Application of Topology Optimization to Satellite Tertiary Structures

Emanuel Filipe Moreira

Thesis to obtain the Master of Science Degree in Aerospace Engineering

Supervisors: Prof. Miguel António Lopes de Matos Neves
Engº. João Daniel Ramos Ricardo
Prof. Giulio Romeo

Examination Committee
Chairperson: Prof. Filipe Szolnoky Ramos Pinto Cunha
Supervisor: Prof. Miguel António Lopes de Matos Neves
Member of the Committee: Prof. José Arnaldo Pereira Leite Miranda Guedes

November 2016
This page has been intentionally left in blank.
Dedicated to my family and friends
Acknowledgments

Firstly, I would like to thank my supervisor from Instituto Superior Técnico, Lisbon, Professor Miguel de Matos Neves, for his support, advisement, bibliographic material, availability and close attention to detail he offered for this thesis.

To my supervisor from Politecnico di Torino, Professor Giulio Romeo, for his support, advisement and availability offered for the realization of this thesis.

To my supervisor from Active Space Technologies®, Engº. João Ricardo, for is guidance, advisement, bibliographic material, confidence and availability.

Active Space Technologies® for the opportunity of internship that allows the realization of this thesis.

To everyone at Active Space Technologies® for the extensive know-how provided and the companionship demonstrated.

Finally, to my family and friends for all the support and motivation for the realization of this thesis, in special to Maria for the support and patience.
This page has been intentionally left in blank.
Abstract

The challenge of this thesis is to use topology optimization software to propose a design for a tertiary structure of a satellite.

Topology optimization is a computational material distribution method for synthesizing structures without any preconceived shape, allowing for the introduction of holes or cavities in structures. This capability usually results in great savings in weight or improvement of structural performance such as stiffness or strength.

The additive manufacturing processes provide geometrical freedom for the design, making a perfect combination of this manufacturing processes with topology optimization results in order to obtain more efficient structures, in terms of mass and structural properties.

The ANSYS Wb v17 - Topology Optimization ACT is successfully verified and compared with the results from literature for MatLab codes.

The topology optimization analysis, with minimum compliance objective, is applied here in a satellite tertiary structure. This optimization is divided into two analysis, a 2D and 3D optimization, in order to save computational time on the 3D optimization.

The optimized structure, compared to the initial tertiary structure proposal, shows considerable stress reductions of 91.79% and a first natural frequencies increases of 504.96% at the expense of a small of 10.43% increase in the structure’s weight. This mass increase is obtained due to an increase of 191.3% in the mass for the 2D optimization followed by a reduction of 62.1% in the mass with the 3D optimization.

Keywords: Topology Optimization, Tertiary Structures, ANSYS, Minimum Compliance, Additive Manufacturing.
Resumo

O desafio desta tese é a utilização de um programa de otimização topológica para propor um desenho de uma estrutura terciária de um satélite.

Otimização Topológica é um método computacional de distribuição do material para sintetização de estruturas, sem qualquer forma premeditada, permitindo a formação de buracos ou cavidades. Esta característica resulta, normalmente, numa poupança de massa ou numa melhoria da performance estrutural, como a rigidez ou a robustez.

Os processos de fabricação aditiva proporcionam uma liberdade geométrica para a conceção da estrutura, fazendo uma combinação perfeita com os resultados da otimização topológica, a fim de obter estruturas mais eficientes, em termos de massa e propriedades estruturais.

O programa ANSYS Wb v17 - Topology Optimization ACT é verificado com sucesso, fazendo uma avaliação comparativa com a literatura dos códigos de MatLab.

A análise de otimização topológica, com o objetivo de compliance mínimo, é efetuada numa estrutura terciária de um satélite. Esta otimização é dividida em duas análises, otimizações 2D e 3D, de forma a poupar tempo de cálculo computacional na otimização 3D.

A estrutura otimizada apresenta consideráveis reduções de 91.79% nas tensões e aumento de 504.96% na primeira frequência natural, em comparação com a estrutura terciária inicial, à custa de um pequeno aumento de 10.43% na massa da estrutura. Este aumento na massa é justificado com o aumento de 191.3% na massa com a otimização em 2D, seguido de uma redução de 62.1% na massa com a otimização em 3D.

Palavras-chave: Otimização Topológica, Estruturas Terciárias, ANSYS, Compliance Mínimo, Fabrico Aditivo.
This page has been intentionally left in blank.
## Contents

Acknowledgments ......................................................................................................................... v

Abstract ........................................................................................................................................ vii

Resumo ....................................................................................................................................... viii

Contents ........................................................................................................................................ x

List of Tables ............................................................................................................................... xiii

List of Figures .............................................................................................................................. xiv

List of Symbols ............................................................................................................................ xix

1. Introduction............................................................................................................................. 1
   1.1. Motivation ...................................................................................................................... 1
   1.2. Objective ........................................................................................................................ 1
   1.3. Thesis Outline ................................................................................................................ 2

2. Background ............................................................................................................................ 3
   2.1. Additive Manufacturing ................................................................................................ 3
       2.1.1. Vat Photopolymerisation ................................................................................ 3
       2.1.2. Material Jetting .................................................................................................. 4
       2.1.3. Binder Jetting ..................................................................................................... 4
       2.1.4. Material Extrusion .............................................................................................. 5
       2.1.5. Powder Bed Fusion ............................................................................................. 5
       2.1.6. Sheet Lamination ................................................................................................. 6
       2.1.7. Directed Energy Deposition ................................................................................ 6
       2.1.8. Comparison of materials between the principal AM processes of metals ............. 7
   2.2. Introduction to Elasticity ................................................................................................ 9
   2.3. Distortion Energy Theory (Von Mises) ........................................................................ 10

2.4. Finite Element Method................................................................................................. 10
   2.4.1. Static Analysis ..................................................................................................... 11
   2.4.2. Modal Analysis .................................................................................................... 11
   2.4.3. Elements .............................................................................................................. 12

2.5. Topology Optimization ................................................................................................... 16
   2.5.1. Density Based Approach ....................................................................................... 16
7.2. Future Work................................................................................................................. 70

8. References........................................................................................................................... 71
List of Tables

Table 2.1 Materials used accordingly to the literature for EBM, DMLS, SLM and LENS processes [12]–[15] ............................................................................................................................................... 7
Table 2.2 Comparison between mechanical properties for Ti6Al4V, AlSi10Mg, Al 7075, Stainless Steel 17-4 PH and IN718 [16]–[19] ............................................................................................... 8
Table 4.1 Topology Optimization Parameters for 2D Academic Problems analysis with ANSYS ..................................................................................................................................................... 27
Table 4.2 Topology Optimization Parameters for 2D Academic Problems analysis with MatLab ..................................................................................................................................................... 27
Table 4.3 Topology Optimization Parameters for 3D Academic Problems analysis with ANSYS ..................................................................................................................................................... 39
Table 4.4 Topology Optimization Parameters for 3D Academic Problems analysis with MatLab ..................................................................................................................................................... 40
Table 5.1 Load Cases inputs for Static Analysis of the Tertiary Structure .................................. 50
Table 5.2 Comparison between mechanical properties for Aluminium Alloy from ANSYS, AlSi10Mg and Al 7075 [16], [19], [27] ......................................................................................... 50
Table 5.3 Initial Tertiary Structure Static Analysis Solution from ANSYS Wb, maximum Von Mises stress [MPa] and Margin of Safety for the Yield $MoSY$ and the Ultimate tensile strength $MoSU TS$ ..................................................................................................................................................... 53
Table 5.4 Initial Tertiary Structure Modal Analysis solutions obtain from ANSYS Wb.............. 55
Table 6.1 Load Cases inputs for Static Analysis of the Tertiary Structure .................................. 57
Table 6.2 Topology Optimization Parameters for 2D Tertiary Structure analysis ....................... 57
Table 6.3 Topology Optimization Parameters for Tertiary Structure analysis ............................ 60
Table 6.4 Optimized Tertiary Structure Static Analysis Solution from ANSYS Wb, maximum Von Mises stress [MPa] and Margin of Safety for the Yield $MoSY$ and the Ultimate tensile strength $MoSU TS$ ..................................................................................................................................................... 65
Table 6.5 Optimized Tertiary Structure Modal Analysis solutions obtain from ANSYS Wb........ 67
Table 6.6 Comparison between Static Analysis from Initial and Optimized Tertiary Structures . 68
Table 6.7 Comparison between Modal Analysis from Initial and Optimized Tertiary Structures 69
Table 6.8 Comparison of Volume and Mass between Initial, Based in 2D TO and Optimized Tertiary Structures ..................................................................................................................................................... 69
List of Figures

Figure 2.1 SLA schematics [11] .................................................................................................... 3
Figure 2.2 PolyJet schematics [10] ............................................................................................... 4
Figure 2.3 3DP schematics [10] .................................................................................................... 4
Figure 2.4 FDM schematics [11] ................................................................................................... 5
Figure 2.5 Powder Bed Fusion [10] ............................................................................................... 5
Figure 2.6 Sheet Lamination schematics [10] ............................................................................. 6
Figure 2.7 LENS schematics [11] .................................................................................................. 6
Figure 2.8 Representation of domain and sub-domain (finite elements) [26] ............................. 10
Figure 2.9 Shell elements schematics [25] ................................................................................. 12
Figure 2.10 Undeformed and deformed geometries of an edge in Classical Plate Theory [26] . 13
Figure 2.11 Undeformed and deformed geometries of an edge in Shear Deformation Plate Theory [26] ............................................................................................................................................... 14
Figure 2.12 SHELL181 schematics [27] ...................................................................................... 14
Figure 2.13 3D elements schematics: linear and quadratic tetrahedral, prism, and brick elements [26] ............................................................................................................................................... 15
Figure 2.14 SOLID185 schematics [27] ...................................................................................... 15
Figure 2.15 Checkerboards Effect in a MBB-Beam Topology Optimization analysis [28] .......... 19
Figure 2.16 Neighborhood (Ne) of element xe and filter size (rmin).......................................... 20
Figure 2.17 Additive Manufacturing Structure with Supports [34] ............................................... 21
Figure 3.1 MBB-Beam (Right) and Cantilever-Beam (Left) 2D problems ................................... 22
Figure 3.2 MBB-Beam Half topology optimization solutions obtained from ANSYS Wb with a mesh of 60x20, E = 1Pa, v = 0.3, p = 3, xmin = 0.001, ConvTer.Rel = 0.01%, MaxIter = 500, f = 50% and a TO post processing with relative density higher than 0.4 a) without smoothing and b) with smoothing................................................................................................................................. 23
Figure 3.3 Traditional FEM optimization analysis, based on [34] ............................................ 24
Figure 3.4 Topology Optimization typical method for component development, based on [5], [34] ..................................................................................................................................................... 24
Figure 3.5 Topology Optimization Analysis: 2D and 3D TO Analysis ......................................... 24
Figure 4.1 MBB-Beam Model and MBB-Beam Model simplification domain and boundary conditions for the static case ..................................................................................................... 28
Figure 4.2 MBB-Beam Half topology optimization solutions obtained: a) from the literature [3], b) from MatLab Optimization; and c) from ANSYS Wb with a mesh of 60x20 and Volume constrain of 50% ......................................................................................................................................... 28
Figure 4.3 MBB-Beam Half topology optimization objective function convergence (compliance) obtained from MatLab Optimization (Top) and from ANSYS Wb (Bottom) with a mesh of 60x20 and Volume constrain of 50% ......................................................................................................................................... 29
Figure 4.4 MBB-Beam topology optimization solutions with compliance (C) for volume constraint study obtained from MatLab Optimization (left pictures) and from ANSYS Wb (right pictures) with a mesh of 75x25 and Volume constrain of 70% (top), 50% (middle) and 30% (bottom) respectively ................................................................. 29

Figure 4.5 MBB-Beam topology optimization solutions with compliance (C) for mesh resolution study obtained from MatLab Optimization (left pictures) and from ANSYS Wb (right pictures) with a Volume constrain of 50% and a mesh of 30x20 (top), 60x40 (middle top), 90x60 (middle bottom) and 120x80 (bottom) respectively ........................................................................................................ 30

Figure 4.6 MBB-Beam Model domain and boundary conditions for distributed load............. 31

Figure 4.7 MBB-Beam with distributed load topology optimization solutions obtained: a) from MatLab Optimization; and b) from ANSYS Wb with a mesh of 80x50 and Volume constrain of 30% ............................................................................................................................................. 31

Figure 4.8 MBB-Beam with distributed load topology optimization objective function convergence (compliance) obtained from MatLab Optimization (Top) and from ANSYS Wb (Bottom) with a mesh of 80x50 and Volume constrain of 30%................................................................. 32

Figure 4.9 Cantilever-Beam Model domain and boundary conditions ................................. 32

Figure 4.10 Cantilever-Beam topology optimization solutions obtained: a) from literature [3]; b) from MatLab Optimization; and c) from ANSYS Wb with a mesh of 32x20 and Volume constrain of 40% ............................................................................................................................................. 32

Figure 4.11 Cantilever-Beam topology optimization objective function convergence (compliance) obtained from MatLab Optimization (Top) and from ANSYS Wb (Bottom) with a mesh of 32x20 and Volume constrain of 40% ............................................................................................................................................. 33

Figure 4.12 Cantilever-Beam topology optimization solutions with compliance (C) for mesh resolution study obtained from MatLab Optimization (left pictures) and from ANSYS Wb (right pictures) with a Volume constraint of 40% and a mesh of 40x20, 80x40 and 160x80 respectively ........................................................................................................ 34

Figure 4.13 Cantilever with Multiple Load Cases Model domain and boundary conditions ...... 34

Figure 4.14 Cantilever with Multiple Load Cases topology optimization solutions obtained a) from literature [3]; b) from MatLab Optimization; and c) from ANSYS Wb with a mesh of 30x30 and Volume constrain of 40% ............................................................................................................................................. 35

Figure 4.15 Cantilever-Beam with multiple load cases topology optimization objective function convergence (compliance) obtained from MatLab Optimization (Top) and from ANSYS Wb (Bottom) with a mesh of 30x30 and Volume constrain of 40% ................................................................. 35

Figure 4.16 Cantilever-Beam with Hole Model domain and boundary conditions ................. 36

Figure 4.17 Cantilever-Beam with Hole topology optimization solutions obtained a) from literature [3]; b) from MatLab Optimization; and c) from ANSYS Wb with a mesh of 45x30 and Volume constrain of 50% ............................................................................................................................................. 36

Figure 4.18 Cantilever-Beam with hole topology optimization objective function convergence (compliance) obtained from MatLab Optimization (Top) and from ANSYS Wb (Bottom) with a mesh of 45x30 and Volume constrain of 50% ............................................................................................................................................. 37
Figure 5.3 Tertiary Structure initial CAD design

Figure 5.4 Initial Tertiary Structure CAD front view with overall dimensions (mm)

Figure 5.5 Initial Tertiary Structure CAD top view with overall dimensions (mm)

Figure 5.6 Initial Tertiary Structure Surface Model

Figure 5.7 Initial Tertiary Structure Boundary Conditions with fixed support (blue) and the distributed mass representing RO (red)

Figure 5.8 Graphics of the mesh size study: maximum displacement (in the RO instrument), total strain energy and first 5 natural frequency for the Initial Tertiary Structure analysis

Figure 5.9 Initial Tertiary Structure Mesh with 36641 SHELL181 elements

Figure 5.10 Initial Tertiary Structure mesh detailed view

Figure 5.11 Initial Tertiary Structure Static Analysis Solution obtained from ANSYS Wb, the Von Misses Stresses (Pa) for the 2nd load case

Figure 5.12 Detail View of Initial Tertiary Structure Static Analysis Solution obtained from ANSYS Wb, the Von Misses Stresses (Pa) for the 2nd load case

Figure 5.13 Initial Tertiary Structure Modal solution obtained from ANSYS Wb, representing the Total Eigenvector Deformation (m) for the 1st Natural Frequency of 11.9 Hz

Figure 5.14 Initial Tertiary Structure Modal solution obtained from ANSYS Wb, representing the Total Eigenvector Deformation (m) for the 2nd(top left), 3rd(top right), 4th(bottom left) and 5th(bottom right) Natural Frequency of 33.0 Hz, 80.1 Hz, 141.8 Hz and 227.6 Hz respectively

Figure 6.1 2D Tertiary Structure Optimization Model

Figure 6.2 2D Tertiary Structure Optimization Boundary Conditions with fixed support (blue) and the distributed mass representing RO (red)

Figure 6.3 2D Tertiary Structure Initial Mesh for 2D Topology Optimization with 6189 SHELL181 elements

Figure 6.4 Design and Exceptions Regions for the 2D Topology Optimization of the Tertiary Structure

Figure 6.5 Tertiary Structure 2D Topology Optimization solution for a Volume constraint of 35% with the total length of relative densities

Figure 6.6 2D Tertiary Structure Topology Optimization objective function convergence (Multiple Compliance) obtained from ANSYS Wb

Figure 6.7 Tertiary Structure 2D Topology Optimization post processed solution for a Volume constraint of 35% with a relative density higher than 0.6

Figure 6.8 Tertiary Structure Model Design with Surfaces of 10 mm thickness based on the 2D Topology Optimization Analysis

Figure 6.9 Tertiary Structure Model Design Interior with Surfaces of 10 mm thickness based on the 2D Topology Optimization Analysis

Figure 6.10 Tertiary Structure Boundary Conditions based on the 2D Topology Optimization Analysis with fixed support (blue) and the distributed mass representing RO (red)

Figure 6.11 Tertiary Structure Model Design with Surfaces of 10 mm thickness based on the 2D Topology Optimization Analysis

Figure 6.12 Tertiary Structure Initial Mesh for Topology Optimization with 17117 SHELL181 elements
Figure 6.12 Design and Exceptions Regions for the Topology Optimization of the Tertiary Structure ................................................................. 61
Figure 6.13 Tertiary Structure Topology Optimization solution for a Volume constraint of 40% with the total length of relative densities ................................................................. 61
Figure 6.14 Tertiary Structure Topology Optimization objective function convergence (Multiple Compliance) obtained from ANSYS Wb ................................................................. 62
Figure 6.15 Tertiary Structure Topology Optimization solution for a Volume constraint of 40% with a relative density higher than 0.4 ................................................................. 62
Figure 6.16 Optimized Tertiary Structure Model Design .......................................................................................................................... 63
Figure 6.17 Optimized Tertiary Structure Boundary Conditions with fixed support (blue) and the distributed mass representing RO (red) ................................................................. 63
Figure 6.18 Graphics of the mesh size study: maximum displacement in the RO instrument, total strain energy and first 5 natural frequency for the Optimized Tertiary Structure analysis .... 64
Figure 6.19 Initial Tertiary Structure Mesh with 146935 SOLID185 elements ......................................................................................... 65
Figure 6.20 Optimized Tertiary Structure Static Analysis Solution obtained from ANSYS Wb, the Von Misses Stresses (Pa) for the 2nd load case ...................................................................... 66
Figure 6.21 Detail View of Optimized Tertiary Structure Static Analysis Solution obtained from ANSYS Wb, the Von Misses Stresses (Pa) for the 2nd load case .............................................. 66
Figure 6.22 Optimized Tertiary Structure Modal solution obtained from ANSYS Wb, representing the Total Eigenvector Deformation (m) for the 1st Modal Frequency of 71.9 Hz ................... 67
Figure 6.23 Optimized Tertiary Structure Modal solution obtained from ANSYS Wb, representing the Total Eigenvector Deformation (m) for the 2nd(top left), 3rd(top right), 4th(bottom left) and 5th(bottom right) Natural Frequency of 103.0 Hz, 188.9 Hz, 209.0 Hz and 277.1 Hz respectively ........................................................................................................................................ 67
List of Symbols

**Acronyms**

ACT  Application Customization Toolkit
ANSYS  Analysis of Systems, Inc.
ANSYS Wb  ANSYS Workbench
AM  Additive Manufacturing
CAD  Computer Aided Design
ConvTer  Convergence Termination Criteria
ConvTer_Abs  Absolute Convergence Termination Criteria
ConvTer_Rel  Relative Convergence Termination Criteria
DMLS  Direct Metal Laser Sintering
DOF  Degrees of Freedom
EBM  Electron Beam Melting
FDM  Fused Deposition Modelling
FEM  Finite Element Method
FoS  Factors of Safety
KKT  Karush-Kuhn-Tucker
LENS  Laser Engineered Net Shaping
LOM  Laminated Object Manufacturing
MatLab  Matrix Laboratory
MaxIter  Maximum Number of Iterations
MBB  Messerschmidt-Bölkow-Blohm
MMA  Method of Moving Asymptotes
MoS  Margin of Safety
OC  Optimality Criteria
RO  Radio Occultation
SHS  Selective Heat Sintering
SIMP  Solid Isotropic Material with Penalization
SLA  Stereolithography
SLM  Selective Laser Melting
SLS  Selective Laser Sintering
SQP  Sequential Quadratic Programming
STL  Standard Tessellation Language
TO  Topology Optimization
UAM  Ultrasonic Additive Manufacturing
UV  Ultraviolet
2D  Two-Dimensions
Greek Symbols

\( \alpha, \beta, \lambda \) and \( \mu \)  
Lagrangian Multipliers

\( \varepsilon \)  
Strain Tensor

\( \eta \)  
Numerical Damping Coefficient

\( \nu \)  
Poisson’s ratio

\( \rho \)  
Density

\( \sigma \)  
Stress Tensor

\( \sigma_{VM} \)  
Von Mises Stress

\( \sigma_{allowable} \)  
Allowable Load/Stress

\( \sigma_Y \)  
Yield strength

\( \sigma_{UTS} \)  
Ultimate Tensile strength

\{\Phi\}  
Mode Shapes

\( (\phi_x, \phi_y) \)  
Plate cross-section angle variation in \( x \) or \( y \)

\( \omega \)  
Natural Frequencies

Roman Symbols

\([C]\)  
Damping matrix

\( C \)  
Compliance

\( c_e \)  
Element Compliance

\( \frac{\partial C(x)}{\partial x_e} \)  
Sensitivity of the Element \( x_e \)

\( c_{new} \)  
Compliance of the new design

\( c_{old} \)  
Compliance of the old design

\( E \)  
Young’s modulus

\( E_e \)  
Element Young’s modulus

\( E_0 \)  
Young’s modulus of the solid material

\( E_{min} \)  
Young’s modulus of the void material

\{\( F \)\} or \( F \)  
Nodal Forces vector

\( f \)  
Volume Fraction

\( H_{ef} \)  
Height Factor

\([K]\) or \( K \)  
Stiffness matrix

\( K_e \)  
Element Stiffness matrix

\( k_e^0 \)  
Element Stiffness matrix for an element with Young’s modulus units

\([M]\)  
Mass matrix
Positive Move-limit

Number of Elements used to Discretize the Design Domain

Neighborhood of the Element $x_e$

Penalization Power

Filter Size

Kinetic Energy

Total Displacements of the Point $(x, y, z)$

Nodal Displacement vector

Nodal Velocity vector

Nodal Acceleration vector

Element Displacement vector

Potential Energy

Material Volume

Design Domain

Volume of Elements $x_f$

Volumic Force

Relative Density Distribution

Relative Element Density

Relative Element Density of the void material

New Design Variable

Old Design Variable
This page has been intentionally left in blank.
1. Introduction

1.1. Motivation

The aerospace industry is looking for new developments and some of the most important concerns are the structural efficiency of the structures at the lowest possible mass penalty. Although these concerns are not restricted to the aerospace and aeronautic industry, obtaining light structures in order to save on fuel consumption and, at the same time, improve the structural requirements are constant goals of the industry. Topology optimization software is a great tool to achieve these goals, but in the past the implementation of a design obtained from topology optimization was difficult or expensive with traditional subtractive manufacturing techniques. Recent developments in additive manufacturing processes in terms of manufacturing of metallic structures made possible the implementation of a structural topology optimization design.

In the area of optimization of structures, in particular with the resource to topology optimization, several studies have been considered. The topology optimization was introduced by the pioneer work of Bendsøe and Kikuchi [1] on homogenization method. Since then several authors have developed the concept, including Bendsøe and Sigmund [2] with a densities based approach. Also several MatLab codes were developed with educational purposes in topology optimization, such as the TopOpt2D from Sigmund [3]. After this introduction and application of the topology optimization in commercial software, several case studies and methodologies appear in order to establish a successful structural optimization.


One success story of Altair® [5] describes the combination of topology optimization with laser additive manufacturing as a new potential for lightweight structures, using a similar method of optimization as described by Brackett, Ashcroft and Hague [4]. The topology optimization also can be applied in aeronautical areas, for example in the design of aircraft wing box ribs, as described in the case study of Krog, Tucker, Kemp, and Boyd [6]

Other case study, also provided by Altair® in combination with EADS [7] describes a topology optimization of an aerospace part to be produced by additive layer manufacturing, using a two cycle design strategy, the first cycle the optimization was for the defining of new constraints and reshape the design, and the second cycle for a optimization with the preconceived design of the first cycle, with improvements in the optimization approach.

1.2. Objective

The main purpose of this thesis is to use topology optimization software to propose a design for a tertiary structure of a satellite. To achieve this goal a two stage optimization methodology is applied, with a 2D and 3D analysis in order to save in the computational time cost, using a
combination of a CAD software, Solid Edge, and a FEM software with topology optimization modulus, ANSYS Wb v17 with Topology Optimization ACT.

This optimization methods are applied with a minimum compliance objective, increasing the stiffness of the structure, being expected a tertiary structure with better structural performances, such as lower stress and higher natural frequency results.

1.3. Thesis Outline

This thesis is composed by seven chapters: Introduction, Background, Methodology, Verifications with Academic Problems, Tertiary Structure Analysis, Tertiary Structure Topology Optimization Analysis and Conclusions.

In the introduction chapter the motivation and objectives of the thesis are defined, highlighting its relevance and providing an overview of the subjects that are going to be discussed.

The second chapter is the background, where information about the several additive manufacture technics is briefly introduced. In addition, an introduction to the elasticity problem with the equation of motion, the strain-displacement relation and the Hooke’s law is presented. The Von Mises stresses is defined and an introduction of the finite element method is added, in order to provide a solid basis to the analysis. Finally, an introduction to the topology optimization analysis is conducted.

The methodology chapter describes the different methods used for the analysis, starting with the approach for the academic problems, where the inputs parameters for the topology optimization are explained and the method for verification of the ANSYS Wb v17 - Topology Optimization ACT is described. After that the traditional finite element analysis method is presented, with application in the initial tertiary structure design. Finally the optimization procedure is introduced, where the 2D and 3D approach for topology optimization is explained.

The verifications with academic problems chapter is a benchmarking of the topology optimization module from ANSYS® software. In this chapter the results from ANSYS are compared with the results from the literature and MatLab optimization codes, TopOpt2D [3] and TopOpt3D [8].

In the tertiary structure analysis chapter the preliminary finite element analysis of the tertiary structure are presented. These analysis correspond to a static and modal analysis to evaluate the stresses suffered in the structure and the natural frequencies of the tertiary structure. These results are verified with the requirements and represent a reference for the optimization results.

In the tertiary structure topology optimization analysis chapter the topology optimization of the structure is described. Firstly the 2D optimization is performed, then the 3D structural topology optimization is executed. Finally the structural results, stress, natural frequency and mass of the new design are compared with the initial design, from the previous chapter.

The final chapter draws conclusions about the methodology and results of this thesis and suggests subjects in need of further development.
2. Background

In this chapter a brief review of the background information in this thesis is presented. The first section explains the additive manufacturing processes, where a categorization of the different processes is described and a brief comparison between the metallic processes is performed. The second section gives an introduction to elasticity problems. The third section gives a brief introduction to the distortion energy theory, introducing the Von Mises stresses used in this thesis. The background information about finite element method needed for the analysis performed in this thesis is presented in the fourth section. The fifth section is a topology optimization review of the problems and methods. Finally, an introduction of the association between additive manufacture techniques and topology optimization analysis is performed.

2.1. Additive Manufacturing

Additive Manufacturing (AM) is an emerging process for the manufacturing of objects. According to ASTM F-42 [9], a committee to standardize AM terminology, AM is defined as:

“The process of joining materials to make objects from 3D model data, usually layer upon layer, as opposed to subtractive manufacturing technologies.” [9]

According with this standardization, the different AM processes can be classified in 7 categories: Vat Photopolymerisation, Material Jetting, Binder Jetting, Material Extrusion, Powder Bed Fusion, Sheet Lamination and Directed Energy Deposition. This section is based on information from Loughborough University [10] and Wong and Hernandez [11].

2.1.1. Vat Photopolymerisation

The Vat Photopolymerisation is a liquid based process that consists in the curing or solidification of a photosensitive polymer when an ultraviolet (UV) laser makes contact with the resin.

The most popular Vat Photopolymerisation process is Stereolithography (SLA), was the first and is the most widely used process of AM, and was developed by 3D Systems, Inc.. This process is illustrated in Figure 2.1 and starts with a platform on the top going downwards while the UV laser is applied to specific regions, making the resin solidify. After that, the excess of resin is drained and can be used again.

![Figure 2.1 SLA schematics](11)
The materials used in this process are limited to Plastics and Polymers, making it a disadvantage. Also, the resin needs to be drained and changed every time a different material is used.

2.1.2. Material Jetting

Material Jetting is a process similar to a 2D ink jet printer. The inkjet head moves along the platform, depositing a photopolymer which is cured by a UV lamp after each layer is finished.

An example of this process is PolyJet, patented by Objet, illustrated in Figure 2.2. First, the liquid photopolymer, and if needed, the support material, are dropped, forming the first layer. After the UV lamp makes the curing of the material the process repeats itself for the following layers.

![Figure 2.2 PolyJet schematics](image)

The materials that are used are polymers and plastics, and there is a possibility of printing with different materials, including a support material. A disadvantage is that the support material is often required, even though it can be easily removed afterwards most of the times.

2.1.3. Binder Jetting

The Binder Jetting process use 2 materials, a powder based material and a liquid binder material that acts like an adhesive between powder layers. It was developed at MIT as 3D Printing (3DP) and uses a powder based material in a building platform that goes downwards after the application of the liquid binder material, where required. This process is illustrated in Figure 2.3

![Figure 2.3 3DP schematics](image)

A disadvantage is the fact that the binding material does not always make the object suitable for structural parts. This process can use a variety of polymers and ceramics.
2.1.4. Material Extrusion

In Material Extrusion the material is drawn through a nozzle, where it is heated and is then deposited layer by layer. The most common process is Fused Deposition Modelling (FDM) and belongs to Stratasys. This process is illustrated in Figure 2.4 and starts with the melted material dropped in a continuous stream on the base, after which the platform drops and another layer can be made on top of the previous one. This process uses polymers and plastics.

![Figure 2.4 FDM schematics](image)

2.1.5. Powder Bed Fusion

Powder Bed Fusion is a process that consists on the melting or sintering and fusing the powder material on the required region. This process starts with a platform with the powder material that can be spread by a roller on the platform and previous layers. A laser or an electron beam is used for the melting or sintering of the powder bed allowing the connection and creation of the layer. This process can be seen in Figure 2.5.

![Figure 2.5 Powder Bed Fusion](image)

A variety of techniques can be used, the most common being the Direct Metal Laser Sintering (DMLS), Electron Beam Melting (EBM), Selective Heat Sintering (SHS), Selective Laser Melting (SLM) and Selective Laser Sintering (SLS).

EBM uses an electron beam to perform the melting and requires vacuum. It uses metals and alloys to make functional parts. One of the future uses of this technique may be the application in space due to the fact that it requires vacuum environment.

SLS and DMLS use the same process, the one of sintering the material instead of completely melting it. The difference between them is that DMLS uses metals and alloys and SLS uses polymers and ceramics.

SHS differs from SLS in the sintering source, using a heated thermal print instead of a laser. The material used is Nylon.
SLM differs from SLS in the connection process. SLM makes the material melt, which gives it a better structural capacity. Its disadvantage is the porosity of the material, because using a full melting of the material does not allow for control of the porosity.

### 2.1.6. Sheet Lamination

The Sheet Lamination Process combines additive and subtractive techniques. The material comes in sheet form and the layers are bonded by pressure, heat application and using thermal adhesive coating or welding. After that a laser cuts the material to the shape of each layer. This process can be illustrated in Figure 2.6 and is most known for 2 different types of processes, the Laminated Object Manufacturing (LOM) and the Ultrasonic Additive Manufacturing (UAM).

![Figure 2.6 Sheet Lamination schematics](image)

The main differences between LOM and UAM are the materials and the type of bound between layers. The LOM uses sheets of paper and an adhesive coating and UAM uses sheets of metal and a welding bond. There is an imposed restriction in sheet lamination processes, which is the need to use a sheet material capable of being rolled.

### 2.1.7. Directed Energy Deposition

An example of Directed Energy Deposition is Laser Engineered Net Shaping (LENS), which consists on an injection of metal powder or wire in specific regions and melting it with a laser beam. The metal solidifies when cooled down, forming the object. This process is illustrated in Figure 2.7 and the big advantage is that the material can be deposited from any angle, which makes it an advantageous technique for repairing and maintaining structural parts. Another advantage of this process is the large variety of metal materials that it can support, like stainless steel and aluminium. A disadvantage of the process is the high concentration of residual stresses due to uneven heating and cooling processes.

![Figure 2.7 LENS schematics](image)
2.1.8. Comparison of materials between the principal AM processes of metals

For main focus of this thesis is the AM processes for metallic materials. Having that into account, a brief compilation of the AM processes that manufacture materials such as Titanium, Aluminium, Steel and Nickel Alloy is presented, as well as the type of alloys they support. A summary of this information is presented in Table 2.1, where the AM processes EBM, DMLS, SLM and LENS are matched to the metallic materials they support.

Table 2.1 Materials used accordingly to the literature for EBM, DMLS, SLM and LENS processes [12]–[15]

<table>
<thead>
<tr>
<th>AM</th>
<th>Titanium</th>
<th>Aluminium</th>
<th>Stainless Steel</th>
<th>Nickel Alloy</th>
<th>Misc.</th>
</tr>
</thead>
<tbody>
<tr>
<td>EBM [12]</td>
<td>CP Ti Grade 2</td>
<td></td>
<td>None</td>
<td>IN 718</td>
<td>CoCrMo super alloy ASTM F75</td>
</tr>
<tr>
<td></td>
<td>Ti6Al4V</td>
<td></td>
<td>17-4 PH</td>
<td>IN 625</td>
<td>CoCrMo super alloy ASTM F75</td>
</tr>
<tr>
<td></td>
<td>Ti6Al4V ELI</td>
<td>AIsi10Mg</td>
<td>15-5</td>
<td>IN 718</td>
<td>Maraging Steel 12709</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>17-4</td>
<td>IN 939</td>
<td>CoBalt-Alloys</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>UNS S31673</td>
<td>IN 625</td>
<td>Cobalt-Alloys</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Tooling grade</td>
<td>IN 625</td>
<td>Carbon Graphite</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Steel</td>
<td>IN 625</td>
<td>Rene</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Hastelloy X</td>
<td>IN 625</td>
<td></td>
</tr>
<tr>
<td>SLM [14]</td>
<td>Ti6Al4V</td>
<td>AIsi10Mg</td>
<td>17-4PH</td>
<td>IN 718</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Ti6Al7Nb</td>
<td>AIsi12</td>
<td>15-5PH</td>
<td>IN 939</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>AIsi7Mg</td>
<td>316L</td>
<td>IN 625</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1.2709</td>
<td>IN 625</td>
<td></td>
</tr>
<tr>
<td>LENS [15]</td>
<td>CP Ti</td>
<td>CP Al</td>
<td>17-4 PH</td>
<td>IN 600</td>
<td>Cobalt-Alloys</td>
</tr>
<tr>
<td></td>
<td>Ti6Al4V</td>
<td>CP Al</td>
<td>17-4 PH</td>
<td>IN 625</td>
<td>Carbon Graphite</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2024</td>
<td>316</td>
<td>IN 625</td>
<td>Rene</td>
</tr>
<tr>
<td></td>
<td>Ti6Al2Sn4Zr2Mo</td>
<td>4047</td>
<td>S7</td>
<td>IN 625</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1018</td>
<td>IN 690</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Ti6Al2Sn4Zr6Mo</td>
<td>6061</td>
<td>H13</td>
<td>IN 718</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>309</td>
<td>IN 718</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Ti48Al2Cr2Nb</td>
<td>7075</td>
<td>420</td>
<td>Ni superalloys</td>
<td></td>
</tr>
</tbody>
</table>

By analyzing Table 2.1 it is observable that the LENS process is the one with more variety of materials to manufacture. The main disadvantage of LENS is the high concentration of residual stresses, as mentioned above. SLM also processes a high variety of materials, but has a restriction on structure size due to the need of having a power bed material in a building platform.
Knowing the range of metal materials used on AM, the main mechanical properties for the most common materials used on this processes are represented in Table 2.2. In example, a titanium alloy, the Ti6Al4V, two types of aluminium alloy, the AlSi10Mg and the Al 7075, a stainless steel, 17-4 PH, and a nickel alloy, IN718 are shown. The represented mechanical properties are density ($\rho$), Young’s modulus ($E$), Poisson’s ratio ($\nu$), Yield strength ($\sigma_Y$) and Ultimate tensile strength ($\sigma_{UTS}$).

**Table 2.2 Comparison between mechanical properties for Ti6Al4V, AlSi10Mg, Al 7075, Stainless Steel 17-4 PH and IN718 [16]–[19]**

<table>
<thead>
<tr>
<th>Material</th>
<th>$\rho$ [Kg/m$^3$]</th>
<th>$E$ [GPa]</th>
<th>$\nu$</th>
<th>$\sigma_Y$ [MPa]</th>
<th>$\sigma_{UTS}$ [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ti6Al4V</td>
<td>4430</td>
<td>120</td>
<td>0.342</td>
<td>950</td>
<td>1020</td>
</tr>
<tr>
<td>AlSi10Mg</td>
<td>2670</td>
<td>70</td>
<td>0.33</td>
<td>240</td>
<td>460</td>
</tr>
<tr>
<td>Al 7075</td>
<td>2810</td>
<td>71.7</td>
<td>0.33</td>
<td>503</td>
<td>572</td>
</tr>
<tr>
<td>Stainless Steel 17-4 PH</td>
<td>7806</td>
<td>169.5</td>
<td>0.27</td>
<td>1172</td>
<td>1310</td>
</tr>
<tr>
<td>IN718</td>
<td>8150</td>
<td>160</td>
<td>0.284</td>
<td>780</td>
<td>1060</td>
</tr>
</tbody>
</table>

For tertiary structures titanium and aluminium are the most used materials due to the greatest strength to mass ratio compared with steel or the nickel alloy. [20]

According to Liu [21], the properties of the materials are different for the AM processed used, depending, for example, on the powder rate, laser power and scan velocity. This number of variables difficult the ability to fully characterize the effect of the process on final product performance. This requires robust and efficient modeling and computational approaches that can improve predictive capabilities in these processes.
2.2. Introduction to Elasticity

The following chapter is based on the work of Landau and Lifshitz [22] and Meyers and Chawla [23] and consists of a brief introduction to elasticity.

Elasticity is the ability of a body to resist a distorting influence or stress and to return to its original size and shape when the stress is removed.

Based on the fundamental physical principles of linear and angular momentum, it is possible to formulate the equation of motion (2.1), which is an expression of Newton’s second law.

\[ \nabla \sigma + X = \rho \ddot{u} \]  

(2.1)

Where \( \nabla \) is the divergence, \( \sigma \) is the stress tensor, \( X \) is the volumic force, \( \rho \) is the density and \( \ddot{u} \) the acceleration.

In a static elasticity problem, the dynamic component is null, \( \rho \ddot{u} = 0 \), obtaining the equation of motion for an elasticity problem in equation

\[ \nabla \sigma + X = 0 \]  

(2.2)

The strain of a body is characterized when the points of the body modify their relative position. In small displacement theory, the strain-displacement relations are given as follows

\[ \varepsilon = \frac{1}{2} \left[ \nabla u + (\nabla u)^T \right], \]  

(2.3)

and can be expressed as

\[
\begin{pmatrix}
\varepsilon_{11} \\
\varepsilon_{22} \\
\varepsilon_{33} \\
\varepsilon_{12} \\
\varepsilon_{13} \\
\varepsilon_{23}
\end{pmatrix} =
\begin{pmatrix}
\frac{\partial u_1}{\partial x_1} \\
\frac{\partial u_2}{\partial x_2} \\
\frac{\partial u_3}{\partial x_3} \\
\frac{\partial u_1}{\partial x_2} + \frac{\partial u_2}{\partial x_1} \\
\frac{\partial u_1}{\partial x_3} + \frac{\partial u_3}{\partial x_1} \\
\frac{\partial u_2}{\partial x_3} + \frac{\partial u_3}{\partial x_2}
\end{pmatrix},
\]  

(2.4)

where \( u_i \) is the displacement in the direction \( i \).

The constitutive law relating the applied force to the resulting deformation is Hooke’s Law, where the strain is assumed to be sufficient small that stress \( (\sigma) \) and strain \( (\varepsilon) \) depend linearly on each other, as in the following equation

\[ \{ \sigma \} = [K] \{ \varepsilon \}, \]  

(2.5)

where \([K]\) is the stiffness tensor.

For isotropic materials, materials characterized with independent properties of direction in space, the matrix form is represented as
\[
\begin{pmatrix}
\sigma_{11} \\
\sigma_{22} \\
\sigma_{33} \\
\sigma_{12} \\
\sigma_{13} \\
\sigma_{23}
\end{pmatrix}
= \frac{E}{(1 + \nu)(1 - 2\nu)}
\begin{pmatrix}
1 - \nu & \nu & \nu & 0 & 0 & 0 \\
\nu & 1 - \nu & \nu & 0 & 0 & 0 \\
\nu & \nu & 1 - \nu & 0 & 0 & 0 \\
0 & 0 & 0 & (1 - 2\nu)/2 & 0 & 0 \\
0 & 0 & 0 & 0 & (1 - 2\nu)/2 & 0 \\
0 & 0 & 0 & 0 & 0 & (1 - 2\nu)/2
\end{pmatrix}
\begin{pmatrix}
\varepsilon_{11} \\
\varepsilon_{22} \\
\varepsilon_{33} \\
2\varepsilon_{12} \\
2\varepsilon_{13} \\
2\varepsilon_{23}
\end{pmatrix},
\] (2.6)

where \( \nu \) is the Poisson’s ratio and \( E \) is the Young’s modulus.

These systems of equation, equation (2.2), (2.3) and (2.5), are the governing equations of the elasticity problem. A full description can be found in several literature, such as [22].

### 2.3. Distortion Energy Theory (Von Mises)

Distortion energy theory states that the material under multi-axial loading will yield when the distortional energy is equal to or greater than the critical value for the material. A full description can be found in several literature, such as [23] and [24]. That definition can be expressed by equation (2.7) in terms of the Von Mises stress \( \sigma_{VM} \) and the yield strength \( \sigma_y \).

\[
\sigma_{VM} \geq \sigma_y,
\] (2.7)

Where the Von Mises stress \( \sigma_{VM} \) is represented as

\[
\sigma_{VM} = \sqrt{\frac{(\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2 + 6(\sigma_{12}^2 + \sigma_{23}^2 + \sigma_{13}^2)}}{2}
\] (2.8)

### 2.4. Finite Element Method

The following chapter is based on the work of Silvestre & Araújo [25] and Reddy [26] and consists of a brief review of the Finite Element Method (FEM).

The Finite Element Method is a numerical method to solve differential equations. In FEM a given domain is discretized in subdomains, the finite elements, creating the finite element mesh, as seen in Figure 2.8.

![Figure 2.8 Representation of domain and sub-domain (finite elements) [26]](image)

Over each subdomain the governing equations are approximated, e.g. by a variational method, determining the element equations of each finite element.

The finite elements are connected by nodes and borders that should obey compatibility conditions, such as equal displacement. To determine the evolution between the nodes of the element, interpolation functions are applied in a way that the boundary conditions are satisfied.
After that discretization, the global system of equations can be found by assembling the equations of all the finite elements of the mesh. This system of equations can be represented as a system of matrices that is expected to be solved when the boundary conditions are imposed.

A general example of a finite element analysis is a dynamic structural finite element analysis, where a differential system of equations is applied, which can be represented by,

\[
[M][\ddot{u}] + [C][\dot{u}] + [K][u] = [F],
\]  

where \([F]\) is the nodal forces vector, \([u]\) is the nodal displacement vector, \([K]\) is the stiffness matrix, \([\dot{u}]\) is the nodal velocity vector, \([C]\) is the damping matrix, \([\ddot{u}]\) is the nodal acceleration vector and \([M]\) is the mass matrix.

In the fundamentals of solid mechanics, solids are assumed to be a continuum, without any voids, imperfections or impurities. For application of the linear elasticity theory, explained in section 2.2, displacements and deflections are assumed to be small when compared with the dimensions of the element. Finally, the mechanical properties of the material are constant, not varying with the applied load.

### 2.4.1. Static Analysis

The static structural finite element analysis is obtained when time dependency is not taken into account in the dynamic analysis, thus generating a linear system of equations, which can be represented by

\[
[K][u] = [F].
\]  

For 3D static elasticity problems it can be derived from the Cauchy law, where

\[
div(\sigma) + X = \rho^\text{Static} \frac{\partial^2 u}{\partial t^2} + X = 0
\]  

One usual technique is to use the residual method to obtain the corresponding weak formulation from

\[
\int \left( \text{div} \left( E_{ijkl} \frac{\partial u_k}{\partial x_l} \right) + X_i \right) v_i dV = 0
\]  

Using integration by parts/divergence theorem and adding boundary conditions. Knowing the boundary conditions the system of equations can be solved, obtaining the nodal displacement and force vectors.

### 2.4.2. Modal Analysis

To get the undamped natural frequencies of a structure in free vibration, the modal structural finite element analysis is reduced to the following problem

\[
[M][\ddot{u}] + [K][u] = \{0\},
\]  

obtained from the Lagrange’s equation,

\[
\frac{d}{dt} \left( \frac{\partial T}{\partial \dot{u}} \right) - \frac{\partial T}{\partial u} + \frac{\partial V}{\partial u} = 0,
\]  

obtained from the Lagrange’s equation,
where $T$ is the kinetic energy and $V$ is the potential energy defined by the following equations.

$$
\begin{align*}
T &= \frac{1}{2} [\dot{u}]^T [M] [\dot{u}] \\
V &= \frac{1}{2} [u]^T [K] [u]
\end{align*}
$$

(2.15)

$[M]$ and $[K]$ are symmetric matrices. For free vibrations, a harmonic response is assumed in the form of equation (2.16) i.e.

$$
\{u\} = \{\Phi\} e^{i\omega t}
$$

(2.16)

Introducing equation (2.16) in (2.13) one obtains an eigenvalues problem as it is represented in equation (2.17). The eigenvalues correspond to the natural frequencies ($\omega_i$) and the eigenvectors to the corresponding mode shapes ($\{\Phi\}_i$) are obtained from the following eigenvalue/eigenvector problem.

$$
([K] - \omega_i^2 [M]) \{\Phi\}_i = \{0\}
$$

(2.17)

2.4.3. Elements

When creating the mesh, both the topology and type of the finite elements are chosen taking into account the problem of choosing the adequate interpolation for the main variable, in this case the displacement field in the finite element. In this subchapter two types of elements are introduced: shell elements and 3D solid elements.

On ANSYS, SHELL181 and SOLID185 elements are used for the analysis of 3D problems. Those elements are described in the following subchapters and based on the information from [27].

2.4.3.1. Shell Elements

The shell elements are used to analyze thin structures, where one of the dimensions is very small compared to the other two, and are represented in Figure 2.9.

Figure 2.9 Shell elements schematics [25]

These elements have 6 degrees of freedom (DOF) in each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. The elements can use a bilinear or biquadratic interpolation, depending on the number of nodes used, e.g. for a rectangular element a bilinear interpolation correspond to a 4 nodes element.

The shell elements are formulated using one of the two different theories, Kirchhoff-Love and Reissner-Mindlin theories, explained in the following subsections based on information from [26].
Classical Plate Theory (Kirchhoff-Love)

The Classical Plate Theory is based on the Kirchhoff-Love hypotheses, which assumes that a straight line perpendicular to the plane of the plate is inextensible, remains straight, and rotates such that it remains perpendicular to the tangent to the deformed surface, as seen in Figure 2.10.

Figure 2.10 Undeformed and deformed geometries of an edge in Classical Plate Theory [26]

The displacement field of this theory is given by the following equations:

\[ u(x,y,z,t) = u_0(x,y,t) - z \frac{\partial w}{\partial x} \]  
(2.18)

\[ v(x,y,z,t) = v_0(x,y,t) - z \frac{\partial w}{\partial y} \]  
(2.19)

\[ w(x,y,z,t) = w_0(x,y,t) \]  
(2.20)

where \((u,v,w)\) denote the total displacements of the point \((x,y,z)\), and \((u_0, v_0, w_0)\) represent displacements of a point on the mid plane \((x,y,0)\) at time \(t\).

The strains in the Kirchhoff-Love theory are given by the following equations:

\[
\begin{bmatrix}
\varepsilon_{xx} \\
\varepsilon_{yy} \\
2\varepsilon_{xy}
\end{bmatrix} = \begin{bmatrix}
\frac{\partial u_0}{\partial x} - z \frac{\partial^2 w_0}{\partial x^2} \\
\frac{\partial v_0}{\partial x} - z \frac{\partial^2 w_0}{\partial y^2} \\
\frac{\partial u_0}{\partial y} + \frac{\partial v_0}{\partial x} - 2z \frac{\partial^2 w_0}{\partial x \partial y}
\end{bmatrix}, \text{and } \varepsilon_{zz} = \varepsilon_{zx} = \varepsilon_{zy} = 0. \tag{2.21}
\]

Shear Deformation Plate Theory (Reissner-Mindlin)

In the Reissner-Mindlin theory we relax the normality assumption of the classical plate theory (i.e. transverse normals may rotate without remaining normal to the midplane) as can be seen in Figure 2.11.
The displacement field of this theory is given by the following equations:

\[
\begin{align*}
     u(x, y, z, t) &= u_0(x, y, t) + z\phi_x(x, y, t) \\
     v(x, y, z, t) &= v_0(x, y, t) + z\phi_y(x, y, t) \\
     w(x, y, z, t) &= w_0(x, y, t)
\end{align*}
\]  

(2.22)

(2.23)

(2.24)

where \((u, v, w)\) denote the total displacements of the point \((x, y, z)\), \((u_0, v_0, w_0)\) represent displacements of a point on the mid plane \((x, y, 0)\), and \((\phi_x, \phi_y)\) designate the angles which the normal to the mid-surface makes with the \(z\) axis at time \(t\).

The strains in the Reissner-Mindlin theory are given by the following equations:

\[
\begin{bmatrix}
\varepsilon_{xx} \\
\varepsilon_{yy} \\
2\varepsilon_{xy}
\end{bmatrix} = \begin{bmatrix}
\frac{\partial u_0}{\partial x} + z\frac{\partial \phi_x}{\partial x} \\
\frac{\partial v_0}{\partial y} + z\frac{\partial \phi_y}{\partial y} \\
\frac{\partial w_0}{\partial x} + \phi_x \\
\frac{\partial w_0}{\partial y} + \phi_y
\end{bmatrix}, \text{ and } \varepsilon_{zz} = 0.
\]

(2.25)

**SHELL181 Elements**

SHELL181 is an element type of ANSYS® that is suitable for analyzing thin to moderately-thick shell structures and is depicted in Figure 2.12. The element uses a bilinear interpolation and is a 4 node element with 6 DOF at each node: translations and rotations.
This element allows for a degenerate triangular topology, but this option should only be used on filler elements in the generation of the mesh. The element is well-suited for linear, large rotation, and/or large strain nonlinear applications. In this case only the linear application is used.

2.4.3.2. 3D Solid Elements

For structures where the characteristic dimensions are similar (none of the characteristic dimensions is smaller than the others) or if geometric complexity requires it, a 3D solid element is typically used to model the structure. Those solid elements can have several shapes as those seen in Figure 2.13, where the most common are represented: the tetrahedral, prism and brick elements.

![Figure 2.13 3D elements schematics: linear and quadratic tetrahedral, prism, and brick elements [26]](image)

These elements have 3 DOF in each node: translations in the x, y, and z directions, and are developed from the solid mechanics elasticity theory. The elements can use either a linear or quadratic interpolation, depending on the number of nodes used, e.g. for a brick element a linear interpolation corresponds to an 8 nodes element.

**SOLID185 Elements**

SOLID185 is used for 3D modeling of solid structures and is depicted in Figure 2.14. The element uses a linear interpolation and is defined by 8 nodes having 3 DOF at each node: translations in the nodal x, y, and z directions.

![Figure 2.14 SOLID185 schematics [27]](image)

This element allows prism, tetrahedral, and pyramid degenerations when used in irregular regions. This characteristic makes SOLID185 generally suitable for modeling solid structures.
2.5. Topology Optimization

The following chapter is based on the work from Sigmund [3]-[28]; Andreassen et al. [29]; Liu and Tovar [8]; Bendsøe and Sigmund [2]; Fernandes [30] and is a brief review of the Topology Optimization problem and methods.

Topology Optimization (TO) is a computational material distribution method for synthesizing structures without any preconceived topology and shape [8]. This technique allows for the introduction of holes or cavities in structures, which usually results in great savings in weight or improvement of structural performance, such as stiffness or strength. In this thesis, the adopted method for the topology optimization problems is a density based approach explained in the next subchapter, known as Solid Isotropic Material with Penalization (SIMP).

2.5.1. Density Based Approach

This method is known as the “power-law approach” or the SIMP and consists on using constant material properties in each element used to discretize the design domain. This formulation is based on the pioneer work from Bendsøe and Kikuchi [1] on homogenization method.

The SIMP method is based on a relation between relative element density \( x_e \) and the element Young’s modulus \( E_a \), and is given by,

\[
E_a(x_e) = x_e^p E_0, \quad 0 < x_{\text{min}} \leq x_e \leq 1, \quad (2.26)
\]

where \( p \) is a penalization power, that should be higher than 1, usually \( 1.5 < p < 7 \), and \( E_0 \) is the Young’s modulus of the solid material or base material. \( x_{\text{min}} \) is the relative element density of the void material, which is higher than zero to avoid singularity of the finite element stiffness matrix, that occurs if all material is removed, so a hole is represented by elements with density of \( x_{\text{min}} \) or near.

A modified SIMP approach given by (2.27) can be used where Young’s modulus of the “void” or weak material \( E_{\text{min}} \) is defined, which is non-zero to avoid singularity of the finite element stiffness matrix.

\[
E_e(x_e) = E_{\text{min}} + x_e^p (E_0 - E_{\text{min}}), \quad x_e \in [0, 1] \quad (2.27)
\]

Using the finite element analysis theory, global stiffness matrix \( K \) is defined by equation (2.28) and the element stiffness matrix \( K_e \) defined by equation (2.29).

\[
K(x) = \sum_{e=1}^{N} K_e(x_e) \quad (2.28)
\]

\[
K_e(x_e) = E_e(x_e) k_e^0 \quad (2.29)
\]

\( k_e^0 \) is the element stiffness matrix for an element with a unitary Young’s modulus, which implies that this matrix is independent of the elastic modulus, and therefore independent of \( x_e \). This \( k_e^0 \) matrix depends on the element type and the Poisson’s ratio \( (\nu) \).
2.5.2. Optimization Problem Formulation

The topology optimization problem that is studied in this work is the minimum compliance problem, in which the objective is to find the design variables, i.e. the density distribution \( x \), that minimizes the structure’s deformation under the prescribed support and loading conditions. The compliance \( C \) can be defined as in equation (2.30), obtained by the inverse of the stiffness \( K' \).

\[
C(x) = F^T U(x), \quad (2.30)
\]

where \( F \) is the vector of nodal force and \( U \) is the vector of nodal displacement. In (2.31) the minimum compliance problem formulation is represented.

\[
\begin{aligned}
\min_x &: C(x) = F^T U(x) \\
\text{subject to:} & \ \left\{ \begin{array}{l}
\frac{V(x)}{V_0} = f \\
F = KU \\
0 < x_{\min} \leq x \leq 1
\end{array} \right.
\end{aligned} \quad (2.31)
\]

Where \( V(x) \) and \( V_0 \) are the available material volume and the volume of the design domain, respectively, and \( f \) is the prescribed volume fraction. So the problem has an equality constraint in the volume, \( \frac{V(x)}{V_0} = f \). \( F = KU \) is the state equation, but numerically it can be more efficient to use \( C = F^T U \). Condition \( 0 < x_{\min} \leq x \leq 1 \) imposes the lateral constraints in the design variables.

Equation (2.32) is obtained by developing the compliance with the definition of the nodal force.

\[
C(x) = F^T U \quad \Rightarrow \quad C(x) = U^T K U \quad (2.32)
\]

The global compliance can be decomposed in the sum of the element compliance \( c_e \) as in (2.33), where \( N \) is the number of elements used to discretize the design domain, \( u_e \) is the element displacement vector and \( K_e \) is the element stiffness matrix.

\[
C(x) = \sum_{e=1}^{N} c_e = \sum_{e=1}^{N} u_e^T K_e u_e \quad (2.33)
\]

2.5.3. Topology Optimization Sensitivity

The sensitivity of the objective function is given by derivation of compliance \( C \) with respect to the design variable \( x_e \) which can be seen in equation (2.34).

\[
\frac{\partial C(x)}{\partial x_e} = \frac{\partial}{\partial x_e} (F^T U) \quad (2.34)
\]

Knowing that the nodal force vector \( F \) is independent of the design variable \( x_e \), equation (2.35) is obtained with the solution for nodal displacement vector derivative.

\[
\frac{\partial F}{\partial x_e} = 0 \quad \Rightarrow \quad \frac{\partial K}{\partial x_e} U + K \frac{\partial U}{\partial x_e} = 0 \quad \Rightarrow \quad K \frac{\partial U}{\partial x_e} = -\frac{\partial K}{\partial x_e} U \quad (2.35)
\]

Applying the assumptions of equation (2.35) on the sensitivity of the objective function in equation (2.34), equation (2.36) is obtained, which relates the sensitivity with the derivative of the stiffness matrix.
\frac{\partial C(x)}{\partial x_e} = \frac{\partial}{\partial x_e} (F^TU) = U^T \frac{\partial U}{\partial x_e} = -U^T \frac{\partial K}{\partial x_e} U \quad (2.36)

On the same way that the compliance in equation (2.33) is decomposed, the global sensitivity can also be decomposed on the sum of the element sensitivity obtaining the equation (2.37).

\frac{\partial C(x)}{\partial x_e} = \sum_{e=1}^{N} \frac{\partial c_e}{\partial x_e} + \sum_{e=1}^{N} -u_e \frac{\partial E(x_e)}{\partial x_e} k_e u_e \quad (2.37)

Equation (2.38) is obtained by having the element stiffness matrix given in equation (2.29) into account.

\frac{\partial C(x)}{\partial x_e} = \sum_{e=1}^{N} -u_e \frac{\partial E(x_e)}{\partial x_e} k_e u_e \quad (2.38)

Applying equation (2.27) in (2.38) the following sensitivity equation is obtained.

\frac{\partial C(x)}{\partial x_e} = \sum_{e=1}^{N} -u_e \int [p_{\text{max}} - (E_0 - E_{\text{min}})k_e u_e] \quad (2.39)

### 2.5.4. Optimization Algorithm - Optimality Criteria Method

Optimality Criteria (OC) methods are one of the approaches to solve non-linear programming problems, such as minimum compliance. Although there are others approaches like Sequential Quadratic Programming (SQP) and Method of Moving Asymptotes (MMA), only the standard OC-method is going be explained in this work.

The OC-method uses the Karush-Kuhn-Tucker (KKT) conditions to find a solution for the optimization problem. The application of the KKT conditions are based on the works in [28]. For the problem formulation described in equation (2.31), the following Lagrangian function is obtained

\[ L(x) = C(x) + \lambda \left( \frac{V(x)}{V_0} - f \right) + \mu(KU - F) + \alpha(x_{\text{min}} - x) + \beta(x - 1), \quad (2.40) \]

where \( \lambda, \mu, \alpha \) and \( \beta \) are Lagrangian multipliers corresponding to the constraints of the problem formulation. Making the stationarity of the Lagrangian function with respect to the design variable \( x_e \) gives the optimality criteria in equation (2.41).

\[ \frac{\partial L}{\partial x_e} = \frac{\partial C}{\partial x_e} + \lambda \frac{\partial V}{\partial x_e} - \alpha + \beta = 0 \quad (2.41) \]

If the inequality constraints \( x_{\text{min}} \leq x \leq 1 \) are inactive, the optimality criteria is given by equation (2.42), because \( \alpha \) and \( \beta \) Lagrangian multipliers are null.

\[ \frac{\partial L}{\partial x_e} + \lambda \frac{\partial V}{\partial x_e} = 0 \quad (2.42) \]

This optimal condition can be expressed as \( B_e = 1 \) in the equation (2.43) where \( \lambda \) is a Lagrangian multiplier that can be found by a bi-sectioning algorithm.
The OC uses a heuristic updating scheme for the design variables that can be expressed in

\[ B_e = \frac{-\frac{\partial C}{\partial x_e}}{\lambda \frac{\partial V(x)}{\partial x_e}} \]  

(2.43)

The termination criteria is defined when the maximum number of iterations \( \text{MaxIter} \) is reached, without ensuring of optimal criteria, or when a convergence termination criteria \( \text{ConvTer} \) value is obtained. In this thesis two different convergence termination criteria. The first is defined by the difference between the new design variable \( x_e^{\text{new}} \) and the old design variable \( x_e^{\text{old}} \), when the difference is lower than the tolerance value \( \text{ConvTer.Abs} \) the criteria of termination is reached, as expressed in equation (2.45).

\[ \| x_e^{\text{new}} - x_e^{\text{old}} \|_{\infty} \leq \text{ConvTer.Abs} \]  

(2.45)

The second convergence termination criteria \( \text{ConvTer.Rel} \) in this thesis is defined by a relative difference between the compliance values of the new design \( c^{\text{new}} \) and the compliance value of the old design \( c^{\text{old}} \) for the TO results. The termination criteria is defined when the expressed of equation (2.46) is obtained.

\[ \frac{|c^{\text{new}} - c^{\text{old}}|}{c^{\text{old}}} \leq \text{ConvTer.rel} \]  

(2.46)

2.5.5. Filters Functions

One of the problems of the topology optimization is the checkerboard effects. This effect is demonstrated in Figure 2.15 on a topology optimization analysis of a MBB-Beam, and is defined as regions with alternating void and solid elements ordered in a checkerboard-like fashion [28].

Figure 2.15 Checkerboards Effect in a MBB-Beam Topology Optimization analysis [28]

Some references to the meaning of the checkerboard can be seen in Neves et al. [32] where the author mention that checkerboard as buckling performance equal to zero.

A way to overcome this problem is to interpret this checkerboards effect as regions with an intermediate uniform relative density on the post-processing phase.
A more complex way to avoid numerical instabilities, such as checkerboards effects, and ensure existence of solutions on topology optimization is the insertion of filter techniques. Two of those techniques are presented in this section: a density filter and a sensitivity filter.

### 2.5.5.1. Density Filter

The basic density filter gives a new relative density for the elements that can be by

\[ x_e = \frac{\sum_{f=1}^{N_e} H_{ef} v_f x_f}{\sum_{f=1}^{N_e} H_{ef} v_f} \]  

(2.47)

where \( N_e \) is the neighborhood of the element \( x_e \) and can be defined in equation (2.48), \( x_f \) is the relative density of those elements and \( v_f \) is the volume of those elements.

\[ N_e = \{ f: \text{dist}(e, f) \leq r_{\text{min}} \} \]  

(2.48)

Here the operator \( \text{dist}(e, f) \) is the distance between the element \( x_e \) and the element \( x_f \), and \( r_{\text{min}} \) is the filter size, given as input data. A graphic representation of the neighborhood of the element \( x_e \) can be seen in Figure 2.16.

![Figure 2.16 Neighborhood \((N_e)\) of element \(x_e\) and filter size \((r_{\text{min}})\)](image)

\( H_{ef} \) is the height factor and is a function of \( r_{\text{min}} \) and \( \text{dist}(e, f) \), obtained by

\[ H_{ef} = r_{\text{min}} - \text{dist}(e,f), \quad \text{with } f \in N_e \]  

(2.49)

### 2.5.5.2. Sensitivity Filter

The sensitivity filter uses a similar process to the density filter. The difference is that the filter is imposed on the sensitivity function defined in (2.38). The sensitivity filter function is defined in equation (2.50).

\[ \frac{\partial C(x)}{\partial x_e} = \frac{\sum_{f=1}^{N_e} H_{ef} x_f \frac{\partial c_f}{\partial x_f}}{x_e \sum_{f=1}^{N_e} H_{ef}}, \quad 0 < x_{\text{min}} \leq x_e \leq 1 \]  

(2.50)

The difference from the terms of the equation (2.47) is \( \frac{\partial c_f}{\partial x_f} \), which is the sensitivity of the neighborhood element \( x_f \). To avoid singularities the element density should be higher than zero, which is the reason for the definition of a minimum relative density \( x_{\text{min}} \) \((0 < x_{\text{min}} \leq x_e \leq 1)\).
2.6. Topology Optimization and Additive Manufacture

This chapter establishes an association between the Additive Manufacture techniques and the Topology Optimization analysis, based on the work of D. Brackett, Ashcroft, & Hague [4] and Oropallo & Piegl [33].

Additive Manufacture technologies provide a strong advantage when compared to conventional (e.g. subtractive) manufacturing processes, which is that it can provide more freedom in the manufacture. This characteristic is interesting to the topology optimization designs allowing to create more complex geometries with good structural performances.

Even though the AM technologies have less constrains that conventional methods, they still have some restrictions such e.g. build accuracy, surface finish and z-direction (direction perpendicular to the plane of material addition) mechanical properties. However, one of the more relevant manufacturing constraint is the support structure requirement. These structural supports are used in manufacture proceedings, being removed after manufacture. Examples can be seen in Figure 2.17.

![Figure 2.17 Additive Manufacturing Structure with Supports](image)

One of the goals of AM processes is to reduce the amount of support material required. The main reasons for that, according to [4], is the reduction of wasted material associated to support materials, the need to set up the STL model to the building requirements for the support materials and the post treatment of removal of the supports that requires a geometric restriction of hand/tool access to the support materials.
3. Methodology

In this section the methodology developed in the thesis to answer to the challenger is explained. The first section refers to the academic problems approach, where benchmarking of the topology optimization problems occurs. Secondly, the method of analysis of the tertiary structure is described. Finally, the last subchapter is dedicated to explaining the topology optimization methodology applied on this tertiary structure.

3.1. Academic Problems for Verifications

Before performing the topology optimization analysis of our tertiary structure it was considered necessary to test and to verify our available optimization software with academic problems. The selected academic problems consist on classic cases of a MBB-Beam and a Cantilever-Beam in 2D and 3D, depicted in Figure 3.1.

![Figure 3.1 MBB-Beam (Right) and Cantilever-Beam (Left) 2D problems](image)

The adopted approach for each academic case is an optimization study (mesh size, volume fraction, design domain, load conditions, etc.) comparing the literature ([3] and [8]) and MatLab optimization, using TopOpt2D [3] and TopOpt3D [8], solutions with the ANSYS Wb v17 - Topology Optimization ACT solution.

For that purpose, a design domain - defined by the mesh size since the element represents the units of measurement (1x1x1) - and boundary conditions (displacements and load conditions) are defined on MatLab and ANSYS software, similarly to the literature [3] and [8]. The material properties attributed to MatLab and ANSYS are the same in order to validate the comparison.

The optimization parameter attributed to the software are the penalization power, minimum relative density, volume fraction, convergence termination criteria, maximum number of iterations and filter size.

The penalization power \((p)\) and the minimum relative density \(\left(x_{\text{min}}\right)\) are explained in section 2.5.1.

Volume fraction \((f)\), or volume constraint, representing the equality constraint in the volume, \(\frac{V(x)}{V_0} = f\), in equation (2.31). This fraction (or percentage) defines the initial distribution of material, as a uniform density of \(f\) in all the domain.
Convergence termination criteria (\textit{ConvTer}) is defined in section 2.5.4 and can have two different approaches, equation (2.45) or (2.46). In MatLab optimizations, the convergence criteria is defined by the absolute difference between the relative densities values, equation (2.45). In the case of ANSYS optimization, the criteria is defined by a relative difference between compliance values, equation (2.46). The fact that the convergence criteria is different for MatLab and ANSYS can be a factor for different results in the convergence compliance.

The maximum number of iterations (\textit{MaxIter}) is a parameter for the MatLab optimization using TopOpt3D [8] and the ANSYS optimization, and it is a parameter that stops the TO, when the convergence termination criteria is not reach, without ensuring the optimal criteria.

Filter size ($r_{\text{min}}$) is explained in section 2.5.5.1 and represents a way to avoid numerical instabilities, such as checkerboards effects. The filter size application imposes a minimum length scale of $r_{\text{min}}$, in a way that, with the increase of the filter size, the results are less detailed, i.e., a more compact design and with less holes (but holes with higher area). In the case of ANSYS a filter size is not imposed, obtaining a more detailed solutions. The ANSYS software uses a post processing tool (smoothing), based on the average relative density on nodes, to display lower relative densities to prevent the checkerboard effect, as can be seen in Figure 3.2.

![Figure 3.2 MBB-Beam Half topology optimization solutions obtained from ANSYS Wb with a mesh of 60x20, $E = 1\text{Pa}$, $\nu = 0.3$, $p = 3$, $r_{\text{min}} = 0.001$, $\text{ConvTer Rel} = 0.01\%$, $\text{MaxIter} = 500$, $f = 50\%$ and a TO post processing with relative density higher than 0.4 a) without smoothing and b) with smoothing](image)

After the topology optimization, a post processing of the TO results in ANSYS is performed, where a relative density is defined for which all the densities with a higher value are displayed.

From these results it is possible to establish comparisons between the results obtained with software and the academic results available in the literature.

### 3.2. Tertiary Structure Analysis

The method of analysis of the tertiary structure uses a traditional FEM analysis methodology. Firstly the structure is defined in Computer Aided Design (CAD). In this thesis the Solid Edge software is used, and the design and material of the tertiary structure are imposed. The structural requirements and the loads and boundary conditions of the structure are also defined.

After structure definition in CAD, the design is converted to a neutral format, STEP file (.stp), and imported for the FEM pre-processing stage, in ANSYS Wb v17, where loads and boundary conditions are applied, and the elements and mesh are chosen, thus defining the mesh model. In this stage a finite element convergence study is performed in order to prevent errors associated with mesh discretization.
After FEM pre-processing, the static or modal analysis are executed and the results are verified in order to assess its compliance with the structural requirements.

In a traditional FEM analysis, represented in Figure 3.3, if the static and modal results do not satisfy the requirements, the design is changed and the FEM model remade until the results are compliant with the requirements.

![Figure 3.3 Traditional FEM optimization analysis, based on [34]](image)

In this thesis the analysis of the tertiary structure is performed in order to have a preliminary idea of the results for further comparison with the optimized structure’s data.

### 3.3. Tertiary Structure Topology Optimization Analysis

The typical method of topology optimization analysis for structures is depicted in Figure 3.4, based on the work of Brackett [34] and the case study from Altair® [5].

![Figure 3.4 Topology Optimization typical method for component development, based on [5], [34]](image)

In this method an initial design is presented, this initial design can be a predefined structure design where the TO is going to be performed in order to optimize its characteristics, such as reducing mass, or the initial design can be a design region, defined by the geometric boundaries of the structure, in order to apply the TO to achieve the optimal design.

In this thesis an alternative approach is proposed in order to optimize the tertiary structure of the satellite, depicted in Figure 3.5.

![Figure 3.5 Topology Optimization Analysis: 2D and 3D TO Analysis](image)

The difference in this approach is that a 2D and 3D topology optimization analysis are performed, instead of a single 3D optimization, as proposed before in the typical approach.

#### 3.3.1. 2D Initial Design

The first step of this method is to define a 2D initial design for the tertiary structure. This design is defined by a 2D representation of the tertiary structure geometric boundaries that delimitate the problem, respecting the domain restrictions, such as the position of constrains, and overdesigning.
the rest of the domain. This stage is performed in a CAD software. For this thesis the software provided by Active Space Technologies was Solid Edge software.

### 3.3.2. 2D TO Analysis

After the 2D design is defined in CAD, the design is imported, in STEP format, to the ANSYS Wb v17, also provided by Active Space Technologies®, where the topology optimization is going to be performed. In ANSYS, the loads and boundary conditions are applied, the material properties are assigned and the elements and mesh are chosen, defining the mesh model.

After the mesh model is prepared, the topology optimization parameter should be defined, where the objective function, in this thesis, is the minimum compliance, described in section 2.5.2. One of these parameters is the design domain, which consists on the portion of the model where the topology optimization is going to be made. The other parameter is the design exclusions, which are the elements of the design domain that are not optimized.

Other parameters for the topology optimization must be defined, as in section 3.1, such as the penalization power ($p$), volume fraction ($f$), maximum number of iterations ($MaxIter$), relative convergence termination criteria ($ConvTer_Rel$) and minimum relative density ($x_{min}$).

### 3.3.3. Redesign 3D Initial Design

After the 2D topology optimization is performed, a post processing of the TO results in ANSYS is performed, where a relative density is defined for which all the densities with a higher value are displayed. This post processing should produce a viable design, in terms of structural integrity, and a STL file is exported with that design proposal for further processing, as explained in [35].

The objective of this 2D topology optimization is to obtain an initial design for the 3D topology optimization analysis. This approach results in a preconceived design for the 3D TO, which contradicts the principal of the topology optimization. Nevertheless the computational time cost is reduced for the 3D optimization and the preconceived design is based on a 2D optimization, making it an engineering solution to the problem.

By importing the STL file of the 2D optimization to the CAD software it is possible to interpret the 2D design to a 3D design, by extrapolating the design on a third dimension, perpendicular to the 2D plane. The result of this interpretation is the initial design for the 3D topology optimization analysis, and can be exported in a STEP format for the ANSYS software.

### 3.3.4. 3D TO Analysis and Redesign

After the 3D initial design is defined the same process of topology optimization is repeated, as in the 2D analysis, with the definition of the mesh model with the loads and boundary conditions, and the definition of the topology optimization parameter, such as the design domain, design exclusions, penalization power ($p$), volume fraction ($f$), maximum number of iterations ($MaxIter$), relative convergence termination criteria ($ConvTer_Rel$) and minimum relative density ($x_{min}$).
After the 3D topology optimization is performed, a post processing of the TO results in ANSYS is performed, as in the 2D optimization, creating a STL file that is interpreted in the CAD software to obtain a new tertiary structure design.

3.3.5. Analysis and Requirements Verification

After redesigning the structure, a similar method of the section 3.2 is adopted, where the FEM model is prepared (elements, mesh and loads and boundary conditions), and a static and modal analysis is performed on the model.

The final step is the requirements verification, which compares the static and modal results with the structural requirements, as in section 3.2, but with the new tertiary structure design.
4. Verifications with Academic Problems

In this chapter two classical topology optimization problems, the MBB-Beam and the Cantilever-Beam, are simulated in two-dimensions (2D) and three-dimensions (3D). These academic simulations are used to compare the literature solutions from [3], [8], the results from the MatLab codes from the same articles, and the results from the software used on this thesis, ANSYS Wb v17 - Topology Optimization ACT.

4.1. 2D Academic Problems

2D analysis uses a material with a Young’s modulus of 1 Pa and a Poisson’s ratio of 0.3. The mesh model is defined with shell elements of the type SHELL181 on ANSYS. Dimensions are defined with length (L), height (H), and thickness (t), (LxHxt), and each element represents the units of measurement (1x1x1). In order to simulate these 2D academic problems, the thickness is unitary (t = 1 element), and the mesh is defined by elements in the length and in the height, LxH.

The topology optimization parameters for ANSYS Wb are defined in Table 4.1 with the penalization power ($p$), maximum number of iterations ($MaxIter$), relative convergence termination criteria ($ConvTer_Rel$) and minimum relative density ($x_{\text{min}}$).

<table>
<thead>
<tr>
<th>$p$</th>
<th>$MaxIter$</th>
<th>$ConvTer_Rel$ [%]</th>
<th>$x_{\text{min}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>500</td>
<td>0.01</td>
<td>0.001</td>
</tr>
</tbody>
</table>

The post processed topology optimization solution from ANSYS is a design with a relative density higher than 0.4 in order to obtain similar optimizations results. Also in ANSYS a filter size ($r_{\text{min}}$) is not imposed, obtaining more detailed solutions, i.e., a design with more holes of lower area.

For the MatLab code from [3], the input variables for the optimization are defined in Table 4.2 with the penalization power ($p$), filter size ($r_{\text{min}}$), absolute convergence termination criteria ($ConvTer_Abs$) and minimum relative density ($x_{\text{min}}$).

<table>
<thead>
<tr>
<th>$p$</th>
<th>$r_{\text{min}}$</th>
<th>$ConvTer_Abs$</th>
<th>$x_{\text{min}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>1.5</td>
<td>0.01</td>
<td>0.001</td>
</tr>
</tbody>
</table>

The parameters that set apart the different optimization problems are the boundary conditions of the model, the size of the mesh and the volume fraction ($f$).
4.1.1. Reproducing the MBB-Beam topology optimization solution

The Messerschmitt-Bölkow-Blohm (MBB) beam is one of the most common study cases for topology optimization. It is composed of a beam with a fixed and pinned constrain on the lower corners and a force (F) of 1 Newton in the upper-middle section of the beam. Due to symmetry properties in static case this model can be decomposed in the middle, with a pinned constrain, as depicted in Figure 4.1.

![Figure 4.1 MBB-Beam Model and MBB-Beam Model simplification domain and boundary conditions for the static case](image)

4.1.1.1. MBB-Beam comparison with literature

The first topology optimization analysis is executed on the MBB-Beam model with a 60x20 mesh, representative of LxH, and a volume constrain of 50%, corresponding to an initial uniform density distribution of 0.5. The solutions obtained with ANSYS Wb, for a post processed TO design with a relative density higher than 0.4, are compared to the solutions from the literature [3] and MatLab optimization, running TopOpt2D [3].The results are depicted in Figure 4.2.

![Figure 4.2 MBB-Beam Half topology optimization solutions obtained: a) from the literature [3], b) from MatLab Optimization; and c) from ANSYS Wb with a mesh of 60x20 and Volume constrain of 50%](image)

As expected, the results from the literature and from MatLab are practically the same. The results from ANSYS are close in its core if only the highest relative density zones are analyzed. The lowest relative density zones are displayed in the ANSYS results due to a checkerboard effect. Also the lack of filter size constrain in ANSYS, causing those checkerboard effect, gives a more detailed structure design.

Graphics of the topology optimization objective function of a half model of a MBB-Beam, as a function of the number of iterations, can be seen in Figure 4.3, for both the MatLab optimization and ANSYS Wb optimization. The objective function of this analysis is the structural compliance in Newton-meter. In both cases, a convergence of the results can be observed. The ANSYS Wb optimization requires a lesser number of iterations to obtain the convergence. This effect is justified due to the stop criteria, which in MatLab is the absolute difference between relative densities, $Conv\_Ter\_Abs = 0.01$, and in ANSYS is a relative difference between compliances, $Conv\_Ter\_Rel = 0.01\%$. The results of compliance for a volume of 0.5 are very similar in both cases, 206.11 Nm for ANSYS and 203.30 Nm for MatLab.
4.1.1.2. MBB-Beam Volume Reduction

One of the most important parameters of the topology optimization analysis is the volume fraction ($f$). In Figure 4.4 the MBB-Beam model with a 75x25 mesh, representative of LxH, for a volume constrain of 70%, 50% and 30%, corresponding to an initial uniform density distribution of 0.7, 0.5 and 0.3, solutions from MatLab optimization and from ANSYS Wb are demonstrated.
When comparing the MatLab and ANSYS solutions it becomes clear that for the lowest volume constrain the results are closer. This effect occurs because, for a higher volume constrain, a checkerboard effect forms, creating an intermediate relative density region, displayed in green. This effect occurs, once more, due to the lack of filter size \( r_{\text{min}} \) in the ANSYS optimizations.

### 4.1.1.3. MBB-Beam Mesh Resolution

Another important parameter of the topology optimization analysis is the mesh resolution. The study of mesh dependency can be seen in Figure 4.5 where the MBB-Beam half model with a volume constrain of 50%, corresponding to an initial uniform density distribution of 0.5, is optimized in MatLab and ANSYS for a mesh resolution \((L \times H)\) of 30x20, 60x40, 90x60 and 120x80.

\[ c = 47.1 \text{ Nm} \]
\[ c = 49.1 \text{ Nm} \]
\[ C = 46.8 \text{ Nm} \]
\[ C = 50.0 \text{ Nm} \]
\[ C = 47.3 \text{ Nm} \]
\[ C = 49.6 \text{ Nm} \]
\[ C = 47.7 \text{ Nm} \]
\[ C = 50.2 \text{ Nm} \]

*Figure 4.5 MBB-Beam topology optimization solutions with compliance \(C\) for mesh resolution study obtained from MatLab Optimization (left pictures) and from ANSYS Wb (right pictures) with a Volume constrain of 50% and a mesh of 30x20 (top), 60x40 (middle top), 90x60 (middle bottom) and 120x80 (bottom) respectively*
The solutions from MatLab and ANSYS are very similar and it is possible to see that for a more refined mesh resolution the solution becomes more detailed, i.e., a structure with more holes of lower area. A possible solution for this problem is to introduce a higher filter size ($r_{\text{min}}$) for MatLab optimization, reducing the chattering effect.

**4.1.1.4. MBB-Beam Distributed Load**

In this section a variation of the MBB-Beam problem is studied. The difference is on the application of the force. Instead of applying the force to a single point, a distributed load applied on the bottom section of the model is used, with a total force (F) of 81 N, corresponding to a force of 1 N per node, as depicted in Figure 4.6.

![Figure 4.6 MBB-Beam Model domain and boundary conditions for distributed load.](image)

The solution for this topology optimization problem in MatLab and ANSYS with a mesh resolution of 80x50 (LxH) and a volume constrain of 30%, corresponding to an initial uniform density distribution of 0.3, is represented in Figure 4.7. Similar results can be seen except that a filter size ($r_{\text{min}}$) constrain is not imposed in the ANSYS solution, thus obtaining a more detailed structure.

![Figure 4.7 MBB-Beam with distributed load topology optimization solutions obtained: a) from MatLab Optimization; and b) from ANSYS Wb with a mesh of 80x50 and Volume constrain of 30%](image)

Graphics of the topology optimization objective function of a MBB-Beam with distributed load as a function of the number of iterations can be seen in Figure 4.8, for both MatLab optimization and ANSYS Wb optimization. In both cases, a convergence of the compliance results can be observed for a volume of 30%, with results of $2.24 \times 10^5 \text{ Nm}$ for ANSYS and $1.35 \times 10^5 \text{ Nm}$ for MatLab. These compliance results have the same order of magnitude, the difference in the value can be explained due to a more detailed design in the ANSYS software, due to the lack of $r_{\text{min}}$. 

31
4.1.2. Reproducing the Cantilever-Beam topology optimization solution

Another classic case for the topology optimization analysis is the Cantilever-Beam. This is composed of a beam with a clamped constrain on one side and a force of 1 Newton in the other side edge of the beam, as depicted in Figure 4.9.

For the most of the Cantilever-Beam topology optimization analysis the filter size ($r_{\text{min}}$) in MatLab is going to be 1.2 instead of 1.5 as in the previous section, and no filter size constrain in ANSYS as in the previous section.

4.1.2.1. Cantilever-Beam comparison with literature

The first topology optimization analysis performed for the Cantilever-Beam model is for a model with a 32x20 mesh (LxH) and a volume constrain of 40%, corresponding to an initial uniform density distribution of 0.4. The solutions obtained with ANSYS Wb are compared to the solutions from the literature [3] and MatLab optimization. The results are depicted in Figure 4.10.
Figure 4.10 Cantilever-Beam topology optimization solutions obtained: a) from literature [3]; b) from MatLab Optimization; and c) from ANSYS Wb with a mesh of 32x20 and Volume constrain of 40%

As expected, the results from literature and MatLab optimization are similar and the result from ANSYS can be considered near to the other two.

Graphics of the topology optimization objective function of the Cantilever-Beam as a function of the number of iterations can be seen in Figure 4.11, for both the MatLab and ANSYS Wb optimization. In both cases, a convergence of the compliance results can be observed for a volume of 40%, with results of 59.6 Nm for ANSYS and 59.5 Nm for MatLab.

Figure 4.11 Cantilever-Beam topology optimization objective function convergence (compliance) obtained from MatLab Optimization (Top) and from ANSYS Wb (Bottom) with a mesh of 32x20 and Volume constrain of 40%

4.1.2.2. Cantilever-Beam Mesh Resolution

The study of mesh dependency can be seen in Figure 4.12 where a Cantilever-Beam model with a volume constrain of 40%, corresponding to an initial uniform density distribution of 0.4, is optimized in the MatLab and ANSYS for a mesh resolution (LxH) of 40x20, 80x40 and 160x80. For this analysis the filter size ($r_{\text{min}}$) of MatLab is 1.5.
The TO solutions from MatLab and ANSYS are very similar and it is possible to see that for a more refined mesh resolution the solution becomes more detailed, as seen for the MBB-Beam mesh resolution in the previous section.

![Figure 4.12 Cantilever-Beam topology optimization solutions with compliance (C) for mesh resolution study obtained from MatLab Optimization (left pictures) and from ANSYS Wb (right pictures) with a Volume constraint of 40% and a mesh of 40x20, 80x40 and 160x80 respectively](image)

4.1.2.3. Cantilever-Beam with Multiple Load Cases

In this section a variation of the Cantilever-Beam problem is studied. The variation differs from the classical example on the application of the force and the number of load cases, which instead of being only a load case with a single force applied on the bottom corner, is the combination of two load cases with separated times of application. The first load case has a force of 1 N applied in the top corner and the second load case the same force applied in the bottom corner, as depicted in Figure 4.13.

![Figure 4.13 Cantilever with Multiple Load Cases Model domain and boundary conditions](image)
The solution for this topology optimization problem in the literature [3], MatLab and ANSYS with a mesh resolution of 30x30 (LxH) and a volume constrain of 40%, corresponding to an initial uniform density distribution of 0.4, is represented in Figure 4.14. The results are similar for the literature and MatLab, and near with ANSYS Wb, only a small checkerboard effect on the right side, represented by lower relative densities (in green).

![Image](image1)

**Figure 4.14 Cantilever with Multiple Load Cases topology optimization solutions obtained a) from literature [3]; b) from MatLab Optimization; and c) from ANSYS Wb with a mesh of 30x30 and Volume constrain of 40%**

Graphics of the topology optimization objective function of the Cantilever-Beam with multiple load cases in function to the number of iteration can be seen in Figure 4.15, for both the MatLab and ANSYS Wb optimization. The compliance results for a volume of 40% are 61.2 Nm for ANSYS and 61.3 Nm for MatLab.

![Image](image2)

**Figure 4.15 Cantilever-Beam with multiple load cases topology optimization objective function convergence (compliance) obtained from MatLab Optimization (Top) and from ANSYS Wb (Bottom) with a mesh of 30x30 and Volume constrain of 40%**
4.1.2.4. Cantilever-Beam with Hole

Another variation of the Cantilever-Beam problem is the introduction of a hole in the beam, as depicted in Figure 4.16.

On this topology optimization analysis the Cantilever-Beam with a mesh of 45x30 (LxH) and a volume constrain of 50%, corresponding to an initial uniform density distribution of 0.5, has a hole with radius of R=15 elements and centered in (X=15, Y=15) elements with reference to the bottom left corner. Figure 4.17 represents the solutions from the literature [3], MatLab Optimization and ANSYS Wb where it is possible to see similar results, as expected. In ANSYS software the hole is applied before the mesh discretization, obtaining a better defined hole on the solution.

Graphics of the topology optimization objective function of the Cantilever-Beam with multiple load cases as a function of the number of iteration can be seen in Figure 4.18, for both the MatLab and ANSYS Wb optimization. The compliance results for a volume of 50% are 67.2 Nm for ANSYS and 52.1 Nm for MatLab.
Another variation of the Cantilever-Beam problem is the symmetric model. The variation differs from the classical example in the application of the force, which instead of being applied in the bottom corner the force of 1 N is applied in the middle of that beam side. This change in the force application gives a characteristic symmetry to the model, as depicted in Figure 4.19.

In this section the influence of the volume fraction \( f \) on the topology optimization of this symmetric Cantilever-Beam is studied. The results from MatLab and ANSYS optimizations for a symmetric Cantilever-Beam with a mesh resolution of 90x50 (LxH) and volume constrain of 70%, 50% and 30%, corresponding to an initial uniform density distribution of 0.7, 0.5 and 0.3, are depicted in Figure 4.20.
The optimizations have a symmetry, as expected, and for a lower volume constrain the results from MatLab and ANSYS are more alike. This effect occurs because, for a high volume constrain, a checkboard effect is observed in ANSYS. Another reason for the different results is the lack of a filter size ($r_{\text{min}}$) constrain in the ANSYS optimization that permits the creation of a more precise structure.

![Cantilever with symmetry topology optimization solutions with compliance (C) for Volume Constraint study obtained from MatLab Optimization (left pictures) and from ANSYS Wb (right pictures) with a mesh of 90x50 and Volume constrain of 70% (top), 50% (middle) and 30% (bottom) respectively](image)

**4.1.2.6. Cantilever-Beam Aspect Ratio**

Another topology optimization analysis that can be performed for the symmetric Cantilever-Beam is the aspect ratio study. This study consists of increasing the length of the model and studying the optimization results of this change.

In Figure 4.21 is presented the results of this study for the topology optimization analysis from MatLab and ANSYS for a volume constrain of 40%, corresponding to an initial uniform density distribution of 0.4, and a mesh resolution (LxH) of 50x60, 110x60 and 180x60. The solutions are very similar for the first aspect ratio and similar in its core for the last aspect ratio. The differences for the last aspect ratio are more significant due to a checkboard effect and the lack of a filter size ($r_{\text{min}}$) constrain in the ANSYS optimization, producing the same effects as discussed in the previous sections.
4.2. 3D Academic Problems

On the 3D analysis, the material has the same mechanical properties as the one from the 2D analysis, with a Young’s modulus of 1 Pa and a Poisson’s ratio of 0.3. The mesh model is defined with 3D solid elements of the type SOLID185 on ANSYS. The dimensions are defined with length (L), height (H), and thickness (t), (LxHxt), and each element represents the units of measurement (1x1x1).

The topology optimization parameters for ANSYS Wb are defined in the Table 4.3 with the penalization power ($p$), maximum number of iterations (MaxIter), relative convergence termination criteria (ConvTer_Rel) and minimum relative density ($x_{\text{min}}$).

<table>
<thead>
<tr>
<th>$p$</th>
<th>MaxIter</th>
<th>ConvTer_Rel [%]</th>
<th>$x_{\text{min}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>500</td>
<td>0.01</td>
<td>0.001</td>
</tr>
</tbody>
</table>

The post processed topology optimization solution from ANSYS is a design with a relative density higher than 0.5 in order to obtain similar optimizations results. A filter size ($r_{\text{min}}$) constrain is not applied in ANSYS in order to obtain more precise design. As explained before, this will result in a checkboard effect in the ANSYS results, which is interpreted as a lower relative densities zones on the post processed solution.
For the MatLab TopOpt3D code from [8], the input variables for the optimization are defined in Table 4.4 with the penalization power \((p)\), filter size \((r_{\text{min}})\), absolute convergence termination criteria \((\text{ConvTer.Abs})\) and minimum Young's modulus \((E_{\text{min}})\).

### Table 4.4 Topology Optimization Parameters for 3D Academic Problems analysis with MatLab

<table>
<thead>
<tr>
<th>(p)</th>
<th>(r_{\text{min}})</th>
<th>(\text{ConvTer.Abs})</th>
<th>(E_{\text{min}} [\text{Pa}])</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>1.5</td>
<td>0.01</td>
<td>(1 \times 10^{-9})</td>
</tr>
</tbody>
</table>

The minimum Young's modulus \((E_{\text{min}})\) of \(1 \times 10^{-9}\) is related to the minimum relative density \((x_{\text{min}})\) of 0.001, with equation (2.26), for a penalization power \((p)\) of 3.

The parameters that set apart the different optimization problems are the boundary conditions of the model, the size of the mesh and the volume fraction \((f)\).

### 4.2.1. Reproducing the Cantilever-Beam topology optimization solution

A classic study case of topology optimization is the Cantilever-Beam. It is composed of a beam with a clamped constrain on one side and a force \((F)\) of 5 N (1 N for each node) in the other side bottom edge of the beam, as depicted in Figure 4.22.

![Figure 4.22 Cantilever-Beam Model domain and boundary conditions](image)

The first topology optimization analysis executed in the Cantilever-Beam 3D is for a model with a 60x20x4 mesh \((L\times H\times t)\) and a volume constrain of 30\%. The results with ANSYS Wb are compared to the solutions from literature [8] and MatLab optimization, as depicted in Figure 4.23.

![Figure 4.23 Cantilever topology optimization 3D solutions obtained a) from literature [8]; b) from MatLab Optimization; and c) from ANSYS Wb with a mesh of 60x20x4 and Volume constraint of 30\%](image)
As expected, the results from literature and MatLab are practically the same. In the ANSYS analysis some additional support structures are created due to the lack of filter size ($r_{\text{min}}$) constrain, turning this result slightly different from literature and MatLab.

Graphics of the topology optimization objective function of a 3D Cantilever-Beam is a function of the number of iteration can be seen in Figure 4.24, for both the MatLab optimization and ANSYS Wb optimization. The objective function of this analysis is the structural compliance in Newton meter. In both cases, a convergence of the results can be observed for a volume of 30%. The ANSYS Wb optimization requires a lesser number of iterations to obtain the convergence, due to the stop criteria. The compliance results are 2486.6 Nm for ANSYS and 2417.7 Nm for MatLab.

![Figure 4.24 Cantilever topology optimization objective function convergence (compliance) obtained from MatLab Optimization (Top) and from ANSYS Wb (Bottom) with a mesh of 60x20x4 and Volume constraint of 30%](image)

4.2.1.1. Cantilever-Beam Volume Reduction

One of the most important parameters of the topology optimization analysis is the volume fraction ($f$). In Figure 4.25 the 3D Cantilever-Beam model with a 60x20x4 mesh and a force of 5 N for a volume constrain of 70%, 50% and 30% solutions from MatLab optimization and from ANSYS Wb are demonstrated.
When comparing MatLab and ANSYS solutions it becomes clear that for the lowest volume constrain the results from ANSYS have a more precise structure, due to the lack of filter size ($r_{\text{min}}$) constrain. In a higher volume constrain the checkerboard effect is displayed with an intermediate relative density region.

4.2.1.2. Cantilever-Beam Mesh

Another important parameter of the topology optimization analysis is the mesh resolution. The study of mesh dependency can be seen in Figure 4.26 where a 3D Cantilever-Beam model with a volume constrain of 30% is optimized in the MatLab and ANSYS for a mesh resolution of 30x10x2, 45x15x3 and 75x25x5, with respectively applied forces of 3 N, 4 N and 6 N (1 N per node).
Figure 4.26 Cantilever topology optimization solutions with compliance (c) for mesh resolution study obtained from MatLab Optimization (left pictures) and from ANSYS Wb (right pictures) with a Volume constraint of 30% and a mesh of 30x10x2 (top), 45x15x3 (middle) and 75x25x5 (bottom) respectively.

The solutions from MatLab and ANSYS are very similar and it is possible to see that for a more refined mesh resolution the solution becomes more detailed, as in the MBB-Beam mesh resolution case from a 2D previous section.

### 4.2.1.3. Cantilever-Beam with Multiple Load Cases

In this section a variation of the 3D Cantilever-Beam problem is studied. The variation differs from the classical example on the application of the force and the number of load cases, which instead of being only a load case with a single force distributed along the bottom edge, is the combination of two load cases with separated times of application. The first load case has a force of 5 N distributed along the top edge and the second load case has the same force distributed along the bottom edge, as depicted in Figure 4.27.
Figure 4.27 Cantilever with Multiple Load Cases Model domain and boundary conditions

The solution for this topology optimization problem in the literature [8], MatLab and ANSYS with a mesh resolution of 60x60x4 and a volume constrain of 40% is represented in Figure 4.28. The results are nearly the same for the literature and MatLab, and near with ANSYS Wb only a small concentration of lower relative densities is observed in the right side of the design.

Figure 4.28 Cantilever with Multiple Load Cases topology optimization solutions obtained a) from literature [8]; b) from MatLab Optimization; and c) from ANSYS Wb with a mesh of 60x60x4 and Volume constraint of 40%

Graphics of the topology optimization objective function of the 3D Cantilever-Beam with multiple load cases in function to the number of iterations can be seen in Figure 4.29, for both the MatLab and ANSYS Wb optimization. The ANSYS Wb optimization requires a lesser number of iterations to obtain the convergence, due to the stop criteria. The compliance results for a volume of 40% are 470.0 Nm for ANSYS and 20.6 Nm for MatLab.
4.2.1.4. Cantilever-Beam with Hole

Another variation of the 3D Cantilever-Beam problem is the introduction of a hole in the beam, as depicted in Figure 4.30.

In this topology optimization analysis the 3D Cantilever-Beam with a mesh of 60x20x4, a total force of 5 N and a volume constrain of 30% has a hole with 6.6(6) elements of radius and centered in (20; 10; 0) elements. Figure 4.31 represents the solutions from the literature [8], MatLab Optimization and ANSYS Wb, with a post processed TO for ANSYS with a relative density higher than 0.3 instead of 0.5 as in the other 3D analysis.
The results from the literature and MatLab are similar, as expected, and the ANSYS optimization results are similar to the other two, although a lower post processed relative density of 0.3 was used in order to ensure a design with structural integrity.

Graphics of the topology optimization objective function of the 3D Cantilever-Beam with multiple load cases in function to the number of iteration can be seen in Figure 4.32, for both the MatLab and ANSYS Wb optimization. The compliance results, for a volume of 30%, are 5573.9 Nm for ANSYS and 2527.7 Nm for MatLab.
4.3. Academic Problems Verification Conclusions

After the Topology Optimization comparison from ANSYS and MatLab of the MBB-Beam and Cantilever-Beam examples demonstrated in this chapter, some conclusions can be put forth.

The first critical point is in the different convergence termination criteria ($\text{ConvTer}$). In the studied cases, the compliance termination of the MatLab optimizations is obtained for a higher number of iterations when compared to the ANSYS optimization. This occurs due to the use of an absolute difference between the relative densities in MatLab, being difficult to compare with the relative difference between the compliance results, in ANSYS. Even with different convergence termination criteria, the compliance results converge to a similar value for both cases.

Another critical point is the filter size ($r_{\text{min}}$), which in the case of the ANSYS software is not imposed, using a post processing tool instead, as already explained in 2.5.5 and 3.1. The major disadvantages in this approach is the necessity of a post processing to avoid a checkerboard effect. On other hand, the lack of the filter size gives more detailed results, open to interpretation in the post processing.

As mentioned before, one of the major advantages in the ANSYS software is the post processing of the topology optimization results, allowing the engineer to adjust the minimum relative density displayed in order to interpret and obtain the most favored design.

In conclusion, topology optimization must be used as a guideline for the design and should not be used without critical judgement, either for the structural requirements verification or for manufacturing.
5. Tertiary Structure Analysis

Structures of a satellite can be classified as primary, secondary or tertiary structures. Primary structures are the major load path structure between the spacecraft’s components and the launch vehicle, corresponding to the main body structure of the satellite and the launch vehicle adapter. Secondary structures include support beams, trusses, antenna dishes and solar panels. Tertiary structures of a satellite include component housing, mounting brackets, cable-support brackets, and connector panels [36].

A mockup of the satellite of this thesis is represented in Figure 5.1 where the different tertiary structures are shown in yellow.

![Figure 5.1 Satellite representation with tertiary structures in yellow](image)

In this thesis the tertiary structure that is to be designed and optimized is a support of a Radio Occultation (RO) instrument. This structure is located on the tip of the satellite as depicted in Figure 5.2.

![Figure 5.2 Tertiary Structure: Support of the RO instrument](image)
5.1. Tertiary Structure Requirements

The initial tertiary structure of the satellite is given in Figure 5.3 where the initial CAD design and the global coordinates reference are shown.

In Figure 5.4 and Figure 5.5 the overall dimensions in millimeters (mm) of the initial tertiary structure in a front and top view are depicted.
It should be taken into account that this design is as provided in the spacecraft’s structural requirements, being only a reference design for the optimization cycle developed for this thesis.

The static requirements for the project of the tertiary structures are an acceleration of 20 G in all directions, having into account symmetrical properties, accordingly with the requirements. Those acceleration inputs are going to be used to test the structural integrity of the tertiary structure and the different load cases are represented in Table 5.1.

<table>
<thead>
<tr>
<th>Load Case</th>
<th>Acceleration x-axis</th>
<th>Acceleration y-axis</th>
<th>Acceleration z-axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-20G</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>-20G</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>0</td>
<td>0</td>
<td>-20G</td>
</tr>
</tbody>
</table>

The first natural frequency of the tertiary structure is going to be taking into account in this analysis in order to prevent a resonance phenomenon, which could cause severe damages on the structure and the components. This first natural frequency of the tertiary structure should be high enough to prevent dynamic coupling during the launch stage.

The Factors of Safety (FoS) for the Yield strength ($\sigma_Y$) and Ultimate tensile strength ($\sigma_{UTS}$) are 1.25 and 2.0, respectively. The requirements state that the Margin of Safety (MoS) should be positive when calculated with equation (5.1) where the FoS are applicable to the allowable load/stress ($\sigma_{allowable} = \sigma_Y$ or $\sigma_{UTS}$) and $\sigma_{applied}$ is the computed load/stress results.

$$MoS = \frac{\sigma_{allowable}}{\sigma_{applied} \times FoS} - 1 \quad (5.1)$$

For the material of the tertiary structure an Aluminium Alloy was selected from the material database of ANSYS [27]. That decision is made taking into account that, when preferable, it is recommended to select an aluminium for the design of tertiary structures in satellites, due to the high strength to mass ratio. Also the aluminium alloy from ANSYS database has close mechanical properties to the different aluminium alloys used on the metallic AM processes [20].

The Table 5.2 shows the mechanical properties of the aluminium alloy from ANSYS and the aluminium AISi10Mg and Al7075 with density ($\rho$), Young’s modulus ($E$), Poisson’s ratio ($\nu$), Yield strength ($\sigma_Y$) and Ultimate tensile strength ($\sigma_{UTS}$) [16], [19], [27].

<table>
<thead>
<tr>
<th>Material</th>
<th>$\rho$ [Kg/m$^3$]</th>
<th>$E$ [GPa]</th>
<th>$\nu$</th>
<th>$\sigma_Y$ [MPa]</th>
<th>$\sigma_{UTS}$ [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminium Alloy</td>
<td>2770</td>
<td>71</td>
<td>0.33</td>
<td>280</td>
<td>310</td>
</tr>
<tr>
<td>AISi10Mg</td>
<td>2670</td>
<td>70</td>
<td>0.33</td>
<td>240</td>
<td>460</td>
</tr>
<tr>
<td>Al 7075</td>
<td>2810</td>
<td>71.7</td>
<td>0.33</td>
<td>503</td>
<td>572</td>
</tr>
</tbody>
</table>
5.2. Tertiary Structure Initial Analysis

For the first analysis of the tertiary structure the CAD design is approximated to a surface model as depicted in Figure 5.6. The thickness of all areas is of 10 mm as illustrated at Figure 5.4 and Figure 5.5.

The boundary conditions of the tertiary structure include a distributed mass, representing the RO instrument with 3.48 Kg, and a fixed support on the connection to the satellite, as depicted in Figure 5.7. The acceleration inputs are the same as in section 5.1, 20 G in all directions, having into account symmetrical properties, represented in Table 5.1.

The structure is modelled as a surface structure where a mesh using SHELL181 elements is performed. This approximation is valid due to the lower thickness dimension, when compared to the other two dimensions.

The numerical analysis has sources of error that can compromise the validation of results, one of them being associated with the mesh discretization. A finite elements convergence study of the numerical results as a function of the number of elements is performed and shown in the graphics of Figure 5.8. With the increase of the number of elements, the value of the maximum...
displacement (verified in the position of the RO instrument), the total strain energy and the first 5 natural frequency converge to a result.

Having this study into account, a mesh with 36641 elements is chosen and represented in Figure 5.9, with detailed view in Figure 5.10.

*Figure 5.8 Graphics of the mesh size study: maximum displacement (in the RO instrument), total strain energy and first 5 natural frequency for the Initial Tertiary Structure analysis*
Figure 5.9 Initial Tertiary Structure Mesh with 36641 SHELL181 elements.

Figure 5.10 Initial Tertiary Structure mesh detailed view

The maximum Von Mises Stresses results for the static analysis of this initial tertiary structure are depicted in Table 5.3. In this table the Margin of Safety (MoS) is calculated, having equation (5.1) into account. The results of this analysis are not compliant with the requirement of having a positive MoS for the structure.

Table 5.3 Initial Tertiary Structure Static Analysis Solution from ANSYS Wb, maximum Von Mises stress [MPa] and Margin of Safety for the Yield (MoSY) and the Ultimate tensile strength (MoSUTS)

<table>
<thead>
<tr>
<th>Load Case</th>
<th>Max. Von Mises Stress [MPa]</th>
<th>MoSY</th>
<th>MoSUTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>40,19</td>
<td>4,57</td>
<td>2,86</td>
</tr>
<tr>
<td>2</td>
<td>621,68</td>
<td>-0,64</td>
<td>-0,75</td>
</tr>
<tr>
<td>3</td>
<td>57,56</td>
<td>2,89</td>
<td>1,69</td>
</tr>
</tbody>
</table>
This Von Mises stresses are a conservative approximation because the maximum stresses are assigned on singularity regions, where a high concentration of stresses occurs due to the nature of the model.

In Figure 5.11 the Von Mises stresses results are analyzed for the load case 2, being the load case with worst results. It is seen that the maximum stress results are in the mechanical interface region with the satellite, due to a singularity in the solution.

A detailed view of these points is depicted in Figure 5.12. The stress in the vicinity of the points is calculated, approximately 400 MPa, but still represents a negative MoS for the structure.

These Von Mises stresses are very high and the tertiary structure should be optimized to acceptable stresses, ensuring the tertiary structure integrity.

In Table 5.4 the results from the modal analysis of this initial tertiary structure are analyzed. For tertiary structures the first five natural frequencies are important. These mode are depicted in
Figure 5.13 and Figure 5.14 with a frequency of 11.9 Hz, 33.0 Hz, 80.1 Hz, 141.8 Hz and 227.6 Hz respectively. The first natural frequency result is too low, being not admissible for a tertiary structure, making necessary to increase it, a drastically change in design.

Table 5.4 Initial Tertiary Structure Modal Analysis solutions obtain from ANSYS Wb

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency [Hz]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11.88</td>
</tr>
<tr>
<td>2</td>
<td>33.01</td>
</tr>
<tr>
<td>3</td>
<td>80.06</td>
</tr>
<tr>
<td>4</td>
<td>141.78</td>
</tr>
<tr>
<td>5</td>
<td>227.61</td>
</tr>
<tr>
<td>6</td>
<td>320.13</td>
</tr>
<tr>
<td>7</td>
<td>346.38</td>
</tr>
<tr>
<td>8</td>
<td>444.64</td>
</tr>
<tr>
<td>9</td>
<td>498.98</td>
</tr>
<tr>
<td>10</td>
<td>539.88</td>
</tr>
</tbody>
</table>

Figure 5.13 Initial Tertiary Structure Modal solution obtained from ANSYS Wb, representing the Total Eigenvector Deformation (m) for the 1st Natural Frequency of 11.9 Hz

Figure 5.14 Initial Tertiary Structure Modal solution obtained from ANSYS Wb, representing the Total Eigenvector Deformation (m) for the 2nd(top left), 3rd(top right), 4th(bottom left) and 5th(bottom right) Natural Frequency of 33.0 Hz, 80.1 Hz, 141.8 Hz and 227.6 Hz respectively
6. Tertiary Structure Topology Optimization Analysis

In this chapter an optimized tertiary structure is analyzed first with a 2D analysis to obtain a good design region of optimization and then a 3D analysis for the final tertiary structure design. Finally, a verification of the optimized structure is performed, similarly to the previous chapter.

6.1. 2D Tertiary Structure Topology Optimization

An expansion of the design is considered to obtain a more efficient design region, for that purpose a 2D design of the structure is considered, as depicted in Figure 6.1. This 2D design is a surface model representative of the section plane x-z of the tertiary structure.

![Figure 6.1 2D Tertiary Structure Optimization Model](image1)

The boundary conditions of the tertiary structure include a distributed mass, representing the RO instrument with 3.48 Kg, and a fixed support on the satellite as depicted in Figure 6.2.

![Figure 6.2 2D Tertiary Structure Optimization Boundary Conditions with fixed support (blue) and the distributed mass representing RO (red)](image2)

The structure is modelled with shell elements (SHELL181) in a total of 6189 elements, corresponding to an overall element size mesh of 10 mm, as depicted in Figure 6.3.
The load case for the optimization is 20 G in all directions, having into account symmetrical properties. Table 6.1 represents the load cases inputs in this 2D optimization.

<table>
<thead>
<tr>
<th>Load Case</th>
<th>Acceleration x-axis</th>
<th>Acceleration y-axis</th>
<th>Acceleration z-axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-20G</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>0</td>
<td>-20G</td>
</tr>
</tbody>
</table>

The topology optimization parameters are defined in the Table 6.2 with the penalization power \((p)\), maximum number of iterations \((MaxIter)\), relative convergence termination criteria \((ConvTer.Rel)\), minimum relative density \((x_{\text{min}})\) and the volume fraction \((f)\). A filter size \((r_{\text{min}})\) is not imposed, obtaining more detailed solutions.

<table>
<thead>
<tr>
<th>(p)</th>
<th>(MaxIter)</th>
<th>ConvTer.Rel [%]</th>
<th>(x_{\text{min}})</th>
<th>(f) [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>500</td>
<td>0.01</td>
<td>0.001</td>
<td>35</td>
</tr>
</tbody>
</table>

The design region (in blue) and exceptions regions (in red) for the topology optimization analysis are depicted in Figure 6.4.
The result for the 2D tertiary structure topology optimization is shown in Figure 6.5 with a representation of the total length of relative densities.

![Figure 6.5 Tertiary Structure 2D Topology Optimization solution for a Volume constraint of 35% with the total length of relative densities](image)

Figure 6.5 Tertiary Structure 2D Topology Optimization solution for a Volume constraint of 35% with the total length of relative densities

Graphics of the topology optimization objective function of a 2D tertiary structure as a function of the number of iteration can be seen in Figure 6.6. The objective function of this analysis is the structural multiple compliance in Newton meter, and a convergence of the results can be observed. The compliance results, for a volume of 35%, is 1.16 Nm.

![Figure 6.6 2D Tertiary Structure Topology Optimization objective function convergence (Multiple Compliance) obtained from ANSYS Wb](image)

Figure 6.6 2D Tertiary Structure Topology Optimization objective function convergence (Multiple Compliance) obtained from ANSYS Wb

The results in this topology optimization analysis should take into account the structural integrity of the tertiary structure. For this purpose, in the post processed topology optimization solution is adopted a relative density higher than 0.6, obtaining the design in Figure 6.7.
6.2. Tertiary Structure Topology Optimization

In this chapter, a new tertiary structure design domain is defined, taking into account the results of the 2D topology optimization of the section 6.1. This new 3D design is represented in Figure 6.8, shaped as a surface model with a 10 mm thickness.

The internal structures of this tertiary structure is shown in Figure 6.9. The internal surfaces, of 10 mm thickness, were based on the 2D topology optimization analysis. This 3D redesign is performed in CAD with the interpretation of the engineer of the 2D results.
The boundary conditions of the tertiary structure include a distributed mass (in red), representing the RO instrument with 3.48 Kg, and a fixed support (in blue) on the satellite connection as depicted in Figure 6.10.

The load case for the optimization is 20 G in all directions, having into account symmetrical properties, as in the Table 5.1 of the requirements chapter in section 5.1.

The structure is modelled with shell elements (SHELL181), in order to reduce the computational time cost, in a total of 17117 elements, corresponding to an overall element size mesh of 10 mm, as depicted in Figure 6.11.

The common topology optimization parameters are defined in the Table 6.3 with the penalization power ($p$), maximum number of iterations ($MaxIter$), relative convergence termination criteria ($ConvTer_{Rel}$), minimum relative density ($x_{min}$) and the volume fraction ($f$).

<table>
<thead>
<tr>
<th>$p$</th>
<th>$MaxIter$</th>
<th>$ConvTer_{Rel}$ [%]</th>
<th>$x_{min}$</th>
<th>$f$ [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>500</td>
<td>0.01</td>
<td>0.001</td>
<td>40</td>
</tr>
</tbody>
</table>
The design region (in blue) and exceptions regions (in red) for the topology optimization are depicted in Figure 6.12.

![Design and Exceptions Regions for the Topology Optimization of the Tertiary Structure](image)

*Figure 6.12 Design and Exceptions Regions for the Topology Optimization of the Tertiary Structure*

The result for the 3D tertiary structure topology optimization is shown in Figure 6.13 with a representation of the total length of relative densities.

![Tertiary Structure Topology Optimization solution for a Volume constraint of 40% with the total length of relative densities](image)

*Figure 6.13 Tertiary Structure Topology Optimization solution for a Volume constraint of 40% with the total length of relative densities*

Graphics of the topology optimization objective function of a 3D tertiary structure as a function of the number of iteration can be seen in Figure 6.14. A convergence of the results can be observed and the compliance result, for a volume of 40%, is 39.63 Nm.
The results in this topology optimization analysis should take into account the structural integrity of the tertiary structure. For this purpose, on the post processed topology optimization is adopted a distribution of relative density higher than 0.4, obtaining the design in Figure 6.15.

6.3. Optimized Tertiary Structure Analysis and Validation

In this chapter, a new tertiary structure design is defined, taking into account the results of Figure 6.15 in the previous chapter. This design is composed with diverse trellis walls, making it a complex structural design, as depicted in Figure 6.16.
The boundary conditions of the tertiary structure include a distributed mass, representing the RO instrument with 3.48 Kg, and a fixed support on the satellite connection as depicted in Figure 6.17.

The structure is model as a solid structure and, for this reason, a SOLID185 mesh is applied, represented in Figure 6.19. The numerical analysis has sources of error that can compromise the validation of results, one of them being associated with the mesh discretization. A study of the numerical results in function to the number of elements is performed and shown in the graphics of Figure 6.18. With the increase of the number of elements, the value of the maximum displacement located in the RO instrument, the total strain energy and the first 5 natural frequency converge to a result.
Figure 6.18 Graphics of the mesh size study: maximum displacement in the RO instrument, total strain energy and first 5 natural frequency for the Optimized Tertiary Structure analysis

Having this study into account, the mesh in Figure 6.19 with 146935 elements was chosen.
The maximum Von Mises Stresses results for the static analysis of this optimized tertiary structure are presented in Table 6.4. In this table the Margin of Safety (MoS) is calculated, taking equation (5.1) into account. The results of this analysis are compliant with the requirement of having a positive MoS for the structure.

Table 6.4 Optimized Tertiary Structure Static Analysis Solution from ANSYS Wb, maximum Von Mises stress [MPa] and Margin of Safety for the Yield ($MoS_Y$) and the Ultimate tensile strength ($MoS_{UTS}$).

<table>
<thead>
<tr>
<th>Load Case</th>
<th>Max. Von Mises Stress [MPa]</th>
<th>$MoS_Y$</th>
<th>$MoS_{UTS}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>19,61</td>
<td>10,42</td>
<td>6,90</td>
</tr>
<tr>
<td>2</td>
<td>51,06</td>
<td>3,39</td>
<td>2,04</td>
</tr>
<tr>
<td>3</td>
<td>21,17</td>
<td>9,58</td>
<td>6,32</td>
</tr>
</tbody>
</table>

This Von Mises stresses are a conservative approximation because the maximum stresses are assigned on singularity regions.

In Figure 6.20 the Von Mises stresses results are analyzed for the load case 2, since this is the load case with worst results. It is seen that the maximum stress results are in the fixed region and the bottom region with the satellite, due to singularity regions of stresses as mentioned before.
A detail view of these points is depicted in Figure 6.21. The stress in the vicinity of the points is calculated, approximately 25 MPa.

These Von Mises stresses results are very efficient, corresponding to a Margin of Safety of 7.96 for the Yield strength and 5.20 for the Ultimate Tensile strength.

In Table 6.5 the results from the modal analysis of this optimized tertiary structure are analyzed. For tertiary structures the first natural frequency is important. This mode is depicted in Figure 6.22 with a frequency of 71.85 Hz. Also the mode shapes of the 2nd, 3rd, 4th and 5th natural frequencies are depicted in Figure 6.23.
Table 6.5 Optimized Tertiary Structure Modal Analysis solutions obtain from ANSYS Wb

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency [Hz]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>71.85</td>
</tr>
<tr>
<td>2</td>
<td>103.03</td>
</tr>
<tr>
<td>3</td>
<td>188.87</td>
</tr>
<tr>
<td>4</td>
<td>209.04</td>
</tr>
<tr>
<td>5</td>
<td>277.11</td>
</tr>
<tr>
<td>6</td>
<td>315.94</td>
</tr>
<tr>
<td>7</td>
<td>354.10</td>
</tr>
<tr>
<td>8</td>
<td>404.31</td>
</tr>
<tr>
<td>9</td>
<td>410.01</td>
</tr>
<tr>
<td>10</td>
<td>435.76</td>
</tr>
</tbody>
</table>

Figure 6.22 Optimized Tertiary Structure Modal solution obtained from ANSYS Wb, representing the Total Eigenvector Deformation (m) for the 1st Modal Frequency of 71.9 Hz

Figure 6.23 Optimized Tertiary Structure Modal solution obtained from ANSYS Wb, representing the Total Eigenvector Deformation (m) for the 2nd(top left), 3rd(top right), 4th(bottom left) and 5th(bottom right) Natural Frequency of 103.0 Hz, 188.9 Hz, 209.0 Hz and 277.1 Hz respectively
6.4. Comparison between Initial and Optimized Tertiary Structure

In the final section of this chapter is presented a comparison between the initial (section 5.2) and optimized tertiary structure (section 6.3) evaluating the impact of the topology optimization in the structure. For that propose it is calculated a relative difference (Difference [%]) between the initial and optimized results of the analysis, using equation (6.1).

\[
\text{Difference [%]} = \frac{\text{Optimized} - \text{Initial}}{\text{Initial}} \times 100\%
\]  

(6.1)

The results of the static analysis are depicted in Table 6.6, where a clear reduction of the maximum Von Mises stresses with the optimization is noted. The more significant reduction occurs in load case 2, with a reduction of 91.79% in the maximum Von Mises stress of the optimized results (in Figure 6.21) in comparison to the initial analysis (in Figure 5.12). The lower reduction occurs in the load cases 1, with a reduction for less than half of the initial stress results, 51.19% reduction.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>40.19</td>
<td>19.61</td>
<td>-51.19</td>
</tr>
<tr>
<td>2</td>
<td>621.68</td>
<td>51.06</td>
<td>-91.79</td>
</tr>
<tr>
<td>3</td>
<td>57.56</td>
<td>21.17</td>
<td>-63.22</td>
</tr>
</tbody>
</table>

The comparison between the results for the modal analysis are represented in Table 6.7 where the relative difference between initial and optimized tertiary structure is calculated. For the first modes, the optimized structure presents a very significant increase, more than the double. In the case of the first mode, the optimized frequency is 5 times higher than the initial frequency, 504.96% increase, making the tertiary structure significant more stiff than the initial design.
Table 6.7 Comparison between Modal Analysis from Initial and Optimized Tertiary Structures

<table>
<thead>
<tr>
<th>Mode</th>
<th>Initial Frequency [Hz]</th>
<th>Optimized Frequency [Hz]</th>
<th>Difference [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11,88</td>
<td>71,85</td>
<td>+504,96</td>
</tr>
<tr>
<td>2</td>
<td>33,01</td>
<td>103,03</td>
<td>+212,10</td>
</tr>
<tr>
<td>3</td>
<td>80,06</td>
<td>188,87</td>
<td>+135,90</td>
</tr>
<tr>
<td>4</td>
<td>141,78</td>
<td>209,04</td>
<td>+47,44</td>
</tr>
<tr>
<td>5</td>
<td>227,61</td>
<td>277,11</td>
<td>+21,75</td>
</tr>
<tr>
<td>6</td>
<td>320,13</td>
<td>315,94</td>
<td>-1,31</td>
</tr>
<tr>
<td>7</td>
<td>346,38</td>
<td>354,10</td>
<td>+2,23</td>
</tr>
<tr>
<td>8</td>
<td>444,64</td>
<td>404,31</td>
<td>-9,07</td>
</tr>
<tr>
<td>9</td>
<td>498,98</td>
<td>410,01</td>
<td>-17,83</td>
</tr>
<tr>
<td>10</td>
<td>539,88</td>
<td>435,76</td>
<td>-19,29</td>
</tr>
</tbody>
</table>

In topology optimization studies a comparison between the initial tertiary structure (Figure 5.3), the tertiary structure design based on the 2D TO (Figure 6.8) and the optimized tertiary structure (Figure 6.16) masses is made. This comparison is shown in Table 6.8 and an increase of 10.43% of the final tertiary structure is seen when in comparison with the initial design.

Table 6.8 Comparison of Volume and Mass between Initial, Based in 2D TO and Optimized Tertiary Structures

<table>
<thead>
<tr>
<th>Structure</th>
<th>Volume [m³]</th>
<th>Mass [Kg]</th>
<th>Difference [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial</td>
<td>0,005749</td>
<td>15,93</td>
<td></td>
</tr>
<tr>
<td>Based in 2D TO</td>
<td>0,016750</td>
<td>46,40</td>
<td>+10,43</td>
</tr>
<tr>
<td>Final</td>
<td>0,006349</td>
<td>17,59</td>
<td></td>
</tr>
</tbody>
</table>

The expansion of the initial design to a design based on the 2D TO have an increase of 191.3% on the mass. This increase is necessary due to requirement purposes, since the initial design is very restrict for optimization. After that expansion, a decrease of -62.1% in the mass occurs with the TO for the final design. The difference between initial and final design is +10.43%, but the difference correspondent to the TO is of -62.1%. This 62.1% reduction is near to the correspondent TO with a volume fraction \( (f) \) of 40%, corresponding to a 60% reduction, where the error for the equality constraint \( \left( \frac{V(x)}{V_0} = f \right) \) is 5.2%.
7. Conclusions

7.1. Achievements

The achievements of the presented work are divided in the following conclusions:

- This thesis presents a brief review of AM processes, but it was not considered at post processing of the topology optimization results.
- In the topology optimization it was conducted a successful comparison between the results in literature for MatLab codes [3], [8] and the ANSYS Topology Optimization ACT.
- Also, a methodology for the topology optimization analysis was introduced, taking advantage of the geometrical freedom provided by AM. A specific characteristic of this methodology is the introduction of a two steps topology optimization, 2D and 3D, requiring also two post processing of the results to obtain the design. This approach gives a preconceived initial design for the 3D optimization, but is an initial design based on a 2D topology optimization and represents a computational time saving for the 3D optimization.
- Finally, a design methodology with a two steps topology optimization and post processing for the satellite tertiary structure was conducted. It was possible to obtain a decrease of 91.79% in the maximum Von Mises stress and an increase of 504.96% in the first natural frequency of the tertiary structure, with only an increase of 10.43% of the mass when compared with the initial tertiary structure design. It should be taken into account that the mass increase is obtained due to an increase of 191.3% on the mass for the 2D optimization and interpretation of the results. In the 3D topology optimization is obtained a decrease of 62.1% in the mass, corresponding to an error for the equality constraint \( \frac{V(\Omega)}{V_0} = f \) of 5.2% for a volume fraction \( f \) of 40%.

7.2. Future Work

In terms of further development there are some relevant subjects:

- Study the influence of the procedures of AM in the material properties, with a better characterization of the effect of the process on final product performance, in a way that the topology optimization analysis have more realistic structural result.
- Study the effects of manufacturing constrain in the topology optimization analysis, such as enclosed voids, minimum feature size and overhanging requirements.
- Verification of buckling critical loads of the optimized structures or even its inclusion as constraint in the optimization.
- Study the use of other objective functions, such as an analysis using natural frequencies and buckling strength as objective functions, or even a multi-objective function analysis.
- Study other types of optimization analysis or algorithms, such as shape optimization analysis or BESO-algorithm, in the design of tertiary structures.
8. References


