CHALMERS
UNIVERSITY OF TECHNOLOGY

Structural optimization of base engine component

Master's thesis in Applied Mechanics

JOHAN SPARLUND

Department of Applied Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2017

MASTER'S THESIS 2017:36

Structural optimization of base engine component

JOHAN SPARLUND

CHALMERS
UNIVERSITY OF TECHNOLOGY

Department of Applied Mechanics
Division of Material and Computational Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2017

Structural optimization of base engine component
JOHAN SPARLUND

Supervisor: Magnus Levinsson, Volvo Car Corporation
Examiner: Magnus Ekh, Department of Applied Mechanics

Master's Thesis 2017:36
Department of Applied Mechanics
Division of Material and Computational Mechanics
Chalmers University of Technology
SE-412 96 Gothenburg
Telephone: +46 31 772 1000

Structural optimization of base engine component
JOHAN SPARLUND
Department of Applied Mechanics
Chalmers University of Technology

Abstract

The purpose of this thesis has been to investigate the potential implementation of structural optimization software in the product development process at the department of engine development at Volvo Car Corporation. This has been carried out by performing two trial cases of optimization on two different base engine components. The models have been created in ANSA and solved with ABAQUS and TOSCA.

The first trial case was of a simpler nature and involved a smaller component inside the bedplate. Its purpose was mainly to bring knowledge about FE-modelling and optimization, which it fulfilled. The second trial case was more complex, and aimed to find the optimal material distribution on the engine block's exterior surface in order to increase the first eigenfrequency of the component. It succeeded in the sense that it increased the eigenfrequency, but the optimized topology does not contain rib-like structures to a large extent, as was hoped. Rather, the majority of the element are spread thinly over the entire surface.

The ultimate conclusion that can be drawn from the results is that it is complicated to implement the use of structural optimization software on larger base engine components. Optimization is better suited for more isolated components of smaller size, with a limited number of load cases that are clearly defined. This would give the optimization solver a better chance to find an improved structure. It would also limit the need for simplifications and therefore allow for greater confidence in the FE-model and in turn the optimization results. The findings in this thesis does not completely reject the potential implementation of structural optimization software in the product development phase, but does show that structural optimization is not without its flaws and limitations.

Keywords: topology optimization, TOSCA, engine development

Preface

This master's thesis has been written as the final part of the master's programme Applied Mechanics during the spring of 2017. It has been carried out a the department of engine development at Volvo Car Corporation. I would like to thank my supervisor Magnus Levinsson for all the help and input, my examiner Magnus Ekh for guidance and all my co-workers for the warm welcome I have received.

Johan Sparlund, Gothenburg 9/6-2017

Contents

1
Introduction

This chapter describes the background, purpose and limitations of the thesis and the methods used. It ends with describing an outline of the thesis.

1.1 Background

As the car industry sees an increasing demand on fuel economy, while at the same time customers wish to retain performance, engines are downsized. That leads to smaller engines with higher specific power which increases the loads. This is a difficult task for the engineers to tackle, especially when lead times are continuously expected to decrease. The currently used design procedure at the department of engine development is an iterative process where a proposed structure is analyzed using computational simulations and redesigned by hand until it meets the criteria. This is a time consuming method that requires both a CAE engineer and a design engineer working together. One way to decrease product development time and the required number of man-hours is to use optimization software, where the iterative design process has been automated. This type of product development aid requires less input from a design engineer and can create multiple design suggestions quickly. Optimization software has successfully been implemented in other departments' product development process and it is therefore believed it could be used for engine development too. Some smaller attempts has been made within the team to evaluate its applicability but no one has had the time to properly dive into the subject.

1.2 Purpose

The purpose of this thesis is to investigate the possibilities of applying structural optimization software in the development process within the department.

1.3 Method

The applicability of topology optimization will be tested by performing two trial cases. The first trial case will be of a simpler kind, to get experience of working with FE-modelling and optimization software, and the second more complex. The FE-model and optimization model will be created in the pre-processor ANSA [1]. The optimization software TOSCA [2] will be used in association with the FE-solver ABAQUS [3]. ABAQUS and TOSCA are both developed by SIMULIA [4] and could therefore be assumed to work well together. Post-processing of the FE-results will be done in META [5] and the optimization results in the viewer supplied by the TOSCA software suit.

1.4 Limitations

The FE-model will not consider any form of temperature dependence and is limited to linear elastic, isotropic material models. Residual stress from manufacturing will not be considered. The influence of the particular choice of optimization software and FE-solver will not be looked into.

1.5 Thesis outline

The thesis starts with chapter 2 explaining the theoretical background to structural optimization and its application in TOSCA. Chapter 3 & 4 presents and discusses the methodology and results from the two trial cases. The final chapter, 5, presents the conclusions that can be drawn from the work and suggests future work. The last page contains the references.

2

Structural optimization

In this chapter structural optimization and its application in CAE software is presented. It starts with a brief explanation of the general concept of optimization, followed by a description of its underlying theory and application in the optimization software TOSCA. For an in depth explanation of the theory behind structural optimization the reader is directed towards e.g. Christensen and Klarbring [6]. The reader is expected to be familiar with the basic concept and purpose of the FE method. For an overview of the FE method the work by Ottosen and Peterson [7] is recommended reading.

In a strictly mathematical approach, the purpose of an optimization scheme is to find the minimum or maximum value of a function for its admissible input values. The application of this concept to the subject of structural mechanics is referred to as structural optimization. This translates to finding the optimal design of a structure and in practice this is performed by discretization of a mechanical problem to a computational simulation model. In this applied case of optimization the function pertains to a structure's mechanical properties and the admissible input values can be interpreted as the limiting outside factors for the structure.

2.1 Theory

In Christensen and Klarbring [6] it is stated that the structural optimization problem consist of the following three parts:

- Objective function, $f(x,y)$. Classifies the goodness of the design, by measuring a quantity such as weight, eigenfrequencies or elastic strain energy.
- Design variable, x. Describes the structures design.
- State variable, y. Represents the structural response, for example displacement or effective stress.

The problem can then be formulated as:

$$\begin{cases} \text{Minimize } f(x,y) \text{ w.r.t. } x \text{ and } y \\ \text{Subjected to } \begin{cases} \text{Behavioral constraints on } y \\ \text{Design constraints on } x \\ \text{Equilibrium constraints} \end{cases} \end{cases} \tag{2.1}$$

The behavioral constraints on the state variable y are often written as as a function of g, as $g(y) \leq 0$. Design constraints are constraints pertaining to the design variable x. The equilibrium equations in a linear elastic FE problem look like:

$$\boldsymbol{K}(x)\boldsymbol{u} = \boldsymbol{F}(x)$$

where $\boldsymbol{K}$ is the stiffness matrix, $\boldsymbol{F}$ the force vector and $\boldsymbol{u}$ the displacement vector corresponding to y in the problem above. Equation (2.1) is referred to as a *simultaneous* formulation. The equilibrium and optimization problems are solved simultaneously and the variables x and y are treated as independent of each other. Another way to formulate the problem is the *nested* formulation, where the equilibrium equation uniquely defines y for a given x. If one treats $\boldsymbol{u}$ as a function of x via the solution of the equilibrium equations then we can write the optimization problem as:

$$\begin{cases} \text{Minimize } f(x,\boldsymbol{u}(x)) \text{ w.r.t. } x \\ \text{Subjected to } g(x,\boldsymbol{u}(x)) \leq 0 \end{cases}$$

The nested formulation is what is used when solving the problem numerically. To find the solution one typically need to find the derivatives of f and g, a process referred to as *sensitivity analysis*.

2.1.1 Multiple objective functions

It is possible for an optimization problem to contain multiple objective functions. For a problem with N objective functions the problem is formulated as:

$$\begin{cases} \text{Minimize } f(f_1(x,\boldsymbol{u}(x)), \ldots, f_N(x,\boldsymbol{u}(x))) \text{ w.r.t. } x \\ \text{Subjected to } g(x,\boldsymbol{u}(x)) \leq 0 \end{cases}$$

As the different objective functions can be contradictory to one another and they are all minimized separately, there will not exist a distinct optimal solution. One way to solve this issue is to assign a weight to each objective function, making them more or less important to the overall optimization objective and therefore end results. A weight w_i is assigned to each objective function. The weights are commonly scaled to be in the range of 0 to 1, and the sum of them are 1:

$$f = \sum_{i=1}^{N} f_i w_i \,, \quad w_i = [0,\ 1], \quad \sum_{i=1}^{N} w_i = 1$$

If a specific objective function is believed to be of greater importance for the design of the structure it is assigned a greater weight – and vice versa.

2.1.2 Classes of structural optimization

Generally speaking, there are three types structural optimization problems. Depending on what is parametrized, it is usually divided into:

- Topology optimization: the design variable x represents connectivity in the domain. It is the most general formulation of a structural optimization problem.
- Sizing optimization: the design variable x represents some type of structural geometry measure, such as cross-sectional area of a truss.
- Shape optimization: the design variable x represents the form of the boundary of the domain. The connectivity of the structure is not changed and no new boundaries are formed.

The focus of this thesis has been on topology optimization.

2.2 Topology optimization in TOSCA

TOSCA is a commercial software [2] and it can be used for topology optimization to determine the optimal material distribution in a given design space, to best meet a the objective function and the constraints. The basis for this is an FE-model that throughout the optimization process will have its topology altered. The flow chart in figure 2.1 shows the general outline of the iterative process of solving the topology optimization problem using TOSCA. Next to the algorithmic flowchart a simple example problem is presented: a clamped bar with a point load. The objective function could pertain to the bar's stiffness and the goal could be to maximize it. The design constraint could be to decrease its area by some amount.

First one has to define the design space, i.e. the set of elements making up the area in which the optimization software is allowed to perform changes in the topology. In the example case, let us assume every element is in the design space. The concept of design space and element's contribution to the global stiffness matrix is presented in section 2.3.

TOSCA initializes the solution process by guessing a design to meet the criteria. The proposed design is analyzed using the FE method. As TOSCA does not supply its own FE solver, a third party solver has to be employed. As stated earlier, in this thesis ABAQUS has been used, but several others are also applicable. The results from the FE analysis are used as the input in what can be considered the core process of TOSCA: the mathematical optimization and consequent update of the design. The details of the optimization algorithms used in TOSCA are described in section 2.5. If, after the design update the solution has converged (see section 2.5.2 on convergence), the problem is solved. If not, the newly proposed design is sent through the process again.

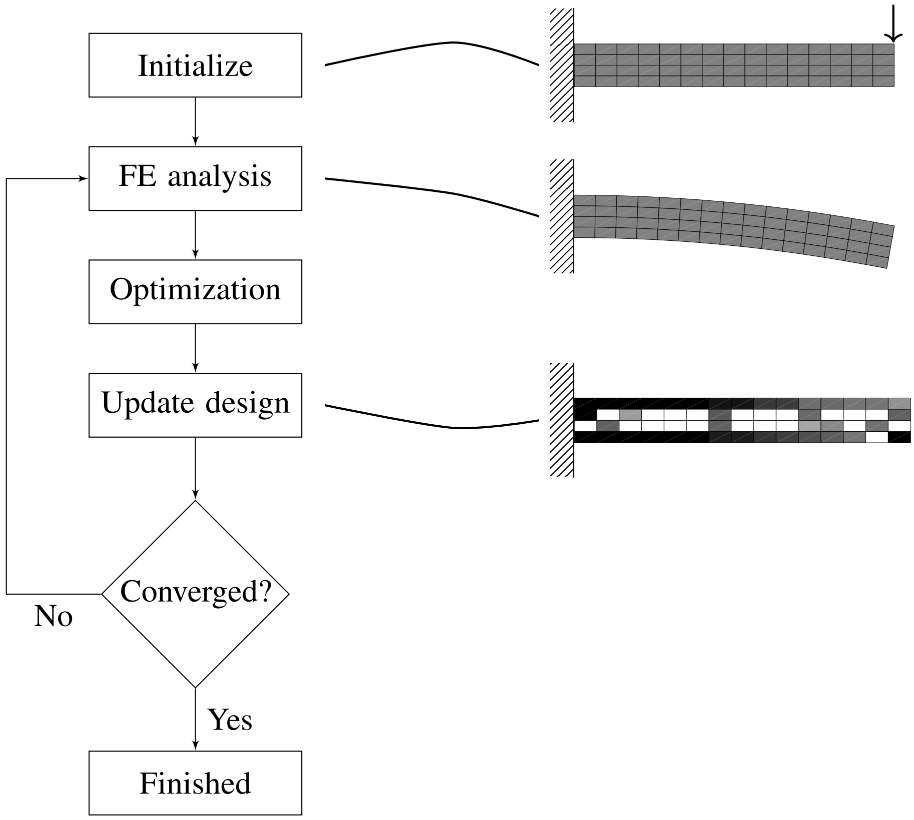


Figure 2.1: Flowchart of the optimization process.

2.3 Material formulation

In order to find which design that minimizes or maximizes the objective function, one has to describe the design in the spatial volume mathematically. The simplest approach is to use an integer material formulation wherein an element exists either as material or void. This is done by letting the design variable vector x represent the *relative densities*, ρ_e. An element having a relative density of 1 is contributing to the global stiffness, while a relative density of 0 means that the element is not contributing. Subscript e denotes individual element number and E_0 is the Young's modulus of a given material. Ω is the design space and one searches for the optimal subset of material points Ω_{opt}. This is referred to as the 1-0 problem and is shown below.

$$E_e = \rho_e E_0, \quad \rho_e = \begin{cases} 1 \text{ if } e \in \Omega_{opt} \\ 0 \text{ if } e \notin \Omega_{opt} \end{cases}$$

For an integer formulation it is not possible to derive the sensitivities used to solve the system numerically, as the design variable vector is not continuous. Therefore one switches to a continuous material formulation where elements can have non-integer densities in the range of 0 to 1, making the design variable vector continuous. The downside of this is that you have to interpret the stiffness contribution of an element having intermediate relative density – basically meaning it is partly void and partly material. This is done using a material penalization scheme, of which TOSCA offers the following two alternatives:

- SIMP (Solid Isotropic Material with Penalization)
- RAMP (Rational Approximation of Material Properties)

Both of the above are described in detail in [8]. They interpolate material stiffness in the following ways:

$$\text{SIMP: } \mathrm{E_e} = \mathrm{E_0}\rho_{\mathrm{e}}^{p}$$

$$\text{RAMP: } \mathrm{E_e} = \mathrm{E_0}\frac{\rho_{\mathrm{e}}}{1+p(1-\rho_{\mathrm{e}})}$$

SIMP is recommended to be used in case of static load cases, and RAMP for dynamic load cases [9]. Figure 2.2 depicts the calculated relative density ρ_e against how the Young's modulus of an element is penalized using the SIMP and RAMP method respectively, for different penalty factors p.

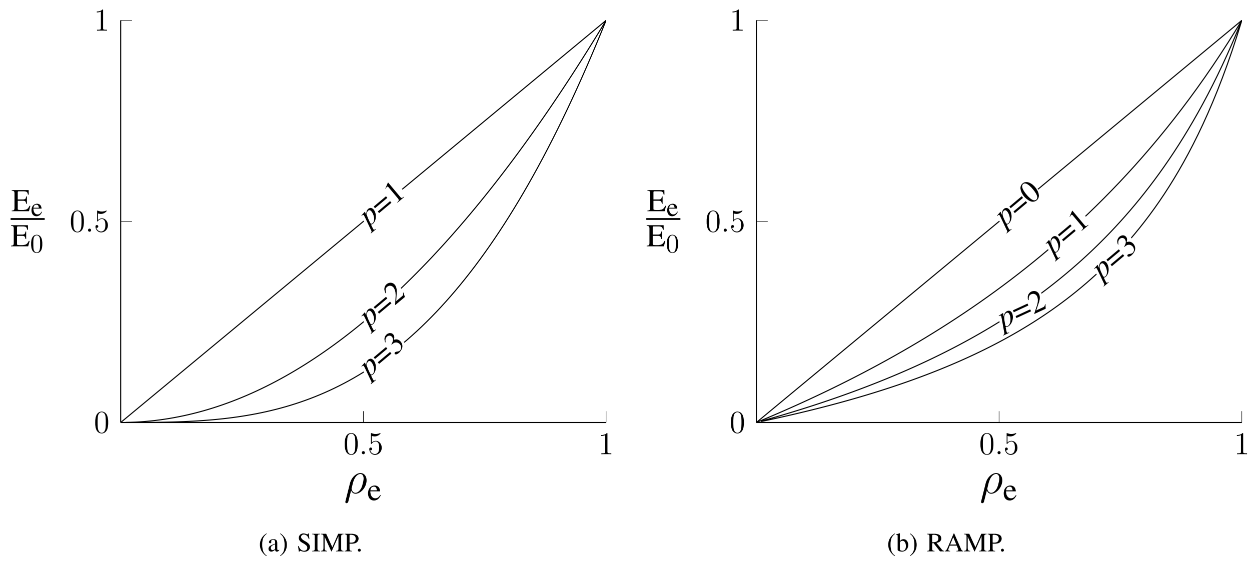


Figure 2.2: Penalized element stiffness against relative element density using SIMP and RAMP with different penalty factors p.

For a higher penalty factor the number of elements with intermediate densities decrease, which is desirable. But a too large penalty factor leads to an increased chance of the optimization algorithm stopping on a local minima rather than the global one [8]. Numerical experiments has proven a good value for the penalty factor p to be around 3 for both schemes [9].

2.3.1 Optimizing for distribution of two materials

It is possible to optimize for the distribution of two materials, rather than just between material and void. This is achieved by limiting the lower end of the range of admissible values for the relative density. Figure 2.3 depicts a material interpolation curve using SIMP with a penalty factor of p of 2, relating relative density to relative stiffness. The shaded area represents non-admissible element properties.

Typically one knows the Young's modulus of the two materials to be optimized for. In figure 2.3 a relative stiffness of 1 corresponds to the stiffer of the two materials and E_{low} the weaker. From E_{low} one can determine ρ_{low} to be used as the lower end of the admissible relative density range.

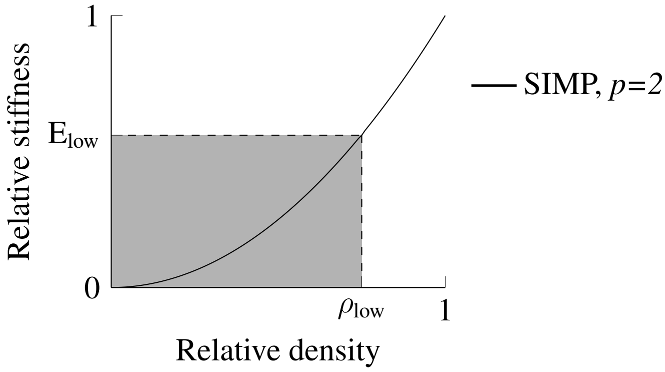


Figure 2.3: The material interpolation scheme SIMP with a penalty factor *p* of 2, with a non-zero lowest value of relative density. The grey area represents the non-admissible element properties.

2.4 Mesh

TOSCA supports a wide variety of element types in the design space: 2D, 3D, different number of nodes and elements of different orders. While the element formulation has a great affect on the FE-results, it is of lesser importance for the optimization solver itself [9]. The number of elements is still important though, as a finer mesh will yield a higher resolution structure than a course mesh will.

2.5 Solving the optimization problem in TOSCA

TOSCA has two distinctly different algorithms for solving the optimization problem, these are referred to as the sensitivity based solver and the controller based solver. Both have their pros and cons, and which one to use depends on the type of optimization problem at hand. The controller based solver is more limited in terms of quantities that can be used, and one often founds that more complex sensitivity based solver is needed. Below follows a brief description of the two:

Sensitivity based solver

The sensitivity based solver uses the Method of Moving Asymptotes (MMA), described in [10]. The sensitivity based approach offers a wider variety of possible constraints and objective functions than its counterpart. The complete list of available design responses to use can be found in [9], but for example it could be: eigenfrequencies, reaction forces or center of gravity. The solver uses semi-analytical sensitivities based on finite differences to update the stiffness matrix:

$$\frac{\partial \boldsymbol{K}}{\partial x} = \frac{\boldsymbol{K}_{0+\mathrm{p}} - \boldsymbol{K}_0}{\Delta x}$$

$\boldsymbol{K}$ is the updated stiffness matrix pertaining to the updated design and $\boldsymbol{K}_0$ is the stiffness matrix for the unchanged current state. $\boldsymbol{K}_{0+\mathrm{p}}$ is the *perturbed* stiffness matrix, where the design parameters has been changed slightly. The denominator Δx is the change in the design variable values. To find the sensitivities TOSCA linearises around the current state by adding so-called matrix steps and pseudo loads to the ABAQUS FE-model. While these steps have to be solved for, they do not affect the FE-results and are only needed in the iteration process to solve the optimization problem. Depending on what quantity is optimized for, TOSCA adds a different amount of matrix steps and pseudo loads. Compliance (a quantity explained in section 2.5.1) and eigenfrequency are good quantities to optimize for, partially because it does not add pseudo loads, unlike what for example a minimization of displacement would.

Controller based solver
The controller based solver does not solve for the sensitivities, instead the strain energy and grid point stresses are used. It is limited to constraints on volume and an objective function pertaining to compliance. The TOSCA manual [9] does not go into any details in explaining the inner workings of the solver, why there are uncertainties regarding its function. The advantage of the controller based approach is mainly that it is faster than the sensitivity based.

2.5.1 Optimization quantities

What quantities to constrain and optimize for depends on the goal of the project and the limitations of the FE-model and optimization solver. The choice of objective function measurement and constraints are crucial as they are the main deciding factors of the end results.

2.5.1.1 Objective function

The following quantities has been used as objective functions in this thesis:

Compliance
Compliance is quantity commonly used in optimization, that is seldom seen elsewhere. It is a scalar quantity that measure the elastic strain energy. It is usually denoted C, and it inversely relates to the structures stiffness. Typically one wants to minimize the compliance, as a lower compliance equals a higher stiffness. It is formulated as:

$$C = \boldsymbol{F}(x)^{\mathrm{T}}\boldsymbol{u}$$

Eigenfrequencies
A structural optimization model involving eigenfrequencies is less straight forward to define than one using compliance. Typically one wishes to include the first or the first few eigenmodes, which leads to a multi-objective function problem. It is recommended to use the supplied Kreisselmaier-Steinhauser method for weighing the objective functions [9]. The objective function is then defined as:

$$\boldsymbol{f}_{\mathrm{KS}} = -\frac{1}{k}\ln(\sum_{\mathrm{i}=1}^{\mathrm{N}} \exp(kf_{\mathrm{i}}))\,, \quad \text{by default } k = \frac{30}{f_{\mathrm{min}}}$$

k is the weight for each eigenmode (see section 2.1.1 for concept of weighted objective function), f_{i} are the eigenfrequencies and N is the number of eigenmodes included in the objective function. This method eliminates the need for mode tracking which is computationally expensive.

2.5.1.2 Constraints

The following constraints have been used in different capacities in this thesis:

Symmetry
A design space can be constrained to various forms of symmetry. Note, this does not require a symmetric mesh in TOSCA.

Volume
Volumetric constrain on a design space is possibly the most straight forward to understand. The design space is constrained to decrease its volume by a specific amount, as to limit extent of the material usage.

2.5.2 Convergence

TOSCA determines convergence based on the change in value in either the objective function or the relative densities. If the current design iteration fulfills the constraints and the percentage wise change in the convergence measurement from the previous iteration is deemed small enough, the solution is said to have converged. Convergence in objective function or relative densities does not necessarily translate to a converged topology, therefore it can be beneficial to force the optimization algorithm to perform at least a certain numbers of iterations.

As there is nothing that says that a converged solution necessarily has reached the global optimum the solution will probably stop on what is one of several local optima. This means that it possibly exists multiple solutions that could have been reached, depending on for example the starting solution guess. Therefore it can be a good idea to run multiple versions of the same optimization problem with different starting guesses on the relative element densities, to confirm that they reach a similar solution.

2.5.3 Filtering

It is necessary to process the optimization solver's results in order to limit mesh dependency and the existence of numerical instabilities, a process referred to as *filtering*. A common numerical issue resolved by filtering is the occurrence of checkerboard patterns. It refers to neighbouring elements alternating between being material and void in a checkerboard-like pattern. This type of material layout makes sense from a mathematical perspective, but not in a mechanical or manufacturing sense. As the elements are only in contact in their infinitesimal corners, they can not carry loads. This numerical issue is resolved with what is referred to as a sensitivity filter, developed by Sigmund [11]. It works by applying a smoothing to the sensitivity analysis. Although its effect on mesh independence and checkerboard patterns has not been proven mathematically, experience has shown it to be useful. In [8] it is expressed as:

$$\frac{\partial f}{\partial \rho_{\mathrm{e}}} = \frac{1}{\rho_{\mathrm{e}} \sum\limits_{\mathrm{i}=1}^{\mathrm{N}} P_{\mathrm{i}}} \sum_{\mathrm{i}=1}^{\mathrm{N}} P_{\mathrm{i}} \rho_{\mathrm{i}} \frac{\partial f}{\partial \rho_{\mathrm{i}}}, \quad \text{with } P_{\mathrm{i}} = \mathrm{r}_{\mathrm{min}} - \mathrm{dist(e,i)}$$

N is the total number of elements in the domain. The function dist(e, i) is defined as the distance between the center of the current element e and the center of element i, see figure 2.4. Only the shaded element in said figure have an affect on the filter and consequently the minimum structure size. The dots are the element center points used by the filter.

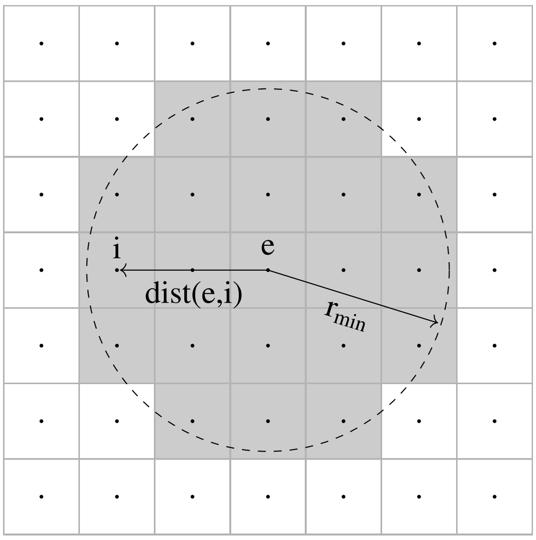


Figure 2.4: An example domain with 49 elements, to demonstrate the sensitivity filter.

TOSCA offers two sensitivity filters referred to as a STANDARD and LOW, where STANDARD is recommended for finer meshes and vice versa. What length of filter radius r_{min} to use is based on experience and the type of task at hand, but an r_{min} twice as large as the mean element edge length is recommended as a starting point [9]. If the filter radius is too large the structure will not be detailed. A too small radius will results in an overly fine structure.

2.5.4 Parameter study

In the previous sections the methods and parameters that make up a structural optimization problem have been presented. With these, the user will have to make choices that affects the end results. It can be beneficial to perform small scale parameter study, i.e. to run several similar optimization jobs with varying settings and various types of constraints. For example one can achieve different results if the eigenfrequencies are to be maximized or simply constrained to be above a specific limit.

3

Trial case 1: bearing support inserts

This chapter presents the methodology and results of performing a topology optimization aiming to find the ideal material distribution in a part cast into the bedplate. After an introduction of the subject matter of the trial case, the FE- and optimization model are explained and the results are presented and discussed.

Figure 3.1 shows the engine block and the bedplate. The block and bedplate are connected by some M8 and M10 bolts, not visible in the figure. When the bedplate is cast, four bearing support structures are cast into it, referred to as inserts. Their purpose is mainly to decrease the risk of fatigue of the bedplate caused by the loads from the crank shaft, that goes through the engine block and the bedplate. The engine block and bedplate are made out of aluminium and the inserts out of the heavier material cast iron. Due to the heavy weight of cast iron it would be desirable to find the optimal distribution of cast iron and aluminium in the volume taken up by the insert.

The CAD-models depicted in this chapter are not the actual geometries used in the optimization model. This is because the used CAD-models are still under development and are therefore not suitable for publishing in a master's thesis. The components depicted are from an older generation, but they still hold the same functions and the geometries are similar. Note that the model colors are not related to material properties, and are chosen only as to more easily distinguish different components.

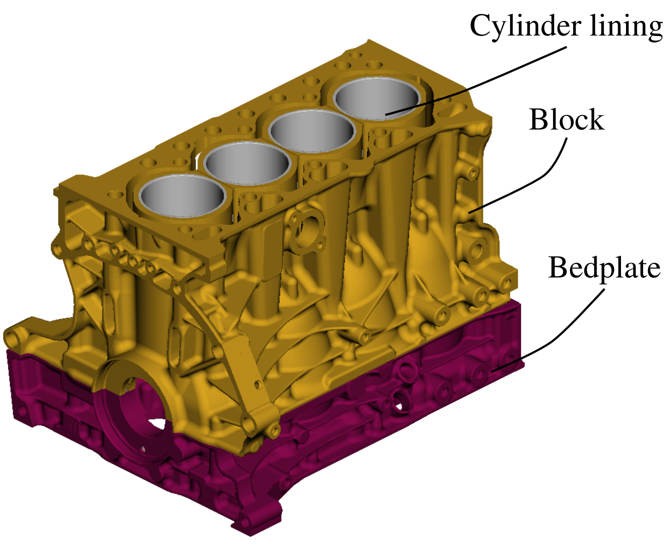
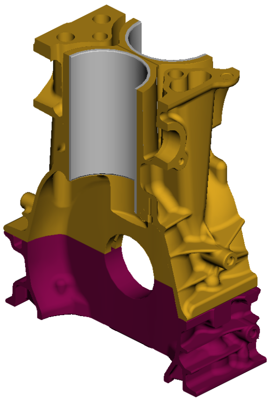


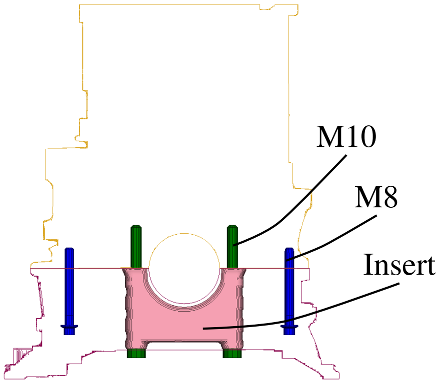
Figure 3.1: Isometric view of the block, bedplate and cylinder lining. Not visible are the main bearing inserts and the bolts.

3.1 FE-model

Given the iterative nature of solving the optimization problem it is a time consuming process, why it is crucial to not have too computationally expensive FE-model. In order to avoid this, simplifications often have to be made. In this case it means to instead of creating a model representing the entire engine block it is represented by the two opposing halves of the middle cylinders, see figure 3.2 and compare with figure 3.1. This reduces the simulation time greatly.



(a) Isometric view of the simplified model with two opposing cylinder halves.



(b) Front view of outlines the simplified model. Block and bedplate drawn as outlines, and cylinder liner not visible.

Figure 3.2: Simplified FE-model.

3.1.1 Mesh

The mesh was created using the pre-processor ANSA and consists of roughly 1.2 million elements. The insert, bedplate and bottom third of the block have an average node spacing of 1.5mm, while the rest of the model has an average of 5mm. The purpose of this is to have a finer discretization where the majority of the deformation will happen. The volumetric mesh for all components, except for the insert, is created with a growth factor of 20% inwards from the surface. The insert has a uniform volume element size to not skew the optimization results, as larger elements would be favored in the optimization process. The element type used is a first order (linear) tetrahedral element with 4 nodes and 1 integration point, referred to as C3D4 in the ABAQUS documentation [12]. It is a fact that using first order elements will yield an overly stiff response compared to the more accurate and more computationally heavy second order elements [7]. But it was found by a colleague that when decreasing the element order from second to first on an engine block FE-model regularly used within the department, the results changed only slightly. Based on that, first order elements were considered accurate enough to be used for this application.

3.1.2 Load cases and boundary conditions

Forces are applied on all the nodes on the inner boundary of the bearing. The load cases are sampled at different crank angles and represent the forces and moments acting on the bearing by the crank shaft. Their magnitude and directions have been obtained by another team within the same department. The number of load cases to include in the simulation is a balance act, as too many would give a too long computational time and too few would not accurately represent reality. Four load cases were included, leading to that the FE-problem took about 45 minutes to solve on the computer cluster. All forms of temperature loads are disregarded as they are believed to have little effect on the outcome of the optimization of the bearing support.

To avoid rigid body motion, the nodes on the top of the engine block are fixed. Before any load is applied, the bolts connecting the bedplate to the engine block are pre-tensioned. This is done in three steps: first a specific displacement is prescribed, then a force corresponding to restoring that displacement and finally a step locking the pre-tension.

3.1.3 Contact

The FE-model has contact conditions in the following locations:

1. Bedplate to bearing support
2. Block to two M8 bolts
3. Block to two M10 bolts
4. Block to cylinder liners
5. Block and bedplate to bearing
6. Bedplate to engine block
7. Bedplate to two M8 bolts
8. Bedplate to two M10 bolts

FE problems with contact conditions are non linear and hence computationally heavy. Item 1-4 in the list above have a contact condition referred to as tie, which 'glues' the connecting surfaces together and essentially makes them one part. This is a simplified form on contact condition used to decrease simulation time. The downside is that it can result in higher stresses. Item 5 and 6 use a linear penalty method with constant friction coefficient. As the bearing is press-fit between the bedplate and the engine block, it will have residual stresses and strain. Item 7 and 8 use the standard ABAQUS surface interaction behaviour (described in [12]), with a constant friction coefficient.

3.1.4 Material formulation

The FE-model consists of four materials:

- Ductile iron (bearing supports)
- Aluminium (engine block and bedplate)
- Steel (bolts)
- GOE (cylinder lining)

All materials are modeled as isotropic and linear elastic. Their only differences lies in the respective Young's modulus and Poisson's ratios. Plastic behaviour has been disregarded to reduce simulation time. Material properties are taken from a state of 20°C and are constant as no thermal analysis is performed.

3.2 Topology optimization model

The following section describes the topology optimization model created and the results from it. The TOSCA-model was set up using ANSA and solved on the VCC computer cluster. The problem is delimited as follows:

Objective function
Minimizing the compliance of the entire model, which is the equivalent of maximizing its stiffness [9].

Design space
All elements making up the main bearing support insert, except for those whose faces are in contact with the bolts or facing towards the bearing.

Constraints
The material distribution has to be symmetric and the design has to decrease its volume by 3%. As the density of cast iron is roughly 2.6 times higher than the density of aluminum a volume decrease of 3% corresponds to a weight decrease of just over 1%.

Solver settings
The sensitivity based algorithm is used together with SIMP, see section 2.5 and 2.3 respectively. The process is forced to performed 40 iterations. The lower end of the range for the relative density is 0.75 as that corresponds to the density relation between aluminum and cast iron, see section 2.3.1.

3.3 Results

Figure 3.3 shows the optimization results from trial case 1. It can be seen that the elements of the design space contributing the least towards the FE-models compliance can be found on the bottom of the insert. The elements having a relative density of 0.75 corresponds to aluminum, and the elements having a relative density of 1 to cast iron. Due to the symmetry constraints the front view is enough to get a complete picture of the material distribution.

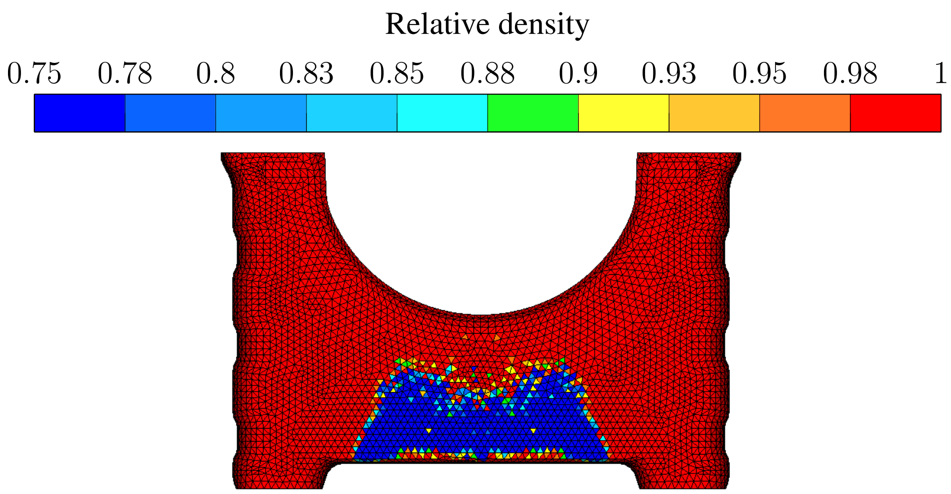


Figure 3.3: Optimization results trial case 1. An element having a relative density of 0.75 corresponds to aluminum, and an element having a relative density of 1 to cast iron.

3.3.1 Convergence

Figure 3.4 shows the normalized objective function convergence and the normalized volume constrain convergence. The objective function is normalized such that a value of -1 equals the initial compliance of the entire model. The optimized topology reaches almost the same compliance as the original, it is basically within computational the margin of error. The volume constraint is normalized such that a value of 1 equals a fulfilled constraint. These figures confirm the objective functions convergence and that the volume constraint is fulfilled. This together with a visual inspection of change in topology between iterations, which is difficult to illustrate in a report, leads to the conclusion of good overall convergence.

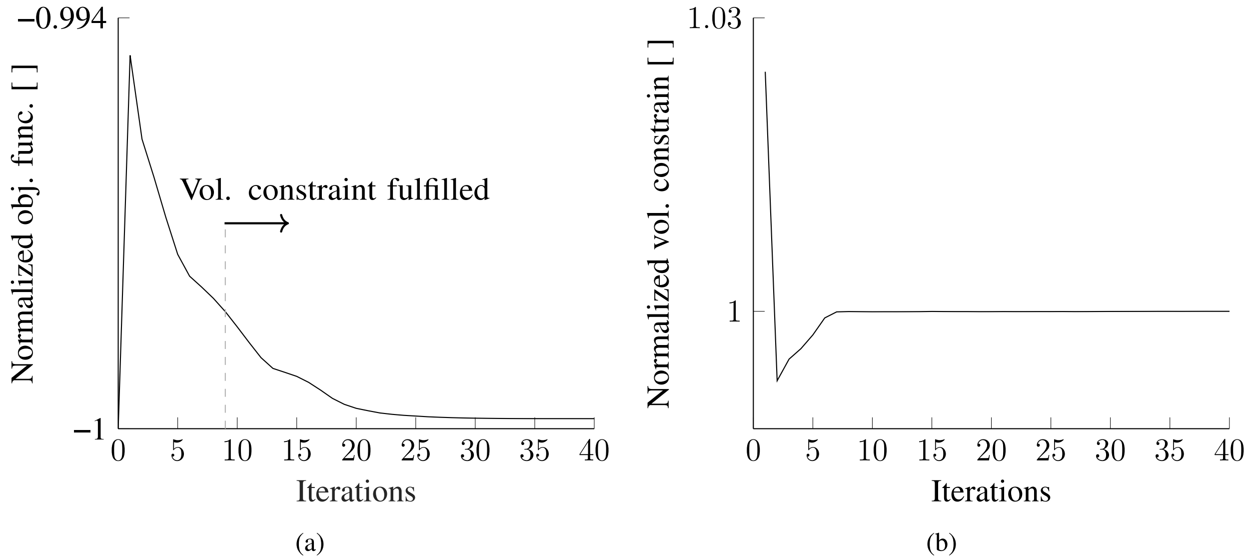


Figure 3.4: Objective function convergence and volume constraint.

3.4 Discussion trial case 1

One of the purposes of the first trial case was to get experience of working with topology optimization and FE-modelling. Much time was put into learning to operate ANSA and understanding ABAQUS and TOSCA.

From figure 3.4 it is clear the models compliance, i.e. the inverse of the stiffness, is sustained while the volume constraint is still fulfilled. This would indicate a potential redesign could be made based on the results. Naturally, there would be some question marks regarding the results. It is possible that it is overly simplistic to only include two cylinder halves, and that the whole engine block needs to be included. Furthermore, the load cases included in the FE-model does not describe all the loads experienced by the inserts and would therefore not be optimized for real life behaviour. But still, it can be seen as an indicator that the inserts could stand to lower its weight.

In the case of an objective function pertaining to maximizing stiffness while optimizing for the distribution of two materials one has to constrain the volume to decrease. If not, the entire design space would be occupied by the stiffer material. The constraint of the volume being forced to decrease by specifically 3% is basically arbitrary. The volume decrease serves as an indicator of which elements contribute the least towards the structures stiffness. It can be seen as a starting point to remove material if a redesign of the component were to be done.

4

Trial case 2: engine block ribs

This chapter presents the methodology and results of performing a topology optimization aiming to find the ideal material distribution on the exterior surface on the long sides of the engine block. After a brief introduction of the trial case, the FE- and optimization models are explained and the results are presented and discussed.

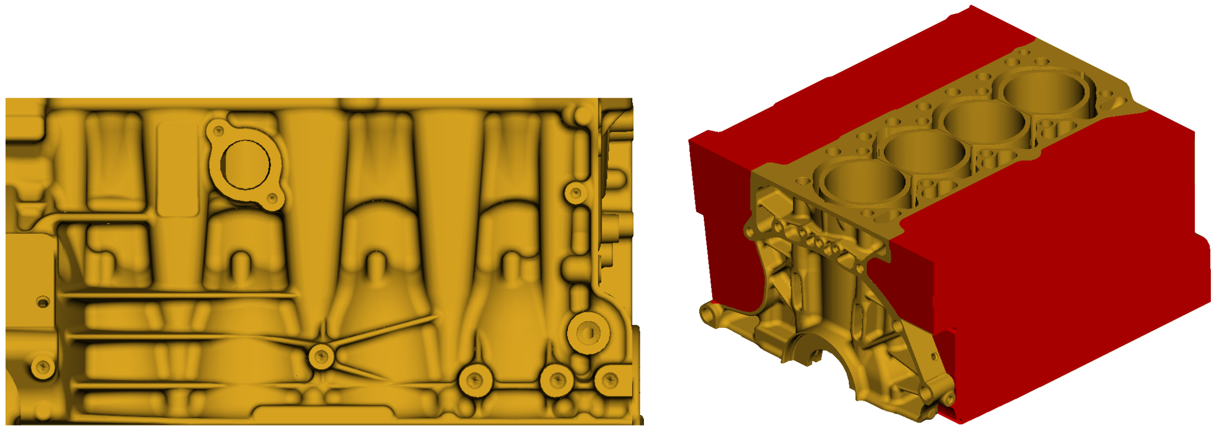
From a structural point of view, the purpose of the ribs on the exterior surface of the engine block is to increase its stiffness in order to minimize deformation during loading and to maximize the components eigenfrequencies. The placement of these is done by design engineers, without the use of optimization software. Most of them are lined up either strictly vertically or horizontally, which is not the type of design one would expect to come out of an optimization software. Basically, the approach in this trial case is to remove rib structures and to replace them with a design space and let the optimization software find the mathematically optimal design.

The engine block used in this trial case is the same as the one used in chapter 3. But once again the CAD-models depicted are not the actual geometries used in the optimization model, but rather they are from an older generation of VCC engines, for the same reason as in the previous chapter.

4.1 FE-model

Figure 4.1(a) depicts on of the long sides of the CAD-model representing the original engine block. For an isometric view of the engine block see figure 3.1 on page 11. The process to create the FE-model is to first delete the faces making up the rib structures and to replace them with faces aimed to create smooth and flat surfaces. This was only done on the long sides of the engine block, and the short sides were left as is. For complex or curved surfaces this can be a difficult and time consuming process. The overall removal and replacement of the ribs was successful, but in a few smaller surface regions previously curved surfaces are now plane. This is believed to have limited effect on the end results, as the eigenmodes were confirmed to be of similar nature when compared in the post-processor META. The engine block without the rib structures is not shown in the report as that work was done on an engine block whose design is still classified.

Figure 4.1(b) depicts the modified engine block without the rib structures, with the added design spaces on each side. The design spaces is created by making an empty box starting at the exterior surface and stretching some distance into the void, perpendicularly.



(a) (b)

Figure 4.1: Side view of the unmodified engine block, and the modified ribless engine block with the added design spaces.

The eigenfrequency used in the optimization model are solved for in ABAQUS using the Lanczos method. It is linearized around an unconstrained state. The only material used is aluminum and it is represented by an isotropic, linear elastic material model.

4.1.1 Mesh

The model consists of three volumes: the engine block itself and the two boxes making up the design space on both long sides of it. The mesh was created in ANSA and consists of roughly 4 million element all together. The element type is the same as in trial case 1, a first order (linear) tetrahedral element with 4 nodes and 1 integration point. The exterior surface of the engine block where the design spaces starts has a node spacing of 4 millimeters. Similarly as in trial case 1 the volumetric mesh has a growth rate of 20% inwards for non-design areas, as to decrease the total number of elements and save on computational time. This is believed to not affect the results in a noteworthy way.

4.2 Topology optimization model

The topology optimization model was created in ANSA and is set up in the following way:

Objective function
Maximize the first eigenfrequency of the engine block. As the Kreisselmaier-Steinhauser method is used, mode tracking is automatically applied.

Constraints
The design space is constrained to decrease its volume on each subdomain to a level that corresponds to the weight difference between the original and modified engine block for that side.

Design space
See figure 4.1.

Solver settings
The sensitivity based algorithm is used together with SIMP, see section 2.5 and 2.3 respectively. Prescribed to perform 100 optimization iterations.

4.3 Results

The resulting topology after the optimization with TOSCA can not be shown in the report due to the fact that the engine block design is still under development. However, the optimization model does not seem to favor the creation of rib-like structures. The majority of the elements were spread thinly and evenly over almost the entire side surfaces, with only small tendencies of creating rib-like structures. This is not in line with what was expected and hoped for.

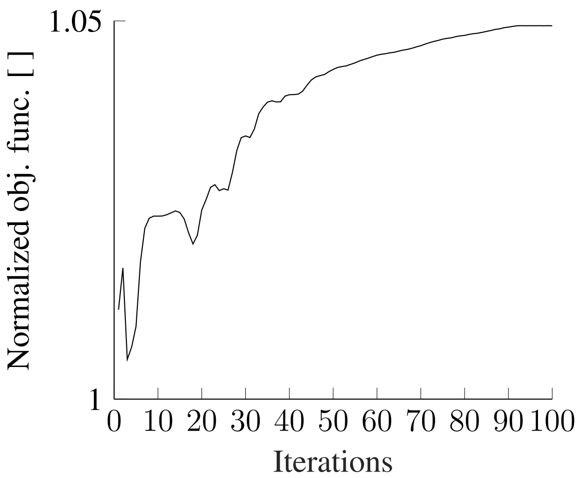
Table 4.1 shows the resulting eigenfrequency values from the modified ribless topology and the resulting optimized topology. Results are in comparison to the original topology's first eigenfrequency. It makes sense that the eigenfrequency for the modified engine block is lower than for the original block as weight is removed. The 5% increase of the first eigenfrequency for the optimized topology is obviously good, but as described earlier the material is not distributed in a way that resembles rib structures. This ultimately renders the results not usable for redesigning the rib structures.

	First eigenfrequency
Modified ribless topology	-2%
Optimized topology	+5%

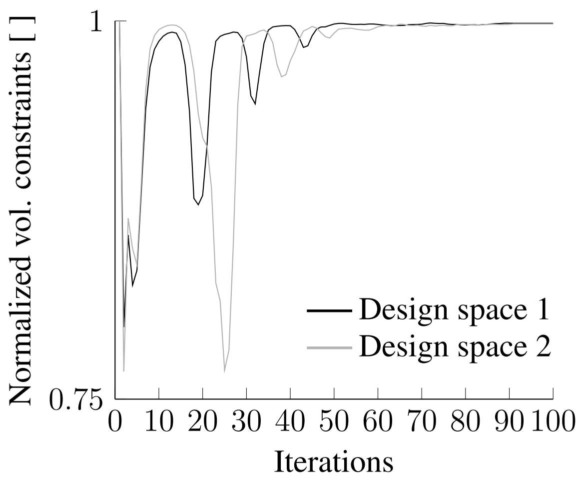
Table 4.1: Comparison of the eigenfrequencies for modified ribless engine block and the optimized engine block to the original design. Note that the optimized topology is heavier than the modified topology.

4.3.1 Convergence

Figure 4.2 displays the convergence for objective function and the volume constraints. The objective function is normalized against the eigenfrequency for the original engine block. The volume constraint is normalized such that a value of 1 equals a fulfilled constraint. Conclusion of good convergence is drawn based on the same argumentation made in section 3.3.1 for trial case 1. Compared to trial case 1 the design space geometry is more complex and the number of elements higher. Therefore it is no surprise that the number of iterations needed to reach convergence is greater.



(a)



(b)

Figure 4.2: Convergence of objective function and volume constraints, normalized against results from the engine block model with the added design spaces.

4.4 Discussion trial case 2

As stated previously, the optimization model does not seem to favor the creation of rib structures but rather it wants to smear out the elements over the entire surface. Many different settings has been tried to create a model more favorable towards the creation of rib structures, such as:

- Smaller and larger radius for the checkerboard filter
- Different material interpolation scheme settings
- Higher and lower mesh densities
- The including of manufacturing constraints

But regardless of setting tweaks, the results were of a similar nature of not creating rib-like structures. It would have been interesting to investigate the possibilities of letting the starting guess be the original design and let TOSCA perform changes based on that. This angle of attacking the problem was not initially considered, as the idea was to let TOSCA create something new and different from the original topology.

It is not overly simplistic or erroneous in itself to consider the eigenfrequencies as objective functions and to constrain the volume. But the eigenfrequency is solved for from a linearization around an unconstrained state, meaning the model is undeformed and free to move in any direction, which obviously is not a physically correct representation. The upside of such a simple model which only does modal analysis is that it is so fast to solve for that even a model consisting of millions of elements can perform upwards of 100 optimization iterations on VCC's computer cluster within a day's time. The current load case scenario means that any topology results that would be realized to a CAD-model and used in for example an engine block fatigue simulation model, would be subjected to loads and boundary conditions that it was not optimized against. It would have been interesting to create a more complete model in terms of load cases and components, but that gets complicated for multiple reasons:

- Complex to accurately describe all load cases experienced by the engine block
- Computationally heavy to include multiple components and many load cases

The load cases experienced by the engine block is a mix of forces originating from the internal combustion and the effect of having components hanging on the engine. The forces from the internal combustion acting on the engine block via the crank shaft are relatively easy to describe. But if a sufficient amount of those loads and the multiple parts making up the entire engine (block, bedplate, inserts, bolts, bearings, etc.) are included in the FE-model it quickly becomes computationally heavy to solve for. The problem with accounting for components hanging on the side of the engine block is not necessarily a computational issue as they could probably be described by point loads. However, it would be difficult to find accurate representations of magnitude and direction of such loads.

It is surprising that the objective function convergence in figure 4.2(a) is not monotonous. Especially considering that the optimization solver uses a gradient method [10]. This has been discussed with the TOSCA support team and it was assured that it is not a concern, as long as the convergence shows a monotonous trend.

5
Conclusions

This chapter presents the conclusions that can be drawn from the thesis work and outline suggestions of future work.

The results from trial case 1 could be used as a basis for a future redesign of the main bearing inserts. The compliance measurement indicates the stiffness would be the same even when the weight was reduced. But as discussed in section 3.4 there is some hesitation towards a realization as the model was quite simplified.

The optimization in trial case 2 was successful in the sense that it managed to increase the first eigenfrequency of the engine block, but unsuccessful in the sense that it did not yield the rib structures that were expected and hoped for. But even if the optimization model would have found rib structures that increased the first eigenfrequency, there would still be hesitation towards using the new topology in a design realization. The proposed design would be based on an optimization model not accurately describing all load cases and boundary conditions experienced by the engine block, as explained in section 4.4. The main conclusion that can be drawn from trial case 2 is that optimization with TOSCA works better on a more isolated component of smaller size, with a limited number of load cases that are clearly defined. This would give less need for simplifications and therefore allow for greater confidence in the FE-model and in turn the optimization results.

The work was set out with the purpose of investigating the possibilities and potential problems of implementing the use of structural optimization software in the product development phase within the department. TOSCA certainly can be used to generate useful new ideas and concepts, but there is no guarantee of return on investment in terms of useful results, as was seen in trial case 2. While the basic application of TOSCA is relatively easy to perform, there exists a myriad of different settings which influences the end results greatly. An engineer using TOSCA would have to have an understanding of these parameters and how they affect the results. That type of knowledge is not necessarily a subset of the traditional knowledge of a CAE engineer working primarily with FE analysis on solids. To avoid an overly time consuming learning curve any future endeavours into structural optimization within the team should to some degree be assisted by an optimization expert.

5.1 Future work

Three specific areas of interest for future work have been found and these are presented below.

5.1.1 Find a more suitable component to perform optimization on

The results from the trial case 2 does not support that further work is put into that specific type of applications. That does not necessarily rule out the overall applicability of optimization on other components relevant for the the department. Guidelines for choosing components to work on, based on the experience from this thesis, would be:

- Simple enough FE-model to be able to perform up to 100 optimization iterations within a reasonable time frame
- Large enough design space to leave room for the optimization solver to "do its magic"
- Comprehensive enough load cases and boundary conditions to give confidence in the results

No consideration has been put into what such a component might be.

5.1.2 Verify results from trial case 1

It would be interesting to realize the optimization results from trial case 1 to a CAD-model and to find a way of comparing the optimized design to the original.

5.1.3 Compare results from different optimization software

The decision of using TOSCA and ABAQUS rather than the optimization software OptiStruct (which is also available at VCC) was made without much research about pros and cons of OptiStruct. It would be interesting to compare the softwares and the results from them.

Bibliography

[1] ANSA. https://www.beta-cae.com/ansa.htm. Accessed: 2017-04-24.

[2] TOSCA. https://www.3ds.com/products-services/simulia/products/tosca/. Accessed: 2017-05-27.

[3] ABAQUS FEA. https://www.3ds.com/products-services/simulia/products/abaqus/. Accessed: 2017-04-24.

[4] SIMULIA 3DS. https://www.3ds.com/products-services/simulia. Accessed: 2017-04-04.

[5] META. https://www.beta-cae.com/meta.htm. Accessed: 2017-04-24.

[6] Peter W. Christensen and Anders Klarbring. *An Introduction to Structural Optimization*. Springer, 2008.

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

[8] Martin Philip Bendsoe and Ole Sigmund. *Topology Optimization: Theory, Methods, and Applications*. Springer, 2013.

[9] FE-DESIGN GmbH. *SIMULIA Tosca Structure Documentation 8.1*. SIMULIA, 2014.

[10] Krister Svanberg. The method of moving asymptotes - a new method for structural optimization. *Simulation*, 1987.

[11] Ole Sigmund. On the design of compliant mechanisms using topology optimization. *Mechanics of Structures and Machines*, 25(4):493–524, 1997.

[12] 3DS Simulia. *Abaqus Online Documentation: Analysis User's Guide*. Dassault Systemes AB, 2016.