Hochschule für Angewandte Wissenschaften Hamburg
Hamburg University of Applied Sciences

Bachelorarbeit

Alexander Mausolf

Weight optimization of a pickup truck frame for the NVH load cases using FEA methods

Fakultät Technik und Informatik
Department Fahrzeugtechnik und Flugzeugbau

Faculty of Engineering and Computer Science
Department of Automotive and Aeronautical Engineering

Alexander Mausolf

Weight optimization of a pickup truck frame for the NVH load cases using FEA methods

Bachelorarbeit eingereicht im Rahmen der Bachelorprüfung

im Studiengang Fahrzeugbau / Karosserieentwicklung
am Department Fahrzeugtechnik und Flugzeugbau
der Fakultät Technik und Informatik
der Hochschule für Angewandte Wissenschaften Hamburg

in Zusammenarbeit mit:
Firma EDAG, Inc.
Abteilung CAE
1875 Research Drive Suite 200
Troy, MI 48033, United States of America

Erstprüfer/in: Prof. Dr.-Ing. Jens Marsolek
Zweitprüfer/in : Prof. Dipl.-Ing. Stefan Bigalke

Industrielle/r Betreuer/in: Velayudham Ganesan

Abgabedatum: 08.10.2014

Zusammenfassung

Alexander Mausolf

Thema der Bachelorthesis

Gewichtsoptimierung eines Pickup Leiterrahmens mithilfe der FE Methode

Stichworte

Optimierung, FEM, FEM Pre-Processing, FEM Post-Processing, NVH, Topometrie Optimierung, Topologie Optimierung, Blechdicken Optimierung, Leichtbau Konstruktion

Kurzzusammenfassung

Da NHTSA eine neue Richtlinie erlassen hat, die es den U.S. Herstellern vorschreibt ab 2016 den Treibstoffverbrauch Schrittweise immer weiter zu senken, besteht ein allgemeines Interesse an der Gewichtsoptimierung von Fahrzeugen.
Eine schnelle und relativ neue Methode um Strukturen zu verbessern, ist die FEM gesteuerte Optimierung.
Pickups sind eine der meist verbreitetsten Autoformen in Nordamerika. Sie sind sowohl für Arbeits- als auch private Zwecke einsetzbar, da sie bequem sind und durch ihre Größe auch als Statussymbol heran gezogen werden. Das Problem dieser Fahrzeuge besteht in ihrem großen Treibstoffverbrauch, der aus der massiven Bauweise und dem bedingten Einsatz eines stabilen Leiterrahmens herrührt.
Diese Thesis befasst sich mit der Optimierung eines Pickup Leiterrahmens für einen besonderen Lastfall, der sich Vibroakustik (NVH) nennt. Die Arbeit behandelt unterschiedliche Methoden und Programme um Ergebnisse zu erzielen, die benutzt werden, um das Gewicht des Rahmens zu reduzieren. Des Weiteren befasst sie sich mit dem Aufbau der unterschiedlichen Optimierungsprozeduren und bewertet den Aufwand den es braucht, um einen gewissen Gewichtsanteil zu reduzieren.

Alexander Mausolf

Title of the paper

Weight optimization of a pickup truck frame for the NVH load cases using FEA methods

Keywords

Optimization, FEA, FEA preprocessing, FEA postprocessing, NVH, Topometry Optimization, Topology Optimization, Gauge Optimization, Lightweight Design

Abstract

In modern vehicle structures consumption and efficiency become more and more important. Since NHTSA introduced its new regulations for the fuel wastage of each car which sets in 2016, a lot of effort is spent to optimize new cars.
A quick and relatively new method to enhance structures regarding the weight is the FEA based optimization.
Pickup trucks are one of the most driven vehicles types in North America. Mostly they aren't used for work, rather than being a comfortable big car and a status symbol. The downside of these massive cars is the fuel efficiency. The typical ladder frame which is normally integrated in the body-in-white of other vehicle types is a particular problem of these cars.
This thesis deals about the optimization of a pickup frame for a special load case called "Noise, Vibration, Harshness" (NVH). It'll deal with different methods and programs to gain reasonable results which are used to minimize the targets weight. Furthermore, it deals with the setup of these optimization processes, and evaluates the effort spent to the increment of weight, which can be saved.

Annotation for publication

To publish this work, some values that are property of EDAG, Inc. are assigned with a uniform offset, to provide the reader comparable results. Every value that is modified with an offset, is marked with a star (*) behind it. Furthermore, legends for figures have been modified, by leaving out the values, only indicating minimum and maximum colors.

Table of Contents

List of Figures

List of Tables

List of Abbreviations

Abbreviations

Symbol	Meaning
ANSA	Preprocessing Program made by Beta
CAD	Computer Aided Design
CAE	Computer Aided Engineering
CFK	Carbon Fiber
COG	Center of Gravity
CPU	Central Processing Unit
DOF	Degree of Freedom
FE	Finite Element
FEA	Finite Element Analysis
GUI	Graphical User Interface
ID	Identification Number
IGES	Initial Graphics Exchange Specification
LC	Loadcase
MPG	Miles per Gallon
MPH	Miles per Hour
NHTSA	Nation Highway Traffic Safety Administration
NVH	Noise, Vibration, Harshness
OEM	Original Equipment Manufacturer
PID	Property Identification Number
SPC	Single Point Constraint
SUPORT1	Special Constraint used for Inertia Relief
TRB	Tailor Rolled Blanks
TWB	Tailor Welded Blanks

Variables

Variable	Dimension	Meaning
A	$[mm^2]$	Area
C	[]	Drag Coefficient
c	$[N/mm]$	Stiffness
f	[]	Rolling resistance
K_{Ic}	$[(N/mm^2) \cdot \sqrt{m}]$	Fracture Toughness
M	$[Nm]$	Torsional Moment
m	$[kg]$	Mass
P	$[N]$	Load
R_{eH}	$[N/mm^2]$	Upper Yield Point
R_m	$[N/mm^2]$	Tensile Strength
$R_{p0.2}$	$[N/mm^2]$	Yield strength with Offset of 0.2 %
t	$[mm]$	Thickness
u	$[mm]$	Displacement Vector
V	$[dm^3]$	Volume
α	$[degree]$	Grade
δ	$[mm]$	Displacement
δP	$[N]$	Residual Load

ϵ	[]	Epsilon
η	[]	Efficiency
Θ	$[degree]$	Rotation
λ	$[dm^3/kg]$	Specific Volume
v	$[m/s]$	Velocity
ρ	$[kg/dm^3]$	Density

Indices

Indices	Meaning
B	Bending Force
D	Drag
f	Front
Frac	Fraction
r	Rolling
T	Torsion Force
t	Transmission
X	Global X direction, parallel to driving direction, pointing backwards
Y	Global Y direction, parallel to the car's axle, pointing in passengers direction
Z	Global Z direction, normal to the ground, pointing up

1 Introduction

Recently, the National Highway Traffic Safety Administration, short NHTSA, which belongs to the U.S. Department of Transportation and is responsible for making the roads more secure, released an updated regulation of the “Corporal Average Fuel Economy” (CAFE), one of today’s most strictly fuel saving regulations. This new regulation will start in the year 2020 and will reduce the consumption of every new produced car step by step up to the year 2025. The fuel economy limit is different for trucks and passenger cars, and also depends on whether it is a small or big vehicle, by using an equation called “Footprint” that can be calculated by multiplying the vehicles wheelbase with its average track width. Table 1-1 shows the set limits that were first introduced 2011 for year 2020 to 2025.

	Passenger cars		Light trucks	
Footprint [sq. ft.]	41 or smaller	55 or bigger	41 or smaller	75 or bigger
2020	49 MPG	36 MPG	39 MPG	25 MPG
2021	51 MPG	38 MPG	42 MPG	25 MPG
2022	53 MPG	40 MPG	44 MPG	26 MPG
2023	56 MPG	42 MPG	46 MPG	27 MPG
2024	58 MPG	44 MPG	48 MPG	28.5 MPG
2025	60 MPG	46 MPG	50 MPG	30 MPG

Table 1-1: CAFÉ regulation for year 2020 to 2025 [1]

If an original equipment manufacturer cannot produce his cars with a better consumption, he has to pay a penalty to the government. “When the average fuel economy of an OEM falls under the defined value, they’re forced to pay a penalty, which is “currently $5.50 per 0.1 mpg under the standard, multiplied by the manufacturer’s total production for the U.S. domestic market.” [1] Because at the same time NHTSA is also concerned about their domestic OEM’s, they pay companies for studies to proof if the set goal can be met with today’s possibilities.

Vehicle name	Features	Model year	Footprint	Average fuel economy	
Chevrolet Silverado 1500	5.3L,Crew Cab, Long Box	2014	73.1 sq. ft.	18 MPG	13.1 l/100km
Dodge RAM 1500 4WD	5.7L, Crew Cab, Long Box	2014	70.6 sq. ft.	17 MPG	13.8 l/100km
Ford F150 4WD	5.0L, Crew Cab	2014	72.8 sq. ft.	16 MPG	14.7 l/100km

Table 1-2: Comparison between the third most bought pickups in the U.S. [2]

EDAG, Inc. as an independent engineering company got one of these studies to see if a pickup truck with a footprint bigger than 75 sq. ft. build with new innovative manufacturing technology from today to the year 2020, can meet the given limit for the fuel consumption. The company’s first step was to find a car that has today’s highest lightweight potential in it. The decision was made for the Chevrolet Silverado 1500 from 2014, which has the best fuel economy compared to its competitors, by using modern lightweight saving methods like an aluminum hood and hydro formed profiles. Table 1-2 shows the Silverado, the second most sold car in the U.S. in comparison to the top seller Ford F150 and the third best sold car, the Dodge RAM. [3]

The actual value for the average fuel consumption of the 2014 model is 18 MPG. That means, by the latest regulations, Chevrolet has to increase the fuel economy in 11 years by 13.12 MPG. In metric units it would be a decrease from 13.06 l/100km to 7.55 l/100km with an increment of 5.5 l/100km.

Dipl.-Ing. H. Timm has made an empirical equation which relates the mass saving of a vehicle to the savings in fuel consumption. He says that for every more 100 kg in a car, the consumption changes about 0.3 to 0.5 l/100km. With the best possible value taken, the Chevrolets Silverado has to get lighter by $\frac{5.5\,\frac{l}{100km}}{0.5\,\frac{l}{100km}} \cdot 100kg = 1100kg$. The Silverado's curb weight is 2366 kg (5218lbs), so it would mean that this car has to go down at least to 1266kg what is a change of 46%. With downsizing the engine and lower requirements, for instance for the towing weight, Chevrolet probably don't have to reduce the full 1100kg. Even with this equation not being accurate, it shows how much light trucks have to be improved, in order get to the set value for fuel consumption. [4]

The pickup truck lightweight saving study was separated by EDAG into two phases:

The first phase is about benchmarking of the baseline model. During this step information will be gathered for the baseline model, so that the new improved lightweight car will have the exact same features as the 2014 pickup truck. The needed data will be gathered to get values like production costs, materials, thicknesses, crash behavior, the structure's stiffness, the weight or aerodynamic behavior.

During the second phase, using on the gathered information before, a new improved truck will be designed, that can meet every given constraints, but needs less fuel for it, using the latest methods for lightweight design.

The work on this thesis began during the first phase of this project. Since there are many different optimization programs and optimization methods available, the question occurred what would be the best way and order to use them.

The thesis was written in the first phase to show a way on how an existing structure can be optimized only using FEA optimizing software. Taken only one specific loadcase for one particular section of the car, it describes step by step, how different methods with different effort can be used, to create a structure that will have the same properties as the initial design but will be lighter. Also a general understanding of how the optimizer works in order to get a better result will be topic of the work.

2 Current State of The Art

The vehicles constructed with the chassis construction method can mostly be separated into four modules: Frame, cabin, suspension and drivetrain.

The main use for the chassis construction of today's cars can be found for trucks. The benefit to connect different cabins and components onto the frame to manufacture it in a simple and cheap way are standing in a good relation to the robustness and durability of this construction. The normal build up is to use two longitudinal beams and connect them with several cross members. The cross sections of the used members are depending on the purpose of the truck. If it should ride mostly off-road U-profiles are chosen to make it resilient to torsion, but rigid against bending, and when it is used mainly for highways, the cross sections should be closed to make it stiffer for global torsion and bending. The longitudinal beams are normally parallel behind the cabin and have the same height because of the mount sub frame. In most build chassis frames the front part under the cabin in bend outside because of the installation space of the transmission and the engine. An often used “Fish-belly shape” was developed because of the local loads applied to the cross members. [5]

Figure 2-1: Half disassembled 1933 Ford Tudor Sedan [27]

Before autos switched to the lighter and more efficient self-supporting construction, they used to be built with frame structures under it, which are still used for trucks nowadays. When the first pickup trucks were designed in the 1920s, they started with the same structure as it was used for normal cars (Figure 2-1). Unlike them, the frames of the pickup trucks never changed to another frame type, because none of them offered the same flexibility to put different cabins, boxes, or even other utilities like towing equipment on it with the same strength as the chassis construction. Different longitudinal beam elements can be simply welded together depending on the desired dimensions of the cab and the cargo box, and the other modules will just be mounted on it.

A complete change for the Silverado construction would decrease the overall weight, with same specifications, but wouldn't be possible for a pickup truck, because it is main advantage results out of the possibilities to have different cabins, boxes and even to leave the box out and mount something else on it. Therefore, also for future trucks, the frame will stay an important element.

To allow the costumer to modify his car, by choosing between different cargo boxes and cabins, the pickup frame is separated into three segments.

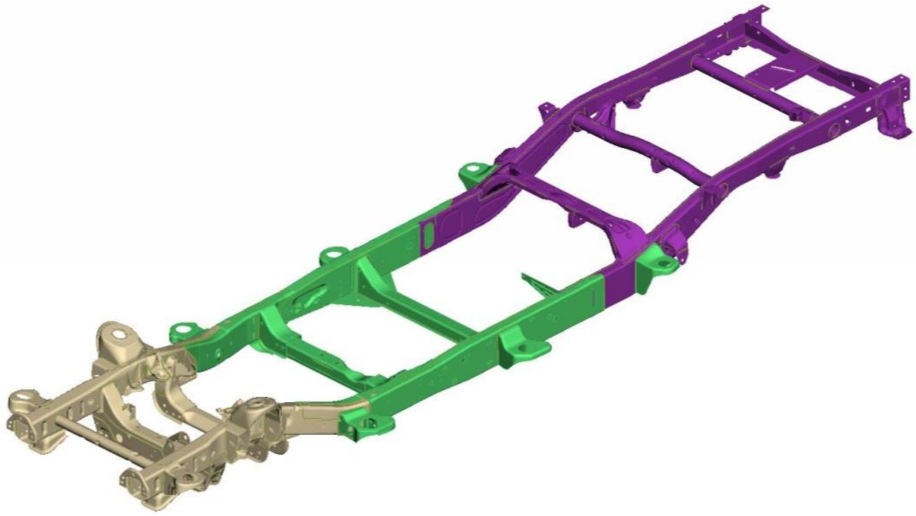

Figure 2-2: FE Model of Chevrolet Silverado 1500, Separated into Sections

Figure 2-2 shows the different segments colored separated. The front part will be the same for every version of the Silverado. It mostly has the task, to hold the engine, absorb the forces coming from the front suspension, and absorb energy from the frontal crash. The middle part varies and its dimension depends on the cabin being chosen by the owner. Six mounts are designed to hold bushings on where the cabin sits. This part is designed to be stiff during the frontal crash, so at first the front beams will collapse. Also it is the last holding for a side crash impact, when the rocker and side doors cannot bend any more. The rear module is dependent from the size of the cargo box being chosen, which is mounted directly onto the frame without any rubber bearings. A bracket at the front and one at the rear part of the rear module are holding the leaf springs for the rear suspension. One part is welded right in the middle of the rear longitudinal beam to restrict the maximal displacement the rear axle can do. All in all, the frame consists out of several sheet metal parts which were bent and welded together as longitudinal beams. An exception is the front longitudinal beam, which is one profile that has been hydro formed for better crash behavior. Welded to the beams there are nine crossmembers, each fulfilling different tasks. Counting from the front, the first crossmember is mostly important for frontal crash loadcases and keeps the longitudinal beam in place for the small overlap crash test. The second and third crossmember are mostly important for the lower wishbones of the front suspension and for the steering support. The fourth crossmember carries the transmission and has only an influence for the side pole impact. The sixth and the seventh member together are holding the gas tank. Welded to the last two crossmembers is a part, where the spare tire is mounted on.

The frame has in total 96 parts* and weights 224.61 kg*. The thicknesses of the used sheet metals are in a range from 1.5 mm to 5 mm, but any information about the alloys cannot be given, because it wasn't available by the time this work was written.

3 Lightweight Design

This chapter will deal about the today’s possibilities to lighten up a vehicle. At first, it briefly deals about the steps for a successful weight reduction and continues with possible materials that can be used to reduce weight of a structure. It shows how to choose the best material for a specific part and ends with today’s possibilities for lightweight design, using new manufacturing methods.

3.1 Lightweight Strategies

In general there are four different methods to realize lightweight design:

- Material lightweight design

In material lightweight design the given structure is optimized by using the right material for the right location. For instance, if a panel has to fulfill optical criteria, but does not have to bear any load, a lighter material with lower yield stress can be used, than for a loadbearing structure, which also has to meet requirements for fatigue.

- Shape lightweight design

The shape lightweight design is all about the way, the structure is designed. A thicker sheet metal can be replaced by a thinner one, when the loadpaths are known and the structure is stiffened by beads against buckling.

- Production lightweight design

The target for production lightweight design is to reduce parts by integrating as much functions as possible into one part which has as less material and connection points as possible.

- Cost efficient lightweight design

Cost efficient lightweight design means, that it is important to save costs by using the sufficient amount of material with the sufficient material quality and simple manufacturing methods.

Often it is not enough to only use one of the four methods. In most cases feasible weight reduction can only be achieved by using the most out of multiple methods. Therefore it is important to let every method interact with each other during the development for best results. [6] [7]

3.2 Steps for Weight Reduction

During the development of a structure, the following steps should be kept in mind to reduce the weight in a right and structured way.

At first, information has to be collected, what kind of loads will happen to the structure. Therefore, tests can be made, or already existing data can be used. This step has to be made carefully, because missing loadcases can lead to a bad result that may fail during its use. Next, limitations and safety factors have to be defined for the part, also called design criteria. Most design criteria are taken out of step one, other criteria are defined by regulations.

Then it comes to designing the part: Design methods should be used to create light structures. The old way of designing structures was, to first design the part and then run the analysis to see, if it meets the requirements. But the usage of today’s FEA optimizing possibilities should be merged with the designing process. Instead of designing a part out of experience, FEA optimizing software can exactly name the loadpaths to use, to get the best structure possible. Also other construction methods can be replaced with the use of optimizing software. Shapes and beads for sheet metal structure can be placed automatically at the best position possible, instead of getting them close to it by running countless versions.

The last step is to realize the developed design. Modern testing technology should be used and modern manufacturing processes with a constant, good maintenance are necessary for using a certain low kept safety factor. [8] [6]

3.3 Material Lightweight Design

In the past decades the traditional use materials have been high strength steel and aluminum alloys. Besides these materials also titan and magnesium alloys have become more and more important. Reducing the weight with different materials can only be successful when one knows the properties of each material, the advantages and disadvantages. The perfect material that fits all these requirements does not exist and is not necessary. It is rather important to find the material that fulfills the gained conditions in the best way. Different alloys and manufacturing method can also help to modify certain metal in a way, that they can meet the given target.

3.3.1 Multi-Material-Design

Different parts in an automotive structure have to meet different requirements to their materials. Not only for every part but also for different locations it can be another requirement to fulfill. Since no material will behave in a perfect behavior in every situation, today’s vehicle tend to have a mixture a several materials, where every location gets the best fitting material possible. The next chapters will describe the way to find and assign the best suitable materials.

3.3.2 Property Values

If the behavior of a material has to be described, different values are needed so that the calculation gets as close to the reality as possible. With only a few values it is already possible to rate if a material is suitable for the use as a lightweight material.

Density:

$$\rho = \frac{m}{V} \, [kg/dm^3]$$

Mechanical values for Stress ($R_m, R_{eH}, R_{p0,2}$), modulus of elasticity (E), transverse contraction (ν) and fracture toughness (K_{Ic})

3.3.3 Specific Characteristic Values

To just compare the materials by their E modulus or density for a decision, if it is suitable for a lightweight task will often have no success, because most of the materials have relations between their values. If, for instance, steel has to be replaced by aluminum, at first, aluminum looks like a promising exchange. But therefore the E modulus is almost proportional low to the lower density. To see these relations of the values, specific characteristic values have to be used.

Specific volume $\lambda = 1/\rho$

This is the simplest characteristic value which can be used. It just describes the volume in dm^3 used by one kilogram. Apparently, heavier materials will use less material for their weight than lighter ones. This value can become interesting when a used material cannot be manufactured thinner. Then another material with the same specific properties but with a higher specific volume could replace the initial material. [7]

Specific stiffness $E/g \cdot \rho$

The specific stiffness is an important value, when it comes to stiffness problems in a structure. By dividing the density from the E modulus, the relation between the material's mass and the stiffness is being made. [7]

Stability resistance $\sqrt{E}/g \cdot \rho$ and $\sqrt[3]{E}/g \cdot \rho$

These characteristic values are an indicator for stability problems. The formula $\sqrt{E}/g \cdot \rho$ shows if the material can handle buckling problems for rods, the formula $\sqrt[3]{E}/g \cdot \rho$ is indicating suitability for bulging of plates and bending of beams. [7]

Breaking length $R_m/g \cdot \rho$

This value should be used when tensile loads are connected to the modified part. Like the word itself implies, the value indicates when a string rips under its own weight. [9]

3.3.4 Table of Materials

When it is necessary to compare an actual material with several other materials, every specific characteristic of the comparing materials should be divided by the baseline material, to show the changes. When it becomes important to compare several specific lightweight values, weighting factors can be used to generate a sum for each material that gives a conclusion about the advantage towards the baseline material (Table 3-1).

		Boron steel	AlMg5Mn	AZ 91 T6	CFK (‖, 55%)
Static strength	$R_m/(g \cdot \rho)$	1.00	0.80	1.03	8.69
Longitudinal stiffness	$E/(g \cdot \rho)$	1.00	0.97	0.96	2.94
Buckling stiffness	$\sqrt{E}/(g \cdot \rho)$	1.00	1.68	2.08	4.06
Bulging/ Bending stiffness	$\sqrt[3]{E}/(g \cdot \rho)$	1.00	2.03	2.69	4.53
	Average:	1.00	1.37	1.69	5.05

Table 3-1: Comparison of current used material to others, using specific criteria [9]

Other interesting values for lightweight verification can also be:

<u>Elastic energy absorption capacity</u> $\frac{R_{p0.2}^2}{E}$

This formula can indicate how much elastic energy a structure can absorb. It becomes interesting when parts have to be compared and the range for the yield strength is important, for example for springs. [9]

<u>Material costs</u> $[\frac{\$}{kg}]$

Besides the properties of each material also the costs are an important factor for a decision. As you can see in Table 3-1, carbon fiber has excellent characteristic values, but if they are set in relation to the prize, which is about \$29 to \$1.4 per kilogram the decision can quickly turn against a material like carbon fiber, even if it has good properties.

3.3.5 Lightweight Materials

Following the philosophy to choose the right material for the right use, leads as a conclusion to a multi-material design. To be able to choose the right material it is important to know the specifications of the lightweight materials. Figure 3-1 shows a selection of materials that can be important for lightweight design.

	Density $[kg/dm^3]$	E modulus [MPa]	Tensile strength [MPa]	Price [$/kg]
Steel	7.9	210.000	1000	1.4
Aluminum	2.8	70.000	500	2
Magnesium alloy	1.74	45.000	250	3
Titanium alloy	4.5	110.00	300-900	19

Figure 3-1: Possible materials for lightweight design in an automotive structure [10]

3.3.5.1 Steel

Steel is still one of the most used materials for automotive engineering. It has good mechanical properties, is cheap and the manufacturing process is so well known and advanced, like for no other material. Only the high density can sometimes lead to more weight for structures. A lot of the aspects of steel can be changed by using alloys, strengthening mechanics or heat treatment. With these methods, steel can meet a lot of desired targets like: stabile for a big range of temperatures, corrosion resistance, high fatigue strength, good surface properties, good use for welding and for stamping and different behavior described by R_m and $R_{p0.2}$. Steel alloys don't meet all these abilities at the same time and have to be chosen carefully by defining their use. Besides these changeable values, all steel types have good recyclability, are environment friendly because of moderate manufacturing costs, can be used for coating, have a high availability and good resistance against aging. [4] [11]

3.3.5.2 Aluminum

One of the most important lightweight materials is aluminum. For a reason, AUDI started to build complete body-in-white structures out of this low density material. If one would just replace the used steel in a BIW with aluminum, 66% of weight can be saved and when it is replaced realistically with the same behavior as steel, about 45% of weight is saved. With about $2 per kilogram to $1.4, aluminum is 42% more expensive than steel what mostly has its reason by the energy intensive manufacturing process called fused-salt electrolysis. Furthermore is aluminum corrosion resistant due to its oxide layer, has good resistance against low temperatures, has strength close to steel, good weldability, good machinability, can be recycled and with extrusion method any desired cross section can be created. The disadvantages are the lower fatigue strength and the higher thermal expansion. With different elements (Magnesium, silicone, manganese) aluminum alloys can get better tensile strength values that even a following heat treatment becomes superfluous. [4]

3.3.5.3 Magnesium

Magnesium as an alloy with aluminum, zinc, manganese, and zirconium, is with his high specific stiffness and a very low density well suited for lightweight purposes and can compete against steel or aluminum. It has good casting possibilities, good weldability, good fatigue behavior, good availability and can be casted 50% quicker than aluminum. Besides this long list of advantages there are also disadvantages which restricts the use of magnesium: it has bad corrosion resistance, can dissolve in contact with other metals and can burn on itself, once it catches fire. [4]

3.3.5.4 Titanium

Titanium is with $19 per kilogram much more expensive than steel which mostly comes out of its manufacturing process. Its density is low than steel but higher than aluminum and its high strength, low thermal expansion and good corrosion and chemical resistance are good reasons for the use of titanium. [12]

3.4 Production Lightweight Design

Not only the lightweight reduction, but also demands for recyclability, more safety and more comfort have become more important for the automotive industry. Besides the development of new materials and alloys, also the development of semi-finished products has made progress. Special needs for the local designing of structures have been made possible due to tailored products. They allow to process with sheet metals, which are already thickness optimized for specific areas, to bring the structure closer to the idea of only as much material as necessary. Not only weight can saved, but also production costs, because of less use of material and less connection steps.

3.4.1 Tailor Rolled Blanks (TRB)

The production process for TRB uses rolls to change the thickness for a coil of sheet metal in a certain range (Figure 3-2). In most cases the result will have different zones of a fixed thickness and transition zones between them, where the thickness will change linear between the two adjacent thicknesses. This process is relatively cheap and can be applied quickly. The disadvantages are, that thicknesses can only be changed over a certain length and only from a certain value to another one. Also, there can only be used one material with its unique properties. Besides coils this technology can also be used to modify tubes over their length.

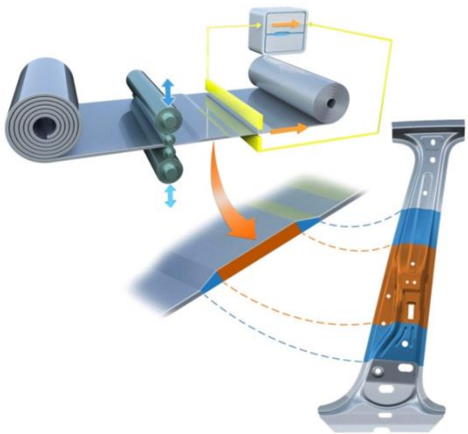

Figure 3-2: Process of TRB [42]

Tailor-rolled tubes have a high potential of their use when being hydro formed afterwards. The final part will be a tube with different thicknesses, different cross-sections and different radii over the complete length. Parts like the A-pillar or the complete roof rail section can be manufactured as one single tube which will fulfill each condition on every location. Because of its good surface compared with other tailored products, this method is suitable for stamped A-surface parts. [13]

3.4.2 Tailor Welded Blank (TWB)

This process uses parts of different coils to weld them together as one sheet metal part. Most of the times laser welding is used to merge parts, but when it comes to aluminum and magnesium sheet metals also friction stir welding is a possible option. For this process the connected edges does not have to be vertical but can also be varied in its angle and shape (called nonlinear tailored blank). Due to the possibilities, to use different materials for one part, there's a huge range of possible combinations to optimize a structure in the best way possible. Typical parts where TWBs are already in use are inner door parts, wheel arches, floor panels, longitudinal beams and cross members. [6]

3.4.3 Patchwork Blank

This process connects patches with a sheet metal, to give support on local areas. The connecting method can either be welding, spot welding, or adhesive bonding. After applying the patch sheet metal onto the surface the stamping process follows. For many parts not only patchwork blanks are used, but often two or more tailored methods, to get the best solution possible. [12]

4 Design Optimization

As mentioned in chapter 1 it is a major interest of automotive companies to optimize their cars, regarding different targets: New regulations defining certain fuel economy and carbon monoxide values, and high quality vehicles for a low price are only a selection of requirements. Lightweight design can help to create cars that match these conditions and in combination with design optimization it gets more effective. Since several years, this time wasting method is replaced by automated numerical optimization tools.

Necessary for a design optimization is a Solver-model. In most cases, and also in this paper, FE models are used, because they are already set up to analyze the results the design optimization is asking for. Generally, the optimizer will try to change the properties and certain values of the Solver-model to get the best possible result.

4.1 Important Words, Used in this Chapter

When it comes to the field of design optimization, there are some words being used, that will be explained in Table 4-1 in order to move on.

Objective	Mathematical formulation of one or more design targets
Constraints	Defined restriction for a response
Responses	A structural answer for a loadcase
Solver file	Used input data which contains a complete analysis setup
Design Variable	Values that can be changed during the process
Initial design	The data which is used for the first cycle (Iteration 0). In most cases the values are used that were already entered in the solver file.

Table 4-1: Terms used for optimization

4.2 A simple Approach

The following example should give a brief introduction to someone who is not familiar with the concept of optimization. Figure 4-1 is showing a hill with set up fences. A blind man wants to get to the top of the hill, but cannot exit the fences. The objective function would be to maximize the height in comparison to his starting position. The design variables are longitude and latitude and they are defining the position of the blind man. The constraint is that the man will stay inside the fence. The man is the optimizer, who will, step by step, climb the hill with its set borders. The optimization ends, when the man has found the highest spot possible. [14]

Figure 4-1: Optimization example [14]

4.3 Optimization Overview

The difference between a design optimization run and a normal analysis run is basically the extended setup, which, in most cases also contains the complete analysis solver model. To create such an optimization procedure, the solver model has to be linked with the optimization algorithm which will change data from the solver model to enhance the structure. Figure 4-2 shows the loop, the program will run through. At first, the initial design will be calculated with Finite Element Analysis, and the result will be compared with the given responses, assigned by the user. If these responses meet a specific convergence value compared with the previous cycle, the optimization stops. If they don’t match, the sensitivity analysis follows. The structural and the sensitivity analysis together will create an approximate model that will be used by the optimizer to generate an improved design. Before the algorithm finds the next solution to continue with, many approximate results are calculated until one of them fulfills the soft convergence requirements, set by the algorithm and the constraints defined by the user. With the improved design starting in a next analysis, also the next iteration has started, which will be documented and can be seen by the user due to constant screening.These optimization procedures can be used in different steps of the development for a structure and therefore can have different targets, variables and intentions. For example, when optimizing a structure from scratch, the target will mostly be to find a rigid structure, but later, when a design was made, the target will be that the structure will meet different specifications with as less weight as possible. Using design optimization can also lead to surprising results, even a good designer cannot imagine of because of its numeric approach. [15] [16]

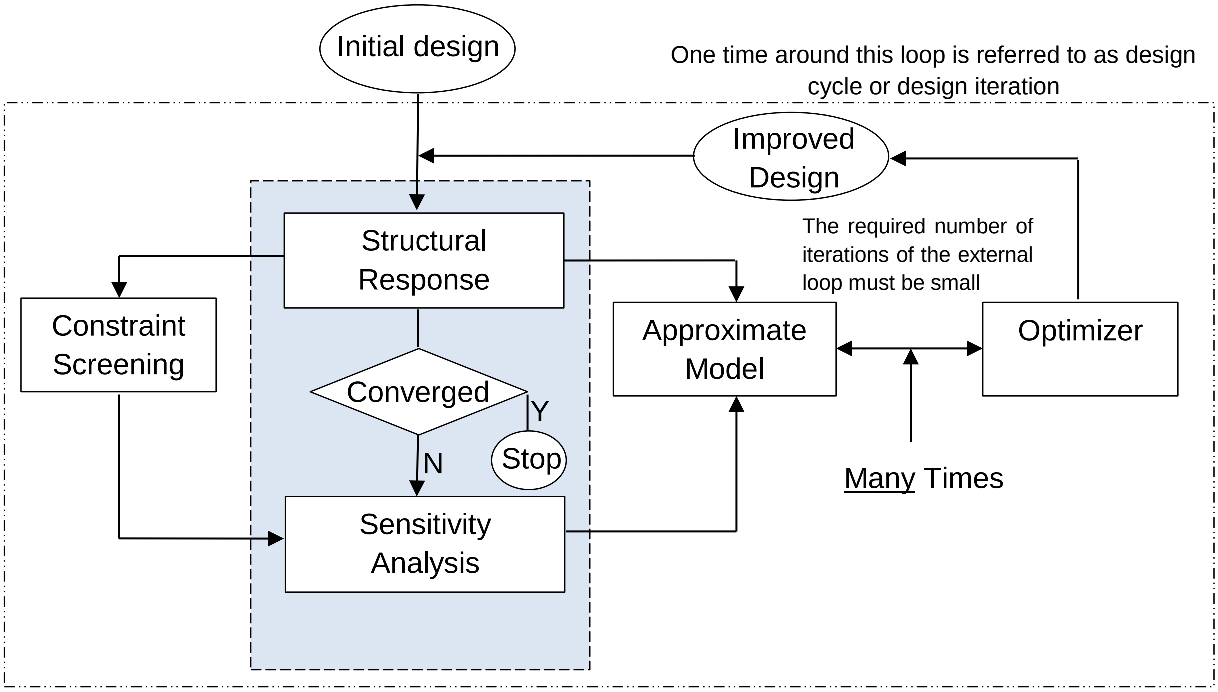

Figure 4-2: Optimizing algorithm [16]

4.3.1 Set up a Design Optimization

When it comes to create a setup for an optimization analysis, every method follows the same order. The following work flow is recommended to exclude any mistakes.

- Define specifications

"The formulation of an optimization problem is extremely important, care should always be exercised in defining and developing expressions for the constraints. The optimum solution will only be as good as the formulation." [17] This citation brings the purpose of creating a specification list to the point. During this step it is high priority to describe the loadcases, variables, responses and objectives for the structure as good as possible. Left out specifications can lead to unpredicted behavior in the structure when it is in use later.

- Gather information for specifications

Information for each specification should be used for defining the constraints of the optimization. They can be gathered by analyzing the baseline model for several loadcases, when the improved structure should have same properties as the initial model but less weight, or they can result out of regulations made for this structure. Examples are defined crash loadcases for a complete car, or anti-theft loadcase for doors. Other information can result out of tests results like loads for the suspension or directly from the customer's request, like a specific trailer that can be towed by a car.

- Create the solver file

Using a FE preprocessor is recommended to set up the solver file. Guidelines should be followed to provide a constant good quality for the generated mesh. Then the created file will be run in order to exclude errors made during the setup to analyze the behavior of the initial model and, should the situation arise, to validate the FE results with the real test results.

- Define the design variables

During this step, the variables for the selected optimization method have to be defined. Depending on the selected method elements, PIDs or particular values can be used as variables. Important is, that they are input values, so that their change will result in a different response during the analysis. For most of the design values, an upper and lower limit is necessary to restrict the optimization to a certain area. This will lead to shorter calculation time and better results.

- Define the responses

In this step, information from the results file of the analysis will be tagged with specific element cards which are called responses. They have the task to identify results and to make them accessible by their identification number. Optimization software like MSC Nastran solution 200 (Chapter 4.5.1) can support about 59 different response types, depending on the loadcase and the optimization method. A rough knowledge about them is required to create a design optimization.

- Assign the constraints

The previous chosen responses now will be equipped with constraints. That secures that the structure will meet the requirements, defined by the specification list. Constraints limit the range of solutions to a certain area and define the area in where the optimizer can do its work.

- Choose parameters and control cards

Considering the manufacturing process, it can be important that the design optimization runs with different options. Draw direction options, for example, can lead to results that can be applied for casting, and symmetry is often an interesting option for the right weight distribution in a car. Also options for the used CPUs, the maximum design cycles, or another criterion when convergence is being made can enhance the design optimization.

4.3.2 Multidisciplinary Optimization

Most of the optimization solvers available on the market are supporting multidisciplinary analysis optimization. That means that multiple analysis tasks, like static analysis and normal modes analysis, can be run at the same time to optimize a structure for both of them. To have this possibility available becomes more important, since most of the module of a vehicle have to stand a huge variety of different loadcases from static to crash related requirements. Special programs can control several input files for different FE solver at the same time and acting thereby as an interface for multidisciplinary optimization between them. [18]

4.4 Optimization Methods

When a structure has to be optimized, the first question is what method should be used to enhance it. Today's programs have several methods to modify different things like beads, thickness or other properties of the selected elements. Knowledge about the best suitable method and how it works is essential for the process of saving weight. Most of the programs which are available have the following possibilities for usage in a FE optimization.

4.4.1 Topology Optimization

The topology optimization method is mainly used to detect load paths for certain loadcases in a defined space, displayed as a solid model (Figure 4-3). It is often used in the early stages of the designing process to find the geometries best shape possible. The programs algorithm will change the density of each element to get to the objective and stay in the range of the constraints. One should keep in mind that where most of the other methods take less than ten iterations to find a solution with hard convergence, topology optimization can easily take up to 50 iterations depending on how many options were given for the design variables.

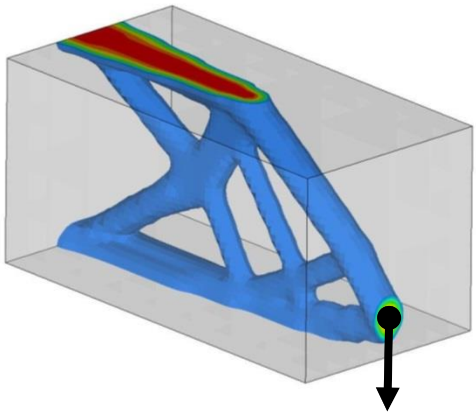

Figure 4-3: Example study for one static loadcase

Often responses called fraction and compliance are used for this process. Fraction, also often called volume fraction or mass fraction means the percentage from zero to one of the space that will be filled with material. A constraint percentage from five to ten percent of the total volume lead in most cases to good results. The upper boundary is always dependent from the used space, the amount and magnitude of the loadcases and from the desired structure, so that the percentage of fraction changes from one run to another and has to be reset in order to get feasible results.

Compliance is the inverse of stiffness and means the overall displacement of a structure for a specific load. Often the objective of a topology optimization will be to minimize the compliance, weighted with different factors for every loadcase. The reason for that is that in a topology optimization one is not looking for a weight minimized structure but for the one which is the stiffest. Responses which are used in other optimization methods like displacements are not recommended because a topology optimized structure often has nothing in common with the final design where the loadcases will be applied. [19]

4.4.2 Topography Optimization

Topography optimization is an advanced form of shape optimization and can be used to stiffen existing shapes with beads, bead patterns and darts. This can be useful if for example a part has to be optimized for higher normal modes or against buckling. The user can set options for the size of these beads, symmetry and what to do with adjacent loads or boundary conditions.

4.4.3 Topometry Optimization

Topometry optimization is the optimization of each element in one assigned region. Every element will become an own design variable that can be changed by its thickness. Unlike topology optimization elements can get thinner or thicker so that there can be holes and thicker areas. Besides the possibility to enter symmetry options, some programs have the opportunity to bundle several elements in one design variable (Figure 4-4). This reduces the processing time and generates results which can be better applied to the actual model.

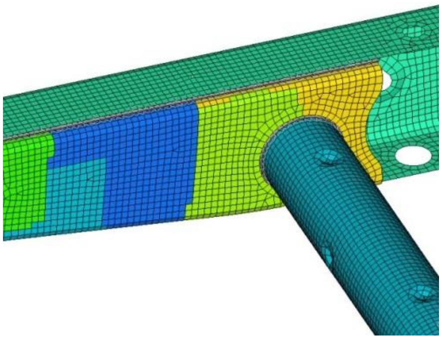

Figure 4-4: Topometry example for the frame

4.4.4 Size Optimization

When speaking of size optimization, most of the programs mean gauge optimization. If the loadcases and the belonging constraint are known, then the model can be optimized for thicknesses in a quick and easy way by using gauge optimization. Other changeable dimensions can be made for a large variety of elements. A structure could be enhanced by changing the moments of inertia, buckling factor or the center of gravity.

4.4.5 Shape Optimization

The algorithm for shape optimization uses the boundaries of the given structure to fulfill the given objective. The specified grid points can move in certain directions and the result will be a change of the shape. This tool can be useful, for example, when holes or radii have too high stresses or areas have to be enhanced for stiffness. When it is used for volume elements, shape optimization can deliver results that produce results almost like in topology optimization, but with the benefit, that it directly changes the final design rather than to be used during the designing process.

4.5 Used Design Optimization Software

During the work with different optimization software is being used. Every program has minor differences. Using the right program for each method can lead to a quicker, more efficient result.

4.5.1 MSC Nastran Solution 200

Since 1992 MSC has equipped their analysis application Nastran with the solution 200 which is used for optimization purposes. Optimization analysis can be setup with either MSC Patran or by defining it manually. For manual configuration of such a problem, MSC provides their own manual called: “Design Sensitivity and Optimization User’s Guide”. [16]

The following analysis methods can be used for optimization: Statics, Normal Modes, Buckling, Direct Complex Eigenanalysis, Direct Frequency, Modal Complex Eigenanalysis, Modal Frequency, Modal Transient, Static Aeroelasticity and Flutter. Nastran SOL 200 allows multidisciplinary analysis, which means, that multiple different loadcases out of different analysis can be run for one particular optimization objective. Table 4-2 shows the pros and cons of the Nastran solution 200 based on the experience made during this work. [20]

Advantages	Disadvantages
External expensive Optimization software can be saved by using the integrated SOL 200 solver of Nastran	Creating a file without a GUI always leaves space for errors, and often a preprocessor like ANSA is necessary to build the file
When ran in a cluster, Nastran SOL 200 delivers results in a short period of time	Features like topography, stamping or the topometry coarse method are currently missing in NASTRAN
When one is used to create normal runs in Nastran or worked with Altair OptiStruct before, it does not take much time to get familiar with SOL 200	

Table 4-2: Advantages and Disadvantages of MSC Nastran SOL 200

<u>Summary</u>

Nastran SOL 200 is a good optimization solver to start with, because it comes with a great manual and contains most of the optimization methods which are necessary to gain good result in a short time. If one is not familiar with Nastran, a lot of canceled runs because of fatal errors can be the result. Also if the task is to quickly analyze small structures, to see the results immediately, change values and start a new run again, the manual input method may be too slow.

4.5.2 Altair OptiStruct & HyperView

Altair OptiStruct is a module integrated in the HyperWorks environment, what means, that it placed right between pre and post processor in one software bundle. It uses almost the same input format as MSC Nastran and has only slight differences in a few optimization cards.

The creation of an optimization analysis works the same as an analysis setup in HyperMesh: The bottom section is used to create different cards for the input file. The graphical area can be used when element selection is needed and the necessary ID is unknown. Created elements and every created card can be seen in the left section of the GUI to give the user an overview of the created file.

The included analysis types are: Linear and non-static analysis, non-linear implicit quasi-static analysis, normal modes analysis, linear buckling analysis, direct and modal analysis, random response analysis, linear direct and modal transient analysis, coupled fluid structure analysis and linear steady-state and transient heat transfer analysis. [21]

The setup can be performed relatively easy. By importing a new FE model it can happen that HyperWorks will have problems to separate between SPC IDs and LOAD IDs. Furthermore, some “PARAM” control cards which are working in MSC Nastran are not supported by the OptiStruct Solver. The experience made with OptiStruct is noted in Table 4-3 sorted by advantages and disadvantages.

Advantages	Disadvantages
If one is using HyperMesh as a preprocessing tool, it makes no effort at all to setup a OptiStruct analysis	The import of Nastran input files does not work perfectly
By creating the necessary commands as cards. Mistakes can be found easily either in HyperWorks or in the .fem file itself	Because it is so close to many of the Nastran SOL 200 functions, an additional license for OptiStruct can be superfluous
Small topology problems can be run and analyzed quickly. Manufacturing constraints are helping to find the perfect solution	OptiStruct can take some time, if one is not familiar with the working environment

Table 4-3: Advantages and Disadvantages of Altair OptiStruct

<u>Summary</u>

Altair OptiStruct was mainly used to define small topology optimization problems, because of good working manufacturing constraints and the possibility to work with it directly in HyperView.

4.5.3 Vanderplaats R&D GENESIS

Genesis is a fully integrated finite element analysis and design optimization software package, which is in its language similar to Nastran SOL 200 and OptiStruct. It has the capability to optimize structures for the following methods: Static, normal modes, direct and modal frequency analysis, random response analysis, heat transfer, and system buckling calculations.

GENESIS uses the BIGDOT and DOT algorithm for solving optimization problems. Both optimizers were also used by MSC Nastran SOL 200 until they were replaced by MSCADS and IPOPT. The GENESIS Solver comes with a Design Studio, in where the complete optimization cycle, from pre- to postprocessing, can be done. It also supports a File Editor in where an optimization process can be built manually, if desired. [16]

The setup of an optimization problem is easy and supported by quick setup trails, if one is not familiar with the GUI of GENESIS. The made experience is gathered and sorted by pros and cons, listed in Table 4-4:

Advantages	Disadvantages
Fast setup for an optimization problem	Graphical area for pre and post processing is slow in comparison to ANSA or HyperView
Possibility to define a optimization analysis even when one is not familiar with optimization Quick results when running the analysis in a cluster	
Special optimization features such as topometry coarse method or free shape optimization	

Table 4-4: Advantages and Disadvantages of VR&D Genesis

Summary

VR&D Genesis is a quick and easy to use software which is especially important when it comes to Topometry and Free Shape optimization where this solver offers special features, the other competitors does not have. It was used for this work to optimize the structure for tailor blanks using the topometry coarse method.

5 Baseline Analysis

After giving an overview about the theoretical aspect of design optimization, this chapter will apply the methods of lightweight optimization to the structure, the Chevrolet Silverado pickup frame. At first, the selected loadcase will be explained, and how the real test is used to validate the simulated results. It then goes on with designing a precise, working FE model that will be used for the analysis. The last step will be the comparison of the calculated results to the actual structure, and how it can be brought in correlation with it.

5.1 Noise, Vibration and Harshness (NVH)

During the use of a vehicle, mechanical vibrations can happen due to many effects. In most cases these vibration are unwanted, but in some cases it is also desired to keep them because it can define the car's character. Also some vibrations cannot be eliminated because they're essential for the behavior of the vehicle for instance vibrations coming due to the road can be suppressed because then the tires would lose their contact to it.

The most important range goes from 20 Hz to 100 Hz. It is a range in where vibrations can be heard and felt. Because of this phenomenon such vibrations are called harsh. For car manufactures it is important to keep the eigenvalues for global bending and global torsion as high as possible because otherwise the car is able to react to incoming vibrations with own vibrational answers which can result in creaking, clattering or other acoustic effects. Such behavior will easily be judged by a costumer as poor quality and can be relevant criteria for the purchase. [22]

Therefore, it is the responsibility for CAE engineers to test structures and modules for the following loadcases, also called NVH loadcases:

- Modal analysis from 0 to 100 Hz

During the modal analysis it is important to find the eigenmodes for global bending and global torsion and to verify that their frequency lies above a specific value.

- Global bending for vehicle structure

For global bending a load will be applied on the middle of the vehicle's structure. Then the maximum displacement in this area will be measured to calculate the stiffness against bending shown in Equation (5.1).

$$c_B = \frac{F_{Bending}}{u_{Bending}} [\frac{N}{mm}] \tag{5.1}$$

- Global torsion for vehicle structures

To measure global torsion, torsional force will be applied on both shock towers so that they turn around the vehicle's center axis. Bearing will be used to fixate the rear axle and then the angle of rotation will be measured in order to calculate the stiffness against torsion, shown in Equation (5.2).

$$c_T = \frac{M_{Torsion}}{\varphi_{Torsion}} [\frac{Nm}{degree}] \tag{5.2}$$

5.2 Real NVH Test Results

To start the simulations with a FE model, the first step is to verify, whether it behaves close to the reality. Therefore the real frame has to be tested under the same load conditions, the analysis will run in afterwards.

Before disassembling the car completely, EDAG assigned a job to another company to perform an analysis to correlate the simulated results with the actual auto. The company built a setup for measuring the frame's reaction to the applied forces for the torsion and bending loadcase at multiple locations during the test. The result of the test is a certain stiffness value for both loadcases.

The bending test is performed by brackets welded under the longitudinal beams right in the middle between the mounting points, where the frame sits on. Both brackets are connected to a tube that goes parallel to the crossmembers. A cable is attached to the middle of the tube and is redirected by a guide pulley that is mounted onto the testing bed. The other end of the cable is attached to an actuator which pulls the rope in order to create the necessary force. The rear mounts for the test are represented by two brackets that are also welded under the frame, similar to the brackets explained before. Those brackets are connected to bearings, allowing the frame to bend. Both bearings are installed on different pods: one tripod and a bipod, allowing one side to move sideways where the other side is locked. For mounting the front, the appliance used for the torsional test is simply locked. The test runs three times with different loads and in multiple steps where the load at first increases and decreases again, once it reaches the maximum value. The displacements are directly measured at the brackets.

For the torsional test a special appliance is used in combination with hydraulic actuators to apply a torsional load to the frame. Every loadcase runs three times, and during the run, the load is applied in nine increments as a full cycle. To hold the auto, a minimum constraint support is chosen by using heim joints, which are hold by brackets, welded directly under the frame where the rear axle used to be. The third constraint for the torsional test is done by the appliance, which has a joint directly between both shock towers and is also used as the center of rotation. To measure the displacements over the whole frame, linear voltage potentiometers are placed along in longitudinal direction during both load tests.

Figure 5-1: Test setup for static tests

Figure 5-1 shows the setup for torsional stiffness. The cargo box is lifted up, to test only the frame. The red beam under the shock towers is used to apply the force into the structure. These tests are not only made for the frame, but also for the frame with the box, the frame with the cabin, and the whole assembly frame with the cabin and the box.

The results are given as tables. They consist out of a plot twist vs. torsion, deflection vs. position, deflection vs. load and deflection vs. position. In every plot, all three cycles are really close together. For the loadcases the results of half of the load cycle are chosen. Table 5-1 shows the extracted results out of the testing report.

Torsion	Twist	0.50 deg*
	Torque	1200 Nm
	Stiffness	3030 Nm/deg*
Bending	Deflection	1.78 mm*
	Load	4448 N
		2495 N/mm*

Table 5-1: NVH Test Results

5.3 Base Model Behavior for NVH Loadcases

Now, that real test result exists, a model has to be built for the frame. Several steps are necessary to create a working FE model. To successfully create and run a FE simulation, basic knowledge about FEA is required that is not subject of this work. [23]

5.3.1 Work before the FE mesh

To optimize the actual Chevrolet Silverado, it has to go through a benchmarking process, also called “reversed engineering”. EDAG bought one Silverado model to analyze the structure and scans every surface of it. This happens in different steps. At first the complete car is scanned, using a white light scanner, to write the data into a geometry file. Then a first disassembling happens to scan every module, like the cabin, the cargo box or the engine. This goes on until it makes no more sense to continue splitting the part again. Every time before scanning, all surfaces have to be prepared by applying chalk powder to it, for better results during the scan process and then sticking black and white orientation targets onto it. These stickers are necessary to adjust the scanned surfaces relative to each other. Information from the previous scan of the modules with the same dots is taken, to set the more detailed scans of the single parts in space. During the scan also information are gathered for the thickness, the weight, the material and the manufacturing method for each part.

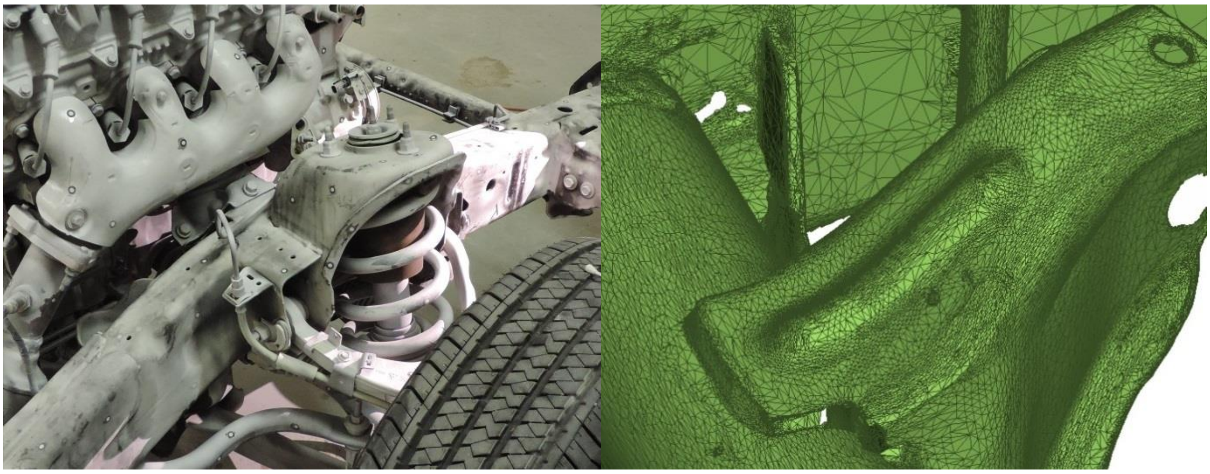

Figure 5-2: Prepared frame before the scan; digitalized scan

Figure 5-2 shows the preparation of the parts from the scan and the resulting data. The better the surfaces are prepared before the scan, the more features can be found in the scan. This is especially important for small details of the structure like laser welds or spot welds.

After the scanning process it is the responsibility of the designers to produce CAD data out of the given surfaces. Where only information for one, either the inner or outer surface exists, the data has to be interpreted the right way in order to project the real car as good as possible. Besides the modeling of the auto the designers also create geometry points for every spot weld of the vehicle. Then sheet metal parts are converted into middle surfaces and exported as IGES file, given to the CAE department to create a model for the FE analysis.

5.3.2 Creating a Working FE Model

The next step is to define the criteria for the mesh that will be created. Because it has to be used for different analysis and every section of it will be analyzed, the mesh size is defined with six millimeter, a value that represents the today's OEM standard for calculations, which can even be used for crash analysis and will still take a manageable amount of time. Besides an average element size, also limits for the elements criteria are defined. What is used is a company's own quality criteria collection that fulfills the given OEM standards and therefore represents the current state of the art for meshing. At the start, the units are set to length in millimeter, force in newton, weight in tons and time in seconds. A new include environment named “frame” is used, in where the given files are imported. Then, step by step, parts are loaded in for meshing.

First thing to do, is repairing the geometry, also called clean up: Either if mistakes are made by a designer, or the preprocessor makes mistakes during the translation of the input file, it happens quite often during the import of a file, that faces are not connected, faces are missing or there is more than one surface. Therefore, every part has to be investigated before one can start to create a grid. The next step that follows is about de-featuring or simplification of the geometry for the right mesh size. Unnecessary lines, also called “CONS”, which are too close together, so that the smallest elements can fit in, will be deleted. Also holes that are too small that it is not possible to fit at least six CQUAD elements around them are investigated. In some cases there are left with four adjacent CQUAD elements and if they are really small or not relevant for the structures behavior the hole will deleted. Next structure’s features are the fillets: Especially if a fillet cannot even allow at least one row of elements, both CONS lines are merged together as one line which still sits on the surface.

After the simplification is done, the meshing can start. Every part receives its own unique ID, following numbering rules which are set up to control the complete assembly, its own name, taken out of the bill of materials that has been made during the disassembling, and a thickness also taken out of the bill of materials. The used material is regular steel, because the complete analysis is all about linear analysis so there is no interest to define any specific values for yield stress. The used information entered for the material card is shown in Table 5-2.

Material: Default steel		
Young’s modulus	Poisson ratio	Density
210,000 $\frac{N}{mm^2}$	0.3	$7.85 \cdot 10^{-9} \frac{t}{mm}$

Table 5-2: Used values for the FE model

Even symmetric parts are assigned with separate tags to get the correct amount of parts later. When the settings are done, the meshing of the FE model can start. A first mesh is created, that is not fulfilling any quality criteria, but has the right element size. For a low number of CTRIA elements it turns out starting in the middle of a part leads to less triangular elements, than by starting at the rim and then going inside, probably, because then the mesh in the middle would has to play along with already set borders. Holes get their own mesh zones, so only CQUAD elements can be put around it. Doing so is important to get correct values for the holes later during the analysis, because holes tend to have stress peaks around them. A misplaced triangular element can result in stresses that are not matching with the real structure’s behavior. Then the created mesh has to be optimized for the set quality criteria. Unfortunately, even the best preprocessing program has still problems, to create a mesh that has the lowest number of CTRIA elements, fulfills all criteria, is relatively close to the geometry and is orthogonal to the centerline of the part, so that manual inputs are still needed for a good model. Doing so, most of the surface can be meshed, excluding fillets, beads, ribs, notches, flanges and hems.

Every fillet should be modeled that it does not contain any CTRIA elements in it. This will get difficult, when there’s a transition in a fillet and the radius is becoming bigger or smaller. Then the change from a certain row of elements to a lesser or greater number has to be done with the lowest number of triangular elements possible. The same goes for crossings of fillets with different radii.

Depending on the sized of beads and ribs, they are either modeled with two or three elements for each cross section. If it is not possible to model them with at least two elements, they will be ignored. Figure 5-3 shows the procedure of meshing beads, and a before and after graphic.

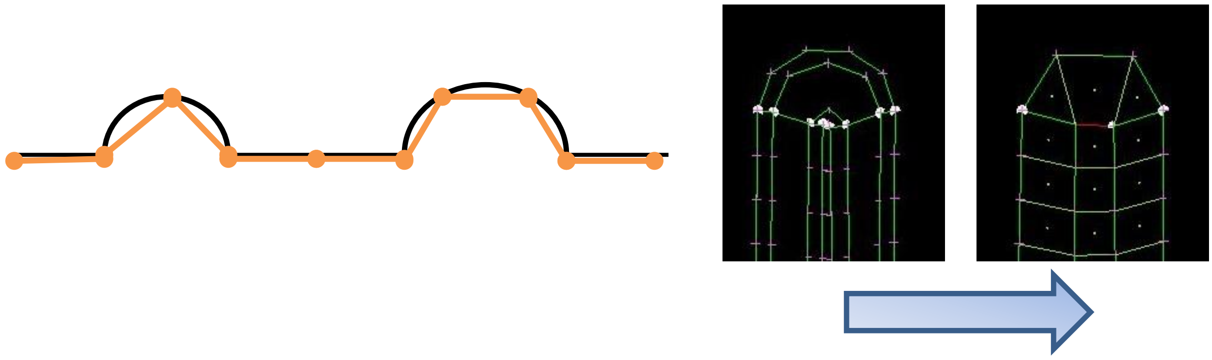

Figure 5-3: Recommended way to mesh beads

Notches tend to have geometry where one cannot model them without the use of triangular elements. Small ones are split into two CTRIA elements, bigger ones are modeled similar to beads, as shown in Figure 5-4. If there is no chance of modeling the bead with at least two triangular elements, they are ignored.

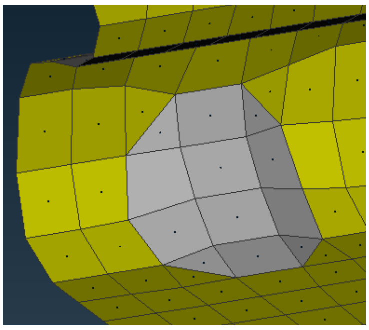

Figure 5-4: Finished FE model of a notch

Flanges, in most cases are modeled similar to fillets. If a flange is too small to receive at least one row of elements, the actual border, where the flange begins to bend, has to be moved more away from the part’s rim.

When a part contains data for hems, more work has to be done: The outer sheet metal part’s rim is trimmed to the point, where it connects with the inner sheet metal. From that point, the flange of the inner part gets a new PID, that goes up to the rim and its thickness is the sum of two times the thickness of the outer part plus one time the thickness of the inner part. Then, every node of the trimmed rim from the outer sheet metal is pasted onto the nodes from the inner part directly beneath it, so that there is a triple connection. Figure 5-5 explains the way of modeling hems with a crosssection and an example of a resulting FE mesh.

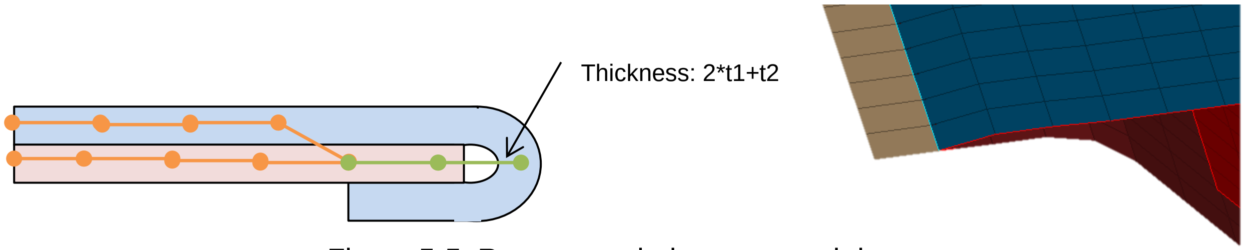

Figure 5-5: Recommended way to mesh hems

During the design of the mesh, the following checks for elements criteria are used:

- Warping
  - Checks if the element is ideal planar. It can only be used for CQUAD elements and controls the angle of the normal vectors for two triangles that are created by splitting the quadrangle.
- Skewness
  - Controls the inner angle and limits the skew of each element.
- Taper
  - The taper check watches the areas from two nodes to the center of gravity of each element, and compares them to each other, so that they are it the same ratio.
- Jacobian
  - The jacobian criteria measures the deviation of the element to a perfect shaped element. The perfect value of 1 is represented by a perfect cube for CHEXA elements, and a square for CQUAD elements. The name of this element criteria refers to the Jacobian Matrix Determinate. [24]
- Aspect Ratio
  - Aspect Ratio controls the ratio between the elements borders. The perfect value for it is 1 and represented by a square.
- Interior Angles
  - The quality criteria for interior angles is similar to skewness, but only controls that every angle of one element is in the set range.
- Percentage of triangular elements
  - FE models that contain a large number of CTRIA elements delivers results for the analysis that are stiffer than the real structure. Therefore it is important to keep the number of traingular elements in a model low.

	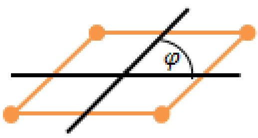	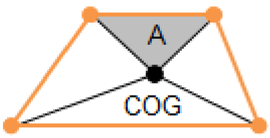	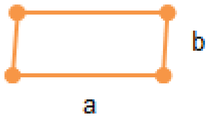
Warping	Skewness	Taper	Aspect Ratio
Criterium= $\max(\frac{h}{(d_1+d_2)/2})$	φ> Criterium	$\frac{4A}{A_{Total}} > Criterium$	$\frac{a}{b} < Criterium$

Table 5-3: Explanation of selected quality criteria [7]

When every element meets the given requirements for element criteria, the designed mesh is released from the geometry, so elements are not refering to the geometry anymore. What followes is a check for penetrations and intersections. If the preprocessor detects an intersection, that mostly means there are parts intersecting each other, because one part is misplaced relative to the other part. The other check for penetration is not necessary for linear static, but since the model will also be used for crash analysis, this check is made. It calculates the half of the thickness to each side of one element to get the actual surface, and checkes if it has contact with another element's calculated surface. If there is any penetration, the nodes have to be moved away from each other. About 79 penetrations has to be fixed that way, what tokes a long time. When there is a penetration for a T-joint, the part's nodes that ends to the other part is moved back and if there is any penetration with surfaces lying on top of each other, other parts has to be considered, when deciding which part can move in a certain direction.

When every part is in the right position without intersections, the creation of the connections starts. Every seamweld, spotweld and bolting connection needs to be represented in the FE model. Bolt connections are modeled by using RBE2 elements, to connect the nodes from the elements that are located around the hole for both parts (Table 5-4). Spot welds are modeled, using a tool, that connects the nodes close to the welding spot with RBE3 elements, and bundles them to one solid element. Seamwelds are modeled similar, but instead of a point, nodes close to a line are connected with RBE3 elements and bundled to a row of solid elements. To get the locations for the seam welds, the original scan file is taken, since the IGES file does not contain any information about connections.

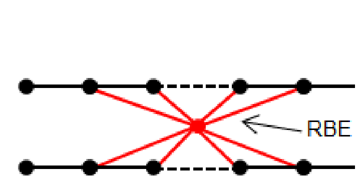	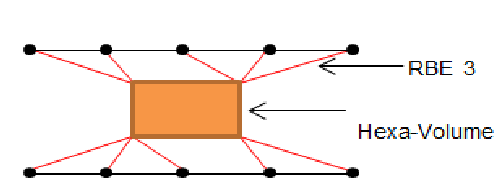	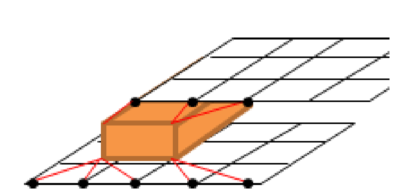
Bolts	Spot welds	Seam welds

Table 5-4: FE modeling for connections

Where the loads for each loadcase are applied on the structure, RBE 2 elements bundles the elements for a collective load distribution. The real test is analysed for the way, the forces are brought to the frame, and how the frame is mounted to the testing tools. For the torsional test, the load is applied with a beam, bolted to the shock towers. Therefore a RBE 2 element is created, whose slave nodes are all up the the point, the real tool sits. Because the bending loadcase is executed with a rod, welded under the frame, for the FE equivalent a RBE 2 element covers the same location. Also for the rear mount, a beam is welded under the longitudinal beam, so there are RBE 2 elements to represent this bearing (Figure 5-6). For the master nodes of the RBE 2 elements, where the load will be applied to, low and easy to use ID numbers are used, that it is easier to create the loadcases later.

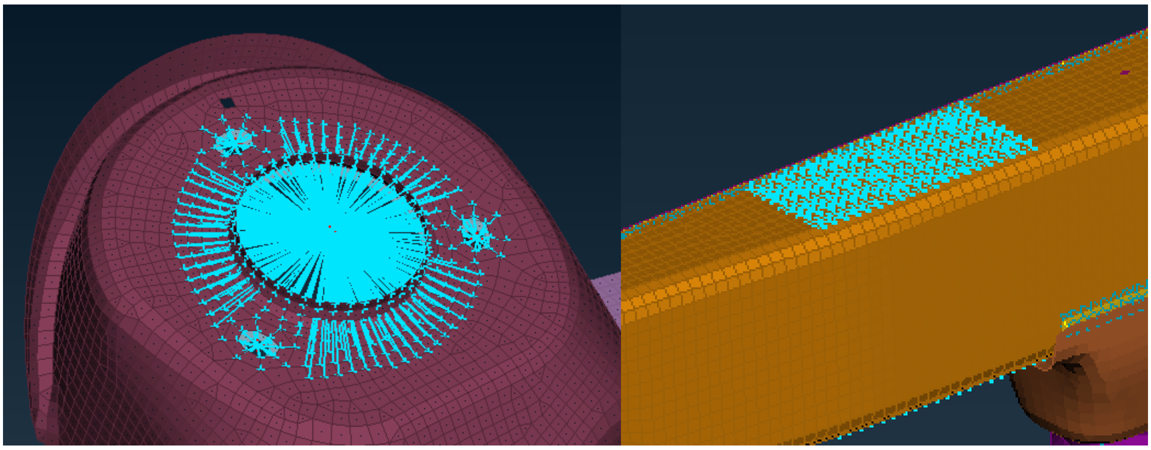

Figure 5-6: RBE 2 for application of force (left) and single point constraint (right)

After all part are connected and the points for the loadcases are created, additional checks for the FE model have to be made, to be sure it will run correctly during the analysis. First of all, the weight for the assembly has to be checked with the actual weight, that was measured during the scanning process. When the mass isn’t correct, parts have to be reviewed for their correct thicknesses. Then it is checked, if any object is still free in space, if the shell elements are homogenous oriented and if there are any triangular elements close to sensitive areas. A special check for RBE elements is also made to see if there are any rigid loops, where a master node is also a slave node, a state that would generate a fatal error. When the checks are succesfully completed, the model is exported as its own include, containing only the FE geometry, PID and material cards.

For the better control of the analysis, the executive control cards,case control cards and the bulk data cards for the parameters and the loadcases are written in an own file, that uses the “INCLUDE” cards to inlcude the designed FE model.

The finished mesh has 278878 QUAD* elements in it, 11403 CTRIA* elements (3.90 %*) and has 10224 CHEXA* elements, used for the welds.

During the next step, loads and constraints are set up for the created model. The objective has to get as close to the real test as possible (Figure 5-7). The necessary values for the forces are taken out of the report, made by the NVH testing company.

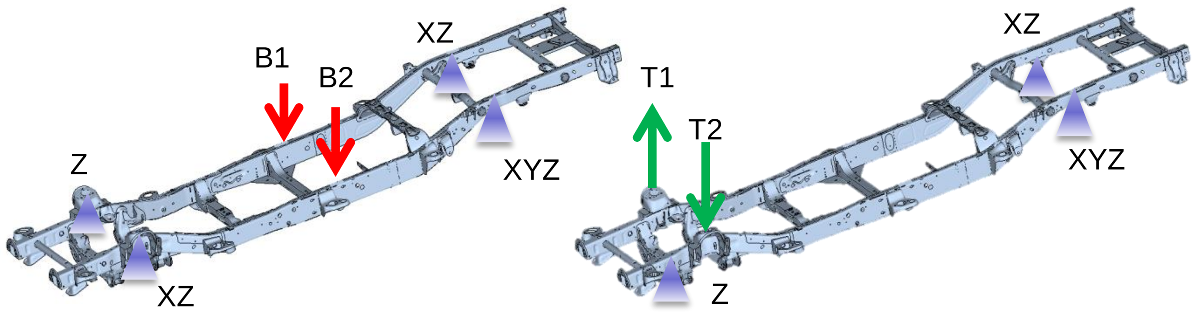

Figure 5-7: Locations for loads and constraints

Then three subcases are assigned for the baseline test:

- Global Torsion
  - 1200 N applied on both shock towers in positive and negative Z direction
- Global Bending
  - 2224 N applied on both longitudinal beams in negative Z direction
- Modal Analysis
  - Range of 0 to 100 Hz to cover the complete range for NVH phenomena's
  - Eigenmodes are calculated using the Lanczos method

Table 5-5 shows the constraints and loads being used for both loadcases. The designed RBE elements are used to apply the shown conditions to it. The RBE master's nodes that intentionally get the lowest IDs possible are assigned to the loads and constraints. Table 5-5 shows the loads being used to create the loadcases. In the following chapters, the node numbers, where the loads are acting, are being used to display results. Therefore, they are also mentioned here once. Besides defining the loads and constraints for the analysis, parameters are set for a better result. The grid point weight generator is used, to add gravity in Z-direction. Also the scaling factor for the penalty stiffness is set to one to avoid additional stiffness in the structure.

Loadcase	Element	Node #	Direction or DOF	Force
Torsion	Force	1	Z	-1200 N
	Force	2	Z	1200 N
	SPC	3	XYZ	
	SPC	4	XZ	
	SPC	5	Z	
Bending	Force	7	Z	-2224 N
	Force	8	Z	-2224 N
	SPC	3	XYZ	
	SPC	4	XZ	
	SPC	1	YZ	
	SPC	2	Z	

Table 5-5: Entered values for static loadcases

5.3.3 Matching the FE Result with the Real Test

The calculation is run with MSC Nastran and it takes 21 min to finish it with four processors. The displacements are measured in Z directions and are taken directly at the independent RBE grid points, where the loads are applied on. The value can be seen in Table 5-6. The given target for an acceptable correlation between the real test and the simulation is a maximum deviation of five percent. This value is used by EDAG as a common value to verify NVH simulations.

Static Displacement Results			
T1	3.501 mm*	T2	-3.500 mm*
B1	-1.766 mm*	B2	-1.766 mm*

Table 5-6: Results for NVH baseline analysis

Torsional stiffness

To calculate the torsional stiffness, the following Equation (5.3) is used:

$$c_T = \frac{M}{\Theta} \tag{5.3}$$

$$c_T = \frac{F_{1,2} \cdot b}{\tan^{-1}\left(\frac{T1 - T2}{b}\right)} \tag{5.4}$$

- M = Applied Moment
- Θ = Rotation
- $F_{1,2}$ = Applied Force on shock towers
- b = Distance between F_1 and F_2

$$c_T = \frac{1200N \cdot 983.6mm \cdot \frac{m}{mm \cdot 1000}}{\tan^{-1}(\frac{3.501mm + 3.500mm}{983.6mm})} = 2894.296\frac{Nm}{\deg} \tag{5.5}$$

The result shown in equation (5.5) matches the real test result of $3030\frac{Nm}{\deg}$* up to 4.7 % and is accepted as a satisfying value.

Bending Stiffness

In comparison to the torsional stiffness, the bending stiffness is only the displacement per force (Equation (5.6)).

$$c_B = \frac{F}{\delta} \tag{5.6}$$

- F = Force, applied to the structure
- δ = Displacement, resulting from the force

$$c_{Bending} = \frac{2 \cdot 2224N}{1.516mm} = 2518.69\frac{N}{mm} \tag{5.7}$$

When comparing the FE result of 2518.69 $\frac{N}{mm}$* to the actual test result of 2495 $\frac{N}{mm}$* the Simulation gets to a difference of 0.939%, a close result that is also accepted.

Modal analysis

To get the test results for the modal analysis, it is part of the work, to identify the modal number, where the structure responses with global torsion and global bending. Therefore, the magnitude of the displacements is set to a higher value, in order to see the desired movements in the post processing program. The values and mode numbers are noted (see Table 5-7) but it cannot be done any comparison, because the frame is missing a real modal test result.

	Mode #	Frequency		Mode #	Frequency
Global Torsion	7	24.35 Hz*	Global Bending	8	29.75 Hz*

Table 5-7: Modal Analysis Test Results

Stress Analysis

Since the simulated displacements are very close to the real test results, the next step is to see, what stresses each loadcase will result in (Table 5-8).The shown stress used to review the results, is called von Mises stress and is the common equivalent stress to use, when metals and other tough materials will be analyzed. Equation (5.8) shows how the von Mises stress can be calculated using the stress in X (σ_x) and Y direction (σ_Y) and the shear stress (τ_{XY}). More detailed stress results for all ongoing analyses can be found in Appendix A.

$$\sigma_v = \sqrt{\sigma_X^2 + \sigma_Y^2 - \sigma_X\sigma_Y + 3 \cdot \tau_{XY}^2} \tag{5.8}$$

Loadcase	Max. von Mises stress	Location
Torsion	70.99 MPa*	Figure 5-8 left
Bending	64.64 MPa*	Figure 5-8 right

Table 5-8: Baseline Maximum von Mises Stress

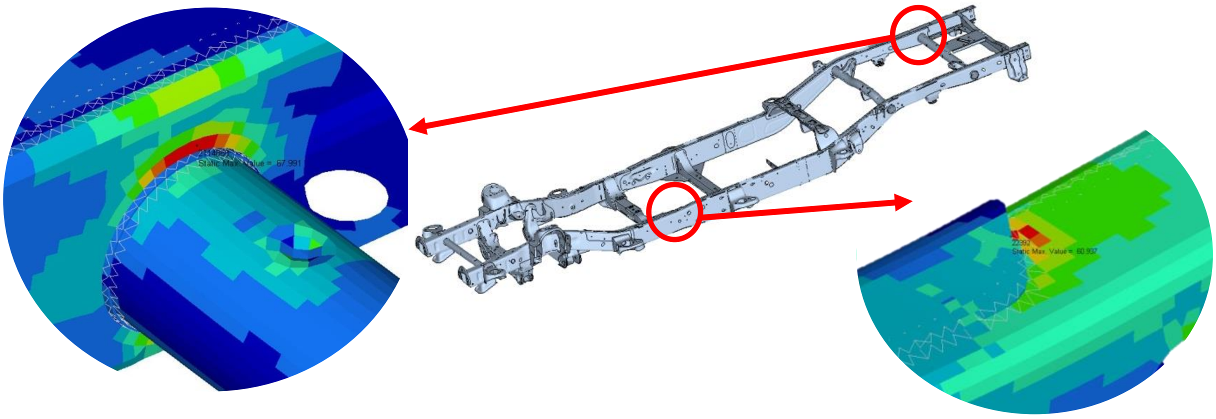

Figure 5-8: Locations for the maximum stresses during both loadcases

Because every NVH loadcase is made only in the elastic area, the stresses occurring on the structure are low. For both static load cases there are two maximum stresses, which are close to each other. The maximum stress during the torsion loadcase is located at the seam weld connection of the last tubular cross member on the passenger's side and the maximum stress for the bending loadcase is located at a seamweld connection between two longitudinal beams. Both measured stresses are relatively low and therefore they won't be considered as a constraint during the design optimization. Nevertheless, they will be checked manually during the post processing step. The resulting values for stiffness and displacements that are proven close enough to the actual frame will be taken as the baseline value to compare the optimized results with.

5.4 Summary

The start for every analysis is to correlate the mode with the actual test results. Therefore it is important that the given geometry is meshed using quality standards and following a checklist so that mistakes can be excluded. Otherwise the process, matching the real test results with the simulation can waste a lot of time with fixing the FE model and the search for mistakes.

6 Optimization of the Frame

Once the baseline model is proven mathematical close to the reality, the actual optimization can start. The initial design is the meshed baseline frame that will also be the structure each new design is compared with. Because of the particular loadcase being used, the first step is to see what part of the structure actually can be used for optimization. Then, one after another optimization process will be done, explaining first, how the optimization will be set up, then reviewing the results and after seeing the results, the output will be evaluated.

6.1 Preparing the Optimization

A very important step, before any structure can be optimized, is to structure the progress. The flowchart for lightweight optimization which is explained in chapter 3 has been found as a good, structured proceeding for an optimization process, without leaving anything out.

At first, the environmental conditions and load assumptions have to be set clear, to make any progress. Like written before, this work will only focus on the optimization for the Chevrolet pickup frame, using the NVH loadcases. By choosing this, parts have to be left out for optimizing the frame because other conditions are involved and they are mainly managing other tasks than the stiffness for global bending and global torsion.

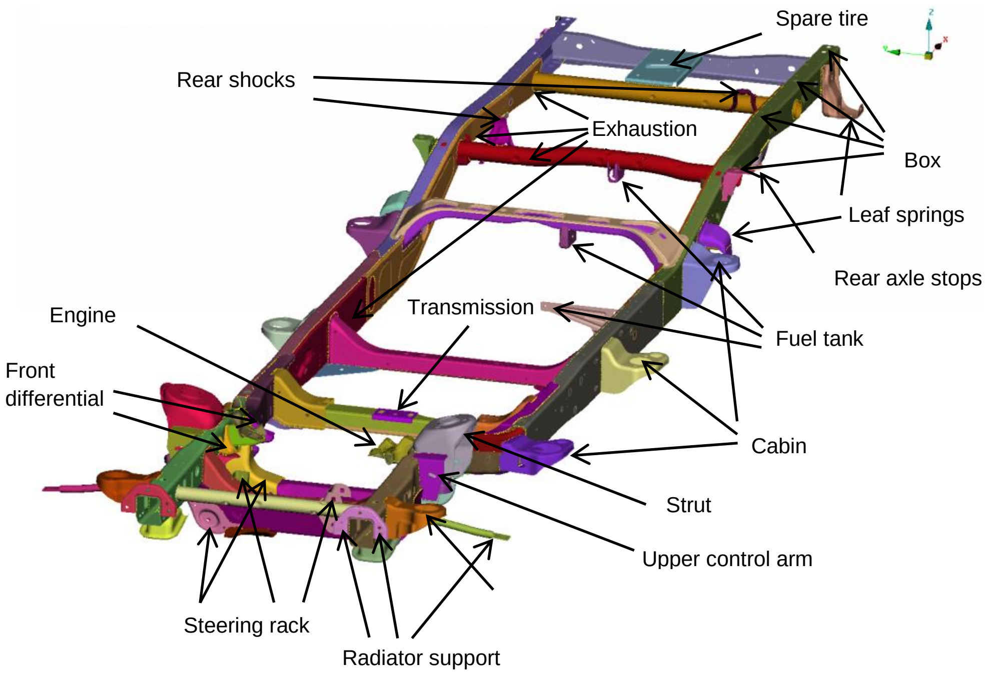

Figure 6-1: Overview of every module connected to the frame

Figure 6-1 shows all the connections, the frame has to any other parts. As you can see, a lot of the 97 parts*, used to build the frame have different tasks than to just stiffen the car. Most of the parts which can be seen are small brackets, mounted on the longitudinal beams of the frame. When performing an optimization for only the NVH loadcases of the bare frame, these brackets thickness would become the lowest value set for the optimization, because the program would detect, that they have no influence to the defined forces. The next part left out are the cross members, which are located under the engine. Both have additional purposes for the suspension, as to let forces, occurring during driving, into the frame. They are essentially for the driving behavior and have also only a minor influence to the frames stiffness. The last part that will not be included is the little part holding the spare tire. This also has a complete different purpose, and therefore is left out. Subtracting all these parts from the area where the optimization will set in reduces the amount of part to 36.

As one can see it is not easy to separate structures into their task, because they often have multiple purposes. The cross members are left in for optimization, because driving behavior, side crashes and other attachments have only a minor influence to them. Please note that this optimization, which will be described in this paper, only deals with selected loadcases. To apply the gained results, further validation can become necessary.

The second step is to define the constraints for the structure. When an optimization of a baseline model, like this frame has to be done, in most cases it is preferable, that the new model has the same behavior as the baseline. Therefore the initial model can be tested for all the loadcases needed, and relevant displacements and stresses will be taken as threshold values in where the change can take place. In this case, the main requirements are that the structure behaves the same for global bending, global torsion and their dynamic frequencies. Displacements at the locations, where the loads will be applied on the frame are the constraints the optimization will focus on. Furthermore these values are already aligned with the real test, so the changes in the following steps will be close the reality. Since the stresses in the structure are low in comparison to any yield stress, they will be still reviewed and compared with the baseline model, but not defined as a constraint. This excludes, that during an optimization process a stress peak, caused by a singularity or elements like CTRIA3 will cause “good” results to fail when their stresses are still in a good area.

6.2 Analyze the Loadpaths in the Structure

During the first attempts of analyzing the results for an optimization run, often the question occurs, why parts change in a way they do. Some members become really thin, others become thick. With no understanding of the actual loadpaths happening in the structure, rating test results can become problematic and solutions can be chosen, that includes mistakes.

To get this overview and to understand the ongoing processes, at first a simple topology optimization will be made. The complete setup is quickly finished, once one knows how to do it, and it saved time late on.

At first, the complete shape of the frame has to be modeled as a solid, using ANSA. Several points of the outer frame contours, without any mounts, are taken as lines for the solid. Tangency lines connect them to a cubic like shape, which is used to model a volume. Then it is important to think about the mesh size being used for this first optimization. On the one hand the simulation should run fast, because topological problems take more iterations than other methods, on the other hand, the used mesh size defines how detailed the design will be. To get good results, the empirical formula that the average element size should be equivalent to one third to one half of the minimum member size is being used. When fewer elements have to act as one member, the result can become inaccurate. For meshing, a coarse grid with 50 mm elements is generated. Because of the simple structure, a hexagonal solid mesh is possible and executed without any effort. Like in the actual model, RBE2 elements are used, to apply the loads to the model (Figure 6-2). The designed FE model then is exported and merged with the command file, by simply changing the incudes' name in the .dat file. The finished Nastran file then is translated into OptiStruct.

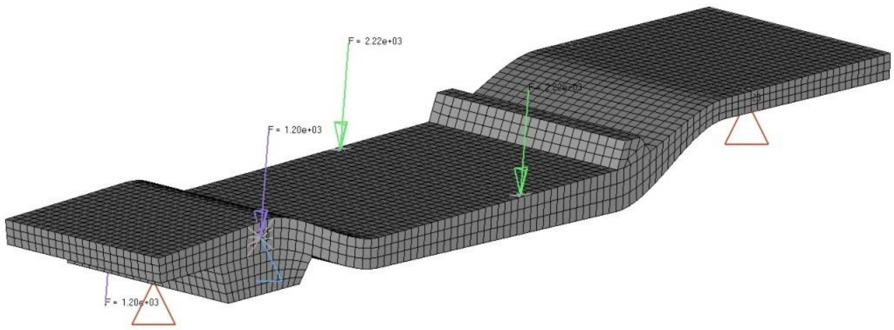

Figure 6-2: FE model which defines the frame's space

After a quick check if the file was imported successfully, the optimization parameters are created. A very simple run is created by using volume fraction and global compliance as responses, and constrain volume fraction to 30% as an upper boundary. Minimum compliance is assigned as the objective, and for the variable the complete solid structure is chosen. For additional variable options, the minimum member size of 100mm, which is roughly two times the average element length and the drawing in z direction are added to the file.

The analysis is executed with the local processor and takes, because of the small amount of 8,558 CHEXA elements only seven minutes and 21 seconds for 45 iterations. Besides the analysis with 30 % fraction, also results are created with 15 % fraction and 15 % fraction without a drawing direction (Figure 6-3).

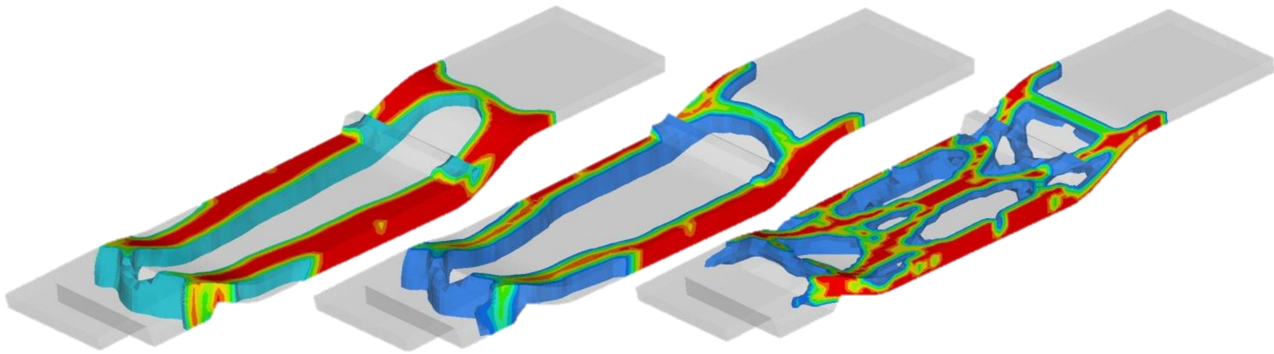

Figure 6-3: The last iteration of the three results after the optimization

Where the first two results only differ in small areas the third analysis give a result, which is completely different from the other ones. The first two runs are matching the actual frame quite good, and the second picture comes with its rear cross member shape so close to the baseline assembly, that it is chosen as the most feasible result. A fourth run is started with 10% fraction and the not changing shape confirms the chosen result of the second analysis with 15 % fraction. The different position of the rear cross member is the most interesting change between both analysis. It results out of the lesser torsional stiffness from the side members, due to less mass, which is constraint for the run. The third analysis is showing multiple crosses between the two longitudinal beams. In the upper level there is one big cross, connecting exactly the points, where the single point constraints are defined and there are two smaller crosser in the bottom level, one close to the rear suspension, and one close to the front suspension. This last run clearly shows a construction which would give the frame a maximum stiffness for bending and torsion. There are the two longitudinal beams that will work against the bending force, and the crosses, which support the structure against torsion. Why the baseline frame is not looking like this result has two reasons: at first the crosses right now are in an area which is occupied by different other parts, like the powertrain, the gas tank, the exhaust muffler and the drive train. The second reason is that this topology optimization is not considering any other loadcases. If more loadcases would be added, and weighting factors would be assigned to them, to define their importance, the result would be a different shape.

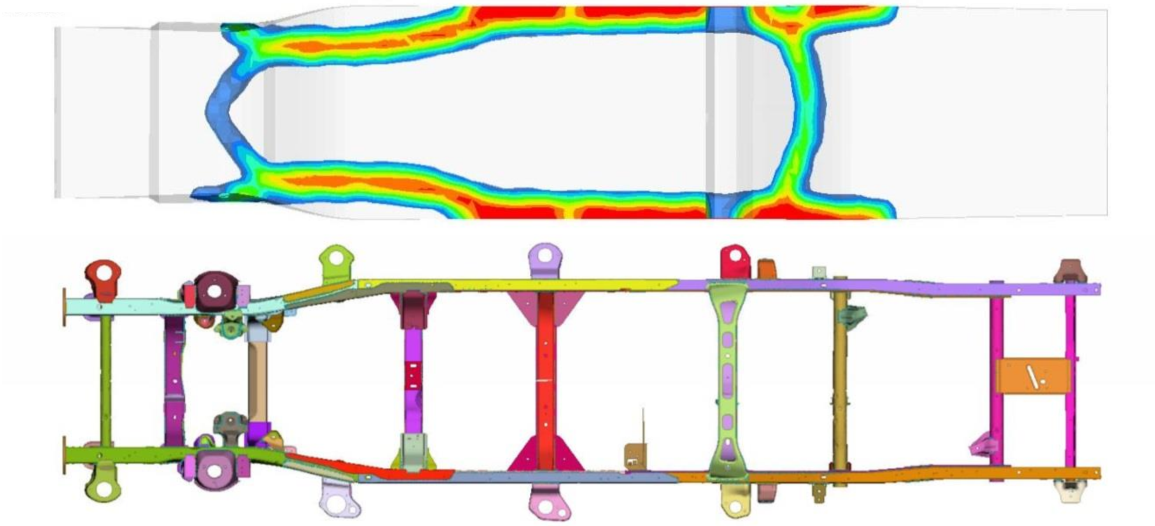

Figure 6-4: Comparison between the actual baseline model and the topological result

The resulting shape of the topology optimization has a rectangle like shape (Figure 6-4). Instead of having several crossmembers all along the frame, only two have managed to stay during the design cycles. The loadpath up front, between both beams, is perfectly represented by the two crossmembers, which are responsible for the front suspension, and will not be changed during the optimization. The rear connection is not perfectly represented by the tubular crossmember, because it is slightly off in X direction. The arch like shape of the result is also an interesting feature and can be considered for a new design. At last there are the strong pronounced longitudinal beams that have a more inside turned shape in the frontal area close to the transmission. All in all the topology optimization is indicating that the majority of the stiffness is made by the two rails on the side, the two crossmembers up front, and the tube in the rear section.

<u>Summary</u>

Doing a topological research before the actual is an important step that will help to understand every other optimization that follows. Therefore it can take some time to get into the procedure of successfully generating a topological result and rating it, because the topology optimization slightly differs from a regular analysis, many different options can be taken in order to produce a result and every used option can lead to a major difference in the output. The explained steps can be compromised to the following work flow:

1. Define the usable workspace for topology optimization, by using the outer lines of the structure
2. Define a sensible element size considering minimum member size and running time
3. Create the analysis input file
4. Create a topology optimization
    a. Define the solid structure as the variable being used for topology optimization
    b. Choose between different additional options for the defined topological variable.
    c. Define the necessary responses and constraints. In most cases it is enough to minimize he weighted compliance by constraining the volume fraction to a specific limit.
5. Run the optimization
6. Review the results using a postprocessor
7. Rate the output, considering manufacturability, level of detail and usability
8. If result is unsatisfying go back to step 3 and change topology option and limits for constraints. In some cases one even has to go back to step 1, changing the model at some locations or changing the element size.

Once one is familiar how to do a topology optimization, an analysis can be finished in a couple of hours. The estimated times for the steps preprocessing (60 minutes), optimization (three different runs, 45 minutes), postprocessing (30 minutes) are made for this example considering, the user knows what he has to do. If the structure gets more complex and also geometry has to be subtracted, the task can quickly be too much for a preprocessing program, and the help of a designer can become handy.

6.3 Gauge Optimization

The first step for weight reduction which is being made, is also the easiest way the save weight. Gauge optimization as part of size optimization will change the thickness of properties in order to full a certain objective, in this case to minimize mass.

6.3.1 Nastran File Editor

Almost every optimizing program can perform these calculations, but not everyone cost the same per usage. MSC Nastran is one of the most used and most reliable programs on the market right now. The configuration of an optimization file has its differences only in an additional section, in comparison with a common analysis. Nastran runs are mostly set up by defining a control file (ending: .dat) which contains all the parameters, loadcases, control section entries and case control entries. This file will call every other file, which is saved as an include file (ending: .nas) during the analysis. Those includes are separated into modules containing FE elements, material cards, property cards and so on. Separating the model makes it easier to run big assemblies, like for a whole vehicle and provides a good overview. Modules can be changed separately without affecting other files, and parameters or loadcases can be changed in the control file quick and easy. This procedure gives the idea to create an Excel spread sheet for Nastran solution 200. The working environment of excel should be used to assemble an optimizing control file which will exclude most of the mistakes made by the user, and can be outputted as a ready-to-run .dat file (Figure 6-5). With the necessary description taken out of the NASTRAN Quick Reference Guide [20], this file is even more precise and helpful than most of the optimizers, which will provide only limited support for their entries.

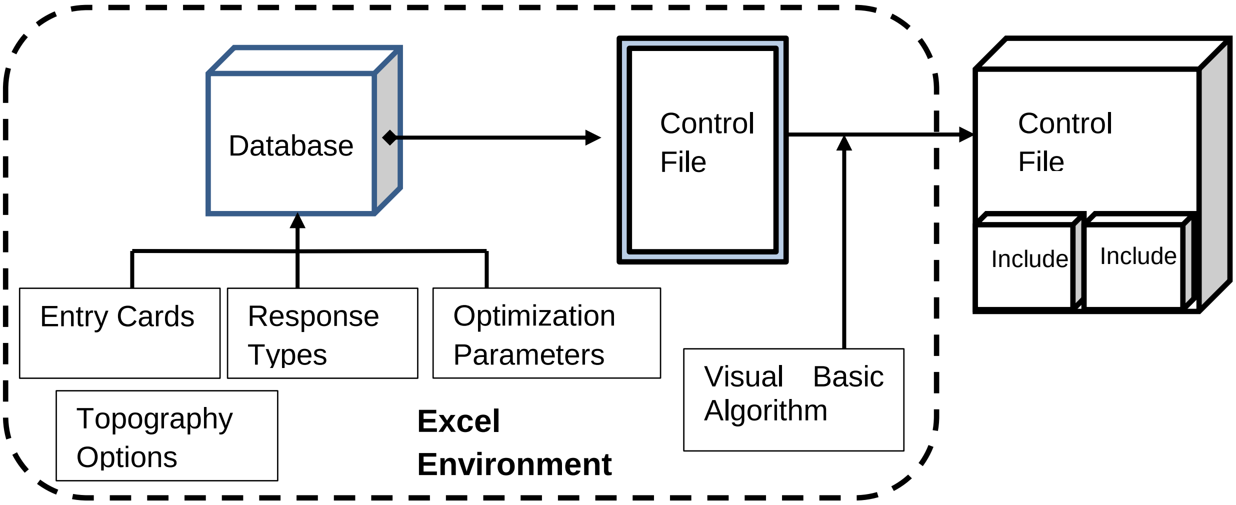

Figure 6-5: Chart of the built Excel Nastran SOL 200 editor

To create a representable database, previous used optimization runs out of more than 15 OptiStruct tutorials are used, whose entry cards are analyzed and then put in the right order into the entry cards section of the Excel file. OptiStruct files are taken, because the used language is close to MSC Nastran. Most of the cards worked, but some have to be looked up because they're called different in Nastran. For example the card “DPTL”, which defines a variable for topology, from OptiStruct wouldn't work in a Nastran analysis, because this card is there called “TOPVAR”. Also other mistakes happened when translating the files. When the total displacement directions are defined in OptiStruct, the program writes “TXYZ” for the same entry cell, are Nastran would type in “123”. Nevertheless, these tutorial files cover a good range of optimization problems to get a database to start with. Then default values are set as placeholders for each entry section and rules are added, so that one cannot enter more than eight characters and that one can only enter real or integer value into specific cells. For the shown boxes in Figure 6-5 for response types, optimization parameters and topography options, worksheets are created, using the “Nastran Design Sensitivity and Optimization User's Guide” [16]. The following worksheet then can be multiplied to create several control files. Each work sheet comes with a rough setup of the most used entries in the “Executive Control” and “Case Control Card” section, so that only the needed parameters and loadcases have to be added.

The next step is to create an algorithm so that the created session can be exported. It takes 200 lines of code to produce a sequence that is able to export the file in a way that every cards characteristics is being considered and that it'll produce a file with a good overview. Especially the different number formats from Excel to Nastran are a problem that has to be solved in a specific loop.

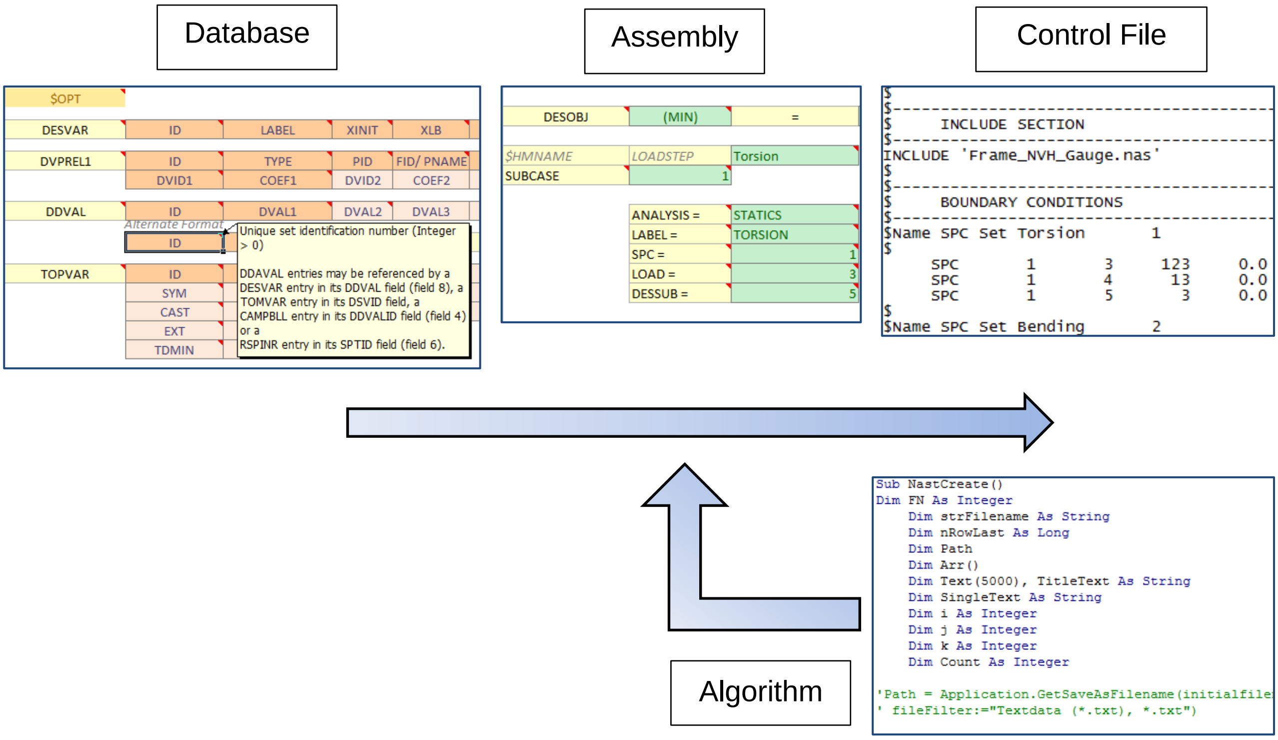

Figure 6-6: Work flow using the Excel Nastran SOL 200 Editor

6.3.2 Optimization Setup

After finishing the Excel spread sheet, a control file for the gauge optimization is created, as shown in Figure 6-6. The property ID for the parts that should be changed is chosen by using ANSA. Since this preprocessor has the ability to output information for visible entities, like in this case PIDs, the necessary thicknesses and PID numbers can be directly pasted into the control file assembly.

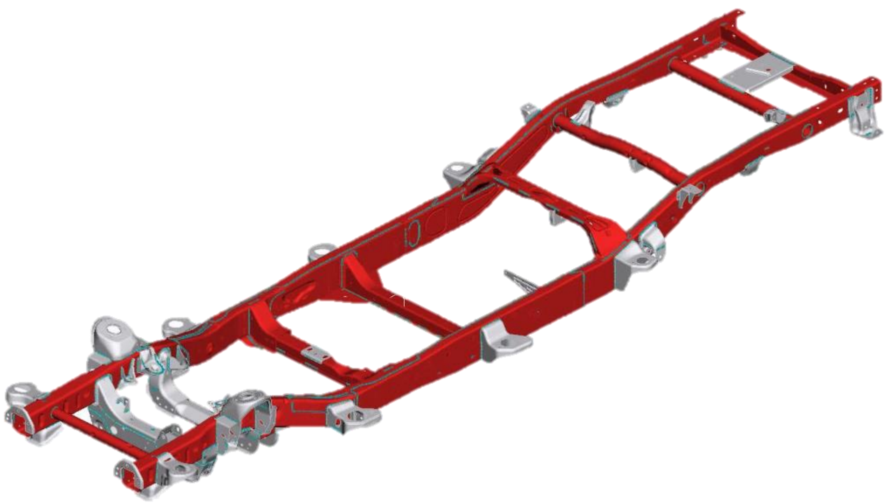

Figure 6-7: Selected parts for gauge optimization

Figure 6-7 shows the selected parts for this optimization. It refers to the chosen parts, dealt in Chapter 6.1. Only for this optimization, the front longitudinal beam is kept in, because it has a major influence for both loadcases bending and torsion. Most parts kept out for optimization are brackets, which would go down to the lowest limit, because they don't have any purpose for NVH loadcases, and the crossmembers under the engine, where too much other loadcases remain unknown.

The loadcases are defined the same as the initial analysis. A discrete value range from 0.6 mm as the minimal manufactural thickness and 5 mm as the maximum available thickness in the frame is chosen to define the space in are the optimization can act. The increment of the discrete values is set to 0.1 mm to get clear result instead of values with too much decimals. To define the design variable, Nastran SOL 200 needs two cards for each PID. One card is called "DESVAR" and is responsible for defining the initial value and the upper and lower boundary for the variable. For the initial value, the baseline thickness is kept. In another run, it is also checked, if an approach from the lowest possible values and the highest have any influence on the result. Since they result in the same values, the baseline thickness is kept as the initial value.

Then seven responses are created. Four responses have the type "displacement" to get the resulting displacements for both sides of the frame for both static loadcases. Two "frequency" responses track the seventh and the eighth mode in order to get their frequencies. The last mode is the "mass" type, and should track the global weight. Six of the seven responses are then constrained for the values, analyzed during the baseline NVH runs. Lower boundaries for displacements stop the structure to get more compliance than the baseline model, and lower boundaries for both frequency values prevent them to get lower. In many weight optimizing analysis, frequency preventing constraints and even frequency loadcases are not necessary, because with less weight in the structure also the frequency will get higher. The equation for eigenmodes (Equation (6.1)) is saying that either the mass has to get lower, or the stiffness has to get higher. Because constraints for global bending and global torsion are already preventing the structure to get lower stiffness, an additional check for frequencies might be superfluous, but only if the eigenmodes have the same deformation as the static loadcases.

$$\omega = \sqrt{\frac{k}{m}} \tag{6.1}$$

This simulation still kept both loadcase and the constraints in it, because the mode analysis is leaving out any constraints, and the shapes for torsion and bending can be slightly different than in the static analysis. In NVH analysis for other modules like the body-in-white, additional to global bending and global torsion, also eigenmodes for the compliant front end can happen. As a universal optimizing method for NVH loadcases, constraint eigenmodes should always kept in mind, despite a lower weight for the result.

Each loadcase can only have one constrained ID. That means that during this run only these constraints will be considered for convergence evaluation. If several constraints should be used for a loadcase, the constraints can be bundled using "DCONADD" cards. Also in this case, these bulk data cards are necessary to merge each two constraints for the three load cases. The last response is used as the objective and will minimize the total mass. As the last step, the additional parameters are defined. For optimization parameters, a maximum amount of 30 iterations is set to exit any simulation which will go over it out of made mistakes, and in addition, the value for responses and constraints for every design cycle is set to be printed out. After the control file is complete it is loaded in the cluster to provide a faster solving with twelve processors.

6.3.3 Read out the Results

When performing an optimization, MSC Nastran creates the following files:

- Filename.f04
- Filename.log
- Filename.op2
- Filename.f06

The .F04 file contains additional information for the analysis such as for the size of the database and the start and stop time for each module.

The .LOG file writes out system information such as the used system for the analysis, the progress for the analysis with the time for each step and if how the analysis ended. [25]

The .OP2 file is optional and has to be called with the parameter “PARAM, POST,-1”. It is a universal file format that can be used to display results using post-processing software.

The .F06 file is the most interesting file because it contains the results, written in text blocks. Most of the file is used to output the results from solving the matrix represented by strain energy entries. But it also gives information about for the grid point weight generator, if it is initiated, for singularities and for an entry called epsilon. The epsilon value gives the user information about the “ratio of the work done by the residual forces to the work done by the applied forces” [26]. The Equation (6.2) explains how the epsilon value is calculated. After running a Nastran analysis it is important to take a look at this value. If it is low, it is an indicator for a mathematical stable problem, but if the value is above the value of 10^{-9} the analysis is ill-conditioning, what means it contains too many effects like singularities, rigid body motion or unconnected degrees of freedom in it. [27]

$$\epsilon = \frac{u^T \delta P}{u^T P} \tag{6.2}$$

- u^T = the calculated displacement vector
- δP = the residual load vector
- P = the applied load

Checking for a low epsilon value after an optimization takes more effort, because it is generated for each loadcase and iteration. If an analysis resulted in an error it is also reported in the .F06 file, tagged with the words “FATAL ERROR”. Nastran also provides an error code that can be looked up in an error report. Often also solutions for an error code are provided.

When doing a design analysis, the results file contains more useful information about the progress of the optimization:

- Design Summary for each cycle
    - Objective
    - Design Variables

    - Inspection of Convergence
        - Soft Convergence Decision Logic
        - Hard Convergence Decision Logic
- Summary of Design Cycle History
    - Objective History
    - Design Variable History

Using the additional design parameter entry “DOPTPRM; P2; 9; P2CR” expands the output with the following entries:

- Design Summary for each cycle
    - Responses
    - Constraints

Where the values are mostly interesting to understand the way, the optimizer worked, the history results for the objective, the different variables and the constraints are interesting for an overview of the whole optimization progress. Unfortunately, the responses are not summarized at the end, so the values have to be taken for every single design cycle. For every response there are two values written, the first one, called input value, is the value that resulted out of the approximation and the output value is the actual response that is calculated for the iteration. At the end where the design history is summarized, there is also a value for the deviation between the approximate model and the exact analysis that explains why the input and output values are always slightly different.

To get an overview for the optimization, it is helpful to generate three plots: the first plot is the objective versus the design cycles. It shows the overall development of the structure during the analysis and can also make show the differences from the initial design to the continuous solution and the discrete result at the end. In addition to the objective, the second plot with the constraint violation versus the design cycles can show when or if all constraint can be met during the optimization. The last plot that can be interesting are selected design variables versus the iterations. The selection of variables should be considered, because otherwise the plot would contain too much information and can easily make no statement at all.

6.3.4 Reviewing the Results

To get a feasible result, Nastran needed seven iterations, and five and a half hour (approx. 47 minutes per iteration).

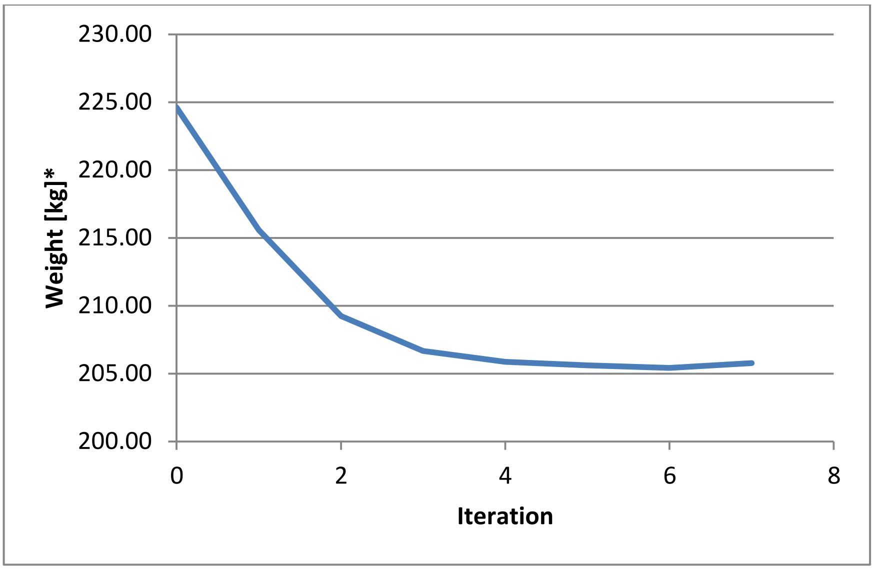

Figure 6-8: Gauge optimization: Objective vs. Iteration

Figure 6-8 shows the weight development during the optimization process. One can see that the first iterations have big changes, while the last iterations will get close to a certain value until the optimization converges. For the last design cycle, there is a slight increase for the objective. That's because discrete values are used. The way, NASTRAN uses these given values is, that it'll perform the complete optimization freely, until it finds a feasible result, and then assign the gained thicknesses to the defined discrete values. After doing this, it'll run a last analysis which is displayed as iteration number seven. The overall weight goes down from 224.6 kg* to 205.8 kg* which is a saving of about 18.8 kg (9.14 %*). Therefore all given constraints are still met.

		New Design	Baseline
Modal Analysis:	#7	25.44 Hz*	24.35 Hz*
	#8	32.375 Hz*	29.75 Hz*
Bending:	7	-1.633 mm*	-1.766 mm*
	8	-1.624 mm*	-1.766 mm*
Torsion:	1	-3.499 mm*	-3.50 mm*
	2	3.499 mm*	3.50 mm*

Table 6-1: Gauge optimization: Results for Responses

Table 6-1 is showing the assigned constraints before and after the optimization. As expected all displacements are kept the same during the run, which is mostly the target for the solver. As written before the decrease of weight and kept stiffness at the same time led to increasing modal frequencies for global torsion and global bending.

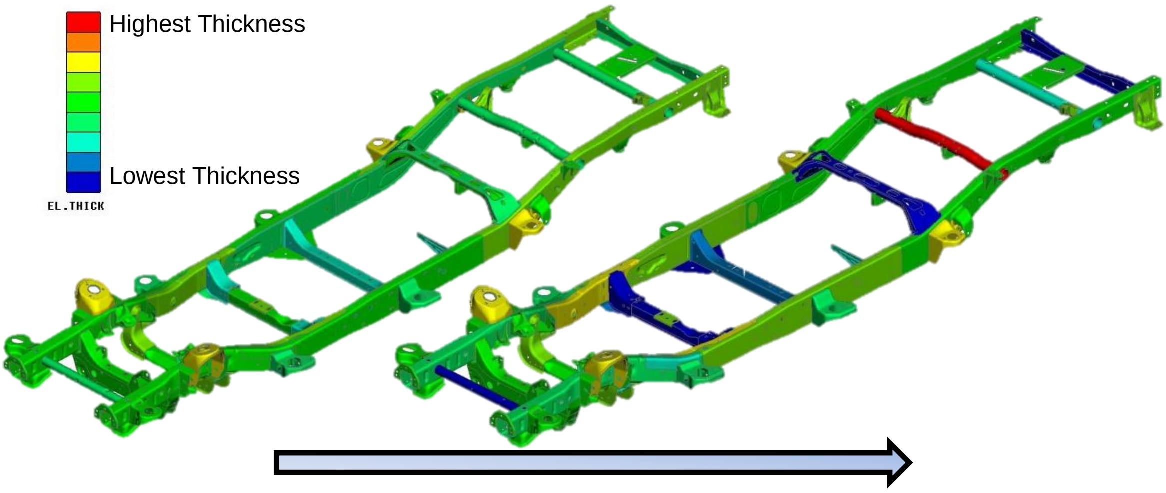

Figure 6-9: Gauge optimization: Changes for thickness

Figure 6-9 compares the thickness of the baseline model to the thicknesses of the optimized result. As the first topology result shows, the majority of the changes are made in the crossmember area. The most important crossmember which is responsible for the global torsion is the one which is located in the rear section and which thickness goes up to the maximum value allowed during the optimization. Especially the longitudinal beam section on both sides can only the slightly optimized using one uniform thickness, because, a major change of the cross section would led directly to a change of the moment of inertia (Equation (6.3)).

$$I_y = \frac{wh^3}{12} \tag{6.3}$$

- h = Height of the beam
- w = Width of the beam

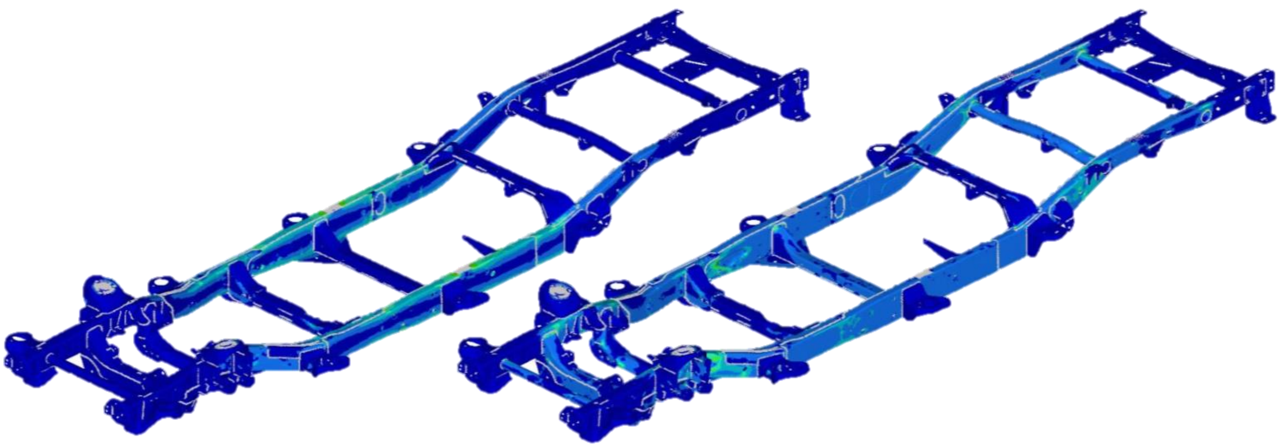

Figure 6-10: Gauge optimization: Stresses for static loadcases

Figure 6-10 shows the stresses for both static loadcases. With a maximum von Mises stress of 75.916 MPa* for torsion and 65.2 MPa* for bending, the stress stays in the same range as the baseline model.

6.3.5 Summary

Gauge optimization for a structure is a great tool to get a first overview about the needed thicknesses, when all the constraints are known. It is set up quickly and can be performed with a lot of different programs. The additional work that has to be done is only slightly different, when a working analysis file already exists and the use of the Nastran integrated solver offers the opportunity to work in an already familiar environment. Knowing the loadcases, a gauge optimization should be the first method to do, to get the perfect thickness needed. Once a gauge optimization input file is build, the setup can be used for other structures as well, only making slight changes for the optimization cards. The following workflow describes to way to set up a gauge optimization.

1. Change current solution to solution 200, considering to modify the loadcases for optimization
2. Decide which parts to be optimized for thicknesses
3. Connect parts using variables, decide for a range in where the algorithm can act
4. Add discrete values, covering the set ranges of thicknesses to make sure, the result is producible
5. Define the responses that should be constrained and working as an objective
6. Set limits for the created responses
7. Assign the constraints to specific loadcases or the complete analysis
8. Define the objective
9. Set additional parameters and control cards
10. Run the optimization
11. Review the result
    a. Check the results file for any fatal errors or high epsilon values
    b. Control if the optimization led to hard and soft convergence and there is no constraint violation
12. Analyze the result as for any static analysis, checking for displacements and stresses in the structure

6.4 Material Optimization

After the first optimization gains good results for optimizing the structure only with one material, the next step for a lightweight design is, to create a model where every material is used at its best position. Right now, only different grades of steel are being used for the baseline.

Both static loadcases are completely in the range of yield stress. Therefore, the criterion for tensile strength is not interesting. The same goes for buckling criteria: All parts of the structure are showing low stresses, and have, except for the cross members, high thickness and are stiffened with beads. Also, the crossmembers, where the thickness goes down, have a small surface, so they are also not likely to bulge. Knowing this, several materials are compared for their usage in situations where stiffness is required. For comparing materials, besides steel, aluminum alloys, magnesium and titanium are used as comparatives. As written in Chapter 3.3.3, the specific value for stiffness tasks is:

$$\frac{E}{g \cdot \rho} \tag{6.4}$$

	Unit	Steel	Aluminum	Magnesium	Titanium	CFK-UD
E modulus	N/mm^2	210,000	70,000	45,000	110,000	125,000
Density ρ	kg/dm^3	7.85	2.7	1.75	4.5	1.7
$\frac{1}{\rho}$	dm^3/kg	0.13	0.37	0.57	0.22	0.59
$\frac{E}{g \cdot \rho}$	km	2,727	2,643	2,621	2,492	7,495
$\frac{E}{g \cdot \rho} \cdot \frac{g_{Steel} \cdot \rho_{Steel}}{E_{Steel}}$		1	0.96	0.96	0.91	2.75
Price	$\$/kg$	1.4	2	2.2	19	29
Corrosive Resistance		Yes, depends on alloy	Yes	Not in combination with stone impact	Yes	Yes

Table 6-2: Material optimization: Possible materials with their relevant properties

Table 6-2 shows the most important criteria for choosing the right material for the frame. When it comes directly to stiffness per mass, the best material would be carbon fiber. The mean costs for saving one kilogram are about \$3 to \$14 [12]. Using this material instead of steel would say that carbon fiber should be save at least 2 kilogram for every used kilogram of CFK in the model. Since the stiffness value is 2.75 times higher, the usage of carbon fiber can become a candidate close to the limit of money spend for new material. But carbon fiber can only be used with this calculation, when it would have homogenous behavior. The E modulus taken for unidirectional CFK is the best value, if the material is tested in drawing direction. The behavior for torsion and for pressure is completely different.

Aluminum is another material, which has almost the same coefficient for stiffness. What is not reconsidered in Table 6-2 is the high usage of volume taken by aluminum. If this material with the same mass as steel is bended under a force, the displacement will be lower because of the higher moment of inertia. For a simple comparison, a full material beam with the total weight of 1 kg and the length of 1m should be applied with 10N (Figure 6-11). Three materials are chosen to compare with each other: Steel, Aluminum and Magnesium.

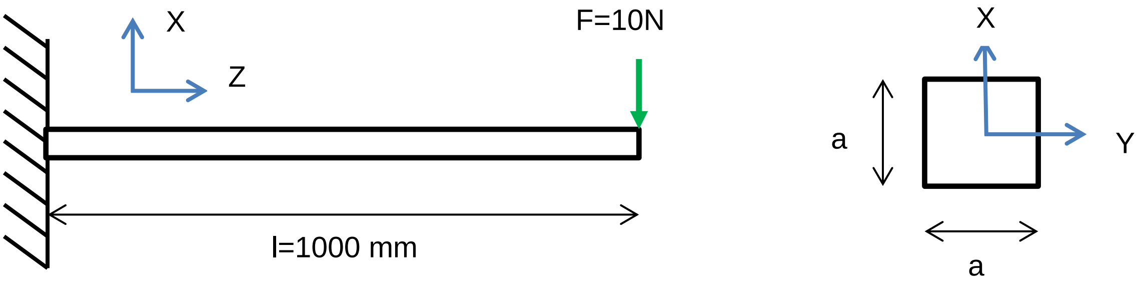

Figure 6-11: Example problem: Beam with bending force

$$V = \frac{1kg}{\rho} \quad (6.5)$$

$$a^2 = \frac{V}{l} \quad (6.6)$$

$$I_y = \frac{a^4}{12} \quad (6.7)$$

$$f = \frac{Fl^3}{3EI_y} \quad (6.8)$$

	E Modulus	Density ρ	Volume V	Cross section area	Length a	Moment of Inertia I_y	Displacement f
	GPa	kg/dm^3	dm^3	mm^2	mm	mm^4	mm
Equation			(6.3)	(6.6)		(6.7)	(6.8)
Steel	210,000	7.85	0.13	130	11.40	1,408	11.27
Aluminum	70,000	2.7	0.37	370	19.24	11,408	4.17
Magnesium	45,000	1.75	0.57	570	23.87	27,075	2.74

Table 6-3: Chosen materials for example and their results

As one can see in Table 6-3, only regarding the specific lightweight value can lead to wrong conclusions about a material. Because of its higher moment of inertia, both, aluminum and magnesium have less displacement with the same usage of weight. Magnesium which has the best results for the maximum displacements, because of its low density cannot be used for parts of the frame, because small stones from the road can damage the coating and then will corrode the magnesium alloy. Titanium and carbon fiber will also not be used for further investigation due to their high price.

After selecting a suitable material for the structure, areas have to be chosen where the material should be changed. The result of the first optimization shows that almost all of the selected crossmembers goes almost completely down to the lowest manufactural thickness for steel. Because aluminum has a lower density, the first optimization is about changing the material of the crossmembers into aluminum which will result in a lighter structure.

6.4.1 Aluminum Crossmembers

As the initial structure, the baseline will be taken except that the material cards for specific parts are different. Several approaches are made by starting with the values of the first optimized result, which also leads to feasible results with less iteration because most of the variables stayed the same or only changed marginal. The baseline is taken as the initial state, to compare the different optimization progress directly to the gauge optimization.

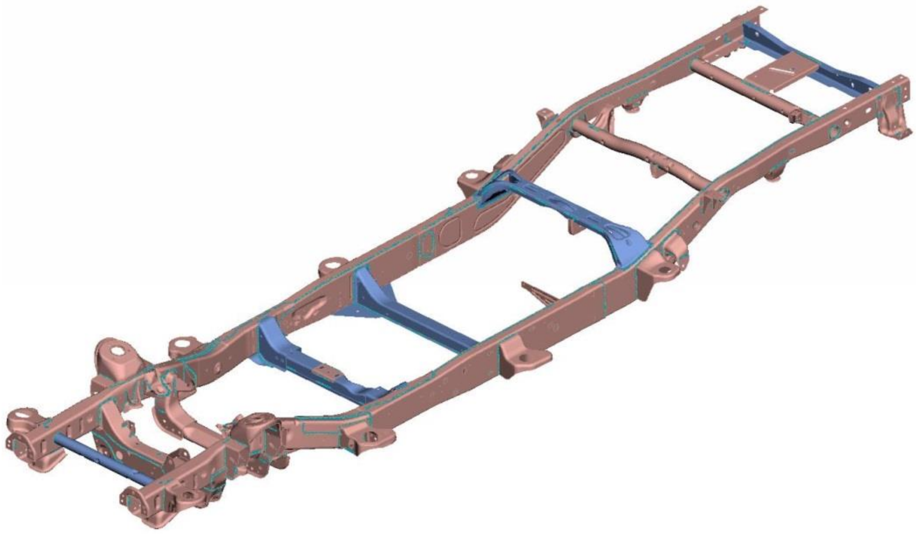

Figure 6-12: Selected parts for material optimization

Figure 6-12 shows the selected parts that are assigned to aluminum instead of steel. The small part on top of the second crossmember up front isn't chosen to be aluminum, because it is a reinforcement for the differential and it would go down to the lowest value due to missing loadcases for it. The second and third last crossmembers also are not chosen for this optimization because of their already high thicknesses they get after the first run. A tube made out of steel which already is about 5 mm thick, would approximately go up to a thickness three times of the steel's one, that is not manufactural anymore.

For the optimization procedure, again, NASTRAN SOL 200 is chosen because the input file for the previous gauge optimization already existed and only has to be modified for material optimization. The parts are assigned with the new material aluminum by just entering the values for pure common aluminum ($E = 70{,}000\ GPA;\ \rho = 2.7\frac{kg}{dm^3};\ \nu = 0.34$) since the structure has to bear only stresses below any yield strength, there is no interest to define a specific aluminum alloy.

The modified FE mesh then is exported as an updated .nas file. Because everything, except the material, stays the same in this file, it is quick work to change the already created command file in the Nastran file editor. Only the entries for the variables have to be changed to the new parts to be optimized. The number of entries and the PIDs for the variables stays the same as in the gauge optimization to give parts around the crossmembers the opportunity to react to the changes being made on them. The ranges for the parts which have to be made out of aluminum needed new defined ranges for their thickness which is defined from 0.7 mm to 6 mm.

The file is exported and run by the Nastran cluster. To gain a feasible result, six iteration are needed what takes two hours and two minutes to solve (approx. 20 minutes per iteration). That this optimization only goes 20 minutes per iteration because it ran completely alone on the cluster and can be taken as regular time for solving this analysis. The overall weight goes down from 224.6 kg* to 203.8 kg* which brings the savings comparing the baseline to 20.8 kg (10.2 %*). Therefore all given constraints are still met.

		Alum. Design	Baseline
Modal Analysis:	#7	25.64 Hz*	24.35 Hz*
	#8	32.8 Hz*	29.75 Hz*
Bending:	7	-1.62 mm*	-1.766 mm*
	8	-1.62 mm*	-1.766 mm*
Torsion:	1	-3.498 mm*	-3.50 mm*
	2	3.498 mm*	3.50 mm*

Table 6-4: Material optimization: Results for Responses

Table 6-4 shows the resulting responses for the optimized multi-material frame besides the given constraints. Interesting outcomes are the values for bending. The assigned discrete values have brought them 0.146 mm away from the lowest border. Since the increment is already 0.1 mm it wouldn't make much sense to define a finer discrete value and so this stiff behavior for the bending loadcase has to be accepted.

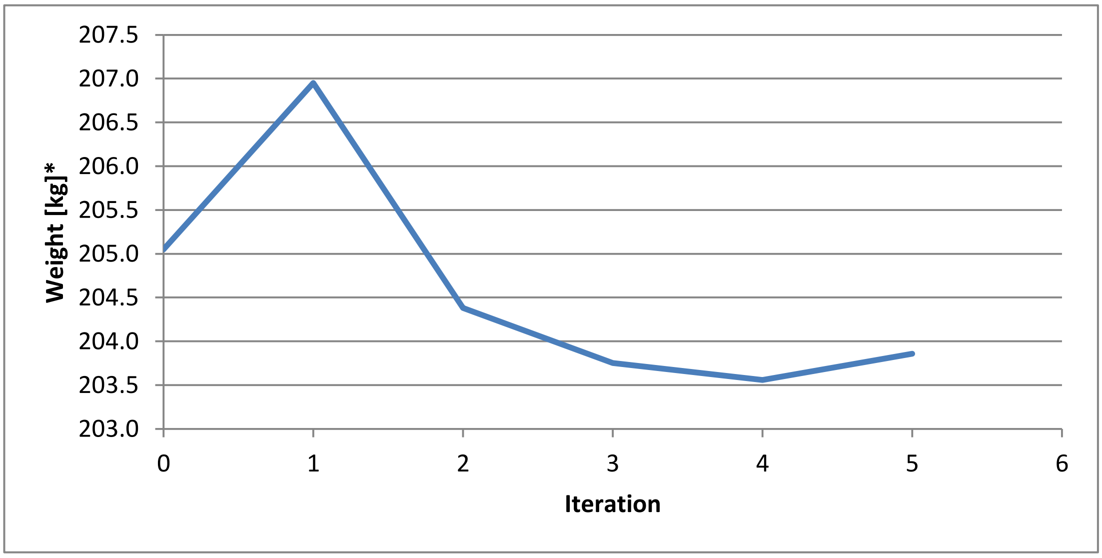

Figure 6-13: Material optimization: Objective vs. Iteration

In comparison to the first optimization run, this time the initial analysis penetrates the given constraints, because aluminum is assigned to the crossmembers, which still have a thickness calculated for steel. To avoid this violation, the algorithm first created a state, where no violation exists (Figure 6-13).

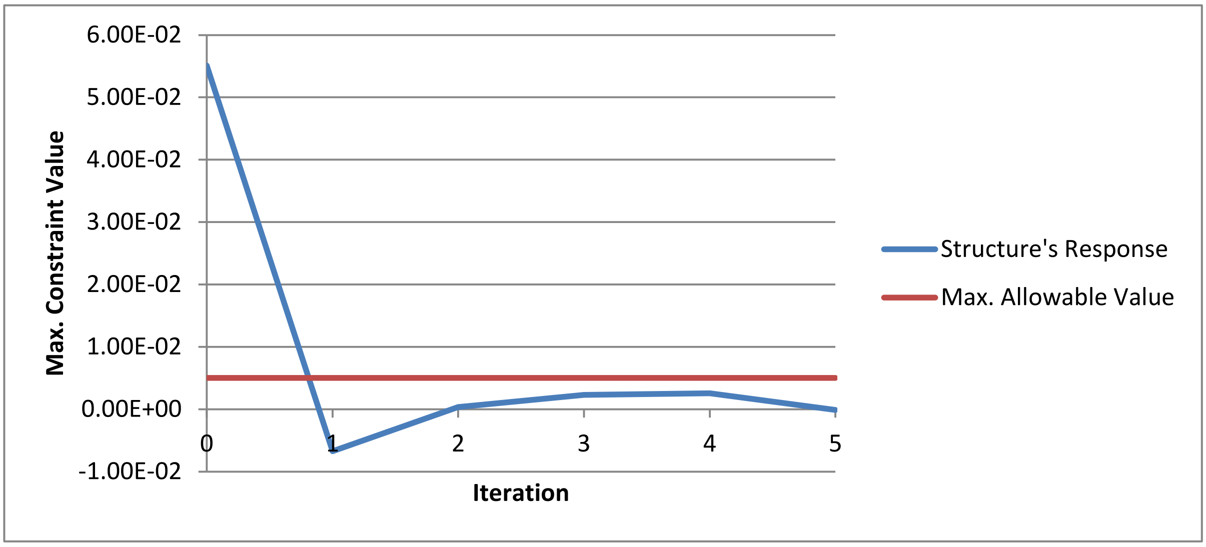

Figure 6-14: Material optimization: Maximum Constraint Value vs. Iteration

The entry for the maximum constraint value (Figure 6-14) explains the behavior of the objective curve. Due to the low initial start, there is a high violation which at first has to be fixed by making the structure a little bit too stiff and then begins with the optimization by getting closer to the allowable value of 0.005. The fourth iteration step brings the structure as close to the limit as possible. Because the last iteration will get the values defined by the discrete value parameter, the structure becomes heavier again and also stiffer which results in the final low value.

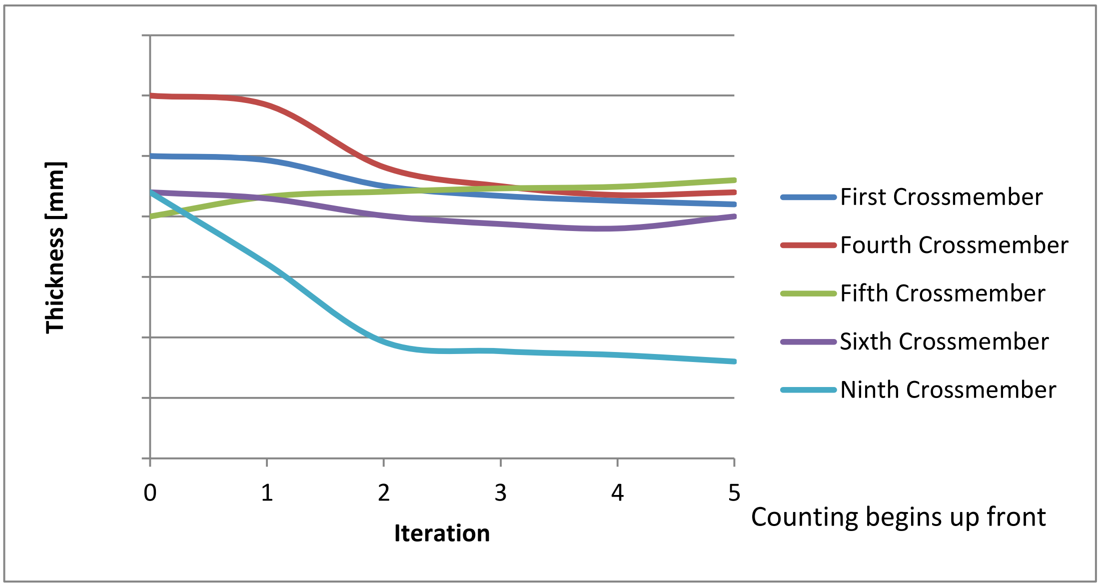

Figure 6-15: Material optimization: Thickness development of crossmembers

Looking at the development of the crossmembers (Figure 6-15), one can see a uniform behavior of the first, fourth and sixth crossmember which all showing lost wheight during the optimization. The sixth and ninth crossmember are showing a different development: The fifth one, who is also the center crossmember, slightly gains thickness during the analysis what results in almost the same thicknesses for the first, fourth, fifth and sixth profile. The ninth and last crossmember goes down to the lowest number what shows, that it does not seem to have any effect on the stiffness for global torsion and bending .It should be considered that loadcases like rear crash and towing are missing, which would affect this last part.

As during the gauge optimization, most of the parts go down with their thicknesses, except the seventh tube shaped crossmember that increases its thickness to a high, but lower value than during the gauge optimization. Also, several parts of the longitudinal beams gain about half a millimeter more thickness to meet the target.

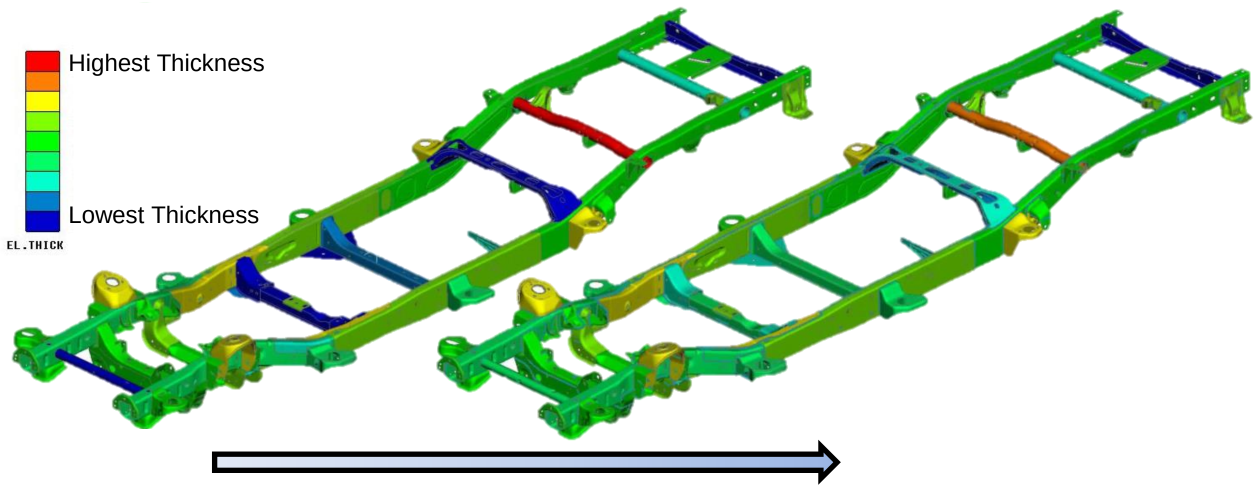

Figure 6-16: Thickness development from gauge optimization to material optimization

Figure 6-16 shows both optimized structures, from the first gauge optimization and from the material optimization. Besides the thicker crossmember, more interesting is that the whole structure looks more uniform again. Even the thick tube goes a little lower, but is still one massive part.

Figure 6-17 shows every von Mises stress of the frame which is above 25 MPa. This value is found during the analysis of several results as a good overview of the stress distribution. Red circles are marking the locations for the maximum stresses.

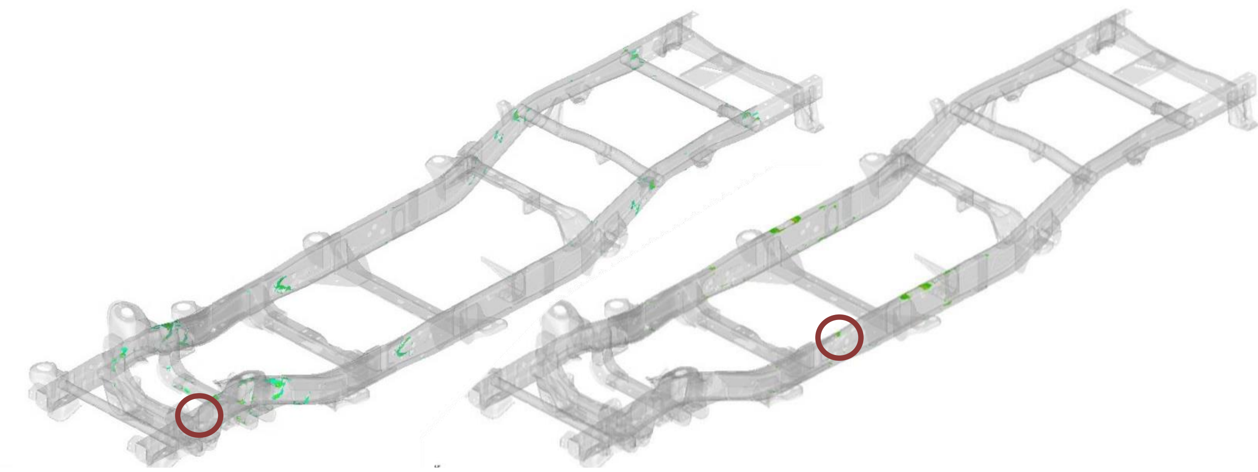

Figure 6-17: Material optimization: Von Mises stresses over 25 MPa

For torsion, the stresses are close to the beams, which are responsible for the front suspension. Like for the baseline test, there are also stresses where the second last crossmember is welded to the longitudinal beams. The maximum stress which is located, same as in the baseline test, at the welding for the engines support at the driver’s side, is with 76.15 MPa* the same as for the initial frame. For the bending loadcase, most stresses are located close to RBE elements, applying the bending load to the structure. The maximum stress occurs at a welding connection between the front longitudinal frame and the middle one on the driver’s side, and is with 57.7 MPa* even lower the in the baseline test.

Overall the additional weight savings that can be made by using aluminum is two kilogram (205.8 kg* for the optimized steel structure, 203.8 kg* for the optimized steel-aluminum structure). Even this small number can be interesting when it comes to weight savings in a vehicle, but in order to use aluminum profiles for crossmembers, each part has to be new designed because of the difficult manufactural connection between steel and aluminum. Instead, connection types like blind rivets, clinching or bolting has to be chosen what will also change the shape of the parts. Because the design of new parts is no longer subject of this investigation, the 2 kilograms can only be a prediction. Also this connection would directly have to deal with the corrosive weather conditions of the road, like water, snow, stone impacts and salt. All these influences can lead to a faster corrosive behavior between steel and aluminum what in the worst case can result in a loose connection and a danger for the passengers.

6.4.2 Whole Aluminum Frame

Besides just replacing a few parts with aluminum, it can also be an opportunity to manufacture the complete frame out of aluminum. The benefits of a lighter structure, which has good recyclability and good corrosive resistance, are good reasons for the use of aluminum. But there is also a disadvantage that comes with aluminum: the lower operational stability than steel. With the concept of the space-frame cars, which have a huge percentage of aluminum in their structure, AUDI showed, that it is possible, to manufacture autos out of aluminum that can stand the daily challenges a lifetime. [4] For designing the frame out of aluminum also analysis for fatigue has to be considered, which will add additional weight to the structure.

To start with making the frame out of aluminum, at first there is the problem that it is impossible to assign every part with aluminum instead of steel, because of the missing loadcases. The actual thickness for all the supports as well as for the crash elements will still remain unknown. A selection is made, that all parts will be assigned with the new material which are actually involved in the NVH loadcases and every other part will still be made out of steel. Again, the same selection as for the gauge optimization is made, referring to the topological analysis that is made in chapter 6.2.

Figure 6-18: Material optimization: Part selection for whole aluminum body

Figure 6-18 shows the selected parts for aluminum colored in green. Like before, the supports, the part for the spare tire and the crossmembers for the front suspension are left out for optimization. Using the preprocessor ANSA, every selected PID gets the new material card for aluminum. The updated file is, again, exported and saved as a .nas file, containing only the needed FE data. Then a command file is added that only contains the information for the normal static and dynamic analysis. This file is used with Genesis, to set up the necessary data for this optimizing process. Since Genesis uses a similar environment as Nastran, the user can quickly define the needed parameter without knowing this program before. A quick setup tool helps by defining every variable. Genesis' Design Studio is simple to use, when the complete file for the regular analysis task already exists. By just importing it into the Genesis environment, the program simply adds all the options and parameters for the optimization progress to the existing file and the exports is as an modified input file, which will be used by the cluster (Figure 6-19).

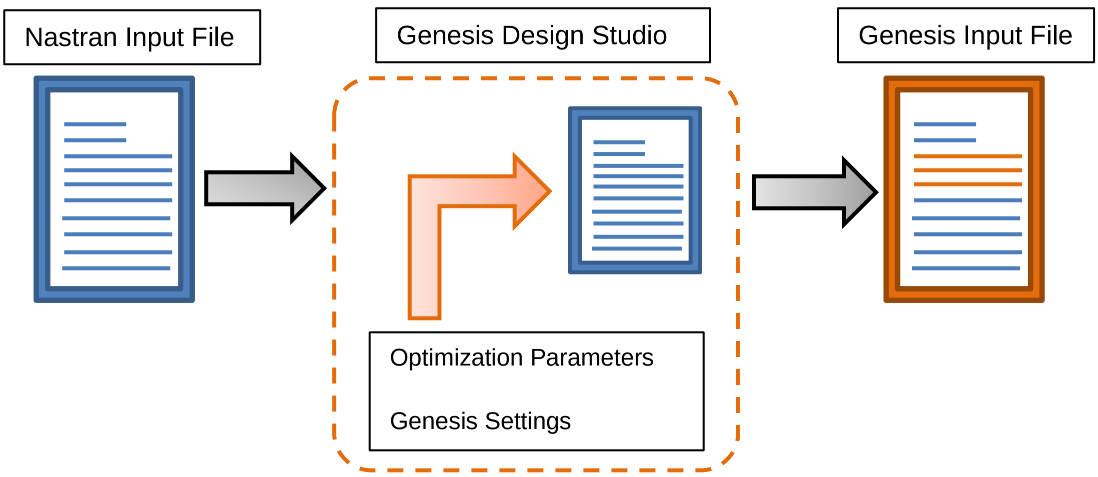

Figure 6-19: Work flow of optimization with Genesis

The allowable range for the thickness of the aluminum parts in where the optimization can take place first is defined from 1 mm to 5 mm. Then discrete values are added to get producible results. In addition, the number of processors and an option, that one gets an updated result file for the last design cycle is selected to save post-processing work. The first analysis leads to the results, that the given range with a maximum thickness of 5 mm is not enough to meet the given targets for the displacements. The analysis ends with a remaining constraint violation and a non-feasible result. The upper limit for the thickness is then corrected to 10 mm [4]. This change makes it possible for the algorithm to find a solution in ten iterations and takes one hour and 38 minutes (about ten minutes per iteration). The analysis leads to a final weight of 182.1 kg* which is a saving of 42.5 kg or 22.44 %*.

		Whole Alum. Design	Baseline
Modal Analysis:	#7	26.83 Hz*	24.35 Hz*
	#8	33.3 Hz*	29.75 Hz*
Bending:	7	-1.984 mm*	-1.766 mm*
	8	-1.733 mm*	-1.766 mm*
Torsion:	1	-3.498 mm*	-3.50 mm*
	2	3.498 mm*	3.50 mm*

Table 6-5: Material optimization: Results for Responses (aluminum frame)

Table 6-5 shows the results for the tenth iteration of the aluminum frame. The values for torsion and bending loadcases are both relatively close to the defined constraint, but both dynamic responses for the seventh and eighth eigenvalue are not significant higher than the previous optimizations. Considering the low weight of only 182.1 kg* the eigenvalue for global torsion is only 1.39 Hz higher than for the gauge optimization.

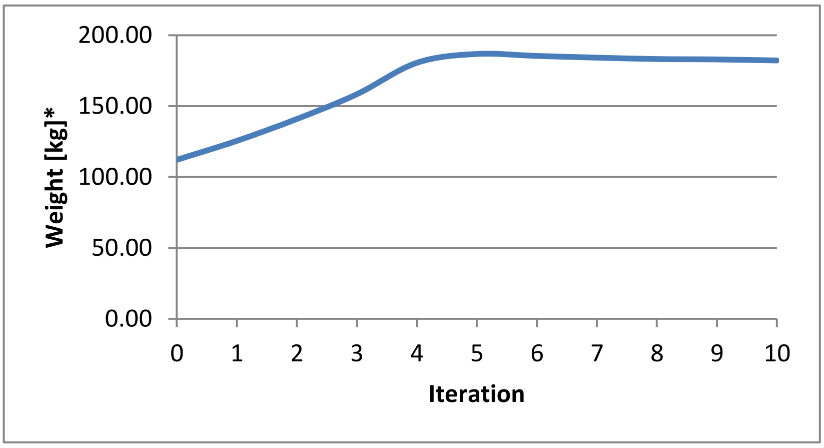

Figure 6-20: Material optimization: Objective vs. Iteration for whole aluminum frame

The curve for the objective this time looks different than the previous optimizations (Figure 6-20). Because the initial thicknesses all are designed for steel, again, the first action, the algorithm did is to get a state where there is not constraint violation. Apparently, it reached this state with the fifth design cycle. Then, the program optimized the structure by getting closer to the set constraints. Knowing this, an user can almost halve the time the analysis needs, just by selecting initial thicknesses which have roughly the same dimension as the final solution.

Figure 6-21: Material optimization: Constraint Violation vs. Iteration

Figure 6-21 displays the violation of all constraints in percentage. That the program takes five iterations to create a system in where the constraint violation is finally zero should be kept in mind for further investigations. In a medium sized structure, like a pickup frame, a first manual approach to get estimated thicknesses, to save time for the calculation can take more time, than the 50 minutes, the cluster takes. But if it comes to bigger structures like a complete car, or the same frame but with a finer grid, one should consider getting the initial state closer to the final result. A start can be to just use the specific density as a multiplier to get to the final value. Steel would have a value of 0.13 and aluminum a value of 0.37 which is 2.85 times the specific density of steel. Multiplying each thickness of the aluminum parts of the frame would bring the structure to the same weight as the steel frame. The optimization can save a couple of iterations by just using this modification.

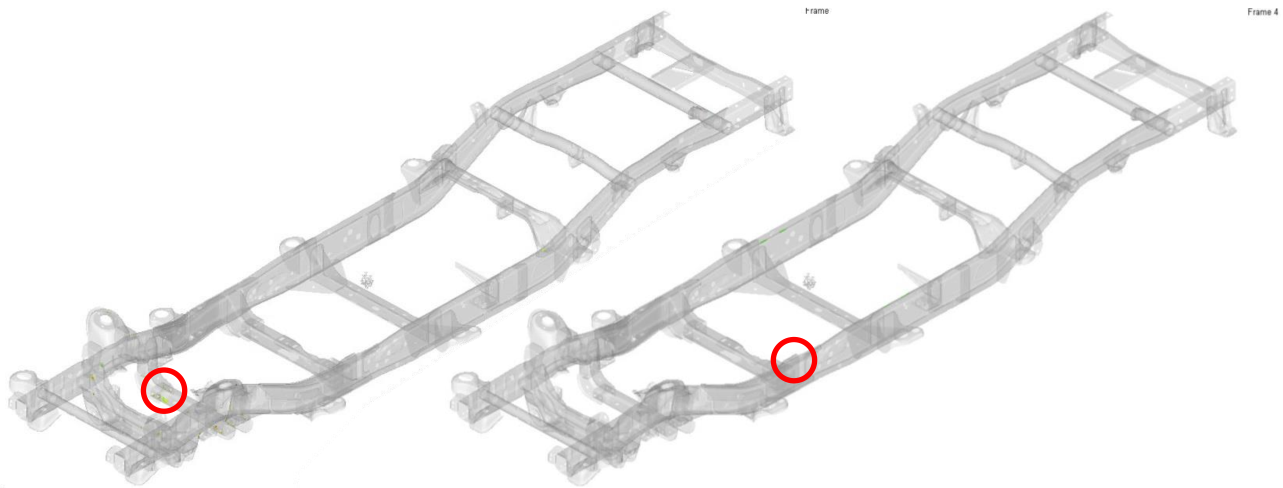

Figure 6-22: Material optimization: Von Mises stresses over 25 MPa (Aluminum frame)

Figure 6-22 shows the stresses for torsion (left) and bending (right) that are above 25 MPa. Where for bending almost no stresses occur in the complete frame, for torsion only a few elements from the third crossmember, close to bolting connections have stresses which are about 50 MPa. The maximum stress for torsion is happening at the inner bolting connection at the passenger's side with 50.728 MPa*. For bending, the von Mises stress of 56.632 MPa* is again at the connection at the seam welding connection between the front and middle longitudinal frame on the driver's side. Both locations are marked with a red circle in Figure 6-22.

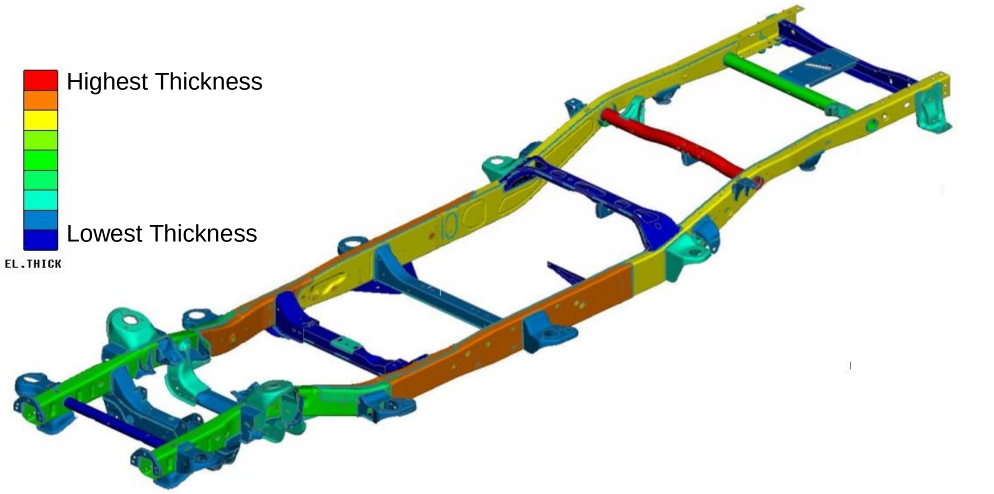

Figure 6-23: Material optimization: Thickness overview (Aluminum frame)

Figure 6-23 shows the thicknesses for the last iteration of the aluminum frame. This result has to be treated with caution. Where all the longitudinal beams are showing reasonable values, the crossmembers go down to the lowest possible value, saying, that they are not necessary for NVH loadcases. This first calculation can only be a prediction of what can be possible with aluminum. If it is compared with the first gauge optimization, the change from steel to aluminum saves, under the same optimizing conditions, additional 23.7 kg. Even if fatigue loadcases will be added, aluminum can stay the lighter material.

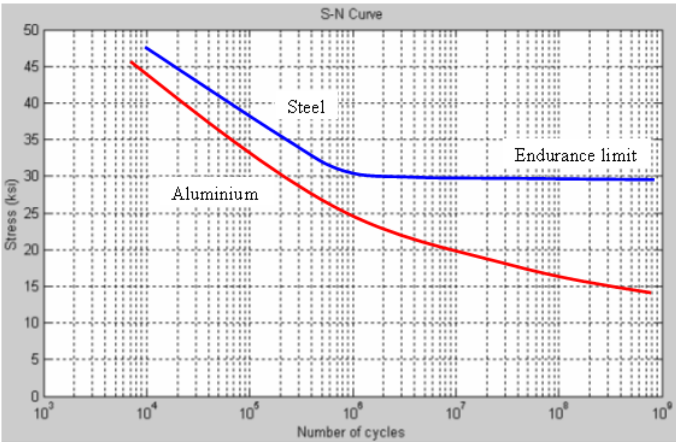

Figure 6-24: Stress vs. Number of cycles for aluminum and steel [28]

Figure 6-24 shows the fatigue behavior of aluminum in comparison to steel. When talking about the disadvantage of aluminum it should also be considered that aluminum alloys will behave the same as steel for up to 10^6 load cycles, a large number where first, tests has to be made to validate how many load cycles of a certain stress the car has to resist, during its lifetime.

Efficient lightweight design always considers a limited lifetime what especially is important for the use of aluminum alloys.

But the bare change from steel to aluminum is just the first step. If a manufacturer would decide to design their pickup frame with aluminum, more work has to be done. To use aluminum, the complete structure has to be new designed, regarding the different manufactural possibilities of aluminum. More extruded profiles for crossmembers and longitudinal beams should be used. They also can be hydro-formed to get shapes which will be adjusted to front and side crash loadcases. Once this design phase has finished, a gauge optimization can be used, to get the exact thicknesses using all the loadcases which are necessary for the frame.

6.4.3 Summary

Without redesigning parts, CAE assisted material optimization can only give a vague prediction of the potential how much weight can be saved, using a better suited material than before. The new material being used, still have to be chosen manually because many aspects are important for the right choice of the material besides the pure behavior for particular loadcases. A carefully made design optimization takes much longer than all the methods shown during this work. The complete new design of the parts can easily take a couple of days up to weeks, depending on the number of parts that have to be redesigned. The following work flow can be used for the progress of a complete material optimization:

1. Analyze the model for parts where a different material can make sense
    a. Hand calculations and the specific lightweight values are a good start for a first decision
    b. Also consider other properties of the materials (corrosive resistance, fatigue strength etc.)
2. Run a sizing optimization for a first impression of the weight saving potential
3. Design the new material related part
4. Change the already created sizing optimization to the updated FE design
5. Rerun the sizing analysis

6.5 Topometry Optimization

When there is an existing structure, which has to be optimized, topology optimization cannot be used, because it is not a good way trying to reduce density of a part with the same displacement as the initial part. In most cases this approach only leads to small changes for areas, where absolutely not stress can be found. This mostly happens for elements which are connected to RBE2 elements, because of their stiff behavior. A topological analysis of the frame only brings low densities for the location, where the load for bending is applied, and for QUAD4 elements which have adjacent slave nodes for seam weld connections. Blue elements in Figure 6-25 are indicating they have zero density.

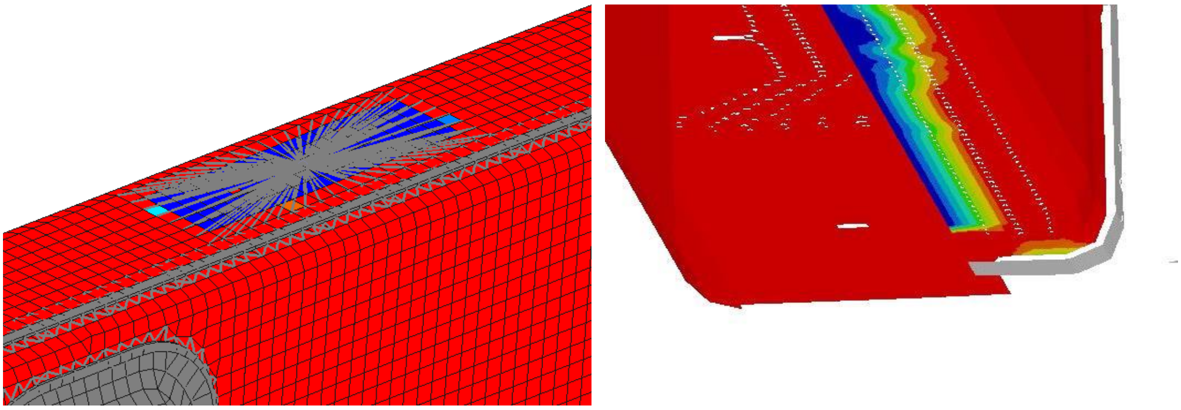

Figure 6-25: Results for topological attempt

Instead of only reducing weight, the optimization tool called topometry optimization gives the opportunity to vary the thickness of each element by defining a variable for every element and balancing the weight between the elements.

Genesis is used to do this operation, since it comes with advanced features for topometry analysis like a coarse option, symmetry option and several more. To continue with the progress being made with the gauge weight reduction, the NVH analysis file has to be updated with the new thicknesses. Unfortunately, Nastran does not come with the option to create an updated file for the last design cycle, so the value has to be entered manually with a preprocessor program. The NVH analysis file with the updated thicknesses from the result of the gauge optimization is imported into the Genesis Design Studio, and the quick sizing setup is used, to define, which part should be modified. The selection of the parts which can be optimized is a difficult progress, because of detailed work of topometry optimization. If only a single element is not used during the loadcases, it will go down to the lowest thickness possible. Several attempts are made to find the right parts for this kind of optimization. The first runs also include parts of the longitudinal beam behind the rear SPCs of the structure. This results in thicknesses which are almost close to the lowest value, except for an area close to the second last crossmember that goes up with the thickness to support this tube (Figure 6-26).

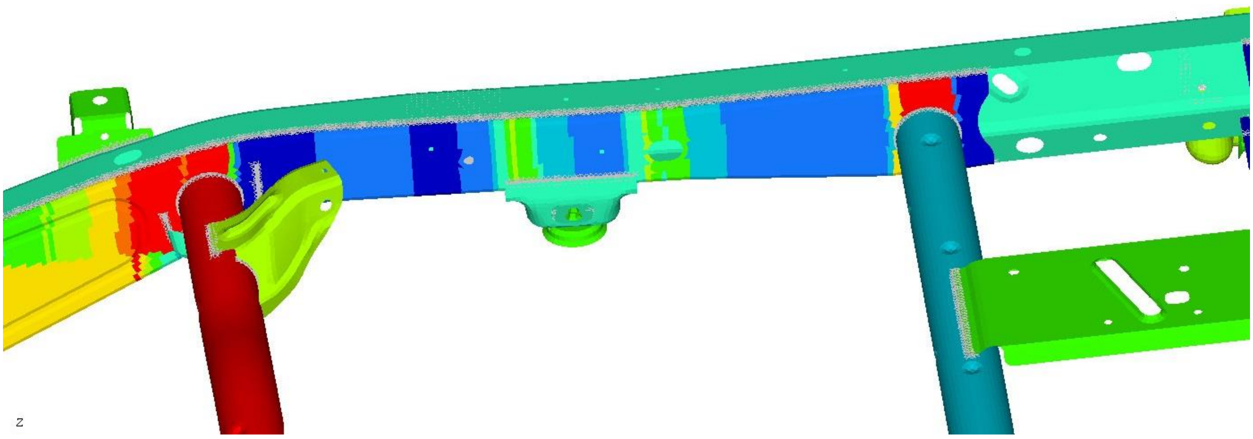

Figure 6-26: Topometry optimization results for the rear section

This leads to the choice to just use the geometry responsible for the loadcases. Therefore, the result from chapter 4.4.1 is taken to define the parts, used for this step. Figure 6-27 is showing the selection of the parts due to the topology optimization. Using ANSA, new PIDs are created which end right after the bearings. This is only made possible by using the topometry tool, since it will modify each element. The red colored parts are used during the following optimizations. It covers almost every loadpath of the topology result except the second crossmember of the front suspension.

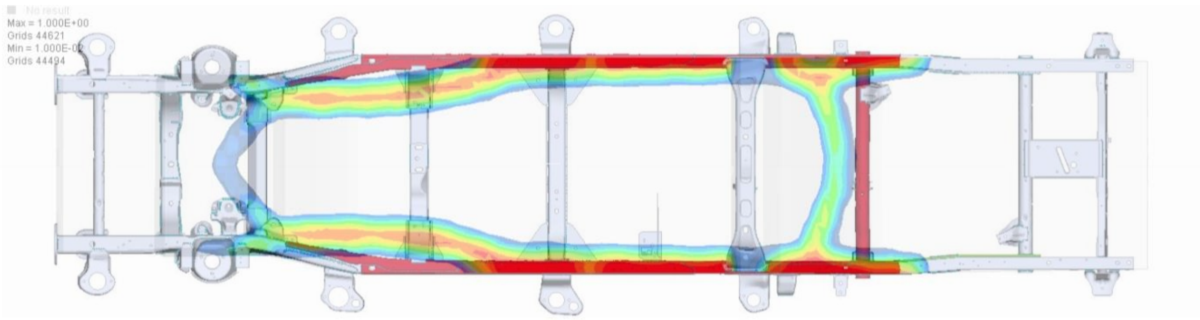

Figure 6-27: Topometry optimization: Merged topological result with frame

Next, the topometry parameters are defined. Every created variable is assigned to a topometry entry card. From all the possible settings, at first just the symmetry for the XY plane is set. For the possible range, 1 mm to 4 mm is chosen which cover all the possible thicknesses for sheet metal. Then responses and constraints are defined, using the same nodes and allowable displacements, like for the baseline model and an objective for minimum mass is created. The last action is to assign the right amount of processors being used during the analysis and that the solver should generate an updated FE geometry file for the last iteration.

Because of the large amount of variables being used, it takes four hours and 26 minutes to produce eight iterations and to find a feasible design (33 minutes per iteration). Every constraint is met as Table 6-6 shows:

		Optimized with Topometry	Baseline
Modal Analysis:	#7	25.69 Hz*	24.35 Hz*
	#8	32.31 Hz*	29.75 Hz*
Bending:	7	-1.715 mm*	-1.766 mm*
	8	-1.718 mm*	-1.766 mm*
Torsion:	1	-3.466 mm*	-3.50 mm*
	2	3.465 mm*	3.50 mm*

Table 6-6: Topometry optimization: Results for Responses

Unlike the previous optimizations, the topometry method managed it to get much closer to the defined constraint for bending which isn't possible with gauge and both material optimization runs. Also the eigenvalues are closer to the baseline, than during the calculations before. This time the maximum von Mises stresses are higher than for the baseline frame. For the bending loadcase, a stress of 103.941 MPa* occurs at a seamweld connection between the middle crossmember and the passenger's longitudinal beam. For bending a value of 100.887 MPa* occurs at the same spot, but on the driver's side.

All in all, this calculation produces a result, which is with 196.63 kg*, 27.97 kg or 14 %* lighter than the baseline frame. But this good result comes along with a thickness distribution, which is not manufactural (Figure 6-28).

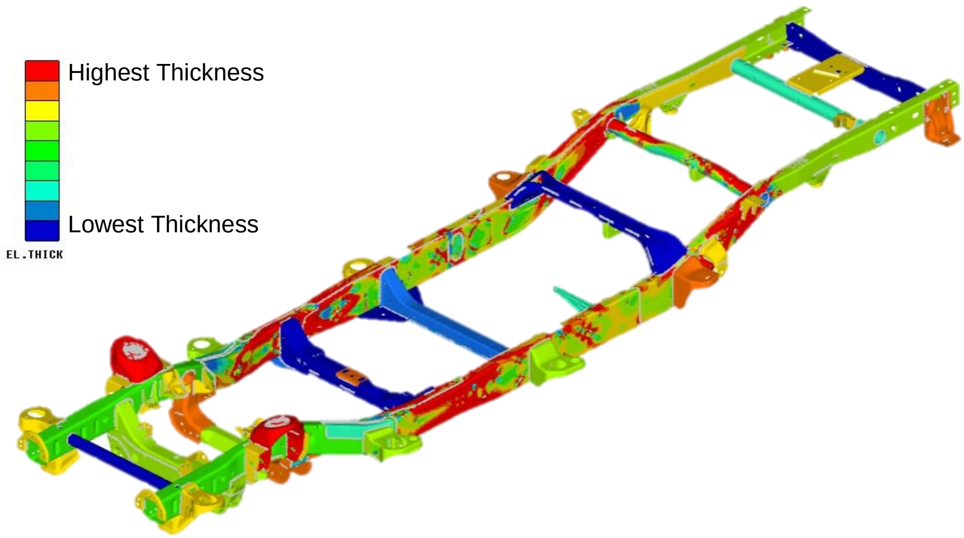

Figure 6-28: Topometry optimization: Thickness distribution

Since every element is now its own PID, it is almost impossible to use this result in a preprocessor in order to create new zones with a uniform thickness. This optimization can only be a more accurate indicator, where mass is needed, and where one can save material. An example can be found by looking at the seventh crossmember that clearly shows that the middle part does not have to be as thick as in the gauge optimization. Furthermore this topometry optimization can also be useful for other parts, where the use of patchwork blanks can lighten the structure. Then a result like this would be taken to define a new PID and variable where much material is located and a patch can be possible and then a following gauge optimization would precisely get the thickness for the patchwork blank and the rest of the structure.

The two left possibilities to save weight for the frame and to keep it producible are Tailor-Welded-Blanks and Tailor-Rolled-Blanks. With the option to join several elements in a defined area to one variable, which will be used for the topometry optimization, Genesis gives the user a way to quickly create both linear Tailored-Blank types. The following investigation will try to show several ways to design them.

The feature which will be used is called coarse option. It allows, to assign one variable for several elements automatically. This can be done either by bundling the elements by a certain number, or by coordinates.

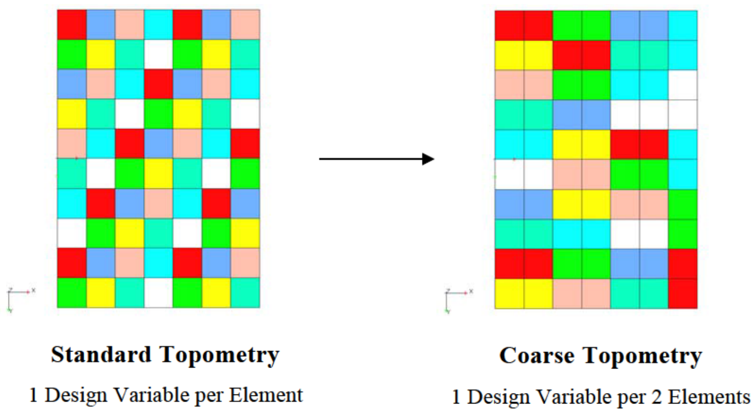

Figure 6-29: Difference between topometry and coarse topometry [29]

Figure 6-29 shows, how Genesis creates variables, using the coarse option for number of elements. It is important to know that this splitting into variables will be done before the actual analysis. The second option, to define variable by coordinates, splits a PID into variables for every value entered in X, Y and Z direction, beginning from global zero. This can be compared to cubes of the size, entered for the coordinates, filling the complete space. Every element which shares one of these cubes with other elements will be joined with them to one variable. Because of these linear borders, the use of this method is limited to linear Tailored-Blanks, where theses linear welding connections are parallel to one of the global planes (XY, YZ and ZX).

6.5.1 Tailor-Welded-Blanks Optimization

6.5.1.1 Automated Tailor-Welded-Blanks with 200mm and 400mm Width

At first, it is important to think about useful sizes for each blank that will be welded together to complete one part. Examples from actual part are reviewed for their width of each blank. [30] Since technically there is almost no limit to their size, but for the costs to manufacture them. The decision ended with two sizes, 200 mm and 400, which will be compared against each other. The Genesis-Input file from the first attempt is imported in the Genesis Design Studio again, and the coarse option with coordinates is added to each PID, chosen for the optimization. The length in global Y and Z direction has to be considered, because only a splitting of the variables in X direction is desired. Therefore the maximum dimension of the frame in Y and Z direction should be looked up and entered for Y and Z so that there’s no split for a part in these directions. Apparently, the seventh crossmember, which also should be optimized, has to be separated in Y direction without any separations in X and Z directions.

A problem with this coarse method is that it comes without any offset to start from. If one wants to separate the parts with a 100 mm width in X direction, but the parts first element has their nodes close to X=90mm, the first welded blank will only be about 10mm wide after the second one follows with the regular 100mm. Often the user then is left alone by spending time and manually creating variables in a preprocessor, by defining PIDs for them on himself.

The following review of the results deals about both chosen sizes 200mm and 400mm because their setup only differs from a change of several topometry input values. Running with 16 processors, the 200 mm analysis takes one hour and 27 minutes for five iterations (17 minutes per iteration) and the 400 mm run needed four hours and 16 minutes for seven iterations (37 minutes per iteration). That eventually another analysis interrupted the second optimization cannot be excluded.

		200mm width	400mm width	Baseline
Modal Analysis:	#7	25.40 Hz*	25.46 Hz*	24.35 Hz*
	#8	32.44 Hz*	32.92 Hz*	29.75 Hz*
Bending:	7	-1.721 mm*	-1.684 mm*	-1.766 mm*
	8	-1.731 mm*	-1.684 mm*	-1.766 mm*
Torsion:	1	-3.493 mm*	-3.264 mm*	-3.50 mm*
	2	3.464 mm*	3.463 mm*	3.50 mm*

Table 6-7: 200mm/ 400mm width: Results for responses

Table 6-7 lists the controlled displacements and eigenvalues for the last iteration. The finer Tailor-Welded blank of 200mm has values that are closer to the given limits, especially for bending, where the better variation in the thicknesses of the middle longitudinal frame gets the optimizing algorithm closer to a lower weight and a frame with an equal stiffness like the baseline model.

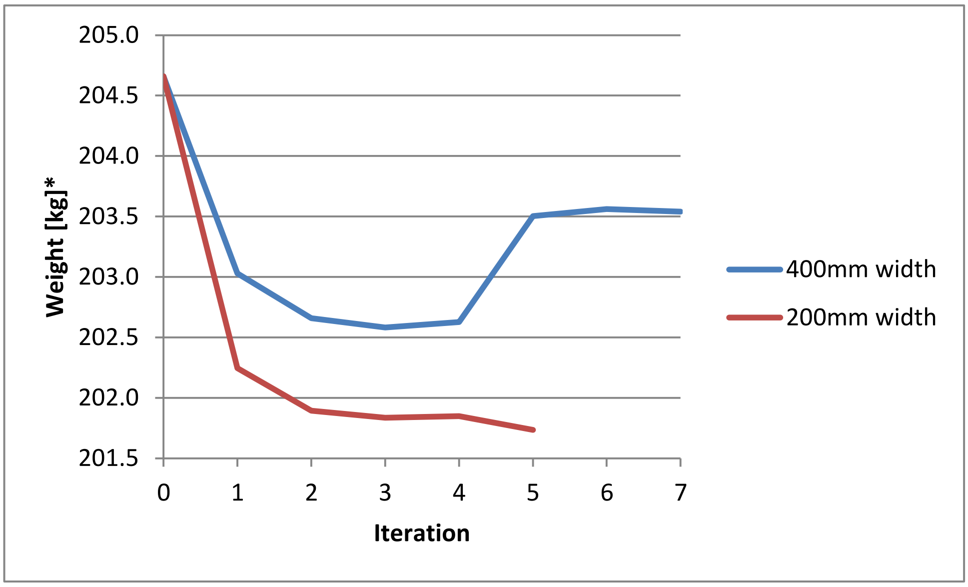

Figure 6-30: 200mm / 400mm width: Objective vs. Iteration

Figure 6-30 shows both weight developments during the optimization. As expected, the finer Tailor-Welded-Blank frame delivers, with its result of 201.74 kg*, a lighter result than the 400mm solution and manages to save additional 4.06 kg in comparison to the gauge result. An interesting development shows the curve for the coarser optimization: After finishing the optimization for the continuous design problem, Genesis starts to optimize the frame for the discrete values given. What at first looks like a missed global minimum is a value that cannot be used because of the given discrete values. This leads to a much higher weight of 203.54 kg* which is a difference of 1.8 kg. During the complete optimization there is not any Constraint violation as shown in Figure 6-31.

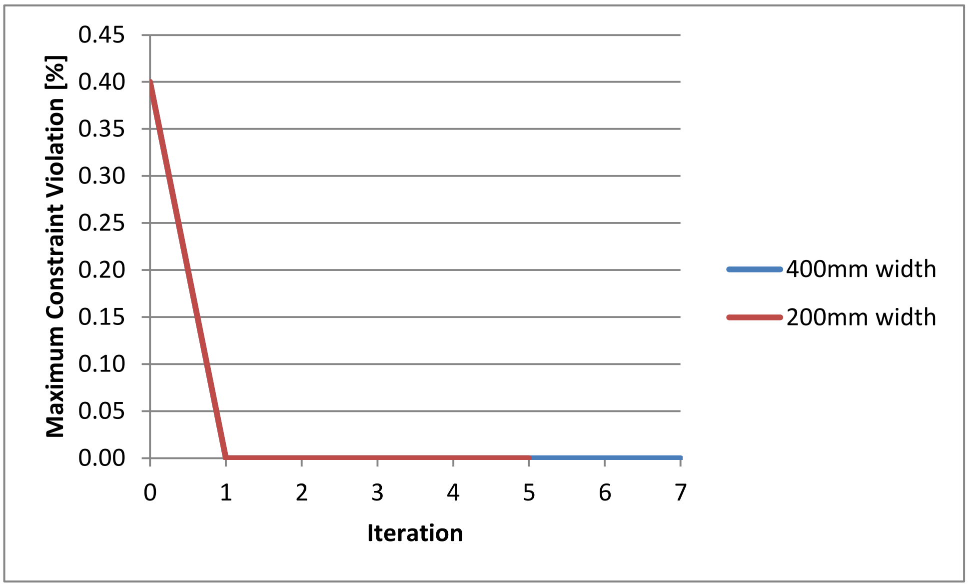

Figure 6-31: 400mm / 200mm width: Maximum constraint violation vs. Iteration

Taking the value from the fourth iteration, there is a difference of only 110 gram between both versions, a value which wouldn't justify twice as many parts to be welded together, but since the result should also be producible, discrete values have to be used what affects the coarser structure visibly. Figure 6-32 displays the last iteration for the finer version (top) and the last iteration for the coarse version (bottom). One can see the differences in the frame, by using different wide blanks.

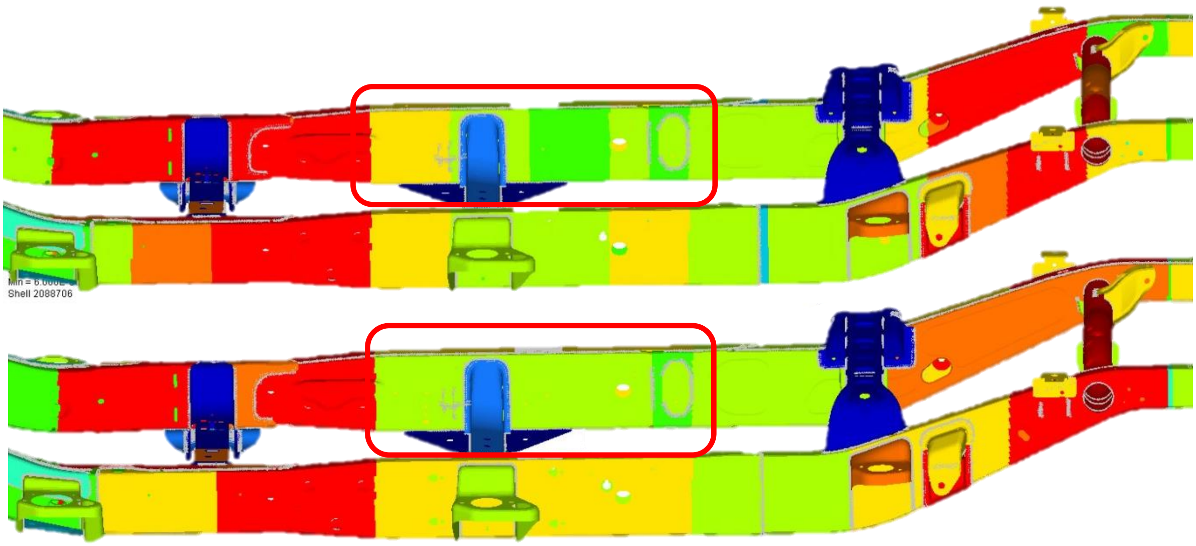

Figure 6-32: 200mm / 400mm width: Comparison between blanks for both results

At first, a review is made, if the increment between adjacent thicknesses is producible. If, for example, there would be a blank of 4 mm right next to a blank of 1mm there would be difficulties during the welding process. The lowest thickness for a blank during the analysis for the 200 mm version is 1.6 mm. This blank only has one neighbor with a thickness of 2.1mm what makes an increment of 0.25 mm for each side of the sheet metal to weld. The thinnest blank for the 400 mm optimization during the fourth iteration is 2.1mm with one adjacent blank of 4mm. This leads to a relatively high increment of 0.95mm for both sides. According to an empirical formula, thankfully provided by WISCO Tailored Blanks GmbH, the maximum possible increment from one thickness to another is 100%, so that even 2 mm can be welded with 4 mm. So the increment of 0.95mm will be producible.

The red boxes in Figure 6-32 are marking an area which will be compared between the two chosen versions and to show the differences. Four variables are used for the finer version, are the coarse one only needed two variables.

	First blank		Second blank	
Thickness 400mm version 4^{th} iteration	2.74 mm		2.44 mm	
Thickness 400mm version 7^{th} iteration	2.8 mm		2.5 mm	
Thickness 200mm version	3.0 mm	2.5 mm	2.4 mm	2.5 mm
Mean Thickness for 200 mm version	2.75 mm		2.45 mm	

Table 6-8: 200mm /400mm width: Comparison of thicknesses

The fourth iteration without discrete values brings closer values to compare the results for the second row of Table 6-8. Both times the average value is only 0.01mm off from the actual value taken for the fourth iteration of the 400mm wide blanks. With this fitting for most of the blanks being designed during the continuous optimization, it explains the minor differences in weight for the continuous result and the 200mm wide blank and also explains why the discrete result for the 400mm Tailor-Welded-Blank frame is so much heavier than the finer version. To meet the requirements for the displacements, most blank thicknesses are rounded up to the next discrete value what results in a stiffer and heavier structure.

The distribution of von Mises stresses is for both versions limited to the locations around the load appliance. Starting with the 200mm wide Tailored-Blanks the maximum stress of 81.118 MPa* occurs during the bending loadcases and the stress of 88.105 is calculated for the torsion loadcases. For the 400mm wide Tailored-Blanks, almost the same values of 77.904 MPa* for the bending loadcase and 90.016 MPa* for the torsional loadcase happen at the same location like for the 200 mm Tailored-Blank frame, in a CTRIA element, which connects the middle crossmember with the longitudinal beam. Because of the behavior of CTRIA elements this value should be treated with caution, the value for the same location with rectangular elements might result in a lower maximum stress.

6.5.1.2 Semi-Manual Designed Tailor-Welded-Blank

Both results which are showed in the chapter before, are created with relatively less effort, can be produced and can save about three kilogram for the frame. But this value is not the best value that can be gained by a Tailor-Welded-Blank manufactured frame. Because the results before are created with variables, which are not considering any thickness distribution as it can be seen for the frame topometry optimization without the coarse method, more weight can be saved by creating blanks which are perfectly covering the relevant areas where stiffer behavior is needed.

The first step is similar to the automated solution. The Nastran analysis file with the updated thicknesses from the gauge optimization is imported in Genesis, and the topometry optimization is set up. Again, the coarse option is used, but this time to create strips that have exactly the average element size, to get a thickness distribution parallel to the seamweld connections which will be made for the Tailor-Welded-Blanks. X=6mm is entered, since the modeled FE grid has 6mm elements size.

Taking nine iterations the analysis ended after two hours and 44 minutes (18.2 minutes per iteration). That there is a total amount of 2140 variables does not seem to have much effect on the time the optimizer takes for solving the problem.

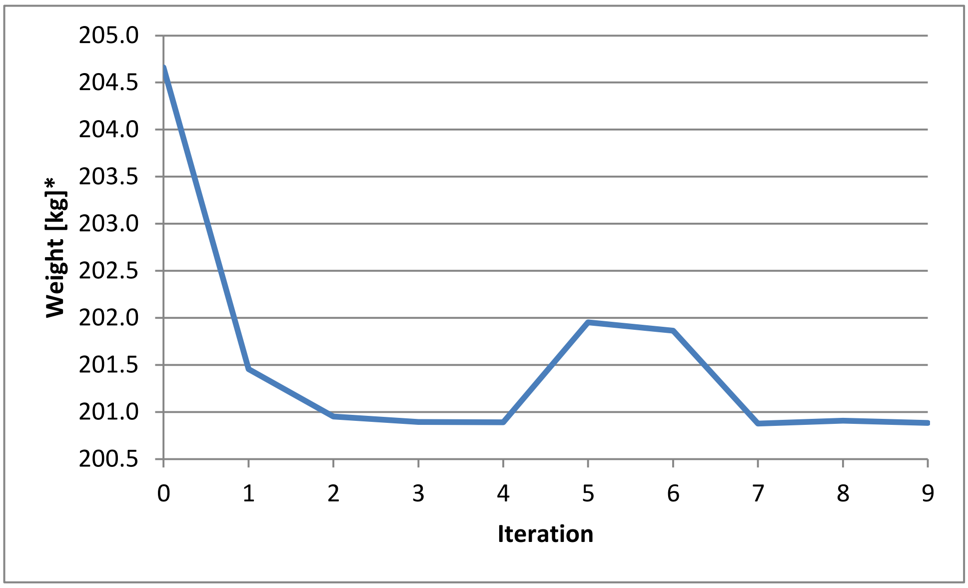

Figure 6-33: 6mm width: Objective vs. Iteration

The weight development shows a run where at first the algorithm solves with continuous variables up to iteration four and then begins optimizing the frame with discrete values (Figure 6-33). In the end, the difference of 200.891 kg* for the fourth iteration to 208.883 kg* for the last, ninth, iteration is almost not visible, but the choice to take discrete values doubles the time for the optimization. The result totally differs from the first topometry try that has a result which is difficult to interpret. This time there is an update FE geometry file that can actually be used to create blanks. Figure 6-34 shows the side view for the last iteration. Looking at the variables, one can see that often there are small strips with different thicknesses than the adjacent strips. Therefore, the following designing of blanks needs rules to go for, to create new zones with uniform thicknesses.

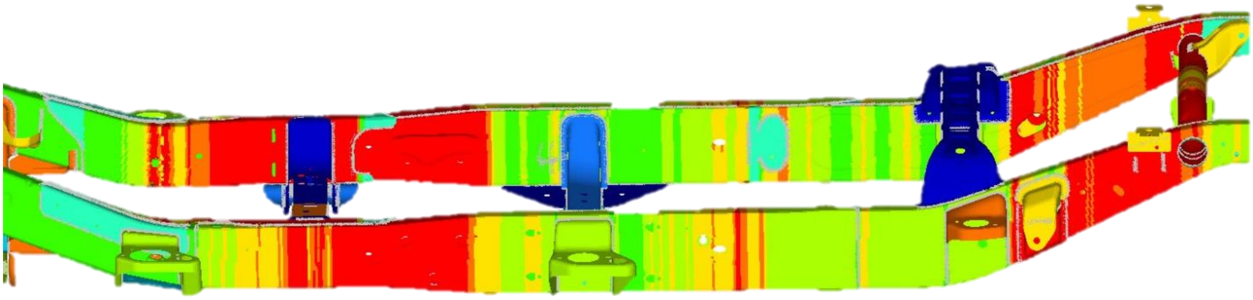

Figure 6-34: 6mm width: Tilted side view of optimized parts

Using the preprocessor ANSA, the result is taken to design the new blanks. All zones which have the same thickness are bundled together to one PID. If there is any small strip within that has a different thickness, it is still merged together. Then areas are merged, where the strips have more or less the same thickness. Left are areas with a lot of different thicknesses in them. Figure 6-35 shows one of these areas.

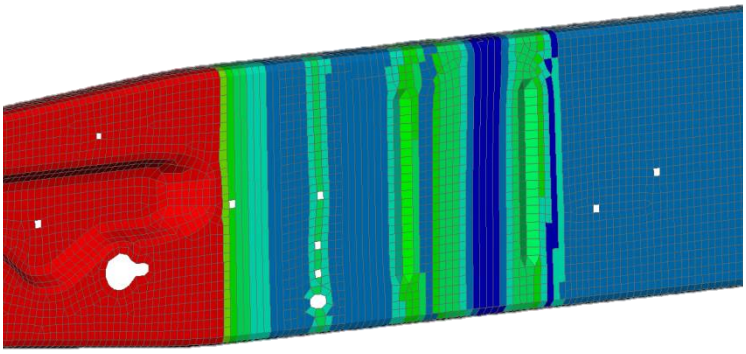

Figure 6-35: Semi-manual TWB: A region that will be merged to one blank

They are the last category of blanks which are created. At the end, every PID being created gets its old thickness back from the initial state to start the optimization from. The outer part for the middle longitudinal beam is displayed in Figure 6-36 as an example to demonstrate the selections being made. Every black box stands for a new PID being created. Here the difference to the automated selection for the borders of the blanks can be seen. Where the second blank covers a wide area there are also some smaller blanks, where less thickness would be necessary. Red boxes in Figure 6-36 are showing the automated way, Genesis defined the blanks for the 400mm version.

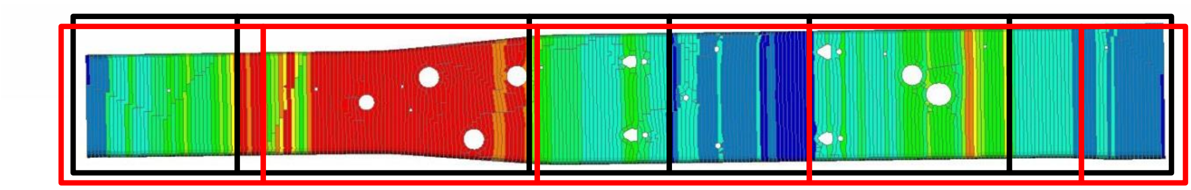

Figure 6-36: Comparison of chosen blanks for automated 400mm and manual result

After defining the variables, the FE mesh is exported and merged with the .dat command file, to do another optimization. This time the file is used, to create another sizing optimization, using the created PIDs from the previous topometry optimization as variables. The setup for the optimization is quickly done by the assisted quick sizing setup from Genesis, adding the constraints, the objective, and an option for symmetry. The range for thicknesses is set from 1 to 4 mm.

The optimization only needs nine iterations and runs three hours and 28 minutes (23.1 minutes per iteration). The result, as shown in Table 6-9, is a frame which is with 200.88 kg* 860 gram lighter than the 200mm automated designed version. The important difference is, that this structure can achieve less weight with less blanks being used (manual designed version: 58 blanks; 200mm width version: 77 blanks; 400mm width version: 44 blanks).

		Topometry optimum	Semi-manual TWB	Baseline
Modal Analysis:	#7	25.69 Hz*	25.44 Hz*	24.35 Hz*
	#8	32.31 Hz*	32.41 Hz*	29.75 Hz*
Bending:	7	-1.715 mm*	-1.717 mm*	-1.766 mm*
	8	-1.714 mm*	-1.726 mm*	-1.766 mm*
Torsion:	1	-3.466 mm*	-3.49 mm*	-3.50 mm*
	2	3.465 mm*	3.489 mm*	3.50 mm*

Table 6-9: Manual TWB: Results for responses

Like for the previous TWBs, the constraints are met better than for the sizing optimizations before, because of the zones, which are much closer to the optimal thickness distribution (Table 6-9). Comparing the designed TWB with the optimal topometry optimization for each element, the optimal result is still stiffer and 4.25 kg lighter, but designing zones manually brought the weight nearer to it.

Next the von Mises stresses are controlled. Again, maximum stresses for the static bending and torsion loadcase happened in the CTRIA element close to the middle crossmember. For bending, the maximum value is 83.92 MPa*, for torsion the value is 94.62 MPa*. Furthermore, looking at the locations are most of the stresses occurring, Figure 6-37 shows every stress over 25 MPa. One can see the low stresses, which are limited to the location of load appliance.

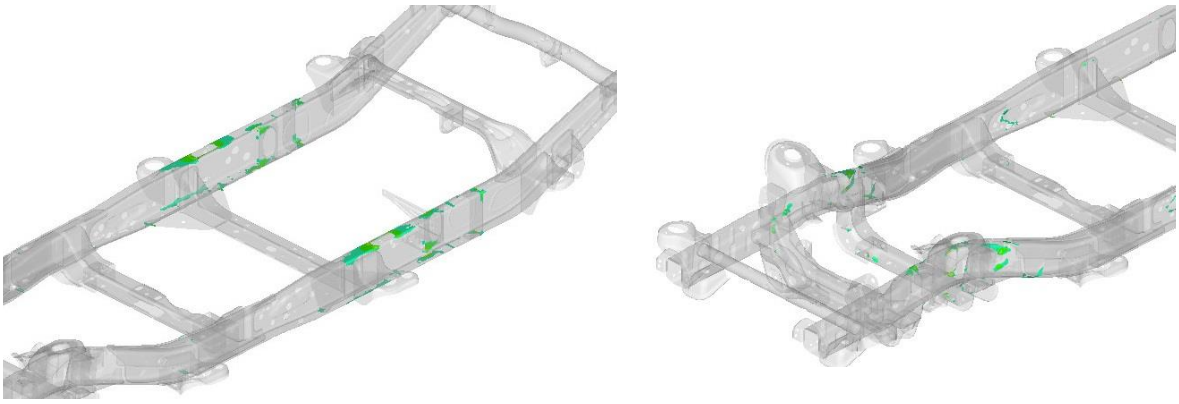

Figure 6-37: Semi-manual TWB: Von Mises stresses over 25 MPa

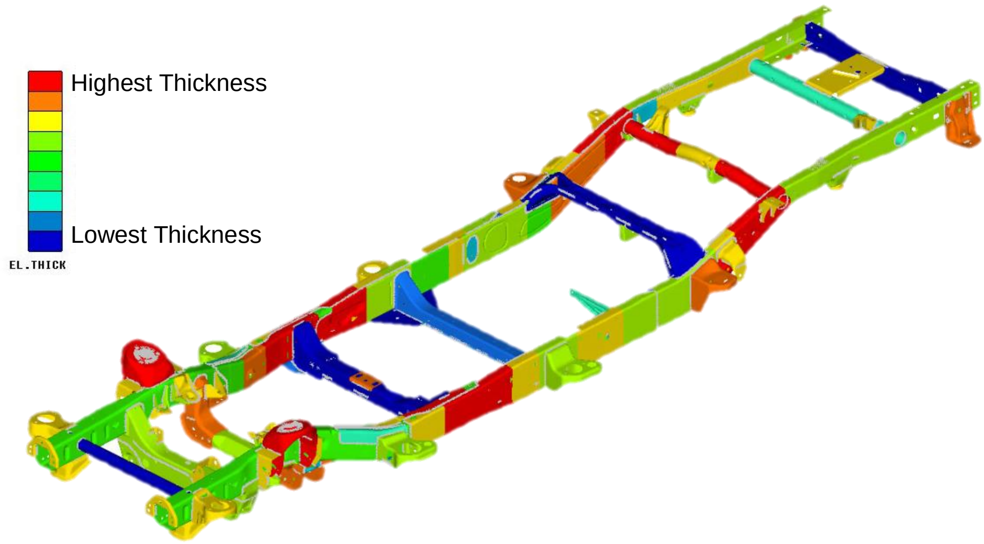

Figure 6-38: Semi-manual TWB: Thickness overview

Figure 6-38 shows the thicknesses of the manual designed TWB frame. Especially the location with thick material is interesting to see: Both longitudinal frames get thicker where the topological result has a different loadpath. The next location with more material is close to the seventh tubular crossmember. Because of them getting stronger, every other blank gets a little thinner than before. At the connection of the middle and rear frame, where an overlaying of sheet metal exists, material goes down even more. For ongoing studies it can be interesting to investigate, if a new design of the locations with thick material can decrease the weight of the structure even more.

6.5.2 Tailor-Rolled-Blanks

Tailor-Rolled-Blanks have the advantage over Tailor-Welded-Blanks, that there is no welding necessary to connect the different sheet metals. The transition is much smoother and therefore TRB have less stress peaks in these zones. The disadvantage of TRB is that they have much more rules to consider when it comes to manufacturability. Today's limitations to this process are the following:

- One area with constant thickness cannot be smaller than 50 mm
- The fastest thickness increase in a transition zone is $\frac{1\,mm}{100mm}$
- The minimal length for a transition zone is 20 mm
- The minimal thickness for tailor-rolled sheet metal is 0.8 mm
- The range for maximum and minimum thickness is also limited:
    - TRB for frame parts are showing that a range from 2 mm to 4 mm is possible [31]

Using the knowledge gained during the TWB optimization, the FE mesh file for the last iteration of the 6mm strip optimization is taken for designing the TRB frame. Unlike the optimization made before for the Tailor-Welded-Blank frame, this time the final FE mesh for the frame has to be exact, because only a normal analysis follows afterwards. The first step is to connect every adjacent PID which has the same thickness. Then the second step follows by designing PIDs for areas with slightly different thicknesses. Now, PIDs with the same values are merged together if they have only small strips between them. The third step is to design the transition zones between the areas already created in step one and two. This time the idea of an average thickness is reversed by creating the transition zone right between zones with constant thickness. Transition zones are also use, to eliminate spaces between two created zones which still have a big variety of different thicknesses.

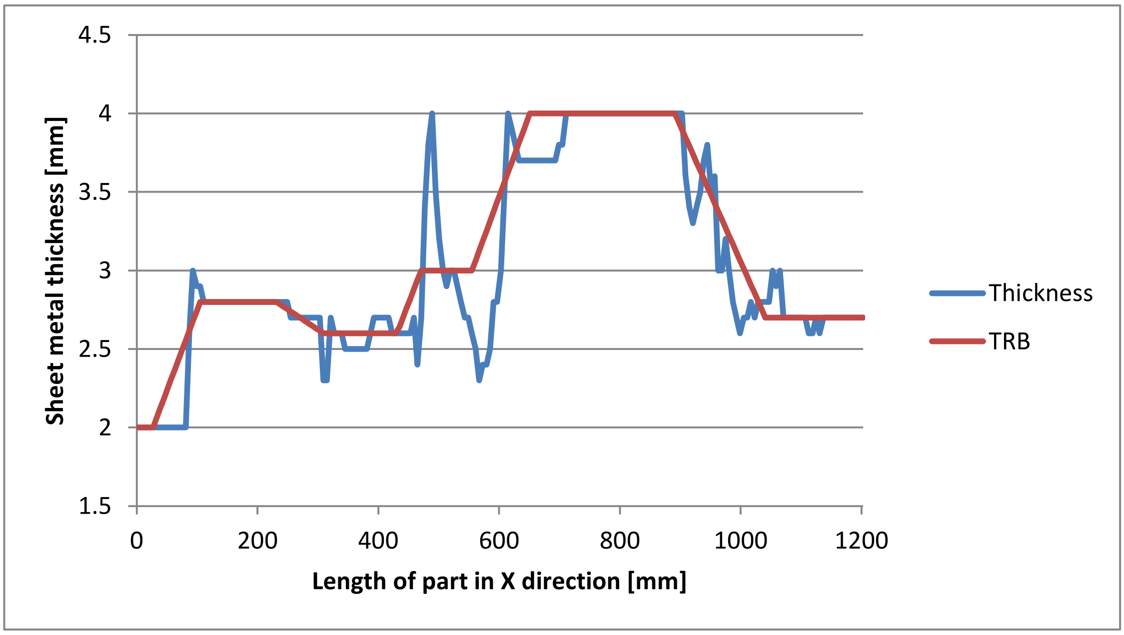

Figure 6-39: Thickness Distribution Before and after the manual FE design

Figure 6-39 shows the difference from the optimized thickness strips of on part from the rear longitudinal frame in comparison to the final TRB design. Every transition zone is created by taking three to four strips and assigning them to a mean value. This reduction is necessary to keep the processing time in a moderate range. Even with this compromise, the overall preprocessing takes about four hours of manual work. If many parts would have to be optimized, it can become handy to write an algorithm which will follow the explained steps and can automatically read the property IDs, bundle them, assign them to every element card which have the old PID before, and export them as a new include file. Excel VBA can be used for such a task. A similar task is executed using a VBA algorithm for sorting 100,000 element cards with several if statements involving multiple calculations. With a run time of about one hour it can become an interesting, more accurate alternative for an engineer who is busy for four hours.

After every part is equipped with the improved TRB design, a static and dynamic analysis followed to prove if the frame still has the correct behavior for every constraint. Running with Nastran, the analysis takes seven minutes. The result is with its 201.48 kg* 600 gram heavier than the TWB result, what partly results out of the manual design of the parts and also comes with the limitations for manufacturability.

		Topometry optimum	Semi-manual TRB	Baseline
Modal Analysis:	#7	25.69 Hz*	Isn't analyzed	24.35 Hz*
	#8	32.31 Hz*	Isn't analyzed	29.75 Hz*
Bending:	7	-1.715 mm*	-1.721 mm*	-1.766 mm*
	8	-1.718 mm*	-1.721 mm*	-1.766 mm*
Torsion:	1	-3.466 mm*	-3.491 mm*	-3.50 mm*
	2	3.465 mm*	3.491 mm*	3.50 mm*

Table 6-10: Semi-manual TRB: Results for Responses

As seen in Table 6-10, every constraint can be met with the manual design, following principles for the TRBs. This time, the bending loadcase gets a little bit more compliance in comparison to the optimal topometry design, but the torsional stiffness gets better than the reference.

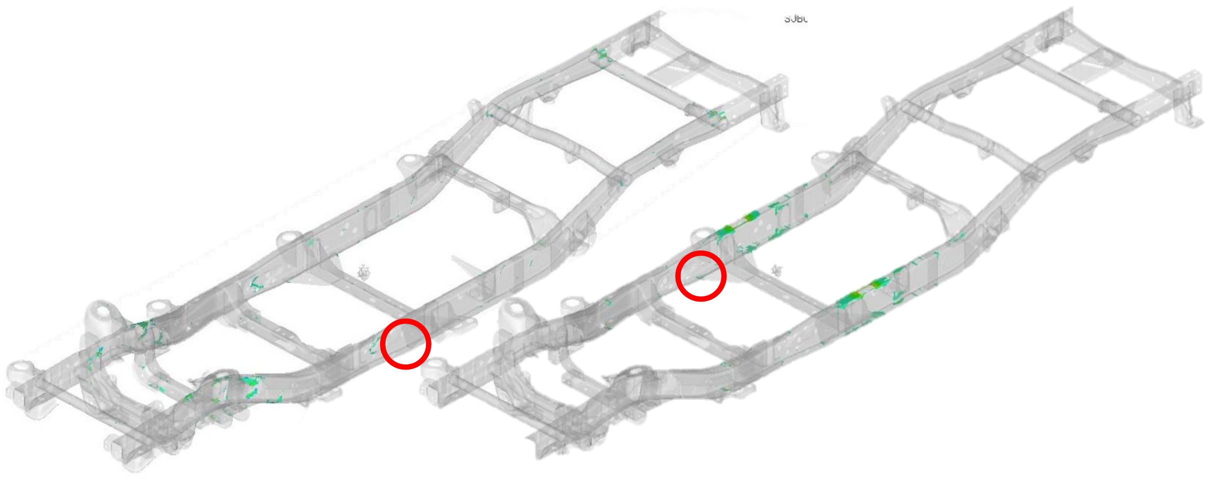

Figure 6-40: Semi-manual TRB: Von Mises stresses over 25 MPa

Figure 6-40 displays the stresses for both loadcases which are over 25 MPa. Like for the baseline model, stresses happen at the seamweld connection, between the rear longitudinal frame and the eighth crossmember for the torsion loadcase. Furthermore, stresses occur close to the third crossmember in the longitudinal frame. The maximum stress of 82.0 MPa* is located, at the connection between the middle crossmember and the longitudinal beam. For the bending loadcase most stresses can be found close to the location, where the load is applied on. The maximum stress of 75.17 MPa* is also at the middle connection but on the passenger's side. Both maxima are marked with a circle in red in Figure 6-40.

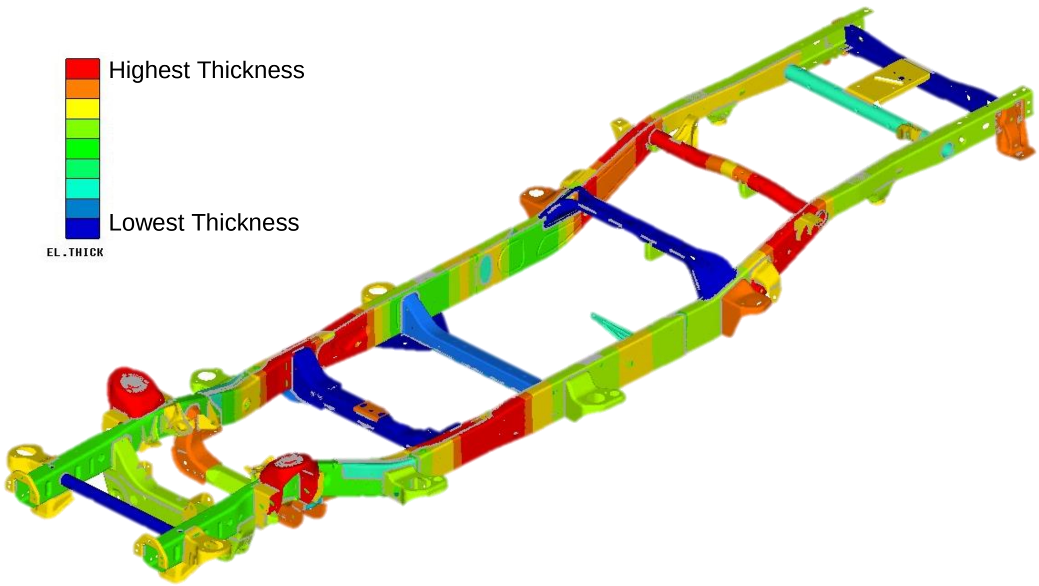

Figure 6-41: Semi-manual TRB: Thickness Overview

Figure 6-41 shows the final result for the TRB optimization. Like the Tailor-Welded-Blank optimization, especially the area between the middle and rear frame and the longitudinal frame section next to the crossmember that supports the transmission, gets really stiff for a better result of the torsion loadcase. Doing so, the middle part goes slightly lighter and makes it possible the get to the 3.59 kg weight saving in comparison to a normal gauge optimization.

6.5.3 Summary

Using topometry optimization expands the possibilities to save weight over the limit where the gauge optimization ends. Today's manufacturing possibilities make this optimizing method producible using TRBs or TWBs. The overall time spent on a topometry optimization with the better manual results takes more time than a normal sizing optimization: Besides the first analysis for the 6 mm strips that takes, the preprocessing included, about four hours, the manual modeling that follows varies from two to five hours, depending on how detailed the result should be, if several analysis are needed and how many parts are included. A following run for verification of the manual design is therefore quickly done in less than ten minutes. This modeling can only be made with linear blanks. When one wants to save even more weight, nonlinear blanks might become an option, but the whole process that start with interpreting results from a topometry optimization without the coarse parameter and goes on with the design of new areas of uniform thickness, that is a time wasting, slowly ongoing work, takes much more time and a quicker solution hasn't been found yet. The following work flow is recommended for a successful topometry optimization:

1. Choose the sections where topometry optimization should take place
    a. Regions with no stresses will lead to a minimum thickness. When on it using selected loadcases, only the areas where stresses are occurring should be considered for topometry optimization
    b. A topological analysis for loadpaths like it was made in chapter 6.2, can help to define regions for topometry optimization
2. Setup a topometry optimization, that will result in strips with the mean length of the used elements
3. Use a preprocessor to analyze the result and bundle strips to blanks of uniform thickness or TRB zones with uniform thickness or transition zones.
4. A normal analysis then will be used to verify every response is under the limit. Instead of an analysis a size optimization may become an option, if the results are not matching.
5. Check the result for its manufacturability using information provided by TRB or TWB corporations.

7 Evaluation

After getting several results for the different optimization method, this chapter is about the evaluation of the different results. First, the results overall will be rated for their usability. Then the different results will be compared to each other and at the end, a conclusion is given.

7.1 Discussion about the Quality of the Results

The first analysis that has been made with the baseline model, is used for a convergence analysis, to be sure, that the FE model matches the reality close enough. Therefore, one can assume that also the following optimization steps are close the actual structure, since elements and geometry hasn't been changed. More problematic is the environment in where this optimization is set. To make sure, that the optimized frame will stand NVH loadcases, a fatigue strength analysis has to proof that the frame can stand the loads for its lifetime. Because of missing data, such an analysis will not be dealt with in this thesis.

More relevant for the optimization of the frame is, that this optimization is only made for one specific loadcase but the frame has to bear much more loads than it is tested for. That leads to significant changes in the structure of the frame: During every optimization, the crossmembers from the middle section go down to the lowest thickness possible. The analysis shows that this works for the investigation of NVH loads, but if one would test the structure with more loads like forces, acting on the structure sideways, due to turnings, these crossmembers can become relevant again. During the work on the optimization, several attempts are made, where also the crossmembers are left out as a variable, but in the end, the decision is made to do the optimization with them, because of the thickness distribution between the crossmembers are an important change of the frame for this selected loadcase. Another effect can take place for the longitudinal beam in the middle section. Because the thicknesses changes at different locations, and the frame does not have a uniform thickness for these profiles anymore, it can happen, that during a frontal crash, the middle section can fail before the front section.

To make a final statement for the weight saving possibilities of the frame more loadcases have to be added. The missing loadcases will be: Full frontal crash, small overlap crash, side pole impact, side crash, fatigue strength analysis, loads occurring during operating conditions (turns, braking, acceleration, potholes and maneuvers), the weight of the modules, sitting on the frame (cabin, cargo box, engine and transmission) and their behavior during accelerations. This collection of loadcases has no reference and may have to be added with more missing loadcases.

7.2 Comparison and Rating of the Concepts

First of all, both aluminum versions cannot be compared with the other results because they are only approaches, and the exact weight remains unclear. Material optimization is still one of the most important methods that are already confirmed by the OEMs. Audi already started in the last century to build parts of their fleet complete out of aluminum and other manufacturer and studies are showing that the future will be multi material design. [32] [33]

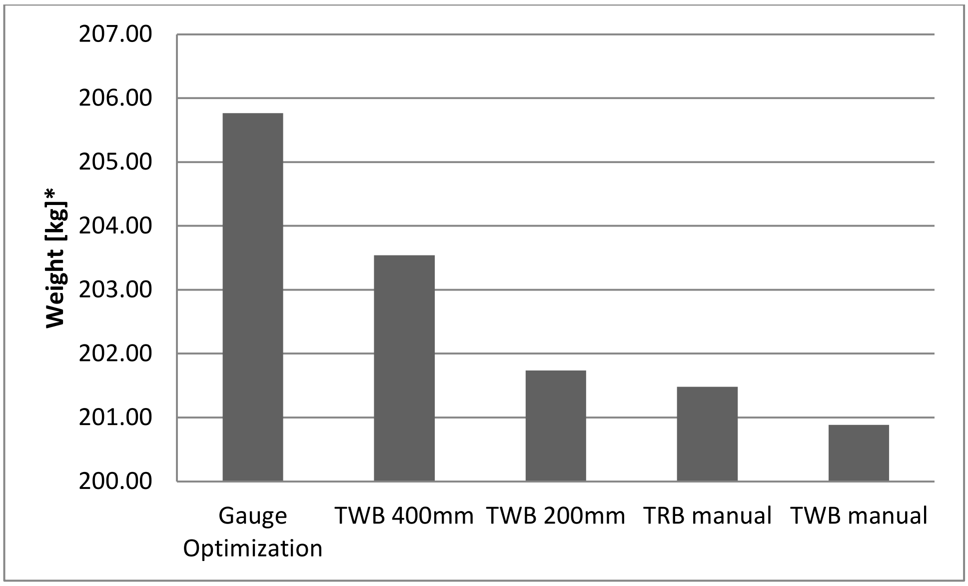

Figure 7-1: Total weight of the calculated results

Figure 7-1 shows the results for the made optimizations, sorted in descending direction. The heaviest result is represented by the gauge optimization that should be nowadays a standard tool for the perfect sizing of a structure. It is directly followed by the two automated attempts for Tailor-Welded-Blanks. Their only benefit is the quick setup of the analysis. In addition, the amount of blanks being used is not perfectly balanced and only set due to parameters. The best results can be achieved by manually modelling Tailor-Rolled-Blanks and Tailor-Welded-Blanks. Still the TRB result is not as light as the TWB result, because of the limits set for manufacturability. The final decision may be the price difference between both results. The only statement available is made by an employee of Mubea, that it is worth using TRB when at least 20% weight saving can be gained. In comparison to the result for gauge optimization, the saved percentage is relatively small and even the difference between the weight of the baseline model to the TRB result is not reaching 20 percent. Again, a clear statement is difficult to make, because not every part can be optimized, and not every loadcase is considered. What's left to say is that the use of TWBs can be affordable, because less scrap is produced during the manufacturing process. The used material is the most expensive cost factor for most of the part, so savings in the segment can justify increasing cost due to new machines and a more complicated manufacturing process. [7]

7.3 Conclusion

Looking at this study with a selected loadcase using the information available the usage of TRBs or TWBs is worth the effort. Even the fact that the manual modeling which takes a decent amount of time only saves 5.11 kg, compared to the gauge result have to be seen in correlation with the savings of the complete vehicle, where the weight of the frame makes only ten percent. Assuming the weight saving in the car is linear, the five kilogram saved for the frame makes about 50 kg additional weight saving for the whole auto by using same optimization methods.

8 Ongoing study for the frame in the NHTSA project

8.1 Redesigning a new module

When the Silverado project comes to stage two, it won't be sufficient to just optimize the existing frame. In order to give a proper forecast about the modern lightweight pickup truck, a complete new frame has to be constructed which fulfills all the loadcase requirements of the baseline model. This chapter should give a first impression how FE generated topology optimization can help in the development progress to find the optimal solution, where the structure should be constructed and where no material is needed. Especially the frame is an important module which has to stand against several important loadcases with high forces.

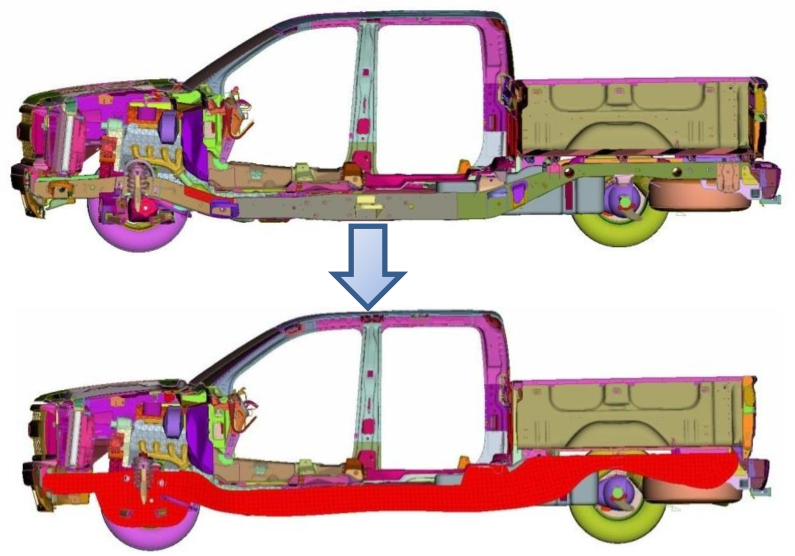

Figure 8-1: Analysis the structure to create an installation space model

The most important loadcases are:

1. Global bending
2. Global torsion
3. Modal analysis
4. Crashworthiness for frontal crash
5. Crashworthiness for rear crash
6. Local stiffness of the engine and gearbox joints
7. The cabins weight with all occupants
8. Towing

The first step to begin with, is to find the space where the frame can be constructed. Important parts like the engine, the transmission, the exhaust filter and the gas tank has to be subtracted from the design space. Smaller objects aren't removed from the structure, when it is also possible to move them to another place. Every connected part has to be considered while defining the possible space for the frame as well as possible movements of them. Also, the ground clearance and the angles from the tires to the front and the back of the auto have to be kept the same (Figure 8-1). The resulting volume is then meshed with 35 mm long tetra elements what makes a total of 145499 elements for this solid FE-model. The complete solid FE model has a weight of 8376 kg. Now the loadcases have to be defined for the model. The global torsion and bending stiffness loads will be used from the NVH run.

The crashworthiness of the frontal and rear crash are both calculated by using the mean acceleration during a crash with 35 MPH. An empirical value during a frontal crash is the acceleration of 25 g. To get to know the force which has to be used for the crash loadcases, the acceleration is translated into a force by multiplying it with the weight of the car.

$$F = m_{frac} \cdot a \tag{8.1}$$

$$F_{crash} = m_{frac} \cdot 25 \cdot 9.81 \frac{m}{s^2} \tag{8.2}$$

$$\frac{0.08 \cdot 8376kg \cdot 25 \cdot 9.81 \frac{m}{s^s}}{2} = 82168.56N \tag{8.3}$$

M_{frac} is the mass, the optimized topological model will have, after mass is distributed from it. Therefore, this input value is depending from the volume/ mass fraction constraint provided by the input data.

The forces reacting to the joints of the gearbox and the engine are calculated by using the weight of the gearbox and the engine with all the liquid and accept that the geometrical center of the three joints is the center of gravity of this assembly. This assumption has to be made, because only the outside surface of the engine is scanned and modeled without weighting and scanning every single part inside, so the actual center of gravity is unknown.

$$\frac{254.09kg \cdot 9.81 \frac{m}{s^2}}{2} = 1246.31N \tag{8.4}$$

$$123.3kg \cdot 9.81 \frac{m}{s^2} = 1209.57N \tag{8.5}$$

Equation (8.4) is applied where the engine's joints are located and equation (8.5) is used to simulate the load for the transmission using RBE2 elements.

For the cabin load, the weight of the complete cabin is measured, added by six passengers and 15 kg luggage and divided by six, because of the number of supports, mounted on the frame (Equation (8.6)).

$$\frac{542.143kg + 70kg \cdot 6 + 15kg}{6} \cdot 9.81 \frac{m}{s^2} = 1597.63N \tag{8.6}$$

In the case of towing, Chevrolet claims that the Silverado is able to tow up to 11,500lbs (5216.31kg). This also has to be done, when the truck is driving uphill, so for defining this loadcase, the following equation (8.7) is used to define, what force the frame has to stand while driving uphill while driving with a constant velocity. [34]

$$P_e = \frac{v}{1000\eta_t}(m_v g f_r \cos\alpha + \frac{1}{2}\rho_a C_D A_f v^2 + m_v g \sin\alpha)(kW) \tag{8.7}$$

		Pickup truck	Trailer
v	Velocity	$35\ MPH = 15.6\frac{m}{s}$	Same as truck
η_t	Transmission efficiency	0.85	Same as truck
m_v	Vehicle Curb Mass	$2315kg + 2 \cdot 70kg + 15kg$	$11500lbs = 5216.3kg$
g	Gravity constant	$9.81\frac{m}{s^2}$	$9.81\frac{m}{s^2}$
f_r	Tire rolling resistance	0.01	Same as truck
α	Grade (Road slope)	6°	Same as truck
ρ_a	Density of air	$1.2\frac{kg}{m^3}$	Same as truck
C_D	Drag coefficient	0.41	0.5
A_f	Vehicle frontal area	$2m \cdot 1.9m = 3.04m^2$	$1.2 \cdot 3.04m^2 = 3.648m^2$

Table 8-1: Explanation of used variables in Equation (8.7) and the used values

Please note that a lot of these values in Table 8-1 are empirical assumptions to get a reasonable dimension for this loadcase. The written equation is only needed for, when one is looking for the necessary engine power. In this case only the force is required to pull the trailer up the slope. This simplifies it to the following equation (8.8).

$$Trailer\ Force = (M_v g f_r \cos\alpha + \frac{1}{2}\rho_a C_D A_f V^2 + M_v g \sin\alpha)(N) \tag{8.8}$$

$$\begin{aligned} Trailer\ Force = (5216.3\ kg \cdot 9.81\frac{m}{s^s} \cdot 0.01 \cdot 0.9945 + \frac{1}{2} \cdot 1.2\frac{kg}{m^3} \cdot 0.5 \cdot 3.648m^2 \\ \cdot 243.36\frac{m^2}{s^2} + 5216.3kg \cdot 9.81\frac{m}{s^s} \cdot 0.1045) \end{aligned} \tag{8.9}$$

$$Trailer\ Force = (508.918N + 267.920N + 5348.940N) = 6127.176N \tag{8.10}$$

Setting up the FE model for topology optimization

After calculating the needed forces for the optimization run, they are defined in the model. Altair OptiStruct v12 is used for this optimization run. The SPC Sets listed in Table 8-2 are created then. Like for the previous optimizations for the NVH loadcases, global torsion and global are used with the same loads as in the real test (Torsion: 1200 N on both shock towers; Bending: 2224 N on each side between front and rear axle). The calculated load for the front and rear impact is applied, using RBE2 elements at the locations where the longitudinal beams would initiate the force to the frame. The SPC set for inertia relief is taken to give the frame the opportunity to act with acceleration against the force. For the loadcase where the weight of the cabin and the engine act on the structure, RBE 2 elements are used at the location where the supports for the engine and transmission used to be. For the constraints, the created SPC set for the global bending loadcase is used, because of the similarities of both loadcases. Similar to the previous setup, for the cabins weight, RBE elements are created where the connection between frame and cabin used to be, and the calculated force divided by six as seen in Equation (8.6) is applied on them. The last left loadcase is the towing simulation. Therefore the exact same RBE2 elements as for the rear impact are used, to apply the calculated load from

Equation (8.10) and the inertia SPC set is used, to simulate the moving car working against the pulling trailer. Figure 8-2 shows the finished FE model with all the explained loadcases and constraints.

SPC Set's Name	Location	Constraint DOFs	Element type
Global torsion	Between shock towers	Z	SPC
	Rear axle co-driver's side	XZ	SPC
	Rear axle driver's side	XYZ	SPC
Bending	Driver's shock tower	YZ	SPC
	Driver's rear axle	XYZ	SPC
	Co-driver's shock tower	Z	SPC
	Co-driver's rear axle	XZ	SPC
Inertia relief	Driver's shock tower	Z	SPC
	Driver's rear axle	Z	SPC
	Co-driver's shock tower	YZ	SPC
	Co-driver's rear axle	YZ	SPC
	Driver's rear axle	X	SUPORT1

Table 8-2: Used SPC sets for Topology Optimization

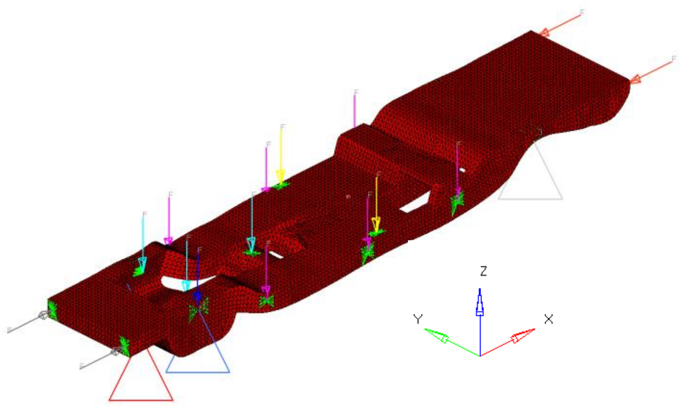

Figure 8-2: Overview of every Load and SPC applied on the model

For both crash and the towing loadcase, an inertia relief elements is used which is called in particular subcases with the INREL=-1 parameter in order to initiate a loadcase which will be done in a movement. For the calculation the constraint X direction will be replaced by acceleration equal to the applied load to keep the system in equilibrium. This is the best way to simulate a moving object, like a vehicle or a plane in a static analysis. The generated model is analyzed in a test run and then the optimization parameters are defined.

The get the final feasible solution more than ten runs are necessary where values like volume fraction, minimum member size and symmetry has to be changed. If a result is good, is rated by its overall appearance, by the definiteness of the loadpaths, by the behavior close to the non-design area and by the symmetry. The optimization starts with only a few loadcases and more are added as soon as the settings of the input data are defined. The shape of the resulting structure changes massively by adding more loads, so that one can say that the final structure can only be as perfect, as the given loadcases are.

The optimization setup is created by defining the designed solid mesh as the variable for the optimization. A parameter is set to define symmetric behavior for the XZ-plane and the minimum member size is set to 50 mm. Next, the responses are defined: Besides the volume fraction only the weighted compliance is needed for the complete optimization. The weighting factors set for the compliance response is full 100 percent for the loadcases global torsion, global bending, engine and transmission loadcase and the cabin's weight loadcase. The left loadcases used with the inertia relief parameter were weighted with 33 percent to bring the three loadcases front and rear impact and towing together to 100 percent. To get the right value to constrain volume fraction, several runs are needed to find five percent fraction getting the clearest results. The last thing to do is to assign the set weighted compliance response as the objective. 42 Iterations are needed and it takes 15 minutes to run the optimization.

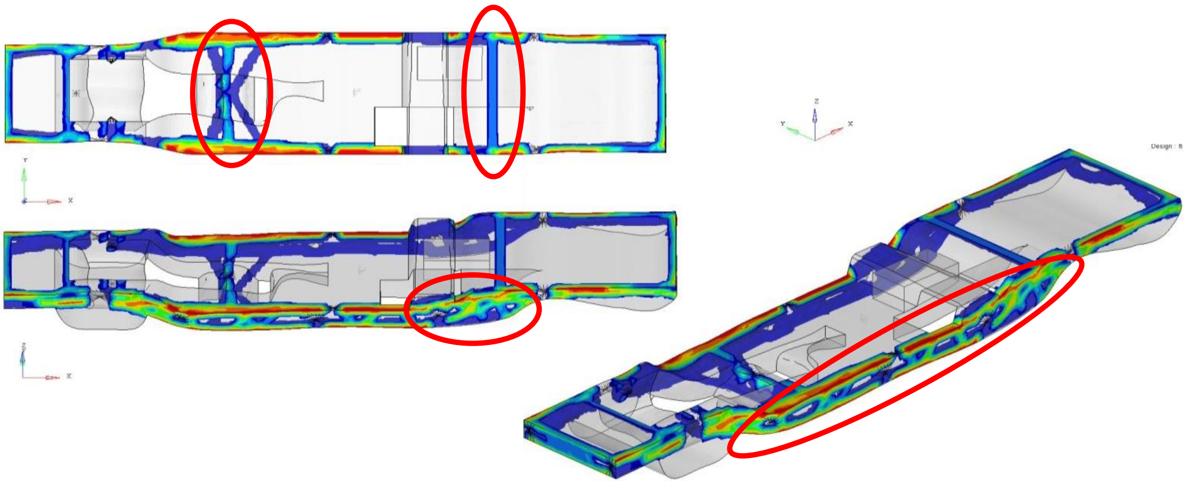

Figure 8-3: Topological result for the last iteration

The resulting shape can be seen in Figure 8-3. At a first glance this output looks almost like the initial baseline structure. But it has also changes that can be interesting for creating a new, stiffer frame. There is, for example, a new x-shape structure under the transmission for supporting the transmission's weight and probably for stiffen the frame against torsion. Another interesting development is the left out mass between the longitudinal beams that can be modeled in the new design by using thinner sheet metal parts in this area, or leaving out material by making holes. The next important region is located on the longitudinal beam, close to the fuel tank. There, the volume tries to use as much space as possible, probably to lower the compliance against bending by increasing the local moment of inertia. Like in the topology analysis for NVH in chapter 6.2, one cross member behind the fuel tank still exists, and is still significant for the overall torsional stiffness of the frame. Instead, almost every other cross member vanishes, so that one can say that the one located above the fuel tank and the filter has only the job to hold their weight. The other members can appear again, when one is using more loadcases. More possible loadcases can be side crash, lateral forces due to sharp turns, the weight of the cargo box or the weight of filter and fuel tank.

The reason why these loadcases are not included is, because of missing information and low influence/ minor changes on the structure. This approach can also be expanded by adding more modules like a cabin, cargo box or fenders to perform an overall topology optimization were every module can interact with each other. The shown topology problem should only be used as a study to indicate the possibilities of topology optimization and to give a first hint, what can be improved in comparison to the baseline frame.

In general, when this step is done, the designer's work begins by using the given loadpaths do invent a new structure. They have to decide, what profiles, connection types, materials and manufacturing methods should the used, to get as close to the produced paths as possible.

8.2 Example Problem – Mount Support

In order to save weight every part of the current frame will be modified totally. With the mount support taken as an example, this chapter shows a simple and efficient way to reconstruct a selected part. Figure 8-4 shows the baseline model and the part that will result out of the topological optimization.

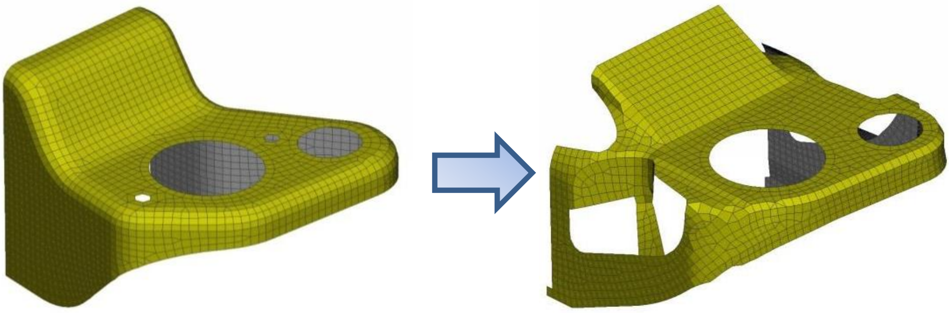

Figure 8-4: Changes being made from baseline model to the optimized result

When a module is changed, every connection to other modules stays the same. Therefore, the supports for every mounting will have to stay at the same place as before. To set up the optimization run, it is important that the new construction has the same or better attributes and behavior as the last one. The decision, if the new part will have the same manufacturing method and the same material as the current one, is also an important step and can have an influence on the whole optimization set up. In this case, material and manufacturing method should stay with steel and stamping.

Three loadcases are assigned to analyze what constraints are needed:

1. The cabin's weight plus six passengers and luggage divided by six

$$\frac{542.143kg + 70kg \cdot 6 + 15kg}{6} \cdot 9.81\frac{m}{s^2} = 1597.63N \quad (8.11)$$

2. The cabin's weight plus six passengers divided by six while braking

3. The cabin's weight plus six passengers divided by six doing turns to each side

The loadcases are defined in a simple way by applying a load to a rigid body element, covering the space, where the mount would sit, while having single point constraints to simulate the seam welds.

8.2.1 Baseline Analysis

The baseline design has a thickness of 2.5 mm* and a total weight of 1.43 kg*. An analysis of the baseline model is necessary to define the constraints in which the optimization algorithm can act. The loadcases listed in Table 8-3 are used for the analysis. Figure 8-5 shows the created FE model with the constraints and loads applied for every loadcase.

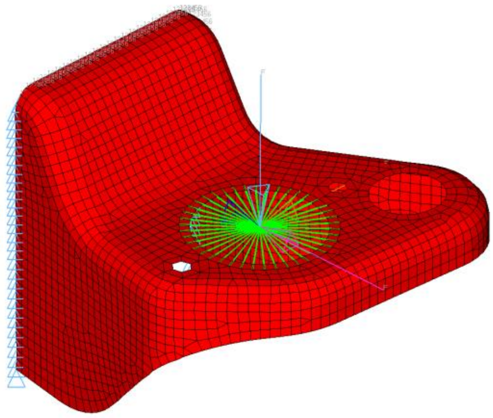

Figure 8-5: Baseline FE model

No.	Loadcase	Force	Direction	Displacement RBE master node
1	Weight	-1597.63 N	Global Z	-0.253 mm Z-direction
2	Braking	-1597.63N	Global X	-0.01201 mm X-direction
3	Turning 1	1597.63N	Global Y	0.006268 mm Y-direction
4	Turning 2	-1597.63N	Global Y	-0.006268 mm Y-direction

Table 8-3: Loadcases considered for the analysis

As seen in table, the displacements for the loadcases 2-4 are relatively small. The main focus of the optimization will be, to optimize the structure for load applied in global Z direction. Especially for this load case, one can see a large displacement for the small flange area on the right side. Figure 8-6 shows this bending under the load and the von Mises stress resulting out of it. Besides the deformation, the colors are indicating a stress concentration for this location. This leads to the conclusion, that at least there the part is not perfectly designed.

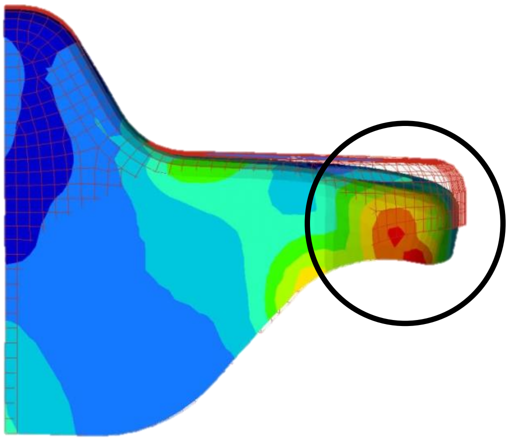

Figure 8-6: Displacement in Z direction for the first loadcase

8.2.2 Optimization Procedure

The following steps are used to optimize the existing structure:

1. Define the space were the new mount support can be constructed
2. Two-dimensional topology optimization with the main loadcase, to understand the loadpaths in a simple way
3. Three-dimensional topology optimization with all loadcases
4. Model the optimized structure in a producible way
5. Using gauge optimization to find the optimal thickness for the mount support
6. Optional: Using topography optimization to add beads in critical areas

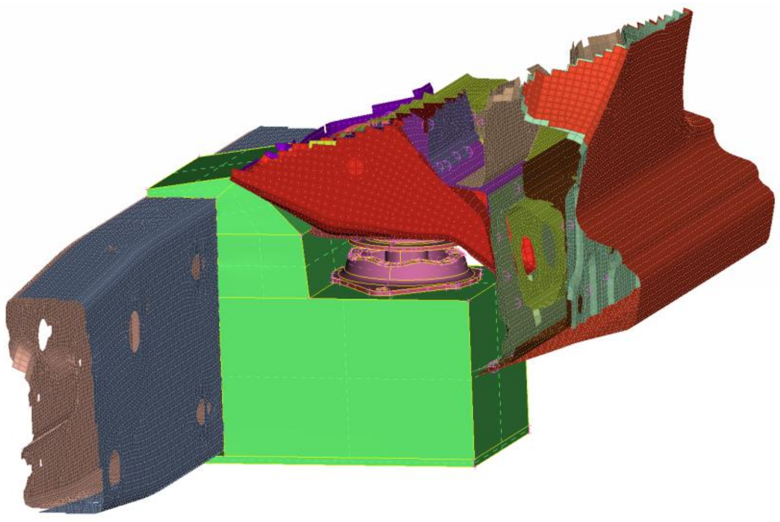

Figure 8-7: Created geometry to define the installation space

8.2.2.1 Space definition

At first the space that can be used to construct a new support has to be defined. Important things to remember are: Connections, close parts, moving parts. In this case the design space, shown in Figure 8-7 in green, will be welded to the longitudinal beam. Therefore the space is set directly onto the beam without any gaps. The cabin, which is located above the support, has to be rated with a certain displacement because of the up and down movement while driving over bumps. The next boundary is set, due to the rocker on the right side. It can also move slightly towards the beam, what has to be kept in mind. At last the design space is defined by the mount, sitting on top of the support. It needs one hole with a diameter of 80 mm to fit in and two holes with a diameter of 12 mm for bolts next to it.

8.2.2.2 Two-Dimensional Topology Optimization

Because the created design space is an extrusion of one face over the length, a quick two dimensional topology study can be used before the real investigation, to understand the given task in a simple way.

The FE model is created by using shell elements which are locked in their normal direction. For the seam weld, SPCs with locked DOFs in all six moving directions are defined for the grids that are adjacent to the beam. The load for the first loadcase is applied, showing in negative Z direction.

The program used to find the needed loadpaths is Altair OptiStruct v12.0. The set up for the optimization run is the following: The created shell surface is chosen to be the design variable, used for the optimization. The minimum member size is set to 12 mm and the minimum thickness is defined with 0 mm to allow the algorithm to design clear loadpaths. The objective is set for minimum compliance and volume fraction is constrained to 20 percent.

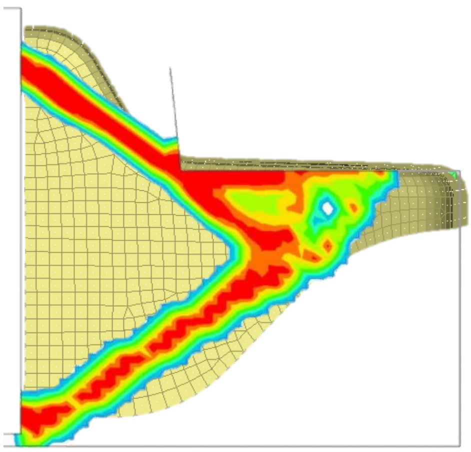

Figure 8-8: Merged side view of baseline model and topology result

The analysis needs 39 iterations to find a feasible design and takes 31 seconds. This first pre run takes a short time to create, and can give a first impression of what can be made to optimize an existing structure. The loadpath in Figure 8-8 shows what parts of the structure are really needed to stand to weight of the cabin with maximum stiffness. A lot of material is applied directly under the load, to avoid bending and to redirect the stresses to the bottom connections.

8.2.2.3 Three-Dimensional Topology Optimization (one LC)

A lot of different attempts are made, to find the optimal topology solution. Not only parameters are changed, the most important part is to define the design space in a proper way. Other parameters which have to be changed to get a good result are: volume fraction, minimum member size, symmetry, stamping option, drawing option, element length size of the mesh and an added non-design area.

A first, simple solution is created by using only the first loadcase and only the area needed for the mount (Figure 8-9). After modeling the shape with ANSA, an OptiStruct Gauge optimization run followed, to find the optimal metal sheet thickness for the design. This first approach has the same displacements as the baseline model, slightly higher stresses, a smaller thickness, and is 1.038 kg lighter than the baseline model (Table 8-4).

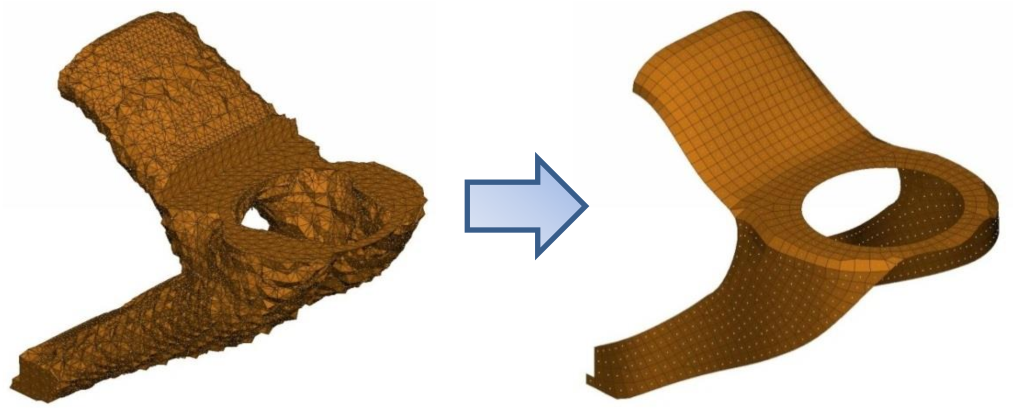

Figure 8-9: Modeling the topology result to a FE model

	Baseline	First attempt
Displacement at center point	-0.2534mm*	-0.2531mm*
Max. von Mises stress	113 MPa*	140 MPa*
Overall weight	1.4261kg*	0.391kg*
Thickness	2.5mm*	1.5mm*

Table 8-4: Comparison between baseline model and the first approach

Please note that the maximum stress for the first optimized model occurs on one side member, what can be fixed by shape optimization and topography optimization to stiffen the side members. This first solution is only used to give a first impression where the optimization is going, and if it is worth it, to continue the analysis.

8.2.2.4 Three-Dimensional Topology Optimization (three LC)

It is necessary to change the design area several times before one can see clear results, where the topology optimization is heading to. At first a model is used, that isn't modified at all. That leads to the problem that mass is trying to go under the hole to support the load directly with the shortest load path possible. For the next runs, the area for the hole is cut out of the volume, to prevent mass going under it. But even this cut out isn't enough, because there is still more hole were no mass should go under it, so the final solution ends with defining an area on top of the mount support, where all the four holes are in it, cut this section out, down to the bottom and leave one row of elements at the top. This one row area at the top is defined as an own property so it will not be used when performing the topology optimization.

For the design variable, the PID is chosen which surrounds the plate, the load is applied on. The parameters have to be changed several times to get producible results. Unfortunately the stamping parameter is not useful finding a feasible solution, because it has trouble creating faces that are orthogonal to each other. Looking up several optimizations made by using this parameters leads to the conclusion, that is can only be used, when the depth for stamping changes slightly. [35] The best result is achieved, using a minimum member size of 12 mm, a set drawing direction along the global Z axis with "split drawing" to allow holes for the drawing result and one symmetry constraint from the center of the model with its YZ plane (see figure 8-5). Next, two responses are created: total volume fraction is used as a constraint to define the clearness of the results. Several runs brought five percent as the best lower constraint value. The second response is weighted compliance, in where the first loadcase is weighted normal, and the other three loadcases are weighted with 1/3. The objective is defined to minimize the compliance.

The final analysis run needs ten minutes for execution and uses 40 iterations, to find the optimal design. This solution seen in Figure 8-10 is chosen, because it is producible in a stamping progress and can be applied for sheet metal.

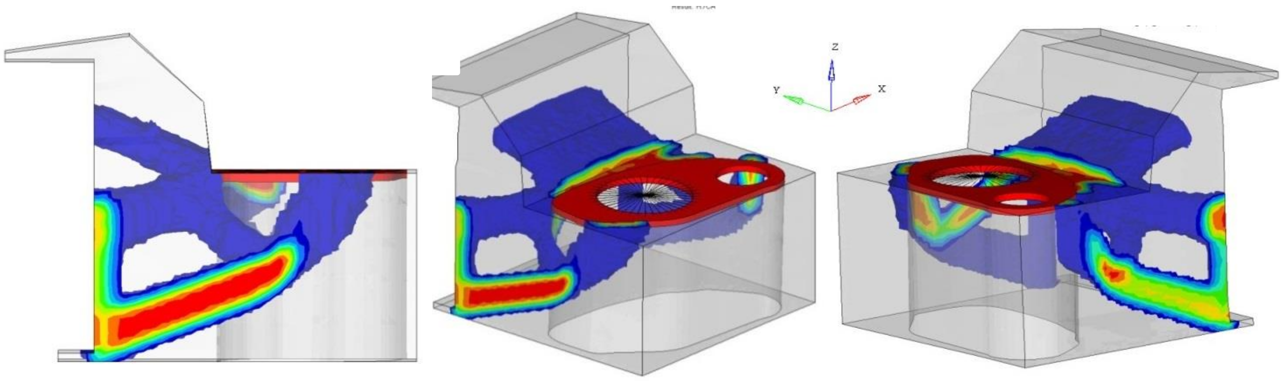

Figure 8-10: Topological result for the last version

The resulting shape is used to design a surface with Siemens NX that can be stamped. The resulting part can be seen in Figure 8-4. After generating a surface, a gauge optimization with all the defined loadcases and the constraints, taken out of the baseline model, is used to get the right thickness. The review of the analysis shows that the upper plate is bending under the applied load, what isn't considered in the topology optimization because of stiffer solid elements. To prevent this effect, a simple flange is modeled at the front to support the structure. The perfect shape of this flange is found by using OptiStruct Free Shape optimization to move the shell's grid nodes along the surface direction.

	Baseline	Optimized structure
Displacement at center point for LC 1	-0.2534 mm*	-0.1929 mm*
Displacement at center point for LC 2	-0.0120 mm	-0.0017 mm*
Displacement at center point for LC 3	-0.0063 mm*	-0.00617 mm*
Displacement at center point for LC 4	0.0063 mm*	0.00617 mm*
Maximum von Mises stress LC1	113MPa*	84.46MPa*
Maximum von Mises stress LC2	16.25 MPa*	19.20 MPa*
Maximum von Mises stress LC3	18.23 MPa*	18.23 MPa*
Weight	1.4261kg*	1.34kg*
Thickness	2.5 mm*	3 mm*

Table 8-5: Comparison between baseline and the final result

As one can see in Table 8-5, the new designed support meets the set requirements perfectly. The weight saving of 80 gram is not that much, but the new design also has better maximum stresses than the baseline model. Because the stresses are not constrained for the optimization, for the last three loadcases, the stresses are slightly off. Comparing the overall stress distribution for the loadcases, stress concentrations still exists, but they are reduced to a small amount of elements (Figure 8-11). Reviewing the results the question raises, if it is necessary to optimize the supports for those bending loadcases, because the displacements and stresses are so low for them, even when they are forced to hold the complete structure in this direction. Those small displacements are mainly the reasons the thickness goes up during the analysis. The optimized design shows not the perfect behavior for the loadcases two, three and four, probably because the topology optimization is being weighted with one third for each of them. Another optimization run showed the optimized behavior for the loads in Z direction. With only this load acting on the structure, the design can achieve a weight saving of 416 gram and has the same maximum stress of 113 MPa*. Only with assumptions being made, it cannot be excluded that instead of 80 gram, more weigh can be saved.

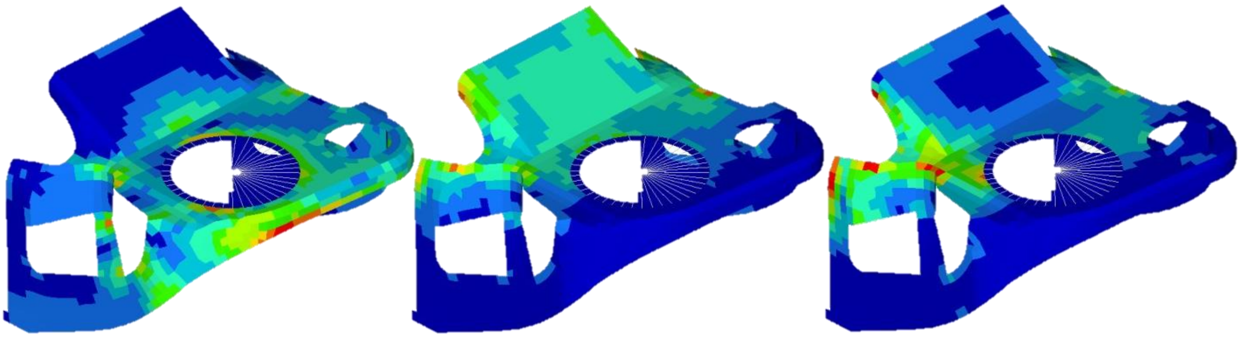

Figure 8-11: Von Mises stresses for: LC1, LC 2, and LC 3

9 Summary

The objective of this paper is the optimization of the Chevrolet Silverado 1500 frame for the NVH loadcases, using FEA methods. The overall optimization process is explained so one can project the gained knowledge to other structures with different loadcases. The prescribed content covers the complete process that starts with the baseline analysis, which at first has to be verified with the real test results and continues with the setup for each optimization. Four optimization methods are chosen for this task: Topology optimization, gauge optimization, material optimization and topometry optimization. At the end the results will be applied to the model.

The first step, the designing of the FE model takes a long time and has to be done carefully, considering a lot of different rules. This is necessary to align the simulation with the actual test results made with the real frame. The calculated results for the approved model can then be taken to start the optimization with. The topology optimization is used to get knowledge about the preferred loadpaths the forces would take for maximum stiffness. During the next steps this pre-run is necessary several times to understand the changes the algorithm decided to make on the structure. Then the gauge optimization followed where a database is created to assist the use of the advanced parameters of MSC Nastran properly and without mistakes. Because of the selected loadcases taken for this optimization, the optimization leads to significantly changes, mostly for the crossmembers. Then attempts are made, to optimize the frame using different materials. A first discussion is made to compare the materials for certain properties and a simple calculation shows the benefit of aluminum due to its lower density. The results for a frame made out of aluminum has the lowest weight but is only an estimated results, because a complete frame made out of aluminum needs to be designed suitable for the different behavior of this material and needs a following analysis for fatigue strength, that can increase the weight again up to an unknown value. The last optimization method used is topometry optimization, which is used to create producible results for Tailor-Welded-Blanks and Tailor-Rolled-Blanks in the middle section of the frame. Several ways to create them are executed and compared considering not only the total weight, but also the suitability for the production.

To optimize an existing structure, using a complete specification list, the described approach can be used to successfully save weight of a structure. Furthermore, it provides a basic understanding how to use FEA based optimizing methods, to successfully improve a structure. During the investigation it is at several steps obvious that the bare use of FEA methods for a successful weight saving is not the best way. Using the given information out of a specification list, and the exact knowledge of the existing structure should rather be used to completely redesign the structure, in where designers use given results out of a topology optimization to create a complete new model. Only the combination of FE based optimization with the intelligent use of lightweight materials and todays design possibilities can lead to a perfectly optimized structure.

10 References

[1] Wikimedia Foundation, Inc., September 2014. [Online]. Available: http://en.wikipedia.org/wiki/Corporate_Average_Fuel_Economy.

[2] U.S. Department of Energy, "Fuel Economy," 08 October 2014. [Online]. Available: http://fueleconomy.gov/.

[3] T. Cain, September 2012. [Online]. Available: http://www.goodcarbadcar.net/2012/09/august-2012-usa-minivan-truck-sales-figures.html.

[4] H.-H. Braess, Vieweg Handbuch Kraftfahrzeugtechnik, Wiesbaden: Springer Vieweg, 2013.

[5] E. Hoepke, Nutzfahrzeugtechnik, 7. ed., S. Breuer, Ed., Wiesbaden: Springer Vieweg, 2013.

[6] B. Klein, Leichtbau-Konstruktion, Wiesbaden: Springer-Vieweg, 2013.

[7] J. Marsolek, "Strukturkonstruktion," HAW Hamburg, Hamburg, 2013.

[8] M. Linke, "Festigkeit im Leichtbau," HAW Hamburg, Hamburg, 2012.

[9] Klein, Leichtbau-Konstruktion, Vieweg Verlag, 2014.

[10] Bargel and Schulze, Werkstoffkunde, Heidelberg: Springer Vieweg, 2012.

[11] S. Keeler, Advanced High-Strength Steels Application Guidelines Version 5.0, World Auto Steel, 2014.

[12] H. E. Friedrich, Leichtbau in der Fahrzeugtechnik, Wiesbaden: Springer Vieweg, 2013.

[13] A. Kröff, "Neue Wege im Karosseriebau," *VDI Konstrktion 9,* 2007.

[14] Vanderplaats Research & DEvelopment, Inc., "Basic Concepts of Optimization," 2014. [Online]. Available: https://www.vrand.com/education.html. [Accessed 5 October 2014].

[15] A. Schuhmacher, Optimierung mechanischer Strukturen, Wiesbaden: Springer Vieweg, 2013.

[16] MSC Software Corp., "MSC Software," 06 November 2013. [Online]. Available: https://simcompanion.mscsoftware.com/infocenter/index?page=content&id=DOC10355&cat=MSC_NASTRAN_DOCUMENTATION_2013&actp=LIST. [Accessed 30 September 2014].

[17] K. Youssefi, "San José State University," [Online]. Available: http://www.engr.sjsu.edu/youssefi/me265/notes/14%20-%20Optimization.pdf. [Accessed 1 October 2014].

[18] red cedar technology, "HEEDS | mdo," [Online]. Available: http://www.redcedartech.com/products/heeds_mdo. [Accessed 7 October 2014].

[19] E. Johnson, W. Wu and X. Yu, "MSC Software," 22 October 2004. [Online]. Available: http://web.mscsoftware.com/events/vpd2004/na/proceedings/pdfs/2004-100-TW_topology.pdf. [Accessed 6 October 2014].

[20] MSC Software Corp., "MSC Software," 06 November 2013. [Online]. Available: https://simcompanion.mscsoftware.com/infocenter/index?page=content&id=DOC10351&actp=LIST_POPULAR. [Accessed 30 September 2014].

[21] Altair, "Altair | Hyperworks," 2014. [Online]. Available: http://www.altairhyperworks.com/Product,19,OptiStruct.aspx. [Accessed 30 September 2014].

[22] P. Zeller, Handbuch Fahrzeugakustik, Wiesbaden: Springer Vieweg, 2012.

[23] W. Dehmel, FEM-Berechnung mit MSC/NASTRAN, Hamburg, 2013.

[24] I. Janzen, "Learning to DO," [Online]. Available: http://www.isaiahjanzen.com/2012/08/the-jacobian-finite-element-mesh-quality.html. [Accessed 26 09 2014].

[25] K. Leke, "MSC Nastran Novice," February 2012. [Online]. Available: http://mscnastrannovice.blogspot.de/2012/02/output-files-from-msc-nastran-part-ii.html.

[26] A. E. Stockwell, "A Verification Procedure for MSC/NASTRAN Finite Element Models," National Aeronautics and Space Administration, Hampton, Virginia, 1995.

[27] Vijesh, "MSC Nastran Useful Resources," 25 November 2008. [Online]. Available: http://nastranusefulresources.blogspot.de/2008/11/epsilon-and-external-work-in-f06-file.html.

[28] Wikimedia Foundation, Inc., "WikiBooks," 2009. [Online]. Available: http://en.wikibooks.org/wiki/Advanced_Structural_Analysis/Part_I_-_Theory/Failure_Modes/Fatigue/Crack_Initiation/The_W%C3%B6hler_Curve. [Accessed 1 Oktober 2014].

[29] Vanderplaats Research & Development, Inc., GENESIS Design Manual, Colorado Springs, CO, USA, 2013.

[30] K. Buchholz, "Automotive Engineering," 5 August 2014. [Online]. Available: http://www.twbcompany.com/Downloads/AE-Aug-2014.pdf. [Accessed 30 September 2014].

[31] Muhr & Bender KG, "Mubea," 2014. [Online]. Available: http://www.mubea.com/de/produkte-technologien/industrie/produkte/tailor-rolled-products/lieferprogramm/abmessungen-und-lieferformen/. [Accessed 7 October 2014].

[32] B. Ingenohl, "Fertigungsautomatisierung und Montage," [Online]. Available: https://wiki.zimt.uni-siegen.de/fertigungsautomatisierung/index.php/Herstellung_einer_Vollaluminium-Karosserie_am_Beispiel_des_Audi_Space-Frames. [Accessed 02 Oktober 2014].

[33] M. Lesemann and M. Broeckerhoff, "Institut für Kraftfahrzeuge," March 2008. [Online]. Available: http://www.ika.rwth-aachen.de/forschung/veroeffentlichung/2008/03-01/lwd_03-

2008.pdf. [Accessed 02 October 2014].

[34] Chevrolet, "2014 Silverado," 7 October 2014. [Online]. Available: http://www.chevrolet.com/silverado-pickup-trucks.html.

[35] M. Zhou and S. P. N. S. D. M. S. G. Raphael Fleury, 13 June 2011. [Online]. Available: ftp://ftp.altair.de/pub/edu/protected/free_papers/WCSMO9_topology.pdf. [Accessed 03 October 2014].

[36] Wikimedia Foundation, Inc., "Wikipedia," [Online]. Available: http://en.wikipedia.org/wiki/National_Highway_Traffic_Safety_Administration. [Accessed 24 09 2014].

[37] T. Cain, "Good Car Bad Car," [Online]. Available: http://www.goodcarbadcar.net/2012/09/august-2012-usa-minivan-truck-sales-figures.html. [Accessed 24 09 2014].

[38] U.S. Environmental Protection Agency, "Fuel Economy," U.S. Department of Energy, [Online]. Available: http://www.fueleconomy.gov/feg/bymake/Chevrolet2013.shtml. [Accessed 24 09 2014].

[39] Chevrolet, "2015 Silverado 1500," [Online]. Available: http://www.chevrolet.com/silverado-1500-pickup-truck/specs/trims.html. [Accessed 24 09 2014].

[40] Vanderplaats Research & Development, Inc., "VR&D Genesis," [Online]. Available: http://www.vrand.com/genesis.html. [Accessed 24 09 2014].

[41] R. Fortier, "ROD & Custom," 13 February 2013. [Online]. Available: http://www.rodandcustommagazine.com/techarticles/1301rc_1933_ford_tudor_sedan_project/.

[42] J. Vondruska, "VWVortex," 24 August 2012. [Online]. Available: http://www.vwvortex.com/features/technical-features/golf-7-technicalpreview/attachment/tailor-rolled-blank/.

Appendix A: Von Mises Stresses of the Results

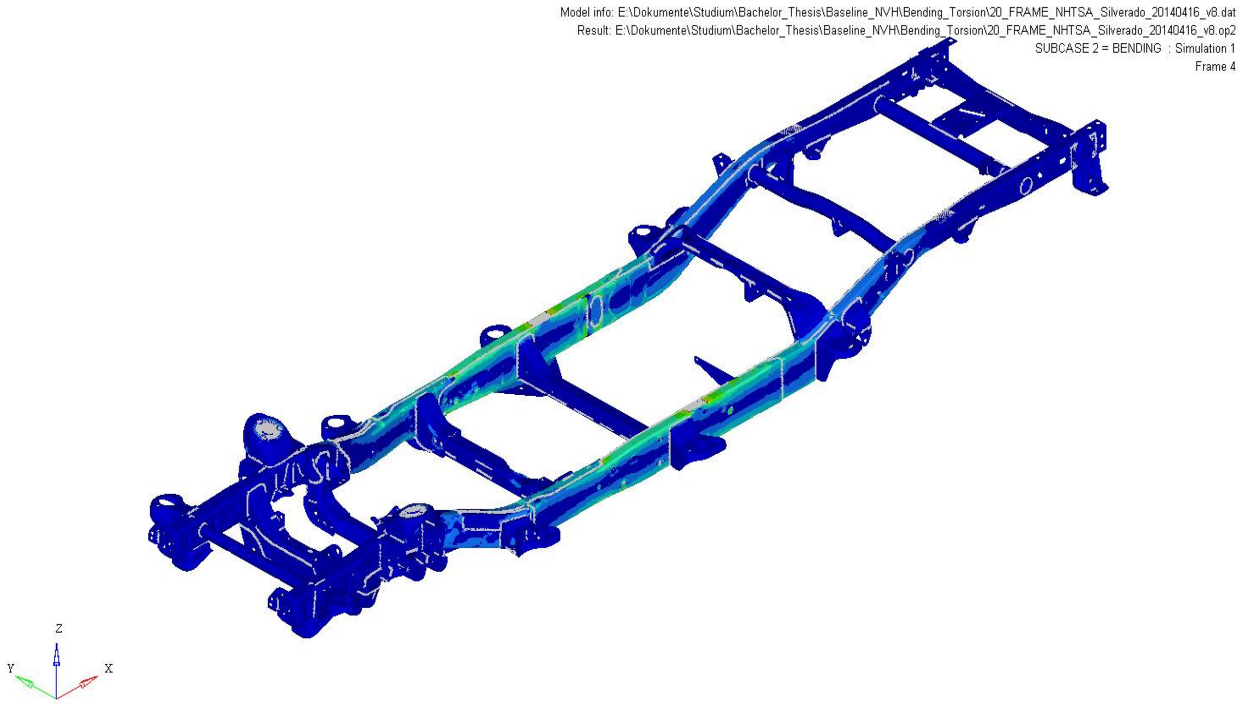

Figure A-1: Baseline result: Von Mises stress for bending LC

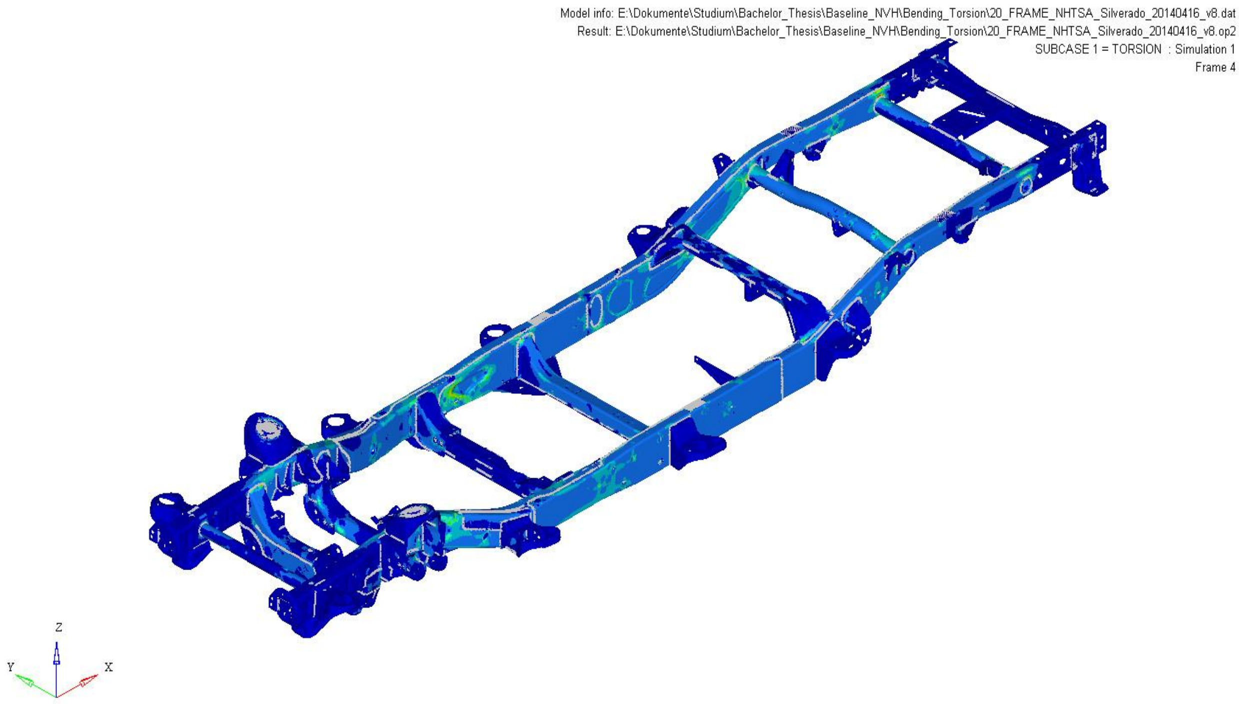

Figure A-2: Baseline result: Von Mises stress for torsion LC

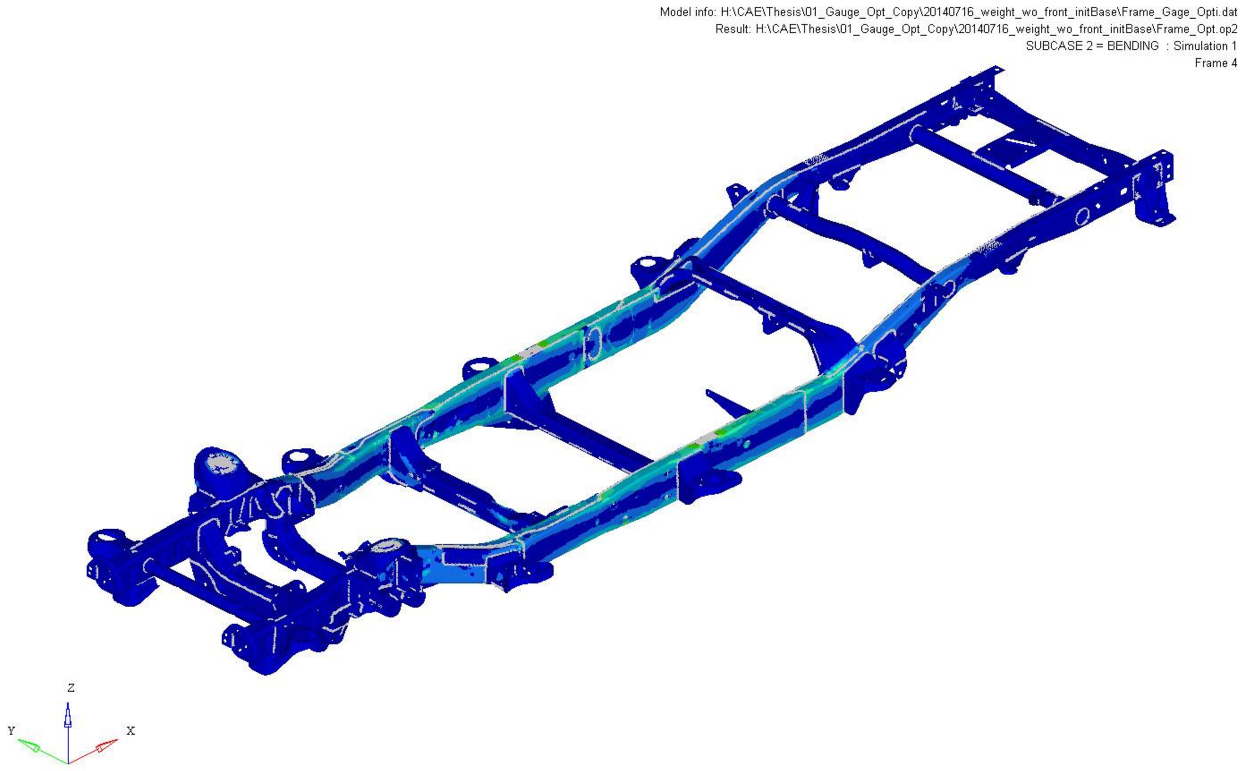

Figure A-3: Gauge result: Von Mises stress for bending LC

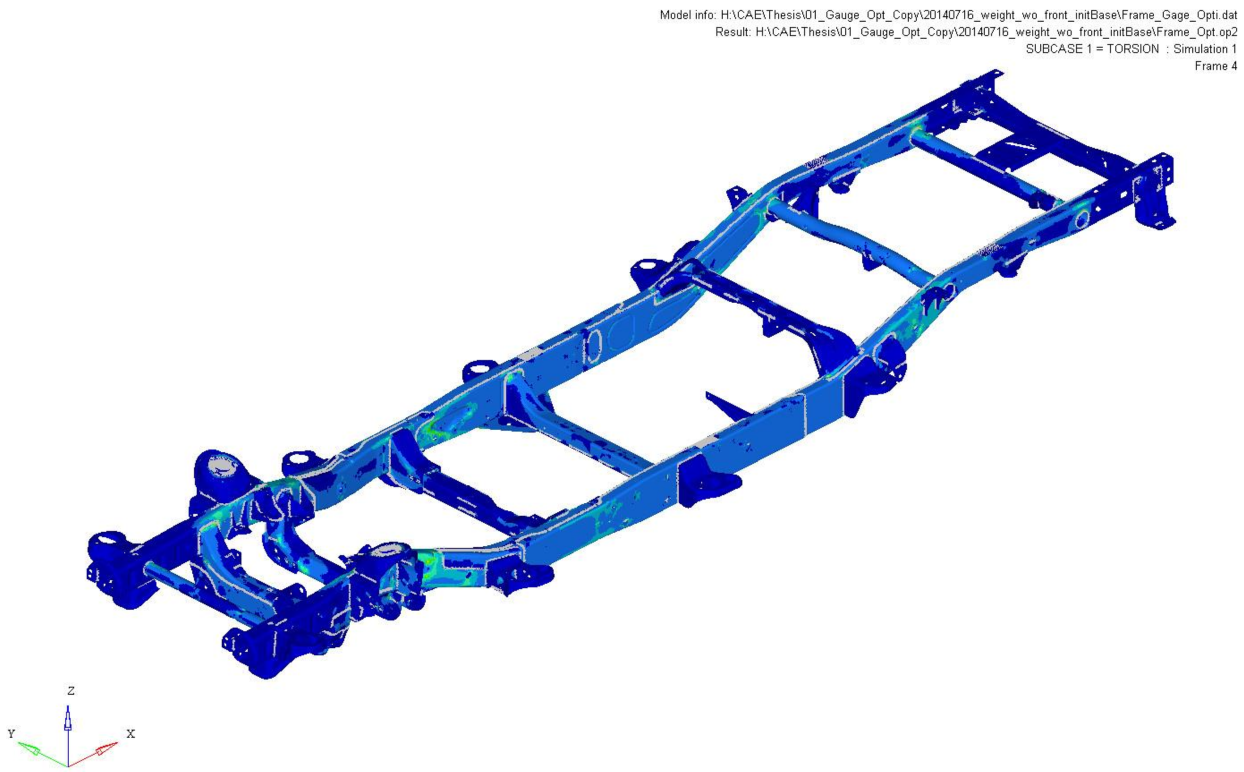

Figure A-4: Gauge result: Von Mises stress for torsion LC

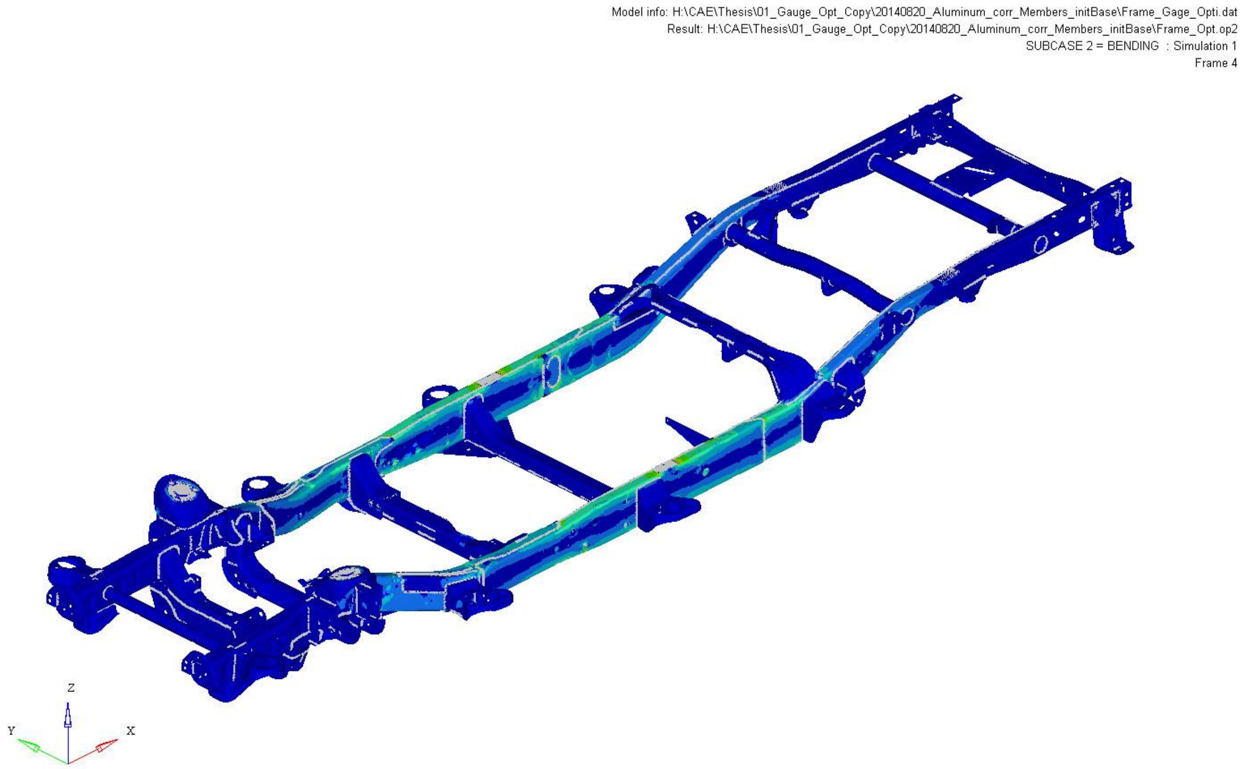

Figure A-5: Aluminum crossmembers result: Von Mises stress for bending LC

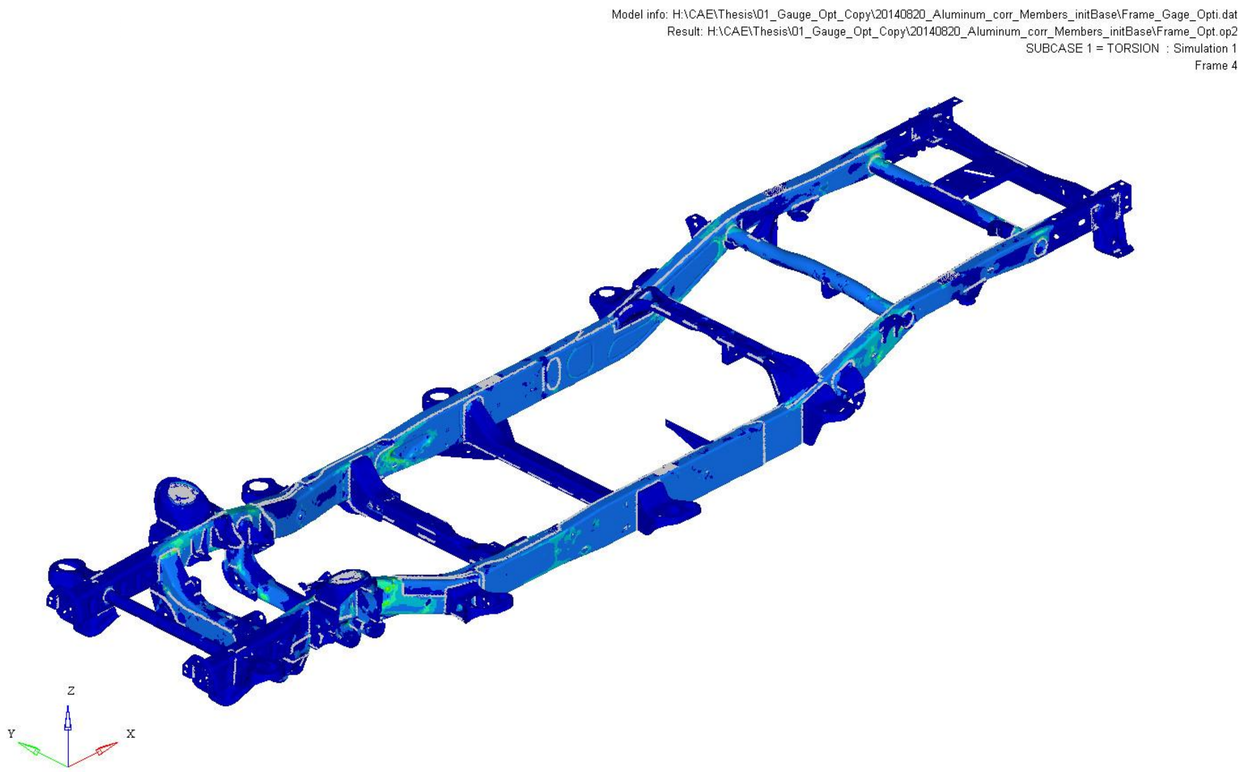

Figure A-6: Aluminum crossmembers result: Von Mises stress for torsion LC

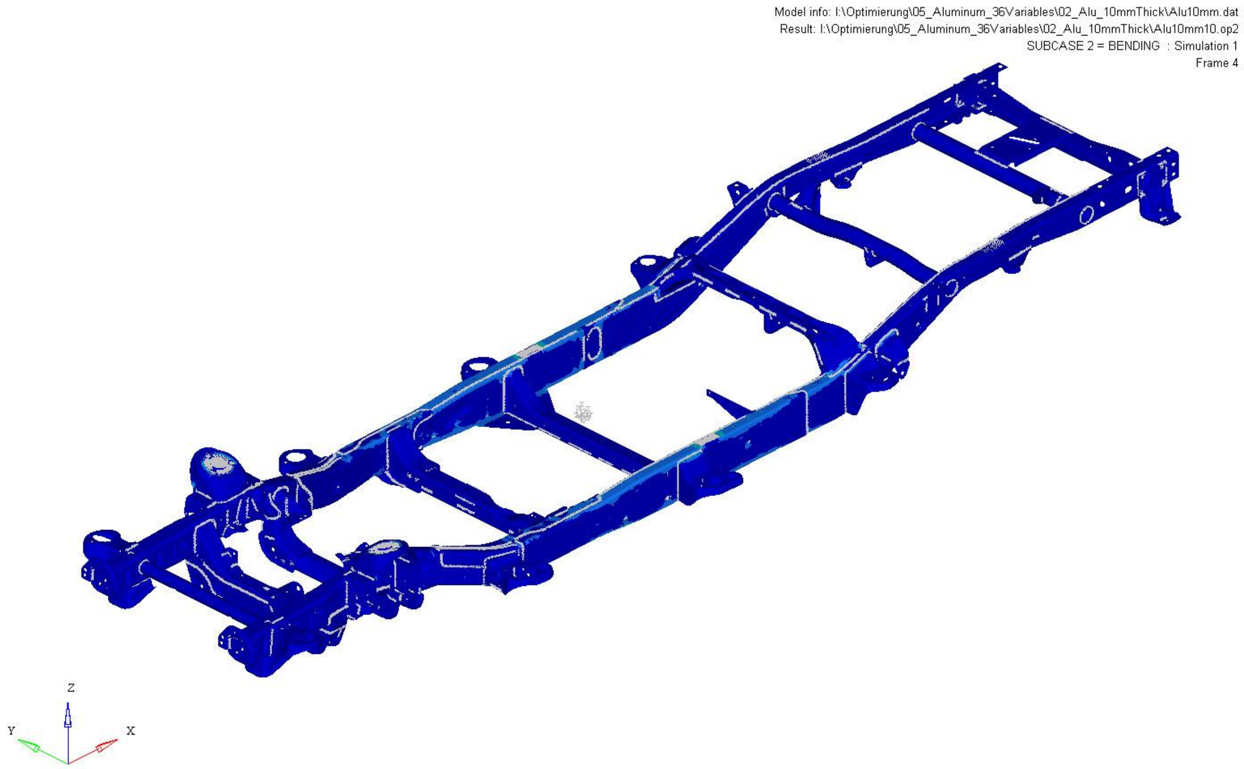

Figure A-7: Aluminum result: Von Mises stress for bending LC

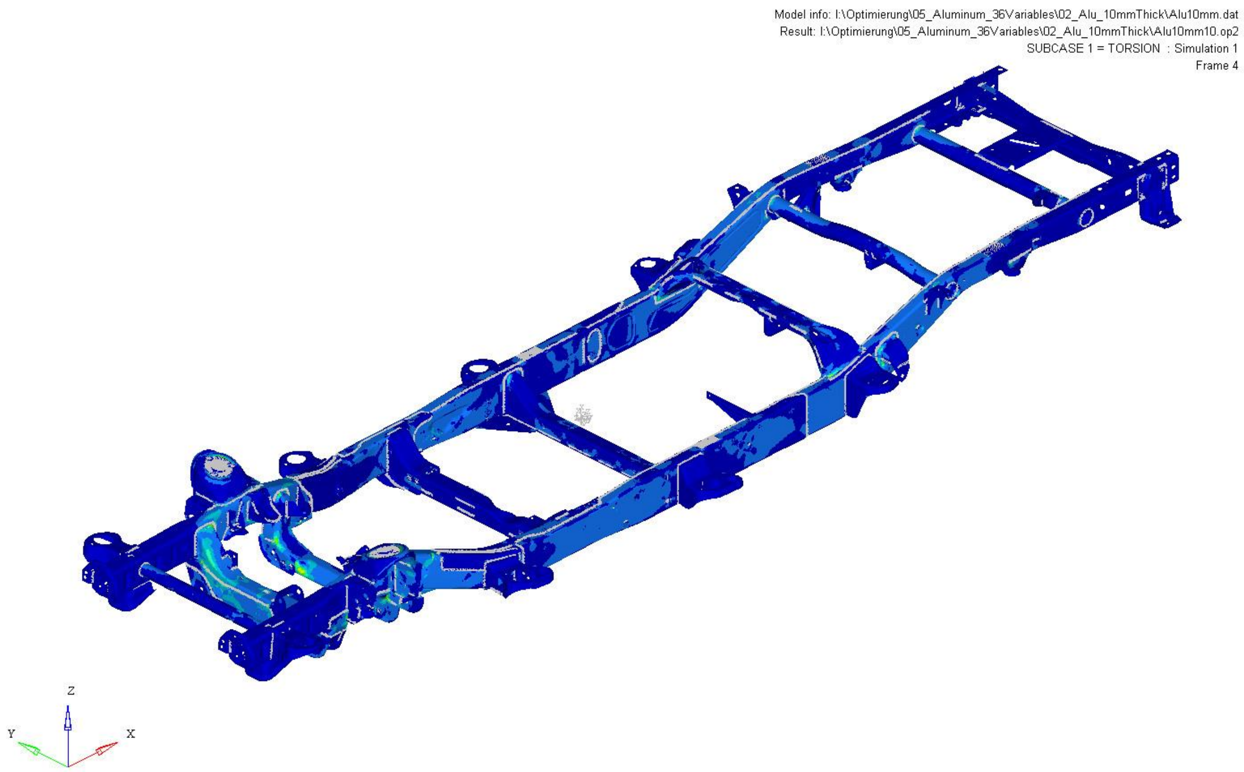

Figure A-8: Aluminum result: Von Mises stress for torsion LC

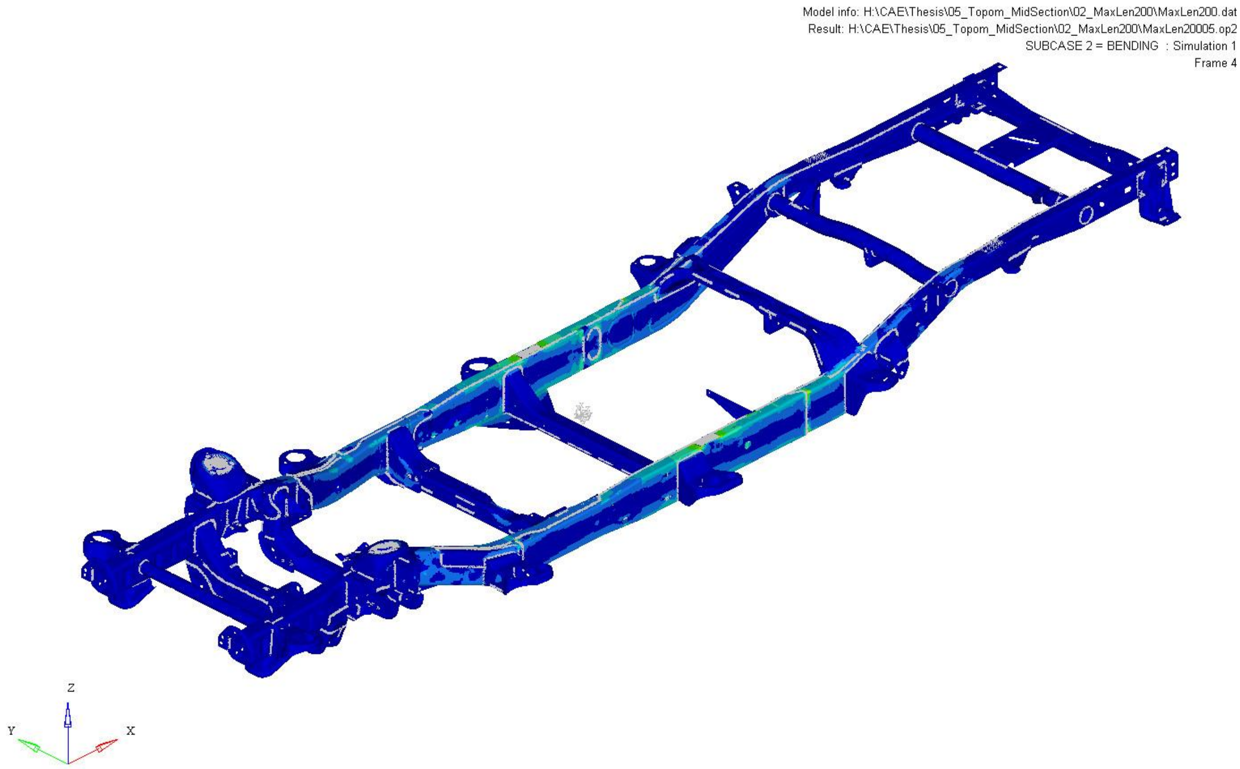

Figure A-9: TWB 200 mm width result: Von Mises stress for bending LC

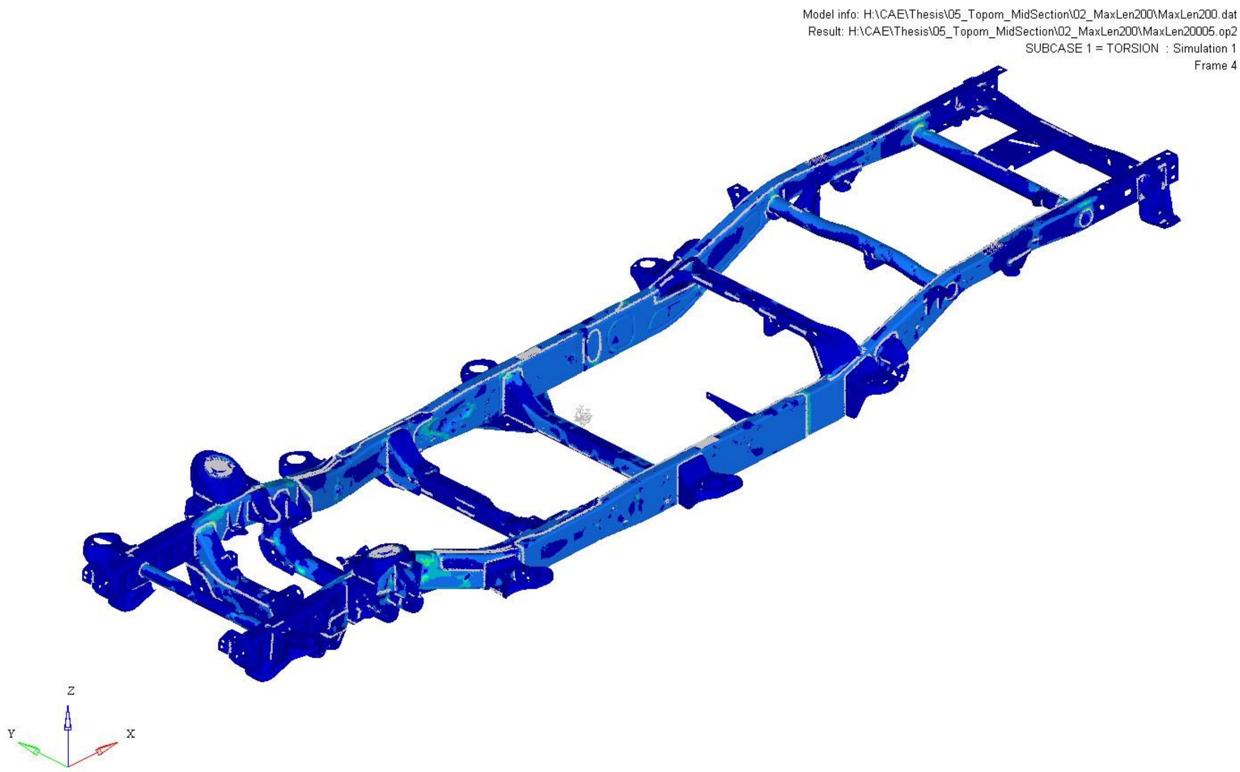

Figure A-10: TWB 200 mm width result: Von Mises stress for torsion LC

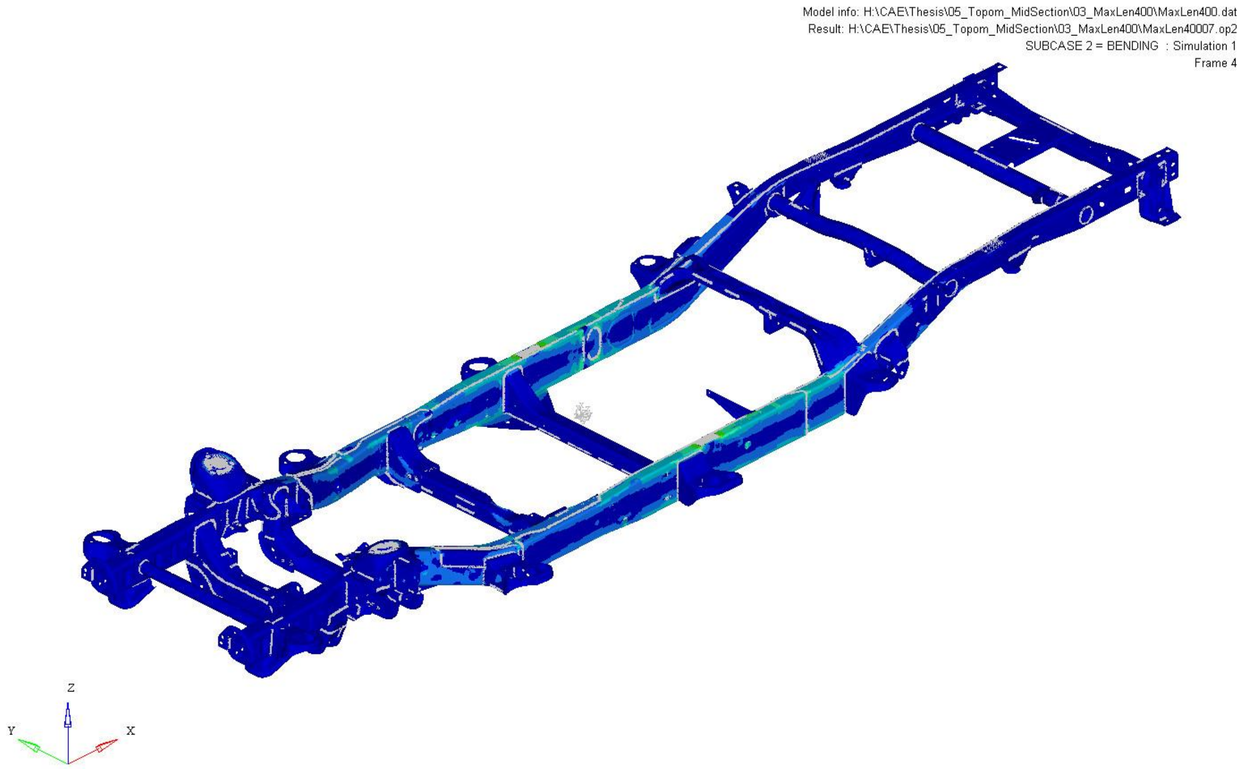

Figure A-11: TWB 400 mm width result: Von Mises stress for bending LC

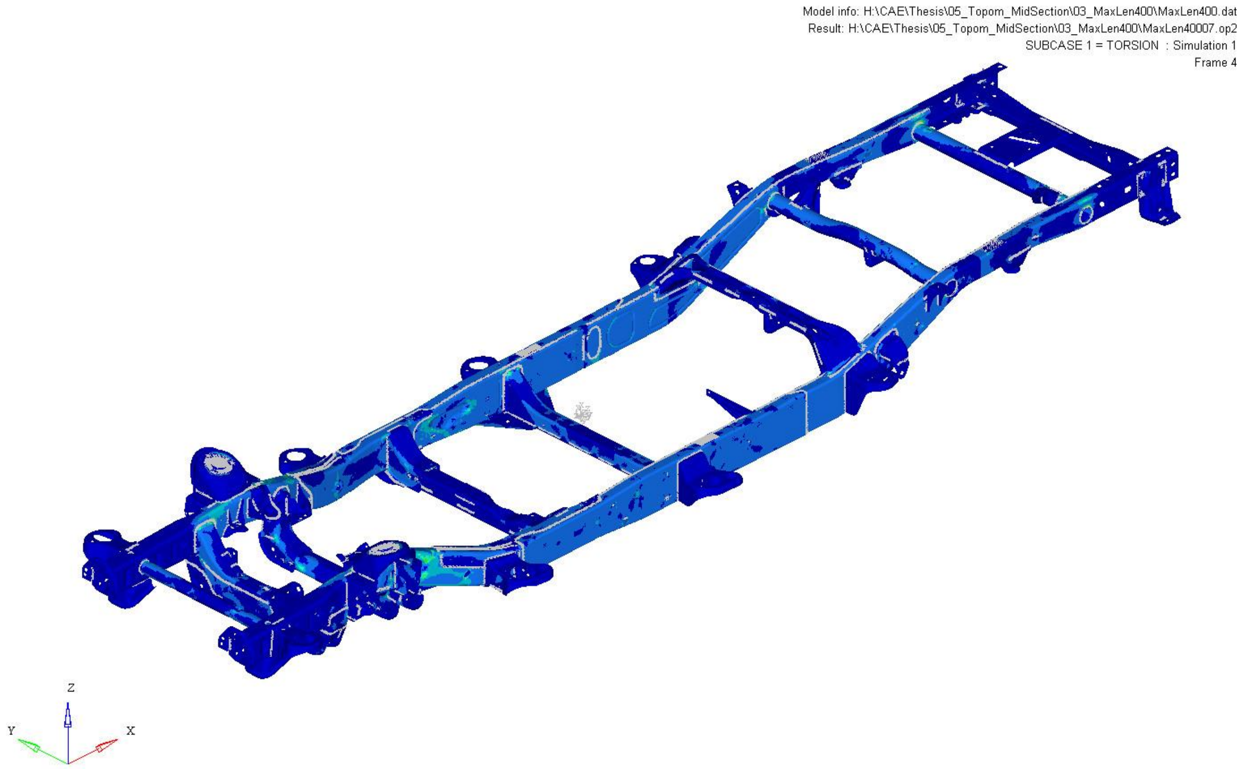

Figure A-12: TWB 400 mm width result: Von Mises stress for torsion LC

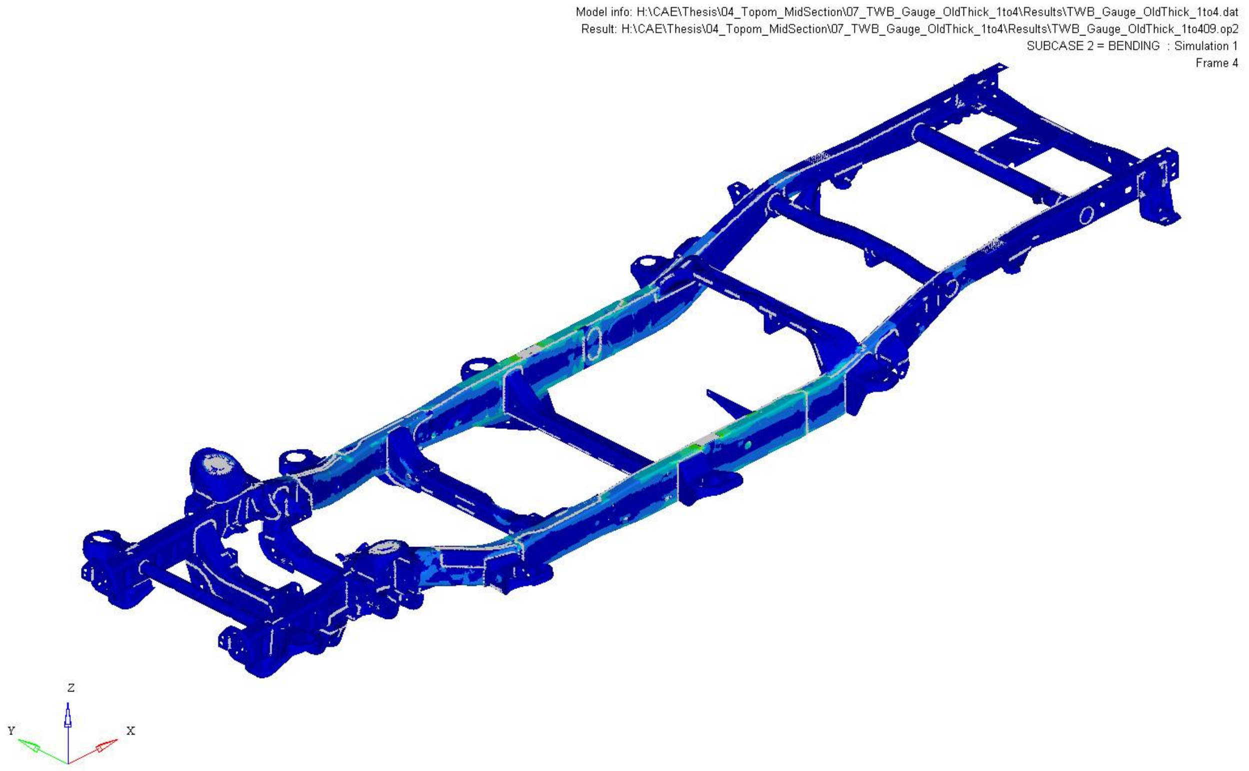

Figure A-13: Semi-manual TWB result: Von Mises stress for bending LC

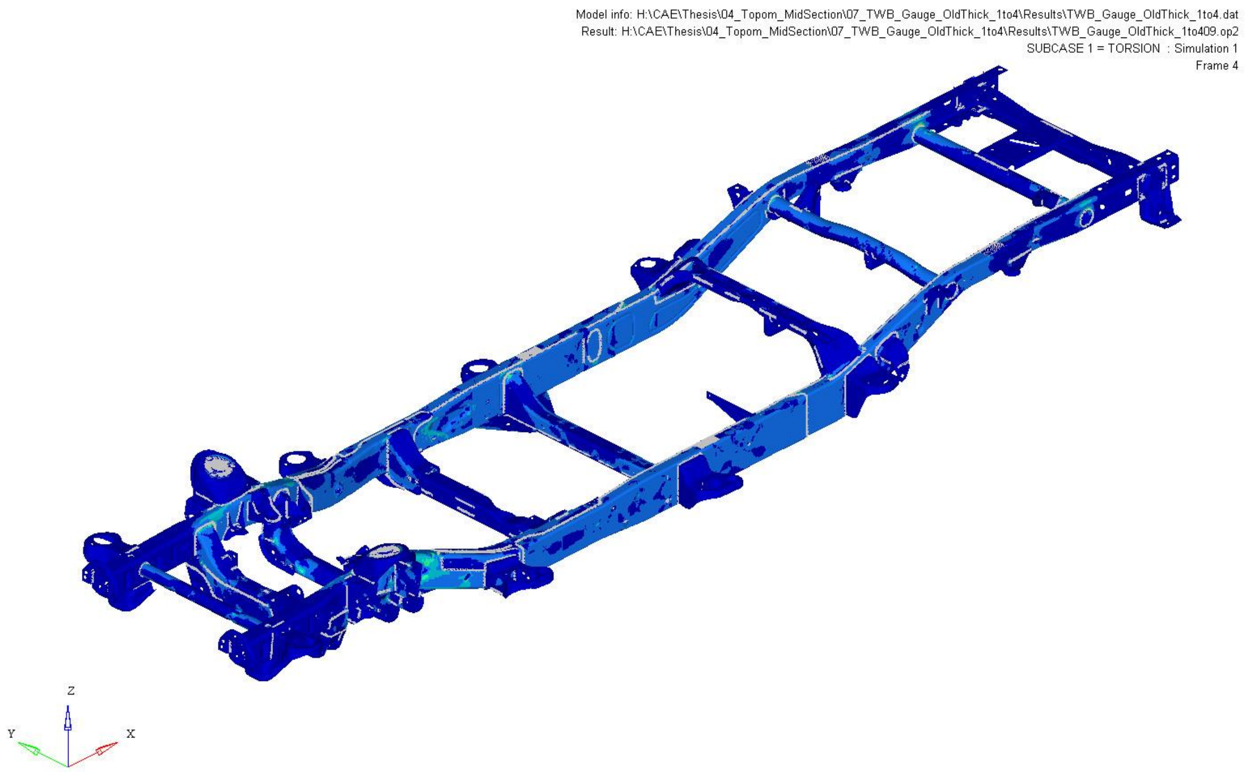

Figure A-14: Semi-manual TWB result: Von Mises stress for torsion LC

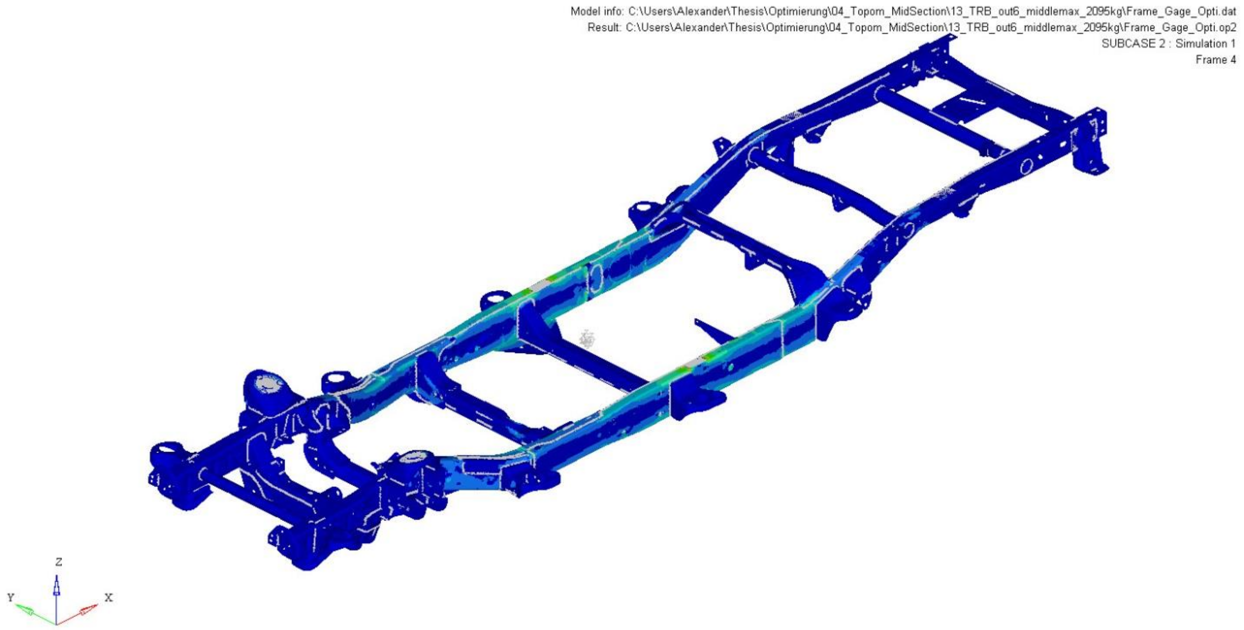

Figure A-15: Semi-manual TRB result: Von Mises stress for bending LC

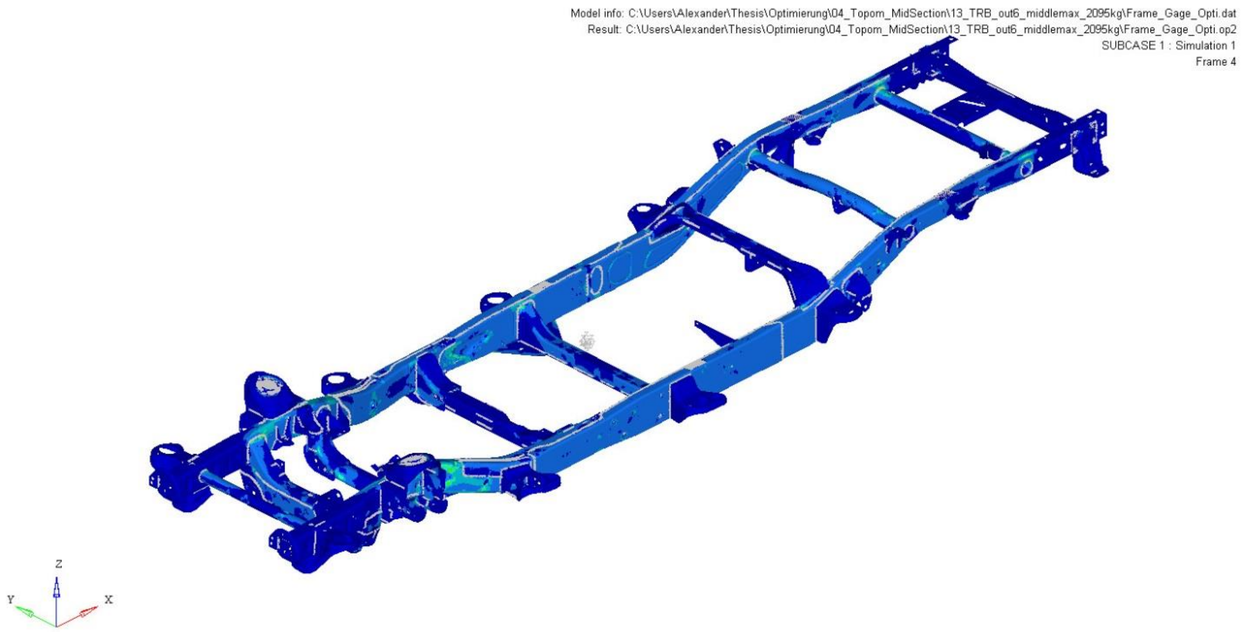

Figure A-16: Semi-manual TRB result: Von Mises stress for torsion LC

Eidesstattliche Erklärung

Hiermit versichere ich,

Name:	Mausolf
Vorname:	Alexander
Geb. am:	05.02.1990
Matr.-Nr.:	2051304

dass ich die vorliegende Bachelorarbeit mit dem Thema:

„Weight optimization of a pickup truck frame for the NVH load cases using FEA methods“

ohne fremde Hilfe selbstständig verfasst und nur die angegebenen Quellen und Hilfsmittel benutzt habe. Wörtlich oder dem Sinn nach aus anderen Werken entnommene Stellen sind unter Angabe von Quellen kenntlich gemacht.

Diese Arbeit wurde in gleicher oder ähnlicher Form noch bei keinem anderen Prüfer als Prüfungsleistung eingereicht.

Datum: ______________________________ Unterschrift: __________________________________