Universidade do Minho
Escola de Engenharia

André Moreira da Silva

Topology Optimization and Genetic Algorithms - application to an EV part based on standard OEM requirements

Novembro 2021

Universidade do Minho
Escola de Engenharia

André Moreira da Silva

Topology Optimization and Genetic Algorithms – application to an EV part based on standard OEM requirements

Dissertação de Mestrado
Mestrado Integrado em Engenharia Mecânica

Trabalho efetuado sob a orientação do (a):
Doutor José Luís Carvalho Martins Alves

Novembro 2021

Acknowledgments

This dissertation, in addition to its associated complexity, was carried out in a period marked in history by uncertainties, fears and isolation related to the COVID-19 pandemic. In this difficult journey there were always pillars that supported my doubts and fears and pushed me to face this challenge with all my commitment as possible. I want to thank these people.

To the team that received me at Continental Engineering Services - Porto, especially to Joel Silva and Claudio Pinto who accompanied me along the way and always motivated me to reach the extra mile necessary to highlight this dissertation from the others. The reception and monitoring by the entire CES Porto team was fundamental for the success of this work.

To professor José Luís Carvalho Martins Alves who was always available to clarify my doubts and to question my certainties in order to be able to reach a more consolidated and detailed path. His experience and assertiveness were fundamental to ensure that this dissertation was always on the defined path and that I didn't hesitate to go even further.

To my parents who always provided me with the best resources I could ask. Without your effort and sacrifice, I would not be able to carry out this dissertation. Finally, to my friends and girlfriend who have always accompanied me in this effort, my thanks.

DECLARAÇÃO DE INTEGRIDADE

Declaro ter atuado com integridade na elaboração do presente trabalho académico e confirmo que não recorri à prática de plágio nem a qualquer forma de utilização indevida ou falsificação de informações ou resultados em nenhuma das etapas conducente à sua elaboração.

Mais declaro que conheço e que respeitei o Código de Conduta Ética da Universidade do Minho.

RESUMO

A presente dissertação de Mestrado foi realizada na Continental Engineering Services (CES Porto), uma empresa do grupo Continental com mais de 1800 engenheiros localizados em 23 países e especialista na consultoria para a indústria automóvel. O objetivo desta dissertação foca-se no desenvolvimento de um novo processo de design mecânico que possa colmatar as falhas existentes nos processos de design tradicionalmente utilizados na indústria automóvel, facilitando o fluxo de trabalho entre as equipas das diferentes áreas e aumentando a qualidade dos resultados obtidos. O processo de design proposto sugere dividir-se todos os procedimentos de design em duas grandes etapas, a etapa inicial onde se irá executar diversos estágios de operações de Topology Optimization obtendo-se, de acordo com as condições exigidas, uma geometria ideal e uma segunda etapa onde se partirá da inspiração obtida da etapa anterior e aplicar-se-á algoritmos genéticos de forma a afinar as medidas de alguns parâmetros geométricos. De forma a comprovar a eficácia desta estratégia aplicar-se-á a mesma a dois casos de estudo, o primeiro denominado como "Initial Case Study – L Beam" com o objetivo de identificar limitações na aplicação prática desta estratégia e o segundo caso de estudo denominado como "Final Case Study – Crossbeam" com o objetivo de reproduzir fielmente o desenvolvimento de um componente mecânico analisando se a estratégia proposta poderá ser mais vantajosa que as estratégias de design tradicionais. No final, será realizada uma breve reflexão sobre as vantagens da utilização deste tipo de áreas de conhecimento, nomeadamente Topology Optimization e Algoritmos Genéticos, e a necessidade de aperfeiçoar os métodos existentes de forma a conseguir acompanhar as exigências cada vez mais elevadas da indústria automóvel.

PALAVRAS-CHAVE

Topology Optimization, Genetic Algorithms, SIMP, MOGA, OEM Requirements

ABSTRACT

This dissertation was carried out at Continental Engineering Services (CES Porto), a company of the Continental group with more than 1800 engineers located in 23 countries and specialist in consultancy for the automotive industry. The objective of this dissertation focuses on the development of a new mechanical design process that can fill the gaps in the design processes traditionally used in the automotive industry, facilitating the workflow between teams from different areas and increasing the quality of the results obtained. The proposed design process suggests dividing all design procedures into two large stages, the initial stage where it will be performed several stages of Topology Optimization operations, obtaining, according to the required conditions, an ideal geometry and a second stage where inspiration obtained through the ideal geometry will be used and genetic algorithms will be applied to fine-tune the measurements of some geometric parameters. To prove the effectiveness of this strategy, the same will be applied to two case studies, the first called "Initial Case Study – L Beam" in order to identify limitations in the practical application of this strategy and the second called “Final Case Study – Crossbeam” to faithfully reproduce the development of a mechanical component, analysing whether the proposed strategy could be more advantageous than traditional design strategies. In the end, there will be a brief reflection on the advantages of using this type of knowledge area, namely Topology Optimization and Genetic Algorithms, and the need to improve existing methods to keep up with the increasingly high demands of the automotive industry.

KEYWORDS

Topology Optimization, Genetic Algorithms, SIMP, MOGA, OEM Requirements

INDEX

FIGURE INDEX

TABLE INDEX

LIST OF ABBREVIATIONS, SYMBOLS AND ACRONYMS

Abbreviations/Symbol/Acronyms	Name	Description/Units
m	Mass	kg
p	Penalty Factor	-
E_0	Young Modulus of the structural material	MPa
E_j	Young Modulus in the step j	MPa
ρ_0	Density of the structural material	kg/m^3
ρ_j	Density of the element j	-
$\rho(x_j)$	Relative Density of the element j	-
SIMP	Solid Isotropic Material with Penalization	-
MOGA	Multi-Objective Genetic Algorithm	-
ETOPO	User Defined Result	-
CES	Continental Engineering Services	-
T.O.	Topology Optimization	-

1. Introduction

This dissertation was developed at Continental Engineering Services (Porto), a company of the Continental group responsible for solving technical challenges in the automotive and industrial sector especially in the areas of Driver Assistance, Interior Electronic Functions, Driveline & Electrification and Brake Systems. The main objective of the present work is to propose and test a new mechanical design process that can respond more adequately to the requirements of the automotive industry than the design process traditionally used. For this, Topology Optimization and Genetic Algorithms will be used to support the technical part of this new design strategy, firstly applying an introductory case study where the proposed strategy will be consolidated and finally validating the design strategy through the development of an automobile component (Crossbeam). The first chapter under analysis in this document is chapter 2, which presents the existing difficulties in the traditional design process and the proposal of the new design process with the incorporation of these two major areas of knowledge, Topology Optimization and Genetic Algorithms. This chapter is represented before the literature review since all the work developed focuses on this proposed strategy, however, from chapter 3 onwards, the chronological order is followed. In the next chapter, chapter 3, is presented the literature review focusing on the best algorithm to be used in the Topology Optimization (SIMP) operation and the compatible Genetic Algorithm (MOGA). Once the design strategy was defined and the algortihms were chosen, it was decided to use two case studies wherever possible to apply the strategy, starting with a simple case study in chapter 4 (L Beam) and ending with a real component for the automotive industry (Crossbeam) in chapter 5. Finally, in chapter 6.1, a brief discussion is presented based on the results obtained following the conclusion of this dissertation in chapter 6.2. In addition, further information can be found in Appendix 1 – Initial Case Study (Experimental Activity Details) and Appendix 2 – Final Case Study (Initial Dimensions).

2. GOAL AND STRATEGY OF THE THESIS

The main objective of this dissertation is based on the use of Topology Optimization tools and Genetic Algorithms to improve the mechanical design process at Continental Engineering Services - Porto (CES), however, to make improvements it is necessary to identify in detail the current process in the automotive industry and the main adjacent difficulties. CES has its main area of action in the development for the automotive industry, an industry traditionally recognized for its high demands, whether in terms of deadlines or terms of safety standards. Furthermore, it is extremely common that development teams have members from different countries working on projects simultaneously and it is necessary to ensure cohesion between the different stages of design.

In this way, it is usual to adopt two types of paths in the design for the automotive industry, represented in Figure 2.1 and Figure 2.2. In the first case, Figure 2.1, it is possible to observe that the Mechanical Designer starts the development through Design Loop 1 and progresses to different stages depending on the incorporation of the client's requirements and the project's objectives. When it reaches a stage where the design is consolidated, then the Mechanical Designer asks the Simulation Engineer to intervene to corroborate the developed geometry. Based on the results obtained from the numerical simulations, the Mechanical Designer makes minor adjustments to the geometry and completes its work. In this situation, sometimes the results obtained through numerical simulation recommend changes in geometry, however the geometry is already conditioned by several design stages and customer requirements, which leads to two types of situations: the restart of a new cycle of Design Loops with the inspiration of numerical results and the work previously done is wasted or the importance attributed to numerical simulation is reduced in order to avoid delays in the process.

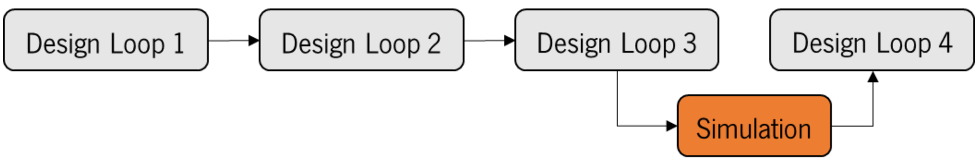


Figure 2.1 – First typical CAD/CAE process with simulation at the end of the design cycle.

Regarding the second path, represented in the Figure 2.2, the Mechanical Designer contacts the Simulation Engineer at the beginning of development and the main objectives of the project are established. Then, the mechanical design and numerical simulations take place simultaneously until an intermediate or final stage of the project where the results obtained are compared. This path also has some disadvantages, as sometimes this process is not fully synchronized, which means that the

Mechanical Designer or Simulation Engineer has to interrupt the development depending on the other, implying a delay in the project. If there is no such interruption, then a situation similar to the procedure represented above arises, where the relevance of the numerical simulations is reduced to be able to meet the established deadlines.

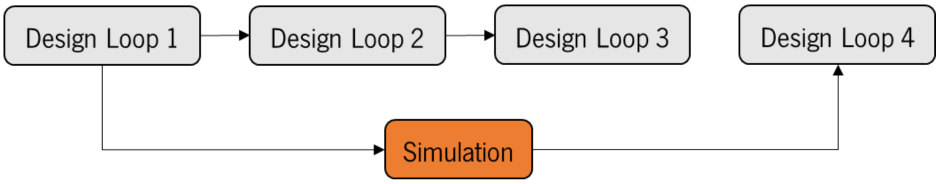


Figure 2.2 - Second typical CAD/CAE process with evolution of design loops during simulation time.

It is based on the difficulties presented in traditional procedures that this attempt to update and improve the classical design process with the incorporation of Topology Optimization and Genetic Algorithms (MOGA) arises, Figure 2.3.

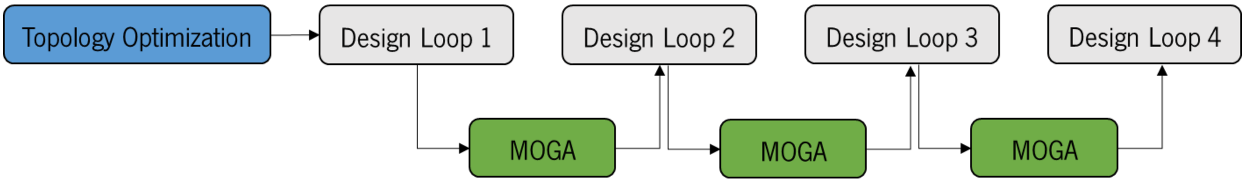


Figure 2.3 - Ideal process with the incorporation of Topology Optimization and Genetic Algorithms (MOGA).

This approach aims to put Topology Optimization first, performed by Simulation Engineer, as a source of inspiration for Mechanical Designer. This way, it is possible to guarantee that the numerical simulations do not end up diminished and it makes the Mechanical Designer process more accessible since it starts with a idea of an ideal geometry even if it is nothing more than an *stl file*. On the other hand, the Mechanical Designer, by incorporating genetic algorithms in the development of the design, allows it to be possible to guarantee that the most important parameters of the geometry have a numerical support that confirms their exact measurement.

If this process is analysed in more detail, as shown in the Figure 2.4, it is possible to understand that this new approach is ambitious and includes new development loops that did not exist before. It is true that for complex case studies the Topology Optimization tools still have some limitations and sometimes it is necessary some experience of the Simulation Engineer to forward the results to the commercial objective. It is important to recall that these projects in the automotive industry attach a high importance to the production of components, practically excluding any type of more exotic geometry that is only feasible to obtain through Additive Manufacturing and hence some development loops within the Topology Optimization component are necessary (Pang & Fard, 2020).

On the other hand, Mechanical Designer and Simulation Engineer must realize the advantages and limitations that genetic algorithms can bring to development. Genetic algorithms are an extremely interesting area at the academic level but still underdeveloped for the industry. If, on the one hand, it is interesting to analyse the different ways of generating populations, crossing chromosomes, mutations, among other factors, it is always necessary to remember that this will only be useful if they manage to fit in with the design process. Thus, MOGA was selected, a genetic algorithm incorporated in Ansys Workbench, which allows the optimization of parameters previously defined in the design in an easy way and with high compatibility with different CAD software (Creo, AutoDesk, SolidWorks, ..) to the detriment of more complete genetic algorithms but difficult to reconcile. Several loops related to the genetic algorithms may be necessary, namely through the definition of the possible ranges of variation of the parameter values and the necessary redesign after each optimization obtained, however it is important to note that MOGA was the best alternative found for the intended accuracy, process compatibility and ease of use.

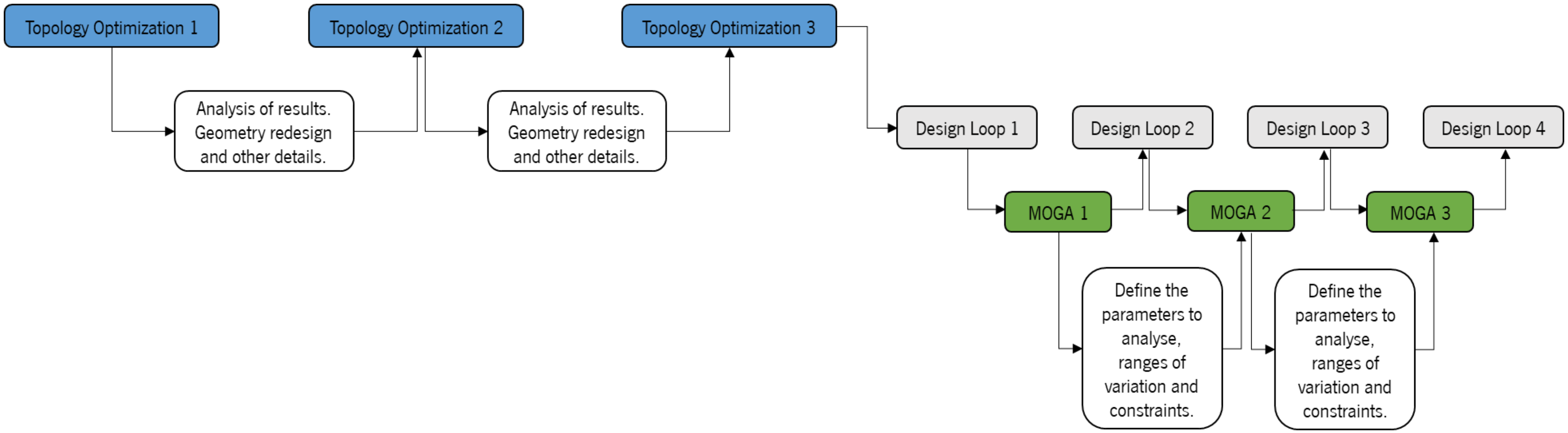


Figure 2.4 - Complete representation of the ideal process with detailed analysis of the Topology Optimization and Genetic Algorithms component.

3. LITERATURE REVIEW

In this chapter, it is intended to present a theoretical context of the areas that will be addressed throughout this dissertation, focusing mainly on the optimization algorithm selected to perform the Topology Optimization operations and on the genetic algorithm (MOGA). In addition, a brief analysis of the challenges that the automotive industry currently presents and the efforts made to ensure lightweight components will be presented. For such interesting areas, a more detailed analysis would be deserved, however, objectivity was privileged.

3.1 Automotive Industry and the Importance of Topology Optimization

In 2018, Portugal emitted a total of 67 417 t CO2eq - thousands, making it the thirteenth most polluting country in the European Union (including the United Kingdom). To be able to understand the reason for this position, when analysing the emissions produced in 2018, Figure 3.1, highlights the emissions associated with Energy (28,14%), Transport (25,45%) and Industry (22,13%). It is possible to observe a similar pattern when analysing emissions from Germany, a more developed country and concerned with environmental issues, dividing between Energy (35.10%), Industry (22.52%) and Transport (18.91%). The conclusion is clear, to be able to achieve the goals defined in the Paris Agreement (2016), high commitment and high innovation will be necessary for the areas of most concern, namely Energy and Transport.

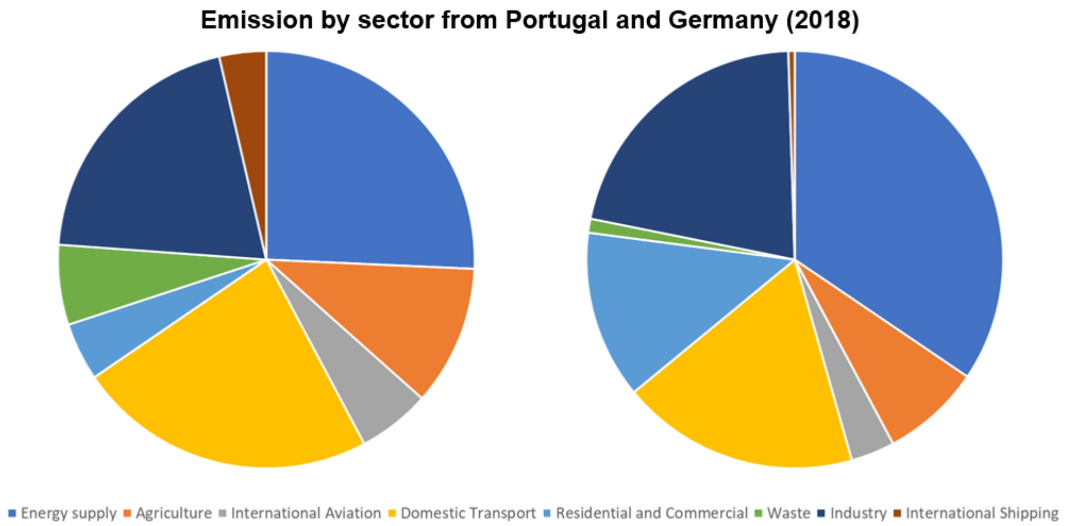


Figure 3.1 - Graphs relating to emissions produced by each sector in the year 2018. On the left is Portugal and on the right Germany. (European Environment Agency, 2021)

Currently, large companies in the automotive industry are dedicating great efforts to make cars more efficient, either through the optimization of internal combustion engines or through the adoption of electric mobility. Although companies have taken different paths to achieve the defined objectives, whether by changing the offer of cars to 100% electric or by developing cars with other sources of energy (hydrogen for example), there is a unanimous factor for making automobiles more efficient: reducing the mass of components. According to the research group ALLIANCE (Affordable Lightweight Automobiles Alliance), a project on automotive lightweight co-funded by the European Commission's Horizon and managed by six leading European carmakers, the weight reduction of 10 kilograms reduces emissions by approximately 1 gram of CO 2 per kilometre driven of an ICE vehicle (Reiland et al., 2020). This factor is also very important for electric cars since the autonomy of these cars is a key factor for market penetration and this is directly related to the mass of the vehicles, since the smaller the mass of the vehicle, the greater their autonomy. To accomplish the objective above presented and increase the decarbonization of automotive mobility through lightweighting it is essential to analyse an increasingly relevant industry trend, the area of Topology Optimization and Genetic Algorithms. This will be the major area of study present in the dissertation and can be analysed in greater detail in further chapters. (Fiebig et al., 2015)

3.2 Topology Optimization

Topology Optimization is a mathematical strategy that allows to achieve an optimal material distribution while respecting the imposed objective function and responding effectively to the restrictions, whether these are mathematical restrictions, boundary conditions or manufacturing restrictions. There are other more traditional structural optimization strategies, such as Size Optimization or Shape Optimization, however these do not allow the freedom of design exploration that Topology Optimization presents, therefore this thesis focus only on this method. To correctly formulate a structural optimization problem, it is necessary to identify the objective function, the design variables and the state variables as represented in the equation (1)

$$\begin{cases} min_x f(x, y(x)) \\ subject\ to \begin{cases} design\ constraint\ on\ x \\ state\ constraint\ on\ y(x) \\ equilibrium\ constraint \end{cases} \end{cases} \quad (1)$$

The design constraint (x) represents the structure design, for example the geometry, and the state constraint represents the structural response normally associated to stress, displacement, or strain and finally the objective function can be described as a profit function (maximize) or a cost function (minimize) (Kazakis et al., 2017). Sometimes the engineers may like to maximize the stiffness of a component while minimizing the mass meaning both objectives can enter in conflict. In these cases, there are two approaches, the weighting criteria and prioritizing one objective and constraint the other objectives. Mathematical speakin, the weighting criteria is equivalent to:

$$\begin{gathered} minimize\ f = [\omega_1 \times f_1(x) + \omega_2 \times f_2(x) + \cdots] \\ subject\ to\ x \in \Omega \end{gathered} \tag{2}$$

where ω_i must be positive and $\sum \omega_i = 1$. On the other way, with the prioritizing criteria is possibile to express this aproach as result of the following expression:

$$\begin{gathered} minimize\ f_1(\mathrm{x}) \\ subject\ to\ f_2(x) \leq c_2 \\ \cdots \\ f_m(x) \leq c_m \\ x \in \Omega \end{gathered} \tag{3}$$

where c_m are the objectives incoporated as contraints and excluded from the main objective function.

There are several approaches adjacent to the Topology Optimization area, which can be easily categorized into element-based, discrete or combined according to the algorithms they use. The most used approaches are element-based, characterized by discretizing the problem in finite elements whose solution should be based on. For this, the correct definition of the CAD model, mesh, and boundary conditions is extremely important since these values will support the Finite Element Analysis (FEA) which will later be integrated into the Topology Optimization (TO) process. Within the element-based category, there are several strategies such as the Rational Approximation of Material Properties (RAMP), the Level Set Method, the Solid Isotropic Microstructures with Penalization (SIMP), among others. In this dissertation, SIMP will stand out, since this methodology is one of the most used in Topology Optimization due to the excellent relationship between quality of results and computational requirement while possuing a higher maturity when compared to the other methodologies. On the other hand, the Level Set Method also has excellent properties that rival Solid Isotropic Material with Penalization (SIMP). This methodology, also element-based, is characterized by having a high capacity to adapt to drastic changes in topology using an excellent definition of the limits of geometry. According to Sigmund & Maute, the boundary condition of the geometry is obtained through the level zero (contour) of the level set function. This zero level derives from the objective function (such as minimize compliance for example), and the ideal configuration will be obtained by combining the movement of the function and the limits previously established. Both methodologies, SIMP and Level Set Method are present in the software used in this dissertation, Ansys Workbench 2020R2, and this subject will be addressed in more detail in later chapters. In Figure 3.2 it is possible to observe a small scheme that represents the differences between the SIMP algorithm and the Level Set Based in a simplified way.

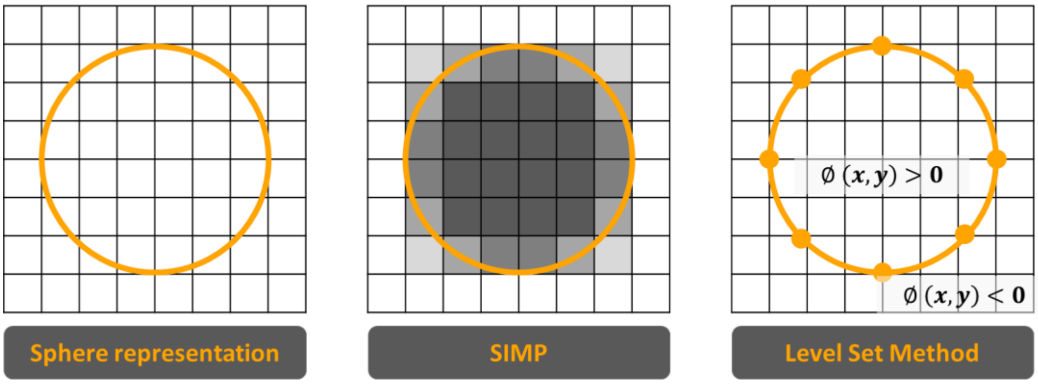


Figure 3.2 - Representation of the interpretation of a sphere through elements based on the SIMP algorithm and the Level Set Method algorithm

Regarding discrete approaches, they present several limitations for more complex Topology Optimization problems. Among several approaches the main spotlight belongs to the Evolutionary Structural Optimization (ESO), the Additive Evolutionary Structural Optimization (AESO) and the Bidirectional Evolutionary Structural Optimization (BESO), resulting from a combination of the ESO method and the AESO (Tyflopoulos et al., 2018). Although academically interesting these are not available in the software to be used so they ended up not being considered in this study. Because SIMP guarantees a higher level of confidence in the results, it will be on this algorithm that the next subchapters will be focused.

3.3 Software Analysis

After analysing the most recommended Topology Optimization algorithms in the literature, it was necessary to advance to a stage where it was necessary to verify which of these algorithms were found in commercial software and why.

For this, the first step was to identify the software capable of performing Topology Optimization operations on the market, as well as their limitations and cost. Thus, it was possible to find several capable software such as Ansys Workbench, COMSOL, SolidWorks, ABAQUS, Altair OptiStruct, SIEMENS NX, AutoDesk NASTRAN-IN-CAD, among others. After identifying the existing software, it was possible to progress to a more refined analysis to obtain the optimization algorithms that each software used and compare with the knowledge obtained in the bibliographic analysis. This point has become fundamental, since several algorithms praised in the bibliographic analysis were not found in the best-known commercial software, such as the AESO or BESO algorithm, and SIMP was present in every single one. This does not mean that the literature is incorrect, simply for commercial software it is needed algorithms that are easy to understand and use, as well as an excellent relationship between computation time and results. Because SIMP is an algorithm that has all these characteristics, it could be an explanation for being the most unanimous from a commercial point of view.

After analysing the different software, three stood out: Ansys Workbench, Siemens NX and AutoDesk Nastran-In-Cad. In the table below, it is possible to observe that SIMP is effectively the most used algorithm due to its maturity and effectiveness, however, several methods such as Lattice Penalty and Level Set Based also present some prominence.

Table 3.1 - Comparison between software with the best cost/quality ratio for Topology Optimization.

AutoDesk NASTRAN-IN-CAD	Ansys Workbench	SIEMENS NX
SIMP	SIMP	SIMP
Level Set Based	Level Set Based	RAMP
	Lattice Penalty	Lattice Penalty

To be able to find the most suitable software among the three mentioned above, several criteria were used, such as cost, ease of obtaining licenses, compatibility with the simulation software used in the company and the possibility of incorporating genetic algorithms. This last factor proved to be decisive since Ansys Workbench had a tool compatible with genetic algorithms incorporated in the simulation panel itself, which avoided the need to extract the results to other software capable of performing operations with genetic algorithms. In conclusion, Ansys Workbench 2020 R2 (professional license) was the selected software.

3.4 Solid Isotropic Material with Penalization (SIMP)

Although the concept of Topology Optimization has always existed in the empirical knowledge of an engineer, the concept of Topology Optimization known today has only begun to materialize and gain importance with the development of machines with excellent computing power. Before 1989, Topology Optimization had only been analysed using the values 0 and 1, that is, through a discrete form. Beginning in 1989, Bendsøe proposed a method of varying design variables continuously, that is to say non-discrete, by assigning more or less influence to intermediate values between 0 and 1. This method became known as Solid Isotropic Material with Penalization (SIMP) and today is the most popular numerical FE-based topology optimization method, both academic and industrial. Even though this method was developed about 30 years ago today, it is still found in most commercial software capable of performing Topology Optimization operations, namely in Ansys Workbench (software used in the present work), SolidWorks, Altair Optistruct, COMSOL, ABAQUS, Siemens NX, among others, which proves the robustness and quality of this method (Manuel & Monteiro, 2017). In this chapter it will be presented its operation, the main associated errors and other details associated with this method. The key factor of the SIMP optimization method is based on the attribution of a relative density to each element obtained in the meshing process of the components. This relative density is an artificial density that must have values between 0 and 1, contributing to the alteration of the component's properties, such as structural stiffness. This density can be obtained by the following formulation:

$$\rho(x_j) = \frac{\rho_j}{\rho_0} \tag{4}$$

Where,

$\rho(x_j)$ is the relative density of the element j

ρ_j is the density of the element j

ρ_0 is the density of the structural material.

To be able to relate the relative density to the structural stiffness, the present method establishes a relationship, for each finite element, between Young Modulus and the relative density through a penalty method. This relationship can be otbained through equation 5:

$$E_j = E_0 \times \left(\frac{\rho_{j-1}}{\rho_0}\right)^p \tag{5}$$

Where,

E_0 is the Young Modulus of the structural material.

E_j is the Young Modulus in the step j

ρ_{j-1} is the density of the element in the preceding step before the step j

ρ_0 is the density of the structural material.

p is the penalization factor (normally this factor is equal to 3 and never inferior to 1)

3.4.1 Penalty Factor

The equation refered previously, in addition to relating stiffness through Young's Module, introduces the concept of the penalty factor. This factor is responsible for influencing the relative densities of each element to approach the value 0 (without material in that element) or value 1 (with material in that element). This factor is fundamental to the success of this method as it facilitates the computing process and allows the user to control it to comply with the desired conditions. It is easy for the algorithm to understand that elements with relative densities of 0.8 are critical for the component and therefore the material cannot be removed, however for situations where the relative density of the element are between 0.4 and 0, 6 it is essential to use this penalty factor to approximate these relative densities of 0 or 1. The value of the penalty factor should never be below 1 and may be extended to infinity, however, usually the value that offers the best results is p=3. It is possible to observe in the figure below the influence that factor may have on Young's Module which corroborates the theory that the results become redundant as the valour of p increases and concluding that p=3 appears to be the most equilibrated approach.

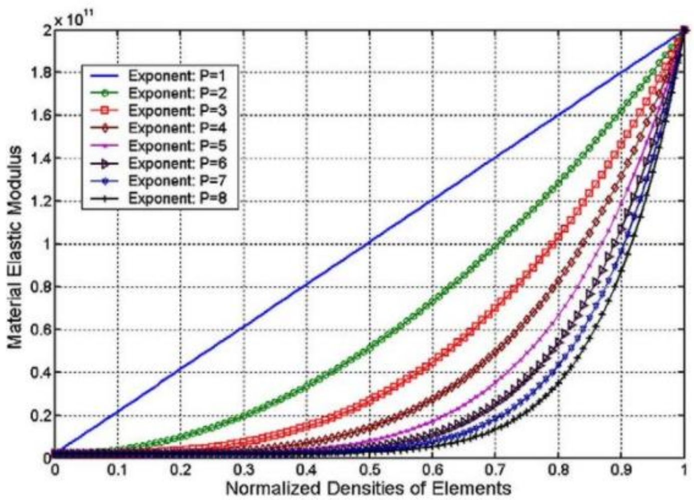


Figure 3.3 – SIMP interpolation scheme. (Deaton & Grandhi, 2014)

3.4.2 Main problems associated with SIMP

Although SIMP is an optimization method with high maturity and international recognition, it may present some associated errors, namely checkerboarding, mesh dependency and local optimum. To guarantee a full understanding of this method, it becomes necessary to analyse the origin of these problems as well as possible solutions or alternatives. This process will be explained throughout this chapter.

Checkerboarding refers to the pattern created by the organization between elements without material (relative density = 0) and elements with material (relative density = 1). This type of problem can be frequent and it is important to eliminate it since it confers a structural rigidity superior to the real one, as well as originating geometric configurations that are quite complicated to manufacture. For this, it is necessary to resort to the use of sensitivity filters that identify these types of patterns and eliminate them, as seen in the figure below. The Figure 3.4 refers to a case study for a cantilever beam where it is possible to observe that in the initial results (figure in the centre) the existence of a checkerboarding pattern and with the application of the filter (figure right) this problem has been minimized.

Figure 3.4 - Checkboarding representation in a cantilever beam. (Koga et al., 2013)

One of the main characteristics of SIMP is the density assignment related to the elements, however, this characteristic can turn out to be a problem since the whole optimization process depends on the elements formed through the creation of meshes. This means that although all the boundary conditions of the problem are the same, models with different meshes may present drastically different geometries. To minimize the impact of this characteristic, it is possible to use the penalty factor incorporated in the SIMP to avoid grey areas, that is, areas in which the elements would not have sufficiently low relative densities to be removed or high enough to be maintained in the final geometry. In addition, it is also possible to use filters to minimize this problem, however, this factor must be considered in the whole optimization problem which is why it is essential to use high-quality meshes. (Jensen, 2018)

Finally, SIMP may present another disadvantage associated with Local Optimum since the problems of Topology Optimization solutions provided by SIMP may present numerous Local Optimum, which makes it extremely complicated to find the best existing solution. This statement is corroborated since small changes in the initial parameters of the problem, such as the geometry of design domains or size of elements, can arrive at drastically different solutions. This problem may not be seen in non-gradient based algorithms since they tend to explore larger boundaries of the problem as opposed to algorithms such as SIMP (gradient-based) which are quite restricted to the initial design. (Sigmund & Petersson, 1998)

3.5 Genetic Algorithms

The area of genetic algorithms, although it has existed for many years, is gaining more and more importance with the evolution of computational capacity and with the requirements existing in the market. In summary, a genetic algorithm (GA) is a method for solving problems based on a natural selection process that mimics biological evolution. There are several interesting factors to analyse concerning genetic algorithms, however, to be able to comply with the objectivity required for this dissertation, only MOGA, Multi-Objective Genetic Algorithm, will stand out since this will be the genetic algorithm used from the Ansys Workbench software.

Of the different possible topics to be addressed concerning MOGA, two will be highlighted: the necessary steps in the functioning of the genetic algorithm and the different strategies it presents to create new populations. Starting with the workflow associated with the operation of MOGA, it is possible to observe in Figure 3.5 a summary of the main steps of MOGA. After the correct definition of the parameters to be optimized, MOGA creates an initial population. This population will serve as a starting point for optimization since it is based on this initial population that the MOGA will generate new populations, a process that will be analysed in more detail throughout this chapter. Once the new populations are created, the algorithm updates the DesignPoints responsible for each combination of parameters and then checks whether the optimization has converged or not. To be able to complete the optimization cycle, the Maximum Allowable Pareto Percentage or the Convergence Stability Percentage must be reached. If this is not possible, the process will generate new populations again until the best possible results are achieved. Note that these convergence criteria can be changed in Ansys Workbench according to the user's requirements.

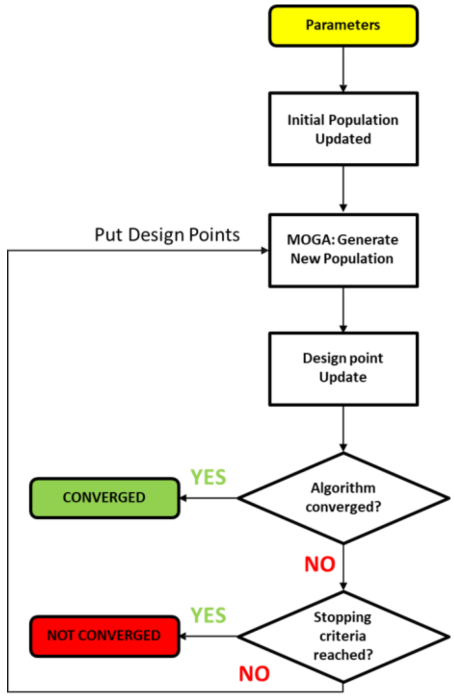


Figure 3.5 – MOGA Workflow. Inspired by Ansys Workbench.

Once the main stages of MOGA are understood, it becomes interesting to analyse the process of generating new populations. For this, there are two important factors: Cross-over and Mutation. Cross-over combines (mates) two chromosomes (parents) to produce a new chromosome (offspring). The main objective of this strategy is to believe that the new chromosome generated can obtain the best characteristics of each parent, creating an optimization of results. There are also two types of Cross-over possible to perform in Ansys Workbench, depending on whether the parameters are continuous or whether they are discrete or have manufacturing restrictions. If the Cross-over is performed for continuous parameters, it is possible to use the equation shown below.

$$Offspring1 = a \times Parent1 + (1 - a) \times Parent2 \quad (6)$$

$$Offspring2 = (1 - a) \times Parent1 + a \times Parent2 \quad (7)$$

If the formula is applied to the example shown below, it becomes simple to understand how new results are generated.

$$Parent1: (0.5)(0.7)(1.6)(3.4)$$

$$Parent2: (0.3)(0.9)(1.3)(3.8)$$

Considering, $a = 0.7$, the results obtained would be:

$$Parent1: (0.44)(0.76)(1.51)(3.52)$$

$$Parent2: (0.36)(0.84)(1.39)(3.68)$$

On the other end, if the Crossover is performed for discrete parameters or continuous parameters with manufacturing constraints then the process becomes slightly more complex. Each parameter is composed of a binary string that corresponds to the number of levels. For this it is possible to use the formula $2^n - 1$, obtaining 15 values (levels) encoded in 4 bits. The concatenation of these chains forms the chromosome, which crosses over with another chromosome. From this point on, there are three possible types of Crossovers: one-point, two-point and uniform. The Cross-over by one point randomly selects a Cross-over point within a chromosome and interchanges two parent chromosomes at this point producing two new offspring, the two-point Cross-over does the same thing but with two selected Cross-over points. Ultimately, uniform Cross-over decides that each parent assigns each gene to the offspring chromosome. In Figure 3.6, taken from the theoretical support of Ansys Workbench, it is possible to observe a scheme which facilitates the understanding of each type of Cross-over.

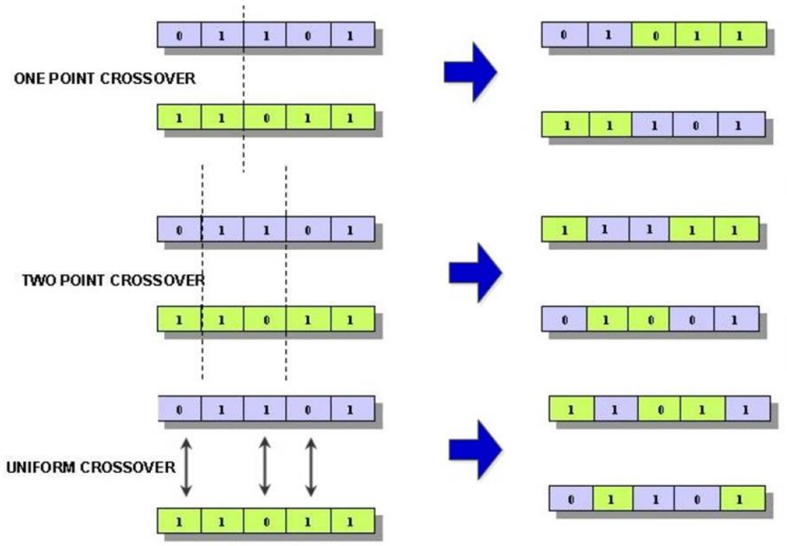


Figure 3.6 - Scheme with the three types of Cross-over available.

Finally, the last major detail to be considered in MOGA is its ability to introduce mutations in the creation of populations. A mutation occurs when one or more genes from the early stage of a chromosome are altered, resulting in a completely different gene being added to the population. This characteristic allows avoiding stagnation phenomena in the creation of populations, leading to better quality results. As with Cross-over, mutation can be applied to continuous or discrete/continuous parameters with manufacturing constraints. Regarding continuous parameters, the mutation can be applied following the following equation:

$$C = p + (UpperBound - LowerBound)\delta \quad (8)$$

Where C is the child, P the parent and δ the small variation. (Algorithm, 2020)

Regarding the other type of parameters, the mutation operator inverts the gene value, that is, if it is 0, it changes it to 1 or vice versa, with a probability of 0.5. This alteration alters the chromosome that will generate new results when crossing with the next chromosomes. In Figure 3.7 it is possible to observe a summary scheme with the main stages of MOGA in the creation of new populations.

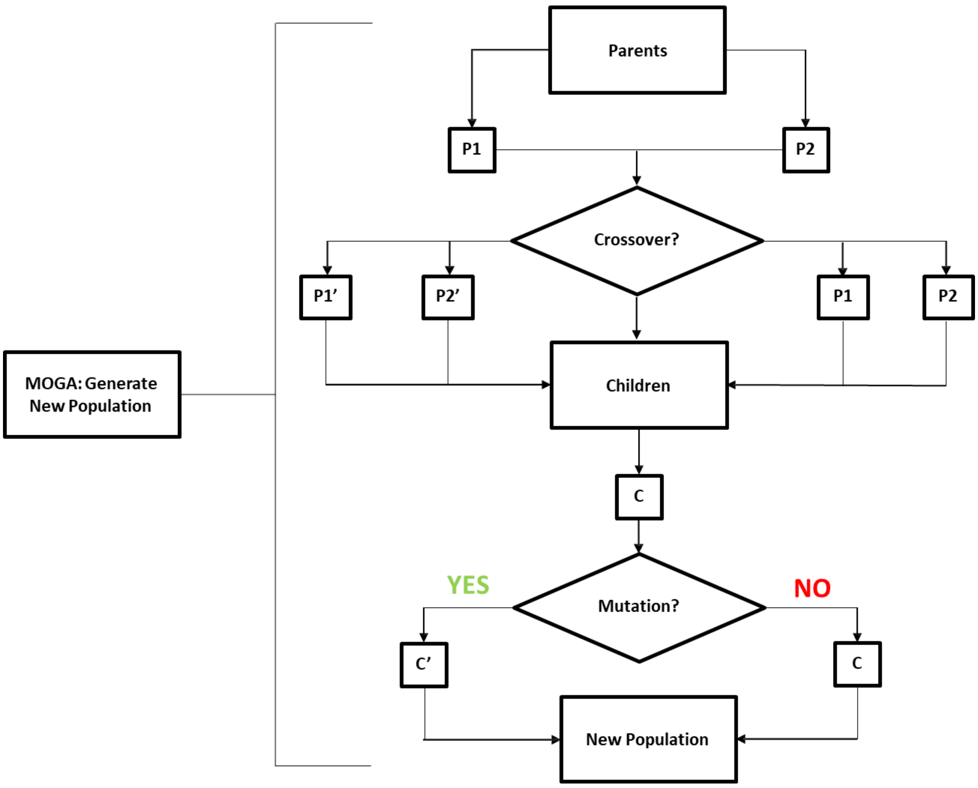


Figure 3.7 - Procedure that the MOGA uses to generate new populations.

4. Initial Case Study – L Beam

The main objective of this dissertation focuses on the development of a new mechanical design strategy using techniques through Topology Optimization operations or Genetic Algorithms to solve the main problems in the development of components in the automotive industry. To prove the effectiveness of the proposed strategy, a component of an electric vehicle named by Crossbeam was considered as an example for the application of the proposed new design process. However, to gain confidence/experience in the optimization process and in the use of Genetic Algorithms, an initial case study (of much lower complexity than the main case study) was established. With the addition of this step, the intention is to gain confidence in using the software, discover its main limitations (and how to get around them) and establish an ideal optimization process (defining the most critical steps and the main details to be considered). Finally, to consolidate this entire path, it is intended to compare the results obtained by numerical simulation with an experimental activity, thus obtaining some support for the defined strategy and validating (or not) the accuracy of the software used. Throughout this chapter, the main stages of the initial case study will be presented, starting with the choice of the case study and ending with its experimental activity.

4.1 Presentation of the initial case study

To be able to incorporate an additional case study in the dissertation and carry out an experimental activity, an attempt was made to find a case study present in the recommended bibliography, with the problem conditions well defined and above all easy to produce and compatible with the existing equipment in the materials testing laboratory at the University of Minho, this last factor being critical for the selection of the case study.

After analysing several examples, the case study presented at the Figure 4.1 was reached. This case study was provided by Erik Holmberg through the University of Linköping (Sweden) in the article entitled "Structural optimization of an L-shaped beam subject to global stress constraints." Although this case study was based on the article mentioned above, it is usually used to exemplify the advantages of using Topology Optimization tools, thus presenting several results that allow comparing and corroborating the results obtained in the dissertation.

It is possible to observe that there are no defined measures, nor quantification of the applied load, however, these factors were obtained through the limitations of the experimental procedure and in order to minimize the component's manufacturing cost. It is possible to conclude that the component is recessed in the upper area and only a load is applied to the right end of it and this is the most important factor to initiate the resolution of the problem.

Figure 4.1 - Structural optimization of an L-shaped beam subject to global stress constraints, Erik Holmberg, *University of Linköping.*

4.2 Problem Definition and Static Analysis

The dimensions of the case study, shown in Figure 4.2, were defined according to the tensile test equipment available and represent the starting point of the beginning of the analysis of the case study. The 5mm hole located in the lower right area of the component stands out since this detail will be used to carry out the experimental activity.

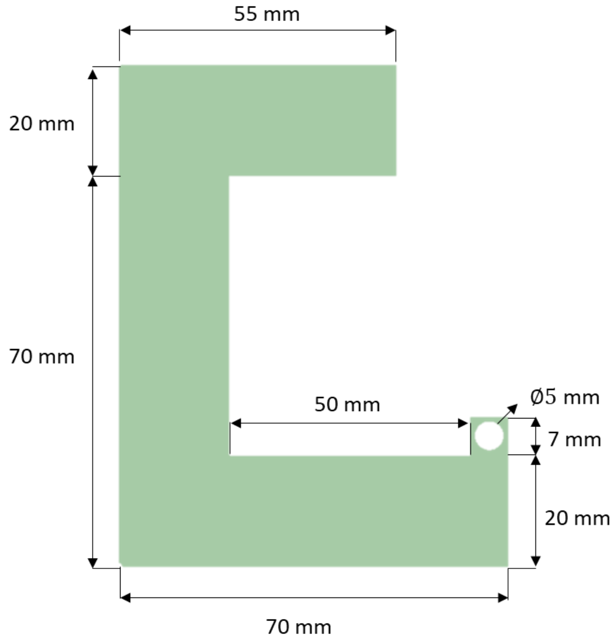


Figure 4.2 - Representation of the assumed geometry for the case study as well as the main measures.

In Figure 4.3 it is possible to observe a representation of the forces assigned to the case study. The area in blue represents the zone that will be fixed, while the hole represented in yellow will withstand a force of 300N.

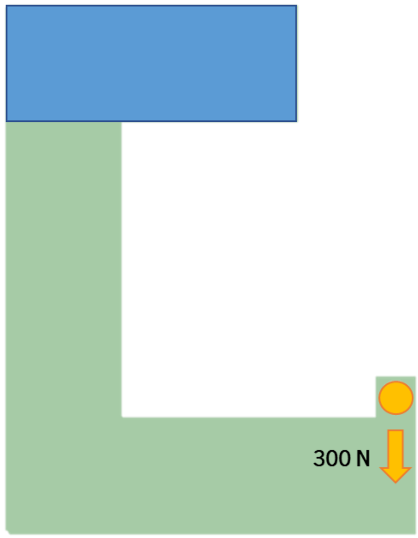


Figure 4.3 - Representation of loads involved in the case study.

At this momento two components still need to be analysed to start the static analysis of the component: its thickness and the material. Regarding the thickness, this will be analysed in more detail later on, however, the value of 5mm was assumed. The component material will depend on the availability of the suppliers, however, to start the study, an aluminium alloy with the properties shown in Table 4.1 was assumed.

Table 4.1 - Characteristics of the material selected to start static simulations of the component.

Material Properties	
Name	Aluminium alloy, wrought, 6061, T6
Density (g/cm^3)	2,713
Tensile Yield Strength (MPa)	259,2
Tensile Ultimate Strength (MPa)	313,1

After defining all the properties, it was possible to start the static analysis of the component using the Ansys Workbench 2020R2 software. The results, shown in Figure 4.4, demonstrate that the component, with a mass of 62.8g, has a maximum von-Mises stress value of 144 MPa. Knowing that only the relationship between mass and maximum stress value will be analysed, the results obtained mark the reference values, 62.8g and 144 MPa.

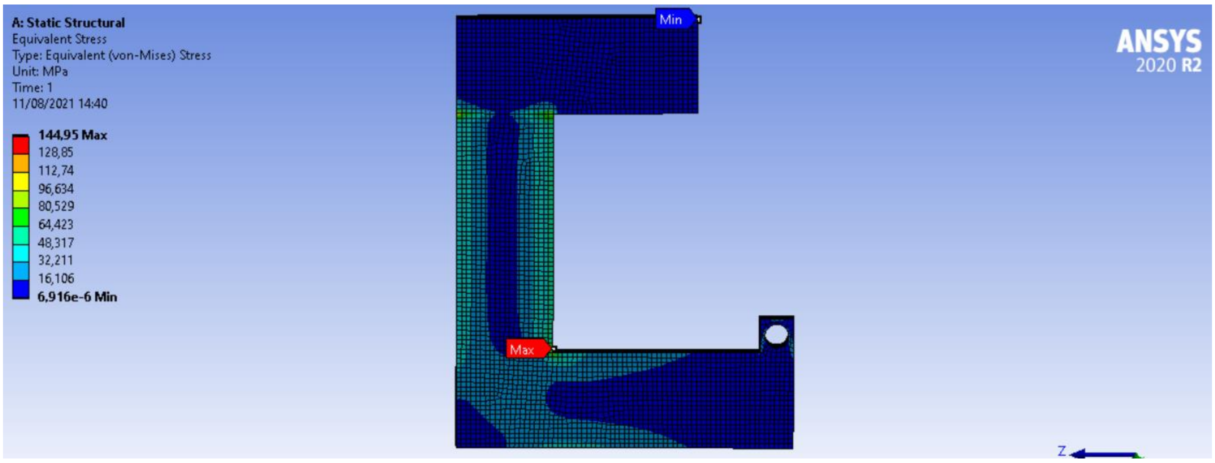


Figure 4.4 - Results of static analysis performed on the component.

4.3 Topology Optimization

With the previous steps completed, it was possible to start the Topology Optimization process. The objective was to reduce as much as possible the mass of the component keeping the maximum stress below 160 MPa. For this, the SIMP algorithm (Solid Isotropic Material with Penalization) presented in chapter 3.4 was used, and the geometry presented in Figure 4.5 was reached. According to the proposed design strategy, it could be necessary to use several optimization stages, however as this case study was quite simple, it was possible to obtain good results through direct optimization (using only one step).

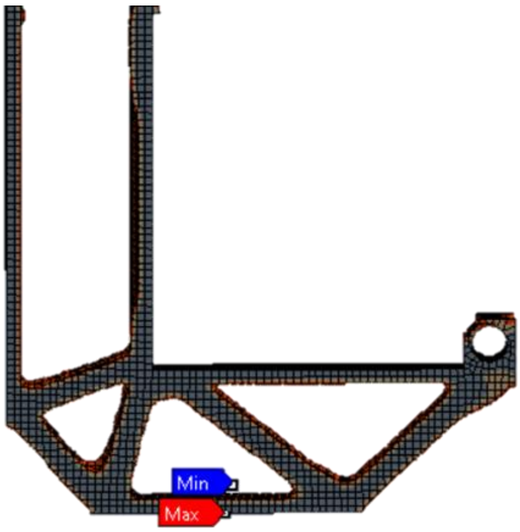


Figure 4.5 - Geometry obtained after Topology Optimization operation. Mass: 15,7g

With this optimization it was possible to achieve a geometry around 75% lighter while maintaining the same response to the imposed stress conditions. To be able to understand in more detail the differences between the optimized geometry and the original geometry, it is possible to observe Figure

4.6 where the elements in red were considered critical by the SIMP algorithm and the elements in blue were considered negligible.

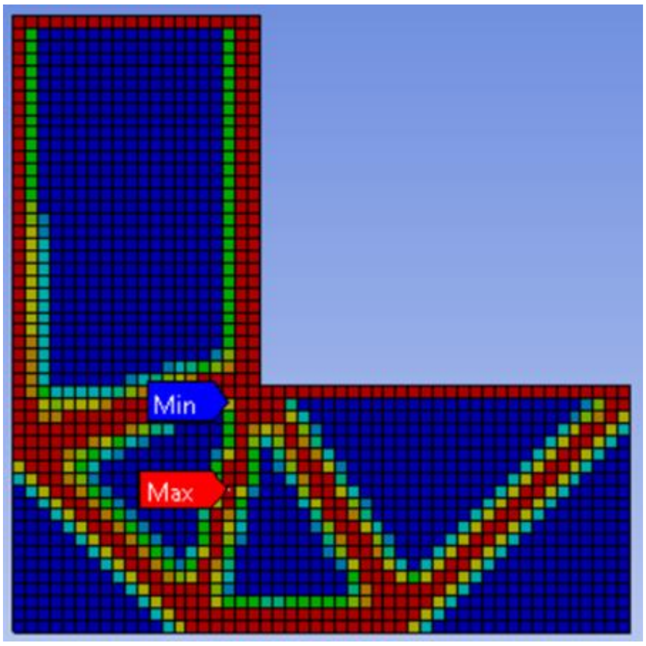


Figure 4.6 - Representation of elements considered fundamental after the Topology Optimization operation in comparison with the elements of the original geometry. ETOPO analysis.

After such interesting results, a redesign process was carried out to transform the present geometry, obtained through a *stl file*, into a geometry performed through a CAD software to be able to carry out future analyses, such as static analysis or optimizations through genetic algorithms. It is at this stage that fundamental details for the success of this dissertation emerge. Although the geometries obtained through the Topology Optimization processes can be extremely interesting, sometimes it is impossible to replicate the same geometry using CAD software. This happens because the geometry of the STL file has small cut-outs and imperfections associated with the elements that make it impossible to obtain geometries through CAD software with the exact shape or mass. To try to minimize this problem and to be able to obtain a geometry as close to the *stl file* as possible, there are two CAD alternatives present in Ansys Workbench, the DesignModeler tool and the SpaceClaim tool. These CAD tools although similar can be critical to the success of the proposed new design strategy. While SpaceClaim, the most modern CAD tool recommended by Ansys Workbench, proved to be the most effective in the faithful representation of the stl file, DesignModeler proved to be the most effective in selecting parameters, which are fundamental for the use of genetic algorithms.

Figure 4.7 shows the geometries obtained using SpaceClaim and DesignModeler, respectively. The drastic differences between the obtained geometries are notorious, however, the following strategy was chosen: for the present case study where the main objective is to know the characteristics and the limitations of the new proposed design process, the DesignModeler was used to facilitate the integration of genetic algorithms in the process of design while in the final case study, presented in the next chapter, SpaceClaim was used to obtain the most reliable geometry possible.

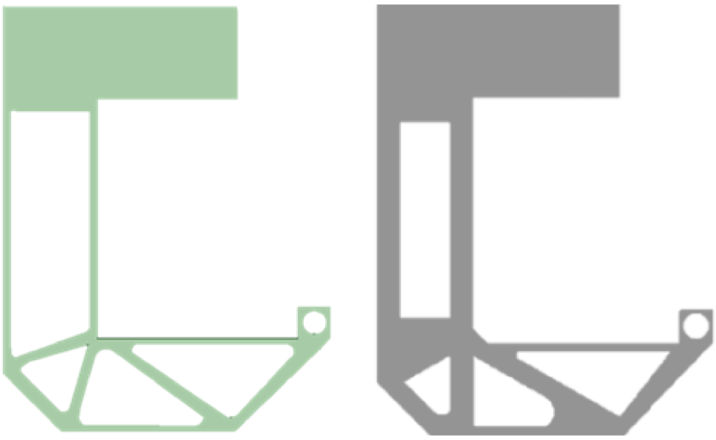


Figure 4.7 - Geometry obtained after a redesign process inspired by the results obtained in the Topology Optimization operation. At the left using SpaceClaim as the CAD tool and at the right using DesignModeler.

Although the study proceeded with the geometry generated by DesignModeler because of the facility to incorporate the Genetic Algorithms, it was decided to carry out a static analysis of the two components to analyse the existing difference. This analysis can be seen in Figure 4.8 and the difference in the stress results can be explained by the increase of mass in the geometry of the DesignModeler (31.4g) when compared to the mass of the geometry obtained by SpaceClaim (22.0g). Furthermore, it is possible to observe that the result obtained in SpaceClaim exceeds the 160 MPa defined as the limit. This can be understood as a failed Topology Optimization operation since it was not possible to fulfil one of the main requirements, however, if one compares the geometry obtained in the stl file, Figure 4.5, and the geometry achieved in SpaceClaim, Figure 4.7, it is possible to conclude that the user considered that it was still possible to reduce the mass of the component. As can be seen in this case, it was preferable to have taken a more conservative attitude (keeping the stress below 160 MPa), however, it was decided to keep this example as a warning for the readers.

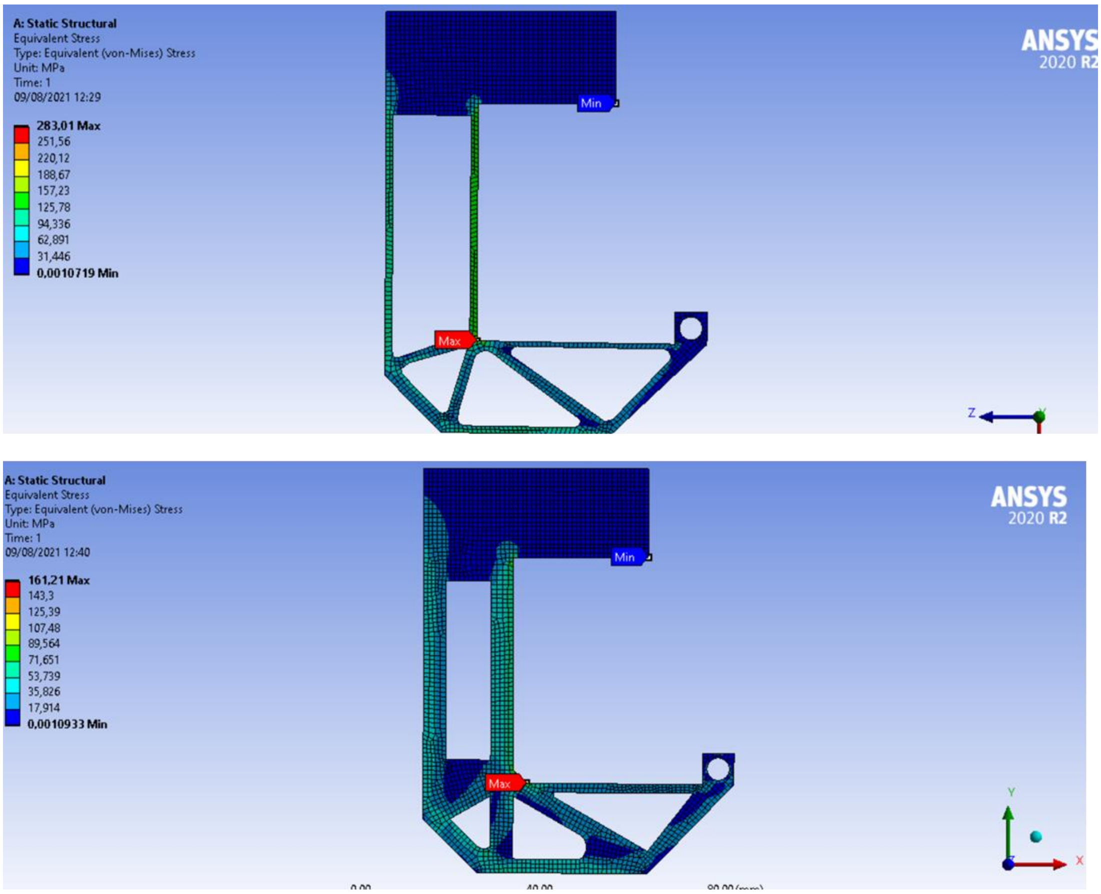


Figure 4.8 - Static Analysis referring to the geometry obtained from SpaceClaim (top) and the geometry obtained from DesignModeler (bottom).

4.4 Genetic Algorithm (MOGA)

The starting point for the analysis by genetic algorithms was the DesignModeler geometry, with a maximum stress of 161MPa and a mass of 31.4g, and the defined objective for this analysis was to decrease the component mass without increase in an uncontrolled way the maximum stress. However, to start the optimization by genetic algorithms and since the objective of this initial case study was to know the software and its main characteristics, it was necessary to define the critical steps for the use of genetic algorithms. For this, it is possible to consult in Figure 4.9 a scheme taken from the Ansys Workbench defining all the tabs necessary to use the genetic algorithms, highlighting the following tabs "Parameter Set", "Design of Experiments", "Response Surface" and “Optimization”.

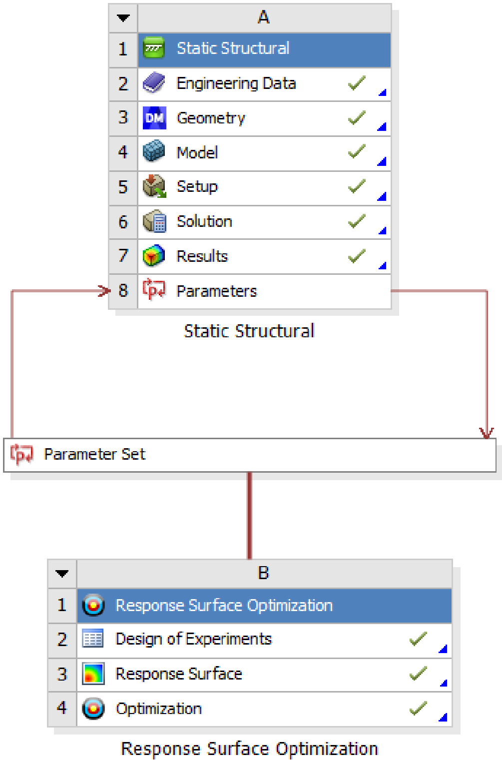


Figure 4.9 - Complete diagram taken from Ansys Workbench of the steps necessary to perform optimization by genetic algorithms.

Ansys Workbench is capable of incorporating the results obtained previously in the static analysis and start the development through the genetic algorithms from these data. From this point onwards, it becomes necessary to identify the parameters to be optimized, a subject analysed in the next subchapter.

4.4.1 Setting parameters for MOGA optimization

The identification of the parameters to be optimized is key to a correct use of the genetic algorithms. These parameters, defined in DesignModeler, have to fulfil several requirements as it is only possible to define parameters such as distances between parallel lines or the radius of a curve. This limitation was one of the main reasons that the geometry obtained in DesignModeler became so distinct from the SpaceClaim geometry. In total, 6 input parameters were defined to be optimized, shown in Figure 4.10, highlighting mainly the thickness, since until now a thickness of 5mm has been considered for all analyses.

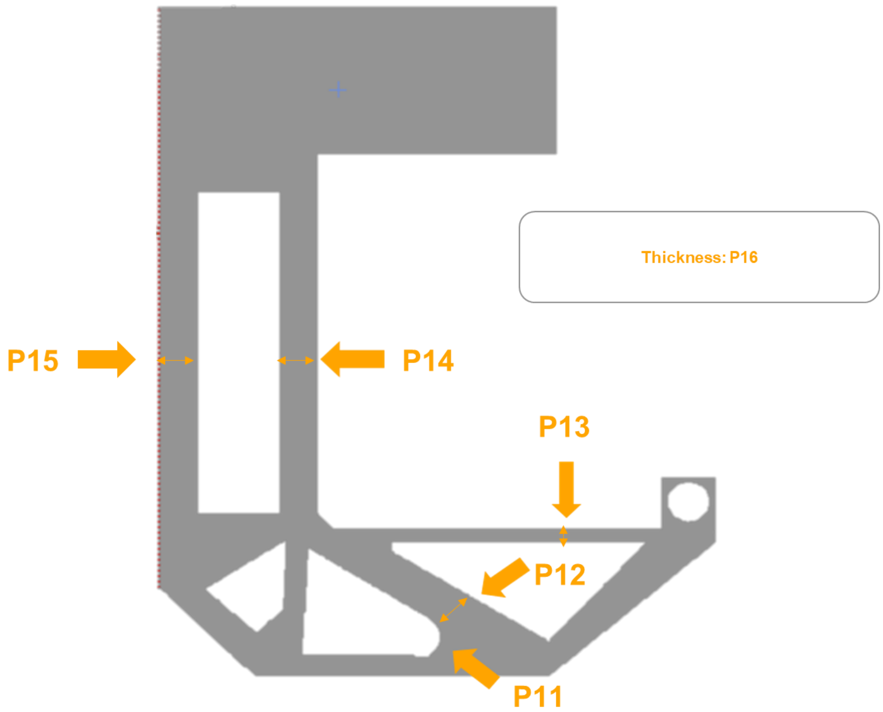


Figure 4.10 - Representation of parameters defined as viable to be used by genetic algorithms.

4.4.2 MOGA Optimization and final results

After completing the selection of parameters to analyse, it is necessary to proceed to the "Design of Experiments" tab. Here, different samples of the problem will be simulated depending on the variation range defined for each parameter. For example, in Figure 4.11 it is possible to observe 28 iterations so that it is possible to obtain standard results that can guide the genetic algorithms. In theory, this phase would be the creation of the initial population, however, in the case of Ansys Workbench, it may be inaccurate to assign this name since this step is not exclusive to genetic algorithms. The key factor in this step is the correct definition of the maximum and minimum values that each parameter can vary since if very large intervals were defined it could conflict with other geometric surfaces and prevent the continuation of the study. If the intervals are too small then the algorithms will have little freedom to be able to work, ending up achieving results that are not relevant. This step could lead to high computation time.

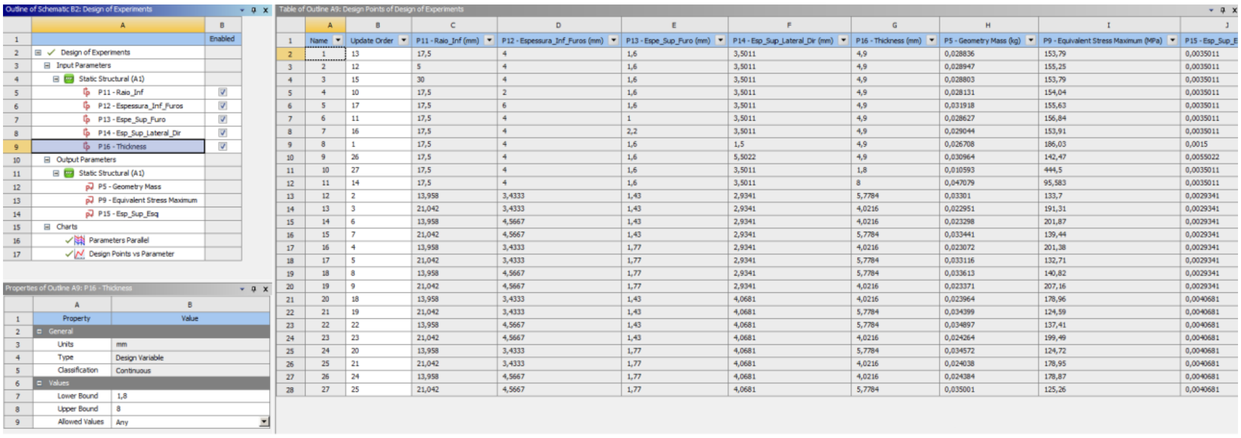


Figure 4.11 – "Design of Experiments" using the defined input and output parameters.

After completing the "Design of Experiments" it is posssible to proceed to the "Response Surface" tab. This step is interesting for the engineer to be able to analyse the influence that the variation of each parameter can have on the final results. An example of one of the different types of analysis can be seen in Figure 4.12 where the influence that each parameter will have on the component mass variation (the inner circle) and the maximum stress (the outer circle) is verified. It is easy to conclude that the variation in thickness should be given more attention, following parameters P14 and P15. (Grebenişan & Salem, 2017)

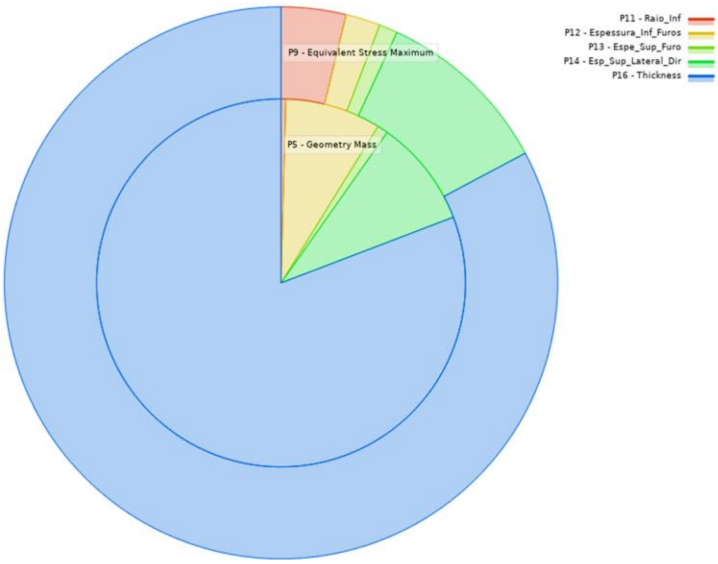


Figure 4.12 – "Response Surface". Pie chart analysing the influence of input parameters on output parameters.

Finally, it is possible to reach the "Optimization" tab. It is in this tab that MOGA, the genetic algorithm present in the Ansys Workbench, will be used, based on the results obtained in the previous steps. Here, the most important thing is to correctly define the optimization objective, having defined that all input parameters could be analysed, however the maximum stress value could not exceed 130 MPa and the parameter relative to the component mass would have to be minimized. In addition, factors related to the use of the genetic algorithm could be defined, such as the maximum number of iterations, convergence criteria among the other factors present in Figure 4.13. In the present case study, due to the genetic algorithms component being a complement to the dissertation and not the main objective, it was decided to use the factors suggested by Ansys Workbench.

Properties of Outline A2: Optimization

	A	B
1	Property	Value
2	⊟ Design Points	
3	Preserve Design Points After DX Run	☐
4	⊟ Failed Design Points Management	
5	Number of Retries	0
6	⊟ Optimization	
7	Method Selection	Auto
8	Estimated Number of Evaluations	24000
9	Method Name	MOGA
10	Tolerance Settings	☑
11	Verify Candidate Points	☐
12	Number of Initial Samples	5000
13	Number of Samples Per Iteration	1000
14	Maximum Allowable Pareto Percentage	70
15	Convergence Stability Percentage	2
16	Maximum Number of Iterations	20
17	Maximum Number of Candidates	3
18	⊟ Optimization Status	
19	Converged	Yes
20	Pareto Percentage	0,1
21	Stability Percentage	1,6553
22	Number of Iterations	11
23	Number of Evaluations	14490
24	Number of Failures	0
25	Size of Generated Sample Set	1000
26	Number of Candidates	3
27	⊟ Design Point Report	
28	Report Image	None

Figure 4.13 - Selected details for optimization by genetic algorithms, using the MOGA algorithm.

In this way, it was possible to obtain three candidate points for the best possible combination of parameters for the defined objectives. These candidate points are present in Figure 4.14 and reveal the accuracy that genetic algorithms can bring to the proposed design process. In this case, it is possible to observe different combinations wherein a candidate point it was privileged to maintain the thickness of the component and vary as much as possible the other parameters (candidate point 1 and 2) and in other candidate points the opposite (candidate point 3).

	A	B	C	D
Table of Schematic B4: Optimization				
1	⊟ Optimization Study			
2	Minimize P5	Goal, Minimize P5 (Default importance)		
3	P9 <= 130 MPa	Strict Constraint, P9 values less than or equals to 130 MPa (Default importance)		
4	⊟ Optimization Method			
5	MOGA	The MOGA method (Multi-Objective Genetic Algorithm) is a variant of the popular NSGA-II (Non-dominated Sorted Genetic Algorithm-II) based on controlled elitism concepts. It supports multiple objectives and constraints and aims at finding the global optimum.		
6	Configuration	Generate 5000 samples initially, 1000 samples per iteration and find 3 candidates in a maximum of 20 iterations.		
7	Status	Converged after 14490 evaluations.		
8	⊟ Candidate Points			
9		Candidate Point 1	Candidate Point 2	Candidate Point 3
10	P11 - Raio_Inf (mm)	10,968	13,798	23,637
11	P12 - Espessura_Inf_Furos (mm)	4,3305	4,6586	2,3636
12	P13 - Espe_Sup_Furo (mm)	1,9752	1,8114	1,2145
13	P14 - Esp_Sup_Lateral_Dir (mm)	4,2035	4,4839	3,1601
14	P16 - Thickness (mm)	4,7728	4,7667	5,2239
15	P5 - Geometry Mass (kg)	0,029071	0,029398	0,029432
16	P9 - Equivalent Stress Maximum (MPa)	126,84	128,71	129,78
17	P15 - Esp_Sup_Esq (m)	0,0042035	0,0044839	0,0031601

Figure 4.14 - Summary of results obtained through optimization by genetic algorithms.

Of the three candidate points, candidate point 1 was selected and the geometry was changed with the new parameter values. In Figure 4.15 it is possible to observe the results of the static analysis and in Table 4.2 it is possible to observe a brief comparison between the parameter values before the optimization through MOGA and after. A detail that is worth highlighting is the difference between the stress value presented in candidate point 1 and that obtained in the static analysis since these should have coincided but were not. This aspect is frequent and usually happens when the zone of higher stress concentration is outside the analysed parameters. This could become a problem, however so far there have been no drastic variations in the recorded stress value.

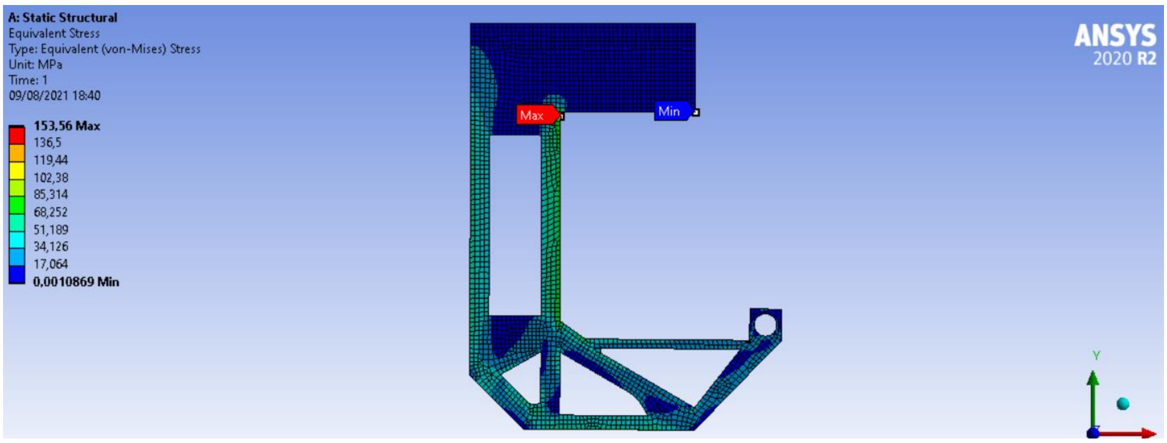


Figure 4.15 - Component static analysis based on changes suggested by the genetic algorithm (candidate point 1).

Table 4.2 - Analysis of the impact of using genetic algorithms.

Parameters	Before MOGA (mm)	After MOGA (mm)	Difference
P11	3.29	10.96	+ 333 %
P12	4.33	4.33	0.00 %
P13	2.00	1.97	- 1.50 %
P14	5.00	4.20	- 16.00 %
P15	5.00	4.20	- 16.0 %
P16	5.00	4.77	- 4.60 %
Mass (g)	31,4	29,1	**- 7.70 %**
Max von-Mises Stress (MPa)	161	153	**- 5.0 %**

4.5 Experimental Activity

The possibility of carrying out an experimental activity on the present case study was the culmination of a complete analysis of the impact of optimization operations, whether by Topology Optimization or by Genetic Algorithms, could have. Unfortunately, despite all stages of the experimental activity having been planned, the company responsible for producing the prototypes to be used in the experimental activity was not able to produce them under the period defined for the dissertation. Nevertheless, it was decided to present a summary of the main stages of the experimental activity on the present case study.

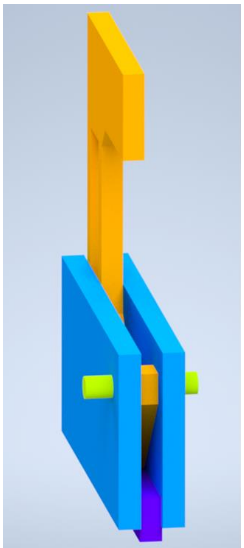
The main objective of the experimental activity was to compare the results experimentally obtained in the original geometry, in the geometry subject to Topology Optimization processes and in the geometry subject to optimization by genetic algorithms, with the numerical simulations obtained in Ansys Workbench 2020 R2. In this way some conclusions could be drawn: Did the optimized prototype manage to support the same loads as the initial geometry prototype? Can the results be compared with numerical simulations? Was it possible to experimentally prove the advantage of adding the optimization operation by genetic algorithms? To be able to answer the proposed questions, it is necessary to present some critical factors of the experimental activity, such as the type of testing, the equipment to be used, the production of prototypes, the accessories designed to be able to replicate the problem loads, among others (Ma et al., 2006). These factors will be presented throughout this chapter.

4.5.1 How to replicate the case study loads?

To be able to replicate the loads of the present case study, it was necessary to identify the type of appropriate laboratory equipment and accessories needed to simulate the situation of the case study. For this, the Material Testing Laboratory of the University of Minho was used and uniaxial traction equipment, present in the Figure 4.16, was identified as the appropriate one for the case study.

Figure 4.16 - Uniaxial tensile testing equipment. Although this photo is not exactly the model present in the University of Minho's laboratories, it is quite similar.

The next step became the idealization of a set of accessories that would allow replicating exactly the proposed loads. In Figure 4.17 it is possible to observe an Assembly with the prototype and the complete accessories for the experiment.



Colour	Name
	Support sheet (5mm)
	Pin (diam=5mm)
	Spacer (6mm)
	Case Study Model (1.1 and 1.2)

Figure 4.17 - Assembly performed using Autodesk Inventor 2021 software in order to represent the accessories needed to carry out the experimental activity in the uniaxial tensile testing equipment.

If we look at Figure 4.17, it is possible to observe two accessories represented in blue and named "Support sheet". These accessories would be responsible for coming into contact with the equipment's mooring area but would be prevented from coming into contact with the component to be analysed since there would be another accessory called "spacer" which would have a thickness greater than that of the component and therefore would be responsible to avoid crushing it. Then, after ensuring that the component is free, the accessory called "pin" would be responsible for applying the force to the component hole indirectly, since a displacement would be imposed on the "support sheets" and as the pin would be the only accessory in contact with the component, the applied displacement would be transmitted to the component. In the end, as the component is fixed at the top, the result of the experimental activity would be similar to the conditions imposed in the study process, and a Force-Displacement graph can be obtained for each component.

4.5.2 How to compare numerical results with experimental ones?

The ideal factor to compare the numerical results with the experimental results was to quantify the stress value experimentally caused in the geometries and compare it with the value obtained numerically, however, to be possible it was necessary several equipment and accessories that were not available. Instead, it was assumed that a good comparison would be through the force-displacement graph obtained through the tensile test equipment and compared with the line obtained in the Ansys Workbench software. In this way, it was expected that the component would not fail up to the numerically stipulated values and, therefore, the comparison between curves would be accessible. In Figure 4.18 it is possible to observe the curve obtained in the Ansys Workbench 2020 R2 for a load of 300N.

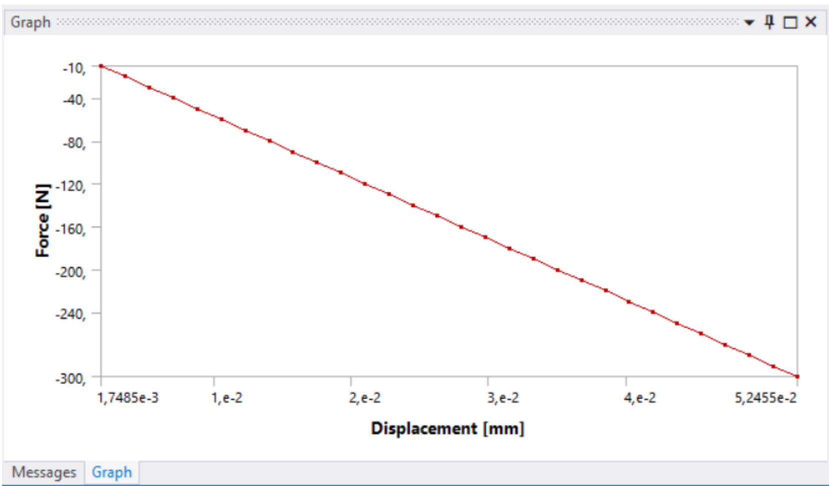


Figure 4.18 - Force-displacement graph obtained in Ansys Workbench 2020R2 software for a load of 300N.

4.6 Conclusions

Knowing that the main objective of this initial case study was to consolidate the steps proposed in the new suggested mechanical design process, Figure 2.3, it is possible to conclude that the developed analysis proved to be fundamental. The difficulty in adopting new strategies sometimes lies in small details, such as limitations in the selection of parameters to be optimized or in the variation ranges for these parameters, which could be understood and resolved with this introductory case study. Furthermore, the selection of the CAD tool, SpaceClaim or DesignModeler, proved to be critical since it is much simpler to replicate the geometry in the Topology Optimization process in SpaceClaim than in DesignModeler. In this case study, it was chosen to use the DesignModeler in an attempt to verify if the genetic algorithms could compensate for the difficulties of replicating the stl file, however it was possible to conclude that for the final case study it is preferable to use only SpaceClaim. On the other hand, the main focus should be on the results obtained through genetic algorithms, since their use allowed a reduction of about 2g, a reduction of almost 8%, to the geometry inspired by the optimization, which highlights the great advantage that this kind of algorithms can bring to Mechanical Designer. It is expected that with the maturation of the proposed design process, the genetic algorithm component may become increasingly simple and effective to use, which could completely revolutionize traditional design processes. Lastly, although all the technical drawings and other production details were carried out, it was not possible to produce these components and therefore the experimental activity was not carried out. It was expected that it would be difficult to obtain exactly equal force-displacement graph curves, however, there would always be the possibility to observe whether the optimized component was able to support the same loads as the original component or not. Nevertheless, all additional information such as technical drawings, production support tables, among others, can be found in Appendix 1 – Initial Case Study (Experimental Activity Details). The execution of the experimental activity may be the subject of future work.

5. Final Case Study – Crossbeam

5.1 Presentation of the case study

After completing the initial case study analysis, a case study where the use of this proposed new design process was tested to consolidate the concept and improve it, it was possible to proceed to a real case study of the automotive industry. This case study aims to develop a Crossbeam for a commercial electric vehicle. This component had already been developed at CES Porto, however this project ended up revealing some difficulties due to the complexity of the problem. Throughout this chapter, the main details of the client requirements for the component and the simplifications made to adjust the problem to this master thesis will be presented.

The Crossbeam is a critical element for the correct functioning of an electric vehicle and although this dissertation is developing the component according to the requirements of an OEM, this component is common to most conventional electric cars. This component is responsible for making the connection between the electric motor and the rest of the vehicle structure, being also submitted to the support of three additional components, namely the Climate Compressor (9kg), CharconPDU - Power Distributing Unit (12kg) and the E-Heater (2kg). In Figure 5.1 it is possible to observe a representation of the electric motor that will be fixed to the component under study. This motor is known as EMR-3, has a mass of 77kg and will be the main reason of concern for the analysis under PotHole and Braking situations.

Figure 5.1 - High voltage axle drive (EMR3). Figure provided by Vitesco Technologies.

Knowing now that Crossbeam is responsible for supporting four components to the vehicle structure, it is necessary to evaluate the limit situations that this component is subject to be able to start the problem analysis. For this, the DOT HS 811 666 report developed by the National Highway Traffic Safety Administration in August 2012 was used. In this extensive report, it is possible to observe all the technical details to consider if you intend to carry out operations to reduce the mass of components, highlighting for this dissertation chapter 5.5 which addresses the main experimental tests that all vehicles must be submitted in order to be approved and considered safe. In total, the report presents 5 experimental activities, namely Fish-hook Test, Double Lane Change Maneuver (ISO 38881), Pothole Test, 0.7G Constant Radius Turn Test and 0.8G Forward Braking Test. For this dissertation and following the client's requirements, only the Pothole Test and the Braking Test were considered, however, to obtain a safety margin and following the procedures of CES Porto, the Pothole Test force was increased from 3.5G to 5G and the Braking Test from 0.7G to 1.5G. Furthermore, it was considered that if there was time available, it would be more interesting to start a small analysis of Crash Optimization instead of the remaining tests presented in the report, since this analysis may have a much greater impact on the final geometry of the component. This analysis would be a bonus to the present work and not an objective, therefore its conclusion is not fundamental.

Another essential aspect for the presentation of the problem lies in the dimensions available and the geometric details necessary to incorporate into the problem. For the execution of this dissertation, it was possible to rely on a preliminary geometry previously obtained by the team at CES Porto, Figure 5.2, however it was decided to start the development of this component completely from the beginning using only general dimensions.

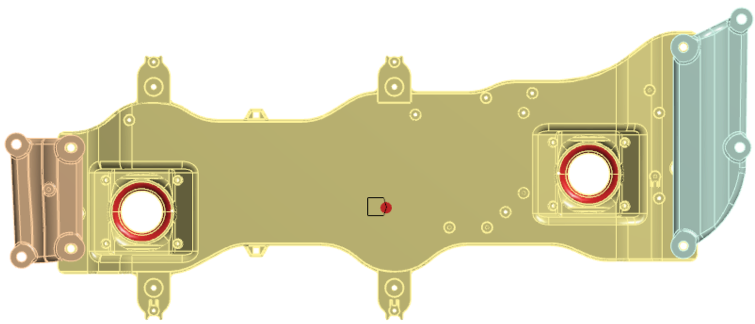


Figure 5.2 - Preliminary geometry obtained by the CES Porto team.

Finally, it is only necessary to identify the main objectives of the Topology Optimization process in the development of this component. Normally, when using Topology Optimization processes, it is essentially intended to reduce the mass of the components while maintaining an acceptable structural rigidity, however, in the present case study this objective is not a priority. Although it may seem obvious that the most important thing in Topology Optimization operations is to reduce mass, in this case the main objective is to get an inspiration of the ideal geometry of the component by reducing the mass of a general geometry only with dimensions of bulk. Thus, after obtaining a geometry with the best possible distribution of the material, it is possible to advance in the chain of the new proposed design process, delivering the results to the Mechanical Designer so that he can develop the geometry of this component faster and with better results, while incorporating the new component of genetic algorithms for this last step.

Briefly, it is possible to summarize the problem in some questions organized in order of importance: Will it be possible to obtain an ideal geometry that can facilitate the design process? Is it possible to strike a balance between mass reduction and ease of production? Will the incorporation of this process prove to be too time-consuming and/or complex? Will it be possible to reduce the component mass considering the Pothole and Braking tests?

5.2 Problem Definition and Static Analysis

5.2.1 Material definition

The definition of the component material can prove to be critical in obtaining the ideal geometry, so its correct definition becomes essential. For this it is necessary to first identify the requirements imposed by the customer, these being the possibility of obtaining the component through a casting process and the preference for the use of an aluminium alloy. Bearing in mind that this case study will serve as a practical example of the application of the proposed new mechanical design process, it was decided that it would be interesting to analyse the impact that the definition of the material could have on the final geometry. For this, the following strategy was adopted. In the first phase, the topology optimization procedure will be applied with a basic aluminium alloy present in the Ansys library. Then, at a later stage, the same procedure will be carried out, but in accordance with the EN AC-44300 aluminium alloy, suggested by the CES Porto team. In the end, the geometries obtained in the different strategies will be compared to draw conclusions about the influence of the material on the ideal Crossbeam geometry. Technical information on the different aluminium alloys used can be found in the Table 5.1 and Table 5.2.

Table 5.1 - Brief description of the aluminium alloy used in the first phase of the project and taken from the materials library incorporated in Ansys Workbench.

Aluminium Alloy (Ansys Library)	
Description	General aluminium alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.
Density	2,77 g/cm^3
Tensile Yield Strength	280 MPa
Tensile Ultimate Strength	310 MPa
Young's Modulus	71 000 MPa

Table 5.2 - Brief description of the aluminium alloy used in the second phase of the project and taken from the CES Porto team.

EN AC - 44300	
Description	Aluminium Alloy for die cast. https://www.manetec-80.de/
Density	2,65 g/cm^3
Tensile Yield Strength	130 MPa
Tensile Ultimate Strength	240 MPa
Young's Modulus	75 000 MPa

5.2.2 Initial Geometry

To be able to start solving the problem, it is necessary to define its limits, which is why it is essential to define the maximum measurements of geometry to be considered in the Topology Optimization process. In addition, small geometric details important to the component must also be incorporated into the initial geometry. Thus, it would be possible to imagine two paths: applying topology optimization operations to the geometry previously obtained by the CES Porto team or starting this procedure with the most basic geometry possible to extract the maximum potential of this type of tool. The second alternative was the chosen one. Throughout this chapter, three distinct initial geometries will be presented, namely Figure 5.3, Figure 5.4 and Figure 5.5. In the case of Figure 5.3, only the maximum measures defined by the customer and the central holes responsible for fixing the Engine and Climate Compressor were considered. In the second figure, in addition to the conditions mentioned above, a detail required by the customer and essential for the assembly of other components on the crossbeam was considered. Finally, for Figure 5.5, four details responsible for fixing components and especially for impact situations also will be considered. The main geometry assumed for the development of the project was the geometry present in the Figure 5.4 since this was the geometry that presents the best balance between the geometric freedom provided to the optimization algorithms and component framing according to the project's requirements. If the reader would like more information about the process of obtaining the maximum measurements for the initial geometry and on the technical drawings of these geometries, please consult Appendix 2 – Final Case Study (Initial Dimensions).

Figure 5.3 - Initial geometry where only the maximum measurements that the component could have been considered. This geometry will be referred to as initial geometry 1.1.

Figure 5.4 - Second version of the initial geometry that will undergo topology optimization processes. In this case, some design limitations were considered, for example, the lower surface of the component was required to be available for fixing other components. This geometry will be referred to as initial geometry 1.2.

Figure 5.5 - This figure represents the third initial geometry assumed in the development of the project. In this case, some details in the component that will only be interesting for Crash Optimization situations were considered. This geometry will be referred to as initial geometry 1.3.

5.2.3 Load cases

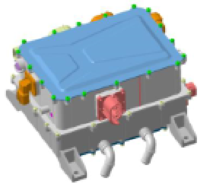
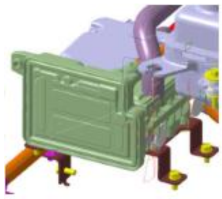
As mentioned in chapter 5.1, the main objective focuses on the component's response to tests in pothole and braking situations. In this chapter, it is expected to present in more detail the loads, as well as the application area, to which the component will be subject. For this, firstly, it is necessary to identify and detail the components involved in the project, namely the Climate Compressor, the CharconPDU (Power Distributing Unit), the E-Heater and the EMR-3 Drive unit.

In the Table 5.3, it is possible to observe the main points to be considered in order to be able to proceed with the analysis of the component. In relation to the place of application of the forces, two scenarios were assumed: in the case of the Climate Compressor and the EMR-3, the fixation zone was assumed as the place subject to the applied loads, while in the case of the PDU and the E-Heater, it was assumed the centre of mass of each component was a guide for the applying loads. This difference is essentially due to the limits of the problem initially defined between the client and the team. On the other hand, the intensity of the loads adjacent to each test is also fundamental to comprehend. As mentioned in the presentation of the problem, chapter 5.1, 5G was assumed as the base value for the Pothole scenario (although the normal is to consider 3.5G) and 1.5G in the Braking scenario. Having these values as a base, the next step was to calculate the resulting forces in each component, through the equation 9:

$$F = m \times G \times 10 \qquad (9)$$

where F represents the net force in N, m the mass of each component, G the acceleration and 10 only as a measure of reduction. The results are also present at Table 5.3.

Table 5.3 - Detailed analysis of components supported by Crossbeam and their possible influence on static analyses.

Name	**Climate Compressor**	**CharconPDU**	**E-Heater**	**EMR-3 Drive unit**
Image				
Load area	Centre holes	-160,5; -135; 395 (mm)	-223,4; 242,6; 380,2 (mm)	Centre holes
Mass (kg)	9	12	2	77
Force (N) Pothole	450	600	100	3 850
Force (N) Braking	135	180	30	1 155

To conclude the analysis of the forces necessary to carry out the static analysis, it is only necessary to refer to the component's fixation zones. This topic could originate an exclusive analysis, however, to be able to get an inspiration through the Topology Optimization process without conditioning it was only assumed that the ends faces would be fixed. The representation of the forces in the simulation software, Ansys Workbench 2020 R2, can be seen in Figure 5.6. It should be noted that the forces existing in the braking test relating to the CharconPDU and E-Heater components were not incorporated as they were considered irrelevant for the static analysis results.

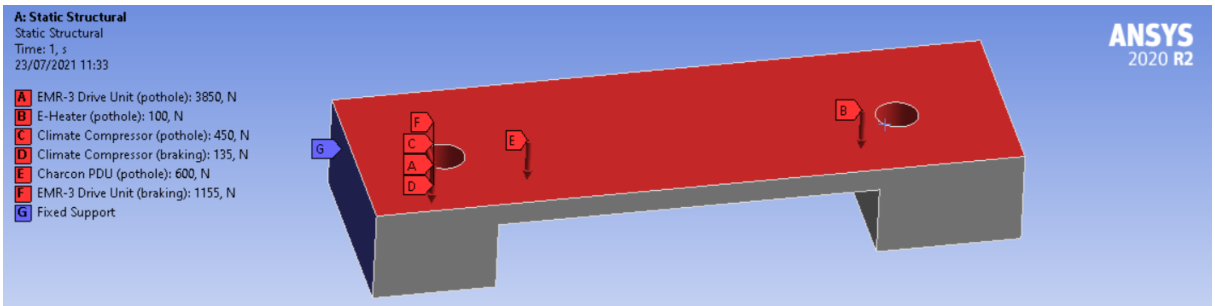


Figure 5.6 - Representation of forces applied in the initial geometry 1.2. Although the software, Ansys Workbench 2020 R2, does not represent the forces on the rightmost hole, these were considered.

5.2.4 Static Analysis

Considering the initial geometries presented in the chapter 5.2.2 and the applied loads in the chapter 5.2.3, it is possible to obtain the results present in Figure 5.7 and Figure 5.8. It should be noted that for these results meshes composed of hexahedrons were used and the material used was EN AC-44300 (highlighted in chapter 5.2.1). With these results, it is possible to define the starting point for the Topology Optimization process since the geometry present in Figure 5.7 has a mass of 75.6kg and the geometry of Figure 5.8 a mass of 41.3kg.

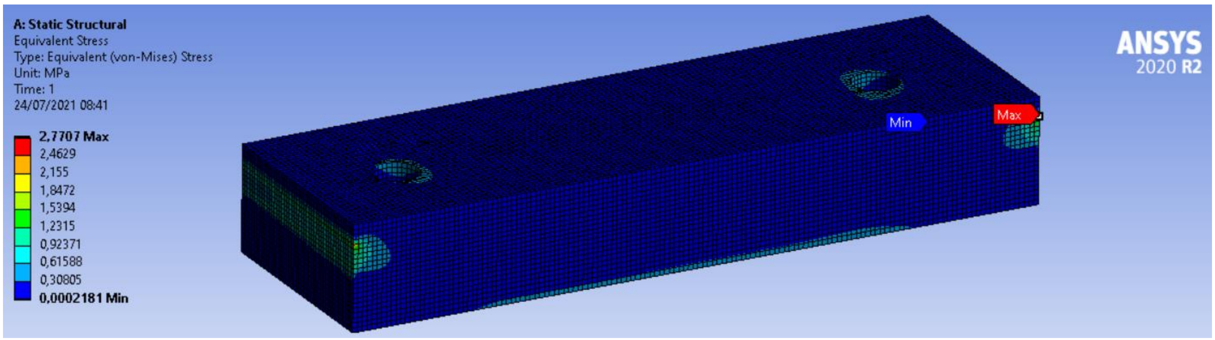


Figure 5.7 - Static analysis corresponding to the loads detailed in the previous chapter on the initial geometry 1.1. Note that for the present simulation a mesh composed of hexahedrons (6mm) and the material EN AC-44300 (75.6 kg) was used.

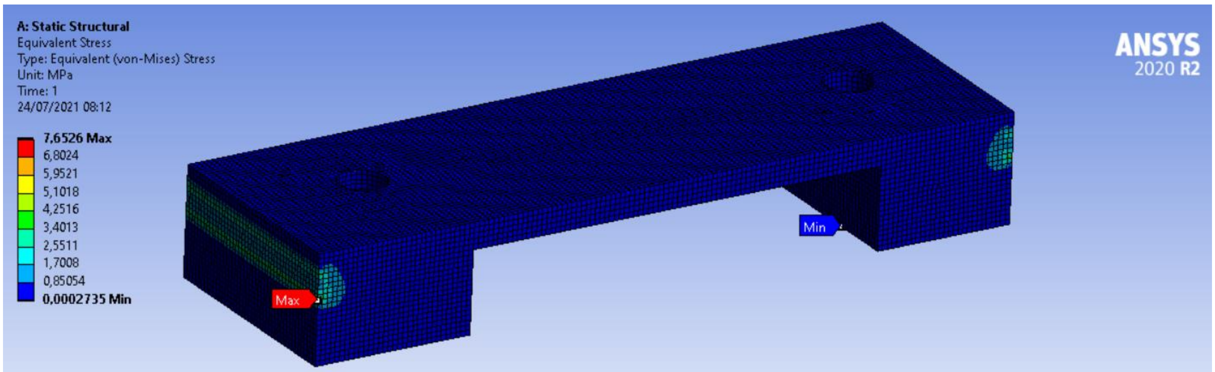


Figure 5.8 - Static analysis corresponding to the loads detailed in the previous chapter on the initial geometry 1.2. Initial geometry 1.3 will not be presented in this chapter because the results are similar to those in this figure.

With the present results, it is possible to verify that SIMP, the algorithm used in the Topology Optimization process, presents a high margin for material reduction until reaching the desired 6/7kg or the 160 MPa required as maximum von-Mises stress.

5.3 Topology Optimization

After the deep analysis of the problem presented, it is possible to start the Topology Optimization process. For this, it is necessary to remember that the optimization algorithm used was the SIMP (Solid Isotropic Material with Penalization) detailed in chapter 3.4 and that the main objective for the project to be developed is not the maximum reduction of material but the possible design inspiration for future developments. The results presented throughout this chapter are chronologically organized so that it is possible to understand the limitations that emerged during the development of the project and the strategies used to reach the final geometry.

The first strategy was using as a starting point the initial geometry represented in Figure 5.3, elementary initial geometry that allows the greatest possible freedom to the optimization algorithm, the optimization process was divided into three stages: optimization only related to the Pothole test, optimization only related to Braking and optimization using both scenarios. The objective in this step was to identify the main geometric changes according to the type of request and verify if these coincided with the team's empirical knowledge. After this analysis, and with the acquired knowledge, the initial geometry 1.2 represented in Figure 5.4 was analysed. At this stage, the constraints imposed was that the geometry must had a final mass in the order of 6/7kg and that it was simple to produce. To be able to compare the results of different stages and scenarios, it was necessary to ensure that the optimization always took place under the same conditions. For this, it becomes necessary to present the conditions used in the Ansys Workbench software. Several parameters can be applied to the Topology Optimization process, namely symmetry relations, Global von-Mises Stress Constraints, Manufacturing Constraints, Displacement Constraints, among others. In Figure 5.9 it is possible to observe the parameters set for all Topology Optimization processes, namely the Analysis Settings, Optimization Region, Objective, Mass Constraint (Response Constraint) and Global von-Mises Stress (Response Constraint 2).

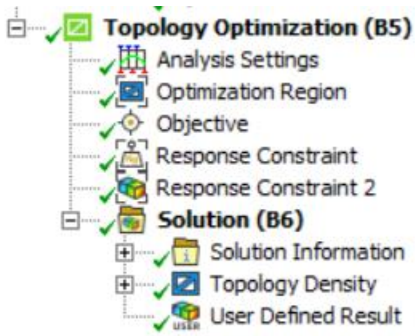


Figure 5.9 - Screenshot of the conditions established in Ansys Workbench 2020R2 for the Topology Optimization process.

The influence of each parameter presented above was exhaustively analysed, however the complete presentation of the results could become too extensive for this dissertation. Thus, throughout this chapter, only the most valued factor of each parameter will be highlighted. Starting with the "Analysis Settings" parameter, the possibility of varying the penalty factor proved to be extremely interesting, however in the end it was used the most commum value, $p = 3$. Another extremely curious parameter was "Optimization Region" since the variation of this parameter can lead to completely different geometries, as will be presented in the next sub-chapter. Continuing the analysis of the optimization parameters, it is possible to observe the "Objective" parameter. This parameter could be varied according to the mathematical criteria presented in the bibliographic review, chapter 2.2, having ended up using only one objective, Minimize Compliance, instaed of the weighting criteria. Finally, there are the "Mass Constraint" and the "Global von-Mises Stress Constraint". These factors were defined as about 15% of the original mass for Initial Geometry 1.1 and 20% 1.2 and 160 MPa as maximum stress value.

5.3.1 Initial Geometry 1.1

For the initial geometry 1.1, it was possible to obtain different results depending on the type of study carried out. In Figure 5.10, it is possible to observe two completely different geometries for the same test with Pothole loads. These results can be explained due to the different "Optimization Region" selected, since in the upper geometry of Figure 5.10 the upper face and the two centre holes were considered as surfaces excluded from optimization and in the lower geometry only the two centre holes were considered as excluded from optimization.

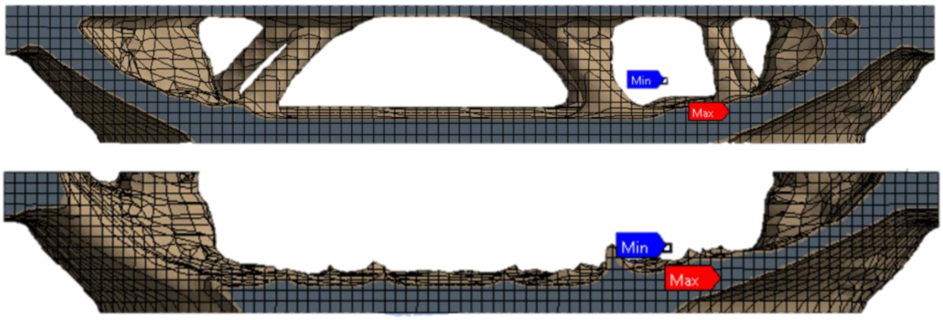


Figure 5.10 - Geometries obtained considering only the Pothole Test loads. In the upper geometry, the top face and the two holes were considered as surfaces excluded from optimization, while in the lower geometry only the two holes were considered.

Continuing the analysis, it is possible to observe in Figure 5.11 the result of the geometry if only loads associated with the Braking Test are considered. In this case, only the two central holes were considered as excluded from optimization.

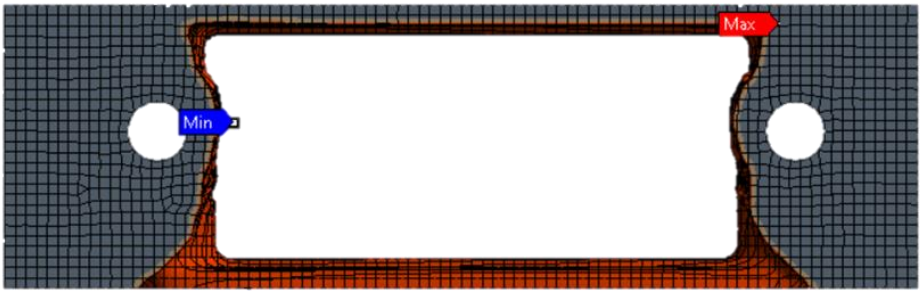


Figure 5.11 - Geometry obtained considering only Braking Test loads. In this case, only the two central holes were considered as an optimization exclusion zone.

Finally, if the loads of the Pothole Test and the Braking Test are considered, it is possible to obtain the geometry of Figure 5.12. As interesting as the geometries may seem, two major problems arise: the difficulty of manufacturing these components and non-compliance with a customer's geometric requirement, the need for the obtained geometry to count the lower surface available for coupling other components. In this way, for many parameters that we could include in the Topology Optimization process, it would be impossible for the algorithm to reach the desired geometries. For this to happen, it is necessary for the engineer to route the obtained geometries through several optimization cycles and the geometry redesign process until the desired results are achieved.

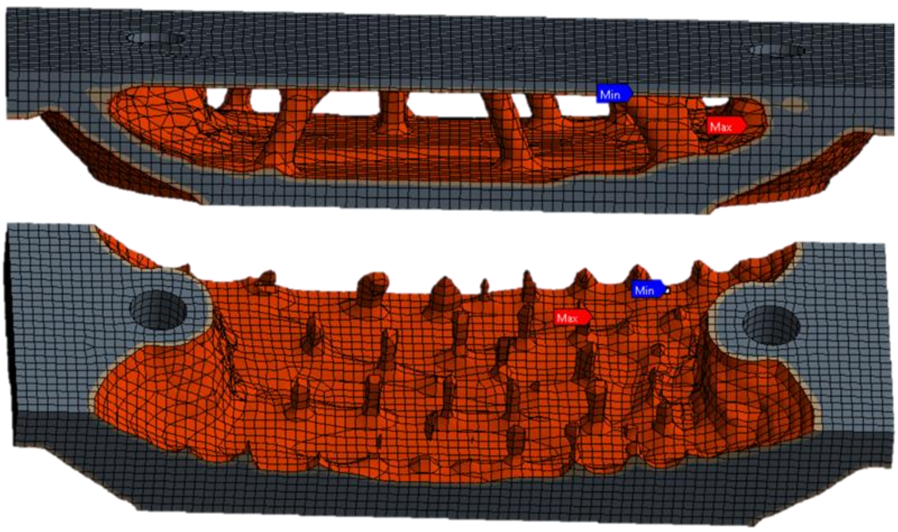


Figure 5.12 - Geometries obtained considering the Pothole and Braking loads. In the upper geometry, the top face and the two holes were considered as surfaces excluded from optimization, while in the lower geometry only the two holes were considered.

The need to include several optimization steps may prove to be a disadvantage of the proposed new mechanical design process, as it makes the process more time-consuming and requires engineers with some experience in this area. However, it was considered worth continuing the process and so the second initial geometry was created.

5.3.2 Initial Geometry 1.2

The initial geometry 1.2, shown in Figure 5.13, was created so that it would be possible to start the second optimization stage. This geometry has a mass of 41.3 kg which will imply an optimization between 80 and 85% in order to obtain a geometry close to the 6/7 kg defined as the ideal objective.

Figure 5.13 – Initial geometry 1.2 with customer-imposed design requirements.

With the development of this geometry optimization, another question arises: Would it be more advantageous to carry out a direct optimization, that is, to go from 41.3kg to 6/7kg in just one phase, or would it be more interesting to carry out several optimization stages intercalary with a redesign process? Certainly, the one-stage optimization will be faster but will the results match the project requirements? A schematic of this issue can be seen in Figure 5.14 and in Figure 5.15.

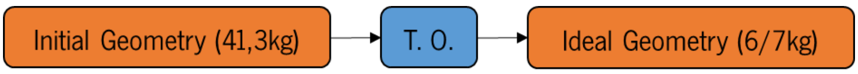


Figure 5.14 - Single-stage optimization scheme (direct optimization).

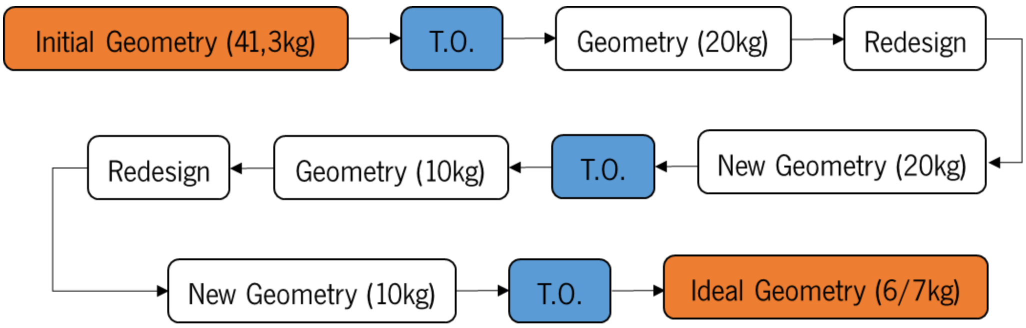


Figure 5.15 - Scheme for multi-stage optimization with the incorporation of intermediate redesign processes.

By following the Topology Optimization process in just one step, it is possible to obtain the geometry shown in Figure 5.16. Although the intended mass has been reached, it is possible to observe a geometry difficult to produce. This happens because the optimization algorithm, SIMP, uses the maximum measurements created in the initial geometry as a reference value leading to features like the outer arms seen in the geometry that do not contribute to structural levels but are mathematical important for the optimization algorithm.

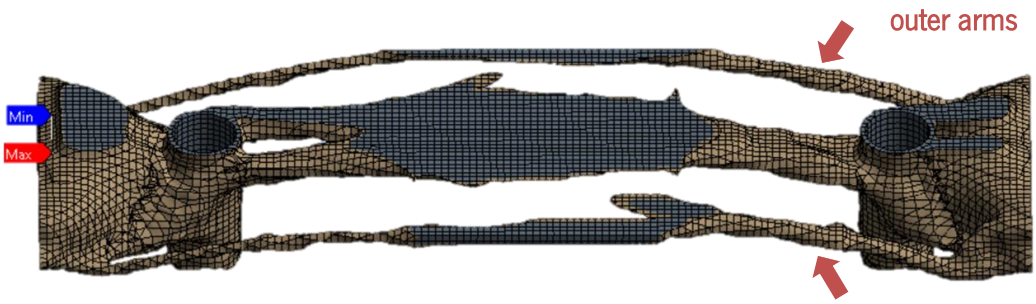


Figure 5.16 - Geometry obtained through direct optimization from 41.3kg to 7kg.

SIMP's tendency to value the mathematical component more than logic is expected and had already been mentioned in the bibliographic analysis. Although this type of problem has not been verified in lower complexity case studies, a way to overcome this difficulty may be in the optimization in several stages.

For te multi-stage strategy proposed in Figure 5.15, three optimization steps were defined. The first step was to apply a Topology Optimization operation to the initial geometry to reduce the geometry's mass from 41.3kg to about half, approximately 20.0kg. Then, an analysis of the obtained geometry would be performed to obtain an inspiration to review a new geometry through the stl file. This new geometry, weighing approximately 20kg, would incorporate small details that could lead the simulations to the expected results. After the completion of the redesign process, a new stage in the optimization process would begin with the objective of reducing the mass of the new geometry, approximately 20kg, to around 10kg. Finally, the last stage of the optimization process would begin with the conclusion of the redesign inspired by the previously obtained geometry and its incorporation into a Topology Optimization process with the objective of reaching 6/7kg.

Figure 5.17 - Geometry obtained after the first cycle of Topology Optimization and based on Initial Geometry 1.2. The geometry represented has a mass of 18.5 kg.

The results of the first optimization cycle can be found in the Figure 5.17 and it is possible to observe the same trend, observed in Figure 5.16 , of the creation of outer arms, however with this geometry is possible to carry out the first comparison with the preliminary Crossbeam created previously by the CES team and present in Figure 5.2. The geometry achieved when compared to the preliminary Crossbeam developed by the CES team makes this strategy quite promising since it is notorious a convergence of details without the user having interfered with the optimization algorithm. These results can be observed in Figure 5.18.

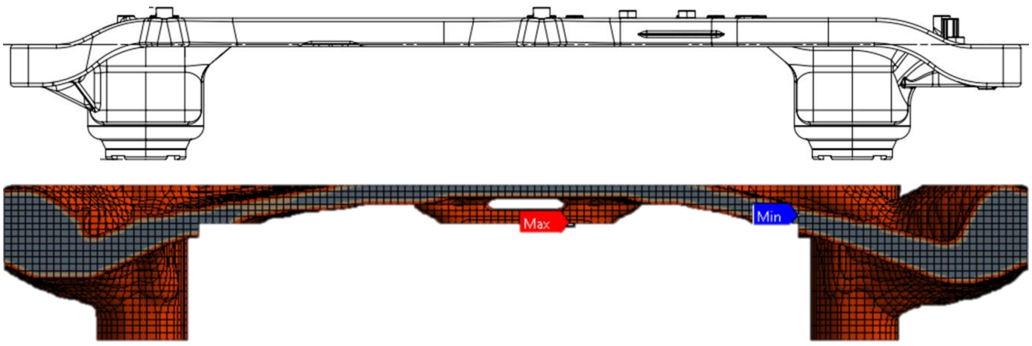


Figure 5.18 - Above is represented a possible final Crossbeam geometry developed by the team at CES and below the geometry obtained with just one optimization cycle.

The results obtained ensured the necessary confidence to move forward with the multi-step optimization strategy. Thus began the redesign process to create a new geometry prepared to be submitted to a new optimization cycle. It should be noted that in this redesign process, attention to geometric detail was not essential, since the main objective was to obtain a geometry that would allow the optimization process to continue quickly.

Figure 5.19 – Geometry obtained at the redesign process and inspired by the stl file obtained through the Topology Optimization operation performed previously.

The use of this strategy presents yet another advantage since to start the next Topology Optimization process it is necessary to perform a static analysis of the new redesigned geometry. This allows a greater understanding of the problem and corroborates details resulting in future geometries. As can be seen in Figure 5.20, the maximum stress demonstrates a high safety margin to proceed with the optimization process. Low-stress levels were to be expected as the geometry still has a mass of 20kg when the goal is to reach 6/7kg.

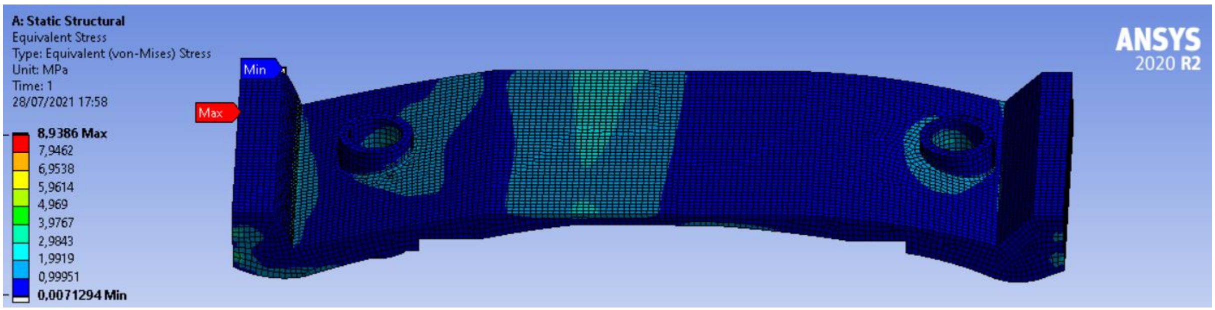


Figure 5.20 - Static analysis performed on the geometry obtained after the redesign process.

Continuing with the second optimization stage and after performing the static analysis, it is possible to proceed to the Topology Optimization operation with the objective of reducing the mass from 20kg to 10kg. The results shown in Figure 5.21 demonstrate that the redesign process was essential to lead the results to the intended objectives. At the moment it is possible to observe a more concrete and detailed geometry, which will facilitate inspiration for the mechanical designer.

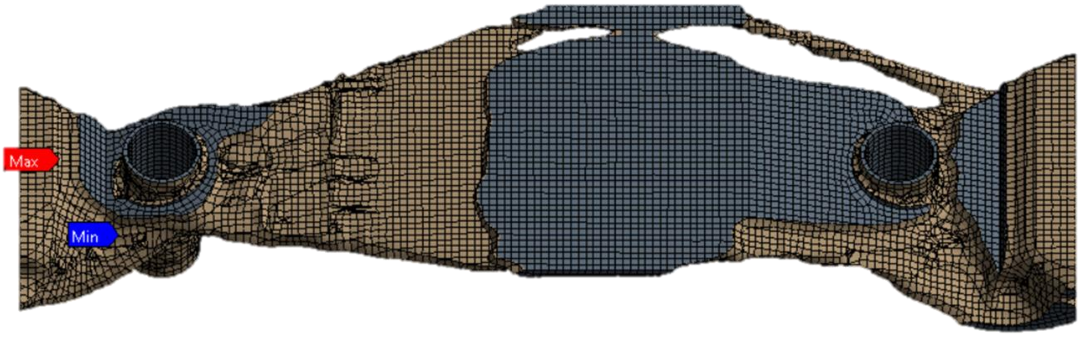


Figure 5.21 - Geometry obtained after the second optimization step with a mass of 10kg.

At this stage, the question can be raised as to whether or not it is advantageous to create another optimization step (from 10kg to 6/7kg) or it is preferable to increase the requirement of the previous Topology Optimization operation, that is, from 20kg to 6/7kg. For this, the geometry obtained through the process of the previous redesign was used and a mass reduction in the order of 65% was stipulated. The results obtained can be seen in Figure 5.22.

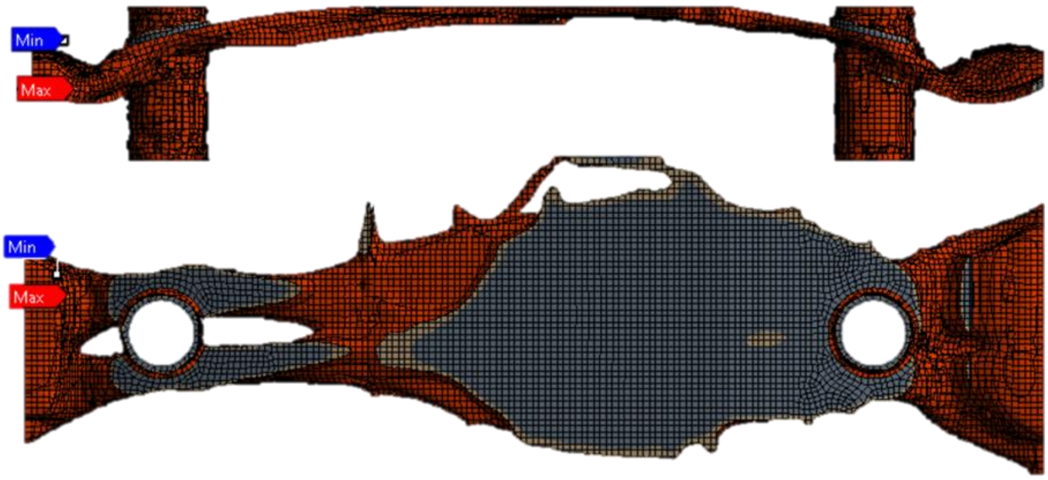


Figure 5.22 - Geometry obtained at the end of the process with several optimization steps. The component mass is still below the mass considered acceptable (5.87kg < 6kg)

With the results revealed to be extremely positive and with some time margin defined for the execution of the dissertation, two possibilities were considered: either to proceed with the optimization of the current geometry, considering only the PotHole Test and the Braking Test, or to carry out initial analysis to the optimization of Crossbeam considering loads related to crash situations. In the end, it was considered more interesting to perform an introductory analysis to crash situations instead of continuing the optimization of Crossbeam, subject only to the two tests mentioned, because it was irrelevant to reach an extremely optimized geometry that does not consider more demanding situations. The results of this introductory analysis can be seen in the chapter 5.5.

Furthermore, as mentioned in chapter 5.2.1, an analysis was carried out on the impact that changing aluminium characteristics could have on the optimized geometry. From this analysis it was observed that the geometry appears to be similar in both situations, however, through empirical knowledge, it is possible to assume that they are not exactly the same due to the critical points. In a short term, although the ideal geometries were similar (and therefore with little relevance for the Topology Optimization analysis), if the component was subjected to fatigue tests, it is natural that the answer may differ significantly depending on the aluminium used because the Tensile Yield Strength of the Aluminium from Ansys is 280 MPa and 130 MPa from the EN AC - 44300.

5.4 Genetic Algorithm (MOGA)

Once the Topology Optimization component defined in the proposed new mechanical design methodology is completed, represented in Figure 2.4, it is time to send the obtained geometries to the Mechanical Designer so that it can get its inspiration to complete the development of the component and consequently start the use of genetic algorithms in this path. In this dissertation, because it is still such an innovative area and because it is a work carried out in an academic context, the use of genetic algorithms was performed by the same person that did the Topology Optimization. In the future, this should be divided between the Simulation Engineer (Topology Optimization) and the Mechanical Designer (Genetic Algorithms).

5.4.1 Ideal geometry obtained after T.O.

As mentioned in chapter 3.5, the purpose of using genetic algorithms for this dissertation favours ease of use and connection with the other stages of the design process in favour of the more advanced genetic algorithm, which is why MOGA was used. In order to proceed with the use of genetic algorithms, it was necessary to start Design Loop 1, referred to in Figure 2.4, to obtain the geometry presented in Figure 5.23.

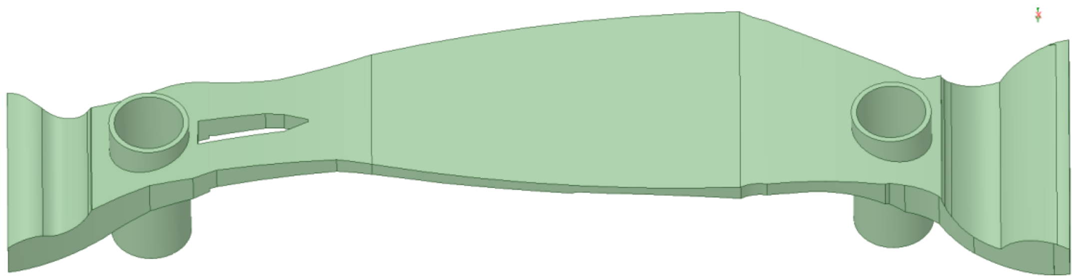


Figure 5.23 - Geometry obtained after inspiration from the Topology Optimization process. This geometry has a mass of 6.31 kg.

Although the geometry was created by inspiration of the final result of the various stages of Topology Optimization, the geometry has a higher mass, around 6.31 kg, than the T.O. one, approximately 6kg. This phenomenon was already expected since the geometry obtained through Topology Optimization operations is composed of a stl file full of small cut-outs and perforations that could not be represented in a conventional geometry. A comparison between the different geometries can be seen in Figure 5.24.

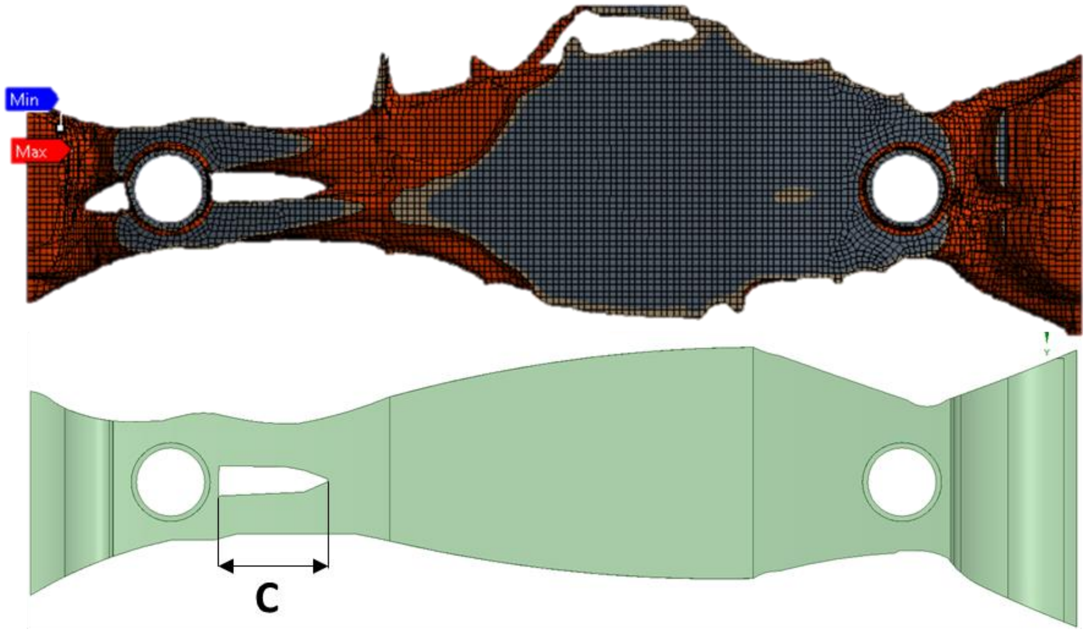


Figure 5.24 - Comparison of the similarities between the geometry obtained through the stl file and the geometry created through inspiration.

5.4.2 Setting parameters for MOGA optimization

After defining the geometry, it is possible to proceed to the next step, the selection of parameters. For this, it is necessary to analyse the geometry and identify areas such as parallel surfaces or curves to be able to define them as the parameter to be changed. This detail can become a major limitation since sometimes the geometries do not have viable areas for the creation of parameters in Ansys Workbench. In the current case of study, the thickness of the Crossbeam fixation zone was analysed. This case is an excellent example to show the advantages of using Genetic Algorithms, since these areas were practically excluded from the Topology Optimization operation, and therefore the ideal thickness for the component is unknown. In addition, the length of the hole introduced in the geometry was considered as the third analysis parameter. These details are represented in Figure 5.25 with the letter A and B and in Figure 5.24 with the letter C.

Figure 5.25 - Identification of the two main parameters to be optimized by genetic algorithms. The letter "A" represents the thickness of the left attachment zone and the letter "B" the thickness of the right attachment zone.

The detail, which can sometimes go unnoticed, considered critical for the success of optimization by genetic algorithms, is the correct definition of the range of variation of the parameter values, that is, for the case of thickness, the minimum thickness that the component could have. This detail is critical because sometimes the variation of the parameter reaches values that interfere with the original geometry, ending up failing the optimization. Although Ansys Workbench automatically defines small ranges of variation, the maximum and minimum values defined are shown in Table 5.4.

Table 5.4 - Definition of the ranges of variation of parameters throughout the optimization.

Parameter	Original value (mm)	Minimum Value (mm)	Maximum Value (mm)
A	29,5	4	30
B	29,5	4	30
C	88,9	85	120

5.4.3 MOGA Optimization and final results

At this point, it is possible to start optimizing the defined parameters. This process encompasses the "Response Surface Optimization" functionality of Ansys Workbench, which is divided into three parts after selecting the parameters. The initial part would be the Design of Experiments, where a first analysis would be carried out by Ansys Workbench of the consequences of the variation of parameters. If this is transposed to the genetic algorithms component this would be equivalent to the generation of the initial population, however, this fact cannot be assumed because the Design of Experiments is universal to different algorithms other than genetic algorithms. This tool allows several interesting analyses, such as the analysis of the influence that the variation of each factor may have on the final result of the geometry. As shown in Figure 5.26, the influence of the variation in the thickness of the Fixation (parameter A and B) is much more relevant for mass reduction and also more critical for maximal stress increase that the parameter C.

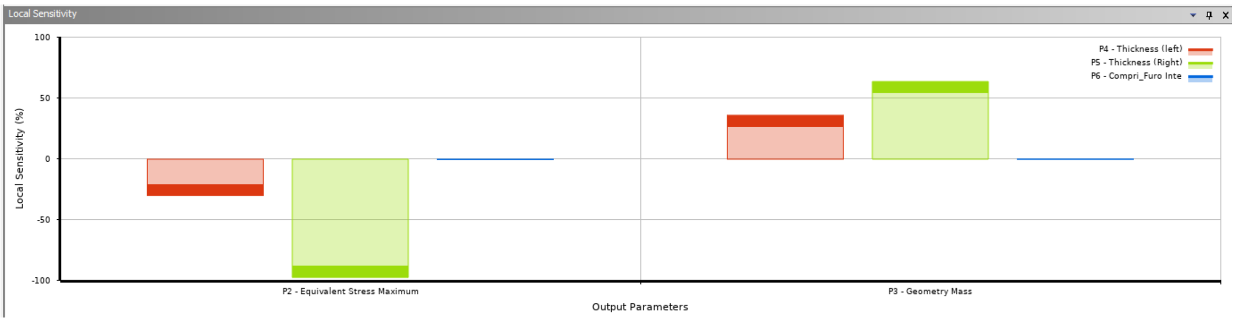


Figure 5.26 - Analysis of the impact of input parameter variation may have on the output parameter (Maximum stress Von-Mises and Mass).

The objective of the analysis of several objectives, where sometimes these are practically indirectly proportional, maybe very difficult. In this case, this balance will be close to that shown in Figure 5.27, where it is possible to achieve the best combination between mass reduction and maximum stress.

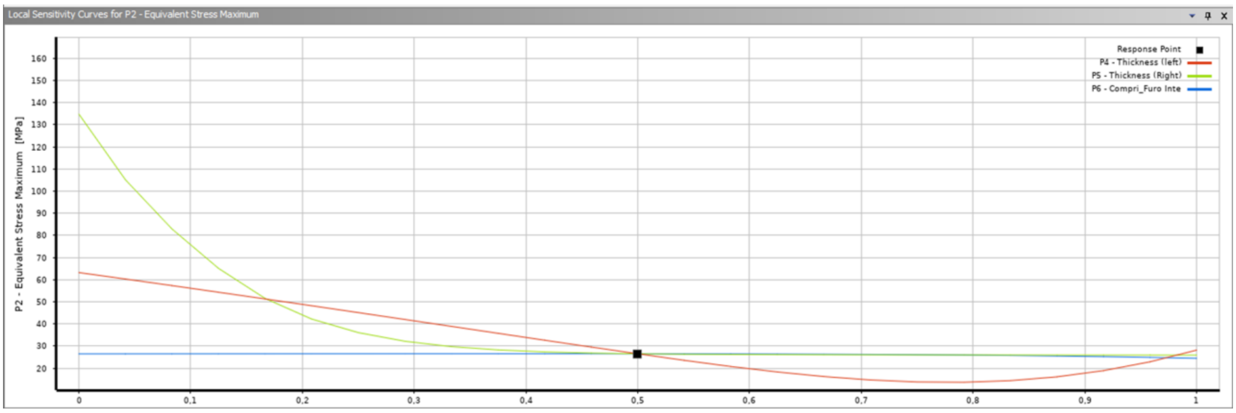


Figure 5.27 - Sensitivity analysis on changing parameters in order to reach the closest point of equilibrium.

Finally, in the Optimization part, it will be possible to use genetic algorithms that will be guided by the steps previously defined. After defining the algorithm to proceed with the optimization, in this case the genetic algorithm MOGA was used, it is necessary to define the objective of the optimization and the constraints. The main objective selected became "Minimize P4 (Mass)" as long as the maximum stress remains below or equal to 150 MPa. There is also the possibility of varying the characteristics of the genetic algorithm such as increasing the initial population, varying the percentage of mutations introduced in the optimization, among others, however as this case study was only intended to validate the new proposed mechanical design process standard Ansys Workbench variables were used. The three points reached by the genetic algorithm after a number of initial samples from 3000, analysing 600 samples for each iteration up to a maximum of 20 iterations, converged after 6609 evaluations can be seen in Table 5.5.

Table 5.5 - Results obtained for the three parameters considered after using the MOGA algorithm.

Parameters	Candidate Point 1	Candidate Point 2	Candidate Point 3	Original
A (mm)	5,0218	5,0513	5,0678	18,756
B (mm)	5,0571	5,0582	5,0544	29,5
C (mm)	77,064	81,585	112,31	88,972
Max stress (MPa)	139,78	139,75	153,7	15,391
Mass (kg)	5,456	5,4564	5,4565	6,1693

With the results obtained, it is possible to observe the clear geometric transformation with the passage of the support zone from 29.5 mm to approximately 5,06 mm, which resulted in a decrease in the total mass of 3.5%. In order to prove the solidity of this optimization, a static analysis present in Figure 5.28 proved the quality of the results. If on the one hand, the results obtained by genetic algorithms are not viable for mass production, on the other hand, they allow the Mechanical Designer to achieve the ideal measurements depending on the restrictions applied throughout development, greatly facilitating the work and improving the component quality.

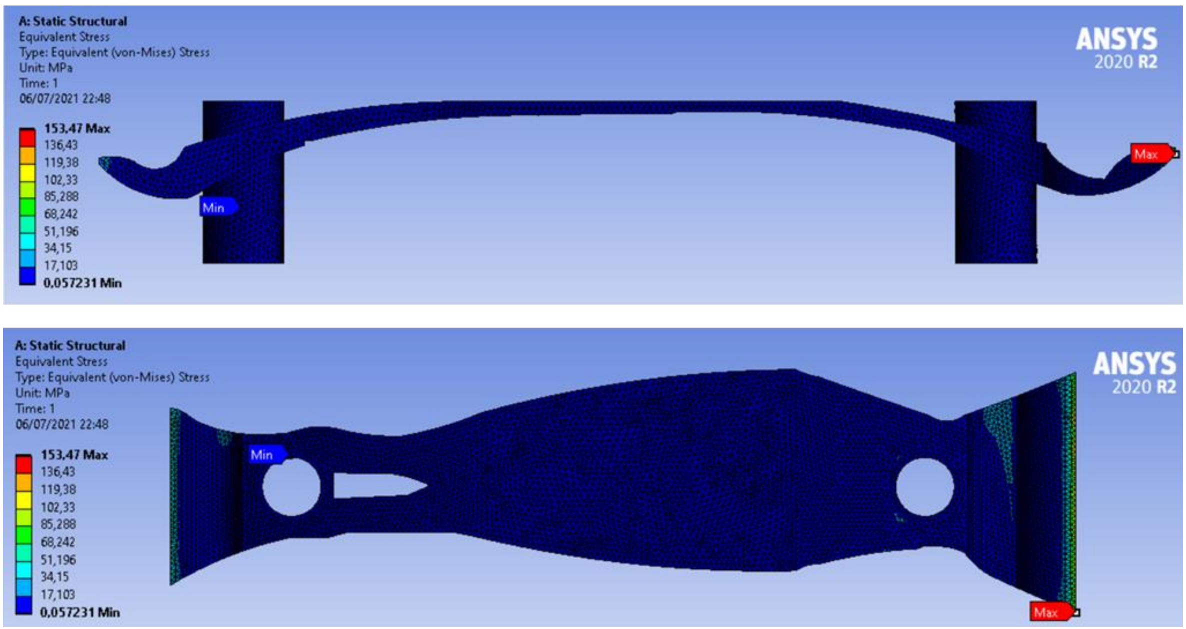


Figure 5.28 - Static analysis based on the values obtained in Candidate Point 1.

5.5 Crash Optimization

As mentioned in chapter 4.5, the main objective of this case study was to analyse the component according to the Pothole test and the Braking Test, however, as the stipulated goals were achieved earlier than expected, a brief analysis of the case component subjected to the loads adjacent to Crash situations was started.

Due to confidentiality issues, it is not possible to fully represent the mechanical requirements imposed by the client, however, it is possible to present a short summary. To submit the component to Crash situations, it is necessary to use the initial geometry 1.3, presented in chapter 5.2.2, since the imported loads will be located in the external support holes. These loads are located in the external holes since, because in case of impact the first geometric detail to come into contact with the other components will be these holes. In addition, each hole is expected to be subjected to two loads, each with an intensity of 10 000N, one being applied to a coordinate axis with a 45° rotation around the *xx axis* and the other to a coordinate axis with a rotation of -45° around the *xx axis*. The final results of all applied loads are shown in Figure 5.29.

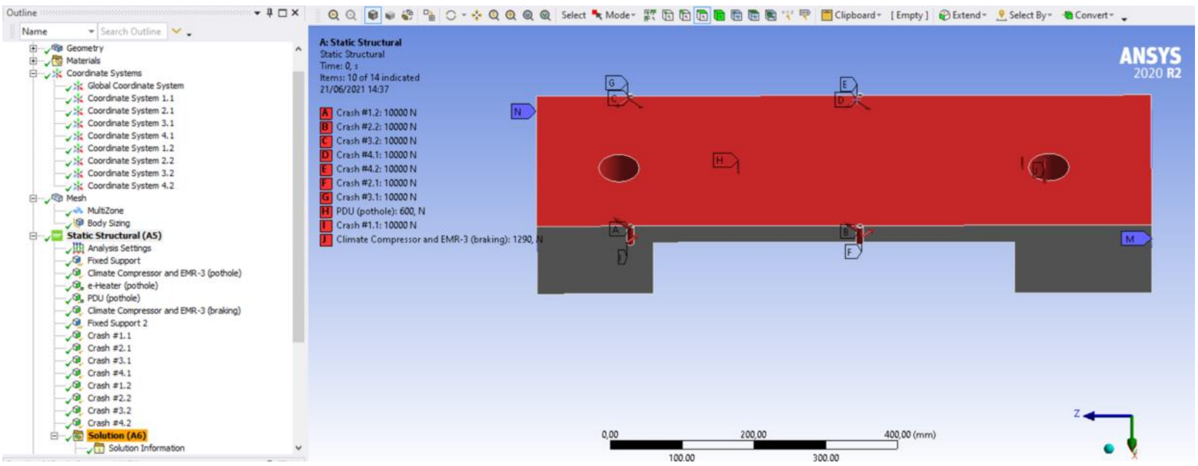


Figure 5.29 - Application of forces related to Crash situations by creating coordinate axes with a 45° rotation of the x axis in relation to the reference, together with the characteristic forces of Pothole Test and Braking Test.

By performing the static analysis including the new requests, it is possible to prove the high impact of Crash forces compared to the previous studies. This fact reinforces the intention of not proceeding with the optimization operations as far as possible considering only Pothole and Braking loads since this geometry would not be able to support the demands of a Crash situation. The results of the static analysis can be seen in Figure 5.30.

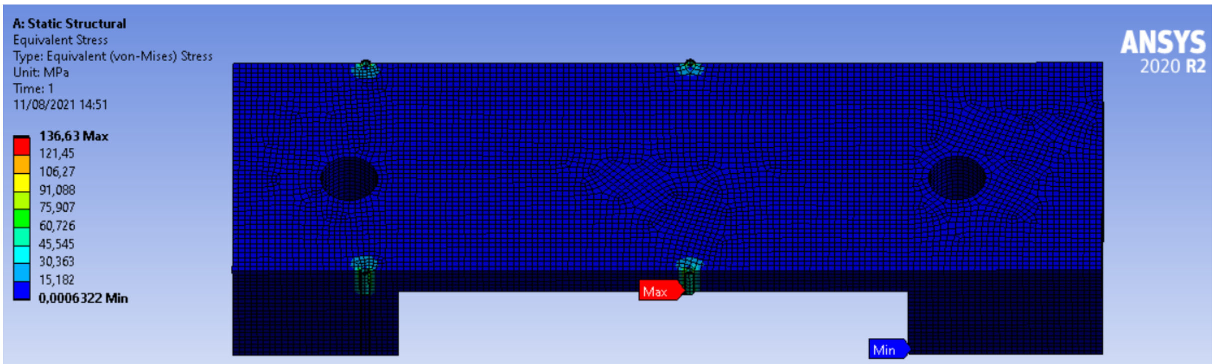


Figure 5.30 -Static analysis of initial geometry 1.3 subjected to Crash, Pothole and Braking.

For this phase, several stages of Topology Optimization were applied with intermediate redesign processes, however, to guarantee the objectivity of this dissertation, only the final results will be presented. These results show a mass of 8.40 kg, which could certainly reach 7.0 kg with more time and with optimization cycles, however there was no longer any availability to proceed with this study. In Figure 5.31 it is possible to see the geometry obtained as well as some details in Figure 5.32. This process demonstrates the flexibility that this new design process can present and the need to have several test and review phases for components with such a level of demand as the case of the automobile industry.

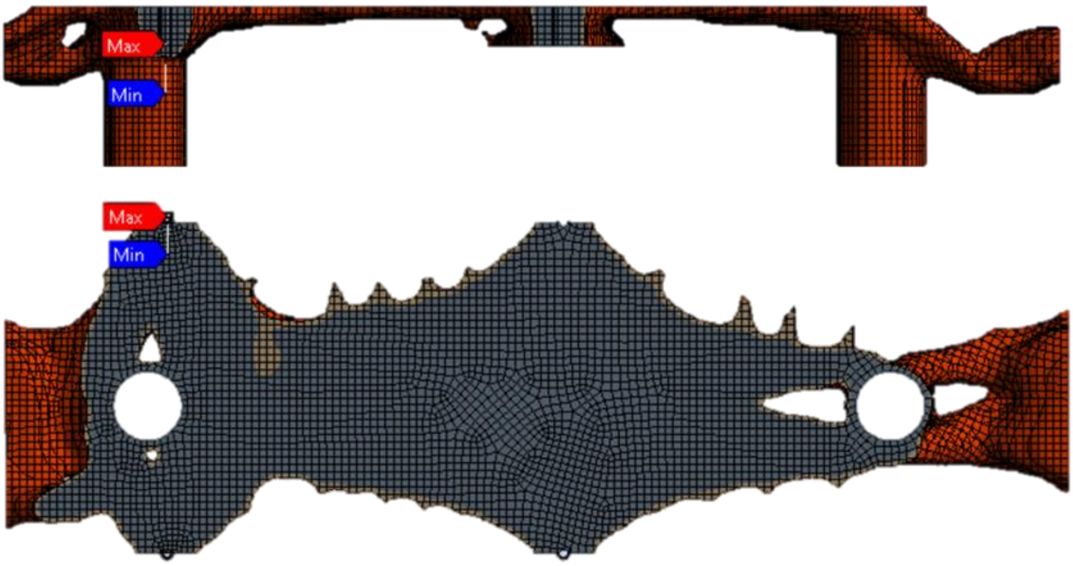


Figure 5.31 - Obtained geometry subject to Crash, Pothole and Breaking loads. It has a total mass of 8.40 kg.

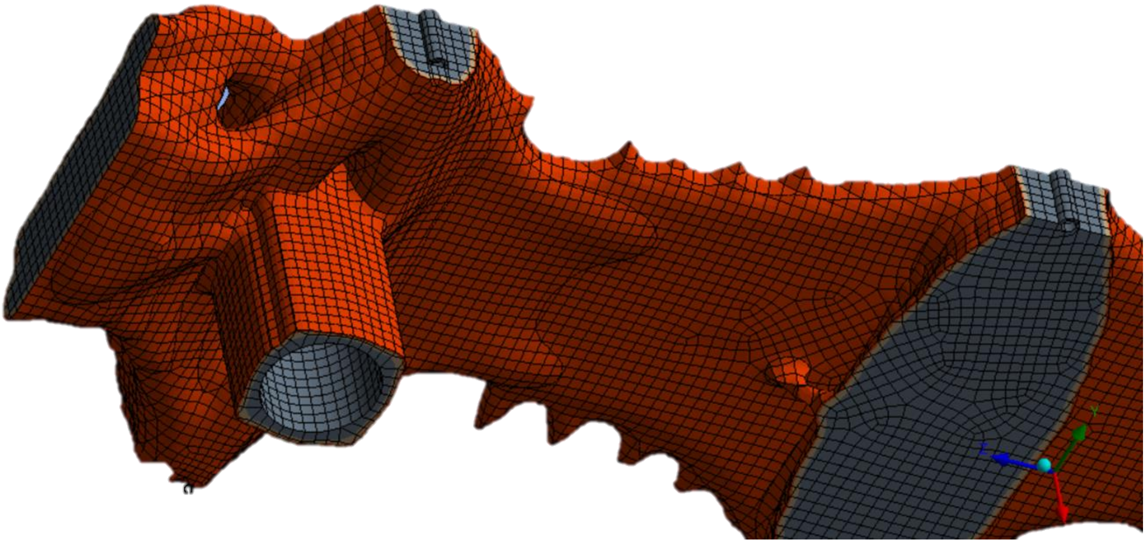


Figure 5.32 - Geometric details relating to critical support zones. This is another advantage of using the proposed new mechanical design process as it offers inspiration to the designer.

Finally, it might be interesting to analyse the impact that loads adjacent to Crash situations had on the final geometry in comparison with the geometry subject only to Pothole and Braking loads. In Figure 5.33, it is possible to observe at the top the geometry considering only Pothole and Braking loads and at the bottom the geometry considering the three scenarios. The impact that the loads adjacent to Crash situations had on the final geometry is evident, favouring the maintenance of material in the vicinity of the outer holes.

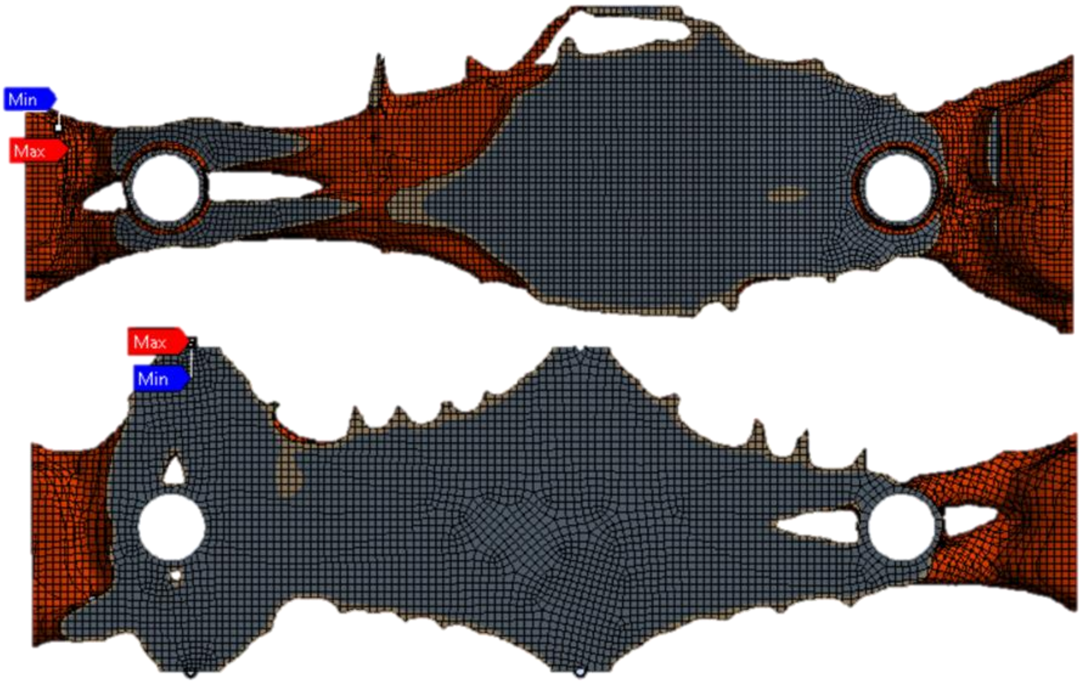


Figure 5.33 - Comparison between the geometry obtained considering only Pothole and Braking loads (upper geometry with a mass of 5.87kg) and the geometry considering Crash, Pothole and Braking loads (lower geometry with a mass of 8.4kg).

6. Conclusion

6.1 Discussion of results

In Chapter 2 was presented a new proposal to adopt the design process traditionally used in the automotive industry to the existing difficulties. Although this goal is ambitious, there are already several articles with new design proposals from other companies, however, none that could combine topology optimization algorithms with genetic algorithms were found. The main disadvantage that the suggested strategy may present will be the time taken to execute it, however, it is expected that the quality of the results can compensate for the cost/hour caused. In order to prove this, the results obtained for the two suggested case studies will be presented below.

In Table 6.1, it is possible to see a summary of the results obtained after the first step, composed of the Topology Optimization operations, and after the second step, composed of the operations with the Genetic Algorithms. As it is possible to see, the Topology Optimization operation was fundamental for the removal of material, removing about 50% of the material compared to the initial geometry. These results correspond to what was expected and there were no great doubts about whether it would be advantageous to be used in the design process. The main doubts centred on the genetic algorithms and on whether they could improve the results previously obtained after the Topology Optimization operation. Regarding the genetic algorithms, it was also possible to prove their effectiveness, since a reduction of 7.7% was possible in relation to the geometry obtained after T.O. while the maximum stress was decreased from 161 MPa to 153 MPa. These results were encouraging, however the final Crossbeam case study would be the real challenge for this proposed strategy.

Table 6.1 - Summary of results obtained after each step for the initial case study.

Initial Case Study		
Stage	Mass	Max von-Mises Stress
Original	62,8g	144 MPa
After Topology Optimization	31,4g	161 MPa
After Genetic Algorithms	29,1g	153 MPa

Concerning the final case study, related to Crossbeam, the results were also extremely positive. In Table 6.2 it is possible to see a summary of the results where a great mass reduction in the Topology Optimization component and a fine-tuning of results by the genetic algorithms can be seen. The rise from 15.40 MPa to 153 MPa after the use of genetic algorithms may be intriguing, but this only occurred for two reasons: the first was the freedom offered to the genetic algorithms in optimizing the defined parameters, since the only restriction that was defined was a limit of 150 MPa, and the second because the operation of topology optimization could have been deeper in the removal of material and consequently increased the maximum stress registered, however, the results were already becoming less interesting due to its manufacturing difficulty.

Table 6.2 - Summary of results obtained after each step for the final case study.

Final Case Study		
Stage	Mass	Max von-Mises Stress
Original	41,3 kg	7,65 MPa
After Topology Optimization	6,31 kg	15,4 MPa
After Genetic Algorithms	5,45 kg	153 MPa

Finally, if there were still any doubts about how the proposed new design process could replace the traditional process, in Figure 6.1 it is possible to observe a comparison between the geometry previously created by the team at CES following the traditional process and the geometry obtained with the new design strategy. The similarities are notorious and although this new design strategy is still in an embryonic stage, there may be a huge margin of progression and, consequently, a huge future for this type of strategy.

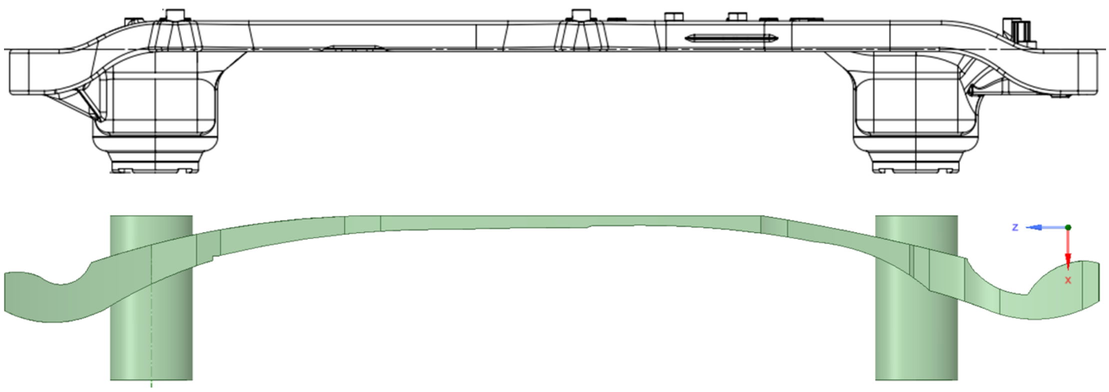


Figure 6.1 – Comparison between the crossbeam created by the traditional design process and the crossbeam created by the proposed design process.

6.2 Synthesis

This dissertation aimed to question the traditional design process, identify the main difficulties in design for the automotive industry, present an alternative design process and in the end demonstrate that this alternative is feasible and advantageous. With the application of this new design strategy to two case studies, it was possible to prove that the results obtained by this new design method may have a higher quality than the results obtained in a traditional process, however, the increase in execution time and the need of sync between the simulation engineer and the mechanical designer is recognized as a challenge. In the end, knowing that this proposed strategy is still in an embryonic stage, it is natural that it may not be able to completely replace the traditional process, however, it is expected that with the evolution of the areas of Topology Optimization and Genetic Algorithms together with the maturation of this proposed design strategy may result in the standard automotive design in the future.

6.3 Future work

Due to the short period of time for the execution of the dissertation, approximately 8 months, and the setbacks influenced by the Covid-19 pandemic, it was not possible to carry out the experimental activity or reach the ideal optimization for Crossbeam when subjected to Crash, Pothole and Braking situations, however, these could likely be completed in the future. With the requirements reaching maximum levels, especially in the automotive industry, and knowing the inevitable electrification of automobiles, design processes that can remove mass from the components (improving the autonomy of these vehicles) while ensuring the safety requirements will certainly be explored and adopted. It is hoped that this dissertation can contribute to these challenges.

Bibliography

Algorithm, M. G. (2020). *MOGA Workflow MOGA Steps to Generate a New Population*.

Deaton, J. D., & Grandhi, R. V. (2014). A survey of structural and multidisciplinary continuum topology optimization: Post 2000. *Structural and Multidisciplinary Optimization*, *49*(1), 1–38. https://doi.org/10.1007/s00158-013-0956-z

Fiebig, S., Sellschopp, J., Manz, H., Vietor, T., Axmann, J. K., Schumacher, A., & Ag, V. (2015). Future challenges for topology optimization for the usage in automotive lightweight design technologies. *11th World Congress on Structural and Multidisciplinary Optimization*, *June*, 1–8.

Grebenişan, G., & Salem, N. (2017). The multi-objective genetic algorithm optimization, of a superplastic forming process, using ansys®. *MATEC Web of Conferences*, *126*. https://doi.org/10.1051/matecconf/201712603003

Jensen, F. (2018). Topology Optimization of Turbine Manifold in the Rocket Engine Demonstrator Prometheus. *Master Thesis at Luleå University of Technology, Luleå, Sweden*, *77*.

Kazakis, G., Kanellopoulos, I., Sotiropoulos, S., & Lagaros, N. D. (2017). Topology optimization aided structural design: Interpretation, computational aspects and 3D printing. *Heliyon*, *3*(10), e00431. https://doi.org/10.1016/j.heliyon.2017.e00431

Koga, J., Koga, J., & Homma, S. (2013). Checkerboard Problem to Topology Optimization of Continuum Structures. *CoRR*, *abs/1309.5*. http://arxiv.org/abs/1309.5677

Ma, Z. D., Wang, H., Kikuchi, N., Pierre, C., & Raju, B. (2006). Experimental validation and prototyping of optimum designs obtained from topology optimization. *Structural and Multidisciplinary Optimization*, *31*(5), 333–343. https://doi.org/10.1007/s00158-005-0530-4

Manuel, A., & Monteiro, D. A. (2017). *Topology Optimization of Microstructures with Constraints on Average Stress and Material Properties*. *November*, 1–10.

Pang, T. Y., & Fard, M. (2020). Reverse engineering and topology optimization for weight-reduction of a bell-crank. *Applied Sciences (Switzerland)*, *10*(23), 1–16. https://doi.org/10.3390/app10238568

Reiland, J., Laszio, B., & Marcos, L. (2020). *A Vision on The Future of Automotive Lightweighting*. *Accelerating the decarbarbonisation of automotive mobility by means of lightweighting*.

Tyflopoulos, E., Flem, D. T., Steinert, M., & Olsen, A. (2018). State of the art of generative design and topology optimization and potential research needs. *Proceedings of NordDesign: Design in the Era of Digitalization, NordDesign 2018*, 1–15.

APPENDIX 1 – INITIAL CASE STUDY (EXPERIMENTAL ACTIVITY DETAILS)

In this appendix it is possible to find information regarding technical drawings, a summary with a brief description of the purpose of each component and also some information about the standard used for the construction of the test specimen.

The technical drawings referring to the initial geometry, the optimized geometry, the tensile test specimen to characterize the material and other accessories designed to carry out the experimental activity can be consulted in the next figures. Note that the technical drawings are not following the NP 204 standard due to the simplicity required for the supplier.

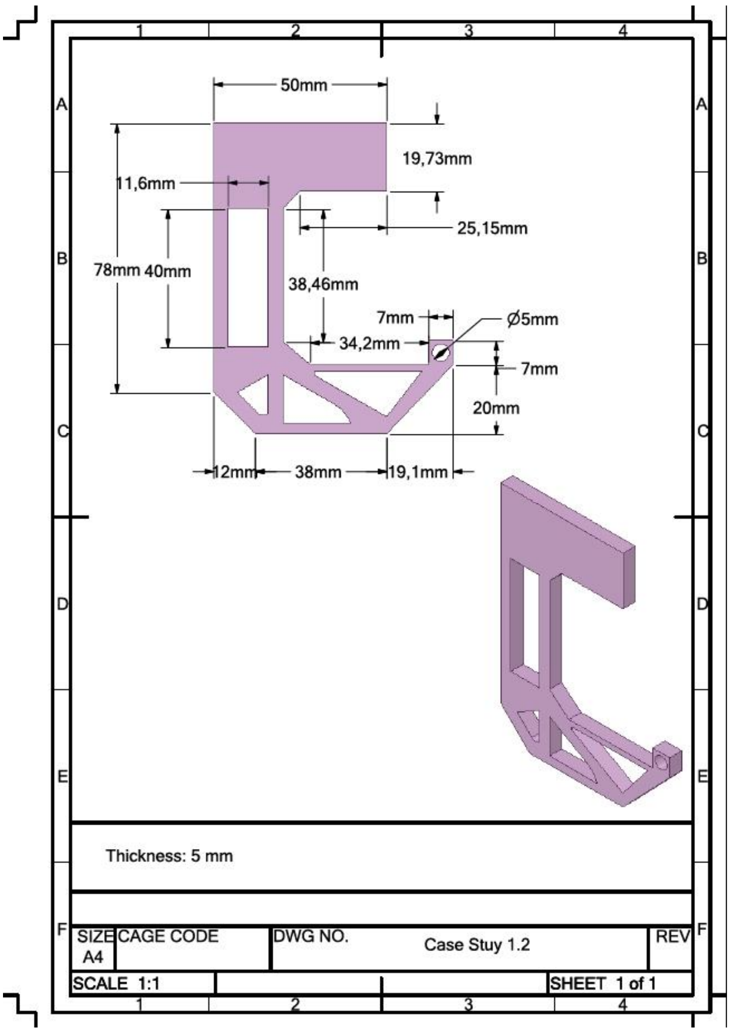


Figure 0.1 - Main dimensions of the optimized geometry (Case Study 1.2).

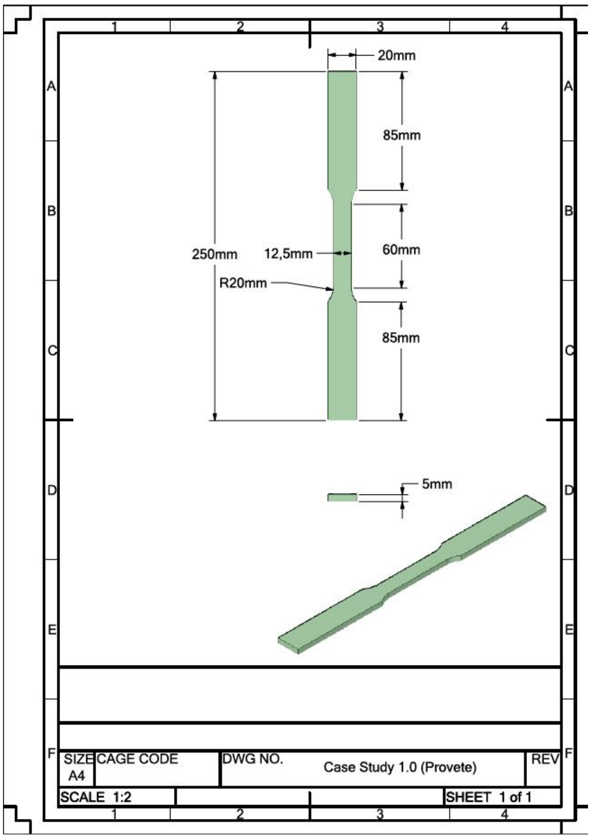


Figure 0.2 – Technical drawing referring to the specimen for the tensile test. (Case Study 1.0)

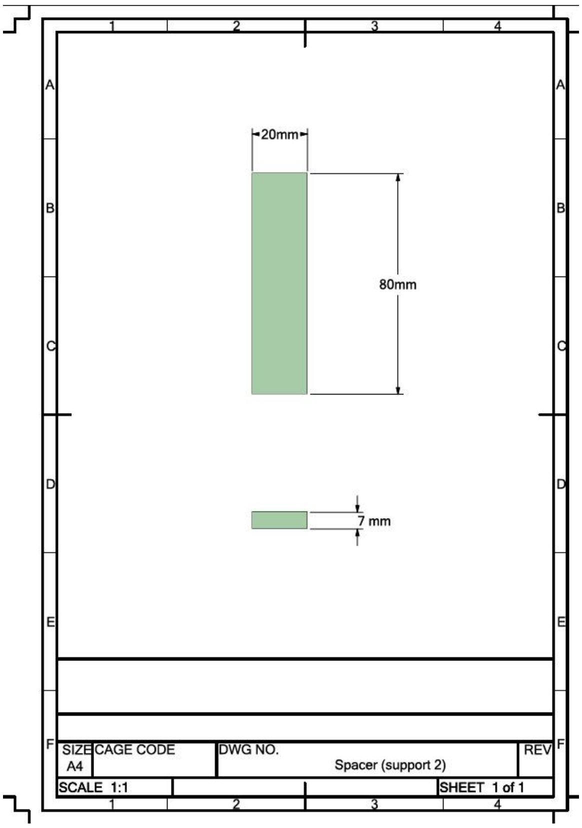


Figure 0.3 - Technical drawing referring to the spacer.

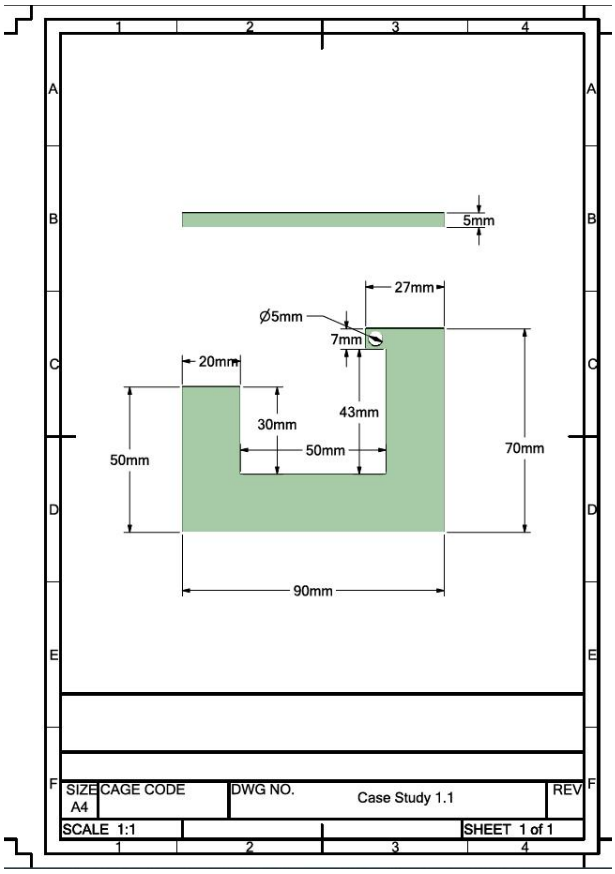


Figure 0.4 – Technical drawing referring to the initial geometry of the case study. (Case Study 1.1)

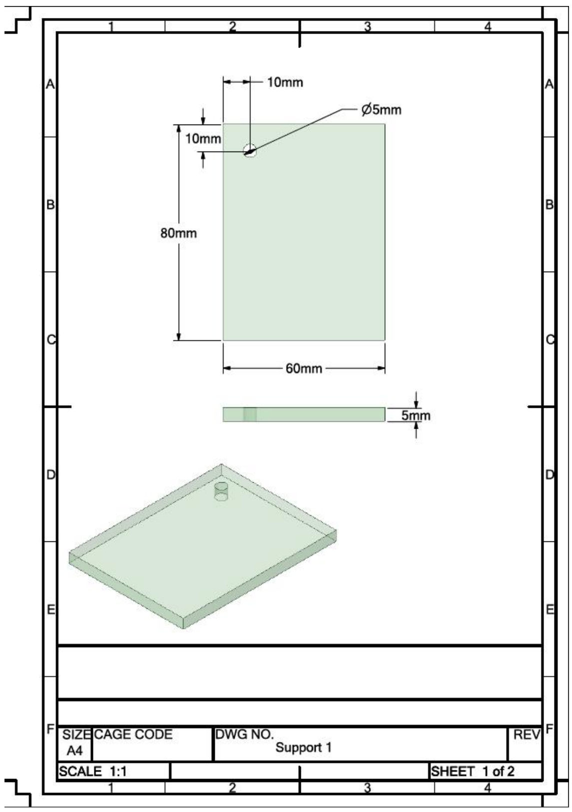


Figure 0.5 - Technical drawing for support 1.

Below, in Figure 0.6 and Figure 0.7, it is possible to consult some information in an objective and essential way for a quick understanding of the production needs of these components.

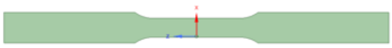
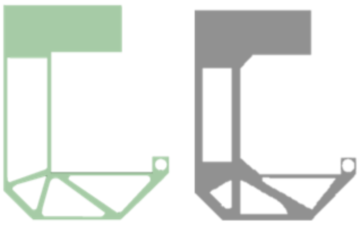
Component	Number of Samples	Objective	Mass (g)
 **Case Study 1.0**	3	The main objective of this case study is to be able to characterize the material. Through the test it will be possible to establish a basis for comparison for the material. The component design was based on the ISO 6892-1 (2009) standard.	
 **Case Study 1.1**	3	This case study represents the initial component without any kind of optimization. This geometry already includes some details that will facilitate the experimental procedure. The objective is to apply the optimization operation, whether topological or genetic, to this geometry in order to reduce the mass and maintain the same structural rigidity. The study carried out assumes that the geometry is supported into the upper part and that a force of 300N will be applied to the lower hole.	46,51
 **Case Study 1.2**	3	In this case study, we started from the geometry obtained through the Topology Optimization operation of Case Study 1.1. After obtaining the first optimized geometry, this geometry underwent a second optimization through genetic algorithms (MOGA).	29,55

Figure 0.6 - Brief summary with the main information related to the component under analysis.

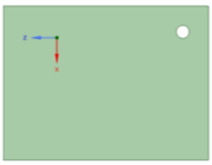
Component	Number of Samples	Objective	Mass (g)
 **Support Sheet (Support 1)**	4 / 12	These support sheets will be responsible for the contact with the equipment's surface. These will have to present a 5 mm hole (coincident with the case study) so that the experimental activity can be carried out.	
 **Spacer (Support 2)**	2 / 6	The purpose of this component is to ensure that the lashing on the support sheets does not collide with the necessary degree of freedom for the lower zone of the component under study. **Note:** **this will be the only component that cannot be the same thickness as the rest**, precisely to guarantee freedom.	
 **Pin (Support 3)**	6	The pin must have a diameter of 5 mm, however great details are not required. This can be **produced at the university.**	

Figure 0.7 - Brief summary with the main information about the accessories needed for the experimental activity.

Finally, it is possible to observe in Figure 0.8, a small excerpt from the ISO 6892-1 (2009) standard used as a reference for the construction of the test specimen that would be essential for the correct definition of the material.

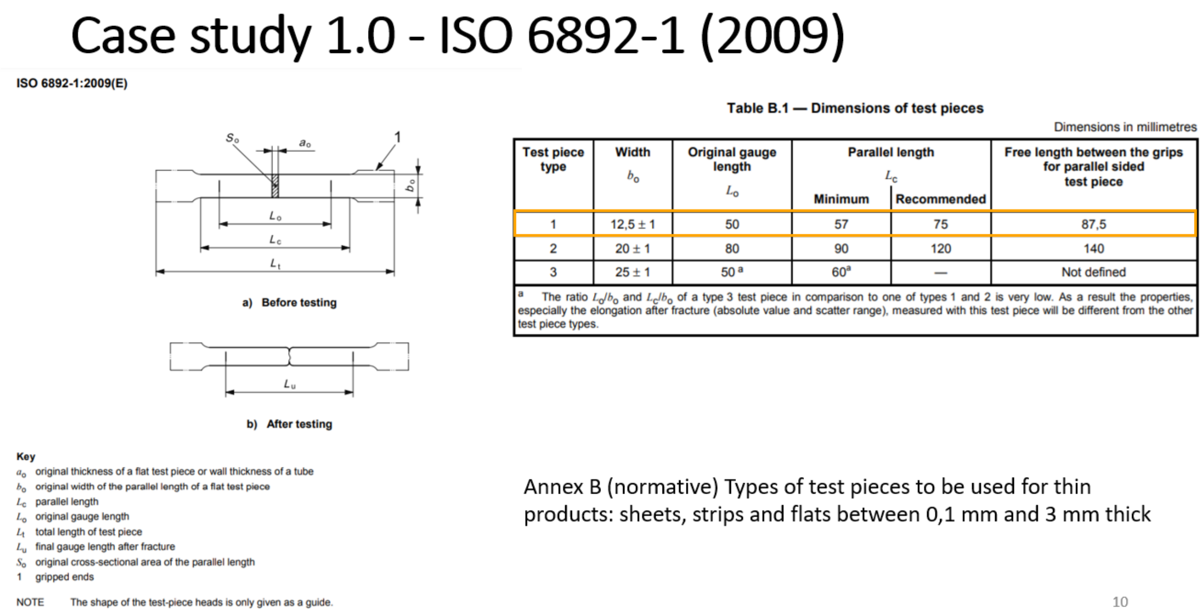


Table B.1 — Dimensions of test pieces

Dimensions in millimetres

Test piece type	Width b_o	Original gauge length L_o	Parallel length L_c		Free length between the grips for parallel sided test piece
			Minimum	Recommended	
1	12,5 ± 1	50	57	75	87,5
2	20 ± 1	80	90	120	140
3	25 ± 1	50 [a]	60[a]	—	Not defined

[a] The ratio L_o/b_o and L_c/b_o of a type 3 test piece in comparison to one of types 1 and 2 is very low. As a result the properties, especially the elongation after fracture (absolute value and scatter range), measured with this test piece will be different from the other test piece types.

Figure 0.8 - Excerpt from ISO 6892-1 (2009) used for the construction of the specimen for the tensile test.

Appendix 2 – Final Case Study (Initial Dimensions)

Although the component under review was guided by an extensive set of requirements and limitations, the general dimensions of the component did not need to be exactly as suggested. To try to create a starting point for the problem that was completely different from the previous study carried out by the team at CES, a strategy was created to obtain the limiting dimensions that the component could have. This strategy consisted of superimposing a parallelepiped on the technical drawing carried out by the CES team and thus obtaining the maximum possible dimensions. This process led to the creation of a parallelepiped with a length of 854.5 mm, a width of 266mm and a height of 128.32mm. In Figure 0.1 - Representation of the strategy used to create the initial dimensions of the component. it is possible to observe a representation of the strategy used. While at first glance this strategy may seem lax, it has actually proven to be extremely effective.

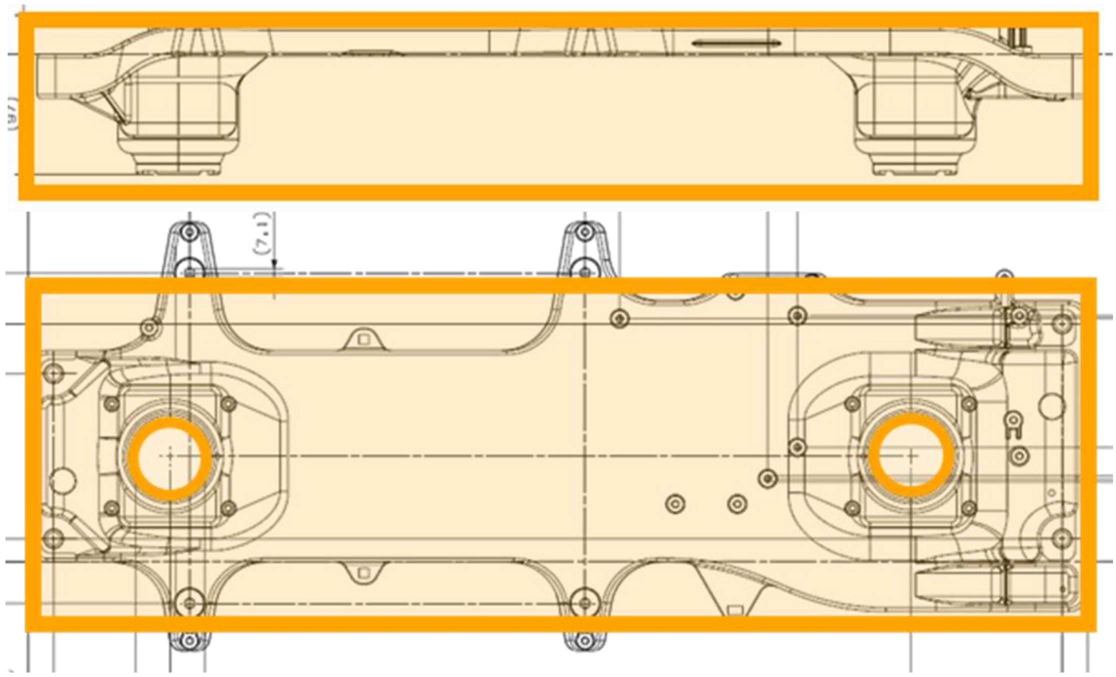


Figure 0.1 - Representation of the strategy used to create the initial dimensions of the component.