Topology Optimization of Vehicle Body Structure for improved Ride & Handling

Emil Norberg
Sebastian Lövgren

Teknisk Mekanik

Spring semester 2011

Examensarbete
Institutionen för ekonomisk och industriell utveckling

ISRN LIU-IEI-TEK-A--11/01158—SE
Abstract

Ride and handling are important areas for safety and improved vehicle control during driving. To meet the demands on ride and handling a number of measures can be taken. This master thesis work has focused on the early design phase. At the early phases of design, the level of details is low and the design freedom is big. By introducing a tool to support the early vehicle body design, the potential of finding more efficient structures increases. In this study, topology optimization of a vehicle front structure has been performed using OptiStruct by Altair Engineering. The objective has been to find the optimal topology of beams and rods to achieve high stiffness of the front structure for improved ride and handling. Based on topology optimization a proposal for a beam layout in the front structure area has been identified. A vital part of the project has been to describe how to use topology optimization as a tool in the design process. During the project different approaches has been studied to come from a large design space to a low weight architecture based on a beam-like structure. The different approaches will be described and our experience and recommendations will be presented. Also the general result of a topology-optimized architecture for vehicle body stiffness will be presented.

Keywords: topology optimization, size optimization, compliance, design space, volfrac, Volvo cars, body structure, body stiffness, struts, beams, bolted components, Altair, hyperworks, hypermesh, optistruct, stepwise optimization, early design phase, benchmarking
# Table of content

Abstract .......................................................................................................................... I
Table of content .............................................................................................................. II
Preface ............................................................................................................................... II
Acknowledgments ........................................................................................................... V
1 Background & Purpose .............................................................................................. 1
  1.1 Objectives .................................................................................................................. 1
2 Theory ........................................................................................................................... 2
  2.1 Structural Optimization ............................................................................................ 2
     2.1.1 Optimization formulation .................................................................................... 2
     2.1.2 Problem formulation .......................................................................................... 3
     2.1.3 Three types of structural optimizations ............................................................... 3
         2.1.3.1 Size optimization ....................................................................................... 3
         2.1.3.2 Shape optimization .................................................................................... 4
         2.1.3.3 Topology optimization .............................................................................. 4
             2.1.3.3.1 SIMP method .................................................................................... 4
             2.1.3.3.2 Sensitivity analysis .............................................................................. 5
             2.1.3.3.3 Compliance and Stiffness ................................................................... 5
             2.1.3.3.4 A topology optimization example ......................................................... 6
3 Prerequisites ................................................................................................................ 7
  3.1 FE-model .................................................................................................................. 7
  3.2 The load cases ......................................................................................................... 8
  3.3 HyperMesh/OptiStruct ........................................................................................... 11
4 Evaluation of current model ....................................................................................... 12
5 Topology optimization ................................................................................................. 13
  5.1 Design space ............................................................................................................ 13
     5.1.1 Connection areas .............................................................................................. 13
     5.1.2 Geometrical restriction ..................................................................................... 13
     5.1.3 Creating the design space .................................................................................. 14
  5.2 Optimization parameters ......................................................................................... 14
     5.2.1 Manufacturing constraints .............................................................................. 14
     5.2.2 Other parameters used ..................................................................................... 15
  5.3 Characteristics of the current optimization problem .............................................. 15
     5.3.1 Large design space ............................................................................................ 15
         5.3.1.1 Influence of low dense element ................................................................. 16
         5.3.1.2 Is beams the optimal use of material? ......................................................... 17
         5.3.1.3 Conclusion ................................................................................................ 18
     5.3.2 Stepwise optimization ....................................................................................... 18
         5.3.2.1 The step method ....................................................................................... 19
         5.3.2.2 Example and evaluation of the method ....................................................... 20
         5.3.2.3 Conclusions of the step method ................................................................. 22
     5.3.3 Stability of topology .......................................................................................... 23
     5.3.4 How to interpret the results ................................................................................ 24
     5.3.5 Load case characteristics .................................................................................... 25
         5.3.5.1 301 .............................................................................................................. 26
         5.3.5.2 303 .............................................................................................................. 26
         5.3.5.3 304 .............................................................................................................. 27
         5.3.5.4 331-332-334 .............................................................................................. 27
         5.3.5.5 341-342-344 .............................................................................................. 28
6 The realization ............................................................................................................. 29
6.1 Realization method .................................................................................................................. 29
  6.1.1 Topology to beams .............................................................................................................. 29
  6.1.2 Verification of the realization method ................................................................................ 31
  6.1.3 The size optimization ........................................................................................................ 32
  6.1.4 Problem formulation ......................................................................................................... 32
  6.1.5 Topology interpreted into beam elements ........................................................................ 33
  6.2 Beam interpretation per load case ....................................................................................... 34
    6.2.1 Load case 301 .................................................................................................................. 34
    6.2.2 Load case 303 .................................................................................................................. 34
    6.2.3 Load case 304 .................................................................................................................. 35
    6.2.4 Load case 331, 332 and 334 .......................................................................................... 35
  6.3 Realization results .................................................................................................................. 36
    6.3.1 Beams optimized for load case 301 ................................................................................ 36
    6.3.2 Beams optimized for load case 303 ................................................................................ 37
    6.3.3 Beams optimized for load case 304 ................................................................................ 38
    6.3.4 Beams optimized for load case 331, 332 and 334 .......................................................... 39
  6.4 Suggested design .................................................................................................................... 40
    6.4.1 Design constraints .......................................................................................................... 40
    6.4.2 Optimization constraints ............................................................................................... 40
    6.4.3 Realization of topology results ........................................................................................ 41
    6.4.4 Evaluation of new design ............................................................................................... 42
 7 Discussion .................................................................................................................................. 43
    7.1 Sources of decrease in performance .................................................................................... 43
      7.1.1 Topology Optimization ................................................................................................. 43
      7.1.2 Realization .................................................................................................................... 43
    7.2 Interpretation of topology results ........................................................................................ 44
    7.3 Introducing topology optimization in the design process .................................................. 44
 8 Conclusion .................................................................................................................................. 45
  8.1 Recommendation to Volvo Cars ............................................................................................ 45
    8.1.1 Starting slow ..................................................................................................................... 45
    8.1.2 Optimization during and before the design process ......................................................... 45
  8.2 Future Work ............................................................................................................................. 45
 9 References .................................................................................................................................. 46
Preface

This is a thesis project conducted by two master students at Linköping University and this is the final assignment before an examination in Master of Science in Mechanical Engineering. The work has been performed and financed at Volvo Cars Corporation and co-financed by Altair. The supervisor at Volvo Cars was Ph.D. Harald Hasselblad and the examinator at Linköping University was Prof. Anders Klarbring, who is head of the Division of Mechanics at the Department of Mechanical Engineering.

Acknowledgments

We would first like to thank our supervisor Harald Hasselblad for the never-ending interest and support in our work. We appreciate the help and assistance given by the other co-workers at Volvo cars. We would also like to express our gratitude to Altair, especially Pär-Ola Jansell and Altair’s support group in Lund for providing us with their software and help us with technical problems. We would give an enormous thank to our beloved girlfriends who supported us at home in the cramped apartment.

Emil Norberg & Sebastian Lövgren
1 Background & Purpose

The car developers are in a hard competition against each other with making the cars lighter for both economical and environmental reasons. Unfortunately, a weight-reduction of the vehicle body will most often lead to a reduction in stiffness, thus giving a negative effect to the cars ride and handling. A car's structure must be stiff enough so it does not wiggle when it is driven in a tough course. The challenge is to remain, or even improve, stiffness, while reducing the body weight. Many car developers have chosen to stiffen the vehicle body with well-placed rods/beams. How to positioning and dimensioning these beams optimally is individual for each car's structure and the way of doing it has until today been done by comparison between competing cars and experienced designers intuition. New tools, like topology optimization, are rarely introduced in the design process of new car models. The designers are constantly under a tough time schedule and to perform optimization on every part often seems to be too time consuming. Another contributing reason is the widespread lack of knowledge and experience in the area.

This thesis will try to address how to perform an early stage design of components with topology optimization and without consuming too much of a construction designer's precious time. Topology optimization could be very useful for a designer and also for CAE engineer. This thesis work will focus on the early design phase, where the level of details is low and the design freedom is big. By introducing a tool to support the early vehicle body design, the potential of finding more efficient structures increases. The results provided by this tool could have the possibility to validate present design, reveal structural weaknesses or even come up with a totally new solution.

1.1 Objectives

The main objective of this thesis is to generate a proposal for a beam layout in the front structure area of a vehicle body, based on topology optimization using the software OptiStruct by Altair Engineering. A realization of the optimized topology will be performed to verify the performance and give the design teams some hints for future application of the results. How to use this tool as a natural part of the design process will also be analyzed.
2 Theory

2.1 Structural Optimization

Structural optimization is used to retrieve an optimum solution to a design problem. Before solving an optimization problem, an optimization formulation has to be made and a model for calculation created.

There are three main types of structural optimization: topology, shape and size. All three may have their purpose in a design phase and how to combine or chose between them depends on what is desirable to achieve.

2.1.1 Optimization formulation

A structural optimization requires an optimization formulation. This should include; design variables, response, constraint and an objective function.

Design variables,

The design variables might be anything that influences the performance of the investigated structure. The number of design variables may vary from the minimum of one to several millions and the structural properties will be dependent on the design variables.

Response,

For any given values of the design variables a response from the structure might be retrieved. Common types of responses are displacement, stress, mass and compliance. Responses are used to evaluate the performance and properties of the structure.

Objective function,

The objective function is a response used to decide how good a design is. In an optimization formulation, the objective function is usually to be minimized (minimization problem). Due to this, only one objective function is allowed per optimization formulation. Although, it do exist multi-objective functions, which are a functions of several responses. Typically used objective functions are mass and compliance. A definition of compliance will be made in chapter 2.1.3.3.

Constraints

Each design variable and response (except ) might be constrained with an allowed minimum and maximum value. Constraints are used on the response and design variables to make sure that the properties of the structure are in an allowed interval. Typical constraints are maximum allowed mass, stress, displacement or minimum allowed stiffness or dimensions.

A constraint that always has to be fulfilled is the static equilibrium equation (1)

For more information about the equilibrium equation, please advice: “Torstenfelt, Bo; Finite Elements – from the early beginning to the very end”, LiU-IEI-S—08/535—SE, 2008”
2.1.2 Problem formulation

From the formulation parts in chapter 2.1.1, a general structural optimization problem might be set up as follows:

2.1.3 Three types of structural optimizations

2.1.3.1 Size optimization

In size optimization, the design variable is some type thickness of the structural parts, typically sheet thickness or cross section area of bars/beams. An example of a size optimization problem is shown in Figure 1 where the cross section area for each bar is a design variable. This optimization needs a pre-defined structure where the size of its members should be optimized.

Optimization formulation

The size optimization in Figure 1 is a good example to illustrate the optimization formulation. The cross section area of each bar in the initial truss is denoted $A_i$ and the length of each bar $l_i$, where $i$ is the identification number of the bar. The cross section of each bar in the new design is denoted $A'_i$. The compliance of the structure is denoted $C$. With the assumption that all bars is made of the same material, the optimization problem is formulated from (2) as (3).
2.1.3.2 Shape optimization

In shape optimization, the design variable is the shape functions of the boundaries for the structure. No new boundaries can be created, i.e. in a 2D case no new holes can be created. In the example shown in Figure 2, the shape function is denoted \( \phi \) and the function that minimizes the mass with the same stress level is considered as the optimum. Shape optimization will not be used in this project; however it is important to know that it exists to increase the available tools when working with structural optimization.

2.1.3.3 Topology optimization

Topology optimization is the most general branch in structural optimization. When using topology optimization it is desired to find where in the design space it is optimal to place material. The design space is a geometrical space where material placement is allowed. A design variable in topology optimization has in theory a discrete value (material or void), and every point in the design space is a design variable. In theory this means that any structural shape can be achieved within the design space. In practice, this is not the case because of the FE-discretization that has to be made for computational reasons.

2.1.3.3.1 SIMP method

An optimization that only allows discrete design variables (material or void) is unfortunately not a realistic alternative when dealing with large numbers of design variables. Because of this, an continuous measure of material existence, \( m \), is introduced. Where \( m = 0 \) is interpreted as void and \( m = 1 \) as solid material. Value in-between might be interpreted as a material with a lower density and Young's modulus. The introduction of a continuous measure of material presence in an element is done for computational reason; however it is still desirable to have discrete design variables (material or void). One way to make the design variable more discrete is to use the SIMP method (Solid Isotropic Material with Penalization). The idea of SIMP is to penalize the stiffness of elements with intermediate values of \( m \). This is implemented in the computation by replacing the Young's modulus for an element, \( E \), with an effective Young's modulus \( E_m \).

\[
E_m = E \left( 1 - (m - m_0)^p \right)
\]

(4)

The variable \( p \) in (4) is a penalization parameter and influence of this parameter is visualized in Figure 3. An increase of \( p \) lowers the stiffness of elements with intermediate values of \( m \). Low stiffness elements are not a favorable way to use material and thus, by increasing \( p \), a more discrete result is received.
2.1.3.3.2 Sensitivity analysis
The optimization is an iterative process, what is included in the process will be discussed later. During the iterations, the design variables are changed until a solution to the optimization problem is found. To decide how to alter each design variable between the iterations, sensitivity analyses are made. In this analysis, the gradient of the response variables is calculated with respect to the design variables. From this result, information in what way and how much they will be affected by an increase or decrease of the value of each design variable are obtained. No in depth mathematical description of this will be made in this report, for more information about this please advice: “Klarbring, Anders; W.Christensen, Peter; An Introduction to Structural Optimization, ISBN 978-1-4020-866-6, 2009”

2.1.3.3.3 Compliance and Stiffness
It is often desirable to receive a stiffness response of a structure when doing an optimization. With OptiStruct one may use compliance, \( C \), to evaluate stiffness. Physically this can be interpreted as a measure of strain energy in the structure. In OptiStruct compliance is calculated from the equilibrium equation (5).

\[
C = \frac{1}{E} \int \sigma : \varepsilon \, dV
\]

(5)

In a case with prescribed load, a structure is said to be stiffer when the displacement is low, thus making minimizing compliance equivalent to maximizing stiffness. When prescribed displacement is considered, a structure is considered stiffer if the force to achieve that displacement is high, thus maximizing compliance corresponds to maximizing stiffness.

Prescribed loads is used in this project, this means that a lower compliance means a stiffer structure. When measuring performance in this project, compliance change to a reference will be used. What the used reference is will be discussed later.
2.1.3.3.4 A topology optimization example

An example of a topology optimization is shown in Figure 4. The objective function that is to be minimized is the compliance of the structure. The design variables are the material presence in each element. Except the equilibrium constraint, the allowed amount of material is limited to maximum 30% of the design space.

![Figure 4 A topology optimization example](image-url)
Chapter 3 Prerequisites

3 Prerequisites

3.1 FE-model

The model that has been used in this study is the S80-BIG (BIG: body in grey, a body structure including the window screen). The vehicle body does only include the body framework which clearly contributes to the body stiffness. The model can be seen in Figure 5.

![Figure 5 Body In Grey model of the front of a S80 with naming of important components.](image)

Due to the complexity and the size of a whole vehicle body the calculation time will be unnecessarily large. Also because this thesis only focuses on the front structure, the whole vehicle body has been cut behind the A pillar.

A body structure is mainly built up with metal sheet and seamed with spot welds and glue. The attachment type and the material of the structure vary from component to component to fulfill their individual function. This will lead to difference in stiffness between the parts of the structure. Because of the asymmetric engine and the fact cars might be right or left hand steered; the front structure does not have any symmetry plane. These asymmetries together with the material differences will affect the topology results.
3.2 The load cases

The chosen load cases for this study are somewhat standard within the car industries. Some are taken from a real scenario and other are used to measure a general stiffness. Since the natural way for the forces to travel into the body structure are through the wheels and wheel suspensions, load cases associated with ride and handling will therefore most often be applied on the spring towers and the spring suspensions attachment points.

All load cases are static linear, which means that no consideration has been taken to dynamic load cases, e.g. crash impacts, NVH (Noise, vibration and harshness) etc. At the cutting line of the A pillar, the end points are fixed in all degrees of freedom. This can be seen as the blue line in Figure 6. This boundary condition is applied for all the load cases.

![Figure 6 Boundary conditions of the model](image)

The following load cases are named by numbers and they will further on in the report only be referenced as those numbers.

301 - Torsion stiffness

Two vertical forces, one on each spring tower, are acting in opposite direction. The forces act on rigid components which are corresponding to the upper connection of the wheel suspension. The rigid components are then bolted to the spring towers.
303 - Lateral stiffness
The horizontal load couple is acting on the connection between the upper bumper and the side rail.

304 - Upward bending
This vertical load couple is, like 303, acting on the connection between the upper bumper and the side rail.

331 - Upper wheel suspension LH
On the left hand side, one load is acting on the upper wheel suspension connection and is directed into the body.

332 - Upper wheel suspension RH
On the right hand side, one load is acting on the upper wheel suspension connection and is directed into the body.

334 - Compressing load on both suspension tower
Combining load case 331 and 332 to achieve a compressing load case on the structure thru the spring towers.
341 - Lower wheel suspension LH

On the left hand side, a load is applied at the connection point of the lower wheel suspension. The force is directed into the body.

342 - Lower wheel suspension RH

On the right hand side, a load is applied at the connection of the wheel suspensions lower part. The force is directed into of the body.

344 - Sub frame compression

Two forces, in opposite direction, are acting on the lower attachment of the wheel suspensions.
3.3 HyperMesh/OptiStruct

This thesis work is co-financed by Altair, which has developed one of the markets leading optimization solver named OptiStruct. OptiStruct has the advantage of being capable to solve the major kinds of optimization problems. It is able to solve problems with millions of design variables, which this thesis will be dealing with. The finite element pre-processor used in this work is HyperMesh, also a product by Altair. HyperMesh is easy to learn and has an interface environment integrated with OptiStruct.

OptiStruct

Solving an optimization problem using computer is most often an iterative process. OptiStruct is no exception and its iterative procedure will briefly be explained with the OptiStruct manual as a reference. The iterative workflow can be seen in Figure 7.

![Figure 7. The iterative process used in OptiStruct](image)

1. First step is an analysis of the FE-model to retain the design’s properties and behavior.

2. Second step is to check the results for convergence. If satisfied, the iteration will terminate and the current design is considered as a solution. Convergence is attained when two subsequent iterations satisfies the convergence criteria e.g. the object function change is less than the object tolerance and the constraints are not violated more than 1%.

3. The optimization problem will then go thru a sensitivity analysis where information how the design variables should be altered to minimize the objective function are retrieved.

4. The design variables will then be given new values and the process will then go back to step 1 and a new iteration begins.
4 Evaluation of current model

The first step in the optimization process is to create a reference so that a beam design can be evaluated. In this project, the car model already contains a beam layout which means that a very clear reference exists and will be used to evaluate new designs. Mass and compliance are considered when discussing the performance of a beam design. An increase in performance means that the weight and/or compliance for all load cases are lower than the original design and not higher. Besides the original design, two additional designs are evaluated. The additional designs and the reasons of the evaluation are:

- **Current design in aluminum.** Done due to that the material used in this project is aluminum.
- **No beams.** All beams are removed to see how the stiffness for each load case is affected

The stiffness evaluation of the two designs are made by calculating the compliance improvement, \( \Delta C \), compared to the original design (reference design).

\[
\Delta C = C_{\text{evaluated}} - C_{\text{front without beam}}
\]

is the compliance of the evaluated design and the compliance for a front without beam. The result can be seen in Figure 8.

The mass evaluation is made by calculating the weight of the beams in the original design with original material choice and the case where all beams are in aluminum. The weight is shown in Table 1.

<table>
<thead>
<tr>
<th>Material</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>2.7 kg</td>
</tr>
<tr>
<td>Aluminium</td>
<td>1.2 kg</td>
</tr>
</tbody>
</table>

Table 1 Weight of original beam design
5 Topology optimization

5.1 Design space

5.1.1 Connection areas
To prevent using an unnecessary large design space, the allowed connection surfaces on the structure should be decided first. Input from departments such as NVH and safety should be heavily considered. For NVH it is important that the attachment surfaces used are not of such nature that vibrations are transmitted into the coupe and reduces the comfort of the driver. For safety reasons it is important that components that is designed to have specific deformation behavior during crash are not changed, an example of this is the crash box where no material is allowed to be attached. The connection surfaces used is shown in Figure 9.

![Figure 9 Allowed connection surfaces](image)

5.1.2 Geometrical restriction
Since this work is on the BIG model, no consideration will be taken to cables, containers, battery and other components that do not contribute to stiffness. The basic idea is that topology optimization should be done during the design phase, where the placements of such components are not decided. However, components that are hard to reposition is considered as a restriction. The two components that are considered as restrictions in this project are the cooler and the engine (with safety margin for vibration). In addition to space restriction from components, the boundary of the car is also taken in consideration, i.e. all beams are to be inside the front. The upper boundary is prescribed from the pedestrian surface for pedestrian safety reason. The restriction used is visualized in Figure 10.

![Figure 10 Geometrical restriction of the design space](image)
5.1.3 **Creating the design space**

When connection surfaces and geometrical restrictions are decided, it is possible to define where material is allowed to be placed and attached. With this information, a design space can be created. In this project the design space was created from the FE model since no geometry model was available. To create the design space from the FE model in HyperMesh is a slower procedure compared to use a geometry model in a suiting environment. The reason is that geometry is more flexible and easier to alternate and modify than elements and nodes. The resulting design space is shown in *Figure 11* and is an enclosed volume mesh within all geometrical restriction.

![Design space](image)

*Figure 11 Visualization of the design space used*

5.2 **Optimization parameters**

5.2.1 **Manufacturing constraints**

There are several types of constraints to use in OptiStruct to control the manufacturability of the final result. This may include constraints on minimum and maximum member size, symmetry, draw direction and more. The two constraints used in this project are minimum dimension and symmetry. When applying these constraints to an optimization formulation, the number of designs gets limited and it is to be remembered that the performance of a solution with more constraints cannot be higher than a solution with less.

**Symmetry constraint**

This is used to receive a topology that is symmetric in one or more planes that are defined in the optimization set up. This constraint should be carefully used because it may prevent the solution to compensate for an unsymmetrical stiff structure. When mentioning symmetry constraint in this report, left hand – right hand symmetry is considered.
Minimum member size.

In 2D, the member size is defined as the width of a member in the topology. This is a useful constraint in this project since it prevents thin none realizable members to form. An example is seen in Figure 12.

![Figure 12 Minimum member size example](image)

5.2.2 Other parameters used

**MINDENS**

MINDENS is the minimum element density that is allowed. According to the OptiStruct manual, very low values will increase the risk of numerical problems. The default value in OptiStruct is 0.01.

**MATINT**

The element density for initial design space is defined through MATINT. Element density is homogenously distributed in the initial design.

**DISCRETE**

This is used to control the penalization factor in the SIMP method.

**OBJTOL**

OBJTOL is the relative convergence criteria. If a relative change in the objective function between two design iterations is less than OBJTOL then optimization is considered to have converged. This parameter is used to control the number of iterations and the influence of this parameter to the number of iterations until convergence is individual for each problem. Normally in this thesis, OBJTOL=5E-4.

5.3 Characteristics of the current optimization problem

5.3.1 Large design space

A solution that occupies less than 1% is to be found in this project to be able to replace the original design without a large increase in weight. Because of this, it is expected that the beams that should be found are very small relative to the design space. One might understand that this increase the importance of using small element size. Topology optimization is a computational expensive procedure which leads to that a limit on mesh resolution has to be set to get reasonable computational time. These two points of interest are in direct conflict.

Due to the limitation on mesh resolution, the topology optimization will have problems finding thin walled beams; more expected are solid beams which make them more expensive from a material usage point of view than what would be necessary in reality. The consequence of is this is that a direct realizable result is not to be expected when running a topology optimization under these conditions.
Since the topology optimization will not be able to find optimal cross sections in this project, a good idea is to not focus on the performance and weight of the topology result. A suggestion is to treat the results as a guide for beam placement. Beam dimensions will be decided later in chapter 6.1.3. Awareness of how to interpret the mass and the stiffness of topology results must also be taken, this will be discussed later.

5.3.1.1 Influence of low dense element

As previously discussed in chapter 2.1.3.1.3, material presence in a topology result is not discrete but continuous. The effect of this is that even if the user interprets the areas in the design space with the lowest material density as a void, it is not. The low dense material one might want to think of as void has both mass and stiffness. This will of course influence the performance and properties of the structure. To see the effect of this, the compliance for the two models in Figure 13 are calculated and compared for all the load cases. The compliance for the low density design space model is denoted \( C_{low} \) and for the empty model \( C_{empty} \). Both \( C_{low} \) and \( C_{empty} \) is the sum of the compliance for all load cases presented in this project. The minimum element density used in the design space is chosen as the OptiStruct default value or lower, a value that is would be considered as a void when interpreting the topology results. The results can be seen in the table below.

![Low density model and Empty model](image)

<table>
<thead>
<tr>
<th>Element density, ( \rho )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Low density model</td>
</tr>
<tr>
<td>Empty model</td>
</tr>
</tbody>
</table>

**Figure 13 Models for investigating influence of low dense elements**

What should be concluded from this experiment is that what an observer might interpret as a void in the design space contributes significantly to the stiffness of the structure. For a topology result interpreter, these two models correspond to the same case, an empty front. Although, in the computations, one is more stiff then the other. This effect will of course increase when the volume of the design space gets bigger in relation to the volume of the material that is allowed in the result. How this influence the topology optimization set up will be discussed later.
5.3.1.2 Is beams the optimal use of material?

It is clear from the problem formulation that the desirable result should consist of beams. This is unfortunately hard to implement as a constraint in the optimization formulation. If it is better to use the available material to reinforcements or plates than to beams, no beams will occur. Beams are expensive from a material usage point of view, especially when thin walled beams have difficulties to appear due to mesh resolution. In a case where only local reinforcement occurs, there are two intuitive actions to take.

1. Remove the connection surfaces where the reinforcements are to prevent them from appearing.
2. Increase the allowed material to increase the possibility for beams to appear.

If one choose the first option, it is hard to know in advance if new reinforcements in different areas will appear in the next optimization. There is a risk that the user gets stuck in a loop of time consuming manually work and thus this is not recommended. If one use option number two and find a suiting volume fraction, the user will receive a result where both local reinforcements and good placement of beams are visible. Both these types of material usage are good for increase the knowledge of the structure and are valuable as an input to designers. It is to be remembered that if a beam structure is found after a lot of material is spent on reinforcements, the beams are not found on the original structure but on a reinforced structure with different properties. The beams therefore might lose their performance if not realizing the reinforcements in the final design. In a later chapter, a method will be suggested how to make topology optimization where a gradual decrease of material usage is used and where reinforcements are prevented during this decrease. An example of a topology result for different material use is shown in Figure 14.

![Figure 14 Example of topology for different material amount used](image)

*Optimization set up is to minimize compliance for load case 301 with a minimum allowed member size of 25mm. Elements with $\rho \geq 0.1$ is shown.*

For this specific example, beams stop appear when allowing somewhere between 2.5% and 1% of the design space to be filled with material. Even if the case where 2.5% of the design space is used is a somewhat beamlike structure it is hard to realize due to the amount of material and the complexity of the structure between the GOR and bumper. It is impossible to guess what material usage (if any) that will give a somewhat clear and realizable beam structure.
5.3.1.3 Conclusion

For the reasons recently discussed, it is clear that an optimization set up should not be made with real performance constraints. The first reason is that the topology result will be stiffer than in reality due to the contribution of the low dense elements. The second reason is that the topology result most often will have a higher weight than needed due to problem finding thin members such as thin walled tubes instead of solid beams.

For the case where the design volume is large compared to what is allowed to be realized, one should treat topology optimization as tool to create an understanding of the structure. The focus should be to find load paths and characteristics of the structure and not as a tool to find direct realizable results.

5.3.2 Stepwise optimization

A method was used in this project to make the topology optimization process as systematic as possible when the design space is large compared to the desirable result. The method will be called “step method” and it is used to increase the control of the optimization process. This is the method that was used in the project to find the placement of beams for each load case.
5.3.2.1 The step method

The idea of the step method is to instead of reducing the design space to a realistic weight in one optimization run (direct method); it is done in several steps. This is to have control of the structure development, that is, in a systematic way find a beam structure that solves the optimization problem.

Optimization run

An optimization is made with a design space, shown in green. Besides from the first run where the original design space is used, a design space is created from a topology result.

Retrieving a topology result

A topology result is retrieved from the optimization run. Elements with density above 0.1 are considered to be a part of the solution. The result is shown in blue.

Removing of isolated reinforcements

All elements that includes in a beamlike structure are kept. The remaining elements considered to be local reinforcements (are shown in red) and will be removed.

Creating the new design space

When reinforcements are removed, a new design space is created from the remaining elements using the shrink wrap tool. This design space has a lower element size than the design space it was created from, but approximately the same number of elements. This design space is used for a new optimization run.

This procedure is repeated until a clear beam structure appears. To create a new design space from a topology result can be done within a matter of minutes. In this project it usually took about 30 minutes to get a new optimization running with a design space created from a previous optimization result. The removal of isolated reinforcements is off course wrong from an optimum point of view since solutions are removed. However, the removal can be motivated by the fact that it is beams that are of interest, not local reinforcements.
5.3.2.2 Example and evaluation of the method

A simplified model and the optimization set up

A simple 2D model will be used as an example in this chapter instead of the full 3D model; this is due to computational time and pedagogical reason. A sketch of the 2D model is shown in Figure 15. The element in the strong frame is twice as thick as in the weak frame. This is to introduce an unsymmetrical stiffness, a property the investigated car model has.

A problem formulation is created such that it is desirable to minimize compliance with 7.5% of the design space filled with material. In this model, it is possible to achieve a beamlike structure with one optimization run that satisfy the material usage constraint. With this result as a reference, the step method will be evaluated.

![2D model used to evaluate the step method](image)
Step method compared to direct method

Three optimizations are done in parallel; one where a solution is found in one optimization run (direct method), one made with one step and one where three steps are made. An overview of the performance and topology is illustrated in Figure 16.

Figure 16 Overview of step method compared to the direct method
The final result (≈7.5% of design space is used) for each of the three optimizations are shown in Figure 17. Elements with a density above 0.1 is considered to be a part of the solution.

![Figure 17 Topologies for the direct method (a) and step method with one step (b) and three steps (c)](image)

5.3.2.3 Conclusions of the step method
The step method would be correct to use from an optimum point of view if a solution with a certain amount of material would contain all the solutions with lower amount of material. This is not the case.

Results from three topology optimizations by the direct method are shown in Figure 18. The results are stacked on top of each other by order of material use, highest amount of material on top. The blue topology with highest amount of material does not completely cover the solutions with lower amount of material. This is the reason for the difference in topology of the three solutions in Figure 17.

![Figure 18 Illustration of the problem with step method](image)

To be able to draw exact conclusions of the method from this test is not possible. It is expected that the direct method will lead to a better result than the step method from a compliance/mass point of view. In this case, the direct method leads to a slightly better use of material but a much more intuitive and convincing topology. This is also expected. In the step method, the design space will gradually get smaller and new holes might be introduced within each step. This removes possible paths for the material from one attachment to another and thus small and complex members might appear to compensate for this.

In this simple example it is shown that the method seems reasonable to use from a performance point of view. The performance between the direct and step method is relatively small. However,
from a topology point of view, as a support for a designer, it might not accomplish what is desirable. The topology of the solution from the step method might be unnecessary complex which leads to realization problems and might not be convincing. The step method will be used when trying to find load paths for each load case. The method increase the chance to find a beam structure and not a lot of experience of topology optimization or OptiStruct is needed to perform this kind of stepwise optimization. If more knowledge and experience are available for this kind of optimization problem and OptiStruct, it is very possible that the direct method can be used.

5.3.3 Stability of topology

It may be understandable that from a topology point of view, one problem might have several different solutions with similar performance. This can be seen as a problem and the pessimist might doubt the use of topology optimization due to that different answer are acquired from what appears to be the same question. An optimist may see this as an opportunity to receive multiple solutions to a problem. If topology optimization is used as a guide for designers, the topology characteristics might be of more interest than exact values of the performance. This is the case in this project and consequently one might ask how sensitive the topology is to changes in optimization set up. This investigation is of course not general, it is more done to get a basic understanding what might influence the topology and in what way.

Topology optimization is made on the 2D model in Figure 15 where the compliance is minimized with mass constraint. The compliance from these optimization is used for a new optimization where mass is minimized and the compliance is constrained. An illustration of the process is shown in Figure 19

![Figure 19 Sensitivity analysis of optimization set up](image)

The two sets of solutions shown in Figure 20 might both be a solution to the problem from a practical mass-compliance point of view. However, the problem is formulated differently in the two cases and the topology differs between them. In reality, both of these results might satisfy the demands on compliance and mass. One solution might suit better than the other from a practical point of view, but their performances are approximately equal. It is important to have in mind that it might exist solutions with similar performance but different topology.
5.3.4 How to interpret the results

When analyzing the visual results from the 3D-topology optimization one must not misinterpret the iso-levels in the result. As explained in chapter 2.1.3.3.1, the element densities cannot reach zero. Therefore, one is forced to sort out those low dens elements to retain a proper view of the topology result. Like it is shown in Figure 4, there will be a fuzzy transition from low to high dense elements. To get a desirable view, elements with densities lower than a certain value (normally ~0.1) are filtered out. It is important that one does not filter out too much so that important load paths get excluded. In Figure 21 it is shown how different filtering affects the view of the result. Note that in the model with no filter in Figure 21, what is shown as red is solid material reaching the boundaries of the design space.
Figure 21. Example of different filter settings. Blue color indicates low-dense material (interpret as no material). Red indicates solid material (actual structure).

One should also keep in mind that internal structures are not shown. Bigger chunks of topology structure could very possibly be hollow.

5.3.5 Load case characteristics

It is chosen in this project to optimize each load case separately; this is done for two main reasons. First, these load cases are used as a standard measure of vehicle body stiffness. It is beneficial to create a knowledge how the structure should look like to make the vehicle stiff for each of these load cases. This knowledge can be used in the future if one wants to increase the performance for a specific load case.

Secondly, it is very complicated to perform an optimization with respect to several load cases, especially if their acting point and magnitude are very different. When doing an optimization with respect to several load cases, it is necessary to weight their importance relative to each other and minimize the sum of the weighted compliance. This is a very hard task to perform. The weighting will greatly affect the topology, thus, the result will be heavily dependent on the complicated decision how to weight the load cases, and this is not desirable. The fact that no recommendation were given from Volvo how to weight the load cases and that it is problematic to use real performance constraints, it is chosen to optimize each load case separately.

The following subchapters will show the final topology solution for each load case. Also the previous steps will be presented. Note that the number of steps needed varies between the load cases.
5.3.5.1 301

Figure 22. Topology result for load case 301. Step 4 to the left and step 5 (final step) to the right

In Figure 22 the fourth step shows a very clear beam structure. Both the upper and lower part of the rear side structure is connected with the topology, creating some sort of deformed cross. This cross absorbs shear stress in the yz-plane and the same applies for beams created inside the cooler frame. The optimization will finally consider the deformed cross to be too expensive to save until the fifth and last step.

The first characteristics of the final result in this load case are the two separate beams that connect the spring towers with the GOR. The connection to the spring tower is supported with an additional beam. The second characteristics are a set of beam crossing the cooler frame. Two main beams are connected to the strong part of the sub frame link and they meet at the center of the GOR. These two main beams are stabilized with two minor, which goes from the center of the main beams to the sub frame. The set builds a net with 45 degrees angle. Reinforcements are created at the spring attachment bolt on the spring tower. The bolts attachments are particularly exposed to the loads on the spring tower whereas reinforcement is created.

5.3.5.2 303

Figure 23. Topology result for load case 303. Step 3 to the left and step 4 (final step) to the right

This load case, 303, corresponds to a test of general lateral stiffness with a side load on the upper bumper. The third step gives two beams from the spring tower to the GOR, similar to the beams in load case 301. In the final step, the topology reached for support from the GOR. A 45 degree net was created with two pairs of beams in an alpha-shape. The optimization did not favor any material from the GOR and back. It concentrates to stiffening the bumper with the side rail and the GOR.
5.3.5.3 304

Figure 24. Topology result for load case 304. Step 1 to the left and step 2 (final step) to the right

This load case was the most difficult one to handle. The vertical loads are exposed on a point at the upper bumper bolts and they are bending the cars whole front section. To resist the bending, the topology wants to form a frame of beams from the load source, straight back to the a-pillar. It uses the original structure whereas it reinforces weak sections and bridges the gaps. The weak points are especially between the corner of the COWL and spring tower and also between the spring tower and the GOR attachment. It is good to connect the bumper to the lower bumper. Also a triangle structure is formed from the GOR’s attachment, passing the side rail and upper bumper, down to lower bumper and the sub frame. Note that no materials are allowed to connect to the crash box.

This load case only had two steps because of the difficulties to receive beams. Further optimization step will give a much more tricky result to interpret.

5.3.5.4 331-332-334

Figure 25. Topology result for load case 331, 332 and 334. Step 2 to the left and step 3 (final step) to the right

This load case is a weighted (normalized) combination of 331, 332 and 334. The loads puts bending force on the upper spring towers and in turn its surroundings. This shows a weakness for bending at the “GOR attachment beam”. This bending is withstanded by an arced beam structure created below the GOR on both sides. There is a cut out at the right hand side rail where the engine is mounted and this load case reveals that this is a weak spot. This is compensated by a beam bridging the spring tower and the GOR attachment bracket. This beam connects to the spring tower with a tripod. A tripod connection distributes the forces over a larger area when the connection surface is weak.

A rally bar is also created between the spring towers and is attached on theirs the upper back corner. It is relative thick and straight and is the absolute optimum for load case 334 but it also add an important support for 331 and 332.
5.3.5.5 341-342-344

**Figure 26. Topology result for load case 341, 342 and 344. Step 3 to the left and step 4 (final step) to the right**

This load case is a weighted combination of 341, 342 and 344. They all impact the lower wheel mountings at the sub frame. The result shows a major distribution inside the cooler frame. The side rail put a big support on the sub frame by forming a V with a base on the symmetry line of the sub frame. The focus in these load cases is to prevent sideway motion of the sub frame. The sub frame is rigid connected to the body frame in the front but has a rubber mounting in the rear part.

Unfortunately the design space has no other connection surfaces on the sub frame except on the lower cooler frame. It is a relative long distance between the impact points of the loads and the closest contact surface. So whatever solutions retained the loads always have to travel thru a long section of structure before it reach a design space. This means that the topology only can achieve indirect effects by the supporting beams for this load case. The possibility to improve stiffness is therefore limited and because of this, the investigation of this load case stops here and will not be realized. If a beam layout for this load case should be purposed, the design space should be change so that a bigger impact on the sub frame can be achieved.


Chapter 6 The realization

6.1 Realization method

The results from a topology optimization on an architecture level will most often tend to be quite organic-looking and nothing a manufacturing engineer would approve to put in production. The beams surfaces are still somewhat undefined because of the fuzzy transition from low to high density. Also, the beams connections to the body structure are modeled as solid casted parts, without realistic connection points. The given results are therefore not applicable in reality and should be interpreted into something that is possible to manufacture.

A realization is also needed to perform a just comparison with the original design. Therefore another goal with thesis is to present a realizable suggestion of a new design. To get the right (optimum) dimensions of the beams, one will find a great opportunity here to use size optimization. This study will continue on a consistent path by letting develop the beams for each load case, see Figure 27 for a overview of the workflow.

![Figure 27 Overview of the workflow](image)

6.1.1 Topology to beams

The method to realize the bars is to interpret the topology bars into 1D beam elements (Bar2 elements in HyperMesh). They are 1st order elements with 2 nodes used to model axial, bending, and torsion behavior. The elements are manually drawn in the centre of the resulting beams from the topology optimization, see Figure 28.
Because the target of this thesis is to obtain beams, the local reinforcement, if they exist, will not be realized. The 1D-elements is given a cross-section (In HyperMesh referred as a property card named "pbeaml"). The optimal type of cross-section for each individual beam will not be investigated in this study. Though it is most probably not optimal, all beams will be given the same type of cross-section. The given profile is a circular hollow tube. The reason of that is because a visual interpretation of the topology results will in most cases give circular beams. However, it is not optimal to allow the beams to be solid. Material in the profile centers do not contribute to torsion and bending stiffness. These profiles are also justifiable in terms of manufacturing and practical implementation. Another advantage of this is provided when the beams are getting size optimized. Only two parameters are required when such profiles are defined and compared with other types of profiles, time can be saved when performing the size optimization. A more detailed description of the size optimization will be given in chapter 6.1.3.

Instead of develop special mounts for the beams; the attachment will consist of simplified 0D-elements. They are commonly known as RBE3-elements (Interpolation constraint element) and consist of one master node and several slave nodes. A point load on the master node is equally distributed to the slaves, just like pressure behaves. The accuracy of the approximation is addressed in chapter 6.1.2 and it is illustrated in Figure 29 below.
6.1.2 Verification of the realization method

Before starting with size optimization to realize the topology results, an evaluation of the 1D modeling used for this was evaluated. To use shell modeling with advanced connection to the frames for the realization would not only be very unpractical and time consuming, it would also be very inflexible with respect to cross sections changes. The cross section dimensions in the reference model were used to create a 1D approximation of the beams from the spring towers to the GOR. The 1D beam approximation model and the reference model are compared to see how well the 1D modeling works. The results are displayed in Table 2 below where the compliance of the original shell model, \( C \), and 1D model, \( C' \), is compared for each load case.

\[
\begin{array}{|c|c|}
\hline
\text{Load Case} & \text{Compliance} \\
\hline
\text{Case 1} & 0.01 \\
\text{Case 2} & 0.02 \\
\end{array}
\]

Table 2 Stiffness comparison between original model and simplified beam model

The difference between the two models is less than 1%, thus it is concluded that the method of realizing beams by 1D bar2 elements is sufficient for this purpose.
6.1.3 The size optimization

To retain the optimal dimensions of the beams, a size optimization was performed. Compared to the topology optimization, this optimization will give a numerical result with exact values of dimension and weight. As explained in chapter 6.1.1, the cross section is set to be hollow circular, defined by two parameters seen in Figure 30.

![Figure 30 The chosen cross section of the beams with the two design variables](image)

Design variables:

Response:

6.1.4 Problem formulation

Objective function:

Subjected to:

The starting values of is set to correspond to the reference model as much as possible if a similar beam exists in the original design. Entirely new beams are set to start from:

Boundaries for all the design variables are and are set as below

\[
\text{[mm]} \\
\text{[mm]}
\]

The other constraints are set as below

\[
\text{[mm]} \\
\text{[kg]}
\]
The design variables and constraints are bounded to be realistic for manufacturing and the maximum dimensions are inside the packaging restrictions.

The given problem formulation obtains some negative effects that one should be aware of. The first is that, which means that a beam cannot disappear completely, even if the beam do not contribute to minimizing the objective function. So, instead of retaining a zero-area cross section of a beam, the beam will have a radius, and thickness, .

The second effect of the problem formulation is that all beams contributing to the objective function will first reach to get maximum radius with a minimum thickness. Then if the beam needs more material it will increase the thickness. This behavior can easily be explained. A cross section, like the one used in this project, can have many solutions for a certain cross section area. Though the topology results in striving for tension and compression, it will unavoidably be exposed for bending and torsion when performing the realization. Tension and compression stiffness is only dependent on the cross section area, but torsion and bending resistance depends on the area of inertia. Thus, the optimum cross section, for a given area, will have the highest available area of inertia. Therefore, the material reaches to be placed as far from profiles centre of mass as possible. This behavior derives from the no existing buckling constraint, which would prevent beams with large radius and small thickness.

6.1.5 Topology interpreted into beam elements

As mentioned in chapter 6.1, the realizations were to be done separated for each load case. It is often desirable to minimize the amount of design variables. In this size optimization there will be two design variables for each beam property. To save work and calculation time symmetric beams are sharing properties. They will therefore have identical cross sections. The following four chapters show the interpretation of the topology results into beam elements. The different properties will be presented with a color code.

There is a strong will within the car companies to form the beam layouts to be symmetrical. It is preferred for a manufacturing point of view but it will also give a more thought-out and appealing appearance. Therefore, thought concerning symmetry has been included when designing the beams.
**6.2 Beam interpretation per load case**

6.2.1 **Load case 301**

![Figure 31 Topology and beam realization of load case 301](image)

In this case the topology result where followed to a great extent and four beam properties were set. The V-beam (301₁) is supported with an additional leg (301₂) at the connection to the spring tower. Even some asymmetries in the beam layout have been preserved, especially 301₄-beams, and may be observed in *Figure 31*. The two beams with property 301₁ do also show small asymmetries. They are both reaching for the middle of the GOR but are limited by the engine.

6.2.2 **Load case 303**

![Figure 32 Topology and beam realization of load case 303](image)

The 303 case were the most advanced result to realize because of the arced form of the net of beams in the front area. The archness of the net was preserved in the interpretation as well as the pair wise beams with the high crossing. The realization is made more symmetrical than the topology result. The structure at the lower bumper has not been realized because of the uncertainty of its influence and because it is wanted to minimize the number of beams to gain a simple and clear characteristic of the case. Two beams, one on each side, support the bumper with the front side member, forming isosceles triangles.
6.2.3 **Load case 304**

![Topology result](image1) ![Beam interpretation](image2) ![Beam property color](image3)

**Figure 33 Topology and beam realization of load case 304**

The beams in this load case follow the load path from the bumpers to the upper back side member. The net resulted from the topology around the crash box is simplified in this realization with a smaller cross connecting. The major asymmetry in this case is the $304_2$ –beam which connect the GOR with the spring tower on the right hand side and connect the GOR with the top of brace on the left hand side.

6.2.4 **Load case 331, 332 and 334**

![Topology result](image4) ![Beam interpretation](image5) ![Beam property color](image6)

**Figure 34 Topology and beam realization of load case 331, 332 and 334**

Most of the topology results have been translated into beam elements. Each leg in the tripod connection have been given a individual property, . The arced beam structures underneath the GOR, , have been simplified on both sides to a single straight beam. The GOR attachment has been strengthen by a beam, , on each side.
6.3 Realization results

In all size optimization cases a weight of all beams of each load case has been set to 2 kg. The weight is of less importance in this study. Since the internal grading of the beams is sought. However, a constraint on 2 kg is somewhat in between the weight of the original design with mixed material and with aluminum.

6.3.1 Beams optimized for load case 301

<table>
<thead>
<tr>
<th>Design variable</th>
<th>Inner radius (mm)</th>
<th>Outer radius (mm)</th>
<th>Thickness (mm)</th>
<th>Area (mm²)</th>
<th>Mass (kg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>16.84</td>
<td>20.00</td>
<td>3.16</td>
<td>731</td>
<td>1.14</td>
<td></td>
</tr>
<tr>
<td>18.47</td>
<td>20.00</td>
<td>1.53</td>
<td>370</td>
<td>0.13</td>
<td></td>
</tr>
<tr>
<td>18.25</td>
<td>20.00</td>
<td>1.75</td>
<td>421</td>
<td>0.65</td>
<td></td>
</tr>
<tr>
<td>4.74</td>
<td>5.74</td>
<td>1.00</td>
<td>66</td>
<td>0.08</td>
<td></td>
</tr>
</tbody>
</table>

Table 3 Beam design for load case 301

The most important beams may be decided by looking at the resulting mass of each beam property. In Table 3 Beam design for load case 301 it can be seen that beam 301 got the highest amount of weight, ~57% of the summed beam weight. The table also shows that the 301 beams (the small cross in the cooler frame) have a rather low importance. As explained in chapter 6.1.4 it is not possible for the cross area to reach zero. Therefore, one can assume that 301 beams reach to not exist. They have not reached the minimum radius, which could derive from a moderate decent in its sensitivity.

The additional leg, 301, for the 301 actually shows some degree of importance. Though its weight is relative low, the area of 301 is fairly average, which gives indication on force absorbance in these beams.

Evaluation of beams optimized for load case 301

Figure 35 Performance of beam design for load case 301
The result of the size optimization for load case 301 shows a clear improvement of the current load case (see Figure 35). This optimization also shows a major improvement of 303. The other load cases show less improvement relative the original design, which is very much possible because no constraint is set concerning the other load cases.

6.3.2 Beams optimized for load case 303

<table>
<thead>
<tr>
<th>Design variable</th>
<th>Inner radius (mm)</th>
<th>Outer radius (mm)</th>
<th>Thickness (mm)</th>
<th>Area (mm$^2$)</th>
<th>Mass (kg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>17,26</td>
<td>20,00</td>
<td>2,74</td>
<td>641</td>
<td>0,51</td>
<td></td>
</tr>
<tr>
<td>17,92</td>
<td>20,00</td>
<td>2,08</td>
<td>496</td>
<td>0,20</td>
<td></td>
</tr>
<tr>
<td>6,89</td>
<td>9,48</td>
<td>2,59</td>
<td>266</td>
<td>0,09</td>
<td></td>
</tr>
<tr>
<td>11,10</td>
<td>14,16</td>
<td>3,06</td>
<td>486</td>
<td>0,11</td>
<td></td>
</tr>
<tr>
<td>18,46</td>
<td>20,00</td>
<td>1,54</td>
<td>372</td>
<td>0,39</td>
<td></td>
</tr>
<tr>
<td>16,64</td>
<td>20,00</td>
<td>3,36</td>
<td>774</td>
<td>0,70</td>
<td></td>
</tr>
</tbody>
</table>

Table 4 Beam design for load case 303

This load case treats an optimization of a complex beam layout, where the beams $303_1$-$303_5$ are very much dependent on each other. If excluding one of these beams, the net built between the GOR and the bumper would collapse. Thus, beams did circumvent to disappear. The only independent beam property is the one dominating. However, it is still unclear how beneficial it is to have an arced bended net layout compared to a straightened one.

Evaluation of beams optimized for load case 303

Figure 36 Performance of beam design for load case 303

This size optimization shows a remarkable improvement for the load case 303 compared with the improvements made by the original design (see Figure 36). However, correlation between 301 and 303 has not been proved as indicated in load case 301. The other load cases show a moderate improvement.
6.3.3 Beams optimized for load case 304

<table>
<thead>
<tr>
<th>Design variable</th>
<th>Inner radius (mm)</th>
<th>Outer radius (mm)</th>
<th>Thickness (mm)</th>
<th>Area ($\text{mm}^2$)</th>
<th>Mass (kg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>15,44</td>
<td>19,70</td>
<td>4,26</td>
<td>941</td>
<td>0,50</td>
<td></td>
</tr>
<tr>
<td>18,27</td>
<td>20,00</td>
<td>1,73</td>
<td>416</td>
<td>0,33</td>
<td></td>
</tr>
<tr>
<td>18,64</td>
<td>20,00</td>
<td>1,36</td>
<td>330</td>
<td>0,18</td>
<td></td>
</tr>
<tr>
<td>18,84</td>
<td>20,00</td>
<td>1,16</td>
<td>283</td>
<td>0,20</td>
<td></td>
</tr>
<tr>
<td>16,22</td>
<td>20,00</td>
<td>3,78</td>
<td>860</td>
<td>0,44</td>
<td></td>
</tr>
<tr>
<td>15,79</td>
<td>20,00</td>
<td>4,21</td>
<td>947</td>
<td>0,35</td>
<td></td>
</tr>
</tbody>
</table>

Table 5 Beam design for load case 304

When comparing the cross section area, the two most important beams are 304$_1$ and 304$_6$ (see Table 5). The resulted dimensions of the two beams with 304$_1$-property indicates a weak spot in the corner of the COWL and the rear side rail for this particularly load case. In this bending load case, it is indicated that the upper bumper should be connected with lower bumper and the sub frame, 304$_5$ and 304$_6$, for improved stiffness.

Evaluation of beams optimized for load case 304

![Figure 37 Performance of beam design for load case 304](image)

As can be seen in Figure 37 the 304-case was greatly improved compared with the original design. This gives strong indications that this load case has not been taken in account when the designers formed the original beam layout. The beams in this case also contribute to certain stiffness for the lateral loads in case 303.
6.3.4 Beams optimized for load case 331, 332 and 334

<table>
<thead>
<tr>
<th>Design variable</th>
<th>Inner radius (mm)</th>
<th>Outer radius (mm)</th>
<th>Thickness (mm)</th>
<th>Area (mm$^2$)</th>
<th>Mass (kg)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>18.91</td>
<td>20.00</td>
<td>1.09</td>
<td>266</td>
<td>0.11</td>
</tr>
<tr>
<td></td>
<td>14.55</td>
<td>15.55</td>
<td>1.00</td>
<td>189</td>
<td>0.14</td>
</tr>
<tr>
<td></td>
<td>18.96</td>
<td>20.00</td>
<td>1.04</td>
<td>255</td>
<td>0.12</td>
</tr>
<tr>
<td></td>
<td>13.74</td>
<td>20.00</td>
<td>6.26</td>
<td>1327</td>
<td>1.60</td>
</tr>
<tr>
<td></td>
<td>3.21</td>
<td>4.32</td>
<td>1.10</td>
<td>52</td>
<td>&lt;0.01</td>
</tr>
<tr>
<td></td>
<td>7.01</td>
<td>8.11</td>
<td>1.10</td>
<td>104</td>
<td>&lt;0.01</td>
</tr>
<tr>
<td></td>
<td>18.95</td>
<td>20.00</td>
<td>1.05</td>
<td>257</td>
<td>0.03</td>
</tr>
</tbody>
</table>

Table 6 Beam design for load cases 331, 332 and 334

The size optimization of this case made it clear that the rally bar, 3314, is of great importance in the ruling beam layout (see Table 6). As for the three-legged connection, no great conclusion is possible to be made. The one leg, 3314, that is almost perpendicular to the main beam is the dominating one, which is not logical due to the moment stress that would occur at the connecting node. The leg, which extends the main beam, was instead expected to rule among the three. The appearance of the three-legged connection itself could be due to the weak contact surfaces on the spring tower.

Evaluation of beams optimized for load case 331, 332 and 334

Figure 38 Performance of beam design for load cases 331, 332 and 334

The result of the size optimization of the ruling beam layout shows a remarkable stiffness improvement for the 334 loads (see Figure 38).
6.4 Suggested design

Good information has been retrieved for each load case, but it is still desirable to try to combine the load cases and finding a design that has higher performance than the original design. To try to accomplish this, a heavily constrained topology optimization is done by the direct method.

6.4.1 Design constraints

The design should be made of thin walled aluminum tubes and their dimension boundary comes from discussion with employees at Volvo CC. The maximum weight is taken from the current design if aluminum were used. All the constraints on the beam design can be seen in Table 7. The weight 1.2kg is chosen due to the fact that it is the weight of the original design if only aluminum is used.

<table>
<thead>
<tr>
<th>Design constraint</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum weight</td>
<td>1.2</td>
<td>kg</td>
</tr>
<tr>
<td>Cross section</td>
<td>Tubes</td>
<td></td>
</tr>
<tr>
<td>Max. beam diameter</td>
<td>40</td>
<td>mm</td>
</tr>
<tr>
<td>Min. beam thickness</td>
<td>1.5</td>
<td>mm</td>
</tr>
</tbody>
</table>

Table 7 Design constraints on new beam design

6.4.2 Optimization constraints

Symmetry condition is applied to try to use the unsymmetrical body to remove local reinforcements. MINDENS is chosen very low compared to the default value to try to reduce the stiffness of low density elements to get a more realistic performance. The connection areas are reduced to avoid areas with overlapping sheets and thin spaces where local reinforcements often occur. The available design space is the same as discussed in this project. The new connection areas are shown in Figure 39 and the optimization formulation in OptiStruct in (7)

\[
\text{Optimization formulation}
\]

\[
\begin{align*}
\text{Minimize Mass} \\
\text{Subject to:} \\
\text{Symmetry} \\
\text{Min. member size} &= 30\text{mm} \\
\text{Compliance} &\leq \text{Ref. for all load cases} \\
\text{With:} \\
\text{Discrete} &= 2 \\
\text{Min.dens} &= 0.00001
\end{align*}
\]
6.4.3 **Realization of topology results**

A beamlike structure is retrieved from the topology optimization. Local isolated reinforcements only appears on the spring tower and are removed instead of realized. During the size optimization of the beams no solution that satisfied the design and performance constraints were found. To remove the reinforcements reduced the performance of load case 301 in such extent that the beams cannot compensate for it. The topology result is shown in *Figure 40*.

The knowledge of each load case is used to increase the performance in the failed load case. Beams from the load case 301-optimum are inserted into the structure and a new size optimization is preformed.
6.4.4 Evaluation of new design

With the use of the new beams a solution is found. An increase in stiffness performance is achieved for all load cases and a weight reduction of 1.5kg (≈ 50%) is achieved compared to the original design. An evaluation of the new design is shown in Figure 41 where the compliance improvement is compared to a car with no beams. The placements of the beams in the new design can be seen in Figure 42 and the dimensions in Table 8.

![Figure 41 Evaluation of new design](image1)

<table>
<thead>
<tr>
<th>Compliance improvement (%)</th>
<th>Original design, 2.7kg</th>
<th>Original design, aluminum, 1.2kg</th>
<th>New design, 1.2kg</th>
</tr>
</thead>
<tbody>
<tr>
<td>0%</td>
<td>Original design, 2.7kg</td>
<td>Original design, aluminum, 1.2kg</td>
<td>New design, 1.2kg</td>
</tr>
<tr>
<td>10%</td>
<td>12.0%</td>
<td>8.8%</td>
<td>14.3%</td>
</tr>
<tr>
<td>20%</td>
<td>15.6%</td>
<td>15.4%</td>
<td>31.4%</td>
</tr>
<tr>
<td>30%</td>
<td>28.0%</td>
<td>9.3%</td>
<td>9.4%</td>
</tr>
<tr>
<td>40%</td>
<td>11.1%</td>
<td>9.1%</td>
<td>14.1%</td>
</tr>
<tr>
<td>50%</td>
<td>11.1%</td>
<td>9.1%</td>
<td>13.9%</td>
</tr>
<tr>
<td>60%</td>
<td>0.1%</td>
<td>0.1%</td>
<td>0.3%</td>
</tr>
</tbody>
</table>

![Figure 42 New beam layout](image2)

<table>
<thead>
<tr>
<th>Color</th>
<th>Diameter (mm)</th>
<th>Thickness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>■</td>
<td>40.0</td>
<td>2.62</td>
</tr>
<tr>
<td>■</td>
<td>17.4</td>
<td>1.51</td>
</tr>
<tr>
<td>■</td>
<td>9.80</td>
<td>1.50</td>
</tr>
<tr>
<td>■</td>
<td>13.1</td>
<td>1.50</td>
</tr>
</tbody>
</table>

![Table 8 Dimensions of new design](image3)
7 Discussion

7.1 Sources of decrease in performance

7.1.1 Topology Optimization

As described in Chapter 5.3.2.3, solutions are removed within each step which will influence the performance of the final solution when using the step method. It has been shown that solutions that have a certain performance might have different topology; this is shown in Figure 20. When solving an optimization problem, it is important to remember that even if the objective function converges, the topology might not. It is therefore recommended to let the topology optimization run for about 50\* iterations even if the objective function change is close to zero. We would once again like to mention to not take the resulting topology literally. It is wise to see what the topology structure is trying to accomplish instead of only looking at the structure itself.

When analyzing the topology results, a validation of the architecture was done by using benchmarking. Remarkably, many of the resulting characteristics can be seen in other car manufactures vehicles. In Figure 43, a comparison has been made with Honda, who has placed a rally beam in the same position as the topology optimization has in load case 331-332-334. Similarities have also been found between our results and BMW.

![Figure 43 Benchmark with Honda (left) and BMW (right)](image)

7.1.2 Realization

In this project, the realization was made with some pre made decisions. No optimization were made to find the beam cross sections for each beam, instead a decision to use circular tubes was made. The boundaries of the design variables (minimum thickness and minimum and maximum outer and inner radius) were retrieved from short discussion with the staff at Volvo CC. These decisions narrow the range of feasible solutions and thus decreasing the performance of the realized result.

The realization method that is shown in this report works well, however it uses a direct translation of the topology to beams. A designer should lead or at least be a part of the realization phase since he/she can either translate the topology in a more realistic way or translate what the topology accomplish to several different design proposal. The size optimization can still be a natural part of the realization, but it is then preferable that the designers are deciding the constraints.

\* The amount of iterations worked well in this project but should not be taken as a specific advice.
7.2 Interpretation of topology results

The results from the topology optimization should, as earlier said, not be taken literally. The surfaces of the resulting beams are uneven because of the limitation given by the element mesh. Also, it could see irrational how some beams are crooked. However, the load paths are, after some iterative steps, quite clear and quite easy for an engineer to read. It is in the nature of topology optimization that the resulting structure is exposed to tension and compression. With a minimum exposure to bending and torsion, along with the manufacturing constraints (MINDIM), it is therefore most natural that the beams are circular.

Their connection to the GOR is not at the symmetry line (y=0), because of the design space boundary made by the engine, but indications says that it could be the optimum. The geometry of the engine that was used defining the design space was made with a limited level of detail. Therefore a mini study has been made.

7.3 Introducing topology optimization in the design process

The design of a new car platform is in itself an iterative process at Volvo Cars. A design is created, a FE model created from the design and then analysis are made of the design. The results of the analysis are used as input for an improved design and the cycle repeats. Since the structure will change for each cycle, it is crucial that the topology optimization is made flexible enough to be made within these cycles. As for now, the biggest problem is the creation of the design space. Much time was spent on finding the best way to create the design space from a FE model. It is possible to create a design space such as the one used in this project in a matter of 1-3 days depending how prepared all geometrical restrictions and connection areas are. However, it is preferable to get designers input on the geometrical restrictions and possible connection areas. Therefore it would be a good option to create the design space in the CATIA geometry environment or similar. It is very important to have an efficient way to create a closed design space from either a geometry or FE model to introduce topology optimization in the design phase. While the FE-model is created the designers could prepare a design space on the area of interest for topology optimization. When the FE-model is ready, a topology optimization could be initiated instantly.
Chapter 8 Conclusion

8.1 Recommendation to Volvo Cars

8.1.1 Starting slow
To use topology is not a "push of a button" science. Experience and knowledge in the tools and subject is needed along with engineering intuition. If topology optimization should be introduced to people without or with slim experience in the field, it is recommended to start using it on a small scale where the design spaces are small, simple and few load cases and very clear demands on performance. From this, designers will get experience in the field and learn to use topology optimization as a natural tool in their daily work.

8.1.2 Optimization during and before the design process
A recommendation is made to introduce topology optimization very early where the big decisions are made where design concepts and load paths are more valuable than a detailed design. This however will demand experience and knowledge of the person responsible but will without a doubt be very valuable step in the car development.

Topology optimization should be done parallel to the design when developing a car. It is a powerful tool to use for delivering input to designers. Topology optimization is best to use in the early design phase where knowledge of the structure is low and the design freedom is high.

By implementing topology optimization early, great knowledge of the load paths and weaknesses in the structure is received early and will help the designer to make good decisions. If topology optimization is introduced in a late stage of the design phase, the influence of previously made decisions about the structures dimensions and geometry will affect the gain of using topology optimization.

No recommendations can be made for what parameters to use (volume fraction, performance; displacement, compliance, stress, etc.) for a general structure and how to relate them to what is desirable to be accomplished by the final design. To use topology optimization as a tool for material placement is more systematic than to use more or less guesses based on experience. This is especially true for such a complex structure that is investigated in this project.

8.2 Future Work
This project is executed with a lot of manually work, no customization and adaptation of the software has been made, also with no previous experience of OptiStruct and car development. The result of this is that a lot of time has been spent on trial and error. There are although two main things that should be addressed to make it easier to use topology optimization as described earlier.

1. Speed up the process of making a design space.
   A design space should be created with the use of geometry in a environment where the car is designed. The use of Catia V5 to create a design space should be investigated. Preferable a design space should be done by or with help of a person that has knowledge of the component or area of the car that is supposed to be optimized.

2. Get designers involved.
   Designers should be involved when interpret topology results and realizing them. This project is done without any considerable deign experience and a improved design is in spite of this still suggested. With help of an experienced designer, the realization step will be both faster and the result better.
9 References

References