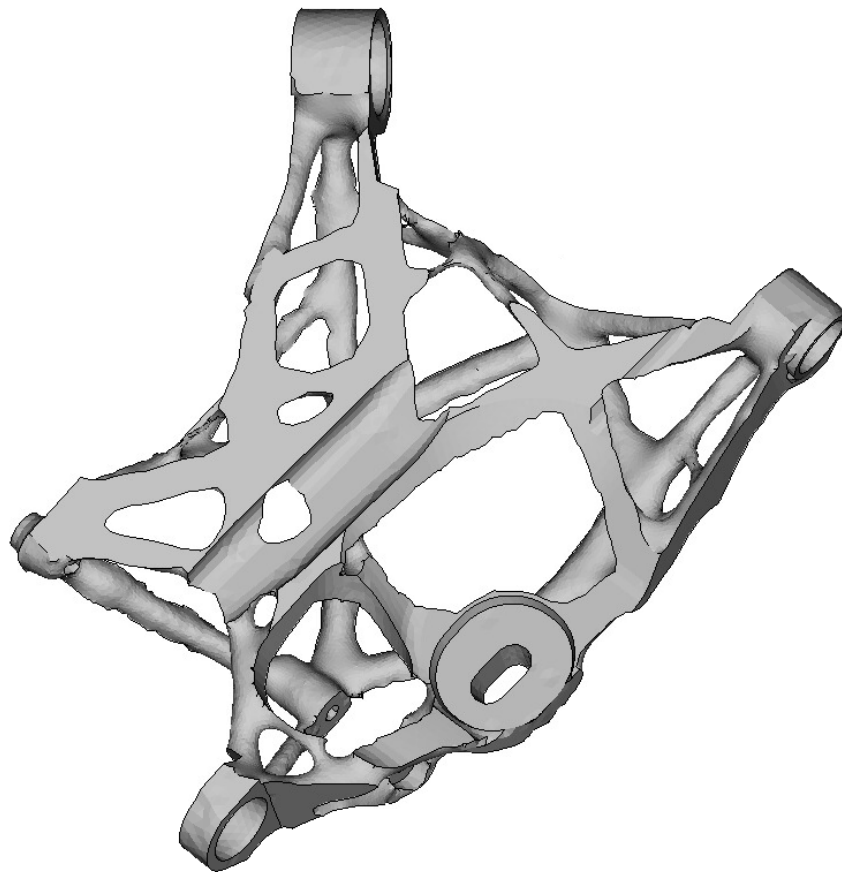




CHALMERS



Methodology for Topology and Shape Optimization: Application to a Rear Lower Control Arm

Master's thesis in Applied Mechanics

ROBIN LARSSON

MASTER'S THESIS IN APPLIED MECHANICS

Methodology for Topology and Shape Optimization: Application to a Rear
Lower Control Arm

ROBIN LARSSON

Department of Applied Mechanics
Division of Solid Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2016

Methodology for Topology and Shape Optimization: Application to a Rear Lower Control Arm
ROBIN LARSSON

© ROBIN LARSSON, 2016

Master's thesis 2016:30
ISSN 1652-8557
Department of Applied Mechanics
Division of Solid Mechanics
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: +46 (0)31-772 1000

Cover:
Topology optimized design of a rear lower control arm.

Chalmers Reproservice
Göteborg, Sweden 2016

Methodology for Topology and Shape Optimization: Application to a Rear Lower Control Arm
Master's thesis in Applied Mechanics
ROBIN LARSSON
Department of Applied Mechanics
Division of Solid Mechanics
Chalmers University of Technology

ABSTRACT

The selection of component material and design is an important topic in industry to produce sustainable and competitive products. To fulfil strength and endurance requirements on a component level, topology and shape optimization can be used as design tools in early phases of the design process. Topology and shape optimization are sub-fields within structural optimization. A component design could actually be constructed based on topology and shape optimization tools throughout the complete component development process. At the Endurance Attribute and Chassis CAE group at Volvo Cars in Gothenburg, the interest of finding a methodology for topology and shape optimization as a natural part of their component development process increases nowadays. The purpose of this thesis is therefore to develop such a methodology with respect to topology and shape optimization. The structural optimization work is carried out in the commercial software TOSCA. In order to define optimization tasks, this requires knowledge of all steps taken during the complete component development process. Furthermore, this thesis focuses on a rear lower control arm component where the structural requirements on that component involves pre-tension, plastic hardening material behaviour and fatigue problems which are treated during the optimization process. Manufacturability and implementation of manufacturability constraints to the optimization tasks are also considered. The topology optimization part involves linear static finite element assumptions whilst the shape optimization part involves both linear and nonlinear finite element analysis. Furthermore multi-objective shape optimization is performed where both equivalent plastic strain and fatigue life are treated. The topology and shape optimized structures requires realization due to manufacturability. These steps are performed by a Design Engineer at the Wheel suspension group at Volvo Cars. Furthermore, it is the Design Engineer who defines the proposed manufacturing method to the component based on the topology optimized design. A component development process using structural optimization tools is suggested and demonstrated by subjecting trial cases to the process. The process is thereafter discussed and further work is suggested, both additional manufacturability verification and simulation are proposed in context of the component development process using structural optimization tools.

Keywords: Structural optimization, Cast components, Multi-objective shape optimization

PREFACE

This master thesis in Applied Mechanics at Chalmers University of Technology comprises 30 credits and was carried out at the Endurance Attribute and Chassis CAE group at Volvo Cars in Gothenburg during the spring 2016. The examiner and academic supervisor was Håkan Johansson, Associate Professor at the Division of Dynamics, Applied Mechanics, Chalmers University of Technology. The supervisor in industry was Iris Blume, CAE engineer at the Endurance Attribute and Chassis CAE group, Volvo Cars, Gothenburg.

This thesis was one of three master thesis projects in a so-called thesis cluster, carried out during the spring of 2016, where the common goal was to increase the usage and knowledge of optimization methods used for automotive wheel suspension development at Volvo Cars in Gothenburg. During the thesis work, meetings and presentations were held where problems and progress were discussed between the groups. The master thesis project titles are listed below:

1. Optimization of a wheel suspension packaging [9]
2. Balancing of wheel suspension packaging, performance and weight [11]
3. Methodology for Topology and Shape Optimization: Application to a Rear Lower Control Arm

ACKNOWLEDGEMENTS

First of all I want to thank my supervisor Iris Blume for her support and helpfulness with the thesis work. I would also like to thank my academic supervisor Associate Professor Håkan Johansson for his inputs and thoughts on the work. I would like to thank Joakim Weglin for his work with design realizations of optimization results and Christoph Schutte for helping me with manufacturability assessment. Furthermore, I would like to thank the thesis cluster coordinator Ph.D Harald Hasselblad and my fellow thesis workers Mattias Linder, Karl Hansen Andreasson, Joakim Skön and Kanishk Bhadani at Volvo Cars for their feedback and support during the thesis work. Lastly, I would like to thank my colleagues at the Endurance Attribute and Chassis CAE group for their support and friendliness during the time spent at Volvo Cars.

ABBREVIATIONS

RLCA	Rear Lower Control Arm
FE	Finite Element
MBS	Multi Body Simulation
RLD	Road Load Data
OC	Optimality Criteria method
MMA	Method of Moving Asymptotes
EPS	Equivalent Plastic Strain
CAE	Computer Aided Engineering
CAD	Computer Aided Design
DOC	Drive Over Curb
ROC	Rearwards drive Over Curb
BIP	Brake In Pothole
SAC	Skid Against Curb

CONTENTS

Abstract	i
Preface	iii
Acknowledgements	iii
Abbreviations	v
Contents	vii
1 Introduction	1
1.1 Background	1
1.2 Purpose	1
1.3 Limitations	1
1.4 Method	2
1.5 Thesis outline	2
2 Theory	3
2.1 Introduction to Structural optimization	3
2.2 Topology optimization	4
2.2.1 Material interpolation	4
2.2.2 The checkerboard problem	5
2.2.3 Problem formulation	5
2.2.4 Multiple load cases	6
2.2.5 Solution methods	7
2.2.6 Application in TOSCA	7
2.3 Shape optimization	7
2.3.1 Application in TOSCA	7
3 The component development process	9
3.1 The current component development process	9
3.2 The component development process using structural optimization tools	10
4 Application to a Rear Lower Control Arm	13
4.1 The current RLCA component	13
4.1.1 FE model	13
4.1.2 FE analysis	14
4.2 CAE requirements	14
4.3 Parameter study	16
4.3.1 Optimization algorithm	16
4.3.2 Member size	16
4.3.3 Demold constraint	16
4.3.4 Load cases	16
4.4 Methodology application	17
4.4.1 Design realization	17
4.5 Trial case 1	18
4.5.1 Design volume	18
4.5.2 Topology optimization	18
4.5.3 Design realization	19
4.5.4 FE analysis	19
4.5.5 Discussion	20
4.5.6 Conclusions	21
4.6 Trial case 2	21
4.6.1 Design volume	21

4.6.2	Topology optimization	22
4.6.3	Design realization	22
4.6.4	FE analysis	22
4.6.5	Shape optimization	23
4.6.6	Design realization	23
4.6.7	FE analysis	23
4.6.8	Discussion	24
4.6.9	Conclusions	24
4.7	Trial case 3	25
4.7.1	Design realization	25
4.7.2	FE analysis	25
4.7.3	Shape optimization	26
4.7.4	Design realization	27
4.7.5	FE analysis	27
4.7.6	Discussion	30
4.7.7	Conclusions	30
4.8	Trial case 4	31
4.8.1	Design realization	31
4.8.2	FE analysis	31
4.8.3	Discussion	33
4.8.4	Conclusions	33
5	Discussion	34
5.1	Trial cases	34
5.2	The component development process using structural optimization tools	34
5.3	Application to other components	35
6	Further work	36
6.1	Trial cases	36
6.2	The component development process	36
	References	37
A	Additional topology parameter study results	39
A.1	Membersize constraint	39
A.2	Demold constraint	39
A.3	A comparison between sensitivity based and controller based solution methods	39

1 Introduction

This chapter introduces the thesis problem formulation to the reader, it starts with a background to structural optimization and its application at the Endurance Attribute and Chassis CAE group at Volvo Cars in Gothenburg. It is followed by the purpose and limitations of the thesis. Thereafter, the method where the disposition and approach of work is explained. Lastly, the thesis outline is explained.

1.1 Background

The selection of component material and design is an important topic in industry to produce sustainable and competitive products. To fulfil strength and endurance requirements on a component level, topology and shape optimization are useful tools to predict an optimal component design in early phases of the design process. A complete design of a component could be constructed by utilizing topology and shape optimization throughout the entire component development process. Although, mathematical concepts used for structural optimization tools are well established, its application in industry is not that well established and needs therefore to be investigated further. At the Endurance Attribute and Chassis CAE group at Volvo Cars in Gothenburg, the interest of finding a suitable methodology for topology and shape optimization as a natural part of their component development process increases nowadays.

A topology optimization in a finite element context modifies the connectivity of finite elements with respect to a pre-defined objective with associated constraints. An example is that the maximum stiffness of a structure is sought for a given amount of material. Furthermore, it is convenient to assume linear isotropic material behaviour with small deformation theory. By performing a shape optimization on a structure, its shape in terms of thickness and radius is varied where non-linear and fatigue material behaviour can be taken into account. As the need to cut lead times in the product development process as well as the need to reduce weight of automotive vehicles increases, it becomes more natural to include topology and shape optimization in early phases of the component development process.

This thesis focuses on a rear lower control arm (RLCA) component, belonging to the rear wheel suspension of an automotive vehicle at Volvo Cars. Requirements for these system components are verified by physical testing of strength events and endurance testing. Directional stiffness at some locations are required. Furthermore, strength events are tested, these represents missuses that could occur during a lifetime of an automotive vehicle. An example of that could be if the vehicle is driven over a curb. Road load data (RLD) is generated by multi body simulations (MBS) of a full vehicle model subjected to strength events and endurance tests, this gives forces and moments on a component level.

1.2 Purpose

The purpose of this thesis is to establish a methodology for structural topology and shape optimization in early phases of the component development process, where the application is a RLCA component. The methodology will include optimization related subjects such as objective function, constraints and also so-called manufacturing constraints for both the topology and shape optimization parts. Furthermore, load cases and boundary conditions which are relevant to include with respect to the optimization process are treated.

1.3 Limitations

The structural requirements to consider are directional stiffness, plastic strain, permanent deformation and damage caused by fatigue below a certain tolerance for different load cases. Hence, the requirements introduces finite element problems of both linear and non-linear structural behaviour. Although, only static linear FE problems are considered in the topology optimization part. The initial design volume is three dimensional and assumed to be fix. Furthermore, the topology optimization considers multiple load case and multiple constraints. The shape optimization part include static elasto-plastic material behaviour, pre-tension and fatigue analysis.

1.4 Method

The thesis work starts with learning software used for Finite Element (FE) modelling. Thereafter, topology optimization tutorials are studied. These are followed by a literature study to increase the knowledge of how topology and shape optimization is used nowadays. The methodology development can be divided into two parts. The first part focuses on topology optimization where design space, boundary conditions, objective function with associated constraints and multiple load cases are treated. Furthermore, design constraints due to the manufacturing process are considered. The influence of the finite element order and grid size are treated. The design volume for the considered component together with stiffness requirements are provided by the Chassis design group at Volvo Cars, from where a three dimensional FE-model is constructed. The CAE load cases and requirements for strength events and the chassis rig cycle are provided by the Endurance Attribute and Chassis CAE group at Volvo Cars. The second part focuses on shape optimization, where plastic deformation and fatigue life are treated.

The commercial software used for pre-process FE modeling is ANSA. Both the linear and non-linear static finite element analysis are solved using ABAQUS whereas TOSCA solves the topology and shape optimization problems. Two methods to solve the topology optimization problem are available in TOSCA, namely the controller based *Optimality Criterion* (OC) and the sensitivity based *Method of Moving Asymptotes* (MMA). Fatigue simulations are performed in nCode Designlife. Post-process work is performed in μ ETA and TOSCA viewer.

1.5 Thesis outline

The thesis starts with explaining the mathematical concepts used for structural optimization used in this thesis and its application in the commercial software TOSCA. It is followed by a description of both of the current component development process with respect to the wheel suspension at Volvo Cars and the component development process using structural optimization tools used in this thesis. Thereafter, the application of the RLCA to the component development process using structural optimization tools is presented where trial cases are subjected to the optimization process. Lastly, the methodology used for structural optimization is evaluated and further work is suggested.

2 Theory

The general mathematical concepts used to formulate the structural topology and shape optimization problems during this thesis work are explained in this chapter. A brief introduction to structural optimization is firstly presented which is followed by an explanation of topology optimization and its application in the commercial software TOSCA. At last, shape optimization and its application in TOSCA are presented.

2.1 Introduction to Structural optimization

To formulate the structural optimization problem, an objective function, design variables and state variables needs to be introduced as described in [7]. The objective function (f), represents an objective that could either be minimized or maximized. A typical objective could be the stiffness or volume of a structure. Furthermore, some structural design domain and state variables associated to the objective function needs to be defined. The design variables ($\mathbf{x}$) describes the design of the structure, it may represent the geometry. The state variables ($\mathbf{y}$) represents the structural response which can for example be recognized as stress, strain or displacement. Furthermore, the state variables depends on the design variables $\mathbf{y}(\mathbf{x})$. The objective function is subjected to the design and state variable constraints to steer the optimization to a sought solution.

$$\begin{cases} \min_{\mathbf{x}} & f(\mathbf{x}, \mathbf{y}(\mathbf{x})) \\ \text{subject to} & \begin{cases} \text{design constraint on } \mathbf{x} \\ \text{state constraint on } \mathbf{y}(\mathbf{x}) \\ \text{equilibrium constraint} \end{cases} \end{cases} \quad (2.1)$$

A state function $g(\mathbf{y})$ that represents the state variables can be introduced, for example a displacement in a certain direction. This state function can be incorporated as a constraint to the optimization task, where it is usually formulated such that $g(\mathbf{y}) \leq 0$. Consider the case where $g(\mathbf{y})$ is represented by a displacement vector $g(\mathbf{u}(\mathbf{x}))$ in a discrete finite element problem. To establish the state function, this requires that nodal displacement are solved for

$$\mathbf{u}(\mathbf{x}) = \mathbf{K}(\mathbf{x})^{-1} \mathbf{f}(\mathbf{x}) \quad (2.2)$$

where $\mathbf{K}$ is the global stiffness matrix and $\mathbf{f}$ is the global load vector. This means that the optimization task can be expressed in a so-called nested formulation where the equilibrium constraint is taken care of by the state function formulation

$$\begin{cases} \min_{\mathbf{x}} & f(\mathbf{x}) \\ \text{subject to} & g(\mathbf{u}(\mathbf{x})) \leq 0 \end{cases} \quad (2.3)$$

The optimization task presented in equation (2.1) is called simultaneous formulation in comparison. Equation 2.3 is usually solved by evaluating derivatives of f and g with respect to $\mathbf{x}$. In this context, $\mathbf{x}$ will represent a geometrical feature. Based on what geometrical feature that is parametrized, the structural optimization problem can be classified into:

- **Size optimization:** the design variable $\mathbf{x}$, represents a structural thickness such as a distributed thickness or a cross-sectional area of a truss model that can be varied. The optimal thickness typically minimizes some physical quantity such as the strain energy (compliance) or the deflection, while the equilibrium constraint has to be fulfilled. The state function may then relative volume.
- **Shape optimization:** the design variable $\mathbf{x}$, represents the boundary of the state equation. In this case, the boundary of the considered domain $\mathbf{x}$ could vary such that some physical quantity is minimized.
- **Topology optimization:** the design variable $\mathbf{x}$, represents the connectivity of the domain. It involves features such as number and sizes of holes in the design domain.

The objective function can also be formulated using several objectives, it is then often called a multi objective or a vector optimization problem:

$$\min_{\mathbf{x}} f(f_1(\mathbf{x}, \mathbf{y}), f_2(\mathbf{x}, \mathbf{y}), \dots, f_n(\mathbf{x}, \mathbf{y})) \quad (2.4)$$

where n is the number of objective functions. Since all objectives are minimized with respect to $\mathbf{x}$ and $\mathbf{y}$, a global optimum is not distinct. The objectives can be formulated as a scalar formulation of the objective functions using weights

$$f = \sum_i f_i w_i \quad (2.5)$$

where i is the single objective function index and the total sum of the set of weights are

$$\sum_i w_i = 1 \quad (2.6)$$

By varying the set of weights, different so-called Pareto optimal points can be found where these solutions are unique with respect to the associated weight set. The set of different Pareto optimal points gives a Pareto set, where no objective can be improved without worsen another.

2.2 Topology optimization

We seek an optimal placement of material points where the reference domain is partitioned into void and solid elements by a finite element discretization.

2.2.1 Material interpolation

In mathematical terms we seek an optimal subset $\Omega_{mat} \subset \Omega$. Where Ω is an available design domain. The design variable $\mathbf{x}$ is now represented by the density vector $\boldsymbol{\rho}$ containing elemental densities ρ_e . The local stiffness tensor $\mathbf{E}$ can be formulated by incorporating $\boldsymbol{\rho}$ as a integer formulation

$$\begin{aligned} \mathbf{E}(\boldsymbol{\rho}) &= \boldsymbol{\rho} \mathbf{E}^0 \\ \rho_e &= \begin{cases} 1 & \text{if } e \in \Omega_{mat} \\ 0 & \text{if } e \in \Omega \setminus \Omega_{mat} \end{cases} \end{aligned} \quad (2.7)$$

and a volume constraint

$$\int_{\Omega} \boldsymbol{\rho} d\Omega = \text{Vol}(\Omega_{mat}) \leq V \quad (2.8)$$

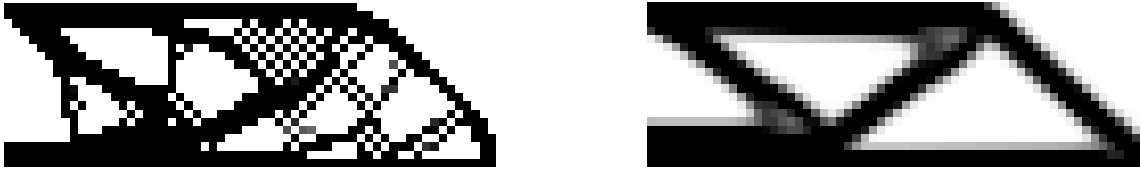
V is the volume of the initial design domain. When $\rho_e = 1$ we consider an element to be filled whereas an element with $\rho_e = 0$ is considered to be a void element. To use a gradient based solution strategy for the optimization problem, the integer problem described in (2.7) needs to be formulated as a continuous function so that the density function can take values between 0 and 1 [4]. The most common method to relax the integer problem is the SIMP (*Solid Isotropic Material with Penalization*) method. The density function is then written as

$$\mathbf{E} = \boldsymbol{\rho}^p \mathbf{E}^0, \quad \boldsymbol{\rho} \in [\rho_{min}, 1], \quad p > 1 \quad (2.9)$$

where p is the penalizing factor that penalizes elements with intermediate densities to approach 0 or 1, ρ_{min} is the lower density value limit to avoid singularities. Thus, the penalization is achieved without introducing any explicit penalization scheme. For materials with Poisson ratio $\nu = 0.3$, it is recommended in [5] to use $p \geq 3$.

2.2.2 The checkerboard problem

Checkerboarding refers to the problem where optimization results shows elements which are alternating solid and void in a checkerboard like pattern. It was earlier believed that these regions represented some optimal microstructure design but proved to be due to poor stiffness representation using finite elements [14]. An illustration of the checkerboard problem for a two dimensional problem is produced by the MATLAB code described in [2]. It is presented in Figure 2.1, where it can be seen that the checkerboard pattern occurs in Figure 2.1a. Looking at Figure 2.1b, where a sensitivity filter to mitigate the checkerboard phenomenon is applied, it can be seen that the material points are placed more homogeneously. Furthermore, higher order elements and mesh refinement could also mitigate the checkerboard problem.



(a) No filter

(b) Sensitivity filter

Figure 2.1: Illustration of the consequence by applying a sensitivity filter or not.

Sensitivity filter

Computational experience has shown that a sensitivity filter is highly beneficial when searching for a mesh independent solution whilst not adding on to much computational time or any extra constraints. The design sensitivity is therefore modified based on a weight average of the neighbourhood elements.

The filter scheme modifies the element sensitivities with respect to the objective function as

$$\frac{\partial f^{\text{new}}}{\partial \rho_k} = \frac{1}{\rho_k \sum_{i=1}^n \tilde{H}_i} \sum_{i=1}^n \tilde{H}_i \rho_i \frac{\partial f}{\partial \rho_i} \quad (2.10)$$

where the weight factor $\tilde{H}_i$ is based on the distance to neighbourhood elements as $\tilde{H}_i = r_{\min} - \text{dist}(k,i)$. Furthermore, $\text{dist}(k,i)$ is the distance between the center of the considered element k and the neighbourhood element i . The neighbourhood elements are defined within a circle with the filter radius $r_{\min}$.

2.2.3 Problem formulation

The optimization problem formulated in a nested formulation in equation (2.3) is now written as

$$\begin{cases} \min_x & f(\boldsymbol{\rho}) \\ \text{subject to} & \begin{cases} \mathbf{0} \leq \boldsymbol{\rho} \leq \mathbf{1} \\ \text{State function constraint} \\ \text{Manufacturing constraints} \end{cases} \end{cases} \quad (2.11)$$

when considering topology optimization using the SIMP interpolation method, $\boldsymbol{\rho}$ is a vector containing the element densities. Two common objectives to be minimized are the compliance (C) and the volume (V). An

example of a state function constraint can be a displacement in a certain direction as mentioned in section 2.1.

Minimize compliance

A possibility to maximize the global stiffness of a structure is to minimize its compliance. The compliance is therefore defined as the equivalent strain energy of the FE solution which yields higher stiffness when minimized. The compliance is defined as

$$C(\boldsymbol{\rho}) = \mathbf{f}^T \mathbf{u} \quad (2.12)$$

where $\mathbf{u}$ solves the equilibrium equation

$$\mathbf{K}(\boldsymbol{\rho})\mathbf{u} = \mathbf{f} \quad (2.13)$$

where $\mathbf{K}(\boldsymbol{\rho})$ is

$$\mathbf{K}(\boldsymbol{\rho}) = \sum_{e=1}^{nel} \rho_e^p \mathbf{K}_e^0 \quad (2.14)$$

$\mathbf{K}_e^0$ is the elemental stiffness matrix with the initial stiffness tensor $\mathbf{E}^0$. To prevent the optimized structure from ending up with the full design volume as a result when searching for its maximum structural stiffness, we need to impose a volume constraint. If a gradient based approach is used, derivatives with respect to $C(\boldsymbol{\rho})$ are evaluated.

Minimize volume

Another possibility is to minimize the volume.

$$V(\boldsymbol{\rho}) = \sum_{e=1}^{nel} \rho_e^p V_e^0 \quad (2.15)$$

where V^0 is the initial volume. To prevent the optimization from minimizing all material, we need for example to impose a constraint for maximum displacement or effective stress. The optimization task is carried out with respect to the objective function and constraints. However, if the objective function is formulated with respect to volume or weight, derivatives are evaluated with respect to the constraints.

If a gradient based solution method is used, the derivatives are evaluated with respect to the constraint instead of the objective. For example, if a displacement vector is imposed, the so-called state derivatives with respect to $\mathbf{u}(\boldsymbol{\rho})$ are evaluated.

2.2.4 Multiple load cases

Similar to the multi-objective optimization formulation described in equation (2.4), the case of multiple load cases can be incorporated to the structural optimization task as

$$f = \sum_{k=1}^M f_k w_k \quad (2.16)$$

by using weights and objective subjected to a specific load case p with index k . M is the total amount of load cases.

2.2.5 Solution methods

Two common solutions methods for topology optimization are the Optimality Criteria (OC) and the Method of Moving Asymptotes (MMA), both of which are described more in detail in [4].

The MMA is similar to other mathematical programming algorithms such as Sequential Linear Programming (SLP) and Sequential Quadratic Programming (SQP) to solve non-linear optimization problems in the sense that they also uses sequences with sub-problems which are approximations of the original problem. For MMA, these sub-problems are constructed by gradient information, furthermore these approximations are assumed to be convex.

The OC method uses the method of Lagrange multipliers to establish the optimization task where compliance is minimized under a volume constraint. This defines an update scheme for the design densities ρ . For simple compliance optimization problems, the OC may be faster but for more complicated problems involving several load cases and constraints, the MMA gives better convergence.

2.2.6 Application in TOSCA

The commercial structural optimization tool TOSCA offers two types of optimization algorithms for topology optimization as described in [3].

- *The controller based algorithm* is a modified OC method which is based on stresses and limited to stiffness optimization under a volume fraction constraint. Sensitivities are not evaluated since the controller uses strain energy and stresses as an input. Furthermore, the algorithm produces clear void or solid elements when solving the optimization problem.
- *The sensitivity based algorithm* utilizes MMA.

Manufacturing constraints can be imposed to enable geometrical design aspects or manufacturing restriction to be taken into account. A manufacturing constraint is not the same type of a constraint as a mathematical optimization constraint. These can be violated, although the optimization strives to fulfil them.

- *Demold constraints* is used to adapt the topology optimized structure to a specific manufacturing procedure. Specific draw directions can be defined to steer where from the elements are eliminated during the optimization procedure.
- *Member size constraints* are used to specify a minimum and/or a maximum member size.

2.3 Shape optimization

The shape of a structure can be optimized by controlling its boundary. A typical shape optimization task could be to minimize the effective stress at some local regions. The available shape optimization methods can be divided into parametric and non-parametric methods as described in [12]. A parametric shape typically represents CAD parameters such as the radius of a fillet or the distance of a geometry section. Most of the commercial CAD software uses parametric shape representation. However, the parametric shapes does not contain any explicit information of the geometry or the topology of the boundary [6]. Non-parametric shape optimization uses implicit parameters, these are defined from a set of chosen surface nodes from the FE model. These nodes are denoted as design nodes. The implicit parameters are defined as the scalar displacements along the optimization vectors belonging to the design surface nodes. The optimization vector is usually taken as the normal vector to the adjacent surface of the design node.

2.3.1 Application in TOSCA

TOSCA uses a controller based nonparametric gradient-less shape optimization method which is based on the optimality criteria method mentioned in section 2.2.5. The procedure is to minimize the deviation of reference objective, such as the effective stress which is based on stress homogenization. It means that the stress along a specified zone strives to be constant. Furthermore, a volume fraction constraint can be imposed if needed. TOSCA uses the following re-design strategy which is based on stress homogenization:

- Design nodes with objective value above the reference value are moved in positive direction relative to its adjacent normal vector.

- Design nodes with objective value below the reference value are moved in negative direction relative to its adjacent normal vector.

The objective function is formulated to minimize the maximum objective for the respective objective

$$\begin{cases} \min_x & \max |F_k(\mathbf{X}) - F_{ref}| \\ \text{such that} & \Gamma^* \in \Gamma \end{cases} \quad (2.17)$$

where F_{ref} is by default in TOSCA set to the average objective value in the design nodes, k is the respective load case. The design boundary is denoted Γ^* which belongs to the full boundary Γ . The heuristic redesign rule could be formulated as

$$\Delta X^{(n)} = \alpha(F - F_{ref}), \text{ with } F = \max_k(F_k) \quad (2.18)$$

where the increment α is determined by line search to find the steepest decent. Due to computational cost, TOSCA uses a controller algorithm for the displacement vector update. A manufacturing constraint can be imposed to specify the movement of displacement scalars, this is named a Grow or Shrink manufacturing constraint depending on if the nodes are moved in positive or negative normal direction.

3 The component development process

This chapter explains the different phases during the component development process. It starts with an identification of the steps taken during the current component development project for chassis components. Thereafter, the component development process using structural optimization tools used during this thesis work is explained.

3.1 The current component development process

The different steps taken during the current component development process are explained with respect to chassis components at Volvo Cars. There are various types of component developments processes going on depending on which project, therefore an example of a component development process is explained in Figure 3.1. The example process considers a traditional approach where structural optimization tools are excluded.

The process is initiated by defining the design volume representing the geometrical domain whereby component material is allowed. The choice is thereafter to either utilize a so-called carry over approach, where the design from a previous project is carried over to a new project. Otherwise, a new conceptual design is proposed. The new conceptual design is either based on engineering judgement or proposed using topology optimization tools. When considering the engineering judgement approach as shown in Figure 3.1, the amount of loops between the Design and CAE engineer could be rather many since the balancing between structural weight and strength is performed manually. If topology optimization is considered, various types of commercial structural optimization software and methods are used depending on which project.

The next step in the process is to fulfil the standard CAE component requirements as seen in Figure 3.1. Most often, problematic regions of the design where for example the high effective stress or short predicted life appears, material is added. This procedure involves communication and looping between the Design and CAE engineer. In some cases, shape optimization is utilized to mitigate these problem regions with high effective stress peaks for example.

If the component requirements are fulfilled, the manufacturing method aspects are considered more thoroughly. If for example casting is chosen as the manufacturing method, the manufacturing process needs to be simulated. Many of the chassis components are manufactured by casting. The material solidification of the cast process is simulated to find out where to place gates for the cast influxes and to verify that the proposed design is feasible to cast.

The design proposal which is modified from the manufacturing process is then to be evaluated again with respect to the standard component requirement. At this stage the component is close to be finished, although the material which is added might not be beneficial with respect to structural performance. Therefore, design changes could be done to mitigate hot spots caused by gates. It follows that the design has to be analysed for strength and manufacturing again. The component design is completed when CAE, Design and the manufacturer has approved the component.

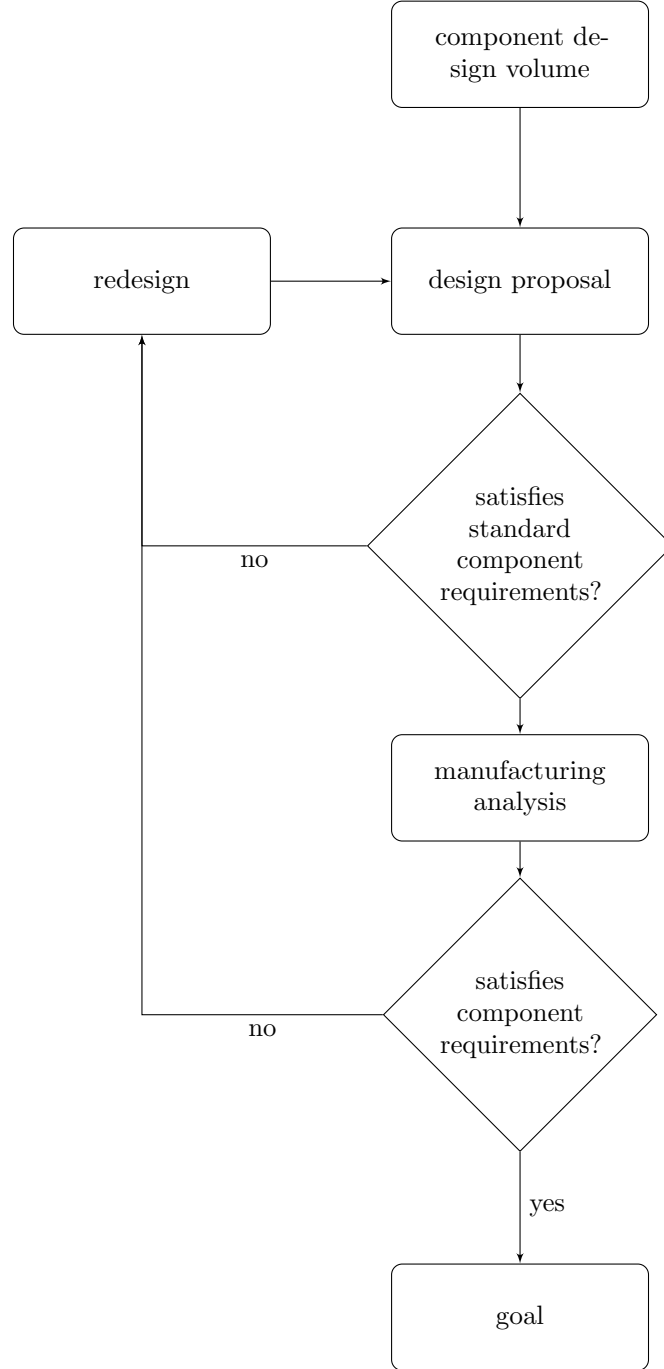


Figure 3.1: *An example of the flowchart using the traditional component development process.*

3.2 The component development process using structural optimization tools

The different phases during the component development process using topology and shape optimization tools are presented in Figure 3.2. The process starts with defining a component design volume. To formulate the design volume of a component associated to a system, one has to consider the expected motions of all components belonging to that system. However, in early phases of a project, all component design volumes can be balanced to find an optimal system behaviour with respect to some objective. This topic is discussed in the on-going master thesis [11] at Volvo Cars.

The next step in the process is to perform a topology optimization. Based on the component design volume, an FE model is constructed to represent the admissible structure of the design volume using linear FE assumptions. Furthermore, boundary conditions and load cases are introduced. For practical reasons, the amount of available load cases generated by MBS to include in a topology optimization is limited due to computational cost. Therefore, the most severe load cases are to be considered. So-called surrogate component requirements can be defined with respect to the topology optimization. Furthermore, the idea is to use a linear FE model for topology optimization. However, only the stiffness requirements are applicable to the topology optimization using a linear FE model. Therefore one of the challenges of this thesis is to find constraints using a linear FE model which can represent the requirements involving non-linear or fatigue material behaviour if so is possible. Due to that simplification, the result after topology optimization is not expected to fulfil all standard requirements. Hopefully, the structure after the topology optimization is not too far away from fulfilling the requirements involving nonlinear and fatigue FE analysis.

The topology optimization results causes most often irregular surfaces due to the elimination of elements during the optimization procedure. The optimized topology structure is then to be translated to a more smooth structure. At this step, the smoothed finite element grid is reviewed by a Design engineer. The smoothed design is redesigned with respect to the manufacturing method proposed by the Design engineer. This design realization step is rather crucial for the outcome of the structural performance of the new component. The topology optimization results are interpreted by the design engineer and a new component design based on the optimized design is constructed. This step is discussed more in detail in section 4.4.1 on page 17 in the context of a RLCA application.

The new component design is subsequently evaluated with respect to the standard FE component analysis. As mentioned earlier, the standard component requirements does not have to be fulfilled at this step. Although, an engineering judgement is required to determine whether the structural performance of the component is sufficient for shape optimization or not. This engineering judgement could be concretized into a so-called surrogate requirement. If the structural performance is too poor with respect to the standard component requirements a change of topology optimization configuration is required. Note that both the standard and surrogate requirements requires the same FE analysis.

Due to the use of linear FE models in topology optimization, this yields that requirements of non-linear or fatigue characteristics needs to be taken care of by another structural optimization tool. Therefore, the shape of the structure is modified to fulfil the standard component requirements. For shape optimization, non-linear finite element and fatigue analysis can be included more conveniently. This means that both EPS and fatigue life can be considered.

A second design realization needs to be performed by the Design engineer to interpret the shape optimized design with respect to manufacturability. Although, minor shape changes are done, these needs to be reviewed. This step is also discussed more in detail in section 4.4.1 on page 17 in the context of a RLCA application.

Finally, the standard component requirements are evaluated. This also requires that a new FE model needs to be constructed based on the shape design realization geometry. If the component fulfils the standard requirements, the component is considered to be ready for delivery to the manufacturer as a build to print project. Else, the procedure is continued from the shape optimization step until the component design fulfils the standard component requirements.

As mentioned in section 3.1, the manufacturing process is simulated for the considered component design. In the case of a cast component, the material solidification of molten material is simulated and gates are placed on the component design. The placement of gates is of importance to control the influence on the structural performance. Furthermore, the quality of the micro-structure and the avoidance of cavities are of high importance with respect to the end performance. A component optimization strategy including cast simulation is proposed in [10]. However, this step is beyond the scope of this thesis.

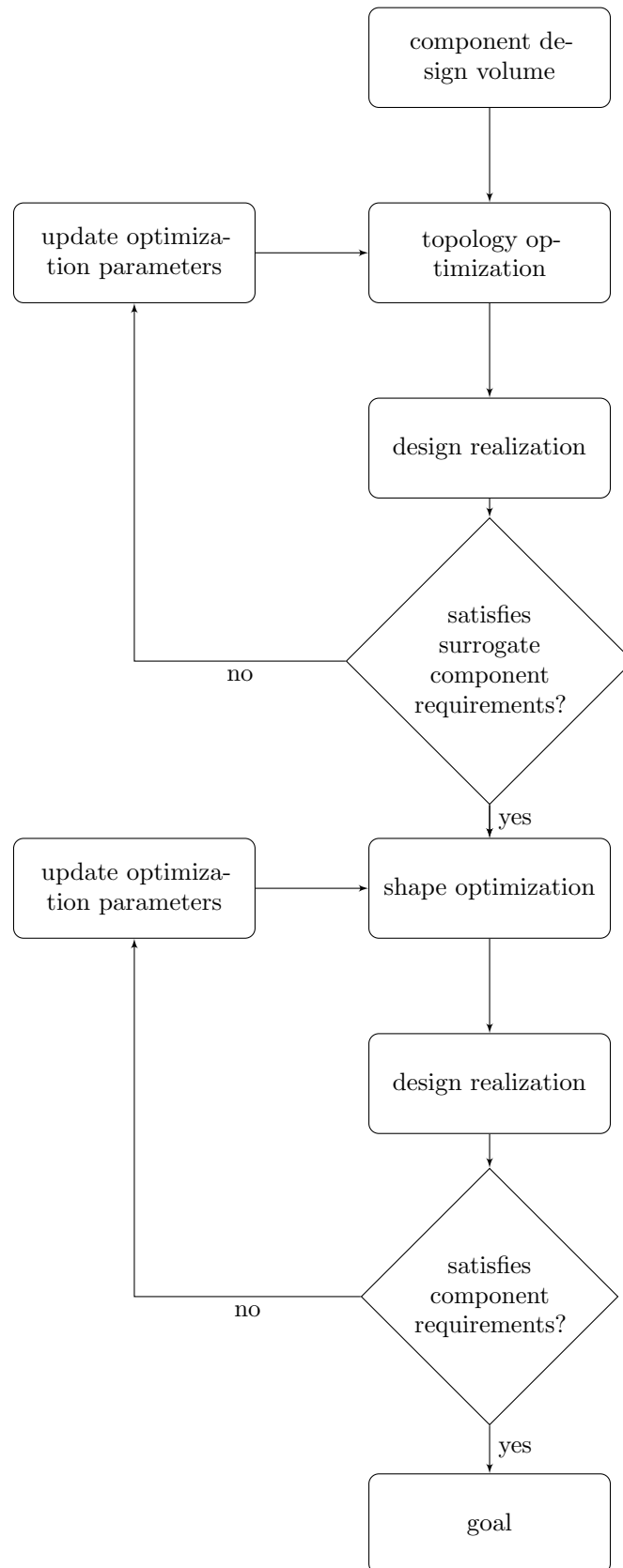


Figure 3.2: *Flowchart of the component development process using structural optimization tools.*

4 Application to a Rear Lower Control Arm

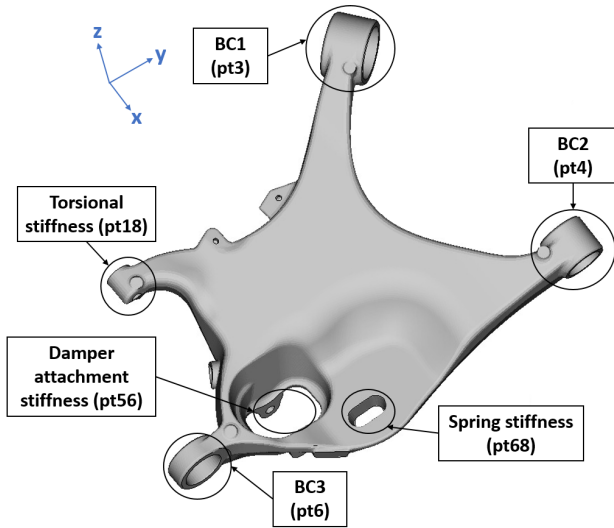
This chapter explains the component development process of a RLCA using structural optimization tools. It starts with a presentation of the current RLCA and its structural performance, which is followed by the application of the RLCA to the component development process using structural optimization tools described in section 3.2. Thereafter, a parameter study is presented to aid the selection of trial cases. Furthermore, both the topology and shape design realization steps are explained in a RLCA context. Finally, the process is demonstrated by running trial cases through the component development process using structural optimization tools.

4.1 The current RLCA component

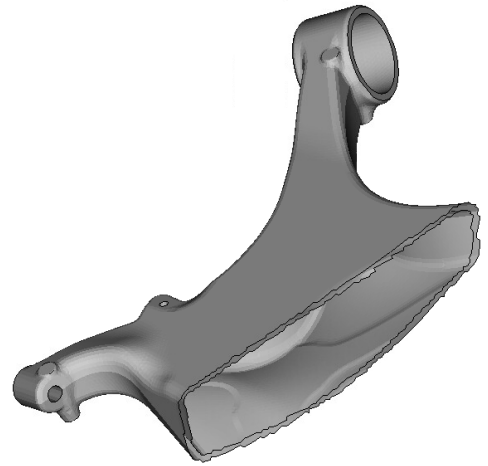
The design of the current RLCA is presented in Figure 4.1, it is a hollow structure which is made out of cast aluminium. The component is bolted to other chassis components named: Subframe, Knuckle, Damper attachment and Leaf spring. The RLCA has a central functionality in the complete chassis system, this brings that the RLCA is subjected to forces and moments via the adjacent components.

4.1.1 FE model

The finite element grid is constructed out of 2nd order tetra solid elements using a target element size of 3 mm. Loads and boundary conditions are subjected via so-called hardpoints. These are the intersection points to the adjacent components and modelled using distributing coupling constraints, the locations are presented in Figure 4.1. The Damper attachment located at pt56 in Figure 4.1 is bolted, it is modelled using beam elements and so-called connector elements. This bolt is named Clevis bracket. Furthermore, the current component weight which is based on the FE model is 4.07 kg. The first boundary condition, BC1 fixates translation in both y and z direction. The second boundary condition, BC2 fixates translation in x and z direction. For the third boundary condition BC3, z direction is fixed.



(a) The current RLCA.



(b) A cross section cut of the current RLCA.

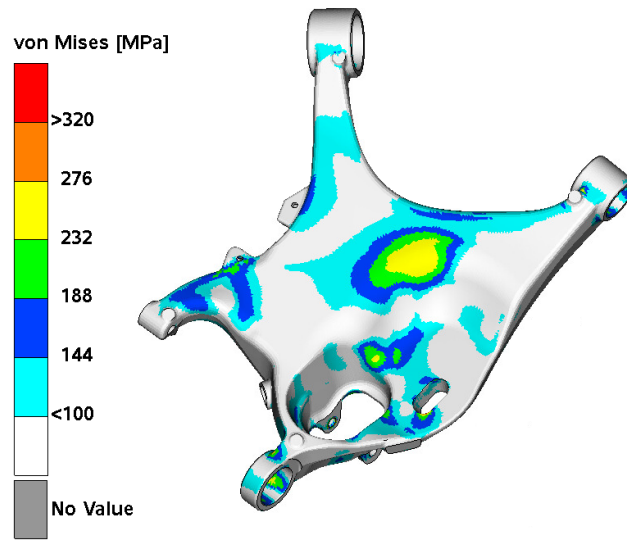
Figure 4.1: A finite element model representing the current RLCA component. Hardpoints are highlighted with given names. The hollow structure can be seen in the cross sectional view in (b).

4.1.2 FE analysis

The stiffness of the current component and requirements are presented in Table 4.1 where it is seen that all requirements are fulfilled. The effective stress is presented in Figure 4.2 for the worst load case. However, note that the effective stress is not included in the standard component requirements. This structural performance is kept in mind during the development of trial cases.

Table 4.1: Stiffness measured at Torsional stiffness (pt18), Damper attachment stiffness (pt68) and Spring stiffness (pt56) together with requirements. The last row shows the discrepancy between the Stiffness requirements and the current component.

Stiffness	pt18	pt68	pt56
Discrepancy [-]	+5.00 %	+6.00 %	+20.00 %



(a) Equivalent plastic strain for the worst considered load case.

Figure 4.2: The worst load case with respect to effective stress after the respective topology design realization.

4.2 CAE requirements

The CAE component requirements can be divided into three categories.

Stiffness

Firstly, the stiffness requirements. A unit force is subjected in z-direction for the Torsional stiffness (pt18), Spring attachment stiffness (pt68) and the Damper attachment stiffness (pt56). These locations are described in Figure 4.1. Thereafter, the displacement at these hardpoints are evaluated with respect to the requirements.

Strength events

The second requirement involves equivalent plastic strain (EPS) and permanent deformation for strength events. Strength events refers to so-called missuses of the car which gives rise to large forces and moments. An example could be when a car drives over a curb, this yields large forces and moments. There are four considered strength events in the optimization process, these are listed below. These are the strength events which gives rise to the largest forces and moments to the current RLCA. Loosely speaking, the DOC and ROC gives rise to large vertical loading whereas SAC and BIP gives rise to large lateral loading.

- Drive over curb (DOC)
- Rearwards driving over curb (ROC)
- Skid against curb (SAC)
- Brake in pothole (BIP)

The strength events are simulated by MBS representing a complete car subjected to these strength events. Road load data (RLD) generated by MBS gives forces and moments on a component level. The time steps for which the largest forces and moments occurs are analysed further using a linear static FE model. Note that several load cases could be generated from one strength event. Effective stress are thereafter evaluated according to von Mises.

$$\sigma^{vM} = \sqrt{\frac{3}{2} \sigma_{\text{dev}} : \sigma_{\text{dev}}} \quad (4.1)$$

where σ_{dev} is the deviatoric part of the stress tensor according to

$$\sigma_{\text{dev}} = \sigma - \sigma_{\text{vol}} \quad (4.2)$$

where σ_{vol} is the volumetric part of the stiffness tensor. Again, the load cases giving rise to high effective stress are analysed using elasto-plastic material behaviour, pre-tension and large deformation theory. The EPS is defined according to von Mises as described in [1].

$$\epsilon_e^p = \sqrt{\frac{2}{3} \epsilon^p : \epsilon^p} \quad (4.3)$$

where ϵ^p is the plastic strain contribution to the strain tensor $\epsilon = \epsilon^e + \epsilon^p$, ϵ^e is the elastic strain contribution. Note that the elastic and plastic strain contributions are decomposed by addition for large deformation theory in Abaqus. Furthermore, the permanent deformation at all hardpoints should be below a certain limit after one loading/unloading cycle separately.

Fatigue

The last requirement involves fatigue. Load history are given by MBS that simulates a physical chassis rig test. The method to calculate damage from stress cycles is the standard EN method, which uses the strain based Coffin-Manson-Basquin formulation. Firstly, the strain tensor history is assembled by scaling and superposition of FE results. Unit static load cases are subjected separately to the structure to generate strain responses using linear elastic material. These strain responses are subsequently scaled with the load history and superimposed. Secondly, an equivalent strain is calculated by the Absolute Maximum Principle value, where the largest magnitude principle strains for each time step are stored. Thereafter, a Rainflow count is performed and local plasticity is estimated using Neuber's rule, the position in each hysteresis loop is tracked simultaneously. Neuber's rule estimates notch stresses and strains which gives a rough estimate of localized plasticity. Finally, accumulated damage D are calculated using Palmgren-Miner's rule. These concepts are discussed more in detailed in [8]. Furthermore, fatigue life is referred to as the inverse of damage. Peak loads from the chassis rig load history can be evaluated by linear static FE analysis, it is called a Robustness check in this thesis. EPS or permanent deformation in hardpoints are not allowed with respect to the Robustness check.

4.3 Parameter study

A brief parameter study is conducted to aid the selection of optimization parameters to the trial cases. The aim of the parameter study is therefore to get a feeling of how the topology optimization works in an RLCA application when varying some of its parameters.

4.3.1 Optimization algorithm

As mentioned in section 2.2.6, two solution methods are available in TOSCA for topology optimization. The controller based algorithm is faster than the sensitivity based algorithm since no gradient information is required. The difference in computational time is significant, although it depends on max iterations, convergence criteria and number of design degree of freedom DOFs. Loosely speaking, 60-70 % reduction of computational time can be expected compared to the sensitivity based algorithm. However, the controller based algorithm is only applicable for the special case of minimize compliance under a volume constraint.

The sensitivity based MMA is more general and sophisticated algorithm compared to the controller based algorithm due to it's general treatment of objectives and constraints. Different design responses can also be combined, for example if the objective is to minimize weight, both the volume and effective stress can be incorporated as constraints.

In Figure A.4 in section A.3 the densities of the resulting structure of two optimization algorithms are presented. The optimization task is to minimize compliance under a volume constraint using the design volume presented in Figure 4.3. The same type of loading is subjected to both optimization tasks. It can be seen that the controller based method produces solid elements whereas the sensitivity based method produces intermediate density elements.

4.3.2 Member size

A possibility to mitigate checkerboard patterns described in section 2.2.2, or too thin rib structure is to use a so-called minimum member size constraint. Furthermore, the maximum member size constraint is available for the sensitivity based algorithm, it was investigated to produce a thin walled structure. Unfortunately, ribs were placed in between of the hollow part which can't be manufactured by sand casting. This problem is presented in Figure A.1, Appendix A where it can be seen that many ribs are thin, which makes the geometry difficult to cast.

4.3.3 Demold constraint

When searching for a suitable demold constraint, the optimization task is to minimize weight while fulfilling the stiffness constraints using the design volume presented in Figure 4.3. The first demold constraint investigated is the auto midplane constraint. The end objective weight was 4.4 kg, this over achieves the weight target. The design of this topology is presented in Appendix A Figure A.2.

The demold constraint is changed to auto midplane using the same optimization task as described above, the design can be seen in Figure A.3 in Appendix A. The objective weight is then 7.67 kg.

4.3.4 Load cases

The purpose of this study is to find a volume constraint for a optimization task where compliance is minimized under a volume constraint. Load cases are varied for the specific optimization task where weight is minimized under maximum displacements and compliance constraints. The stiffness requirements are used to define the displacement constraints. Furthermore, the design volume presented in Figure 4.6 is used. Compliance constraints are defined via the equivalent strain energy (compliance) of the associated load case subjected to the current FE model separately by a linear static analysis. The objective weight when varying some of the load cases presented in Table 4.2.

Table 4.2: The objective weight of topology optimization when subjected to the loading described in the first column.

Load case	Weight [kg]
All stiffness	1.98
All stiffness+DOC	1.99
All stiffness+DOC+BIP	2.47
All stiffness+DOC+BIP+ROC	2.62
All stiffness+DOC+BIP+ROC+SAC	2.67

4.4 Methodology application

To develop a methodology for component development with respect to a RLCA, different trial cases which are based on specific topology optimization configurations are tested and analysed throughout the proposed component development process using structural optimization tools. These trial cases are based on the parameter study presented in section 4.3 and lessons learned from previous trial cases. The trial cases are thereafter subjected through the component development process using structural optimization described in chapter 3. Since the aim of this thesis is to establish a suitable optimization process starting from an initial design volume to the end result, trial cases which shows non promising tendency is aborted due to the time frame of this thesis. The design realization of the topology optimized design involves rather extensive work by the Design engineer, therefore the amount of trial cases during this thesis are limited. The starting approach is to keep restrictions such as filters and manufacturing constraints of the topology optimization as loose as possible to be open minded to the prospective manufacturing method. Furthermore, to actually achieve a change of design for the RLCA component using the current procedure of production, besides from fulfilling the structural component requirements, a reduction of weight which is approximately greater or equal to 10-15 % of the current component weight is required. It corresponds to a weight of approximately 3.6 kg. This criteria is kept in mind when searching for a methodology including structural optimization tools. Additionally, it is believed that the placement of gates due to the sand cast method could add approximately 200 grams.

4.4.1 Design realization

In this section, the design realization procedure is explained from a Design engineer perspective.

Topology design realization

The design realization performed based on the topology optimization design includes several steps. The prospected manufacturing method is chosen based on the topology optimization design. In this thesis, sand casting is proposed for all trial cases. The following design realization steps are therefore explained with respect to sand casting. Firstly the mould tool direction is set. Thereafter, a split line which separates the mould into two parts is set. Any holes coming from topology optimization design are considered when defining the split line which separates the two mould parts. Furthermore draft angles are defined, these are usually provided by the manufacturer. A sand core is constructed to define the component geometry. Here, the Design engineer strives to resemble the topology optimized geometry as much as possible. A constant cross-sectional thickness is defined for most of the component regions. Finally machining is considered with respect to the adjacent component surfaces. At this stage the topology design realization is done. However, the Design engineer can only give a conceptual design proposal since the cast material solidification needs to be verified further.

Shape design realization

Looking into design realization based on shape optimization, the main challenge here is to follow the proposed shape without making too sharp or large variations of the component thickness. Furthermore, draft angles are considered. Note that if the design is constructed with the minimum cross sectional thickness due to the cast process, the shape is not allowed to shrink.

4.5 Trial case 1

The purpose of the first trial case is to investigate if it is sufficient to only use the three stiffness requirements and load cases in the topology optimization to fulfil all the standard component requirements in the end of the process. This approach has been successful for another chassis component at the Endurance Attribute and Chassis CAE group.

4.5.1 Design volume

The initial design volume is presented in Figure 4.3, it can be seen that the kinematics of other system components has been taken into consideration. The material used is aluminium. The targeted element size when constructing the mesh is 6 mm using 1st order tetra elements.

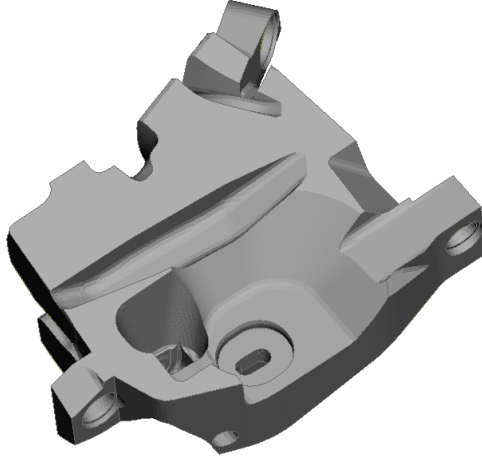


Figure 4.3: *The initial design volume of the RLCA component. Tracks in the structure are generated by a kinematic analysis of adjacent system components.*

4.5.2 Topology optimization

The topology optimization set-up is presented in Table 4.3. The optimization task is to minimize weight of the design volume under maximum displacement constraints based on the stiffness requirements. A sensitivity filter radius of two times the mean element size is used. The finite elements are partitioned into both design and non-design element sets. The non-design elements are located adjacent to boundary conditions, these elements are not modified during the topology optimization procedure. Furthermore, all stiffness load case and displacement constraints are subjected in z-direction.

Table 4.3: Configuration of the optimization task for Trial case 1.

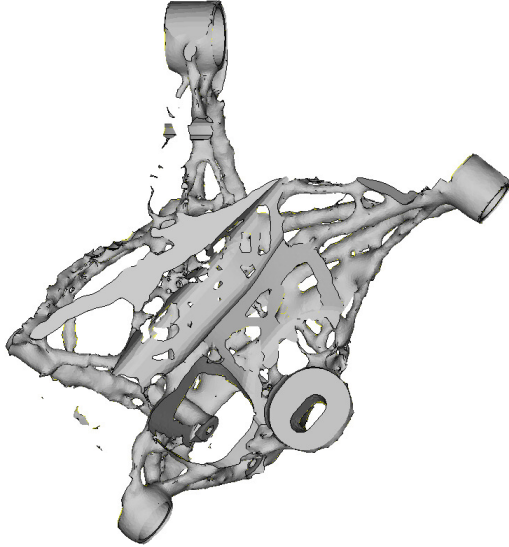
Optimization algorithm	Sensitivity based MMA
Objective function	Minimize weight
Constraint	$u_z \leq \bar{u}_{z,pt18}$ at <i>pt18</i> $u_z \leq \bar{u}_{z,pt68}$ at <i>pt68</i> $u_z \leq \bar{u}_{z,pt56}$ at <i>pt56</i>
Load case	Torsional stiffness Damper attachment stiffness Spring stiffness
Filter radius	2x mean element size
Density update scheme	normal

The objective weight is minimized to 1.98 kg. The finite elements generated by the topology optimization are thereafter modified using TOSCA.SMOOTH to produce smooth surfaces. An isosurface is created at elements

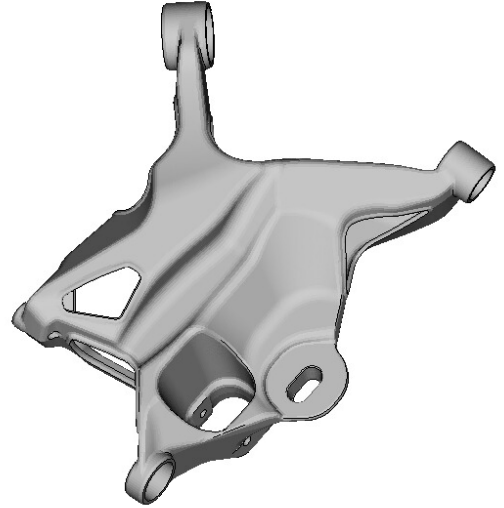
with intermediate densities greater or equal to 0.3, from where the new smoothed geometry is generated. The isosurface value is determined by testing different parameter values and observing the most continuous structure design. The new geometry created from the isosurface is presented in Figure 4.4a, it can be seen that the geometry looks rather edgy including many small holes and loose elements. The loose elements is a consequence of the smooth procedure.

4.5.3 Design realization

The geometry is realized by a Design engineer, where manufacturability is considered further. It can be seen in Figure 4.4b that extensive design changes are made during the design realization step when comparing with the topology optimized design presented in Figure 4.4a. This design realization step is necessary to achieve a feasible component design with respect to the considered manufacturing method. However, the Design engineer considers the realized design in Figure 4.4b at this stage to be conceptually feasible due to sand casting. Additionally, this design realization was reviewed by a person at Volvo Cars with expertise in casting. To achieve the design presented in Figure 4.4b, modifications had to be performed manually after the first design realization. A significantly large effective stress hotspot was manually identified after the first linear standard analysis. This hotspot caused divergence for the non-linear FE simulation due to too large EPS. Subsequently, material was added to reinforce this region. The weight of the final topology design realization is 3.66 kg.



(a) Prior to design realization



(b) Final design realization

Figure 4.4: The design before and after the topology design realization step. The design in (a) is interpreted by the Design engineer to fulfil manufacturing constraints to the new design (b). Note that the general outline in (a) is kept while small holes are filled with material. Loose structure elements observed in (a) comes from the smooth procedure, these are disregarded in the design realization step.

4.5.4 FE analysis

The directional stiffness response for the three load cases of the design realized component is presented in Table 4.4. Note that the discrepancy of the stiffness between Trial Case 1 and the requirement are rather high.

Table 4.4: Stiffness measured at Torsional stiffness (pt18), Damper attachment stiffness (pt68) and Spring stiffness (pt56) together with the requirements. The last row shows the discrepancy between the Stiffness requirement and Trial case 1 after the topology design realization step.

Stiffness	pt18	pt68	pt56
Discrepancy [-]	+46.33 %	+50.64 %	+62.60 %

By looking at Figure 4.5a, it can be seen that the normalized EPS requirement of 1 is not fulfilled. So-called hotspots which are local regions with high accumulation of EPS or fatigue life occurs mainly at regions close to sharp geometry changes. The fatigue life is presented in Figure 4.5b when the component is subjected to one chassis rig cycle. It can be seen that many regions on the component surface are below this normalized criteria of 1.

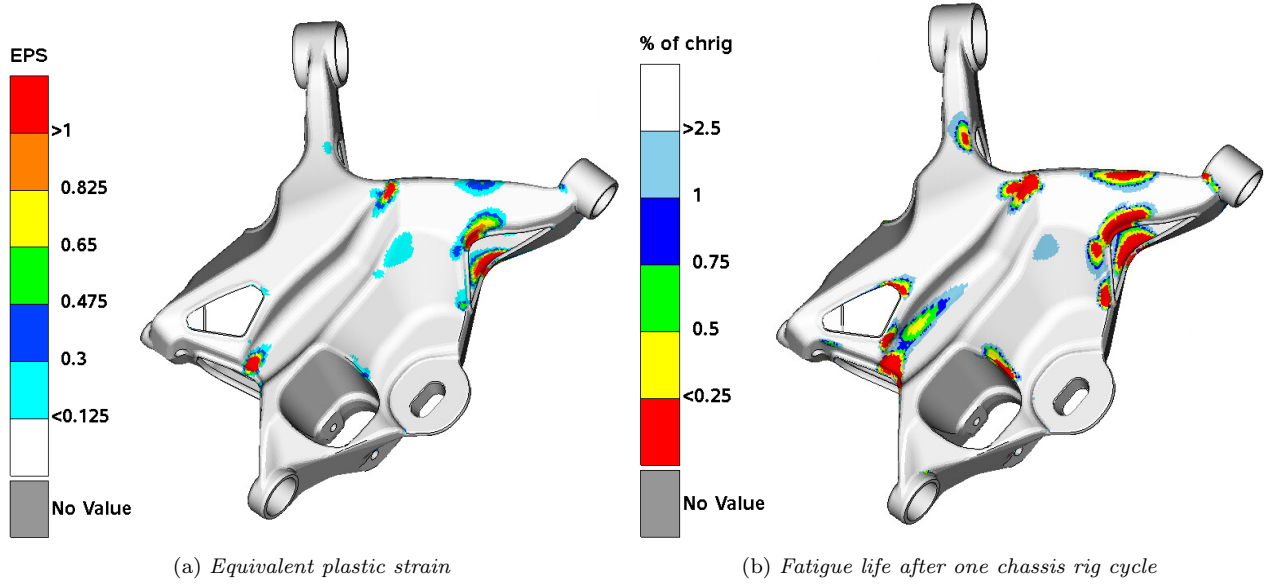


Figure 4.5: Regions where the EPS exceeded the requirement threshold are marked with red color. The fatigue life after one chassis rig cycle shows many regions where the fatigue requirement of 1 is not fulfilled.

4.5.5 Discussion

The load cases subjected to the component design volume yields both twisting and bending moment to the component structure. It seems therefore efficient in a weight saving sense to have ribs placed as a shell of a hollow structure while fulfilling the stiffness requirements. Sand casting which is the current manufacturing method for the current RLCA is suggested by the Design engineer. Since the design realization requires an additional reinforcement of material due to too large EPS, this indicates that this problem needs to be taken care of in the topology optimization configuration if so possible. Furthermore, the optimal structure wants to expand to the geometrical boundaries of the initial design volume. This brings that some of the geometrical features from the design volume geometry are transferred to the optimized design. Some of the sharp edges coming from the design volume shape are therefore kept, these features could give rise to stress concentration regions. One reason for this is that stress is not considered during the optimization whilst it is optimal due to minimal weight to put material close to the outer design volume boundaries. Lastly, Trial case 1 is aborted due to too heavy weight (3.66 kg) in comparison with the current component weight (4.07 kg). Furthermore, the stiffness over achieves the requirement rather much at the topology design realization step.

4.5.6 Conclusions

- The stiffness requirements used as a constraints for the topology optimization task is too high set when looking at the component stiffness after the design realization.
- If additional load cases and constraints are introduced to the optimization task used for this trial case, this will most likely increase the weight further.
- The topology optimization design is extensively modified during the design realization step. This means that too small variation of topology optimization parameters and requirements might not affect the outcome of the topology design realization.
- One possibility to lower the weight while including more load cases is to modify the design volume geometry.
- Another possibility to lower the weight is to change the optimization task in order to achieve a more clear topological design which enables more or larger holes to the component structure during the topology design realization.
- The weight is 3.66 kg at the topology design realization step.

4.6 Trial case 2

To reduce the amount of material added during the design realization step and increase lateral stiffness of the structure, the second trial case uses a smaller design volume compared to the design volume used for Trial case 1. Furthermore, BIP and SAC are included to increase lateral stiffness compared to Trial case 1. It should also be mentioned that the other possibility proposed in section 4.5.6, to reduce weight by changing the optimization task and keeping the initial design volume was considered but not chosen to proceed with. This is due to the rather much amount of material that is necessary to add due to the sand casting method.

4.6.1 Design volume

The design volume used for Trial case 2 is presented in Figure 4.6, where the height is lowered and sharp edges are smoothed out.

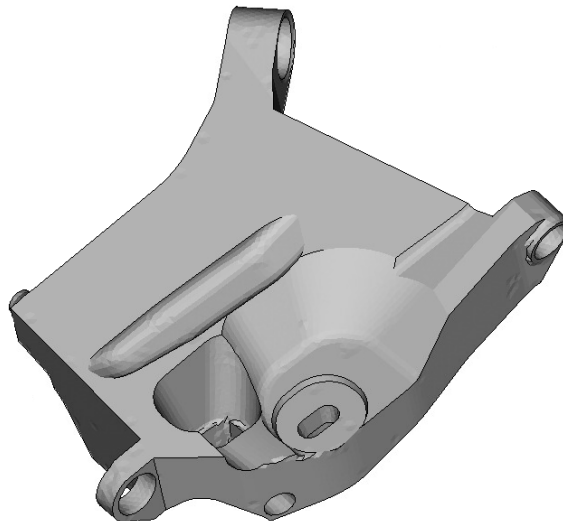


Figure 4.6: *The second design volume of the RLCA component. The height is reduced and sharp edges are smoothed out from the initial design volume.*

4.6.2 Topology optimization

The topology optimization configuration is presented in Table 4.5. The optimization algorithm is the sensitivity based MMA, same as used for Trial case 1. The optimization task is now to minimize compliance for all load cases described in Table 4.5 under a volume constraint. This optimization objective gives often more smooth transitions of intermediate densities compared to when minimizing weight. The density update scheme is changed to conservative since this formulation is more stable. The volume constraint is based on the parameter study section 4.3.4, a weight between two and three kg seems reasonable by looking at Table 4.2. This, since the weight of an optimal structure fulfilling the stiffness and compliance constraints when subjected to the stiffness and strength event load cases is 2.67 kg. The strength events BIP and SAC are included to increase the lateral stiffness compared to Trial case 1. Note that forces and moments are applied in all hardpoints for each strength event respectively, this yields a more complex type of loading to the structure compared to the stiffness load cases. The structure is thereafter smoothed with an isosurface value of 0.3, before it is sent to the topology design realization step. This isosurface parameter value is estimated by variation of the parameter and then observing a topology design with continuous structure.

Table 4.5: The configuration of the optimization task for Trial case 2. The volume fraction constraint correspond to a weight of 2.53 kg.

Optimization algorithm	Sensitivity based MMA
Objective function	Minimize compliance
Constraint	$V_f \leq 0.09$
Load case	Torsional stiffness Damper attachment stiffness Spring stiffness BIP SAC
Filter radius	2x mean element size
Density update scheme	conservative

4.6.3 Design realization

The topology optimized design presented in Figure 4.7a is interpreted by the Design engineer to the design presented in Figure 4.7b. The minimum thickness with respect to the cast method is used for most of the geometry during the design realization. The weight of the topology design realized is 2.84 kg.

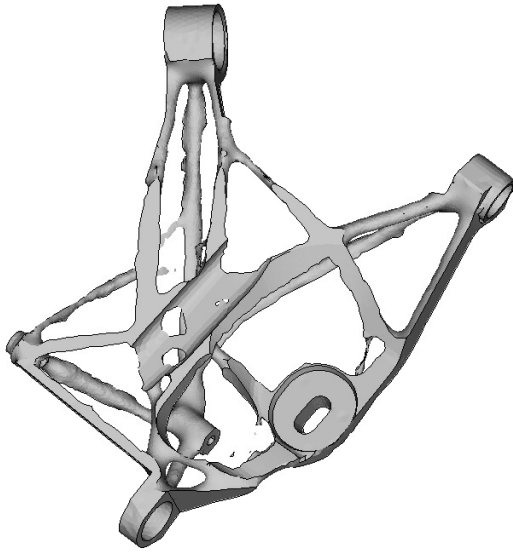
4.6.4 FE analysis

The directional stiffness is presented in Table 4.6. It can be seen that the stiffness are under achieved compared to the stiffness requirement.

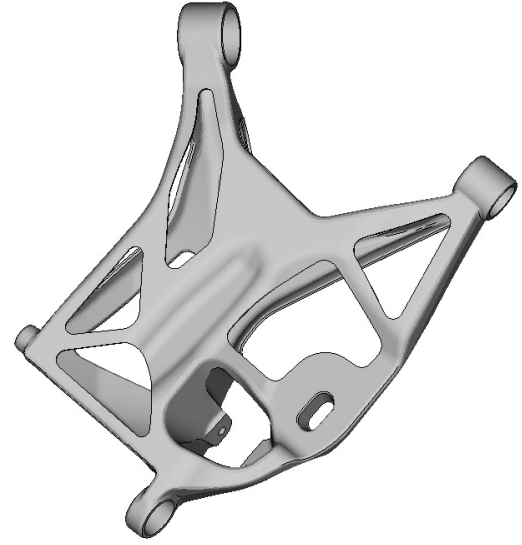
Table 4.6: Stiffness measured at Torsional stiffness (pt18), Damper attachment stiffness (pt68) and Spring stiffness (pt56) together with requirements. The last row shows the discrepancy between the stiffness requirement and Trial case 2 after the topology design realization. The different hardpoints where stiffness are measured can be seen in Figure 4.1 on page 13.

Stiffness	pt18	pt68	pt56
Discrepancy [-]	-36.67 %	-46.40 %	-36.08 %

The linear FE analysis of all considered strength events shows significantly high effective stress compared to the effective stress of the current component presented in Figure 4.2. Recall that effective stress is not included in the optimization task, however too high effective stress peaks indicates that this problem needs to be addressed before focusing on EPS and fatigue life. The most severe strength events are DOC and ROC with respect to effective stress. The DOC is presented in Figure 4.8a on page 24, where high effective stress regions can be spotted.



(a) Before design realization



(b) Design realization

Figure 4.7: The design before and after the design realization step. The design in (a) is interpreted by the Design engineer to fulfil manufacturing constraints to the realized design (b). Loose structure elements observed in (a) comes from the smooth procedure, these are disregarded by the design realization step.

4.6.5 Shape optimization

The approach for shape optimization of Trial case 2 is to focus on high effective stress firstly. Subsequently, EPS and fatigue life can be considered. Furthermore, the effective stress is evaluated by a linear FE model which is less demanding due to computational cost compared to an optimization involving non-linear or fatigue analysis. The shape optimization configuration is presented in Table 4.7. The effective stress of the worst load case DOC, is minimized with respect to the default effective stress value determined by TOSCA. Recall that the default effective stress in TOSCA is equivalent to the average effective stress in all design nodes. A volume fraction constraint V_f is imposed to control the increase of material. The objective function is minimized from 1512 to 681 MPa, where these are peak values.

Table 4.7: Configuration of the shape optimization task for Trial case 2.

Optimization algorithm	Controller based OC
Objective function	Minmax $ \sigma^{vM} - \sigma_{ref} $
Constraint	Volume fraction $\leq 5\%$
Load case	DOC

4.6.6 Design realization

The shape optimized design is interpreted by the Design engineer to fulfil manufacturing constraints while keeping the design as similar as possible to the shape optimized design. The weight of the structure is 3.00 kg after the design realization step. The increase is 5.63% compared to the topology design realization.

4.6.7 FE analysis

The stiffness results are presented in Table 4.8, it can be seen that the stiffness of the realized design does not fulfil the requirement. However, a stiffness analysis was performed prior to the shape design realization step to get a feeling for how much the stiffness can be expected to increase. These results are presented on the two rows in the middle of Table 4.8.

The peak effective stresses are still rather high for DOC and ROC, although a shape optimization to

Table 4.8: Stiffness measured at Torsional stiffness (pt18), Damper attachment stiffness (pt68) and Spring stiffness (pt56) before and after design realization together with requirements. The discrepancy between the stiffness requirement, both pre and post shape design realization are shown. Note that the * indicates that this stiffness is measured prior to the design realization, this in order to get a feeling for how much the stiffness increases during the shape design realization.

Stiffness	pt18	pt68	pt56
Discrepancy* [-]	+19.00 %	-4.60 %	-6.56 %
Discrepancy [-]	-13.00 %	-36.40 %	-23.96 %

minimize effective stress was performed. The effective stress is visualized in Figure 4.8 when subjected to the strength event DOC. It can be seen that high effective stress regions in Figure 4.8a are minimized to Figure 4.8b during one shape optimization step.

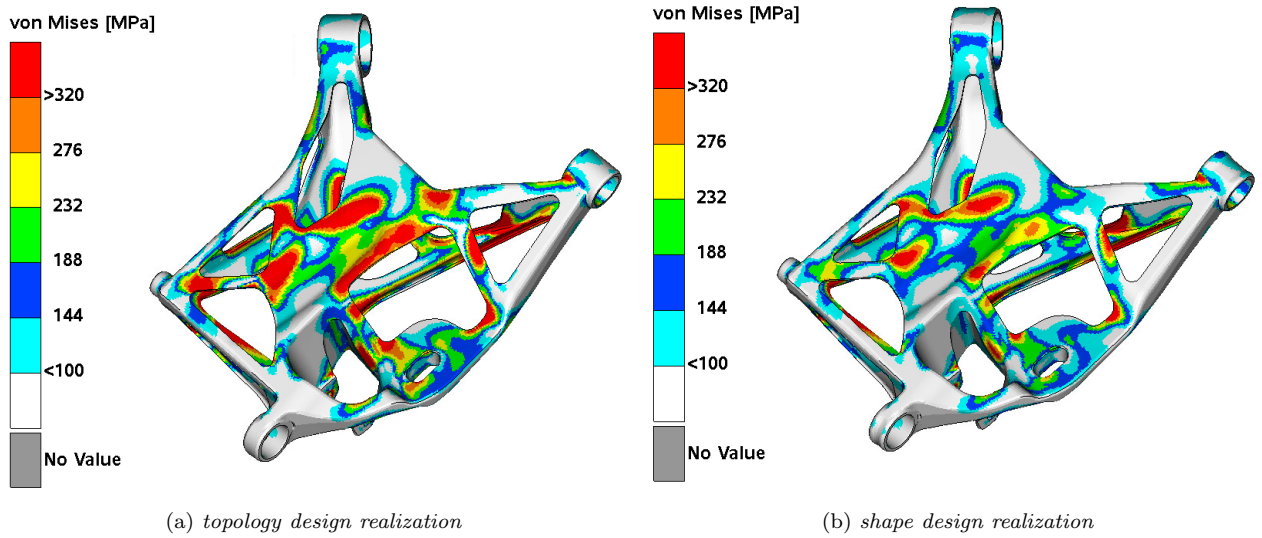


Figure 4.8: Effective stress for the worst strength event DOC.

4.6.8 Discussion

The topology optimization task where compliance is minimized under a volume constraint gives a more clear and distinct rib structure. This is easier to realize by the Design engineer compared to Trial case 1 when applying the sand cast method to the topology optimization design. The reduced design volume brings that not as much material needs to be filled out with respect to the cast process compared to Trial case 1. It can also be seen that the reduction of sharp features of the geometry gives less potential stress concentration regions. The linear FE analysis subsequent to the topology design realization shows that effective stress are rather high, especially for the strength events DOC and ROC. This indicates that it can be worth to incorporate these load cases to the topology optimization task. The shape optimization performed yields both lower effective stress and higher stiffness. Unfortunately, the effective stress of the design realization proposal is not minimized as much as the shape optimization objective shows due to the shape design realization. However, more than one linear shape optimization loop could be performed. Lastly, Trial case 2 is aborted due to too high effective stress levels and the time frame of this thesis.

4.6.9 Conclusions

- The choice of compliance as an objective contributes to a more clear rib structure design which is easier for the Design engineer to interpret.
- A shape optimization using linear FE analysis is used instead of a nonlinear FE analysis which is suggested

in the component development process. This seems necessary due to too high effective stress. Furthermore, linear FE analysis are less computational costly to evaluate compared to a nonlinear FE analysis.

- The stiffness of the component increases during the shape optimization.
- DOC is identified to give significantly high effective stress.
- The weight is 2.84 kg at the topology design realization step.

4.7 Trial case 3

The purpose of Trial case 3 is to incorporate the DOC and ROC strength events to the topology optimization task used for Trial case 2 to mitigate high effective stress caused by these load cases. Both the design volume and the topology optimization configuration are the same as used for Trial case 2, except from the additional DOC and ROC load cases.

4.7.1 Design realization

The topology optimized design is presented in Figure 4.9. An isosurface is created for the topology optimized design at the elements with intermediate densities greater or equal to 0.2. This isosurface value seems to give the most continuous structure by variation of isosurface parameters. The topology optimized part in Figure 4.9a is interpreted by the Design engineer to the design shown in Figure 4.9b. The weight of the topology optimization design is approximately 2.53 kg, after design realization it is 3,13 kg. Furthermore, the Design engineer uses the minimum available thickness with respect to the sand cast method for most of the component design regions when constructing the design proposal.

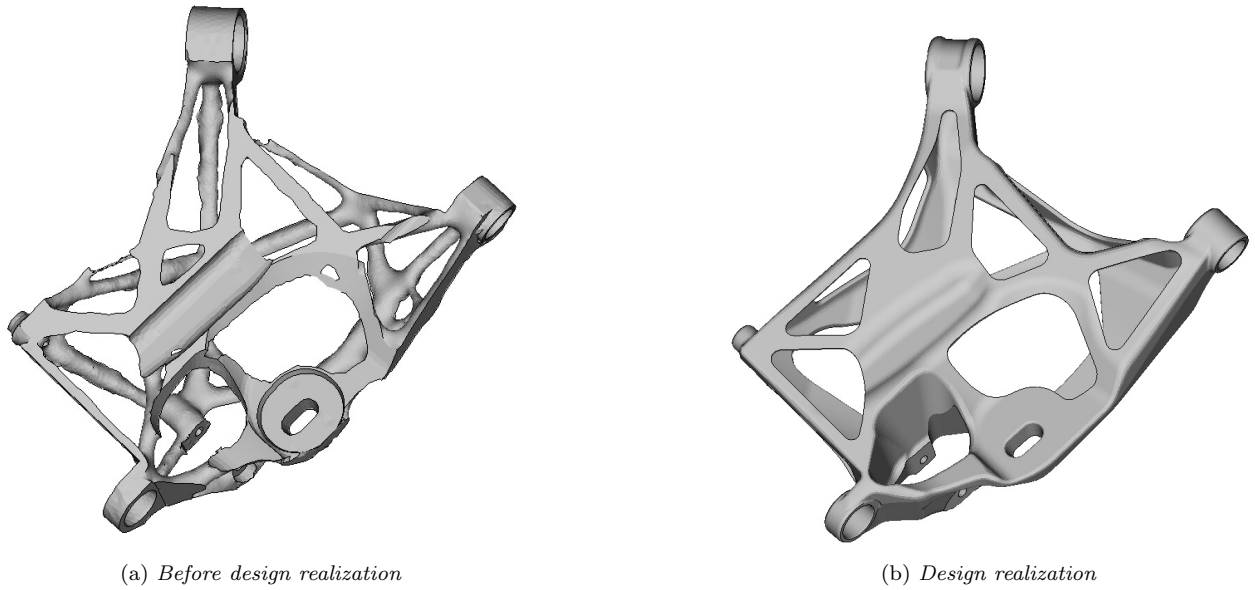


Figure 4.9: The design before and after the design realization step. The design in (a) is interpreted by the Design engineer to fulfil manufacturing constraints in the realized design (b).

4.7.2 FE analysis

The stiffness results are presented in Table 4.9. It can be seen that the Torsional stiffness (pt18) requirement is further away from fulfilling the requirement than the others.

The worst case strength event due to effective stress, DOC is presented in Figure 4.11a on page 28. It can be seen that the effective stress peaks are rather high.

Table 4.9: Stiffness measured at Torsional stiffness (pt18), Damper attachment stiffness (pt68) and Spring stiffness (pt56) before and after design realization together with requirements.

Stiffness	pt18	pt68	pt56
Discrepancy [-]	-26.67 %	+1.00 %	-1.08 %

4.7.3 Shape optimization

The approach is to minimize effective stress hotspots firstly, before focusing on EPS and fatigue life. The shape optimization is therefore divided into four shape optimization steps. One step involves a shape optimization, a design realization based on the shape optimized design and a FE analysis. The reason why this is done in four steps is due to the limited time frame of this thesis. Although as many steps required to fulfil the standard component requirements would be preferable. Furthermore, the order of different steps in the process is set based on each other. The shape optimization steps are presented in Figure 4.10.

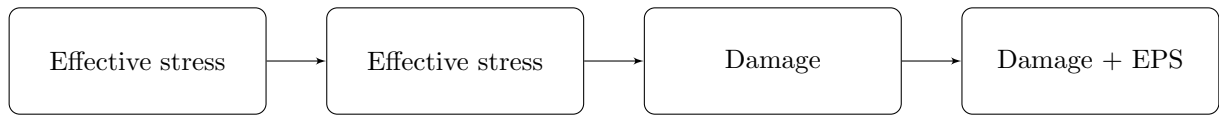


Figure 4.10: Flowchart of the shape optimization steps.

The configuration of the first shape optimization is presented in Table 4.10. The effective stress objectives of the most severe strength event load cases DOC and BIP are minimized from 941 and 631 to 579 and 397 MPa respectively. A volume fraction constraint of 4 % is imposed to control the increase of weight. This value can be varied to find a good balance between volume fraction and objective value. However, the influence of this parameter is rather small if a volume fraction above 4% is chosen. Furthermore a grow control condition is incorporated, it was mentioned in section 4.4.1 that the design is realized with the lowest applicable minimum material thickness, hence the design surface is not allowed to shrink.

Table 4.10: Configuration of the first shape optimization task for Trial case 3, u_d is the scalar displacement of a design node.

Optimization algorithm	Controller based OC
Objective function	Minmax $ \sigma^{vM} - \sigma_{ref} $
Constraint	Volume fraction $V_f \leq 4\%$
Manufacturing constraint	Grow control $u_d \in [0,5]\text{mm}$
Load case	DOC BIP

The difference between the first and second shape optimization is that the only DOC is considered and the volume fraction constraint is increased from 4 to 8 %. The objective is minimized from 839 to 559 MPa.

Table 4.11: Configuration of the second shape optimization task for Trial case 3.

Optimization algorithm	Controller based OC
Objective function	Minmax $ \sigma^{vM} - \sigma_{ref} $
Constraint	Volume fraction $V_f \leq 8\%$
Manufacturing constraint	Grow control $u_d \in [0,5]\text{mm}$
Load case	DOC

The third shape optimization step focuses on minimizing damage. The configuration is presented in Table 4.12. No specific volume constraint is specified whereas the grow control manufacturing constraint is used as done for the second shape optimization. The most critical fatigue life is increased from 2.56 to 4.16 %.

The fourth shape optimization step focuses on minimizing both damage and plastic strain. Both objectives are normalized to use the minmax formulation properly, since we minimize the maximum objective value, both objectives are normalized to avoid that one of the objectives becomes much higher than the other. Furthermore, the material added during shape optimization step one to three brings that the design surface is now allowed

Table 4.12: Configuration of the third shape optimization task for Trial case 3.

Optimization algorithm	Controller based OC
Objective function	Minmax $ D - D_{ref} $
Manufacturing constraint	Grow control $u_d \in [0,5]$ mm
Load case	One chassis rig cycle

to both shrink and grow as specified in the manufacturing condition in Table 4.13. The objectives EPS and damage are minimized, this yields EPS and fatigue life from 0.015 and 4 % and to 0.012 and 18.9 %. Note that these objective values are the extreme nodal values.

Table 4.13: Configuration of the fourth shape optimization task for Trial case 3.

Optimization algorithm	Controller based OC
Objective function	Minmax $w_1 \frac{ D-D_{ref} }{D_{val}} + w_2 \frac{ \epsilon_e - \epsilon_{e,ref} }{\epsilon_{e,val}}$
Manufacturing constraint	Grow control $u_d \in [-1,5]$ mm
Load case	One chassis rig cycle DOC

4.7.4 Design realization

The weight after the fourth shape design realization is 3.28 kg. The increase of material is 4.8 % compared to the topology design realization step.

4.7.5 FE analysis

The stiffness results of the fourth shape optimization design are presented in Table 4.14. It can be seen that the stiffness requirements are fulfilled at this stage.

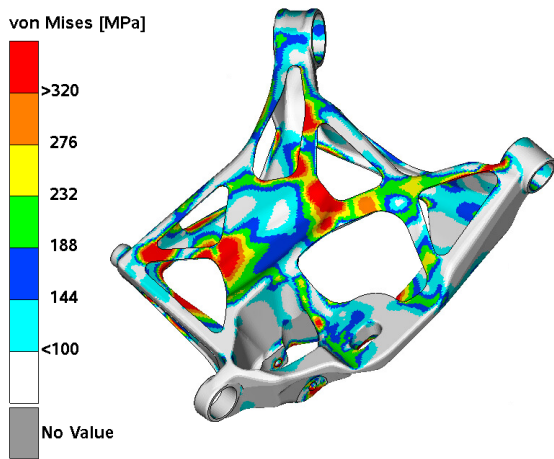
The effective stress for all design realization steps of Trial case 3 is presented in Figure 4.11. Looking at the effective stress of the topology design realization, it can be seen that high effective stress is spread out on large regions of the component. The next result shows the first shape design realization where the objective was to minimize the effective stress of the two worst load cases, it can be seen that rather large effective stress regions are reduced. By comparing the first and the second shape design realization, it is seen that the overall effective stress is reduced further. Note that it is the damage which is minimized when comparing the difference between the second and third shape realization, although this brings further reduction of effective stress. By comparing the third and fourth shape design realization where EPS and damage are minimized, it is seen that the reduction of effective stress is not as significant compared to the previous steps.

The fatigue life is presented in Figure 4.12b to 4.12d. Firstly, the fatigue life of the topology design realized part is presented, rather large regions where the requirement is not fulfilled can be observed. By comparing the topology and second shape design realization where the effective stress has been minimized in two shape optimization steps, it is seen that the fatigue life is significantly improved. Between the second and third design realization where the objective was to minimize damage, it can be seen that the fatigue life has improved further, the same goes for the third to fourth design realization. However, note that the standard component requirement are not fulfilled at this stage.

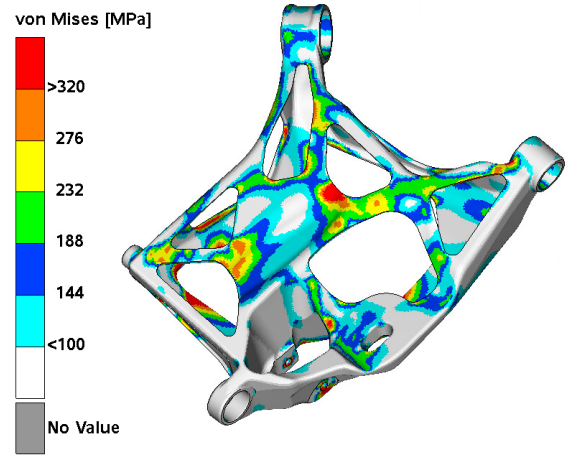
It should be mentioned that the nonlinear FE analysis for EPS after the topology design realization did not converge when subjected to the worst load case with respect to effective stress, therefore it is not showed. Looking at Figure 4.12e to 4.12f, it is noticeable that the EPS is reduced somewhat. Note that neither this requirement is fulfilled at this stage.

Table 4.14: Stiffness measured at Torsional stiffness (pt18), Damper attachment stiffness (pt68) and Spring stiffness (pt56) are compared to the stiffness requirements.

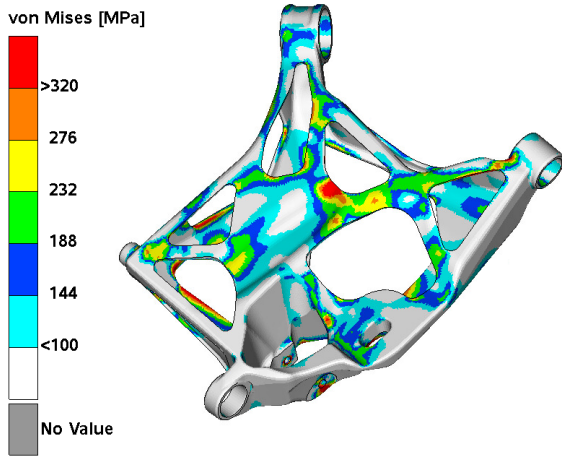
Stiffness	pt18	pt68	pt56
Discrepancy [-]	+11.34 %	+11.00 %	+12.40 %



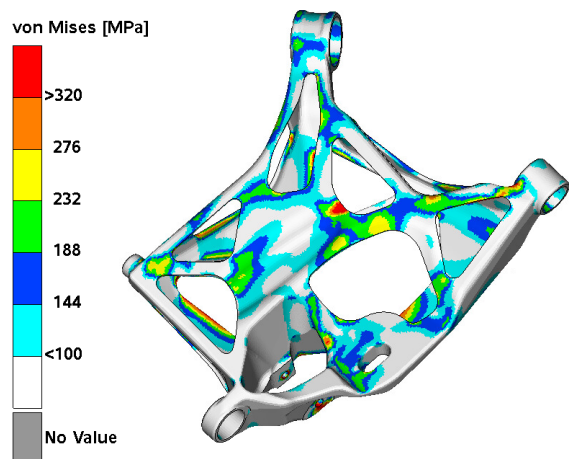
(a) *Topology design realization.*



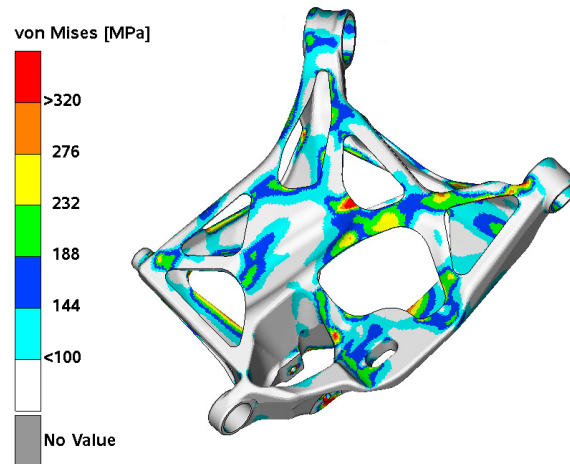
(b) *First shape design realization.*



(c) *Second shape design realization.*

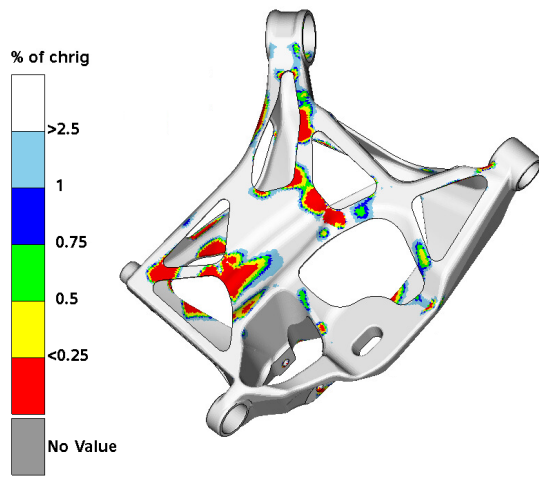


(d) *Third shape design realization.*

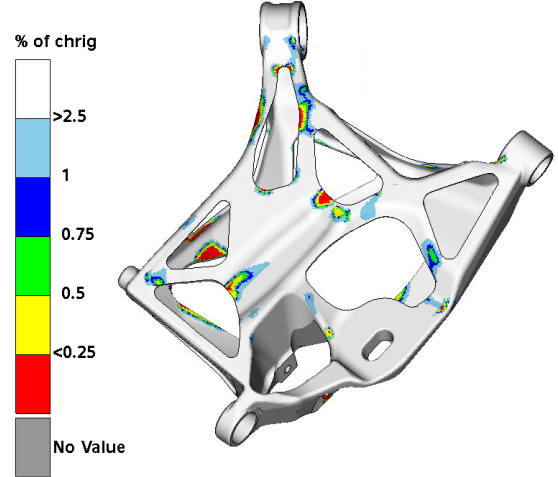


(e) *Fourth shape design realization.*

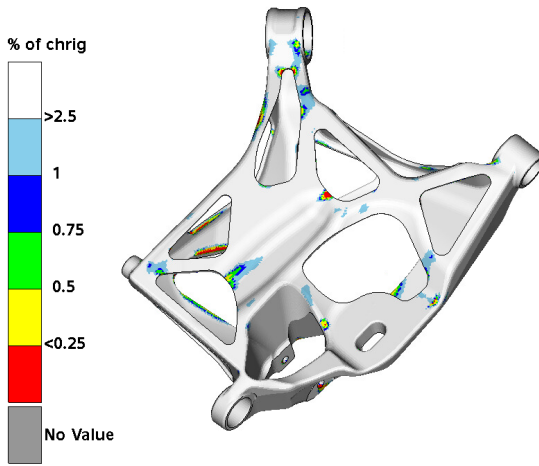
Figure 4.11: *The worst strength event with respect to effective stress after the respective shape design realization.*



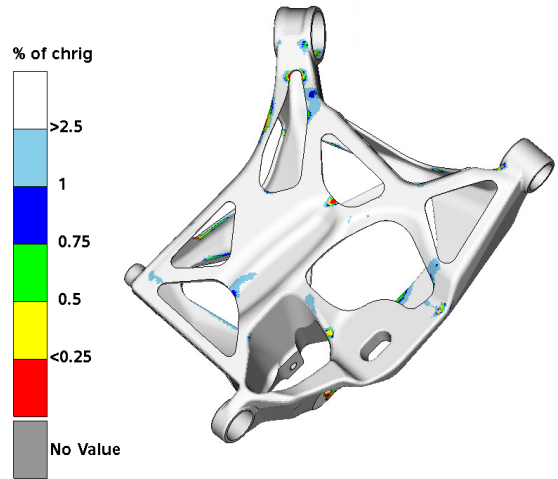
(a) Topology design realization.



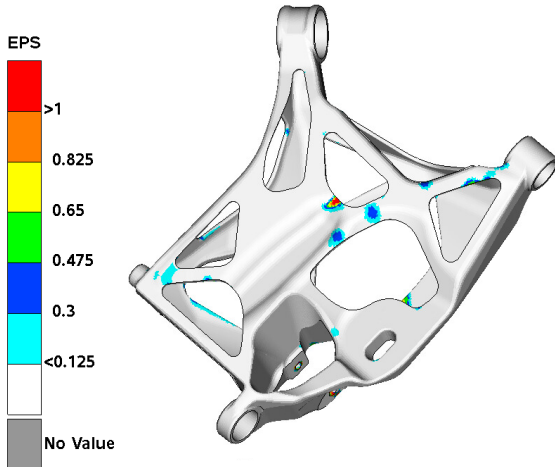
(b) Second shape design realization.



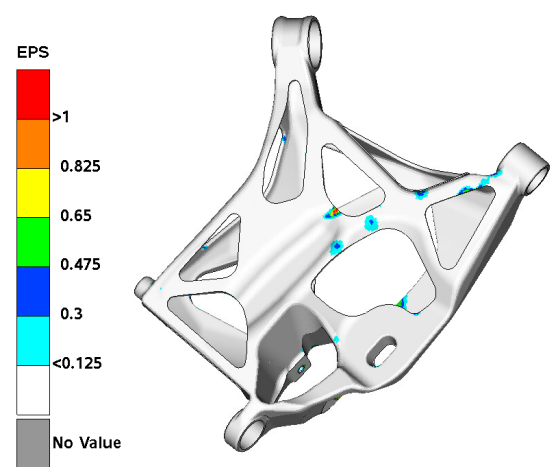
(c) Third shape design realization.



(d) Fourth shape design realization.



(e) Third shape design realization.



(f) Fourth shape design realization.

Figure 4.12: FE analysis of Trial case 3 with respect to fatigue life and EPS. Note that the two first rows shows fatigue life whereas the last row shows EPS.

4.7.6 Discussion

The addition of the DOC and ROC load cases to the topology optimization task gives a change of design compared to Trial case 2. However, the increase of weight during the topology design realization step is slightly higher compared to Trial case 2. Furthermore, the effective stress of the worst load case is lower compared to Trial case 2 after the topology design realization, that is the purpose of Trial case 3. Although the effective stress peak values are still rather high at this stage. Furthermore, the EPS analysis does not converge due to too high EPS after the topology design realization. The shape optimization part are divided into several steps, where one step contains of one shape optimization, design realization and a FE analysis. One reason for this is the FE mesh quality, in shape optimization nodes are compressed or elongated, this gives a limitation of nodal movement with respect to FE mesh. The increase of material during these steps are rather low, (4%) increase of material in total for these four shape optimization steps with respect to the topology design realization weight. Furthermore, the grow control manufacturing condition proved to work well. The FE analysis of the fourth shape design realization fulfils the stiffness requirement whilst the EPS and fatigue requirement are not fulfilled. However, the remaining hotspots are rather small. Since the increase of material during each shape optimization step is rather low, this enables that the shape optimization step strategy can be continued further. Although, it is has not been investigated how the process of shape optimization steps is supposed to be set up to fulfil the requirements with as low weight as possible.

4.7.7 Conclusions

- By adding DOC and ROC to the topology optimization task optimization task, this gave lower effective stress and somewhat more weight compared to Trial case 2 at the topology design realization step (+290 grams).
- The effective stress peaks for the worst load case are still too high in order to perform multi-objective shape optimization directly after the topology design realization step.
- The approach presented in Figure 4.10 seems rather efficient in the sense that all considered quantities are gradually reduced during the shape design realization steps whilst not adding too much weight. Although it is not said that this shape optimization step process is the most efficient one.
- The weight is 3.13 kg at the topology design realization step. The weight has increased 150 grams during the four shape optimization steps, starting from the topology design realization step.

4.8 Trial case 4

The purpose of Trial case 4 is to investigate if load cases from the robustness check can improve the fatigue life and also give a more robust design due to effective stress and EPS. Recall the robust check from section 4.2. This gives in total 25 static load subjected in the topology optimization task. The design volume and topology optimization configuration are the same as used for Trial case 3, except from the robustness check load cases.

4.8.1 Design realization

The topology optimized design in Figure 4.13 is smoothed using a isosurface value of 0.18. The design is thereafter interpreted by the Design engineer with respect to manufacturability. The weight is 3.15 kg after the topology design realization.

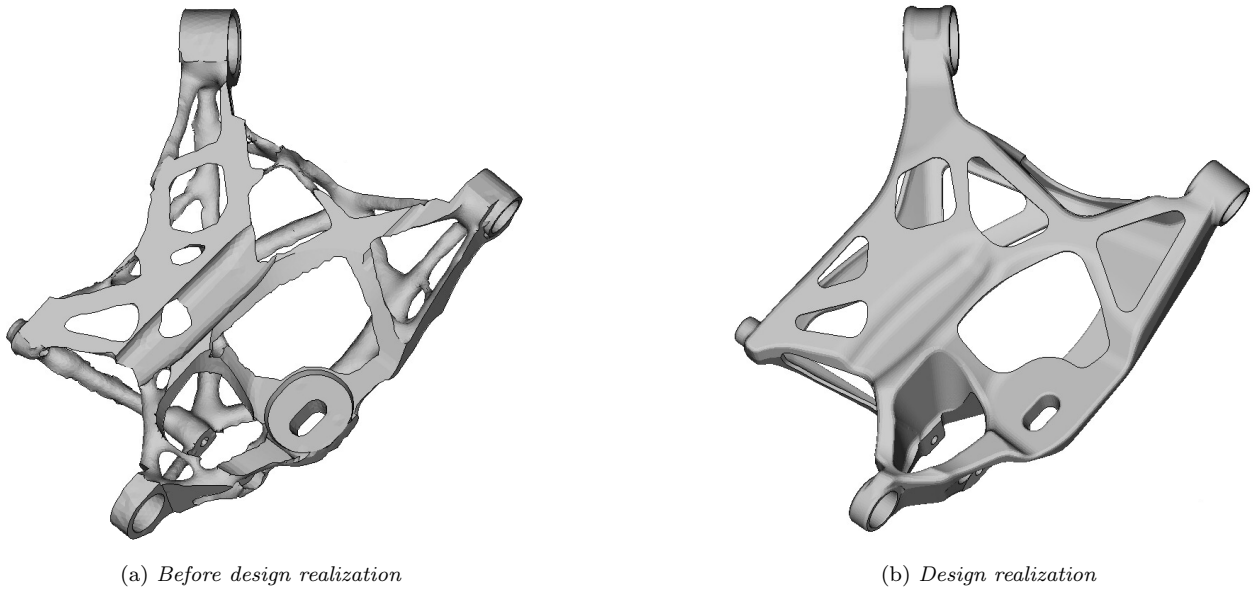


Figure 4.13: The design before and after the design realization step. The design in (a) is interpreted by the Design engineer to fulfil manufacturing constraints to the realized design (b).

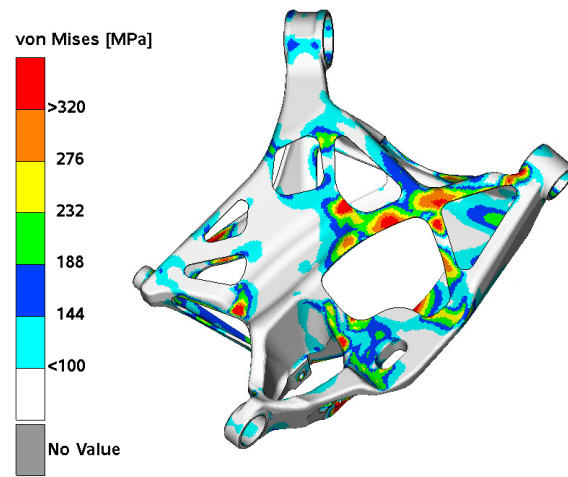
4.8.2 FE analysis

The stiffness results is presented in Table 4.15, it can be seen that all stiffness requirements are fulfilled.

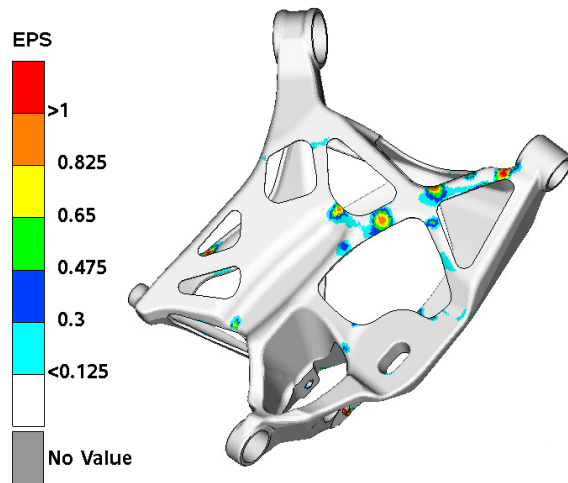
Table 4.15: Stiffness measured at Torsional stiffness (pt18), Damper attachment stiffness (pt68) and Spring stiffness (pt56) are compared to the stiffness requirements.

Stiffness	pt18	pt68	pt56
Discrepancy [-]	+8.33 %	+4.90 %	+7.80 %

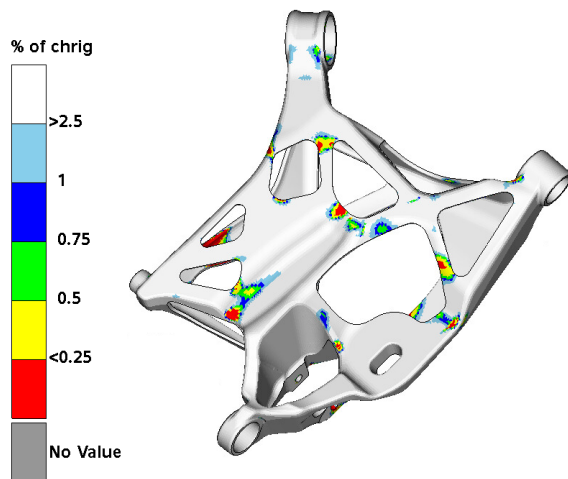
The effective stress, EPS and fatigue life are presented in Figure 4.14 where FE analyses are preformed after the topology design realization. Note that EPS is evaluated directly after the topology design realization step. The overall impression is that peak EPS and fatigue life are not too far away from fulfilling the standard component requirements.



(a) *Topology design realization.*



(b) *Topology design realization.*



(c) *Topology design realization.*

Figure 4.14: *The worst load case with respect to effective stress after the respective topology design realization.*

4.8.3 Discussion

The weight of the topology design realized part is almost the same as for Trial case 3, the increase is 0.6 %. Note that the topology optimization designs of Trial case 3 and 4 looks rather similar.

By comparing the effective stress between Trial case 3 and 4 after topology optimization, see Figure 4.14a and 4.11a, it is seen that Trial case 4 shows significantly smaller regions of high effective stress. This tendency is also seen when comparing fatigue life in Figure 4.12a and Figure 4.14c. The EPS can not be compared since it does not converge for the other Trial cases at this stage. Finally, Trial case 4 is aborted due to the time frame of this thesis.

4.8.4 Conclusions

- By including peak loads from the chassis rig cycle (robustness check) to the topology optimization task, it is possible to not only improve fatigue life but also reduce effective stress and EPS compared to Trial case 3 at the topology design realization step.
- The weight is 3.15 kg at the topology design realization step.

5 Discussion

This chapter discusses the component development process using structural optimization tools with respect to the RLCA application.

5.1 Trial cases

Firstly, it was believed that the largest design volume would yield the best performing component design in the end of the component development process, which is intuitive from a mathematical perspective. The manufacturing constraints investigated in TOSCA were not successfully implemented in the sense that the objective weight could be minimized further than the weight of the current component while achieving the same or better structural performance. It was also discovered that the initial design volume gives rise to two problems. The first is the sharp edges and tracks in the design volumes giving rise to problem areas such as high accumulation of effective stress when loaded. Secondly, the sparsely placed ribs of the topology optimized design requires that rather much material needs to be filled out in order to fulfil the manufacturability of the sand casting method. Therefore, a smaller design volume for this application proved to work better since not as much material needs to be filled out with respect to sand casting manufacturability. Note that filled out material due to the proposed manufacturing method is not placed with respect to topology optimization.

By comparing the four trial cases tested in this thesis, one can notice that Trial case 4 gives the best performance amongst the trial cases with respect to effective stress, EPS and fatigue life after the topology design realization step. Trial case 4 considers all stiffness, strength event and the robustness check loading to its topology optimization task. However, Trial case 4 gives the highest weight amongst the trial cases based on the reduced design volume subsequent to the topology design realization. It was discussed in section 4.3.4 that a volume constraint could be defined based on a topology optimization where the objective is to minimize weight under maximum displacement and compliance constraints. However, this specific volume constraint is not verified to be the most efficient one with respect to the full component development process using structural optimization tools. Furthermore, it should be noted that the considered topology optimization task where compliance is used as an objective gives a more continuous and distinct solution compared to having the weight as an objective.

It is seen that shape optimization is a rather powerful tool for this type of application when looking at the FE analysis in section 4.7.3. Unfortunately, all standard component requirements are not fulfilled due to the time frame of this thesis. However, the potential of shape optimization tools is demonstrated. Furthermore, the shape optimization step was divided into four sub-steps, this was not the idea from the start but proved to be necessary due to too high effective stress peaks when subjected to the worst case strength event. However, Trial case 4 has not been investigated with respect to shape optimization yet. It should also be noted that a growth control manufacturing constraint was incorporated, this facilitates the design realization step from a Design engineer perspective.

5.2 The component development process using structural optimization tools

By comparing the component development process using structural optimization tools to the traditional example presented in chapter 3, it can be noticed that the role as a CAE engineer is more involved throughout the component development process when using structural optimization tools. By using the traditional approach, the role as a CAE engineer is mainly focused on structural analysis. Hopefully, the component development process will be faster in terms of time using structural optimization tools since the placement of material is proposed by optimization methods compared to a traditional trial and error approach.

5.3 Application to other components

In this thesis, sand casting was proposed as the manufacturing method for all considered trial cases, although this might not be the case for other components subjected to this methodology. The main differences by subjecting another component to the process would most likely be the design realization step where manufacturability is considered or whether manufacturing constraints can be successfully incorporated to the topology optimization task. By changing the manufacturing method to additive manufacturing for example, this would more or less remove the design realization step because of the capability in complex geometry with this method. Although the process for both topology and shape optimization would still be necessary due to the fatigue and EPS requirements which requires the nonlinear FE and fatigue analysis handled by shape optimization. Another example could be a component where the load cases and boundary conditions subjected enables manufacturing constraints to be successfully incorporated to the optimization task in the sense that the structural performance fulfils requirements where the weight is lower compared to a current component. As mentioned in section 4.5 for Trial case 1, another chassis component was constructed by topology optimization using manufacturing constraints. This strategy was not applicable for a RLCA component however.

A general recommendation for how to use this methodology with respect to other chassis components is suggested:

1. Start with an initial design volume. Trim sharp edges which might lead to stress concentration regions. Note that the outer surface of the design volume could remain to the outer surface of the optimized component.
2. Perform topology optimization without manufacturing constraints.
3. Discuss and propose a manufacturing method together with a Design engineer. If several manufacturing methods are applicable, evaluate these different concepts separately throughout the topology optimization loop. Use manufacturing constraints if the topology optimization using manufacturing constraints gives a lower weight with better or equal structural performance compared to a current component.
4. Perform a standard FE analysis and evaluate the structural performance of the component by engineering judgement to determine if the component is ready for shape optimization or not. If the component is "too" far away from fulfilling the standard CAE requirements, the optimization task can be re-formulated with tighter constraints. If the weight is too high after topology design realization due to the manufacturing method, the design volume can be reduced. If the components over achieves with respect to stiffness requirements, this means that the weight could be reduced further.
5. Lastly, choose the concept with the lowest weight together with best structural performance if several concepts are evaluated. Thereafter, verify the manufacturing feasibility and proceed with shape optimization to fulfil standard CAE requirements. In the case of cast component, a person with cast expertise can be consulted to verify the component design due to manufacturability if needed.

6 Further work

This chapter proposes some of the further work that can be done based on this thesis.

6.1 Trial cases

Trial case 4 shows the most promising structural performance after the topology design realization amongst the trial cases. Unfortunately, there was no time to proceed in the process with this candidate. It would therefore be interesting to proceed through the component development process with Trial case 4.

Furthermore, the design volume could be reduced further to produce more trial cases that could lower the weight compared to the trial cases produced while fulfilling the standard component requirements. When looking at stiffness requirements, this seems possible if one argues that a smaller design volume yields lower stiffness while keeping the optimization task constant.

The verification of feasible design due to manufacturing is done by the Design engineer. In this thesis sand casting is proposed, this yields rather complex problems which needs to be treated including material solidification and design of moulds amongst other issues. This step could therefore be improved by including a person with expertise in the area of sand casting to verify the feasibility at the topology design realization step. Furthermore, a specific volume constraint is used for Trial case 2 to 4, this volume constraint could be investigated further.

Looking into shape optimization, the work flow presented in Figure 4.10 on page 26 for Trial case 3 has not been verified to be the most efficient work flow. Therefore, the order of these processes can be investigated to find the most efficient shape optimization process flow. This also requires that the process needs to be completed in the sense that the standard component requirements are fulfilled.

6.2 The component development process

Firstly, the "satisfies surrogate component requirement?" step in the process where the structural performance status after topology design realization is estimated by engineering judgement has not been defined during this thesis work, see Figure 6.1. This requires further investigation where additional trial cases can be introduced and completed with respect to standard component requirements to define such a so-called surrogate requirement.

A manufacturing related issue when considering casting, is how to include cast simulation to the component development process. This will enable that the component is so-called build-to-print at the end of the process. A proposed strategy for this is presented in Figure 6.1 where the proposed process in Figure 3.2 on page 12 has been modified. The idea is to perform a cast simulation after the topology design realization to avoid that the shape optimization work needs to be performed twice due to added cast gates or change of rib thickness due to solidification. However, this yields only for cast components as the manufacturing method.

Lastly, this methodology can be investigated by application to other components.

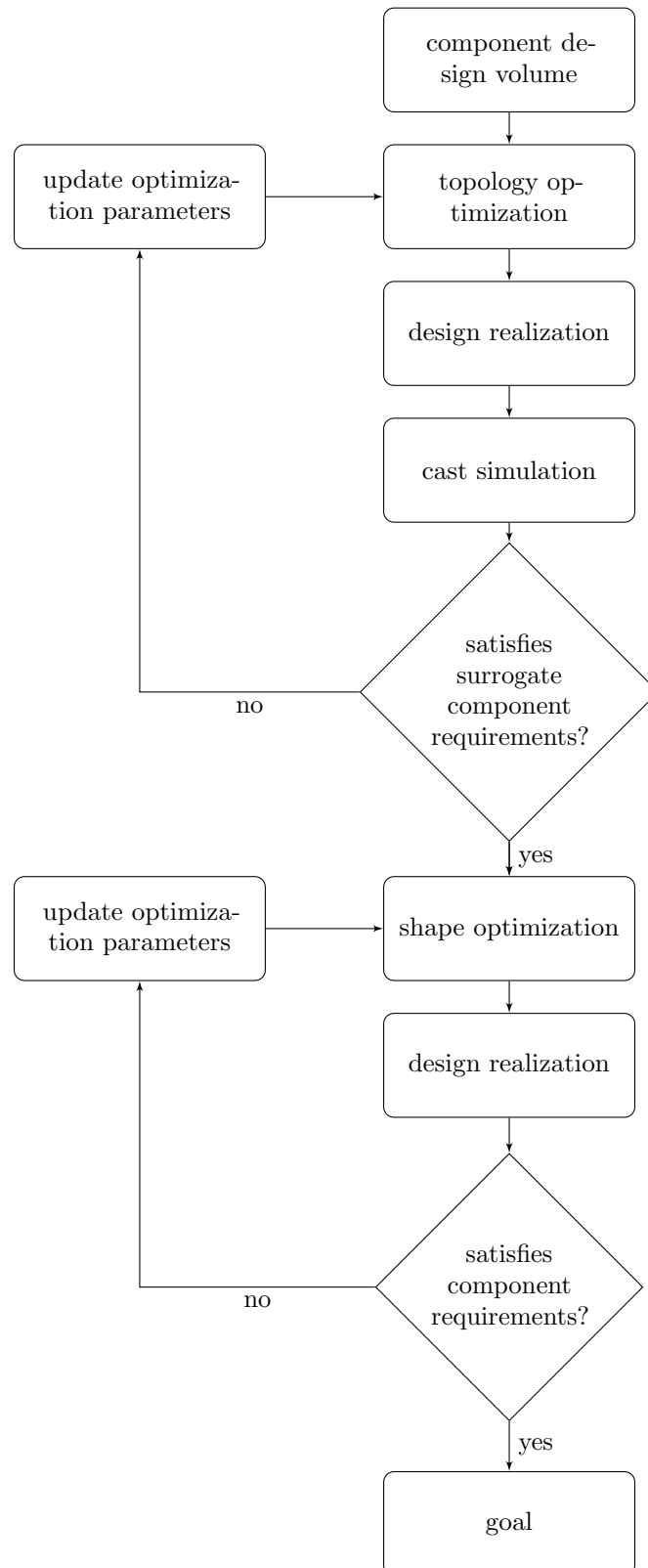


Figure 6.1: *Flowchart of the component development process using structural optimization tools with respect to casting.*

References

- [1] *Abaqus 6.12 Online Documentation*. Version 6.12. Dassault Systèmes, 2012.
- [2] E. Andreassen et al. Efficient topology optimization in MATLAB using 88 lines of code. English. *Structural and Multidisciplinary Optimization* **43.1** (2011), 1–16.
- [3] M. P. Bendsøe, N. Olhoff, and O. Sigmund. *IUTAM Symposium on Topological Design Optimization of Structures, Machines and Materials : Status and Perspectives*. English. Vol. 137. Dordrecht: Kluwer Academic Publishers, 2006;2014;
- [4] M. P. Bendsoe and O. Sigmund. *Topology optimization : theory, methods and applications*. English. 2.;2; Berlin: Springer-Verlag, 2003;2004;
- [5] M. P. Bendsøe and O. Sigmund. Material interpolation schemes in topology optimization. English. *Archive of Applied Mechanics* **69.9** (1999), 635–654.
- [6] J. Chen et al. Shape optimization with topological changes and parametric control. English. *International Journal for Numerical Methods in Engineering* **71.3** (2007), 313–346.
- [7] P. W. Christensen et al. *An introduction to structural optimization*. English. Vol. 153. Dordrecht: Springer, 2009;2008;
- [8] *DesignLife Theory Guide*. Version 9.1. HBM United Kingdom Limited, 2014.
- [9] K. Hansen Andreasson and M. Linder. *Optimization of a wheel suspension packaging*. English. M.Sc. thesis, Institutionen för produkt- och produktionsutveckling, Chalmers tekniska högskola, 2016.
- [10] A. Holmström. *Optimized Development Process for Cast Components*. Lic. thesis, Institutionen för tillämpad mekanik, Material- och beräkningsmekanik, Chalmers tekniska högskola, 2005.
- [11] B. Kanishk and J. Skön. *Balancing of wheel suspension packaging, performance and weight*. English. M.Sc. thesis, Institutionen för produkt- och produktionsutveckling, Chalmers tekniska högskola, 2016.
- [12] R. Meske, J. Sauter, and E. Schnack. Nonparametric gradient-less shape optimization for real-world applications. English. *Structural and Multidisciplinary Optimization* **30.3** (2005), 201–218.
- [14] O. Sigmund and J. Petersson. Numerical instabilities in topology optimization: A survey on procedures dealing with checkerboards, mesh-dependencies and local minima. English. *Structural Optimization* **16.1** (1998), 68–75.
- [15] *SIMULIA Tosca Structure Documentation 8.1*. Version 8.1. FE-DESIGN GmbH, a Dassault Systèmes company, 2014.
- [16] K. Svanberg. The method of moving asymptotes—a new method for structural optimization. English. *International Journal for Numerical Methods in Engineering* **24.2** (1987), 359–373.

A Additional topology parameter study results

In this appendix, additional parameter study results are presented.

A.1 Membersize constraint

In Figure A.1, a member size constraint is used. The stiffness load cases and all strength events are subjected. The optimization task was to minimize compliance. It introduces many thin ribs in between of the general outline of the structure which makes it difficult for the Design engineer to realize.

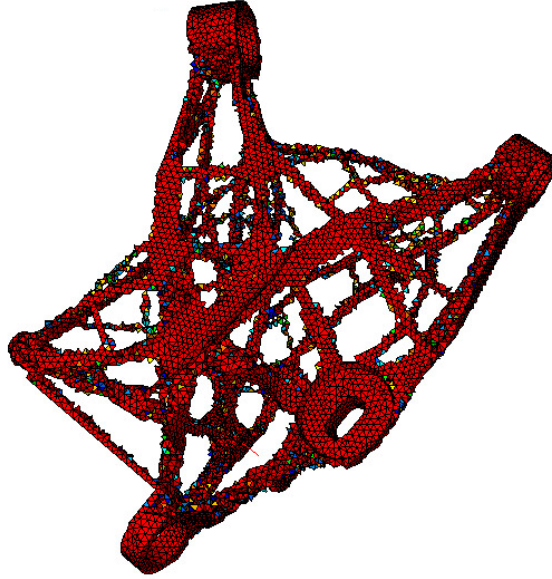


Figure A.1: *The topology optimization design using a member size constraint.*

A.2 Demold constraint

In Figure A.2, the demold constraint auto is used. The stiffness load cases and requirements are subjected, the optimization task was to minimize mass. The objective mass is here 4.4 kg, this is above the current component weight. The demold constraint is changed to auto midplane using the same optimization as described above, the design can be seen in Figure A.3. The objective mass is then 7.67 kg.

A.3 A comparison between sensitivity based and controller based solution methods

In Figure A.4 the element densities of the resulting structure of two optimization algorithms are presented. The optimization task is to minimize compliance under a volume constraint. The same type of loading is subjected to both optimization tasks. It can be seen that the controller based method produces solid elements whereas the sensitivity based method produces intermediate densities.

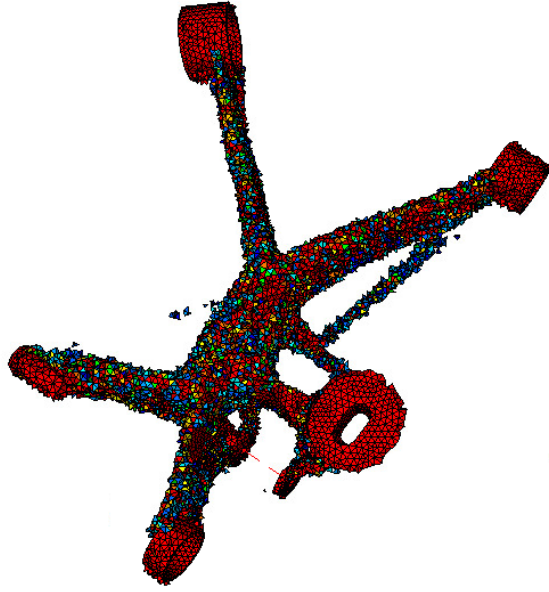
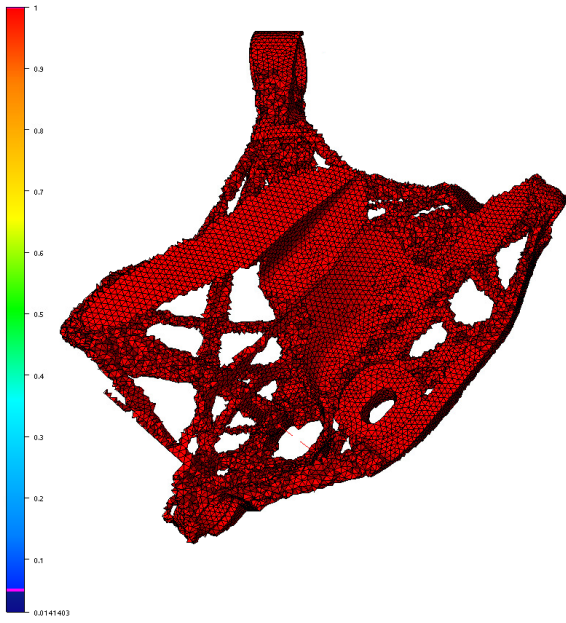


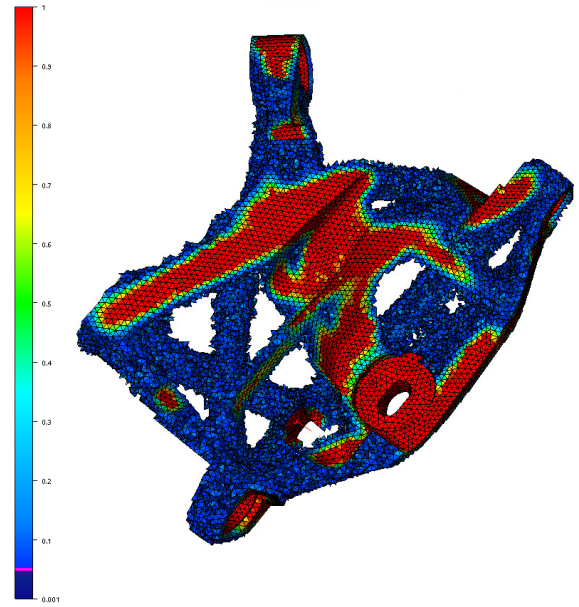
Figure A.2: *The topology optimization design using auto demold constraint.*



Figure A.3: *The topology optimization design using auto tight demold constraint.*



(a) Controller based OC method



(b) Sensitivity based MMA

Figure A.4: A comparison between the controller based OC and sensitivity based MMA method. Red color indicates solid element whereas blue color indicates void element.