Application of Topology Optimization to a Satellite Tertiary Structures

Emanuel Filipe Moreira
emanuel.moreira@tecnico.ulisboa.pt

Instituto Superior Técnico, Lisbon, Portugal
November 2016

Abstract

The challenge of this work 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 a geometrical freedom for the design, making a perfect combination with topology optimization.

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, here applied 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, 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

1. Introduction

The aerospace industry is looking for new developments and some of the most important concerns are to obtain lighter structure in order to save on fuel consumption, while improving the structural efficiency of the structures. Both these goals can be achieved by combining a structural topology optimization with an additive manufacture process.

Topology Optimization (TO) was introduced by the pioneer work of Bendsøe and Kikuchi [1] on homogenization method, and since then several authors have developed the concept, including Bendsøe and Sigmund [2] a density based approach.

Additive Manufacturing (AM) is a growing process for the manufacturing of objects and is define by a process of joining materials to make objects from 3D model data, usually layer upon layer, as opposed to subtractive manufacturing technologies [3].

Brackett, Ashcroft and Hague [4] describe opportunities for the application of TO methods for AM, presenting a workflow approach.

The challenge of this work is to use TO software to propose a design for a tertiary structure of a satellite. To achieve this goal a TO with minimum compliance objective was divided in a two stage optimization methodology is applied, with a 2D and 3D analysis, using a combination of a Computer Aid Design (CAD) software, Solid Edge, and a TO software, ANSYS Wb v17 with Topology Optimization ACT.

2. Fundamentals
2.1. Finite Element Method

The Finite Element Method (FEM) 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. In this work, using ANSYS [5], two types of elements are used: shell elements (SHELL181) and 3D solid elements (SOLID185).

Over each subdomain the governing equations are approximated determining the element equations (of each finite element). After that discretization, the global system of equations can be found by assembling the equations of all the finite elements of the mesh [6].

2.1.1. Static Analysis

The static structural finite element analysis for 3D static elasticity problems is derived from Cauchy law, using a residual method and imposing boundary conditions thus generating a linear system of equations, represented by

$$[K]\{u\} = \{F\}. \qquad (2.1)$$

where $\{F\}$ is the nodal forces vector, $\{u\}$ is the nodal displacement vector and $[K]$ is the stiffness matrix.

2.1.2. Modal Analysis

To get the undamped natural frequencies of a structure in free vibration, the modal structural finite element analysis is reduced, from the Lagrange's equation and the assumption of a harmonic response, to the following eigenvalues problem

$$([K] - \omega_i^2[M])\{\Phi\}^i = \{0\}, \qquad (2.2)$$

where eigenvalues correspond to the natural frequencies (ω_i) and the eigenvectors to the corresponding mode shapes $(\{\Phi\}^i)$.

2.2. Topology Optimization

The following section is based on the works from Sigmund [7]-[8]; Andreassen et al. [9]; Liu and Tovar [10]; Bendsøe and Sigmund [2]; Fernandes [11] and Neves et al. [12].

Topology Optimization is a computational material distribution method for synthesizing structures without any preconceived topology and shape [10].

In this work, the adopted method for the TO problems is a density based approach, known as Solid Isotropic Material with Penalization (SIMP). This method consists on using constant material properties in each element used to discretize the design domain. The SIMP method is based on a relation between relative element density (x_e) and the element Young's modulus (E_e), and is given by,

$$E_e(x_e) = x_e^p E_0, 0 < x_{min} \leq x_e \leq 1, \qquad (2.3)$$

where p is a penalization power $(p \geq 1)$, E_0 is the Young's modulus of the solid material and x_{min} is the relative element density of the "void" material.

A modified SIMP approach given by (2.4) can be used, where E_{min} is the Young's modulus of the "void" material.

$$E_e(x_e) = E_{min} + x_e^p(E_0 - E_{min}), \qquad (2.4)$$
$$x_e \in [0,1]$$

Using the finite element analysis theory, global stiffness matrix (K) is defined by equation (2.5) and the element stiffness matrix (K_e) defined by equation (2.6).

$$K(x) = \sum_{e=1}^{N} K_e(x_e) \qquad (2.5)$$

$$K_e(x_e) = E_e(x_e)k_e^0 \qquad (2.6)$$

k_e^0 is the element stiffness matrix for an element with a unitary Young's modulus, depending on the element type and the Poisson's ratio (v).

The TO 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) for punctual forces can be defined as in equation (2.7).

$$C(x) = F^T U(x), \qquad (2.7)$$

where F is the vector of nodal force and U is the vector of nodal displacement. In (2.8) the minimum compliance optimization problem is formulated as

$$\min_x : C(x) = F^T U(x)$$
$$subject\ to: \begin{cases} \dfrac{V(x)}{V_0} = f \\ F = KU \\ 0 < x_{min} \leq x \leq 1 \end{cases} \qquad (2.8)$$

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, or volume constraint.

So the problem has an equality constraint in the volume, $\frac{V(x)}{V_0} = f$. This fraction (or percentage) defines the initial distribution of material, as a uniform density of f in all the domain.

$F = KU$ is the state equation, but numerically can be more efficient to use $C = F^T U$. $0 < x_{min} \leq x \leq 1$ impose the lateral constraints in the design variables.

Equation (2.9) is obtained by developing the compliance with the definition of the nodal force and decomposing in the sum of the element compliance (c_e), where N is the number of

elements used to discretize the design domain and u_e is the element displacement vector.

$$C(x) = \sum_{e=1}^{N} c_e = \sum_{e=1}^{N} u_e{}^T K_e u_e \qquad (2.9)$$

The sensitivity of the objective function is obtained by derivation of compliance with respect to the design variable. Knowing that the nodal force vector is independent of the design variable and decomposing the sensitivity the following sensitivity equation is obtained.

$$\frac{\partial C(x)}{\partial x_e} = \sum_{e=1}^{N} -u_e{}^T[p x_e^{p-1}(E_0 - E_{min})k_e^0]u_e \qquad (2.10)$$

Optimality Criteria (OC) methods are one of the approaches to solve non-linear programming problems, such as minimum compliance. The OC-method uses the KKT conditions to find a solution for the optimization problem. For the problem formulation described in equation (2.8), with the application of the stationarity of the Lagrangian function with respect to the design variable (x_e), the optimality criteria is expressed when $B_e = 1$ and is given by [2]

$$B_e = \frac{-\dfrac{\partial C}{\partial x_e}}{\lambda \dfrac{\partial V(x)}{\partial x_e}} \qquad (2.11)$$

where λ is a Lagrangian multiplier.

The OC uses a heuristic updating scheme for the design variables that can be expressed in

$$x_e^{new} =$$
$$= \begin{cases} \max(x_{min}, x_e - m), if\ x_e B_e^{\eta} \le \max(x_{min}, x_e - m) \\ x_e B_e^{\eta}, if\ \max(x_{min}, x_e - m) < x_e B_e^{\eta} < \min(1, x_e + m) \\ \min(1, x_e + m), if\ \min(1, x_e + m) \le x_e B_e^{\eta} \end{cases} \quad (2.12)$$

where m is a positive move-limit and η is a numerical damping coefficient, for minimum compliance problems $m = 0.2$ and $\eta = 0.5$ are recommended for minimum compliance problems by Bendsoe [13] and Sigmund [8].

The termination criteria in ANSYS optimization is defined when the maximum number of iterations $(MaxIter)$ is reached, without ensuring of optimal criteria, or when a convergence termination criteria $(ConvTer\_Rel)$ value is obtained. This work has two different convergence termination criteria. The first, used in MatLab optimizations, is defined by the difference between the new design variable (x_e^{new}) and the old design variable (x_e^{old}), expressed in equation (2.13).

$$\|x_e^{new} - x_e^{old}\|_\infty \le ConvTer\_Abs \qquad (2.13)$$

The second convergence termination criteria, used in ANSYS, is defined by a relative difference between the compliance values of the new design (c^{new}) and the compliance value of the old design (c^{old}) for the TO results, represented in (2.14).

$$\frac{|c^{new} - c^{old}|}{c^{old}} \le ConvTer\_Rel \qquad (2.14)$$

One of the problems of the TO is the checkerboard effects. This effect is defined as regions with alternating void and solid elements ordered in a checkerboard-like fashion [7], [12].

A way to avoid numerical instabilities, such as checkerboards effects, and ensure existence of solutions on TO is the insertion of filter techniques, such as sensitivity filter.

The sensitivity filter is imposed on the sensitivity function (2.10), being defined by

$$\frac{\partial C(x)}{\partial x_e} = \frac{\sum_{f=1}^{N_e} H_{ef} x_f \dfrac{\partial c_f}{\partial x_f}}{x_e \sum_{f=1}^{N_e} H_{ef}}, \qquad (2.15)$$
$$0 < x_{min} \le x_e \le 1$$

where N_e is the neighborhood of the element x_e and can be defined in equation (2.16), x_f and $\frac{\partial c_f}{\partial x_f}$ are the relative density and the sensitivity of those elements.

$$N_e = \{f : dist(e, f) \le r_{min}\} \qquad (2.16)$$

The operator $dist(e, f)$ is the distance between the element x_e and the element x_f.

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

$$H_{ef} = \mathrm{r}_{min} - dist(e, f), \qquad f \in N_e \qquad (2.17)$$

r_{min} is the filter size, given as input data in MatLab, and imposes a minimum length scale of r_{min}, in a way that, with the increase of the filter size, the results are less detailed, i.e., a design more compact and with less holes (but holes with higher area).

In the case of ANSYS a filter size isn't 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 avoid the checkerboard effect, as shown in Figure 2.1.

After the TO, a post processing of the TO results is performed in ANSYS, where is defined a relative density for which all the densities with a higher value are displayed.
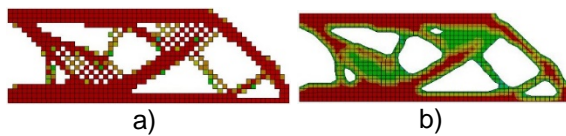
Figure 2.1 MBB-Beam Half TO solutions obtained from ANSYS Wb with a mesh of 60x20, $E = 1Pa$, $v = 0.3$, $p = 3$, $x_{min} = 0,001$, $ConvTer\_Rel = 0,01\%$, $MaxIter = 500$, $f = 50\%$ and a TO post processing with relative density higher than 0.4 a) without smoothing and b) with smoothing

3. Methodology

In this section the methodology developed in the work to answer to the challenger is explained.

3.1. Academic Problems for Verifications

Before performing the TO analysis of the tertiary structure it was considered necessary to test and verify the procedures implemented to use the optimization softwares with academic problems. The academic problems chosen here consist on classic cases of a MBB-Beam and a Cantilever-Beam in 2D and 3D.

The adopted approach for each academic case is an optimization study comparing the literature and MatLab optimization, using TopOpt2D [8] and TopOpt3D [10], solutions with the ANSYS Wb v17 - Topology Optimization ACT results.

For that purpose, a design domain, defined by the mesh size since each element represents the units of measurement (1x1x1).

All the boundary conditions, material properties and optimization parameter are defined on MatLab and ANSYS software, similarly to the literature.

3.2. Tertiary Structure Analysis

The method of analysis of the tertiary structure uses a traditional FEM analysis methodology, represented in Figure 3.1.
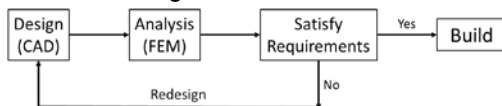
Figure 3.1 Traditional FEM optimization analysis, based on [14]

Firstly the structure is defined in CAD and the design and material of the tertiary structure are imposed.

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 the loads and boundary conditions are applied, and the elements and mesh are chosen, 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 if are or aren't compliant with the structural requirements. If the static and modal results don't satisfy the requirements, the design is changed and the FEM model remade until the results are compliant with the requirements.

In this work the analysis of the tertiary structure is performed only to have a relative idea of the results for further comparison with the optimized structure's data.

3.3. Tertiary Structure Topology Optimization Analysis

In this work an alternative approach is proposed in order to optimize the tertiary structure of the satellite, depicted in Figure 3.2.
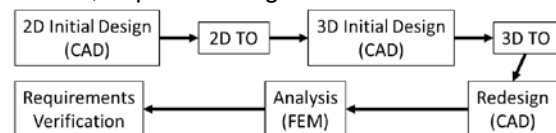

Figure 3.2 TO Analysis: 2D and 3D TO Analysis

The difference in this approach is that a 2D and 3D TO analysis are performed, instead of a single 3D optimization.

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.

After the 2D design is defined in CAD, the design is imported, in STEP format, to the ANSYS Wb v17, where the TO is going to be performed. In ANSYS, the loads and boundary conditions, the material properties, the elements and mesh and the optimization parameters are defined.

One of TO parameters is the design domain, which consists on the portion of the model where the TO is going to be made. Other parameter is the design exclusions, which are the elements of the design domain that aren't optimized.

After the 2D TO is performed, a post processing of the TO results in ANSYS is performed. This post processing should produce a viable design, in terms of structural integrity, and a STL file is exported to the CAD software for further processing.

4

The 2D optimization design is interpreted, by extrapolating the design on the third dimension, perpendicular to the 2D plane, to a 3D design. The result of this interpretation is the initial design for the 3D TO analysis, and can be exported in a STEP format for the ANSYS software.

After the 3D initial design is defined, the parameter for TO are defined (mesh model, loads and boundary conditions and TO parameters), as in the 2D analysis.

After the 3D TO 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.

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 section, classical TO problems are simulated in 2D and 3D.

4.1. 2D Problem (MBB-Beam)

The MBB beam 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.
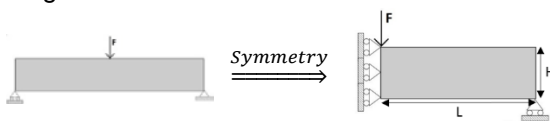
Figure 4.1 MBB-Beam Model and MBB-Beam Model simplification domain and boundary conditions for the static case

In order to simulate these 2D academic problems, the thickness is unitary (t=1), and in ANSYS the mesh is defined with shell elements of the type SHELL181.

The results are depicted in Figure 4.2, and the graphics of the objective function history can be seen in Figure 4.3, with compliances of 203.30Nm for MatLab and 206.11Nm for ANSYS.
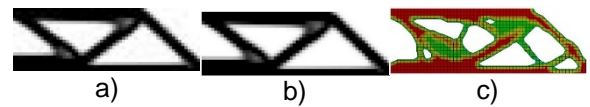

Figure 4.2 MBB-Beam Half TO solutions with a mesh of 60x20, $E = 1Pa$, $v = 0.3$, $p = 3$, $x_{min} = 0,001$ and $f = 50\%$ obtained: a) from the literature [8], b) from MatLab Optimization with $ConvTer\_Abs = 0,01$ and $r_{min} = 1,5$; and c) from ANSYS Wb with $ConvTer\_Rel = 0,01\%$, $MaxIter = 500$ and a TO post processing with relative density higher than 0.4
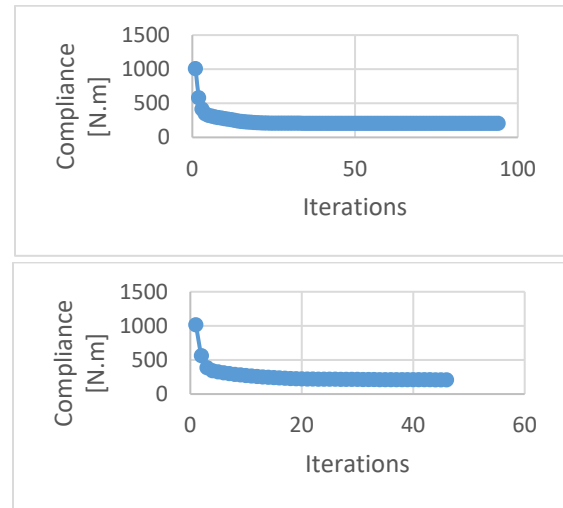

Figure 4.3 MBB-Beam Half TO objective function convergence (compliance) obtained from MatLab Optimization (Top) and from ANSYS Wb (Bottom) with a mesh of 60x20 and $f = 50\%$

4.2. 3D Problem (Cantilever-Beam)

The Cantilever-Beam is composed of a beam with a clamped constrain on one side and a force (F) of 5N, 1N for each node, in the other side bottom edge of the beam, as depicted in Figure 4.4.
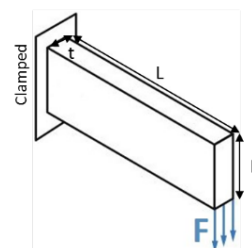

Figure 4.4 Cantilever-Beam Model domain and boundary conditions

In ANSYS the mesh model is defined with SOLID185 element type. The results are depicted in Figure 4.5 and the graphics of the objective function history can be seen in Figure 4.6, with compliance results of 2417.7Nm for MatLab and 2486.6Nm for ANSYS.
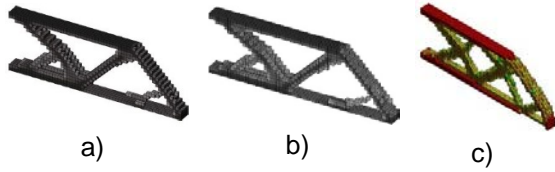
a)   b)   c)

Figure 4.5 Cantilever TO 3D solutions with a mesh of 60x20x4, $E = 1Pa$, $\nu = 0.3$, $p = 3$ and $f = 30\%$ obtained a) from literature [10]; b) from MatLab Optimization with $ConvTer\_Abs = 0,01$, $E_{min} = 1 \times 10^{-9} Pa$ and $r_{min} = 1,5$; and c) from ANSYS Wb with $ConvTer\_Rel = 0,01\%$, $MaxIter = 500$, $x_{min} = 0,001$ and a TO post processing with relative density higher than 0.5
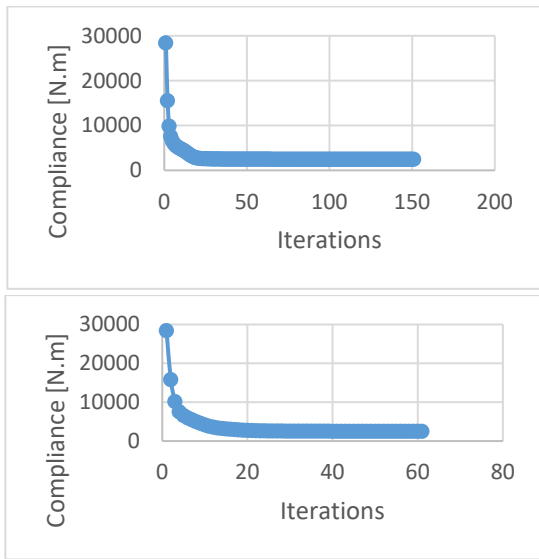

Figure 4.6 Cantilever TO objective function convergence (compliance) obtained from MatLab Optimization (Top) and from ANSYS Wb (Bottom) with a mesh of 60x20x4 and Volume constraint of 30%

4.3. Discussion

The first critical point is in the different convergence termination criteria. 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. Despite this, the compliance results converge to a similar value for both cases.

Other critical point is the filter size, where in the case of the ANSYS software isn't imposed, as already explained in section 2.2. 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.

One of the major advantages in the ANSYS software is the post processing of the TO results, allowing the engineer to adjust the minimum relative density displayed in order to interpret and obtain the most favored design.

5. Tertiary Structure Analysis

The tertiary structures of a satellite include component housing, mounting brackets, cable-support brackets, and connector panels [15].

The initial tertiary structure of the satellite is a support of a Radio Occultation (RO) instrument with 3.48 Kg, given in Figure 5.1 where the initial CAD design with overall dimensions (mm) and the global coordinates reference are shown.
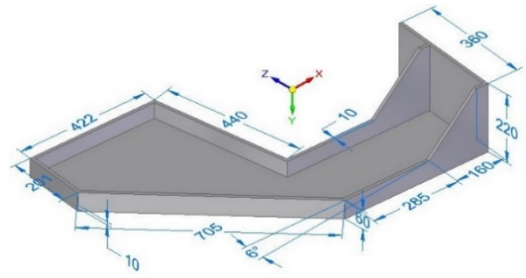

Figure 5.1 Tertiary Structure initial CAD design with overall dimensions (mm)

For the material of the tertiary structure this study choose the Aluminum Alloy from the material database of ANSYS [5], with mechanical properties shown in Table 5.1.

ρ [Kg/m^3]	E [GPa]	ν	σ_Y [MPa]	σ_{UTS} [MPa]
2770	71	0.33	280	310

Table 5.1 Mechanical properties for Aluminum Alloy from ANSYS [5]

The static requirements for the project of the tertiary structures are an acceleration of 20G in all directions, having into account symmetrical properties, represented in Table 5.2.

Load Case	Accel. x-axis	Accel. y-axis	Accel- z-axis
1	-20G	0	0
2	0	-20G	0
3	0	0	-20G

Table 5.2 Load Cases inputs for Static Analysis of the Tertiary Structure

The Factors of Safety (FoS) for the Yield strength (σ_Y) and Ultimate tensile strength (σ_{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

6

load/stress ($\sigma_{allowable} = \sigma_Y \ or \ \sigma_{UTS}$) and $\sigma_{applied}$ is the computed load/stress results.

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

5.1. Tertiary Structure Initial Analysis

For the first analysis of the tertiary structure the CAD design is approximated to a surface model with a thickness of all areas of 10mm as illustrated at Figure 5.1.

The boundary conditions of the tertiary structure include a distributed mass, representing the RO, and a fixed support on the connection to the satellite, as depicted in Figure 5.2. The acceleration inputs are represented in Table 5.2.
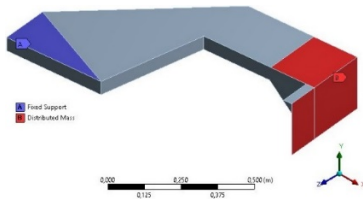
Figure 5.2 Initial Tertiary Structure Boundary Conditions with fixed support (blue) and the distributed mass representing RO (red)

The structure is modelled as a surface structure where SHELL184 mesh is performed. This approximation is valid due to the lower thickness dimension in reference to the other two dimensions. Having into account a finite elements convergence study, a mesh with 36641 elements is chosen and represented in Figure 5.3.
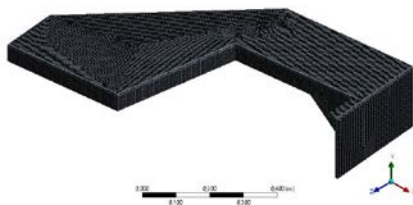
Figure 5.3 Initial Tertiary Structure Mesh with 36641 SHELL181 elements.

The maximum Von Mises Stresses (σ_{VM}) results for the static analysis of this initial tertiary structure are depicted in Table 5.3.

Load Case	Max. σ_{VM} [MPa]	MoS_Y	MoS_{UTS}
1	40,19	4,57	2,86
2	621,68	-0,64	-0,75
3	57,56	2,89	1,69

Table 5.3 Initial Tertiary Structure Static Analysis Solution from ANSYS Wb.

In Figure 5.4 the stress results are analyzed for the load case 2, being the case with worst results.
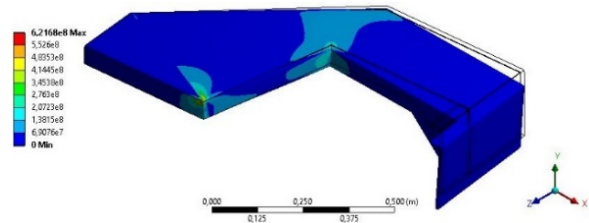

Figure 5.4 Initial Tertiary Structure Static Analysis Solution obtained from ANSYS Wb, the σ_{VM} (Pa) for the 2nd load case

It's seen that the maximum stress results are in the fixed region with the satellite, due to a singularity in the solution. The stress in the vicinity of the points is calculated, approximately 400 MPa, but still represents a negative MoS.

The first natural frequencies, depicted in Figure 5.5, is of 11.9Hz. This frequency isn't admissible for a tertiary structure, making necessary a drastically change in design to increase it.
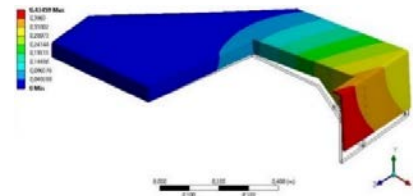

Figure 5.5 Initial Tertiary Structure Modal solution obtained from ANSYS Wb, representing the Total Eigenvector Deformation (m) for the 1st Natural Frequency of 11.9Hz

6. Tertiary Structure Topology Optimization Analysis

In this section is applied the TO methodology in the tertiary structure.

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.
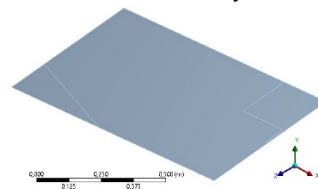

Figure 6.1 2D TO Tertiary Structure Model

The boundary conditions are shown in Figure 6.2. The exceptions regions for the TO correspond to the fixed and mass distributed areas.
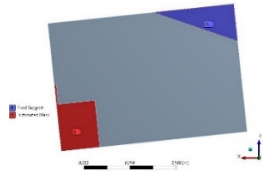
Figure 6.2 2D TO Tertiary Structure Boundary Conditions with fixed support (blue) and the distributed mass representing RO (red)

The load case for this optimization is the load cases 1 and 3 from Table 5.2.

The structure is modelled with shell elements (SHELL184) in a total of 6189 elements, as depicted in Figure 6.3.
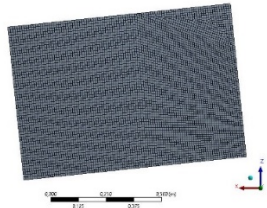

Figure 6.3 2D Tertiary Structure Initial Mesh for 2D TO with 6189 SHELL181 elements

The TO parameters are defined in the Table 6.1, with the TO result and post processed solution shown in Figure 6.4.

p	$MaxIter$	$ConvTer\_Rel$ [%]	x_{min}	f [%]
3	500	0.01	0.001	35

Table 6.1 TO Parameters for 2D Tertiary Structure analysis
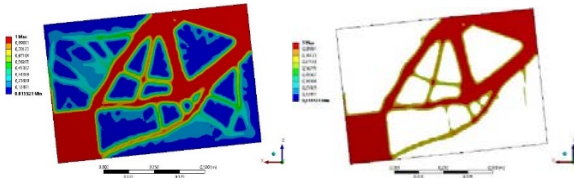
Figure 6.4 Tertiary Structure 2D TO solution with the total length of relative density (left) and a relative density higher than 0.6 (right)

Graphics of the TO objective function history can be seen in Figure 6.5. The compliance result, for a volume of 35%, is 1.16Nm.
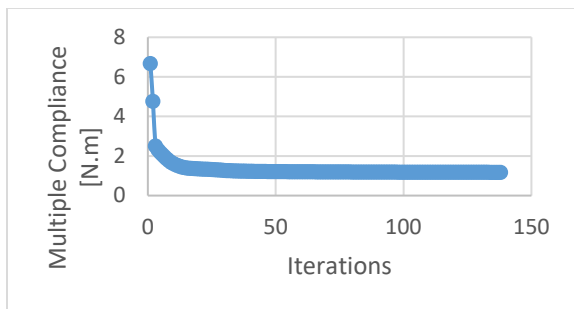

Figure 6.5 2D Tertiary Structure TO objective function history obtained from ANSYS Wb

6.2. Tertiary Structure Topology Optimization

A new tertiary structure design domain is defined, taking into account the results of the 2D TO of the section 6.1. This new 3D design is represented in Figure 6.6, shaped as a surface model with a 10mm thickness.
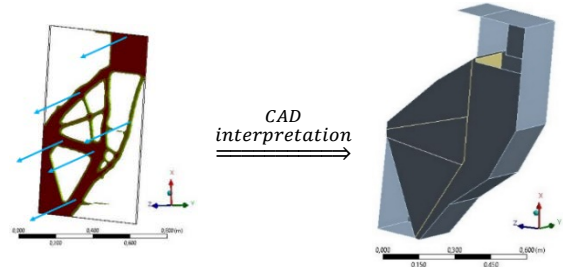

Figure 6.6 Tertiary Structure Model Design with Surfaces of 10mm thickness based on the 2D TO Analysis

The boundary conditions are seen in Figure 6.7. The exceptions regions for the TO correspond to the fixed and mass distributed areas.
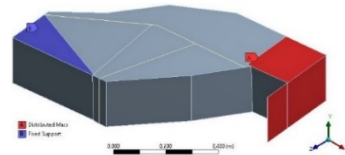

Figure 6.7 Tertiary Structure Boundary Conditions based on the 2D TO Analysis with fixed support (blue) and the distributed mass representing RO (red)

The load case for the optimization is 20G in all directions, as in the Table 5.2.

The structure is modelled with shell elements (SHELL184), in order to reduce the computational time cost, in a total of 17117 elements, as depicted in Figure 6.8.

Figure 6.8 Tertiary Structure Initial Mesh for TO with 17117 SHELL181 elements

The TO parameters are defined in the Table 6.2

p	$MaxIter$	$ConvTer\_Rel$ [%]	x_{min}	f [%]
3	500	0.01	0.001	40

Table 6.2 TO Parameters for Tertiary Structure

The result of TO and the post processed solution are shown in Figure 6.9.
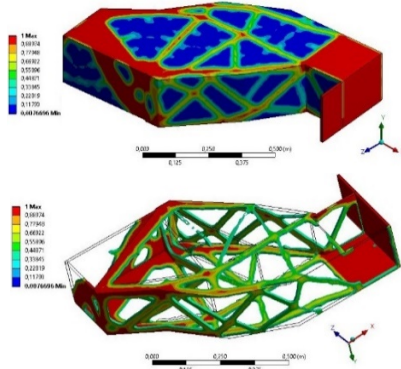
Figure 6.9 Tertiary Structure TO solution with the total length of relative density (top) and a relative density higher than 0.4 (bottom)

Graphics of the objective function history can be seen in Figure 6.10. A convergence can be observed with a compliance result of 39.63Nm.
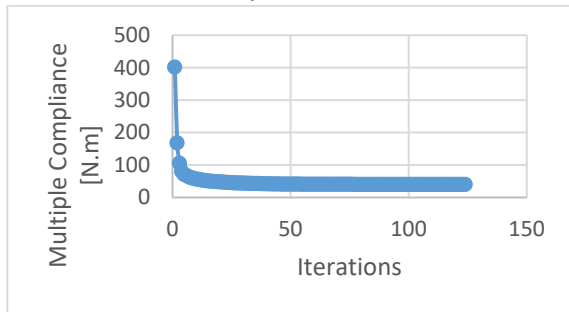

Figure 6.10 Tertiary Structure TO objective function history obtained from ANSYS Wb

6.3. Optimized Tertiary Structure Analysis and Validation

A new tertiary structure design is defined, taking into account the results of Figure 6.9. The boundary conditions of this structure are depicted in Figure 6.11.
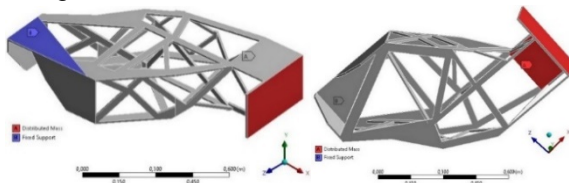

Figure 6.11 Optimized Tertiary Structure Boundary Conditions with fixed support (blue) and the distributed mass representing RO (red)

The structure is model as a solid structure where a SOLID185 mesh is applied. Having into account a finite elements convergence study, a mesh with 146935 elements is chosen and represented in Figure 6.12.
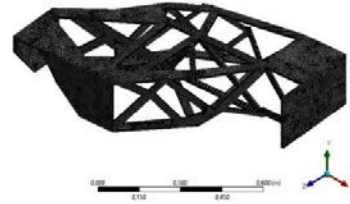

Figure 6.12 Initial Tertiary Structure Mesh with 146935 SOLID185 elements

The maximum stress results for the static analysis of this optimized tertiary structure are depicted in Table 6.3.

Load Case	Max. σ_{VM} [MPa]	MoS_Y	MoS_{UTS}
1	19,61	10,42	6,90
2	51,06	3,39	2,04
3	21,17	9,58	6,32

Table 6.3 Optimized Tertiary Structure Static Analysis Solution from ANSYS Wb

In Figure 6.13 the stress results are analyzed for the load case 2, being the case with worst results.
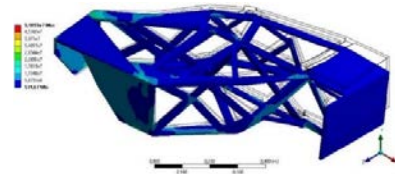

Figure 6.13 Optimized Tertiary Structure Static Analysis Solution obtained from ANSYS Wb, the σ_{VM} (Pa) for the 2nd load case

Analyzing the vicinity of the maximum stress locations (fixed region and bottom region with the satellite) the stress is 25MPa, approximately, corresponding to a $MoS_Y = 7.96$ and $MoS_Y = 5.2$. The first natural frequency, depicted in Figure 6.14 is of 71.85 Hz.
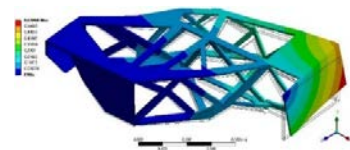

Figure 6.14 Optimized Tertiary Structure Modal solution obtained from ANSYS Wb, representing the Total Eigenvector Deformation (m) for the 1st Natural Frequency of 71.9Hz

6.4. Comparison between Initial and Optimized Tertiary Structure

In this section is presented a comparison between the initial (section 5.1) and optimized (section 6.3) tertiary structure. The results of the static and modal analysis reveal a decrease of 91.79% in the maximum Von Mises stress for load case 2 and an increase of 505.0% in the 1st natural frequency of the structure.

9

Also a comparison between the initial, the design based on the 2D TO (Figure 6.6) and the optimized tertiary structures masses is made and shown in Table 6.4.

Structure	Mass [Kg]	Diff. [%]
Initial	15,93	
Based in 2D TO	46,40	+10,43
Final	17,59	

Table 6.4 Comparison of Mass between Tertiary Structures stages

The difference between initial and final design is +10.43%, but the expansion of the initial design to a design based on the 2D TO have an increase of 191.3% on the mass. After that expansion, a decrease of 62.1% in the mass occurs with the TO for the final design. This reduction is near to the expected reduction, $f = 40\%$, corresponding to an error in the equality constrain of 5.2%.

7. Conclusions

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

- A successful TO comparison was conducted with the result in literature for MatLab codes [8], [10] and the ANSYS TO module.
- A methodology for the TO analysis was introduced. A distinguish characteristic is the introduction of a two steps TO, 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 TO and represents a computational time saving in the 3D TO.
- Finally, using the proposed methodology, the design of the satellite tertiary structure was optimized.

In terms of further development there are some relevant subjects:

- Study the use of other objective functions, such as 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. Acknowledgments

9. References

[1] M. P. Bendsøe and N. Kikuchi, "Generating optimal topologies in structural design using a homogenization method," *Comput. Methods Appl. Mech. Eng.*, vol. 71, no. 2, pp. 197–224, 1988.

[2] M. P. Bendsøe and O. Sigmund, *Topology Optimization: Theory, Methods, and Applications.* Springer Berlin Heidelberg, 2003.

[3] E. Herderick, "Additive manufacturing of metals: A review," *Mater. Sci. Technol. Conf. Exhib. 2011, MS T'11*, vol. 2, no. 176252, pp. 1413–1425, 2011.

[4] D. Brackett, I. Ashcroft, and R. Hague, "Topology Optimization for Additive Manufacturing," *J. Chem. Inf. Model.*, vol. 53, no. 9, pp. 1689–1699, 2011.

[5] ANSYS Inc., "ANSYS Help Viewer." 2015.

[6] J. N. Reddy, *An Introduction to the Finite Element Method*, 3rd ed. New York: McGraw-Hill, 2006.

[7] O. Sigmund, "Design of Material Structures using Topology Optimization," no. December 1994, 1994.

[8] O. Sigmund, "A 99 line topology optimization code written in matlab," *Struct. Multidiscip. Optim.*, vol. 21, no. 2, pp. 120–127, 2001.

[9] E. Andreassen, A. Clausen, M. Schevenels, B. S. Lazarov, and O. Sigmund, "Efficient topology optimization in MATLAB using 88 lines of code," *Struct. Multidiscip. Optim.*, vol. 43, no. 1, pp. 1–16, 2011.

[10] K. Liu and A. Tovar, "An efficient 3D topology optimization code written in Matlab," *Struct. Multidiscip. Optim.*, vol. 50, no. 6, pp. 1175–1196, 2014.

[11] P. R. A. Fernandes, "Optimização da topologia de estruturas tridimensionais," UTL, Instituto Superior Técnico, Lisbon, 1998.

[12] M. M. Neves, O. Sigmund, and M. P. Bendsøe, "Topology optimization of periodic microstructures with a penalization of highly localized buckling modes," *Int. J. Numer. Methods Eng.*, vol. 54, no. 6, pp. 809–834, 2002.

[13] M. P. Bendsøe, "Optimization of structural topology, shape, and material." Springer Berlin etc, 1995.

[14] D. Brackett, "Exploiting the design freedoms & constraints of additive manufacturing." Advances in Metals Manufacturing Technologies Conference, 2014.

[15] G. F. Abdelal, N. Abuelfoutouh, and A. H. Gad, *Finite Element Analysis for Satellite Structures: Applications to Their Design, Manufacture and Testing.* Springer Science & Business Media, 2012.