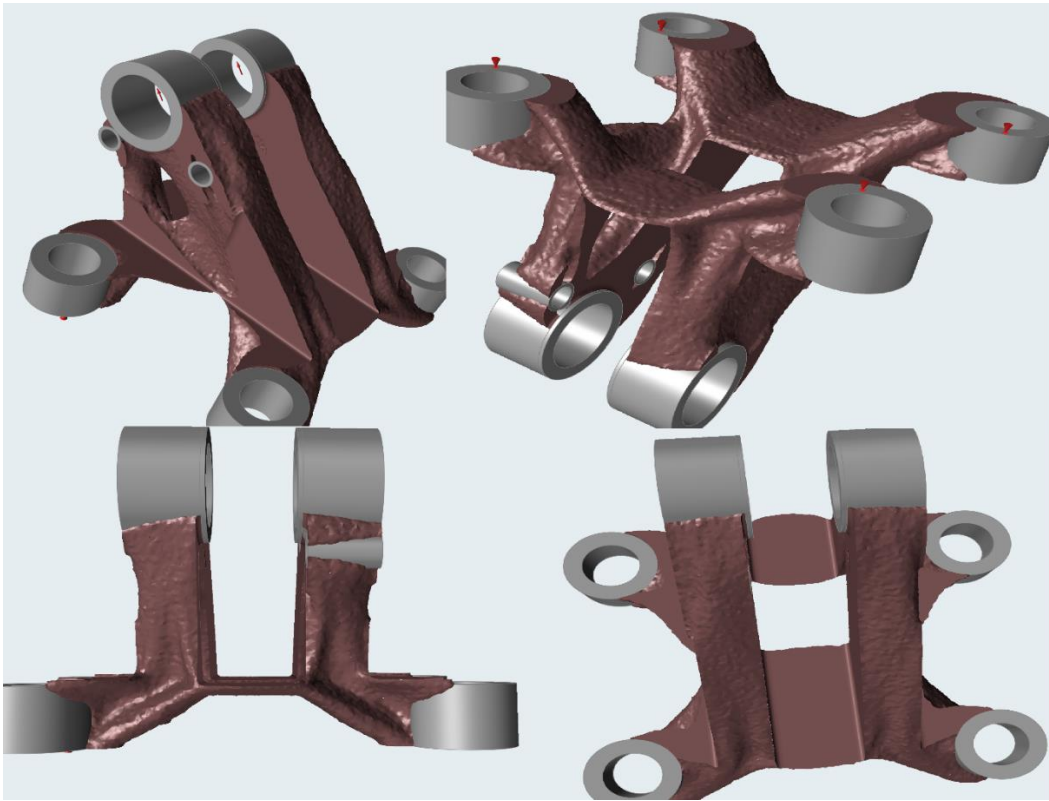


Figure 12. Topology optimized bracket (Oliver Folger, 2019)

3.5 Post processing

Analysing the optimized part showed most of it having a stress of 0.18 MPa , including the pivot pin hole, indicated by the blue colour. The Factor of Safety for this area was 6. However, as in the FEA- analysis before the optimization, the edges joining the two back bottom plate mounting holes to the rest of the part showed higher stresses at 422 MPa . The factor of safety value for this red area was reported by Inspire to be 1.

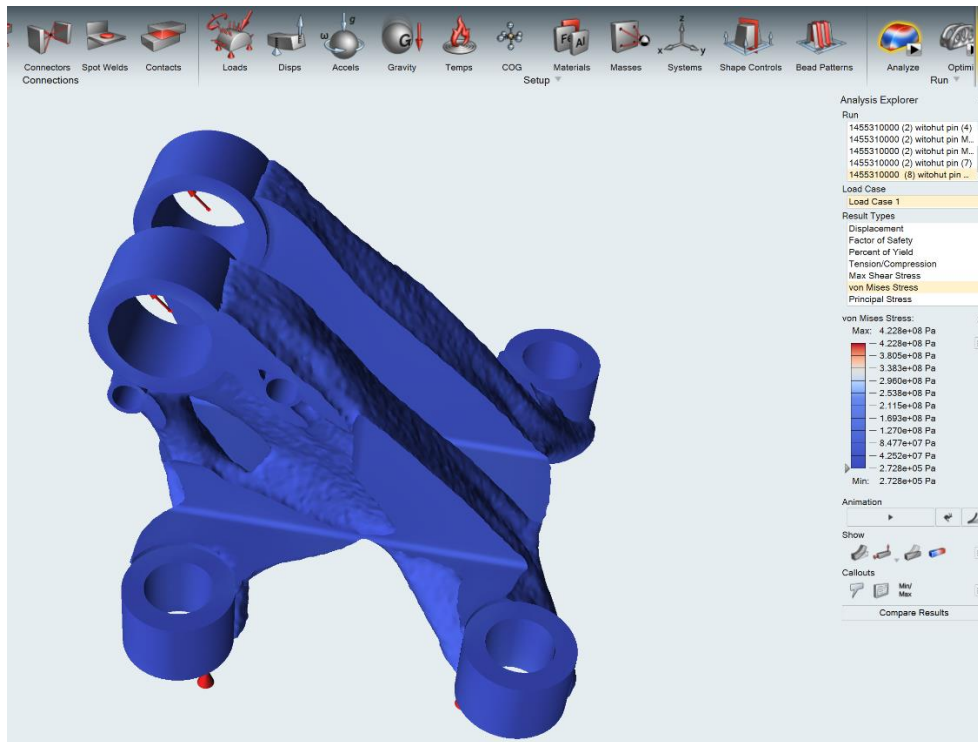


Figure 13. FEA- analysis after optimization

The maximum displacement concentrated to the top of the part including the pivot pin holes was reported to be $4.357 \cdot 10^{-5}\text{ mm}$ by Inspire.

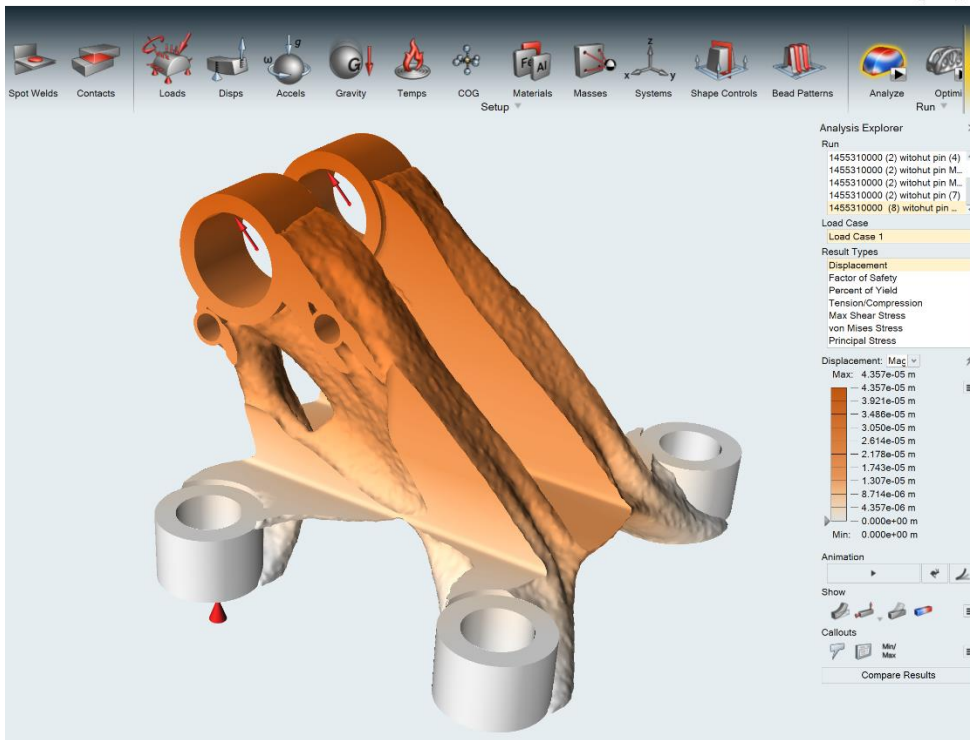


Figure 14. Displacement

After running the topology optimization and analysing the part Inspire gave the option to fit the part. This meant automatically wrapping the part with triangles making the surface smoother and could now be exported as STL- file ready to 3D Print.

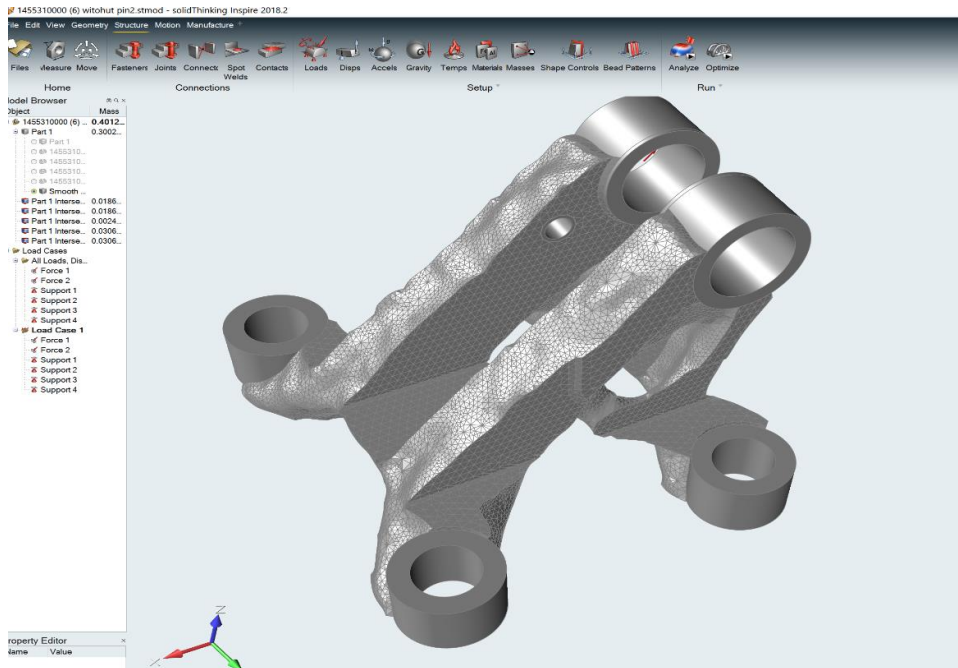


Figure 15. Automatic surface smoothening created by the Fit- command (Oliver Folger, 2019)

3.6 Topology optimizing Mounting bracket using SolidWorks 2019

The standard Parker mounting bracket was opened in SolidWorks and saved as SLDPRT-file. Before running the topology study a static stress analysis was performed to make sure the part could withstand the forces applied. It was performed by selecting Simulation → New study → Static. This was followed by placing fixtures as each mounting hole and a load of 6250 N at each pivot pin hole. The stress analysis showed the mounting bracket being able to withstand the forces placed on it with the highest stresses being on the two back mounting holes.

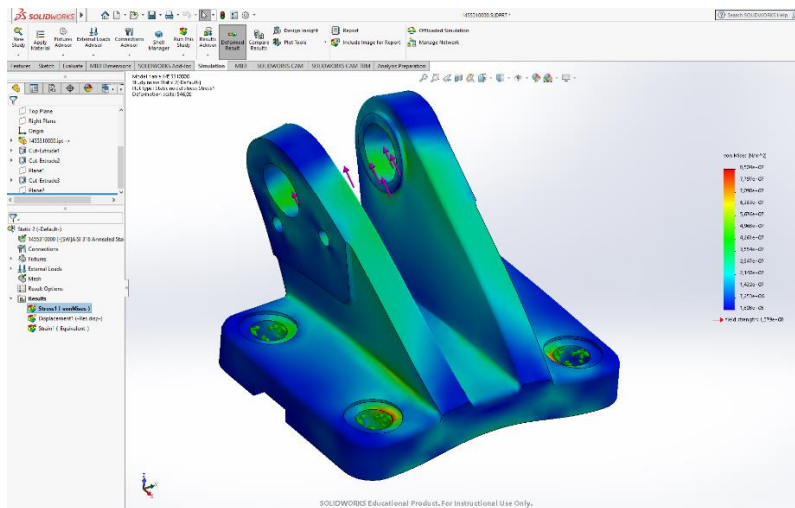


Figure 16. Static stress analysis simulation (Oliver Folger, 2019)

Next the SolidWorks topology optimization tool was located under SolidWorks Add in-ribbon and choosing SolidWorks Simulation. Then, a new study was selected and the topology study using best weight to stiffness ratio specified as study method. The mass was set to be reduced by 70 % matching the Inspire optimization and a Factor of safety margin was set to 1.2 to make sure the final result would withstand the stress from the forces.



Figure 17. Selecting new study and Topology study

The standard Parker mounting bracket was opened in SolidWorks and saved as SLDPRT-file. First, the material was set as AISI 316 steel, the same used in Inspire. The fixtures simulating parts of the bracket being fixed to a wall or a base were set choosing the four mounting holes in each corner.

Next the forces were set 6250 N for each pivot pin hole. The direction of the force was set to follow a plane placed on the top slope of the bracket and choosing selected direction.

The preserved region was specified. This acted as design and non-design space as in Inspire. The two pivot pin holes, two locking plate holes and four counterbore holes were selected. Any surfaces specified as fixture were automatically set as preserved region by the topology study.

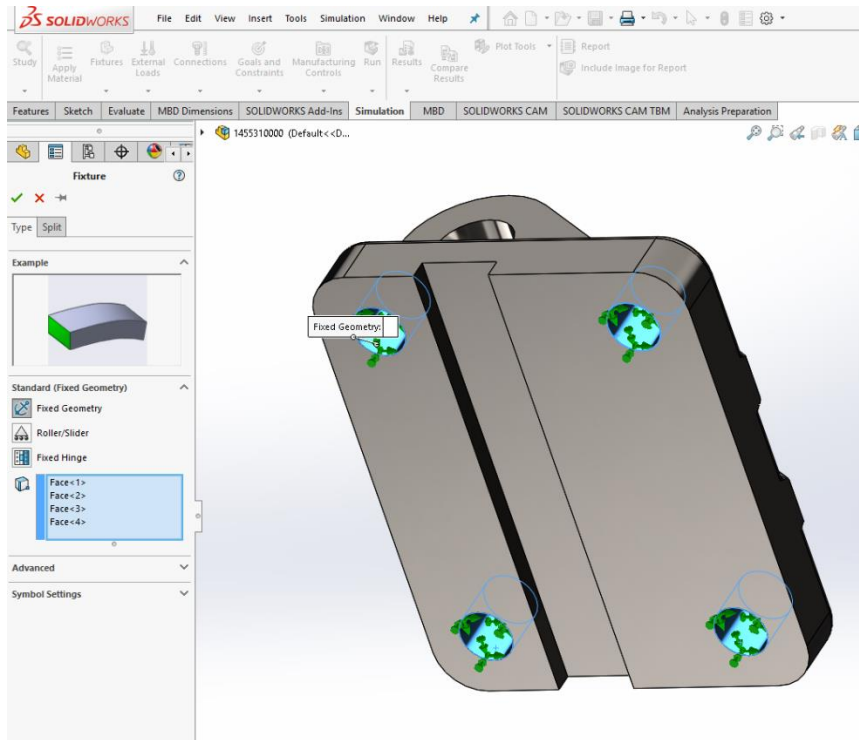


Figure 18. Setting fixtures (Oliver Folger, 2019)

The thickness control was set to 8 mm. This made sure the topology study would not leave any bodies or members generated by the study less than 8 mm making it more suitable for 3D Printing. Symmetry plane could also be specified, making both sides of the part symmetrical after optimization however this was not seen necessary since it would defeat the purpose of being an organic shape.

The mesh was then generated creating small triangles around the part. The default mesh set by SolidWorks was chosen. A finer mesh would give a more accurate result but, would lead to a larger file size and longer optimization time.

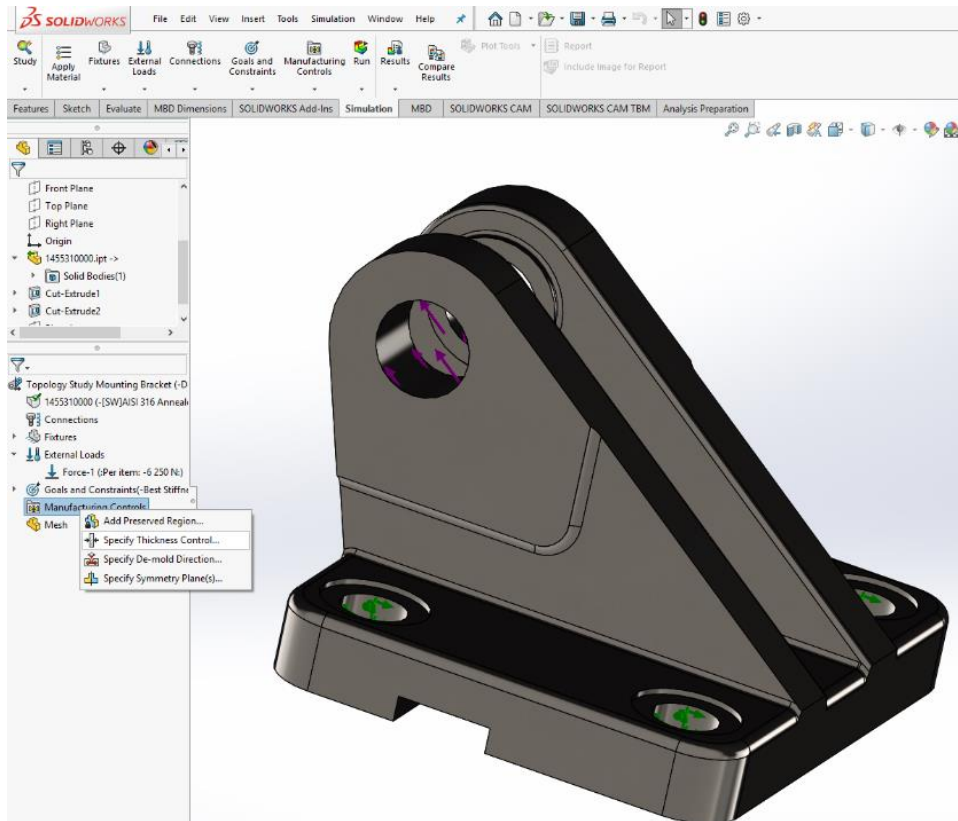


Figure 19. Adding forces, 6250 N per mounting hole (Oliver Folger, 2019)

The study was then started by selecting run study, taking 3 hours and 30 minutes to complete. The optimized part contained many sharp angles and therefore the part was automatically smoothed using the calculate smoothed mesh- function. The material mass could still be adjusted, however the default setting showed the topology study retaining 33% of the original mass. The different areas of the optimized part were shown according to a specific colour (yellow, light blue, blue or purple) indicating which areas would be removed and which would stay. After completing the smooth mesh function the part could then be saved as a STL-file to be 3D Printed.

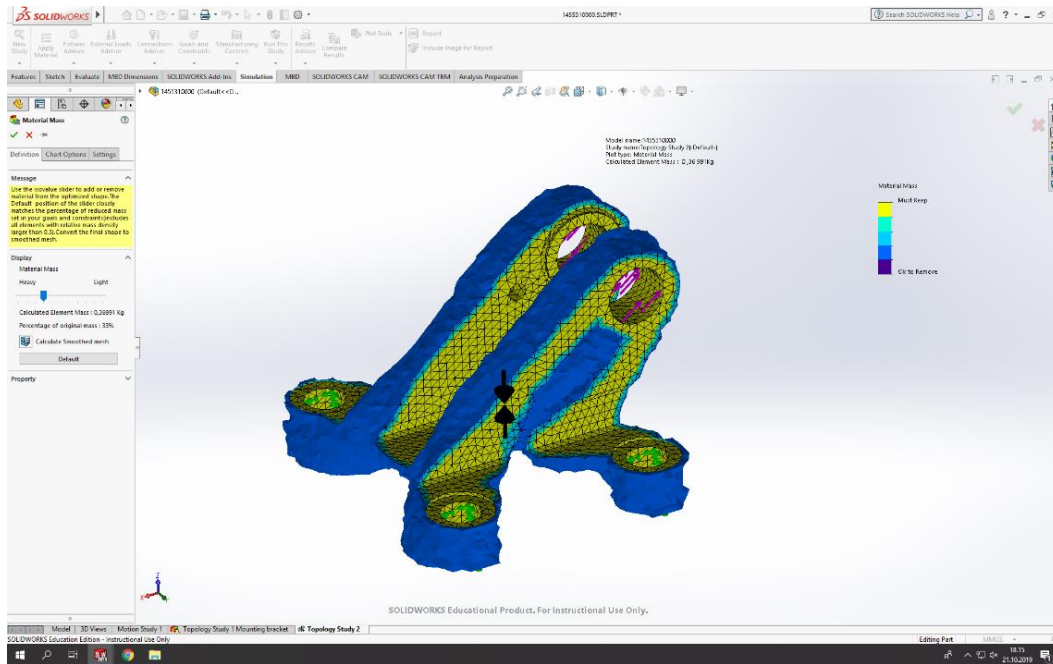


Figure 20. Selecting areas to be preserved (Oliver Folger, 2019)

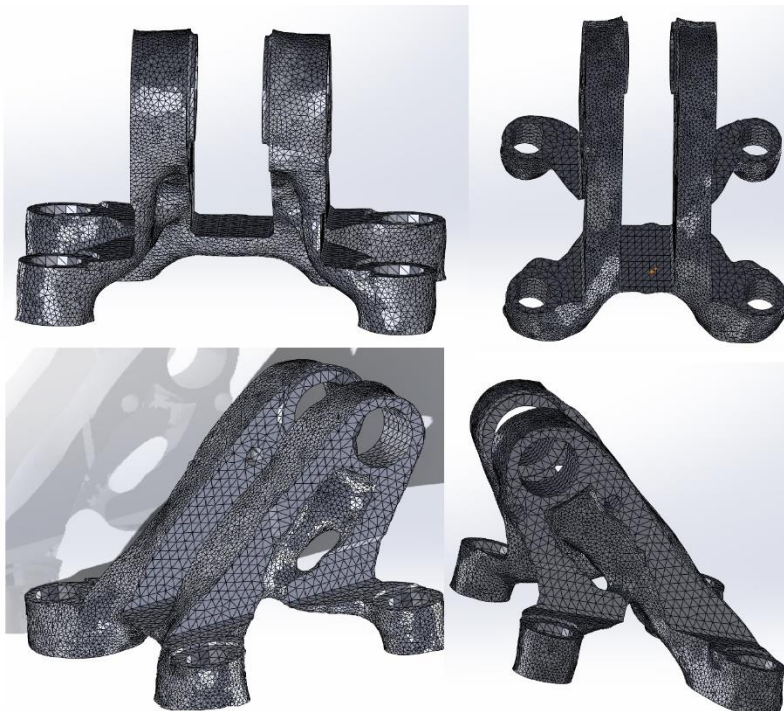


Figure 21. The mounting bracket saved as STL-file (Oliver Folger, 2019)

3.7 3D Printing Mounting Brackets

All models were printed using MakerBot replicator 5 Printers because of the availability in Arcadas plastic lab and ease of use.



Figure 22. MakerBot replicator 5 3D Printer

The models served only as visual demonstrations and thus did not require to be printed with 100% infill saving both time and material. Instead 20 % infill was chosen as this produced a thick enough surface supported by a diamond grid inside to give the part enough strength yet making it 80 % hollow saving both material and printing time. They were printed using white PLA- material. The main settings used for printing were 20 % infill and 215 °C nozzle temperature. The number of shells, meaning how thick the outer surface would be was kept at default value of 2 making the surface strong enough. Because of the overhangs in the optimized models supports were required. All models were printed using a raft at the bottom securing it to the build plate and minimizing warping. The MakerBot 3D printers use their own software called MakerBot Print. The STL- files were imported into the program and the printing settings applied. After the settings were applied a final check to make sure the model would print accordingly was done by pressing the print preview button. The program would then calculate each toolpath and layer necessary to complete the print. After it was loaded the user could manually with the slider choose to see all the layer.

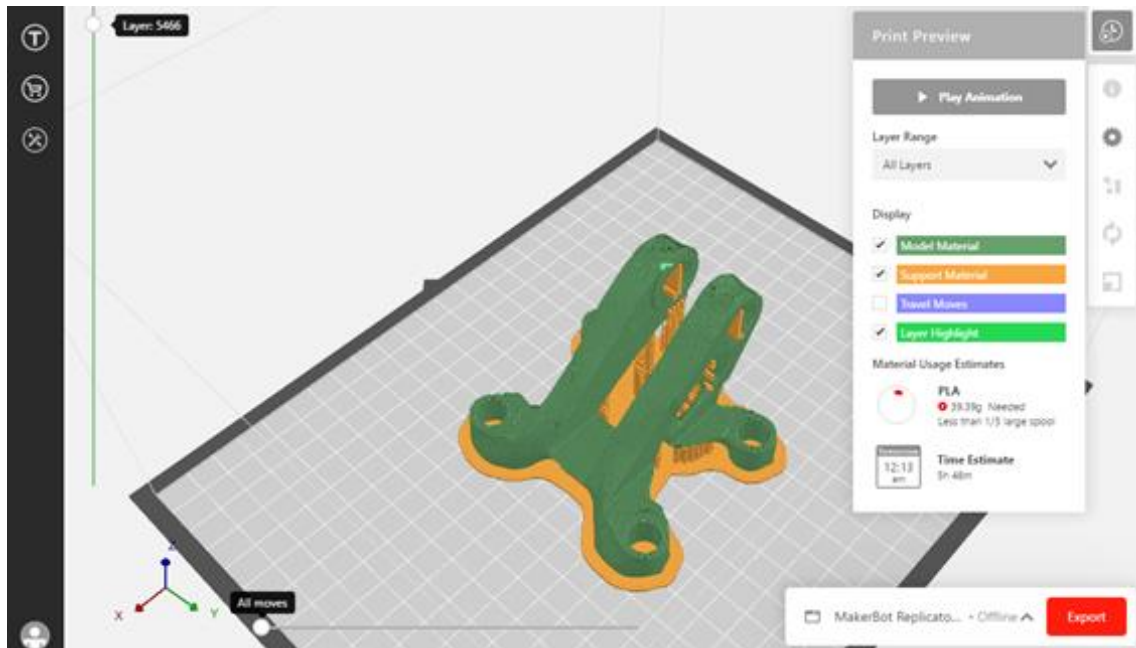


Figure 23. SolidWorks Topology study model prepared for 3D Printing in MakerBot Print. Model in green, raft and supports in orange (Oliver Folger, 2019)

4 RESULTS

The same Parasolid mounting bracket model was used in both cases. Best stiffness to weight ratio study method was selected and the mass reduction was set to 70 % in both cases. The same forces and fixture points were used in both optimization cases. Default mesh size was selected affecting the optimization run time. Also, how optimized and developed the programs were affected the run time. All parts used AISI 316 material. Reported masses before and after topology optimization were shown by respective software. Both programs provided easy mesh smoothing settings after the optimization process. This made is easy to produce a smooth presentable part ready to be exported to STL-file and 3D Printing.

4.1 Inspire

The mounting bracket optimized with Inspire saw a reduction in weight of 66 %, from 1180 g to 400 g. The bottom plate saw the most mass reduction removing material between each mounting hole. However, in this case the topology optimization left material between both the front- and back pair of mounting holes. Inspire left a more apparent border between the transition from design space to non- design space, making the mounting holes stand out.

The FEA-analysis before optimization showed the areas around the pivot pin holes and mounting holes with stresses of 0.18 *MPa* before and 0.42 *MPa* after the optimization. The joints between the two back mounting holes saw most stress at 145 *MPa* before and 422 *MPa* after the optimization. The respective Factor of Safety values were 6 and 1.

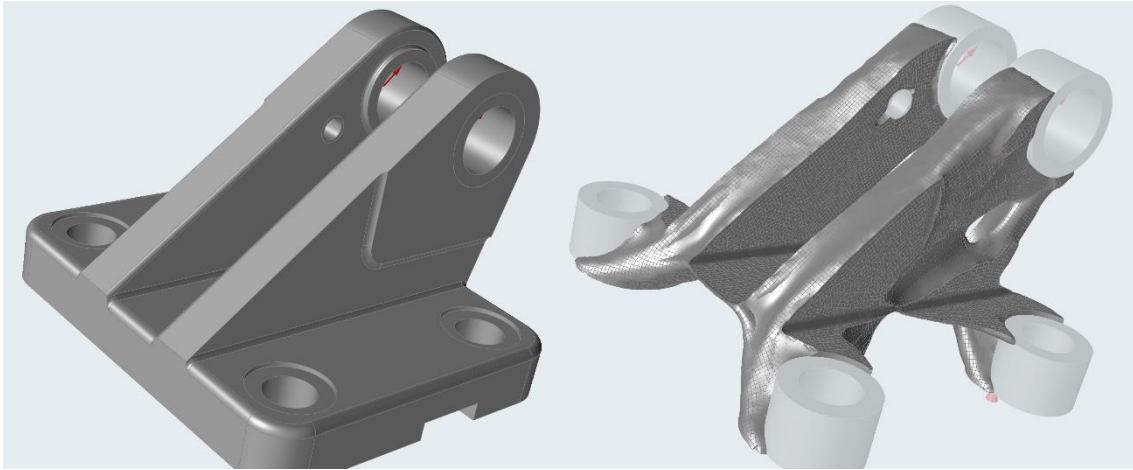


Figure 24. Topology optimized mounting bracket using solidThinking Inspire (Oliver Folger, 2019)

4.2 SolidWorks

The SolidWorks topology optimized part reduced the weight by 67 %. The optimization study removed material mostly from the base of the mounting bracket leaving the only connection point between all members at the back feet.

Overall the result looked similar the mounting bracket optimized with Inspire.

The two diagonal surfaces leading up to the mounting pin holes were less affected by the topology optimization as the surfaces were linear to the applied forces. The outside round surfaces of each mounting saw a small mass reduction as well as the top pivot pin holes. The transition from design space to non-design was fluid and blended both parts together.

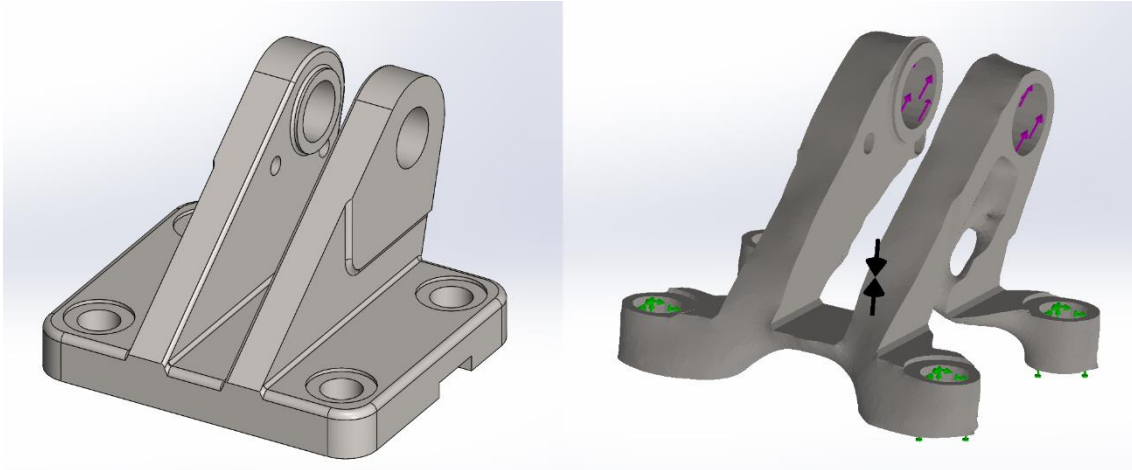


Figure 25. Topology optimized mounting bracket using SolidWorks Topology study (Oliver Folger, 2019)

Table 2. Comparing the topology optimized parts to the standard part (Oliver Folger, 2019)

Part	Mass before Topology optimization	Mass after Topology optimization	Reduction	Optimization time
Parker Hannifin	1300 g			
solidThinking Inspire	1180 g	400 g	66 %	4 h
SolidWorks	1120 g	369 g	67 %	3 h 30 min

Table 3. FEA-analysis results

Part	Stress before Topology optimization around pivot pin holes	Stress after Topology optimization around pivot pin holes	Maximum displacement before Topology optimization	Maximum displacement after Topology optimization
solidThinking Inspire	0.18 MPa	0.42 MPa	$3.074 \cdot 10^{-5} \text{ mm}$	$4.357 \cdot 10^{-5} \text{ mm}$
SolidWorks	0.21 MPa	X	X	X

4.3 3D Printing

The 3D Printing settings used to print the models were the same for all three models except the layer height which for the optimized parts were 0.1 mm while the standards part used 0.2 mm. The printing times for the optimized parts were similar. The printing

time of the standard part increased due to its larger area and the increased travel time and travel distance for the printing nozzle.

According to 3D Formtech, a 3D Printing company based in Jyväskylä, Finland, the price to print each optimized part was 590 € + tax (Lipponen, 2019), while the Parker Hannifin mounting brackets retail price according to retailer Espoon Teollisuuspalvelu Oy was 370 € (Valo, 2019)

Table 4. Comparing 3D Printed mounting brackets (Oliver Folger, 2019)

Part	Parker (standard part)	solidThinking Inspire	SolidWorks Topology study
Printing time	7 h 27 min	5 h 37 min	5 h 48 min
Material usage	80 g	46 g	39 g
Size, x, y, z-axis	90 x 95 x 66 mm	82 x 88 x 61 mm	84 x 86 x 62 mm



Figure 26. 3D Printed mounting brackets with Parker standard part to the left, solid Thinking Inspire on the top and SolidWorks optimized to the right (Oliver Folger, 2019)

5 DISCUSSION

The target weight reduction even though set to 70 % was not reached with default settings. The bracket optimized with Inspire reached a 66 % weight reduction with the SolidWorks version reaching 67 %. It is to be noted the masses reported by both programs differed from Parker Hannifins own mass specification with both programs reporting 1.1 kg compared to 1.3 kg.

A Factor of safety value of 1.2 was set in both cases however not fully met. This could be due to the way the program presented the results. Since a slider at the side was used to “scroll” through the part, removing material corresponding to factor of safety an accurate reading could not be attained. In the Inspire optimization case the stiffnesses in the different parts were checked by running an FEA- analysis after the optimization. Part stiffness increased in the topology optimized mounting bracket from 0.18 *MPa* before and 0.42 *MPa* after the optimization around the pivot pin holes. Between the two back mounting hole joints the maximum stress increased from 145 *MPa* to 422 *MPa* after the optimization. The lowest factor of safety reported was 1 around the two back mounting holes.

Due to the lack of response of SolidWorks during optimization no post FEA- analysis was available. It is presumed the SolidWorks mounting bracket met the stress requirement since a Factor of safety value of 1.2 was defined before running the optimization. However, performing a post- FEA- analysis is recommended in future topology optimization cases. Therefore, the topology optimization was partially successful since the lack of post-FEA analysis in SolidWorks gave no comparable results.

The maximum displacement increased from $4.357 \cdot 10^{-5} \text{ mm}$ to $3.074 \cdot 10^{-5} \text{ mm}$. However, these displacements were very small and in practice would not affect the function of the part.

The way the different software were used and the workflow was basically the same, although they looked different and had different tool names. The way design and non-design space was set gave different results in the end, with Inspire producing a clear difference

between the mounting holes and the rest of the part while SolidWorks blended everything together.

The programs differed in the way they produced design and non-design space. Inspire required the extra step of making the non-design space areas to separate bodies something which was not necessary in SolidWorks.

Since Inspire did not support models created in the SolidWorks student version it is highly advised to save them as Parasolid- files before opening them in Inspire because the Parasolid-format retained all part information and parameters. In case the processing time was to be kept under 15 minutes the element size in the mesh created before running the optimization had to be above 4 mm. The part size was kept constant throughout each optimization process and no additional sizing was required. For someone already familiar with using Inspire and SolidWorks, learning the basic features of both topology optimization tools would only require following a few tutorials.

There were many instances during operation when SolidWorks would load for a long time and doing any other task at the same time could make the program freeze. Even with a well performing computer used in Arcada performance was still slow. The need for storage space proved to be a major factor since a 32 GB USB- drive had to be used to store the result which ended up taking 25 GB of space. The largest hurdle in producing a result with SolidWorks was the constant error message during optimization saying, “not enough disk space to run topology”. This problem would appear at random stages and there were instances when 92% and even 99% of the topology study was complete when the error would occur.

SolidWorks uses its topology study as an add-in, something only recently added to the program in 2018. Therefore, it may not be as well integrated into the software causing problems. SolidThinking Inspire was easier to use and ran more consistently. It had in general more intuitive user interface and experienced no errors during optimization.

For future improvement and expanding on the thesis, the standard part design space could have been enlarged to make the possibility for a better optimization. Giving more design

space would result in a better result, however first the motion of the pivot pin when connected to the bracket would have to be studied in order not to cause material interfering with the pivot pin. In addition, design- and non-design space should be specified to make sure there is enough material around mounting holes. In this case the small front side hole intended for the locking plate was left thin. If produced, the wall thickness should be sufficient for adding threads and avoid risk of breaking when tightening the screw.

3D Printing all parts was straight forward and using default printing settings proves adequate to produce rigid models with enough detail to show the organic surfaces of the optimized parts. The price to 3D Print the optimized parts still is expensive.

Looking at the cost of 3D Printing the optimized brackets versus buying one from Parker Hannifin showed a price difference of 220 €. The higher cost of 3D Printing is to be expected since the brackets are custom made. The high price of the standard mounting bracket is due it also being a custom part and only available through certified Parker Hannifin distributors. However, the price only involved one bracket. If the number of 3D Printed brackets were more the price could be lower and closer to the price of the standard part. And so, if the optimization would have been performed on a standard “of the shelf” part found in any hardware store the price difference would have been greater.

6 CONCLUSION

Topology optimization is a powerful tool in reducing part weight while retaining stiffness. It enables a seamless and efficient optimization for design engineers to improve an existing design. When used in great numbers, the combined weight reduction leads to cost savings in operation such as in the airline industry. Not only does the topology optimization reduce weight but also parts no longer requiring welding seams further reduces the weight.

Two topology optimized versions of a standard mounting bracket were produced using solidThinking Inspire and SolidWorks. The successful steps included importing a CAD-model, performing a FEA- analysis, defining optimization method and part design- and non-design space, running topology optimization and generating smooth mesh. The specified optimization type used in both optimization cases was to reduce weight while maximizing part stiffness.

In the Inspire optimization case the part weight reduced as well as the maximum part stiffness, even though the factor of safety was specified to 1.2. The maximum and minimum factor of safety varied between 6 and 1, with the edges joining the two back mounting holes to the rest of the body showing 1 and taking most stress. Depending on the safety requirements the part is accepted, or these areas should be addressed by adding more material to relieve some of the stress.

For someone already familiar with using Inspire and SolidWorks, learning the basic features of both topology optimization tools would only require following a few tutorials.

The retail price for the 3D Printed brackets was more expensive compared to the standard part since it is a custom part however, the difference was not as large as expected.

7 INTRODUKTION

Topologisk optimering är en matematisk metod att optimera delar och strukturer. Begreppet presenterades för första gången på 1980- talet och används idag i bil-, luftfarts-, marin- och den biomedicinska industrin. Dess funktion är att optimera material distribuering i ett givet design rum med bestämda krafter, randvillkor och begränsningar med det slutliga målet att maximera delens styrka och samtidigt minimera massan. (Gould, 2015)

Det optimerade resultatet har ofta former som kan hittas i naturen och som har effektiva strukturer dvs. det finns inte mycket onödigt material. Optimeringen kan justeras till att ge resultat som är lättare att producera med vissa produktionsmetoder som t.ex. gjutning och 3D Printning.

Den topologiska optimeringsprocessen följer efter design-processen där själva referensdelen skapas. Denna process innehåller 3D modellering, även känd som Computer Aided Design (CAD).

- Demonstrera den Topologiska optimerings processen
- Optimera ett elementfäste med solidThinking Inspire och SolidWorks
- Jämföra delarnas form och styvhet till det originala
- 3D Printa båda optimerade delar

7.1 CAD- Computer Aided Design

En CAD modell, även känd som en Solid Modell, fungerar som startpunkt för den topologiska optimeringsprocessen. En CAD- model skapas m.h.a. 2D ritningar som projekterar olika sidor av ett objekt. Dessa olika sidor som kan representera toppen, sidorna och botten hjälper att beskriva det 3-Dimensionella objektet. Solid Modelling förvandlar dessa 2- Dimensionella sidor till en elektronisk 3D -modell. Denna 3D- modell

har en volym, yta, hål och gömda tomrum. Modellern kan roteras och vändas för att man skall kunna inspektera dem och även skäras för att inspektera den inre strukturen. (Pechter, n.d.)

Då man designar en mekanisk del har man traditionellt använt ett CAD – program för att först 3D modellera delen och sedan visa den till en ingenjör som analyserar delen genom att använda den finita element analys metoden. Efter att delens styrka blivit analyserad och godkänd har en fysisk prototyp framställts och dess mekaniska egenskaper testats. På basis av resultaten förbättrar man formen för att delen bättre skall motstå den mekaniska stressen. Den nya modifierade delen testas igen och en ny omgång för justering görs på basis av de nya resultaten. Detta slag av design process kan kräva mycket tid och den slutliga formen är nödvändigtvis inte den optimala p.g.a. de många omgångar av justeringar som inte optimalt fungerar tillsammans. (Olason & Tidman, 2010)

8 TEORI

8.1 Topologisk Optimering

Topologisk optimering är en form av strukturell optimering vars syfte är att hitta den optimala material distributionen. Området i delen som man vill optimera kallas för design rum eller ”design space” och det man inte vill att skall ändra kallas för icke-design rum eller ”non design space”. I optimeringen används FEM för att avskilja området till ett nätverk av finita element som kallas för isotropic solid microstructure. (Dassault Systemes, n.d.)

Topologiska optimeringsmetoden kräver en finit element model som består av triangulära ytor. Dessa utgörs av 3D modellerna och uppskattar den optimala formens yttre gränser. Den nuvarande standarden är att skapa en CAD modell vars 3D modell uppskattar de yttre gränserna för den optimala delen. (Larsen & Jensen, 2009)

Det som gör den topologiska optimeringen effektiv är den automatiska processen som skapar konceptuella former vilka ersätter den traditionella ”trial and error” metoden. Optimeringen kan användas för olika design problem p.g.a. algoritmerna som bestäms enligt begränsningar tagna från mekaniken, strömningstekniken och termodynamiken. Topologisk optimering hjälper designers att skapa nya ideér och former även för väl anpassade produkter. (Liu & Yongshen, 2016)

Områdets avskilning till nätverk av finita element, utförs för att göra funktioner, variabler, modeller och ekvationer anpassade till numerisk evaluering och implementering för datorer. Optimeringsmetoder används för att lösa problemet genom att visa vilka element som är tomma och vilka som är solida i nätverket, vilket betyder att man bestämmer vilka element som har material och vilka som inte har material i sig. Detta är ett problem där varje element har två möjliga nivåer, antingen 0 eller 1. Antalet element gör problemet opraktiskt att lösa då antalet möjliga kombinationer är 2^N där N = antalet element. De två huvudmetoderna som kan lösa problemet är densitet metoden (SIMP) och homogeniserings metoden. (Olason & Tidman, 2010)

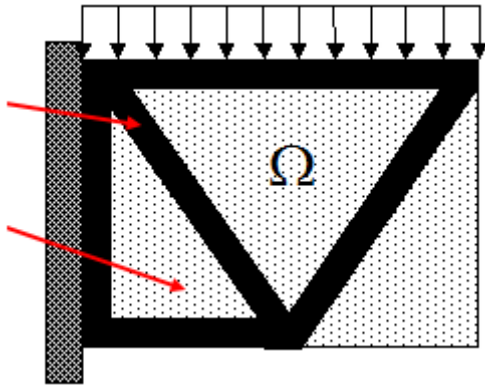


Figure 27. Tomma element i vitt, solida i svart

9 METOD

Ett standard fästelement från Parker Hannifin valdes för att demonstrera den topologiska optimeringsprocessen. Optimeringen gjordes med både solidThinking Inspire 2019 och SolidWorks 2019.

Fästelementet fungerade tillsammans med en styrled och kunde monteras mellan ett system och en vägg. Delens nummer enligt Parker Hannifin var 455310000 och hade en angiven styrka på 12.5 kN. Dess material var AISI 316, även känd som rostfritt stål och dess rapporterade massa var 1.3 kg.

För att demonstrera den topologiska optimeringsprocessen 3D modellerades fästelementet först i SolidWorks enligt ritningen från Parker Hannifin. Fästelementets ena halva modellerades först och eftersom den var symmetriskt kunde man med mirror – verktyget spegla den andra halvan vilket sparade mycket tid. I detta fall p.g.a att det saknades mått i ritningen kunde inte ett identiskt fästelement skapas och därför valdes Parker Hannifins egna 3D modell för att köra topologiska optimeringen på. En 3D CAD process av fästelementet kan hittas i bilagorna.

9.1 Inspire

Fästelementet som fanns tillgängligt i .IPT- format sparades i parasolid- format för att kunna bearbeta det i Inspire.

En FEA-analys gjordes före optimeringen för att få reda på den mekaniska påfrestningen.

Delen måste först förberedas för den topologiska optimeringen genom att ange design och icke-design rum. De fyra monteringshålerna och de två stora hålen för anslutningspinnen gjordes till icke- design rum genom att använda Inspires sketch verktyg för att rita ringar runt hålen och utvidga dem till cylindrar så att de täcktes totalt. Efter det kunde man med RMB välja områden som skulle bli icke-design rum. Det som inte valdes till icke-design rum blev automatiskt design-rum och kom därmed att påverkas av topologiska optimeringen. Efter att design och icke-design rummen definierats kunde krafterna och stödpunkterna anges. Inspire hade under structure- fliken ett Loads- verktyg där krafterna valdes och angavs till de två stora hålens inre sidor där fästelementet

kopplades till styrledet. En kraft på 6250 N per hål angavs som simulerade kraften som drar på fästelementet. Riktningen valdes linjärt med fästelementets övre kanter som var i ca. 45° vinkel med bottenplattan. Då allt var klart kunde topologiska optimeringen påbörjas och valdes ur Structure → Optimize. Optimeringsmetoden som valdes gav bästa styvhet och 30 % av den originala massa skulle sparas vilket betydde att vikten skulle minskas med 70 %

Efter att delen var optimerad var ytan rätt ojämn och kunde göras slätare med Fit-funktionen som Inspire presenterade efter optimering. Detta medförde att Inspire gjorde ytans tringel i 3D modellen ännu noggrannare genom att öka antalet trianglar och minska dess storlek. Efter att ytan gjorts slätare kunde det optimerade fästelementet sparas som en STL-fil och 3D Printas. Efter optimeringen utfördes en ny FEA-analys.

9.2 SolidWorks

Samma fästelement öppnades i SolidWorks. Den topologiska optimering var en add-in som kunde hittas under simulation → new study → topology study.

Optimeringsmetoden "Best weight to stiffness ratio" valdes tillsammans med en vikt förminskning på 70 %. Som följande angavs fixsturen och krafterna som var de samma som i föregående optimering.

Factor of Safety på 1.2 tillsattes för att säkra att den optimerade delen skulle klara av krafterna. Tjockleksbegräsning valdes till 8 mm, detta betydde att ingen struktur eller element i den optimerade delen skulle vara mindre än 8 mm tjockt för att förkorta optimeringstiden och göra det lättare att 3D Printa. Som följande valdes "generate mesh" som skapade små trianglar runt delen. Default mesh valdes för att inte förlänga optimeringstiden och behålla filstroleken i rimlig storlek.

Optimeringen påbörjades genom att välja "run study" och det tog 4 timmar. Det optimerade fästelementet hade grova ytor och kanter som behövde jämnas ut. SolidWorks topology study gav möjligheten till automatisk utjämning och även graden av ytjämnhet enligt behov. Även vikten kunde ändras men det förhandsvalda yjämnhetsnivån valdes för att inte ändra på massa. De olika delområden presenterades enligt gul, blå, lila färger som visade vilka grova områden som skulle raderas. Efter det automatiska ytjämnandet

kunde delen sparas i både STL-format och SolidWorks egna .SLDPR- fil och förberedas för 3D Printning.

9.3 3D Printning

Man valde att 3D Printa modellerna med Arcadas MakerBot Replicator 5. MakerBot 3D Printern använde FDM- tekniken där materialet värmdes upp och sprutades ut genom ett upphettat munstycke. Eftersom modellerna behövdes endast för visuella ändamål valdes fyllningsgraden till 20 % vilket betydde att varje del var till 80 % ihåligt. Som material valdes PLA och munstyckets temperatur bestämdes att vara 215 °C. Modellerna printades med fästelementets bottenplatta mot printerns ”build plate” eftersom det gav mindre mängd stödstruktur och sparade på printtiden. MakerBot 3D Printers använder dess egna 3D -Printnings program som heter ”Makerbot Print” där modellen öppnas i STL-format och sparas i .makerbot- fil. Endast modeller som sparats som .makerbot- fil kan printas med Replicator 5 modellen. Modellerna öppnades i Makerbot Print i STL- format varefter printnings inställningarna justerades och modellerna exporterades som .makerbot- filer.

10 RESULTAT

Samma fästelement för 3D modellen användes i ett parasolidformat, i båda optimeringar. Optimeringsmetoden ”best stiffness till to weight ratio” valdes i båda fallen med en 70 % vikt förminskning. Lika stora krafter placerades på samma områden. Programmens förinställda mesh storlekar användes vilket hade en effekt på optimeringstiden. Även graden av utveckling av de båda programmen påverkade användarupplevelsen. De båda delarnas material valdes till AISI 316.

10.1.1 Inspire

Det Inspire optimerade fästelementet hade en 66 % reduktion av massan, från 1180 g till 400 g. Delen såg som helhet likadan ut som det SolidWorks optimerade fästelementet. Majoriteten av material och vikt hade raderats från botten plattan och lämnat endast de fyra monteringshålens kvar.

FEA- analysen före optimeringen visade att största delen av fästelementet utsätts för 0.18 MPa mekanisk påfrestning. Den största mekaniska påfrestningen fanns under de två bakre monteringshålens kanter där påfrestningen var 145 MPa. Efter optimeringen visade samma områden 422 MPa vid monteringshålens och 0.42 MPa vid styrledens hål med en Factor of safety- värde på 6 och 1.

10.1.2 SolidWorks

SolidWorks minskade fästelementets vikt med 67 %, från 1120 g till 369 g. Optimeringen raderade material från bottenplattan och lämnade det enda fäststället mellan de bakre fötterna. Optimering hade mindre effekt på de två diagonala elementen. Monteringshålens yttre sidor hade även blivit tunnare lika så med de två stora hålen. En FEA-analys efter optimeringen kunde inte utföras p.g.a problem i SolidWorks.

Tabell 5. Jämförelse mellan optimerade och standard fästelementen gällande vikt och ptimeringstid

Fästelement	Massa före TO	Massa efter TO	Förminskning	Optimiserings tid
Parker Hannifin	1300 g			
solidThinking Inspire	1180 g	400 g	66 %	4 h
SolidWorks	1120 g	369 g	67 %	3 h 30 min

Table 6. Skillnader i mekanisk påfrestning och flyttning

Fästelement	Påfrestning före Topologisk optimering	Påfrestning efter Topology optimering	Flyttning före Topologisk optimering	Flyttning efter Topologisk optimering
solidThinking Inspire	0.18 MPa	0.42 MPa	$3.074 \cdot 10^{-5} \text{ mm}$	$4.357 \cdot 10^{-5} \text{ mm}$
SolidWorks	0.21 MPa	X		

10.2 3D Printning

Alla modeller printades med samma inställningar förutom lagertjockleken som för de optimerade fästelementen var 0.1 mm och för Parker Hannifin fästelementet 0.2 mm. De optimerade delarnas printningstider var liknande och tog nästan 6 timmar. Det tog längre att printa standard delen eftersom den hade större ytor vilket ökade distansen för 3D Printerns munstycke. Enligt 3D Formtech, ett 3D Printnings företag från Jyväskylä skulle de båda optimerade modellernas metall 3D Printning kosta 590 € + moms. (Lipponen, 2019) jämfört med standard delen som enligt Espoon Teollisuuspalvelu Oy kostade 370 € (Valo, 2019)

Tabell 7. De tre olika modellernas 3D Printningstider

Fästelement	Parker (standard del)	solidThinking Inspire	SolidWorks Topology study
Printningstid	7 h 27 min	5 h 37 min	5 h 48 min
Materialkonsumtion	80 g	46 g	39 g
Storlek, x, y, z- axis	90 x 95 x 66 mm	82 x 88 x 61 mm	84 x 86 x 62 mm

11 DISKUSSION

Med det valda fästelementet från Parker Hannifin lyckades man optimera de båda programmen och producerade slutlig reduktion av massan på 66 % respektive 67 % dock uppnådde man inte det valda 70 %.

De båda fästelementen hade blivit optimerade på samma sektioner i elementet. Största skillnaden kunde ses i hur väl programmen sammankopplade design- och non-design områden. Inspire lät fästhålerna sparas som skilda delar och sättet som de var fästa till resten av delen gav en tydligare gräns. SolidWorks bearbetade fästelementet mera som en hel del och hålen för fästen var kopplade till resten av delen med slätare ytor. Programmens användningssätt liknade dock varandra även om verktygen kunde ha olika namn och hittas i olika områden. Själva optimeringsprocessen var tydligt planerad i båda programmen och de olika stegen kunde lätt följas efter lite övning.

Vid de båda optimeringarna användes Factor of Safety- värde på 1.2. I inspire granskades delens mekaniska påfrestning och förflyttning före och efter optimeringen. Den största stressen hittades under de två bakre monteringshålens kanter där påfrestningen ökade från 145 MPa till 422 MPa . Efter optimeringen visade resten av delen samma områden $3.805 \cdot 10^5 \text{ Pa}$ med en Factor of safety- värde på 1. Resten av delen, även hålen för styrleden visade ett Factor of Safety värde mellan 5 och 6 och en ökning i mekanisk påfrestningökande från 0.18 MPa till 0.42 MPa . Delens förflyttning p.g.a kraften ökade från $3.074 \cdot 10^{-5} \text{ mm}$ till $4.357 \cdot 10^{-5} \text{ mm}$ vilket i praktiken är så liten att dess funktion inte påverkas.

Mesh storleken påverkade även det slutliga resultatets noggrannhet men förde även med sig en större filstorlek kopplad till en längre optimeringstid. Det förekom flera fall där SolidWorks slutade att fungera och endast laddade under optimeringstiden. Även med en välpresterande dator verkade användandet långsamt. I SolidWorks optimeringsprocessen spelade lagrings utrymmet en stor roll eftersom det ofta förekom ett error som klagade på för lite lagringsutrymme för att kunna fortsätta optimeringen. Detta error- meddelande förekom även efter att optimeringen var 99 % färdig. Istället användes en 32 GB USB-sticka dit alla resultatfiler sparades istället.

Detta problem påverkade möjligheten att inspektera delen efter optimeringen eftersom SolidWorks inte ville visa den optimerade delens FEA- analys och därefter kunde inte en ny optimering föras till slut.

I framtiden kunde standarddelens yttre ytor utökas för att ge optimeringen ett större design rum att hitta den optimala strukturen. Det skulle dock medföra behovet att försäkra styrelsens rörelse så att det inte kolliderar med design rummet.

SolidWorks Topology Study- verktyg är integrerat i programmet som en add-in och introducerades till programmet år 2018 vilket kunde ha effekt på hur väl det fungerade. 3D Printningen av alla delar gick väl och inga problem uppstod under printningen. Av någon orsak kunde MakerBot Print programmet inte identifiera den Inspire optimerade delen och därför öppnades modellen först i Cura, sparades till STL-fil varefter MakerBot Print kunde läsa modellen. Priset att 3D Printa är så högt att endast en tillräckligt kombinerad viktminskning kunde berättiga köpet. Modellstoleken behölls lika genom hela processen och därför behövdes inte delarna skalas

12 SAMMANFATTNING

Topologisk optimering är ett effektivt redskap för att minska delars vikt men samtidigt behålla dess styvhet. Det tillåter ett effektivt sätt för design ingenjörer att förbättra existerande utformning. I användningsområden där massan spelar en stor roll kan en topologisk optimerad del ha en stor effekt ifall den används i stora mängder i en konstruktion eller ifall den optimerade delen är stor och fungerar som en självständig del. Ett sådant fall skulle föra med sig betydliga besparingar i drift som i t.ex. inom luftfartsindustrin.

Utav standard fästelementet lyckades två nya topologiskt optimerade fästelement skapas genom att använda solidThinking Inspire och SolidWorks. Till optimeringsprocessen hörde importen av CAD- modellen, utförande av FEA- analys, bestämning av typen av optimering, utse design- och icke design utrymme, köra den topologiska optimeringen och jämna ut delens yta. Båda optimeringar minskade vikten och minimera delens styvhet.

Med Inspire minskade delens massa med 66 % medan dess styvhet ökade. Största mekaniska påfrestningen hittades i de två bakre monteringshålerna som dock behöll en Factor of safety på 1. Beroende på fästelementets krav kunde denna optimerade del användas eller genom att tillsätta material i dessa områden kunde påfrestningen minskas.

För någon som redan kan använda solidthinking Inspire eller SolidWorks kräver det inte mycket övning att lära sig använda de inbyggda topologiska redskapen och utnyttja hela den topologiska optimeringsprocessen. För studerande som är intresserade av 3D CAD är detta ett värdefullt redskap som är värt den investerade tiden.

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14 FIGURES

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APPENDICES

Since there are many different ways of reaching the same final result when creating a part in CAD, the method demonstrated below was one way of creating the mounting bracket based on the drawing provided by Parker Hannifin. Since the mounting bracket was mostly symmetrical time could be saved by only modelling half of the bracket and using the mirror command to create the other half. The bracket was designed in the following way: first, the baseplate for half the bracket was sketched to 47.5 mm x 80 mm and extruded to 13 mm thick. Now the gap width in between the two bracket halves was cut out by sketching a rectangle over the edge and making sure 1 mm was covering the top of the base. This sketch was then used to cut out 7 mm deep using the extrude cut- tool.

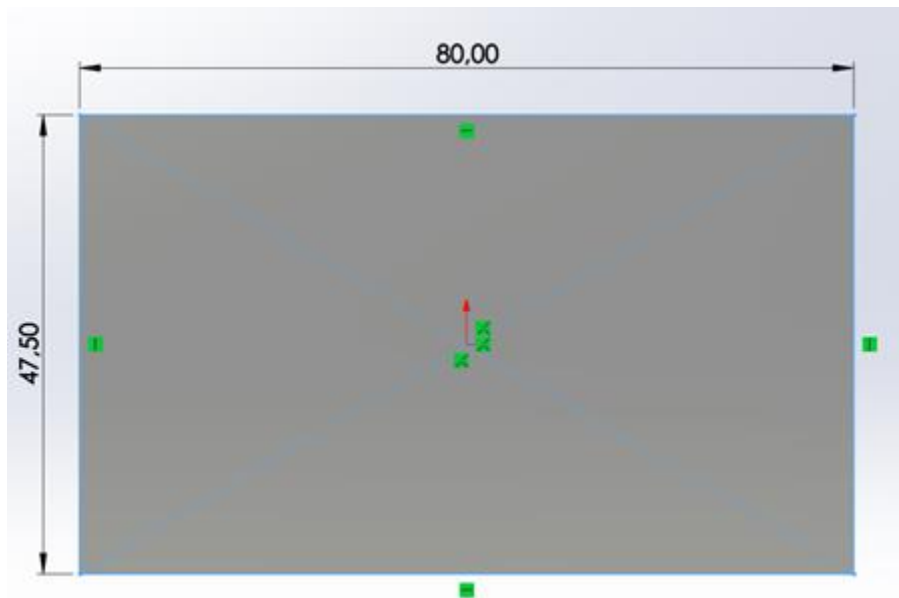


Figure 28. Sketching the base (Oliver Folger, 2019)

Now a circle could be drawn on the same level as the previously cut out part. The 16 mm diameter circle was positioned in place according to the measurements, 74 mm from the back of the base and 50 mm up from the bottom. This circle could now be used as reference to draw the actual bracket part which would hold the pivot pin. A circle with 32 mm diameter was sketched on top and two lines reaching from the back end and front end of the base plate were connected using tangent constraints. This gave a smooth transition

from a straight line to the curve of the circle. Now the extra lines from the circle could be trimmed of using the trim- function thus leaving a sketch for one whole bracket half ready to be extruded. The sketch was then extruded 13 mm. Now the bracket was more recognizable.

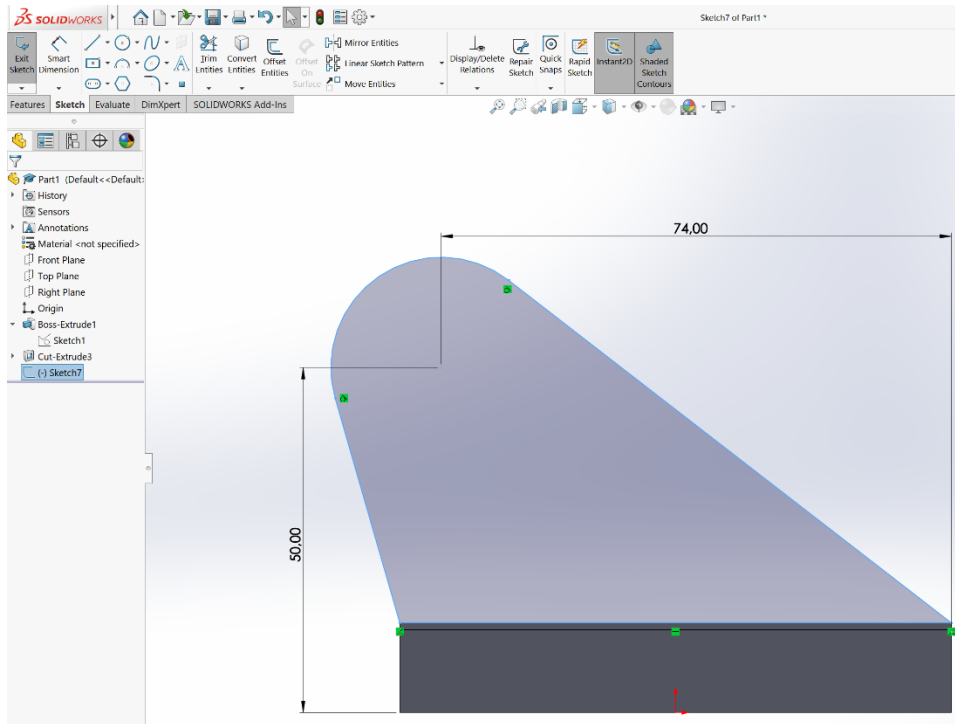


Figure 29. Sketching the mounting bracket upper half (Oliver Folger, 2019)

The same hole which held the pivot pin was redrawn and cut out.

Next the sketch was extruded cut according to the specified thickness of 13 mm.

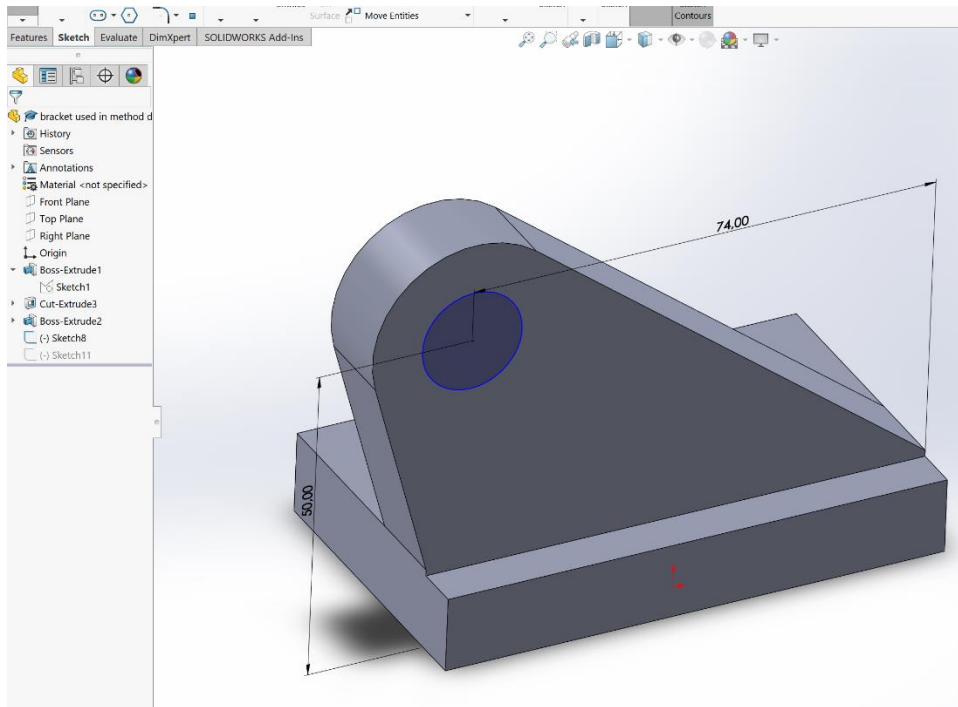


Figure 30. Mounting hole placement dimensions (Oliver Folger, 2019)

The mounting holes were sketched on the base plate and dimensioned according to the drawing. They were then cut out again using the extruded cut command.

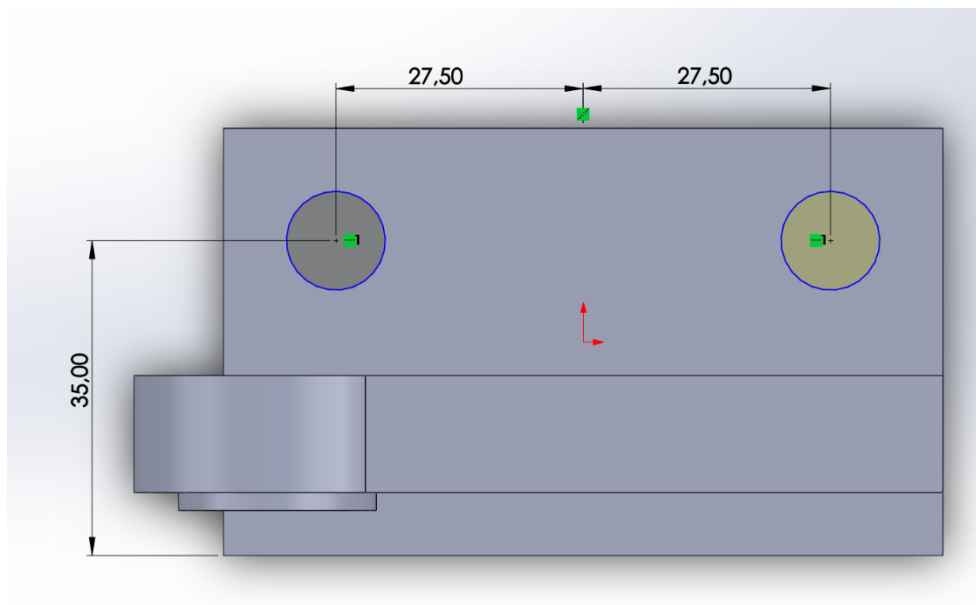


Figure 31. Setting correct hole placements with the dimension- tool (Oliver Folger, 2019)

A few details were added such as the extruded spacer around the pivot pin hole securing the pin in place when mounted and adding an additional 1 mm thickness to the outside of the bracket. At this stage half the mounting bracket was complete and could be mirrored to create the other half. Only a plane had to be created on the inner surface of the base plate. To create the mirrored part the mirror tool was chosen, plane 2 set as mirror face and all features created thus far selected.

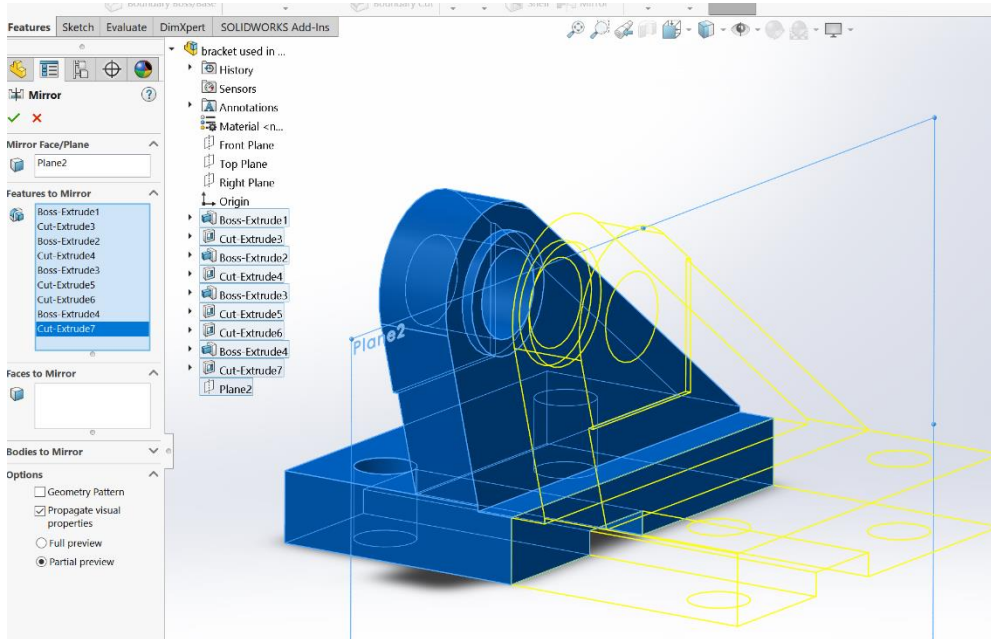


Figure 32. Using mirror tool to create the other half (Oliver Folger, 2019)

The part was then completed and could be saved and imported to Inspire for the topology optimization process.

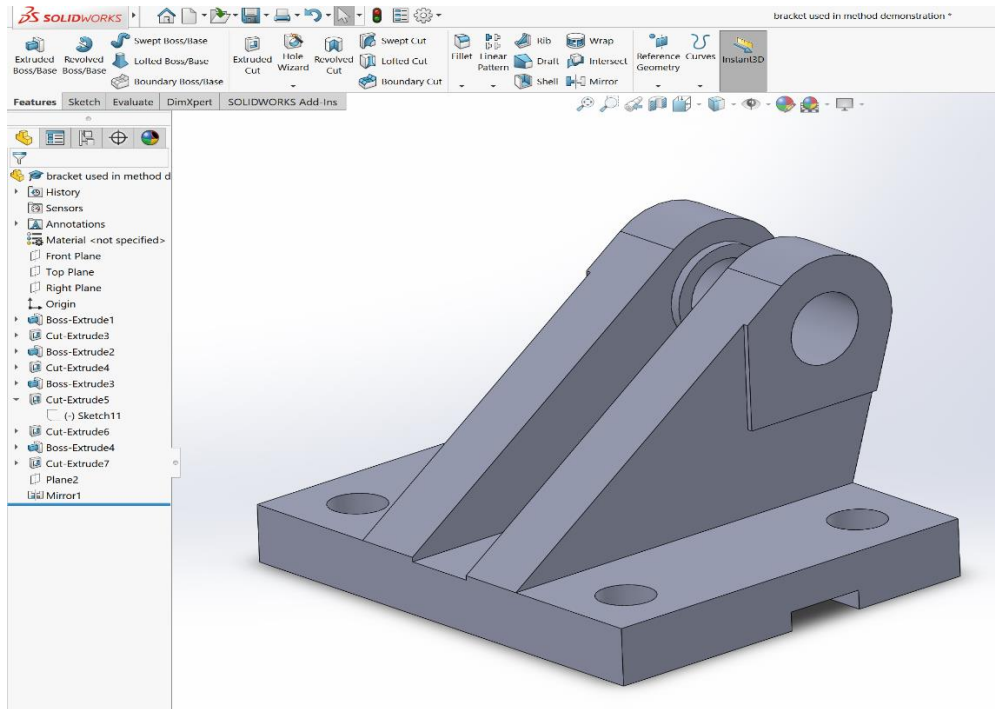
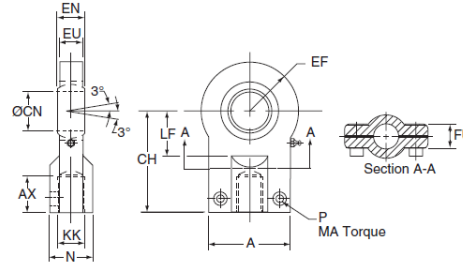


Figure 33. Finished bracket 3D modelled in SolidWorks (Oliver Folger, 2019)

Metric Accessories

Rod Eye with Spherical Bearing, Mounting Bracket and Pivot Pin

KK Thread	Rod Eye with Spherical Bearing	Mounting Bracket and Pivot Pin	Nominal Force kN	Weight kg
M10x1.25	1452540000	1455300000	8	0.2
M12x1.25	1452550000	1455310000	12.5	0.3
M14x1.5	1452560000	1455320000	20	0.4
M16x1.5	1452570000	1455330000	32	0.7
M20x1.5	1452580000	1455340000	50	1.3
M27x2	1452590000	1455350000	80	2.3
M33x2	1452600000	1455360000	125	4.4
M42x2	1452610000	1455370000	200	8.4
M48x2	1452620000	1455380000	320	15.6
M64x3	1452630000	1455390000	500	28.0



Rod Eye with Spherical Bearing

All spherical bearings should be re-packed with grease when servicing. In unusual or severe working conditions, consult the factory regarding the suitability of the bearing chosen.

Rod Eye with Spherical Bearing Dimensions

Part No.	A max	AX min	EF max	CH	CN Ø	EN	EU	FU	KK Thread	LF min	N max	MA max Nm	P
1452540000	40	15	20	42	12 -0.008	10 -.012	8	13	M10x1.25	16	17	10	M6
1452550000	45	17	22.5	48	16 -0.008	14 -.012	11	13	M12x1.25	20	21	10	M6
1452560000	55	19	27.5	58	20 -0.012	16 -.012	13	17	M14x1.5	25	25	25	M8
1452570000	62	23	32.5	68	25 -0.012	20 -.012	17	17	M16x1.5	30	30	25	M8
1452580000	80	29	40	85	30 -0.012	22 -.012	19	19	M20x1.5	35	36	45	M10
1452590000	90	37	50	105	40 -0.012	28 -.012	23	23	M27x2	45	45	45	M10
1452600000	105	46	62.5	130	50 -0.012	35 -.012	30	30	M33x2	58	55	80	M12
1452610000	134	57	80	150	60 -0.015	44 -.015	38	38	M42x2	68	68	160	M16
1452620000	156	64	102.5	185	80 -0.015	55 -.015	47	47	M48x2	92	90	310	M20
1452630000	190	86	120	240	100 -0.020	70 -.020	57	57	M64x3	116	110	530	M24

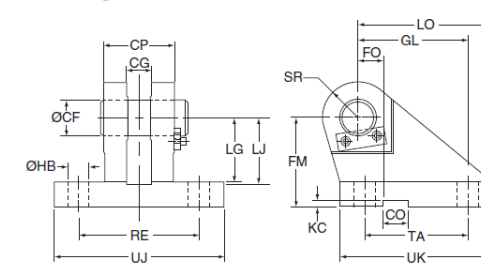
Mounting Bracket and Pivot Pin Dimensions

Part No.	CF Ø K7/h6	CG +0.1, +0.3	CO N9	CP	FM js11	FO js14	GL js13	HB Ø	KC 0, +0.30	LG	LJ	LO	RE js13	SR max	TA js13	UJ	UK
1455300000	12	10	10	30	40	16	46	9	3.3	28	29	56	55	12	40	75	60
1455310000	16	14	16	40	50	18	61	11	4.3	37	38	74	70	16	55	95	80
1455320000	20	16	16	50	55	20	64	14	4.3	39	40	80	85	20	58	120	90
1455330000	25	20	25	60	65	22	78	16	5.4	48	49	98	100	25	70	140	110
1455340000	30	22	25	70	85	24	97	18	5.4	62	63	120	115	30	90	160	135
1455350000	40	28	36	80	100	24	123	22	8.4	72	73	148	135	40	120	190	170
1455360000	50	35	36	100	125	35	155	30	8.4	90	92	190	170	50	145	240	215
1455370000	60	44	50	120	150	35	187	39	11.4	108	110	225	200	60	185	270	260
1455380000	80	55	50	160	190	35	255	45	11.4	140	142	295	240	80	260	320	340
1455390000	100	70	63	200	210	35	285	48	12.4	150	152	335	300	100	300	400	400

Cap Mounting Bracket and Pivot Pin

Bore Ø	Mounting Bracket and Pivot Pin	Nominal Force kN	Weight kg
25	1455300000	8	0.6
32	1455310000	12.5	1.3
40	1455320000	20	2.1
50	1455330000	32	3.2
63	1455340000	50	6.5
80	1455350000	80	12.0
100	1455360000	125	23.0
125	1455370000	200	37.0
160	1455380000	320	79.0
200	1455390000	500	140.0

Mounting Bracket and Pivot Pin



All dimensions are in millimeters unless otherwise stated.