Strategy Development of Structural Optimization in Design Processes

Ahmad Mansouri
David Norman

Examensarbete LIU-IEI-TEK-A--08/00480--SE
Institutionen för Ekonomisk och Industriell utveckling
Avdelningen för Teknisk Mekanik
Abstract

This thesis aims toward developing strategies in the area of structural optimization and to implement these strategies in design processes. At GM Powertrain Sweden where powertrains are designed and developed, two designs of a differential housing have been chosen for this thesis.

The main tasks have been to perform a topology optimization of a model early in a design process, and a shape optimization on a model late in a design process. In addition the shape optimization strategies have also been applied on a fork shifter. This thesis covers the theory of different optimization strategies in general. The optimization processes are explained in detail and the results from the structural optimization of the differential housings as well as the fork shifter are shown and evaluated.

The evaluation of the thesis provides enough arguments to suggest an implementation of the optimization strategies in design processes at GM Powertrain. A Structural Optimization group has great potential of closing the gap between structural designers and structural analysis engineers which in long terms mean that better structures can be developed in less time. To be competitive in the automotive industry these are two of the most important factors for being successful.

Keywords: Topology Optimization, Shape Optimization, Solid Mechanics, Differential, Fork Shifter
Sammanfattning

Detta examensarbete är utfört i syfte att utveckla strategier då strukturoptimeringar genomförs för att senare implementera dessa strategier i designprocesser. På GM Powertrain Sweden där man huvudsakligen designar och utvecklar drivlinor, har två designer av ett differentialhus samt en växelförare utvalts för projektet.

Huvuddelen består av en topologioptimering av en modell tidigt i en designprocess, och en formoptimering av en modell senare i en designprocess. Utöver detta har strategierna för formoptimering implementerats på en växelförare. Detta projekt behandlar den generella teorin bakom olika strukturoptimeringar. Optimeringsprocessen är utförligt beskriven och resultaten från strukturoptimeringarna av differentialhusen samt växelföraren visas och utvärderas.

Utvärderingen av detta examensarbete visar på tillräckligt starka argument för att implementera optimeringsstrategierna i designprocesser inom GM Powertrain. En strukturoptimeringsgrupp har stor potential att minska gapet mellan strukturdesigners och strukturanalysingenjörer vilket på längre sikt medför att bättre strukturer kan utvecklas på kortare tid. För att vara konkurrenskraftig inom bilindustrin är dessa nyckelfaktorer för att lyckas.

**Nyckelord:** Topologioptimering, Formoptimering, Hållfasthetslära, Differential, Växelförare
Acknowledgements

We would like to thank the people that provided us with help along our way during this thesis. We have had a good time from the very first day at the office in Trollhättan.

Anders Klarbring
Alejandro R Martinez
Johanna K Olsson
Martin Schagerlind
The Structure Analysis Group

Trollhättan Mars 2009

Ahmad Mansouri and David Norman
# Table of Contents

Table of Contents .................................................................................................................. 5

1 Introduction ......................................................................................................................... 10

  1.1 Background ....................................................................................................................... 10

  1.2 Purpose ............................................................................................................................. 10

  1.3 Limitations ......................................................................................................................... 10

  1.4 Software used .................................................................................................................... 10

  1.5 Methodology ..................................................................................................................... 11

    1.5.1 Introductory part ......................................................................................................... 11

    1.5.2 Executing part ............................................................................................................. 11

    1.5.3 Compilation part ......................................................................................................... 11

  1.6 General Motors Corporation (GM) ................................................................................... 12

  1.7 Overview of the Powertrain .............................................................................................. 12

    1.7.1 Introduction to the differential .................................................................................. 13

    1.7.3 Overview of a differential ......................................................................................... 14

    1.7.4 Function & mechanics of a differential .................................................................... 16

  1.8 Demands and tests of powertrains ................................................................................... 17

2 Introduction to structural optimization .................................................................................. 18

  2.1 Formulation of an optimization problem .......................................................................... 18

  2.2 Different types of structural optimization ....................................................................... 19

    2.2.1 Size optimization: .................................................................................................... 19

    2.2.2 Shape optimization: .................................................................................................. 19

    2.2.3 Topology optimization: ............................................................................................. 20

  2.3 Theory of penalization technique ...................................................................................... 20

3 Creating a FE-model ............................................................................................................. 21
3.1 Meshing
3.1.1 Meshing for Topology Optimization purposes
3.1.2 Meshing for Shape Optimization purposes
3.2 Assigning material properties to FE-model
3.3 Assigning Load and Constraints to FE-model
3.3.1 Rigid Body Elements
3.3.2 Assigning rigid body elements
3.3.3 Assigning Constraints to FE-model
3.4 Loadings
3.5 Creating load cases
3.6 FE-analysis
4 Topology optimization of the Differential housing
4.1 Introduction to Topology Optimization of the differential housing
4.2 FE-model of the differential housing for Topology Optimization
4.2.1 Description of whole model
4.2.2 Description of parts of the differential housing
4.2.3 Contacts
4.2.4 Applying gap elements
4.2.5 Models for consideration
4.3 FE-analysis
4.3.1 FE-analyses of model with design space
4.3.2 FE-analyses of model without design space
4.3.3 Evaluation of FE-analysis
4.4 Topology optimization in OptiStruct
4.5 Controlling the optimization
4.5.1 Setting up the optimization ......................................................................................... 43

4.5.2 Objective function Minimizing Weighted Compliance ................................................. 43

4.5.3 Constraints on responses ............................................................................................... 44

4.5.4 Constraints on feasible designs ..................................................................................... 44

4.5.5 Constraints on quality of optimization ........................................................................... 47

4.5.6 Penalization .................................................................................................................. 48

4.6 Evaluation of Topology optimization .................................................................................. 50

4.6.1 Analyzing stress levels of different designs ................................................................. 50

4.6.2 Influence of Calculation times ....................................................................................... 50

4.6.3 Analyzing the Weight reduction ................................................................................... 50

4.6.4 Mesh size influence ...................................................................................................... 51

4.7 Results .............................................................................................................................. 52

4.7.1 Chosen design proposal ............................................................................................... 53

4.7.2 Approximate FE-analysis of stress levels of final design ............................................. 54

4.7.3 Weight comparison of different designs ....................................................................... 55

4.7.4 Approximate FE-analysis of displacements of final design ......................................... 56

4.7.5 Comparison of stresses with initial FE-analysis ......................................................... 57

4.7.6 Redesigning of Final design proposal .......................................................................... 58

4.8 Conclusions ...................................................................................................................... 59

5 Shape optimization ............................................................................................................. 61

5.1 Introduction to Shape optimization .................................................................................. 61

5.2 Shape optimizing in Optistruct ....................................................................................... 62

5.2.1 Free Shape ................................................................................................................... 62

5.2.2 Hypermorph ................................................................................................................ 63

5.2.3 Domains and Handles ............................................................................................... 64
5.3 FE-analysis of the differential housing ................................................................. 66
5.4 The affect of the size of the design space when shape optimizing ...................... 67
5.5 The influence of mesh size .................................................................................. 68
5.6 Optimizing by using Hypermorph ..................................................................... 69
5.7 Results ................................................................................................................. 70
  5.7.1 The affect of the size of the design space when shape optimizing ..................... 70
  5.7.2 Influence of mesh size ..................................................................................... 72
  5.7.3 Comparison between the hypermorph and the free shape functions ................ 74
5.8 Analysis & Conclusions ....................................................................................... 77
  5.8.1 Evaluation of the size of the design area ......................................................... 77
  5.8.2 Evaluation of the influence of the mesh size ................................................... 78
  5.8.3 Comparison between freeshape and hypermorph ......................................... 79
5.9 Shape optimization of a fork shifter .................................................................... 80
  5.9.1 Setting up the optimization ............................................................................ 81
  5.9.2 Evaluation of the size of the design area ......................................................... 82
  5.9.3 Results ............................................................................................................ 84
  5.9.4 Conclusion ..................................................................................................... 84
6 Summary ............................................................................................................... 85
References ................................................................................................................. 86
Appendix 1 ................................................................................................................ 87
Appendix 2 ................................................................................................................ 89
Appendix 3 ................................................................................................................ 90
Appendix 4 ................................................................................................................ 92
Appendix 5 ................................................................................................................ 94
Appendix 6 ................................................................................................................ 95
1 Introduction
The main subject of this thesis is to evaluate the capability of OptiStruct\textsuperscript{1} when performing structural optimizations. It is done as a master thesis in Mechanical Engineering in the area of Technical Mechanics.

1.1 Background
To design a detail in a gearbox, experience has been the best tool for the designers. When the design is made it is sent for analysis where possible flaws are found on the design and these are pointed out to the designers. A refinement of the design is made and sent again for analysis and this will go on until a satisfying design is found. Every loop is time demanding and of course costly. With today's technology regarding computer software it is possible to create an optimum design with few flaws that the designers can easily fix.

1.2 Purpose
The main purpose of this thesis is to develop structural optimization strategies in design processes. OptiStruct is known to be a good tool when performing structural optimizations, but is mainly used for a few minor procedures at \textit{GM Powertrain} today. It is of interest to investigate if OptiStruct can deliver satisfying results and if a strategy can be developed so that it can be implemented in future design processes.

The aim is to develop strategies so that structural optimization can be implemented, and to suggest when and how they should be used. The main goal is to lower the amount of loops between the design engineer and the structure analysis engineer.

1.3 Limitations
If simplifications on the model and load cases had not been made, the accessible computer power would be insufficient to perform the optimizations. Also to keep the calculation times reasonably low simplifications have been made on the models used in this thesis.

1.4 Software used
GM has a lot of licenses for different software used as tools for analyses and development. The software used in this thesis are:

- Ultra Edit (IDM Computer Solutions, Inc.): Text editing software used for editing input data and reading output data.
- SimLab 6.0 (SimLab corp. USA): Meshing software used for creating a first mesh of models.
- HyperMesh 8.0 (Altair): Preprocessing software used for creating meshes, applying loads and constraints and for setting up the optimization. \textit{Optistruct} is set as user preference which makes it compatible with the optimization solver \textit{Optistruct}.

\textsuperscript{1} Processing software used for solving structural optimization problems. (See section 1.4 Software used)
• Optistruct 8.0 (Altair): Processing software used as a solver for the optimization problem stated in Hypermesh.

• HyperView 8.0 (Altair): Post processing software used for visualization and analysis of results from FE-analyses or optimizations.

1.5 Methodology
To easily understand the methodology of this thesis it can be divided into three parts. The first part can be called the introductory part. The second part can be called the executing part. The third part can be called the compilation part.

1.5.1 Introductory part
The introductory part of this thesis consists of the following tasks:

• Defining sub- and main goals of thesis.
• Setting up a time plan.
• Gathering knowledge regarding the powertrain.
• Get acquainted with the software available and learning how they can be applied.

1.5.2 Executing part
The executing part of this thesis consists of the following tasks:

• Receiving geometry of designated parts.
• Creating FE-models
• Perform analyzes and optimizations.

1.5.3 Compilation part
The compilation part of this thesis consists of the following tasks:

• Compile the results from the executing part.
• Draw conclusions.
• Suggest future actions and implementations of structural optimization in design processes at GM Powertrain.

2 In this thesis the powertrain mainly refers to the transmission, however a powertrain consist of all the components that generate and transfers torque in a vehicle. (See section 1.7 Introduction to Powertrain)
1.6 General Motors Corporation (GM)

GM was founded in Flint, Michigan, USA, September 16, 1908. The first brand produced was Buick and the same year their second brand Oldsmobile began production. Today GM is the owner of 13 car brands such as Buick, Cadillac, Chevrolet, Daewoo, GMC, Holden, HUMMER, Opel, Pontiac, SAAB, Saturn, Vauxhall and Wuling. For 77 years (up to 2008) GM has been the car company that has sold most cars. And as of January 2008 GM is the world’s largest automaker, measured by global industry sales. As for today GM has grown to have around 284 000 employees in 35 different countries.

GM in Sweden started year 1927 in Stockholm with an assembly plant where Opel and US cars where produced. That plant was closed in 1956. SAAB started its production in 1949 in Trollhättan and it was not until 1989 that GM bought 50 percent of SAAB. 11 years later GM bought the rest of the stocks and as of then fully owns SAAB Automobile AB.

GM Powertrain which is responsible for the powertrain R & D and production has about 50 000 employees in 17 different countries. GM Powertrain Sweden has about 600 employees located in Trollhättan, where areas such as hybrid, engine and transmission development among others are a part of the GM R & D family.

1.7 Overview of the Powertrain

When a car is driven, power is generated in the engine. The torque is transferred to the transmission with the help of the clutch\textsuperscript{3}. The power is transferred through the front differential onto the front wheel shafts, thus transferring torque from the engine to the front wheels and onto the ground. This kind of powertrain is called a FWD\textsuperscript{4} and this is the most commonly used powertrains in cars today. There are several kinds of powertrains and another common one in GM cars is the AWD\textsuperscript{5} powertrain. This powertrain distributes the torque to all four wheels. The transferring of the torque to the rear wheels goes through the transfer case\textsuperscript{6} and rear drive shaft. See figure 1.7.1.

This thesis focuses on the part that transfers torque to the wheel shafts, namely the differential.

\begin{footnotes}
\item[3] Here are cars with single clutch systems considered.
\item[4] Front Wheel Drive
\item[5] All Wheel Drive
\item[6] Also called PTU, Power Take-off Unit
\end{footnotes}
1.7.1 Introduction to the differential

It is common among kids to build box cars and race down the streets. The wheels are skidding and screaming in curves when they are trying to finish first. Typically for these cars is that the front wheels are locked to each other by a single solid shaft. The consequence of this is that the front wheels will always rotate with the same angular velocity. This causes the skidding and screaming of the box car. This problem can be solved by dividing the solid wheel shaft into two shafts connected to a differential.

To be able to have different angular velocities of the left and right front wheel shafts a front differential is mounted in the gearbox which transfers the torque equally to the two front wheels. There are different kinds of differentials used in cars. In this thesis an open differential is considered.
1.7.3 Overview of a differential

The differential is positioned in the gearbox so that torque can be transferred to the wheel shafts. See figure 1.7.3.1.

Figure 1.7.3.1 Overview of a gearbox in the GM Powertrain product family.

The differential is a complex part which consists of several components. To fully understand the function and mechanics of the differential, these components needs to be explained. The different components are as following (also see figure 1.7.3.2):

- Differential Housing: Contains interior gears and attaches to the ring gear and the bearings.
- Ring gear: Connecting the final drive\(^7\) and the differential housing.
- Side gears: Also called Bevel gears, connects to the wheel shafts
- Pinion gears: Interior gears which are connected to the side gears and the pinion.

\(^7\) The final drive refers to the last gear that transfers torque to the differential.
- Pinion: Connects the pinion gears
- Bearings: Connected to the differential housing allowing rotation.

Figure 1.7.3.2 Example of a differential within the GM Powertrain product family.
## 1.7.4 Function & mechanics of a differential

When driving a car, the differential allows the two front wheels to rotate with different angular velocities. The differential is affected by the torque transferred from the final drive and this torque is distributed equally to the front wheel shafts. The upper limit of the torque that can be provided to a differential can be explained in two cases.

1. **Unlimited traction**, which means that the tires will not slip against the ground and that the engine will determine the maximum ingoing torque.

2. **Limited traction**, which means that at least one tire will slip when the torque exceeds a threshold value. As explained earlier the torque in the differential housing is distributed equally to the front wheels and therefore if one tire slips it will determine the maximum torque on both shafts.

When driving straight forward the two front wheel shafts have the same angular velocity. Turning, or heavily accelerating (spinning wheels), causes a difference in angular velocity of the two front wheel shafts made possible by the differential.

For example let us say that a car makes a right turn. The right wheel will have a smaller turning radius than the left wheel in the curve. This means that the left wheel must make more revolutions than the right wheel. To compensate the difference in velocity between the left and right wheel shaft, the left side gear in the differential housing will start rotating in the same direction as the housing, thus it will add angular velocity to the left wheel shaft. On the other hand whilst the left side gear adds velocity to the left wheel shaft, the right side gear will start rotating in the opposite direction of the right wheel shaft, thus reducing the angular velocity on the right wheel shaft.

The following explanation refers to figure 1.7.3.2. Rotations refer to rotation, relative the differential housing, around the z-axis following the right hand rule if else is not specified.

When the left side gear experience a rotation relative the differential housing, the right side gear will rotate in the opposite direction due to the function of the pinion gears. This will result in a reduction of angular velocity for one wheel shaft and adding of angular velocity for the other wheel shaft. A conclusion that can be made is that when one side gear rotates and gains angular velocity the other side gear will experience the same magnitude in reduction of angular velocity. The total velocities are explained in following equations:

---

8 This example implies a FWD powertrain.
\[ \omega_{LA} = \omega_{DIFF} \pm \omega_{LSG} \]
\[ \omega_{RA} = \omega_{DIFF} \pm \omega_{RSG} \]

When infinite traction and no curve: \( \omega_{LA} = \omega_{RA} = \omega_{DIFF} \), thus \( \omega_{LSG} = \omega_{RSG} = 0 \frac{\text{rad}}{s} \)

When \( + \omega_{LSG} \) then \( - \omega_{RSG} \) and \( + \omega_{LSG} = [-\omega_{RSG}] \)

When \( + \omega_{RSG} \) then \( - \omega_{LSG} \) and \( + \omega_{RSG} = [-\omega_{LSG}] \)

\( \omega_{DIFF} = \) Angular velocity of the differential [rad/s]

\( \omega_{LA} = \) Angular velocity of the left axle shaft [rad/s]

\( \omega_{LSG} = \) Angular velocity of the left side gear [rad/s]

\( \omega_{RA} = \) Angular velocity of the right axle shaft [rad/s]

\( \omega_{RSG} = \) Angular velocity of the right side gear [rad/s]

### 1.8 Demands and tests of powertrains

During development of powertrains, rigorous tests are performed to ensure quality and durability of all parts within the GM Powertrains product family. These tests yield valuable input data for future analyses and simulations in the research and development team. There are a lot of standard tests, however this thesis considers an abuse load testing. These are the worst case scenarios and are used when testing the powertrain to the extreme. During a car's life cycle these abuse loads are not likely to occur. The powertrain must be able to withstand these loadings without failure.

The tests are divided into two groups which are known as Coast and Drive, and a clutch release of 10 milliseconds is used. The quick engagement of the clutch will cause high torque peaks in the gear box. There are four different abuse load modes as following:

**Drive**

1. From stationary position the engine speed is increased up to a very high rpm\(^9\). The transmission is shifted into first gear and then the clutch is engaged.

2. From stationary position the engine speed is increased up to a very high rpm. The transmission is shifted into reverse gear and then the clutch is engaged.

**Coast**

3. When a specified velocity is achieved at first gear, the clutch is disengaged and then engaged whilst the rpm of the gearbox is still as high as before disengaging the clutch.

4. When a specified velocity is achieved at reverse gear, the clutch is disengaged and then engaged whilst the rpm of the gearbox is still as high as before disengaging the clutch.

---

\(^9\) Revolutions per minute.
2 Introduction to structural optimization

The idea of optimization is finding the best solution to the problem stated. First the definition of “best” must be defined and it varies from case to case and it is up to the optimizer to decide this definition. It could be money, available material, design space etc. that is the main restraining factor in the design process. So “best” should be interpreted as the optimum solution under the existing circumstances. The solution depends on the objective function. What is the purpose of the optimization? This could for instance be minimizing mass, maximizing stiffness or minimizing peak stresses.

When working with an optimization problem, one has to remember that after the optimization process it is important to check all of the demands on the detail that is going to be designed. Because in real life all demands and constraints cannot be implemented in the optimization process and must therefore be analyzed separately on the new design.

2.1 Formulation of an optimization problem

An optimization problem always consists of an objective function, design variables and state variables. The problem is often subjected to various constraints in order to receive good results. Often the formulation is set up so that the structural optimization problem is to minimize the objective function with respect to the design and state variables.

Minimize Objective function \( f \) with respect to design and state variables \( x \) and \( y \) respectively

\[
\begin{align*}
\text{Subject to: } & \text{Design constraints on } x \\
\text{Subject to: } & \text{Behavioral constraints on } y \\
\text{Subject to: } & \text{Equilibrium constraints}
\end{align*}
\]

Objective function \( f \):

- Function measuring the quality of the design, i.e. the quality of the optimization.

Design variables \( x \):

- Describes the design of the structure. This can include geometry, material and other variables that may be allowed to change during the optimization.

State variable \( y \):

- Represent the response of the structure for the given design. Any value on the design variable \( x \) will yield a response \( y \) on the structure. The response can be for instance displacement or stresses.

The design and state variables are often subjected to constraints. These are known as design constraints on design variables \( x \) and behavioral constraints on the state variables \( y \). The structure is also subjected to equilibrium constraints.
The equilibrium constraint looks like:

\[ K(x)u = F(x) \]

where \( K(x) \) is the stiffness matrix as a function of the design variable \( x \), \( u \) represents the displacements and \( F(x) \) is a force vector which can also depend on the design variable \( x \). The displacement can be written as \( u(x) = K(x)^{-1}F(x) \) if \( K(x) \) is invertible for all given values of the design variables \( x \). This is valid when the state problem uniquely defines \( u \) for any given value on the design variable \( x \). Now it is possible to leave out the equilibrium equation and also substitute this function \( u(x) \) for the state variable. We can assume that all state and design constraints can be written on the form

\[ g(x, u(x)) \leq 0 \]

The so called nested formulation can then be written as:

\[
\begin{align*}
\text{Min } f(x, u(x)) \\
\text{Subject to } g(x, u(x)) \leq 0
\end{align*}
\]

2.2 Different types of structural optimization
There are three types of structural optimization problems. These are Size, Shape and Topology optimization:

2.2.1 Size optimization:
The design variable represents some sort of structural thickness. For instance the thickness of a sheet or a radius of a rod can be varied.

2.2.2 Shape optimization:
The design variable represents some part of the boundary shape of the structure. This boundary may form into other shapes but no new boundaries are created. While sizing could optimize the thickness of a sheet, the shape optimization can also let the thickness vary along the boundary.

Shape optimization is a tool of great use when aiming to improve the structure locally. From analyses local areas with high stresses can be recognized. These areas can be shape optimized in order to lower the stress level.

The local area is defined as a design area (design space) which means that it is allowed to change its shape during optimization. The areas that are not included in the shape optimization are called non-design space.
2.2.3 Topology optimization:
As the name implies the topology is allowed to change during the optimization. This means that new boundaries can be formed. In the 3D-case, the design variable now represents a density variable which can take values between an upper and lower bound. If the density of an element reaches zero it becomes a void. Often this density variable is normalized so that the variable has a lower bound of 0 and upper bound of 1. 0 represents a void whereas 1 represents a fully dense solid element.

When the density of an element is between the densities of a void and solid it is called semi dense. Semi dense elements are not wanted in the design due to impossibilities in producing such structures. A cure against presence of semi dense elements is penalizing the values densities are allowed to take. A fully discrete design will only consist of elements with relative densities of either 0 or 1. However it is hard to find a fully discrete solution when performing the optimizations, so typically a large amount of semi-dense elements exist in the final solution.

Topology optimization is most commonly used when trying to find new designs not intuitively clear to the designer. It is therefore a good tool for making design proposals early in the design process. The elements for which density is allowed to vary are called design elements. For these elements the stiffness is linearly dependent on the density of the element, if no penalization\textsuperscript{10} is present.

2.3 Theory of penalization technique
Penalizing intermediate densities will provoke the final design to be represented by element densities of either 0 or 1. The penalization technique used for the density approach is based on the so called “Power law representation of elasticity properties”. It can be expressed for any solid 2D or 3D element as follows:

\[ K(\rho) = \rho^p K \]

Where \( K \) is the penalized stiffness matrix, \( K \) is the real stiffness matrix, \( \rho \) is the element density and \( p \) is the penalization factor which is always greater than one (or 1 for no penalization). The penalization works in a way that element with low density will have a dramatically lower stiffness contribution to the structure than fully dense element. This will provoke a more discrete solution due to the fact that the optimization tends to place fully dense elements where most necessary and have zero density elsewhere. In other words it is “uneconomical” in a mass sense to have elements with semi densities if they do not at the same time contribute to the stiffness of the structure.

\textsuperscript{10} See chapter 2.3 Theory of penalization technique.
3 Creating a FE-model

All components in the powertrain are stored as CAD models. These are computer based representations which are used when creating FE-models. The geometry consists of surfaces enclosing a volume and a surface mesh can be generated following this geometry. The volume mesh can then be generated by filling up this volume with 3D elements in an automated or manual way, depending on mesh preferences.

3.1 Meshing

The first step is to import the geometry to the user preferred software for meshing\(^\text{11}\). A good start is to define the element type and the size of the elements. If the mesh consists of large elements the analysis will be rough and results may not be accurate enough. One should try to find an element size that gives short calculation time, without losing accuracy.

There are a lot of factors\(^\text{12}\) that influence the quality of the mesh. Therefore it is important to perform an element check, where the quality for each element is measured. Some elements may not meet the quality demands specified by the user and remedial actions must be taken. However the information given by just looking at the mesh should not be underestimated. A rule of thumb is that if the mesh looks good it often is.

When just performing an analysis the demands on the mesh is usually the same from case to case. However the demands on the mesh differs for different types of optimizations.

3.1.1 Meshing for Topology Optimization purposes

Usually when analyzing larger structures a volume mesh is created with smaller elements near the surface and larger elements in the midst of the model. This is sufficient for most analyses.

However, when the aim is to perform a topology optimization, it is important to have a somewhat constant element size, beginning from the surface and going throughout the entire designable part of the model. This is called an interpolated mesh.

The structural members that can be created when topology optimizing will be dependent on the mesh size of all elements in the designable region. It is therefore important to use a fine mesh throughout the design space. A fine mesh provides more accurate calculations of responses and further more it will yield better optimization results. Also more control of the design of new structural members are given.

\(^\text{11}\) The software used in this thesis for meshing purposes are SimLab and Hypermesh, see 1.4 Software used

\(^\text{12}\) These factors can be aspect ratio, Min/max angle, element size, skewness, jacobian among others.
3.1.2 Meshing for Shape Optimization purposes

When the aim is to perform a shape optimization, it is important to have a fine element size on the surface and a couple of layers deep. The reason for wanting a finer mesh locally is to obtain more accurate results, in order to obtain smoother finishing surfaces and more easy interpreted results.

When the shape changes the nodes of the design elements will move, thus deforming the elements. This often leads to a deteriorated element quality, which leads to poor results when the elements get highly distorted. To prevent highly distorted elements, the amount of movement of the nodes can be distributed to a number of layers, moving each layer according to a specified function. Instead of moving on layer of elements a certain distance, numerous layers of elements are moved the same amount of distance, keeping element quality on a sufficient level. When the element quality violates the user defined demands, the optimization will end, with the last design as final design.

Figure 3.1.2.1 Example of a Triangular 2D mesh

Figure 3.1.2.2 Example of a Triangular 3D mesh
3.2 Assigning material properties to FE-model

The materials that are used in this thesis are cast iron and steel. Cast iron is used for the differential housing and steel is used for the ring gear. These material properties are assigned to the elements created when meshing the FE-model.

In this thesis small displacement theory is used and linear elastic material properties can therefore be assumed. This simplification ignores the fact that in reality the material will plastically deform when subjected to stresses above its yield limit.

The material properties are listed in Table 3.2\textsuperscript{13}

<table>
<thead>
<tr>
<th>Material</th>
<th>E [GPa]</th>
<th>$\nu$</th>
<th>$\rho$ [kg/mm$^3$]</th>
<th>$\sigma_{\text{yield}}$ [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cast Iron</td>
<td>190</td>
<td>0,27</td>
<td>$7.8 \times 10^{-9}$</td>
<td>800</td>
</tr>
<tr>
<td>Steel</td>
<td>210</td>
<td>0,3</td>
<td>$7.9 \times 10^{-9}$</td>
<td>980</td>
</tr>
</tbody>
</table>

Table 3.2 Material properties

\textsuperscript{13} Values are provided by GM Powertrain material lab.
3.3 Assigning Load and Constraints to FE-model

In order to simulate realistic behavior, great care is needed when applying loads and boundary constraints on the model. All aspects that affect the model should be taken into consideration.

3.3.1 Rigid Body Elements

Two kinds of rigid body elements are used in this thesis to simulate various behaviors. A description of the rigid body elements is as follows:

*Rigid link:* All the nodes connected to a mother node are restrained to move relative to this node in the degrees of freedom specified. This can, for instance, simulate the presence of bolts in bolt holes or the PTU shaft in the AWD connection.

*Distribution coupling:* The load applied to the mother node is distributed equally\(^\text{14}\) to all dependent nodes. The load can for instance be a force or a torque.

---

\(^{14}\) Weighting factors of the distribution to each dependent node can be user specified. However in this thesis only equal distribution is needed.
3.3.2 Assigning rigid body elements

These rigid elements simulate the contact with the bearings.

These rigid elements simulate the contact to the PTU shaft.

These rigid elements simulate the contact to the pinion.

These rigid elements simulate the bolts, which connects the differential housing to the ring gear.

This distributed coupling simulates the contact to the PTU shaft.

Figure 3.3.2.1
These distribution couplings represents the contact between the differential housing and the side gears.

These distribution couplings represents the contact between the differential housing and the pinion gears.

Figure 3.3.2.2
### 3.3.3 Assigning Constraints to FE-model

The movement of the differential is directly coupled to the rotation of the wheel axes, i.e. the velocity of the car. In other words, the faster the car moves the faster the differential spins around its z-axis\(^ {15} \). In this thesis, dynamic effects will not be taken into account when analyzing. In a static FE simulation, rigid body motions must be prevented by applying constraints on the degrees of freedom for some parts of the boundary.

In a static case the FE-model has to be in a state of equilibrium. At the contact between the wheel shafts and pinion to the differential housing, some degrees of freedom are removed. At these contacts, reaction forces and moments will arise when loads are applied to the differential. In the static case the loads applied will equal the reaction forces and moments in opposite direction.

There are six constraints of the degrees of freedom:
1. Locking any displacement along the x-axis.
2. Locking any displacement along the y-axis.
3. Locking any displacement along the z-axis.
4. Locking any rotation around the x-axis.
5. Locking any rotation around the y-axis.
6. Locking any rotation around the z-axis.

For the FE-model of the differential, following constraints of the degrees of freedom are specified. Note that the constraints will differ depending on the direction of rotation of the differential:

![Figure 3.3.3.1](image)

This constraint simulates the locking of the left wheel shaft and the differential housing. The constraints on the left side of the differential are defined as following (see figure 3.3.3.1):

- **Forward**: DOFs 1,2,3,6
- **Reverse**: DOFs 1,2

\(^{15} \text{See figure 1.7.3.2} \)
This constraint on the right side of the differential is defined as following (see figure 3.3.2):
Forward: DOFs 1,2
Reverse: DOFs 1,2,3,6

The constraints simulate the locking between the pinion and the differential housing. Both of the constraints are locked in following way (see figure 3.3.3):
Forward: DOFs 6
Reverse: DOFs 6
3.4 Loadings

From abuse load tests\textsuperscript{16} a highest torque value transferred through the differential was measured to 9628 Nm\textsuperscript{17}. Discussions with the author led to the conclusion of using 9000 Nm as the worst case torque. Measured values from GM Powertrain show that 75 % of the ingoing torque is distributed equally to the right- and left side gear. The remaining 25 % is distributed to the PTU (Power Take of Unit), thus this is an all wheel drive differential. Figure 3.3.2.1 shows where the torque that is distributed to the PTU is applied. The torque that is distributed to the rear differential is calculated in following way:

When transferring torque from the final drive to the ring gear, reaction forces will arise in the contact point. By using the measured torque value and knowing the radius of the ring gear, the reaction force was calculated, as following:

\[
F = \frac{M}{r} = \frac{0.75 \cdot 9000}{0.1153} = 58\,040\,N
\]

Figure 3.4.1 represents a ring gear where the blue part represents a tooth. The \(\alpha\) angel represents the pressure angle of 21.5 degrees whilst the \(\beta\) angle represents the helical angle of 27 degrees. Values given from GM Powertrain, gear division.

\[
F_T = \cos(27) \cdot \cos(21.5) \cdot 58\,040 \approx 48.1\,kN
\]
\[
F_Z = \sin(27) \cdot \cos(21.5) \cdot 58\,040 \approx 24.5\,kN
\]
\[
F_R = \sin(21.5) \cdot 58\,040 \approx 21.3\,kN
\]
\[
F = \sqrt{F_T^2 + F_Z^2 + F_R^2} \approx 58\,kN
\]
\[
M_{\text{rear}} = 0.25 \cdot 9000 = 2250\text{Nm}
\]

\textsuperscript{16} Explained in section 1.8 Demands and tests of powertrains.

\textsuperscript{17} Value obtained from test report xxx, which is GM confidential.
When the differential is loaded the pinion and side gears will cause a pressure to the inside wall of the differential housing. See figure 3.3.2.2.

A test has been made where the pressure values from a smaller gearbox\(^{18}\) have been measured. A simplification has been made in the way that these pressure values are linearly dependent on the maximum engine torque. To obtain exact pressure values, a hardware test of the designated gearbox would be required. However the simplification that has been made is considered adequate enough for this purpose.

The measured area and pressure values\(^{19}\) from the testing of the smaller gearbox, with a maximum engine torque of 170 Nm, were obtained for several different scenarios. The worst case scenario was during a coast test for the pinion gears which resulted with a pressure of 38.5 N/mm\(^2\). The worst case scenario was also a coast test for the side gears which resulted with a pressure of 24.5 N/mm\(^2\). To ensure that the right load is applied on the differential housing, the pressure is recalculated into forces which are applied through distribution couplings.

\[
\text{Ratio, } R = \frac{M_{\text{PTU}}}{M_{\text{F17}}} = \frac{400}{170} = 2.35
\]

\[
\begin{align*}
P_{\text{pinion}} &= P_{S,F17} \cdot R = 24.5 \cdot 2.35 \approx 58 \text{ N/mm}^2 \\
P_{\text{Side}} &= P_{S,F17} \cdot R = 24.5 \cdot 2.35 \approx 58 \text{ N/mm}^2 \\
P_{\text{pinion}} &= P_{\text{pinion}} \cdot A_{BF17} = 90.6 \cdot 766.7 \approx 70 \text{ kN} \\
P_{\text{Side}} &= P_{\text{Side}} \cdot A_{S,F17} = 57.5 \cdot 1731 \approx 100 \text{ kN}
\end{align*}
\]

### 3.5 Creating load cases

As the worst case scenarios for forward and reverse are considered, two load cases are created. For these two load cases the applied pressures, caused by interior gears, are equal. The difference is the direction of the applied force on the ring gear, and the direction of the torque representing the PTU. Following cases are determined:

<table>
<thead>
<tr>
<th></th>
<th>(F_t) [kN]</th>
<th>(F_s) [kN]</th>
<th>(F_i) [kN]</th>
<th>(M_{\text{PTU}}) [Nm]</th>
<th>(F_{\text{pinion}}) [kN]</th>
<th>(F_{\text{Side}}) [kN]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Forward</td>
<td>48.1</td>
<td>24.5</td>
<td>21.3</td>
<td>2250</td>
<td>100, 100</td>
<td>70, -70</td>
</tr>
<tr>
<td>Reverse</td>
<td>-48.1</td>
<td>-24.5</td>
<td>21.3</td>
<td>-2250</td>
<td>100, 100</td>
<td>70, -70</td>
</tr>
</tbody>
</table>

For definition of constraints applied for the two different load cases, see section 3.3.3 Applying constraints to FE-model.

\(^{18}\) It is known as the F17 gearbox.

\(^{19}\) Values obtained from test report yyy, which is GM confidential.
The differential housing is designed in such a way that it is not cyclically symmetric. This causes a difference in the stress distribution, which is dependent on the position of the interlocking between the ring gear and final drive. If we assume eight interlocking positions, with a 45 degree interval, the differences in the stress distribution can be recognized, even though accuracy can be improved by considering more interlocking positions. However, in this thesis, eight interlocking positions are considered to be sufficient for obtaining reasonable results. Furthermore, two planes of symmetry can be recognized. The conclusion can be made that instead of assuming eight interlocking positions, only three positions with a 45 degree interval on one fourth of the model will yield the same information. For every interlocking position, the two load cases have to be applied, creating three load steps for each load case. It is desired to have as few load steps as possible, both from a modeling as well as from a processing point of view.

![Figure 3.5.1 Display of two plane symmetry.](image)

The different load steps are displayed in figures 3.5.2 - 3.5.7.
Figures 3.5.2 - 3.5.7

Load step 1, first gear

Load step 4, reverse

Load step 2, first gear

Load step 5, reverse

Load step 3, first gear

Load step 6, reverse


3.6 FE-analysis
There are several reasons why a FE-analysis needs to be performed. Basically there is a need to obtain the responses and behavior of the model. Also, it is common to visualize the analysis to grasp the results easier.

For all load steps FE-analyses has to be performed prior to optimization. When post processing the results there are several evaluation criterions. For the models in this thesis there is a demand that the highest stresses have to be lower than the material’s yield stress\(^{20}\).

In the post processing stage the following procedure is done:

1. Evaluate the general stress distribution.

2. Highlight areas with stress concentration.

3. Evaluate if results are reasonable and can be considered to be sufficiently accurate.

The FE-analysis is an important foundation when devising appropriate implementations of structural optimization.

\(^{20}\) Yield stresses for materials used can be found in table 3.2 Material properties
4 Topology optimization of the Differential housing

4.1 Introduction to Topology Optimization of the differential housing.
In this chapter a topology optimization of a differential housing is being performed. Loads and constraints are applied according to previous chapters. Here instead focus will lie on the setup of the optimization and how it can be controlled. Some specific features in Optistruct will be explained. The topology optimization were an iterative process and decision of a final proposal was made and sent to a CAD engineer.

In the following sections the set up of the model and the optimization will be described. This is followed by the FE-analyses, optimization results, discussions and conclusions.

4.2 FE-model of the differential housing for Topology Optimization
Two CAD geometries were received where one contained the maximum design of the differential housing and the other one the minimum design. The advantage of this is that it is easy to build a model which has predefined design spaces. This makes it possible to divide the model into design and non design spaces with minimum effort.

The best design must be somewhere between the minimum and maximum designs. Here the idea is to come up with a design that combines the high stiffness of the maximum design and low weight of the minimum design. By performing topology optimization, the best design will be found according to the constraints specified.
4.2.1 Description of whole model

The whole model consists of a ring gear and a differential housing. The housing is divided into two parts which are the designable and the non-designable part of the domain. In figure 4.3.1 the whole model is shown:

Figure 4.2.1
4.2.2 Description of parts of the differential housing

According to specifications the minimum design is developed in a CAD-program. This design is the smallest functional design that can be accepted. By functional it means that support for the ring gear, bearings, interior gears, axes and pin exists. Another demand is that the interior gears of the differential must fit and have room to be assembled. See figure 4.2.2.1 and 4.2.2.2.

The design space represents the volume in which the material distribution is allowed to change during the optimization. Its outer surface is limited by the confined space in the gearbox and the inner surface is limited by the outer surface of the non-design volume. See figure 4.2.2.3 and 4.2.2.4.

A model which consist of both the design and the non design space can be seen in figures 4.2.2.5 and 4.2.2.6.
Figure 4.2.2.5 Design and non design space

Figure 4.2.2.6 Design and non design space
4.2.3 Contacts

It is of great importance that all contact areas are well defined to achieve a model that behaves in a logical and realistic way. Further and more in important it will yield valid results. For the topology optimization, a design where the ring gear is welded to the differential housing is chosen.

The area of contact in this model is between the housing and the ring gear. The connection will be modeled using two different methods. One is by applying weld and non-linear gap elements, and the other one is by simply meshing them together into one solid component.

FE-analyses are performed for both contact definitions but only the connection defined by meshing the parts together will undergo optimization. This is due to the fact that non-linear contacts are not applicable when optimizing. If non-linear contact are applied while optimizing the solver will face convergence problems.

Modeling contacts with Weld and Gap elements.

The weld connector simulates a weld between two components and acts as a rigid link. It will be used to weld the differential housing and the ring gear together instead of using bolts.

![Weld area shown with white circle](image)

Figure 4.2.3.1 Weld area shown with white circle

The area of contact between the ring gear and the housing that is not welded together needs a contact definition as well. This is done by applying gap elements. The gap contact allows the components to move relatively to each other with a connection that has high stiffness in compression and low in tension. The function of gaps provides a contact which makes the model behave in a logical and realistic way.
4.2.4 Applying gap elements
Performing a FE analysis is a good way of checking the quality of the connector. Test loads can be applied to check penetration, high displacements or transferring of stresses and strains in a correct way.
Figure 4.2.4.1 is showing gap elements when the structure is loaded with a test load for testing tension. No tensile stresses in the contact area where gap elements are applied which proves that the contact definition is correct for tension.

Figure 4.2.4.2

The stress distribution is showing that the Non-linear gap elements are working properly in compression. Also it is shown that no penetration occurs. Penetration of surfaces means that in compression the surfaces would overlap and thus cause unrealistic behavior. The load case used is a test load for testing compression.

4.2.5 Models for consideration
Analyses of models with both contact definition are desired, even if only one can be used when optimizing. A comparison between them will show how the difference in the contact definition will affect the behavior and response of the structure. This will be important information when evaluating the results from the topology optimization.

The different models will be referred to as simple or advanced models based on connector definition. The FE analyses will be carried out with second order elements.

The two FE-analyses which will be performed are named simple and advanced with the following attributes:

**Simple Model:** Second order elements, Meshed connection

**Advanced Model:** Second order elements, Non-Linear Gap elements, weld elements
4.3 FE-analysis

4.3.1 FE-analyses of model with design space
FE-analyses were performed on the maximum design model. All of the designable material is present. The FE-analysis shows a high stiffness and low stress state, which shows that this model is suitable for topology optimization. The FE-analysis are shown in the appendices.

Figure 4.3.1 Maximum design model

FE Analysis Simple model: See appendix 1
FE Analysis of Advanced model: See appendix 2

4.3.2 FE-analyses of model without design space
FE-analyses were performed on the minimum design model. None of the designable material is present and it yields a low stiffness and high stresses of the structure. From the stress distribution information can be obtain of where the material should be placed. The FE-analysis are shown in the appendices.

Figure 4.3.2 Minimum design model

FE Analysis Simple model: See appendix 3
FE Analysis of Advanced model: See appendix 4 (First order elements)
FE Analysis Simple model showing stresses over 800 MPa: See appendix 5
FE Analysis of Advanced model showing stresses over 800 MPa: See appendix 6
4.3.3 Evaluation of FE-analysis

**Calculation times for models with design space:**

Simple model:
- CPU time: 12:07:21

Advanced model:
- CPU time: 3 days + 18:37:21

**Calculation times for models without design space:**

Simple model:
- CPU time: 16:52:26

Advanced model:
- CPU Time: 2 days + 14:09:27

**Comparison between Simple and Advanced model**

In the advanced model the welded area will be affected the most and it will have the highest stress state. The gap elements will allow some movement between the ring gear and the housing and stresses are only transferred in compression. This is a more realistic modeling but the computer power and the calculation time needed is far greater than for the simplified model.

For the reverse load cases there will be no tension in the gap elements of the advanced model and therefore the stresses in the welded area will be greater. This is the major difference of the two different ways of modeling the structure.

The question is if the simple model provides a sufficient base for topology optimization. Evaluating the results from the FE analysis of the simplified model shows us that the overall stress state is similar to the advanced model but in the contact area between the ring gear and the housing there are differences. In these areas result cannot be considered trustworthy.

**Comparison between the Maximum and Minimum design**

The FE-analyses of the maximum design models shows that the stress level is low and that there is material to remove in a topology optimization while still maintaining satisfactory low stress levels.

The FE-analyses of the minimum design models show that the stress level is too high and also give a hint of where material should be kept.
4.4 Topology optimization in OptiStruct
Optistruct uses the finite element mesh to calculate material properties for each element. An algorithm within the program modifies the distribution of the material.

The process of setting up the Topology Optimization can be summed up in the following steps when aiming towards minimizing the objective function:

1. Specify areas as design and non design areas.
2. Specify symmetry planes if needed.
3. Specify draw direction constraints.
4. Define the objective function, responses, design variables and constraints.
5. Perform a topology optimization by using OptiStruct solver.
6. Post process by using HyperView and HyperMesh.
7. Repeat the processes until satisfying results are reached. This can be done by fine tuning optimization parameters, refining the mesh further, changing the design area, constraints and design variables.

4.5 Controlling the optimization
Controlling the optimization is the user’s way of restraining the results from the optimization runs. By specifying constraints on the structure or altering the solving technique or procedure it is possible to obtain solutions that meet standards and demands already in the optimization stage. At the moment the controlling features are basic but in the future it can be possible to have an advanced control of the optimization in such a way that all demands and standards can be met already in the optimization process. This involves preserving radii, defining variable radii, optimize mesh quality, advanced manufacturing constraints and so on.

In the following chapters some of the ways of controlling the optimization will be presented.

4.5.1 Setting up the optimization
First the target of the optimization must be specified. This is done by introducing the objective function which is to minimize the weighted compliance of the structure. By minimizing the compliance the stiffness will be optimized. Constraints on the responses and on the structure are then applied. The Opticontrol parameters can be specified or be kept at their default value.

4.5.2 Objective function Minimizing Weighted Compliance
By minimizing the compliance of the structure, the stiffness will be optimized. A downside is that it will not optimize the weight of the structure directly as this must be specified. This means that the structure can be made as stiff as possible while reducing the weight by specific percentage of the original weight. This is however a good way of producing design proposals as the results will visualize the load paths of the structure. Examples could be to investigate where and how a rib structure should be placed on an existing part.
In order to take every load case into account, a weighted compliance must be specified which will consider all load cases. By letting the weights be equal for all load cases they will have equal impact on the solution. If experience shows that one load case typically limits the life of the model, an increment of the weight factor could be considered.

4.5.3 Constraints on responses

Only one constraint on the response of the structure is considered. This is the mass fraction constraint which restrain the amount of design material present.

The mass fraction constraint is a constraint where a specified amount of designable material is available for optimizing the structure. It is often the case that all of this designated material will be used to find the optimum solution to the optimization problem but this depends from case to case.

The weight of the structure will not be optimized directly but as an early step in the design process it is a good way of coming up with new designs. It is also possible to find out where and how a rib pattern should be placed.

Chosen values for the maximum mass fraction are 0.05, 0.10, 0.20 and 0.30. By comparing the results it will be shown how much the mass fraction influence the result of the optimization. More material will yield a stiffer structure but with a higher mass as a consequence. From the result it is possible to see if the solutions look similar or if complete different solutions are obtained. It is often the case that an increment of the mass fraction value will simply add more material to the already designed members rather than making new designs.

4.5.4 Constraints on feasible designs

The constraints on feasible designs which have been applied are symmetry constraints and manufacturing constraints. These constraints remove solutions which are not feasible due to the fact that they will be impossible to manufacture or that the design needs to meet certain demands.

**Symmetry constraints**

When optimizing towards a producible design, symmetry constraints are often desired. Even though an initially designed structure is symmetrically loaded and constrained, the topology optimization in OptiStruct does not guarantee a perfectly symmetric design as a result. By using symmetric constraints one can however provoke an optimization leading to a symmetric design regardless of symmetry on constraints loads and initial design.

As previously mentioned, two planes of symmetry can be recognized for the differential housing. The two planes can be seen in figure 2.7.5. It is desired to maintain this symmetry while optimizing. This is especially important since the loading of the model is asymmetric\(^\text{21}\).

\(^{21}\text{See section 3.5 Creating load cases}\)
Figure 4.5.4.1 A two-plane symmetry constraint will be applied to the differential housing.

**Draw direction constraint**

Designs that are obtained from a topology optimization without manufacturing constraints are often not practical, when considering the manufacturing process. A main problem is that the designs often contain cavities that are not viable for casting.

The manufacturing constraints will remove solutions that are not feasible due to manufacturing methods such as casting. It can however be informative for the designer to see the solution without these constraints to get a good visualization of where the material should be placed in order to get the best results, even if it is not possible to manufacture it. This will yield a better understanding of the problem and the behavior of the structure. Maybe it is possible to change the original structure in some way or change the manufacturing method in order to make the design producible.

Optistruct with its manufacturing tools allows consideration of die casting manufacturing processes. When applying the draw direction constraint, the feasible solutions will be reduced into designs that are applicable on casting. The die can slide in user defined directions. The draw option allows the user to choose from two different choices:

- **Single**: This option assumes a single die to be used. It is defined in the way that it slides in the given drawing direction. A base is defined and from that a direction is determined.
- **Split**: This option assumes that two dies splitting apart in the given draw direction. A base is automatically defined and from that two different directions are determined (two vectors).

Non-designable parts of the domain can be defined as obstacles in the draw direction constraint. This preserves the casting feasibility of the final structure.

A split-die manufacturing constraint is applied on the differential housing in the draw directions shown in figure 4.5.4.2.
The following pictures show an optimized design proposal without the influence of manufacturing constraints. A structure is obtained which is not feasible when manufactured through casting.

Figure 4.5.4.3 No draw direction constraints

The following pictures show the influence of applying draw direction constraints. The result is a structure with no cavities which makes it suitable for casting.
For other reason these designs might be bad from a casting point of view. Then constraints have to be applied on dimensions of the created structural members.

**4.5.5 Constraints on quality of optimization**
As long as the optimization is converging, the process will continue to improve the quality of the optimization. The quality is the same as the value of the objective function. In theory the amount of iterations could be almost endless but after a while the iterations will only improve the solution very little. Somehow the quality of the optimization must be restrained.

By specifying a maximum number of iterations or an objective tolerance it is possible to decide how accurate the optimization process should be. By specifying a low tolerance and a maximum number of iteration the optimum would be improved but it would be time consuming.

By specifying a maximum number of iterations, the optimization process will stop after the last specified iteration regardless of the quality of the solution unless it has not stopped before this iteration.

By specifying a value on the objective tolerance, the optimization process will stop when the difference in the quality of the solution between three consecutive iterations is less than the objective tolerance.
4.5.6 Penalization
Discrete factor

The discrete factor will provoke a more discrete solution by penalizing the values which the densities are allowed to take\(^\text{22}\). A high value of the discreteness factor will yield a solution where an element either is close to fully dense or filled with void. This is important so that a solution with clearly defined structural members can be achieved.

Semi dense elements will be penalized so that these elements will not contribute to the stiffness matrix in the same way as usual. Usually the stiffness contribution is made linearly dependant of the density of the element but with penalization this dependency is made non linear. This is typically applicable when minimizing compliance is the objective function of the optimization problem since this is directly coupled to the stiffness of the structure.

It is up to the user to decide which level of discretion is wanted and semi dense elements can be filtered out manually in post processing stage.

**Minimum dimensions of structural member, (MinDim)**

The feature MinDim penalizes the formation of small members. The minimum dimension specification allows the user to choose a minimum dimension of the smallest structural member created during the optimization.

Specifying the MinDim parameter will increase the chances of a good design proposal as solutions with members smaller than the specified value will be discarded.

In order to obtain good and useful results, a minimum dimension should always be specified. This is especially important when a fine mesh is used since smaller elements can form smaller structural members. In this project the mesh size is so large that the minimum dimension will always be either 5 mm or specified through the default value of three times the average element size.

**Maximum dimensions of structural members, (MaxDim)**

The feature MaxDim penalizes the formation of large members. This feature is a new research development and the technique is still undergoing improvement\(^\text{23}\). Therefore usage of MaxDim should be used with care and only if it is truly desirable. The results should also be seen critically as they are not 100 % trustworthy.

The advantages of using Maxdim are many. It is possible to achieve a rib structured solution or to obtain a design with thicknesses of the largest member that are suitable for casting to name a few. If a rib

\(^{22}\) Also see chapter 2.4 *Theory of penalization technique*

\(^{23}\) According to Altair. See 1.4 *Software used*
structure is desired specifications of the minimum member as well as the maximum member size control should be specified.

The penalization does not take direction into account which means that if a structural member has a thickness smaller than the specified MaxDim value in any direction the constraint is considered to be satisfied. Controlling of thicknesses of specific directions is not yet possible but might be implemented in future Optistruct versions.

In order to obtain good results it is recommended to use a fine mesh. The minimum value of the MaxDim parameter is six times the average element size and less than half the size of the thinnest part of the designable region.

When using the parameter MaxDim in combination with the mass fraction constraint, the maximum mass fraction value can be 0.5 as the highest. This is due to the penalization and a forced spacing between structural members. Optimizations will be performed where the mass fraction takes values between 5 % and 30 % so this will not cause any problems.

A downside that has been noticed is the presence of more intermediate densities of the elements in the final solution when MaxDim is activated. In order to obtain better results an increased discrete factor could be applied.

Figure 4.5.6
Influence of MaxDim parameter. The left picture has Maximum dimensions of 15 mm and the right has a no specified maximum dimension. A nice rib structure with enforced spacing is obtained when MaxDim is used. The setup is the same otherwise.
4.6 Evaluation of Topology optimization

4.6.1 Analyzing stress levels of different designs
Note that no constraints on the stress level are applied. The topology optimization will only aim at increasing the overall stiffness of the structure as much as possible with the mass fraction available. However, the stress levels benefit from a structure with an optimized stiffness and this will be shown at the FE-analysis following the topology optimization. Regions with elevated stress levels can be further optimized in a later stage by performing a shape optimization.

The stress levels of all designs can be analyzed during the post processing in Hyperview. The design is chosen by filtering out semi dense elements below a user specified threshold value. These filtered out elements will only be masked out and still contribute to the stiffness. This will cause an underestimation of the stress levels and an overestimation of the stiffness. Furthermore, the kept elements are semi dense as well and since all those elements will be fully dense when produced it will cause an overestimation of the stress level and underestimation of the stiffness. This however is a good preliminary check of the stress state of the structure. Increasing the discreteness factor can be considered in order to obtain a more discrete the solution.

The only way of analyzing the true stress level is by redesigning the chosen design and perform a FE-analysis of the chosen design.

4.6.2 Influence of Calculation times
With every kind of analysis or optimization the computational calculation time needed is of great importance. A certain amount of loops are always needed and the engineer is always dependant on the results from the latest run in order to take the next step or to fix errors or undesirable behavior. For the single run time might not be hugely important but if a process consists of 10, 50 or more loops, than these calculation times build up to a considerable amount time. A process that could be done in days can take several weeks or months. Therefore it should be the target to minimize the calculation times as well as the number of loops to save time for the complete project.

For this project, the desirable maximum time for an optimization run is less than seven hours which allows for overnight calculation as well as getting results in the afternoon from morning calculations.

4.6.3 Analyzing the Weight reduction
When using the mass fraction constraint it is typically the case that all the available material will be used in order to find the best solution to the optimization problem. It is easy to believe that a mass fraction value of for example 0.2 will yield a 80 % reduction of the designable mass which is true in one way but not in another. All the semi dense elements will be fully dense if produced and all the elements below an element density filter value will be removed which makes it more difficult to say something about the weight of the design proposal.
The true weight reduction can be done by analyzing the chosen design again by exporting the design and perform an analysis with fully dense elements.

4.6.4 Mesh size influence
A finer mesh would provide for a more accurate calculation of responses, and further more yield better optimization results. More control over MinDim and MaxDim is also retrieved. MinDim is especially important when the mesh is very fine as the small members of the structure can become smaller and this must be restricted. A downside of having a very fine mesh is that calculation times are increased.
4.7 Results

Final Design chosen
The final design is considered the best design proposal and is chosen to be sent to the designers for further redesigning. After that it is possible to do a FE-analysis and a shape optimization.

The design chosen has the following specification:

Objective function:
Minimize Weighted Compliance

Responses:
Mass fraction of design material

Constraints:
Mass fraction ≤ 0.2 of design mass
Stress: No stress constraints active
Symmetry: Two-plane symmetry
Manufacturing: Split die with obstacles
Minimum structural member size: 5 mm
Maximum structural member size: 15 mm
Discreteness factor: 3
Max number of iterations: 30

Order of elements:
Second order

Load steps:
Six load steps weighted equal

Iterations performed: 16
Status: Successful optimization
Calculation time: 3 days
4.7.1 Chosen design proposal

The post processing is done in Hyperview. A filter removes elements with densities below 0.68. This filter was chosen by simply testing different filter values and choosing the one that looked best. Elements below this specified value were masked out in order to visualize and analyze approximate stress levels.

The pictures below show the chosen design according to the filter value 0.68 and is exported to Hypermesh. The purple area is the proposed kept material of the design volume and the light blue is the minimum design which could not be altered.

Figure 4.7.1.(1-6)
Support for the side holes are obtained. The side where the pin is applied has a larger non design space which is reflected over the planes of symmetry to the other side. This material may be placed here just due to this and not that the structures stiffness benefits from this optimally. A rib structure can also be noticed between the side holes and the large holes. The gap underneath the bearing support is filled out. More material is placed near the side holes than over the large hole. A nice and smooth radius is preferable in this area.

A nice rib structure is obtained on the bottom side of the housing. These ribs support the housing under the large hole. This side where the large hole is can be seen as the weaker side from a stiffness point of view.

4.7.2 Approximate FE-analysis of stress levels of final design

As mentioned before an approximation of the stress levels can be done already in the post processing stage. By filtering out elements below a density threshold and mask them out it is possible to do stress analysis of the chosen design. These masked out elements will still be present in the background and provide stiffness to the structure. This in combination with the fact that semi dense elements exist above this threshold value causes a numerical fault the stress levels obtained in the analysis. However it will provide a good approximation in an early stage of the design process.

FE-analysis of optimized model, See appendix 7

The approximate overall stress state is low. The areas where the stress levels are elevated are in the non design region. It is the 90 degree turn on the bottom side (see figure 4.7.2.2) and the area around the pinhole (see figure 4.7.2.1). These areas should be investigated further and solution to this issue cannot be found through this topology optimization but rather by shape optimization or new innovative design regarding the pinhole function. A pin and pinhole which follows the shape of the housing instead of the opposite could be one solution.

Figure 4.7.2.1
This area around the pinhole has a disruption of the smooth radius of the housing which causes stress concentrations. This in combination with no design material causes remaining stress concentrations even after the optimization.

![Figure 4.7.2.2 Pinhole](image1)

![Figure 4.7.2.3](image2)

Area with stress concentration. This is caused mainly by the torque take out from the PTU. No design material in the vicinity makes it hard to reduce the stress levels just by topology optimizing. A local shape optimization is needed.

### 4.7.3 Weight comparison of different designs

<table>
<thead>
<tr>
<th>Design</th>
<th>Weight</th>
<th>Design material</th>
<th>Design material (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum design</td>
<td>7,19 kg</td>
<td>0 kg</td>
<td>0 %</td>
</tr>
<tr>
<td>Maximum design</td>
<td>9,56 kg</td>
<td>2,37 kg</td>
<td>100 %</td>
</tr>
<tr>
<td>Optimized design</td>
<td>7,66 kg</td>
<td>0,47 kg</td>
<td>20 %</td>
</tr>
</tbody>
</table>

Table 4.7.3 Weight comparison
4.7.4 Approximate FE-analysis of displacements of final design

Figure 4.7.4 Interlocking of Ring gear and Final drive in the red area on the ring gear.

<table>
<thead>
<tr>
<th>Displacement</th>
<th>Minimum Design</th>
<th>Maximum Design</th>
<th>Optimized Design</th>
</tr>
</thead>
<tbody>
<tr>
<td>Loadcase 1</td>
<td>1.435 mm</td>
<td>0.35 mm</td>
<td>0.78 mm</td>
</tr>
<tr>
<td>Loadcase 2</td>
<td>1.061 mm</td>
<td>0.2856 mm</td>
<td>0.65 mm</td>
</tr>
<tr>
<td>Loadcase 3</td>
<td>0.9244 mm</td>
<td>0.2986 mm</td>
<td>0.64 mm</td>
</tr>
<tr>
<td>Loadcase 4</td>
<td>1.517 mm</td>
<td>0.3573 mm</td>
<td>0.77 mm</td>
</tr>
<tr>
<td>Loadcase 5</td>
<td>1.189 mm</td>
<td>0.3075 mm</td>
<td>0.66 mm</td>
</tr>
<tr>
<td>Loadcase 6</td>
<td>1.046 mm</td>
<td>0.3539 mm</td>
<td>0.68 mm</td>
</tr>
<tr>
<td>Difference highest - lowest</td>
<td>0.5926 mm</td>
<td>0.0717 mm</td>
<td>0.14 mm</td>
</tr>
</tbody>
</table>

Table 4.7.4.1

By investigating the absolute displacements of the structure it is possible to see how the stiffness has increased from the topology optimization. As mentioned it will only be an approximate value of the displacements and will only acts as a rough estimation. By comparing the results with the initial design it will be shown if the stiffness has increased. The highest displacements will be found in the interlocking between the ring gear and the final drive. This is where the torque is applied to the ring gear from the gear box. A comparison between the minimum, maximum and the optimized design show the displacements as well as the stiffness distribution. This is seen in table 4.7.4.1.
Table 4.7.4.2 Reduction of displacements of ring gear at final drive mesh when comparing Minimum and optimized design.

The displacements are lowered by as much as 49 % when comparing the minimum and the optimized design. The reduction of the difference between the highest and lowest displacements is 76,4 % which shows that a better distribution of the stiffness is achieved. What this means is that regardless of the interlocking position between the ring gear and the final drive the stiffness of the structure is optimized.

### 4.7.5 Comparison of stresses with initial FE-analysis
Comparisons show that a provoked reduction of 80 % of the design material resulted in stress levels of roughly the same magnitude as before according to the approximate FE-analysis of the optimized structure. It is also the case that the highest stresses were located in the same areas as before. These areas are in 90 degree turn on the bottom side and around the pinhole. These were areas that were hard to affect through topology optimization since no design material were present in the vicinity.

The important things to see when comparing the FE-analyses is that the stress levels are roughly the same and that areas with stress concentration are in the non design area and are therefore hard to reduce.

A comparison of the displacements of the final design proposal and the initial design show that stiffness has increased and the displacements been reduced dramatically.
4.7.6 Redesigning of Final design proposal
When satisfied with a design it is time to send the model to a CAD designer who can make a proposal of a producible design. Then it is sent back for FE-analysis and possibly a shape optimization. In this thesis timing made it difficult to perform the latter two steps. The design proposal can be seen in figure 4.7.6.2.

Repeat process until satisfying design is reached.

Figure 4.7.6.1 Optimization in a design process

Figure 4.7.6.2 Redesigned CAD model of differential housing
4.8 Conclusions

The most interesting part of the results is the rib structure on the bottom side of the housing. It can be seen that it supports the weakness that the large holes will bring to the structure. It is also a design that is very easy for CAD engineers to implement and it is also producible. Once the ribs are done by the designer it is possible to optimize the shape of them by performing a shape optimization. This is a good example of how topology optimization can prove to be a very powerful tool early in design processes.

The top side design is not as easy to grasp and further analyses and optimization probably has to be done to come up with a suitable design. Furthermore it is hard for the designer to interpret the design and make a CAD model with defined radii and smooth shapes that still maintain good stress levels. This yields more loops between optimizer and designer and is therefore done only when truly needed.

Areas with high stresses are mainly in the non design region where the geometry is not always beneficial for the stress state. It can be hard to reduce these stresses by just topology optimizing the design space. The peak stresses are due to the geometry rather than lack of support. The possibilities of altering the shape of the minimum design could also be considered.

A lot of optimization runs were performed where different setups were used. This resulted in a large number of different design proposals. All of them are not presented in this report but rather the examples that show the influence of changing different parameters. Some parameters are clearly affecting the outcome of the optimization. These are parameters like MinDim, MaxDim, draw direction and symmetry constraints. The conclusion is that parameters that restrict the feasible design affect the outcome of the optimization in a great way and should therefore be used with more caution and only when truly needed.

All these results of the optimization runs led to the choice of one design that felt interesting enough to go further with. This was sent to a CAD engineer for redesigning into a part that meets production standards. This will then be sent back for FE-analyses and possibly shape optimization. The design proposal was made but time made it difficult to perform FE-analyses and shape optimization.

Since the housing and the ring gear was meshed together the connection between them was modeled too stiff. This behaves in a somewhat other way than a welded model would. The influence of this should be investigated thoroughly. The question is if the result can be validated or if the structure must be modeled in another way.

Choosing different mass fraction values will influence the design in such a way that structural members become larger rather than creating new ribs. With the maximum dimension parameter however there is a limit of how large they can become and creation of new structural members can be provoked.

The issue of semi dense elements can also be discussed. All the optimization runs show that there will always be semi dense elements regardless of the discrete parameter. In the post processing stage, elements with low density can be filtered out. A recommendation is to always use a higher discrete factor when using maximum dimension constraints as this feature produces a lot of semi dense
elements. In other cases the discrete factor can be used but the affect is not that great. The default value of this factor will often be sufficient to produce good optimization results.
5 Shape optimization
This section will include some introductory theory about the shape optimization function in Optistruct. Furthermore test and evaluations are made on different shape optimization functions, that Optistruct provides. The tests are made on a differential housing and they consist of:

- Evaluating the affect of the mesh size, when shape optimizing.
- Evaluating the size of the design space, when shape optimizing.
- Evaluating a shape optimization feature called "Hypermorph".
- Confirming the conclusions, from the tests and evaluations, on a fork shifter.

5.1 Introduction to Shape optimization
Free shape optimization is a tool of great use when aiming towards local refinement of a model. A local region where for instance an enhanced stress level is recognized during analysis is found and this local area will be free shape optimized in order to lower the stress level to an acceptable level.

The local area is defined as a design area (design space) which means that it is allowed to change its shape during optimization. The areas that are not included in the shape optimization are called non-design space.

Figure 5.1 Red colored elements represent the design space and the blue colored elements represents the non-design space. The white dots (nodes) represent the boundary between these elements.
5.2 Shape optimizing in Optistruct

There are two different tools for performing a shape optimization in Optistruct. The tools are called "Free Shape" and "Hypermorph". The tools differ in the way you set up the optimization. The difference can be shortly explained in following way:

**Free Shape:** When using this tool the optimizer only needs to define the design space and the objective of the optimization. Of course some minor and easy constraints can and should be added. These are explained more detailed in section 5.2.1 Free Shape

**Hypermorph:** When this tool is used, the optimizer needs to define the boundaries of the design space. With the help of the tool, a new shape is created by morphing the design space and determining the boundaries. Hypermorph is explained more in detail in section 5.2.2 Hypermorph

5.2.1 Free Shape

The free shape tool has its boundaries when allowing the optimizer to influence the design of the new shape. Instead it has its strength when constraining how the design space can change. In other words, this tool is of great use when the optimizer wants Optistruct to freely determine the new shape, whilst controlling the way the new shape can be created.

The figure below displays a good function within Free shape. This function allows the user to define how the design space should change shape. The function allows the user to constrain the shape change in the way that it only can grow, shrink or do both. For example, by constraining the design space in the way of only letting it shrink, violation of boundaries can be prevented.

1. **GROW** – grids cannot move inside of the initial part boundary.
2. **SHRINK** – grids cannot move outside of the initial part boundary.
3. **BOTH** – grids are unconstrained.

![Figure 5.2.1 from Altair Engineering OptiStruct Tutorial](image)

---

24 The word morph originates from the greek word morphe meaning shape or form. Morphing can then be described as the process of altering the shape or form of the FE-model.
Another good function with Free Shape is the possibility of constraining the nodes within the design space. These constraints basically determine where and how in space the new shape can change. There are several constraints of such kind and they are:

- **Fixed**: A set of nodes are not allowed to move at all which is typically the case for nodes on the boundary of the design area. See figure 5.1 for an illustration.

- **Vector**: The nodes that form a grid are forced to move along a user defined vector. When defining the vector, no consideration of which way it is directed needs to be done. The grid will move in direction of the vector (no need to rotate the vector 180 degrees if the arrow points in the wrong way).

- **Planar**: The grid that is chosen as a group of nodes can only move on the specified plane where a vector defines the normal.

### 5.2.2 Hypermorph

This tool works in the way of morphing the design space by altering the mesh. The shape change of the FE-model is done with minimum mesh distortion. Even though the mesh is distorted, it is not ruined and fulfills the demands that earlier are defined. That shape is saved and only used when the model is sent for optimization. The shape is created in such a way that the volume between the original shape and the new shape defines the new design space. With other words the new optimized design will find a solution (if possible) within the boundaries of the old shape and the design created by the optimizer.

With Hypermorph, unlike Free Shape where the new shape is defined in the way of how it should change, the shape is designed before the optimization. The strength with Hypermorph is that fillets, radii and other complicated geometries can withhold their form while undergoing an optimization. For example wanting to maintain the shape of an outer surface, whilst changing the volume below it.

Hypermorph consists of four different methods depending on the shape changes required. These are as following:

- **Domain and handles**: This method creates domains that consist of elements. Furthermore handles that are coupled to the domains are created. When moving the handles, the elements that attached to the domain will be inflicted. This method is the most frequently used one. It is applicable in most cases and has a wide range of functions.

- **Morph volumes**: This method involves Morphing the model while keeping tangency to the shape. It creates boxes, known as morph volumes, with handles at each corner as well as the edges. The edges can furthermore be chosen to be Lagrange or Spline curves. When the handles move the interior nodes of the morph volume will be affected correspondingly causing a shape change of the model.
- **Map to geometry:**
  This method allows the user to map nodes in the model to a line, plane or surface by using handles in the morphing process.

- **Freehand morphing:**
  This method is suitable for basic shape changes of the model. Applications could be translation, rotation, move to vector, move normal, map to line etc.

### 5.2.3 Domains and Handles

There are different kinds of domains. Depending on the case following domains can be determined:

- **Global domain:** consists of entire model
- **1D domain:** contains 1D elements such as bars or rigid elements.
- **2D domain:** contain a group of shell elements.
- **3D domain:** contain a group of solid elements.
- **Edge domain:** contains a series of nodes commonly found along edges of 2D or 3D domains.

The domains shape can be modified by placing moveable handles on the domain edges. When the handles are moved the shape of the domain changes, i.e. the nodes within the domain changes their position. The mesh is modified simultaneously in a logical way by letting nodes closer to the moved handle be more affected then nodes further away from it. The mesh stretches or compresses according to the shape change of the domain between moved and unmoved handles.

Handles can in the same way as domains be of a local or global form.

- **Global handles** influence every node of the model and is recommended to be used only for large scale shape changes of the model.

- **Local handles** can be applied on local domains and influence the nodes in the local domains, which they are associated with. Local handles is used when making small shape changes of the model locally.

Handles can furthermore be classified as independent or dependent, also see figure 5.2.3.1-3.

- **An independent handle** only moves by the influence applied to this certain handle.

- **A dependent handle** moves by the influence applied to it and also by the influence from the handles associated to it. This means that the dependent handle is affected by the movement of
the handles to which it has a linkage. There are possibilities of creating a great number of layers of these dependencies. See following figures for an illustration of how the domain and handles works.

Figure 5.2.3.1 No handles has been moved and nodes are equally distanced from each other

Figure 5.2.3.2 One handle has been moved and the nodes between the moved and the fixed handle moves correspondingly. Note that the nodes closest to the moved handle have moved a greater distance than those further away.
One handle has been moved and the handles to the right moves correspondingly due to the dependencies.

5.3 FE-analysis of the differential housing
From the earlier mentioned six different load steps, indications of high stresses reveal where local reshaping needs to be done. From appendix 8, where the results from the FE-analysis is displayed, the Von Mises scaling shows with color and values where attention needs to be drawn. The dark blue colored parts represent areas where stresses are close to 0 and the red colored areas display the highest affecting stress levels.

From the FE-analysis it can be seen that several loadcases have a stress level around 1100 MPa. If this model were to be sent for production, all areas that violate 800 MPa should be shape optimized. However if a great deal of areas are in the need of a shape optimization, a change of the original design should be taken into consideration.
5.4 The affect of the size of the design space when shape optimizing
As seen from the FE-analysis in appendix 8, different areas are in need of a shape optimization. These areas have a high stress peak, which creates a need of reducing them as they are violating the demands. From earlier sections it is mentioned that when performing a shape optimization it is preferable to refine the mesh locally. However it is never mentioned how large the design space can be. It is also not mentioned whether several areas can be shape optimized at once, saving time not optimizing area by area. It seems that it would be a lot easier to include all those areas in one optimization. But then again it is important to keep in mind how much computer power that is needed when refining the mesh. Also how much longer the optimizations will take.

From the two figures below it is seen where the mesh is refined to 1 mm elements and which nodes that are defined for a shape optimization. Could the entire differential (nodes that are free from rigidity constraints) be defined for a shape optimization? There are several arguments against that. One which is not mentioned yet is that there is a bigger risk of some elements violating demands, when unnecessary nodes are defined in the optimization.

To evaluate the above mentioned, different optimizations were executed on the differential housing. Also two different objectives were chosen for the evaluations. One objective was minimizing the weighted compliance and the other one minimizing the maximum stress level.

Figure 5.4.1 Refined elements
Figure 5.4.2 White dots illustrate nodes that are defined in the shape optimization
5.5 The influence of mesh size

When performing a shape optimization in optistruct one might wonder how big the influence of the mesh size is. Sometimes when a finished model is sent to the designers they have to guess the correct fillet size, thus could the optimizer provide the right information so that the right fillet size is obvious? A test will be conducted on load step 4, see figure 3.5.5, where two different mesh sizes will be tested for optimization. The first model will have the mesh size locally improved from 2 mm to 1 mm, whilst the second model will have the mesh size locally improved from 2 mm to 0.5 mm. From the figures below, the difference in mesh size is very obvious. But how long will the optimizations take for the different mesh sizes and how adequate will they be?

Another mentionable issue is which order should the elements have (how many nodes). It is preferable to have a second order element which will create 10 (four corner nodes and six mid nodes) nodes per element when it consists of a tetra mesh. When measuring and testing stress levels the results get more precise with more elements and more nodes. However it is known that Hypermesh 8.0 cannot handle a free shape optimization with second order elements. Keeping the elements in the first order will create a stable way of solving free shape problems.

![Figure 5.5.1 Mesh size 1 mm](image1)

![Figure 5.5.2 Mesh size 0.5 mm](image2)
5.6 Optimizing by using Hypermorph

From section 5.2.2 Hypermorph it can be regarded that there are several ways of morphing and depending on the case some are easy to use and some not. In this case two different types were used, “Morph volumes” and “Domains And Handles”. These two types were chosen because of the geometry of the area that will undergo a shape optimization. For comparison reasons the two different types were sent for optimization. These two different types are displayed in the figures below.

It is earlier seen from the FE-analysis that there are several areas in need of a shape optimization. However the load step with the highest affecting stress is chosen for this optimization.

Figure 5.6.1 shows the morph volume that is inside the box. The arrows display how the box is deformed to create a new shape. The elements within the box will deform when the box is deformed.

Figure 5.6.2 shows the domains and handles. Like the previous figure the arrows illustrate how much and how the elements have deformed during the morphing. The elements within the red line lie within a domain and the yellow nodes display the handles on the domain boundary.

When choosing the area where it is wanted to create domains and handles, Optistruct will automatically apply domains and handles. Depending on what is wanted for the shape optimization, the domains and handles need to be user defined in the way of deleting and adding them. This will make it much easier to create the shapes that are wanted. Also, often it is wanted to add some geometry dependent handles, for a smoother transition. However in this case none is needed and adding them will just take more time. When the area is setup and ready for a reshape, the translation of the handles or domains can begin in order to create the shape wanted. When applying handle perturbations it is quite important to consider the bias values.

When using a handle perturbation it is often tricky to acquire soft and smooth surfaces without ruining the mesh (reducing the mesh distortion). The bias value helps when having troubles with this. By determining the bias value, the influence of the handle relative the nodes within its is determined.
5.7 Results

5.7.1 The affect of the size of the design space when shape optimizing
One affect from tests is clearly seen in the way that the larger the areas are the fewer the optimization iterations will be. This is strictly because if you have a small area to refine the mesh, you could refine the mesh better than for a large area. Again a larger area demands more calculating time and computer power which will for today's technology put a limit on the element size. However the results show that a large area can be marked as a design space when shape optimizing.

One should recognize that different cases have different strategies. From the FE-analysis the areas with the highest stresses from three different load steps have been chosen.

Results when the objective was minimizing the maximum stress:

![Maximum stress level has decreased with 18 %.
Figure 5.7.1.1 load step 4](image1)

![Maximum stress level has decreased with 6 %.
Figure 5.7.1.2 load step 2](image2)
Results when the objective was minimizing the weighted compliance:

- Maximum stress level has decreased with 11%

Figure 5.7.1.3 load step 3

- Maximum stress level has decreased with 23%

Figure 5.7.1.4 load step 4

- Maximum stress level has decreased with 4%

Figure 5.7.1.5 load step 2
5.7.2 Influence of mesh size
When changing the mesh size in the way of making them smaller the FE-analysis gets more accurate. The highest stress level when using a 1 mm improved mesh area with load step 4 is 1074 MPa. The number of iterations from the optimization was 6 and the time for execution was 7 minutes. The final result showed from previous section that the highest stress level was less than 800 MPa. The FE-analysis of the model with a locally improved mesh size of 0.5 mm shows that the highest stress level of 1050 MPa.

The model with the improved element size had a total calculation time of 17 minutes and 12 optimization iterations. This model is more accurate with its analysis and it is obvious that there are areas where the elements violate the guideline of 800 MPa. The dark yellow elements have a stress level of just over 800 MPa.

When the model was sent for optimization the constraints were defined in such way that it was unconstrained. With other words the optimization can add and remove material how it wants, as long as the maximum stress levels are minimized. By applying that constraint only 0.4 grams material was added. This means that if a new design based on these results were to be made only a total of 0.12 grams would be added to the differential, hence there are four similar corners that are affected with the same stress level.

The results are shown in the figures 5.7.2.1-2.
Figure 5.7.2.1  1 mm mesh size

Figure 5.7.2.2  0.5 mm mesh size
### 5.7.3 Comparison between the hypermorph and the free shape functions

First of all, the evaluation of the two different hypermorph methods are explained. The results that were achieved from the two different hypermorph methods were exactly the same. Even though the shapes differed, the final result and number of optimization iterations were exactly the same. Following results were achieved:

![Figure 5.7.3.1 Volume Morph](image1)

![Figure 5.7.3.2 Domains and handles](image2)

The figures from above show clearly the same result for both types of hypermorphing. It is also seen that the high stress peak of 1074 MPa are decreased to the highest stress level of just around 780 MPa. However the element size has only been refined from 2 mm to 1 mm. The test took around seven minutes to execute and the number of iterations were five.
The differences between the free shape optimization and the hypermorph optimization are very small, regarding this case. The differences are seen in figure 5.7.3.3 compared to figures 5.7.3.1-2 and is mainly noticeable on the yellow elements. when using the free shape method, the number of yellow elements were decreased. In other words the average stress peak was decreased more by using the free shape function.

Figure 5.7.3.3 free shape method
A comparison is made on the following table:

<table>
<thead>
<tr>
<th>Comparison</th>
<th>Free shape</th>
<th>Hypermorph</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calculation time</td>
<td>≈ 7 min</td>
<td>≈ 7 min</td>
</tr>
<tr>
<td>Setup time</td>
<td>Easy &amp; fast</td>
<td>Time consuming</td>
</tr>
<tr>
<td>Stress level before opt.</td>
<td>1074 MPa</td>
<td>1074 MPa</td>
</tr>
<tr>
<td>Stress level after opt.</td>
<td>750 MPa</td>
<td>780 MPa</td>
</tr>
<tr>
<td>Change in percent</td>
<td>30 %</td>
<td>27 %</td>
</tr>
<tr>
<td>Mass change</td>
<td>+0.76 grams</td>
<td>+0.76 grams</td>
</tr>
<tr>
<td>Nr of iterations</td>
<td>6</td>
<td>5</td>
</tr>
</tbody>
</table>

Table 5.7.3 Comparison between the Free shape and Hypermorph feature.
5.8 Analysis & Conclusions

5.8.1 Evaluation of the size of the design area
A careful analysis shows that the differential housing is a poor example for evaluating the size of the design area. If every load step had several areas that needed a shape optimization and the mesh distortion from importing the model from SimLab was not that great, the differential housing would have been a good example. Therefore a confirmation of the results needs to be done on another example, where there are several areas with high stresses, within one load step. There is a fork with this known problem and a test will be executed it, to validate the conclusions.

The optimization case puts the limitation on the size of the design area. If there are no constraints and there are several areas that need a shape optimization, one could make one optimization case out of it. However if there are constraints, one has to create different optimization cases depending on the constraints. Which kind of objective to use is also dependent on the optimization cases. If the violation is high stresses, minimizing the weighted compliance or minimizing the maximum stress could be used.

**Suggestion:** If several areas are in need for a shape optimization and there are no constraints such as optimizing along a plane or vector, all violated areas should be added to the optimization problem and be optimized with the objective of minimizing the weighted compliance. However if there are constraints, different optimization cases should be created with the objective of minimizing the maximum stress level.
5.8.2 Evaluation of the influence of the mesh size

The difference of the different mesh sizes are obvious and are seen in the figures below.

![1 mm element mesh size](image1.png) ![0.5 mm element mesh size](image2.png)

Figure 5.8.2.1  1 mm element mesh size  Figure 5.8.2.2  0.5 mm element mesh size

Following differences were obtained:

- When locally remeshing a smaller element size results with a smoother surface. The surface in figure 5.8.2.1 is quite angular compared to the surface in figure 5.8.2.2.
- The corner that actually is a radius is much easier seen and noticed with the smaller elements. These facts can help a designer when trying to re-designing a certain part or an entire detail.
- The time difference from executing the optimizations between the smaller element mesh and the bigger element mesh is 2.4 times. In total the model with 0.5 mm element mesh took only 17 minutes which still is an excellent execution time. This short time is more than satisfactory for an optimizer.
- The better meshed model shows better results, which is obvious.

**Suggestion:** It is apparent that the result from the more advanced model is much better and the influence of mesh size is big. We believe that it is not even questionable to not use as good (small) mesh size as possible dependent on how large the area undergoing an optimization is.
### 5.8.3 Comparison between freeshape and hypermorph

The Free Shape function is easy to use and time saving. When the model is analyzed from the FE-analyses, areas that need shape optimization are spotted by the optimizer. It is more common for the optimizer to use Shape Optimization for several reasons like: faster method, easy to set up etc. However when using the Free Shape method it could get very tricky to determine how the shape can change and sometimes it is impossible. For instance it is difficult to maintain the geometry of a radius or even the size. Maintaining productivity demands could also be an insolvable issue. In these just mentioned cases Hypermorph is easier to use and will give good results.

An example follows for the above mentioned problems. If this area is to be optimized, one can see that the shape could change in a way that would conflict with productivity demands. In this case the productivity demand is to obtain one size throughout the whole fillet. Usually the constructor guesses the fillet size which will lead to a new FE-analysis and perhaps a new shape optimization. If these iterations between optimizer and constructor could be reduced or removed, one can save a lot of time. Hypemorph could be a good tool to use when aiming for this goal. There are other applications where Hypermorph is a good tool such as: changing radiuses, extending a shaft, bending geometries etc.

![Figure 5.8.3](image)

**It is often preferable that the fillets or corners have the same radius throughout.**

**Suggestion:** Only use hypermorph when the free shape function is inadequate or when wanting to change the geometry in such way as: shifting radiuses, extending or bending geometries etc.
5.9 Shape optimization of a fork shifter

This section is added to confirm section 5.8.1 Evaluation of the size of the design area. An earlier FE-analysis made within GM revealed nine areas in need of a shape optimization, within one load step. Some of these areas have the same geometry and are divided into three areas. These areas are displayed in figures 5.9.1-3. The target value for the highest stress peak is 170 MPa. Area 1 and 2 violates this demand. However area 3 is on the limit, but is still undergoing a shape optimization.

![Figure 5.9.1 Results from the FE-analysis made on the fork shifter.](image)

![Figure 5.9.2 Displays area 1 and area 2.](image)

![Figure 5.9.3 Displays area 3.](image)
5.9.1 Setting up the optimization

the original mesh consists of an average mesh size of 1 mm. For better accuracy, the red colored areas in figure 5.9.1.1 are remeshed to a new mesh size of 0.5 mm. Since this model is analyzed and tested in another project, the constraints and loads are already validated. Therefore only a change in the mesh has been made in this thesis before setting up the optimization. The same optimizations that were made on the differential housing were made on this model. The pink elements are the design spaces and the yellow elements the non design spaces.

Figure 5.9.1.1

Figure 5.9.1.2

Figure 5.9.1.3
5.9.2 Evaluation of the size of the design area

Following results were obtained when minimizing the maximum stress on all areas:

<table>
<thead>
<tr>
<th>Area 1</th>
<th>Area 2</th>
<th>Area 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stress level before opt.</td>
<td>Highest 200 MPa</td>
<td>Highest 176 MPa</td>
</tr>
<tr>
<td>Stress level after opt.</td>
<td>Highest 166 MPa</td>
<td>Highest 166 MPa</td>
</tr>
<tr>
<td>Change in percent</td>
<td>17 %</td>
<td>6 %</td>
</tr>
<tr>
<td>Mass change</td>
<td>≥ 0 grams</td>
<td>≥ 0 grams</td>
</tr>
<tr>
<td>Nr of iterations</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>Time for optimization</td>
<td>≈ 30 minutes</td>
<td>≈ 30 minutes</td>
</tr>
</tbody>
</table>

Table 5.9.1

Following results were obtained when minimizing the maximum stress only on area 2:

Area 2

| Stress level before opt. | Highest 176 MPa |
| Stress level after opt. | Highest 130 MPa |
| Change in percent | ≈ 26 % |
| Mass change | 2 grams |
| Nr of iterations | 7 |
| Time for optimization | ≈ 40 minutes |

Table 5.9.2
Following results were obtained when minimizing the maximum stress only on area 3:

**Area 3**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Stress level before opt.</strong></td>
<td>Highest 169 MPa</td>
</tr>
<tr>
<td><strong>Stress level after opt.</strong></td>
<td>Highest 157 MPa</td>
</tr>
<tr>
<td><strong>Change in percent</strong></td>
<td>7 %</td>
</tr>
<tr>
<td><strong>Mass change</strong></td>
<td>- 10 grams</td>
</tr>
<tr>
<td><strong>Nr of iterations</strong></td>
<td>2</td>
</tr>
<tr>
<td><strong>Time for optimization</strong></td>
<td>≈ 17 minutes</td>
</tr>
</tbody>
</table>

Table 5.9.3

Following results were obtained when minimizing the weighted compliance on all areas:

<table>
<thead>
<tr>
<th></th>
<th>Area 1</th>
<th>Area 2</th>
<th>Area 3</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Stress level before opt.</strong></td>
<td>Highest 200 MPa</td>
<td>Highest 176 MPa</td>
<td>Highest 169 MPa</td>
</tr>
<tr>
<td><strong>Stress level after opt.</strong></td>
<td>Highest 160 MPa</td>
<td>Highest 150 MPa</td>
<td>Highest 130 MPa</td>
</tr>
<tr>
<td><strong>Change in percent</strong></td>
<td>20 %</td>
<td>15 %</td>
<td>23 %</td>
</tr>
<tr>
<td><strong>Mass change</strong></td>
<td>13 grams</td>
<td>13 grams</td>
<td>13 grams</td>
</tr>
<tr>
<td><strong>Nr of iterations</strong></td>
<td>4</td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td><strong>Time for optimization</strong></td>
<td>≈ 25 minutes</td>
<td>≈ 25 minutes</td>
<td>≈ 25 minutes</td>
</tr>
</tbody>
</table>

Table 5.9.4
5.9.3 Results
The results from minimizing the weighted compliance can be seen in the figure 5.9.3.

![Figure 5.9.3](image)

Figure 5.9.3 It is seen that all the red colored elements and violated areas are gone and only a couple of few elements are close to violating the stress demand are left. These elements are orange colored.

5.9.4 Conclusion
From the results it is confirmed that several and large areas can be chosen to perform only one shape optimization. The conclusions and suggestions made in section 5.8.1 Evaluation of the size of the design area are valid and correct.
6 Summary
The thesis aim was towards developing strategies in the area of structural optimization and to implement these strategies in design processes. This thesis can act as a guideline for future projects within GM Powertrain Sweden.

The main tasks have been to perform a topology optimization of a model early in a design process, and a shape optimization on a model late in a design process. In addition the shape optimization strategies have also been applied on a fork shifter. The thesis covered the theory of different optimization strategies in general. The optimization processes were explained in detail and the results from the structural optimization of the differential housings as well as the fork shifter are shown and evaluated.

**Suggestion:** The evaluation of the thesis provides enough arguments to suggest an implementation of the optimization strategies in design processes at GM Powertrain. Following is recommended for future work:

1. Use topology optimization early in the design process to create new design concepts.
2. Use shape optimization when it is needed later on in a design concept.
3. Create standard works of how to use optimization strategies in design processes.
References

1. Anders Klarbring & Peter W. Christensen, An Introduction to Structural Optimization. 2007

2. Karl-Olof Olsson, Maskinelement. 2006

3. Institutionen för Hållfasthetslärare KTH, Bengt Sundström och övriga författare, Handbok och formelsamling i Hållfasthetslärare. 1998


6. GM Powertrain Sweden, Confidential reports.
Appendix 1
Appendix 3
Appendix 4
Appendix 5
Appendix 7
Appendix 8