XL CILAMCE
IBERO-LATIN AMERICAN CONGRESS ON COMPUTATIONAL METHODS IN ENGINEERING
NOVEMBER 11-14, 2019
Praiamar Natal Hotel & Convention
Natal, RN-BRAZIL

Topological Optimization Software Tools: Literature Review and Real Application

Nágera, Maicon Júnior B.
maicon.j.nagera@hotmail.com
Toniasso, Pedro H.
pedro.toniasso@hotmail.com
Slaviero, Guilherme M.
GuilhermeSlaviero@hotmail.com
de Souza, Carlos E.
carlos.souza@ufsm.br
Rodríguez, Rene Q.
rene.rodriguez@ufsm.br
Department of Mechanical Engineering, Federal University of Santa Maria, Brazil.

Abstract. The sake for the optimal condition or shape is inherent in us. Some optimization examples are visible even in nature. In metals, for example, atoms tend to take positions of minimum energy in order to form unit cells, which define the crystalline structure of these materials. In a zero gravity condition, a liquid droplet forms a perfect sphere, which has the least surface area for a given volume. The first historical example of structural optimization is present in Galileo's work, in which he sought the optimal structural shape of a cantilever beam. Since then, the optimization field has growth exponentially. In this context, not only significant evolution in the theoretical field is necessary, but also, computational tools capable to follow this growth are compulsory. New numerical methods have also been developed, such as the Finite Element Method (FEM), distinguished for its accuracy and high adaptability for a broad type of physical problems. Several software tools for topology optimization based on the FEM are available on the market, however, few or none information about the optimization method or optimization parameters are explicitly presented. This work aims to fulfill this conceptual lack, as well as, to show an extensive review of topology optimization tools available for common users, such as commercial softwares or apps. Thus, a realistic example of topology optimization is presented to show the feasibility of constructing an optimized mechanical component using the computational tools described on this work.

Keywords: Topological Optimization, Software, Finite Element Method (FEM).

1 Introduction

In recent years there has been an increasing demand in the area of structural design using software as practical tools for optimization, especially Topological Optimization. To respond to this recent needs, Finite Elements Analysis software developers have been putting a lot of effort to make their programs more efficient and accessible. Complex processes that once required numerous lines of code today can be performed without prior knowledge of optimization theory. With the broader access of this feature, the conceptual design process has become more efficient in early design stages, as the optimal form of a model can be obtained via computational simulation, without the need to invest time and money in prototyping, for example.

The present work aims to define topological optimization, differentiating it from other types of optimization, reinforcing its advantages and limitations, with examples of use in the real world. It will also address different methods and algorithms for topological optimization used by leading softwares available in the market such as ANSYS, ABAQUS, CREO, among others. Finally, the application of these tools shows that the optimization task can be easily accomplished using the appropriate methodology. Modeling results in conjunction with additive manufacturing can make theoretical examples take shape in a realistic way. Two examples of the use of optimization tools are herewith presented. This to show the feasibility of constructing an optimized mechanical component using computational tools described on the present work.

2 Theoretical Background

2.1 Optimization

The optimization process can be defined, according Cheng [1], as the search for the values of certain variables that lead to an optimal design, given conditions and constraints. This process is commonly interpreted as a set of numerical and mathematical techniques that aim to find the best performance, which is particular to each project, Silva [2]. Historically, firsts steps were based on analytic methods, which restricted the application and implementation to a limited class problem. It stands out from these first methods, in the 19th century yet, the work of Maxwell [3], in which he sought to minimize the volume of axial structures submitted to a single load.

Already in 1904, Michell continues Maxwell's work, generating expressive, even nowadays, results in the theory of Topological Optimization. Michell optimized the weight of truss structures given the stress in each bar. Although poorly recognized by contemporaries, Michell's work came to be referenced in the late 1950's in Cox's works [4]. However, the spread and popularization of numerical optimization techniques in engineering projects came only in 1960 with Schmit [5], and later in 1965 with Fox; by applying nonlinear optimization methods to structures. But, in practice, they were very intuitive and lacked a mathematical solidity; problem that only comes to be solved due to the computational technological revolution that is triggered at the end of the 20th century.

In a mathematical approach, the optimization process is basically a minimization or maximization of one or more functions, called objective functions, such as weight, stress, or even cost. These functions are defined by a parameter groups, that characterize the structural system and are called project variables. Added to the objective function and project variables, the constraints are restrictions, physical and mathematical, on the variables imposed by the problem. These constraints can be equalities or inequalities. A generic classical formulation of an optimization process takes the following form:

Minimize (or Maximize) $\rightarrow f(x)$, subject to:

$$g_j(x) \leq 0, \qquad j = 1, 2, 3, \ldots, p$$
$$h_k(x) = 0, \qquad k = 1, 2, 3, \ldots, q$$
$$x_i^L \leq x_i \leq x_i^U \qquad i = 1, 2, 3, \ldots, n$$

where x is the design vector; $f(x)$ is the objective function; $g(x)$ is the vector with inequality restrictions; $h(x)$ is the vector with equality restrictions; x_i^L and x_i^U are the low and upper design limits, respectively.

According Bendsøe and Sigmund [2], structural optimization is classified into three categories: parametric or dimensional, shape and topological. In parametric optimization, there is no change in shape or topology of the structure, varying only constructive material characteristics and/or geometric dimensions. In shape optimization, the topology remains fixed, changing the internal and external contours in the structure. This type, according to Coutinho [6], requires greater sophistication in the numerical implementation compared to the first. Already in topology optimization, the idea is to look for the best distribution of material in a given domain given certain conditions and constraints. According to Fonseca [7], in this optimization type, particular care should be taken with the physical and mathematical stability of the problem, since certain elements removal can make the structure hypostatic, for example.

Another interesting way to classify optimizations is by the method, and it is divided into deterministic and probabilistic or heuristic. In deterministic, the exactly optimal point - or the closer than its possible, is sought in a continuous range. Its reasonable, in these problems, the objective function be continuous and first-derivable in the domain. In heuristic methods, probabilistic analyses are used to determinate viable solutions for a specific problem. However, this cause-effect relationship is not totally defined; therefore, the method covers problems that reproduce, in a simplified way, phenomena and processes of nature, and due this they can also be called Natural Computing (also called Meta-Heuristics) [8].

2.2 Topological Optimization

Structural optimization seeks to achieve the best performance for a structure while satisfying various constraints such as a given amount of material [9].

Topology optimization may greatly enhance the performance of structures for many engineering applications. It has been exhaustively studied and various topology optimization methods have been developed over the past few decades [10–12].

Optimal structural design is becoming increasingly important due to the limited material resources, environmental impact and technological competition, all of which demand lightweight, low-cost and high-performance structures [9].

Different methods for topological optimization have been developed in recent decades, and computational implementation has made it a rather laborious and repetitive method, fast. Among these methods, evolutionary structural optimization (ESO) and bidirectional evolutionary structural optimization (BESO) stand out.

The ESO method was first proposed by Xie and Steven in 1992 [10]. This method consists in remove inefficient elements in a structure, forming a set of structural and global elements. The number of elements removed is related to the rejection ratio (RR). This method can be stress or strain based.

The stress level of a structure can be obtained from a finite element analysis. The efficiency of each frame element can be measured by the stress level, which leads to a rejection criterion based on the local stress level, where low stress material is considered inefficient and thus removed from the structure [13].

Stiffness is one of the key factors that must be taken into account in the design of such structures as buildings and bridges [9]. With this in mind, ESO based on strain level emerges. In this method, a finite element analysis is performed and elements with lower deformation energy are removed, remaining only elements with high importance in the stiffness of the structure.

The bi-directional evolutionary structural optimization (BESO) method allows material to be removed and added simultaneously. The initial research on BESO was conducted by Yang et al. [14] for stiffness optimization [9]. This method consists of gradually removing inefficient elements, and estimating the efficiency of elements that would be in the voids, then elements with low structural function are removed, and estimated elements with high structural function are added. The number of elements added and removed are not necessarily equal, and are related to rejection ratio (RR) and inclusion ratio (IR).

From these main methods, and others less known, different software were developed for engineering application, and some of these were analyzed and their functionality will be discriminated during the present work.

3 Softwares for Topological Optimization

There exist several optimization tools available in the literature, some of which are accessible directly by well-known CAD/CAE softwares, as part of or as an extension of the main software. Some of these softwares and their optimization features are presented, with the objective of giving the reader an initial description of these tools. Thus, a satisfactory choice, of which one is more adequate for a specific problem, can be made. Softwares analyzed include CREO, ABAQUS, ANSYS, COSMOS and NASTRAN.

3.1 CREO

CREO is an interconnected software's group develop by PTC, used as CAD, CAE and CAM tools to support manufactured products development, components and mechanical systems projects; structural, thermal or modal simulation, among others. The first version, CREO 1.0, was released in January 2011; while the latest, CREO 6.0, in March 2019. All of them were develop for Windows operational system. Among the many features available, stands out solid 3D modeling, Finite Elements analyses, simulation and topology optimization.

The latter one is a relatively recent tool, since its predecessor version (5.0.0.0) did not has this option. However, this version came with an optional Beta extension, which was a precursor of topological optimization on CREO. The following version (5.0.1.0) already has the Creo Topology Optimization available, which comes to replace extension Beta. From 5.0.1.0, it exists two possibilities of optimization tools: Creo Topology Optimization & Creo Topology Optimization Plus. Although each one has its particularities, the process employed is common. Figure 1 shows a schematic representation of this process.

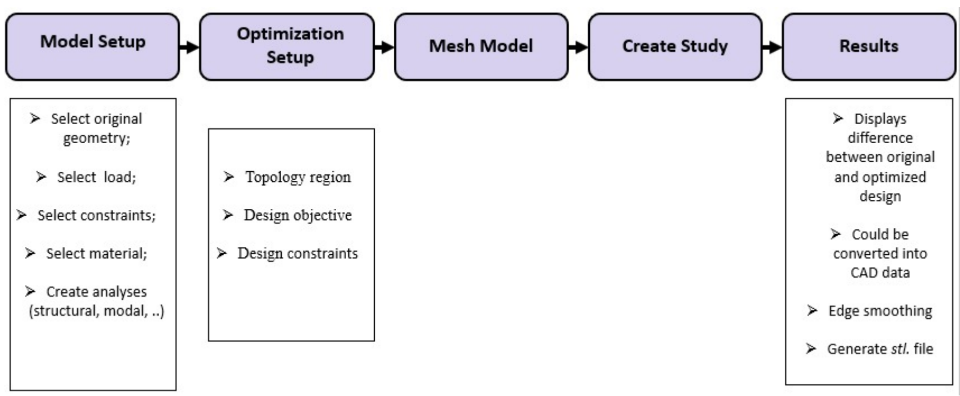


Figure 1. Schematic representation of the topology optimization process on CREO.

In Setup Model, inputs are provided, such as geometry, material, loads and constraints. Then, the analysis type is selected: modal, thermal or structural. Until 5.0.0.0 version, those were all the features available for optimization. The Beta extension allowed, after the analysis completion, a wide range of optimization features, in which was essential to define three parametrers: optimization region, design objective and design constraints.

In optimization region parameter its defined which volume of the body must be topologically optimized, in other words, the volume whose material will be removed, given some stopping criterion. For this, should be chosen some manufacturing and fabrications constraints. Its related to the original body geometry construction. At this point, is useful to distinguish between Creo Topology Optimization & Creo Topology Optimization Plus modes, since one of the advantages of the second over the first is more manufacturing restrictions options. Table 1 brings this difference, besides the analysis type and number of possible analyses,

Table 1. Mainly differences between Creo Topology Optimization & Creo Topology Optimization Plus modes.

	CREO Topology Optimization	CREO Topology Optimization Plus
Fabrication Constraints	Symmetry; Cyclic Symmetry; Extrusion; Filling.	Symmetry; Cyclic Symmetry; Extrusion; Filling; Stamping; Uniform; Filling Symmetrically; Radial Filling; Radial Spokes; Periodic.
Number of analyses	3	Unlimited
Analysis Type	Modal and Structural	Modal, Structural and Thermal

In optimization setup, the design objective refers to the main parameter to maximize, minimize or maintain; for example: total mass, Von Misses stress, stiffness, among others. Design constraints should also be set up in the analysis, such as imposing maximum limits for stress or strain.

After these definitions, the software generates the results, modelling the mesh and creating an automatic analysis; then, presenting the results to the user. Finally, straight edges can still be smoothed to soft curves, thus, avoiding stress concentrators. It is also possible, posteriorly, to make new analyzes on the already optimized body in order to compare with the initial study done in the Model Setup step. Figure 2 presents a topology optimization didactic example showing the model and optimization setups and, subsequently, the results obtained and the comparison with the original geometry.

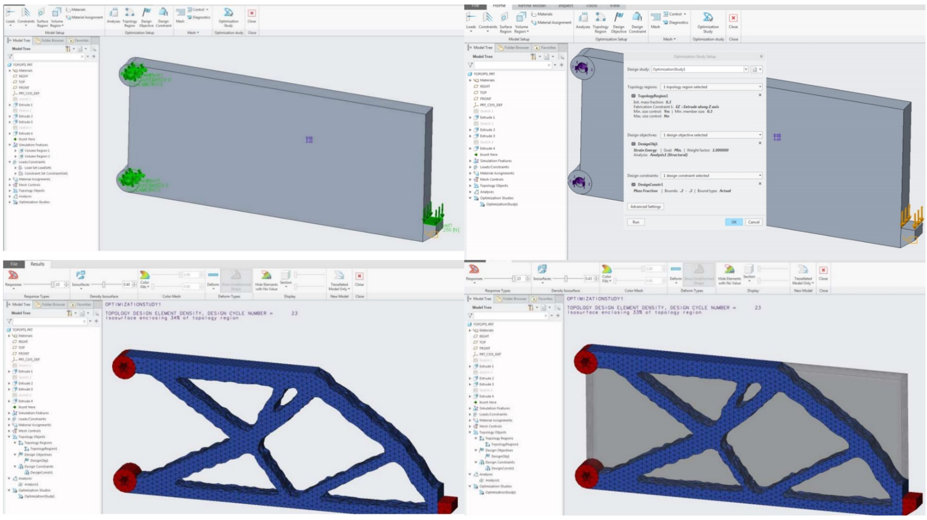


Figure 2. CREO Topological Optimization example.

3.2 ABAQUS

Abaqus co. was founded by professors at Brown University in 1978 under the original name of HKS Inc [15]. The company's first major challenge was in 1970, when the US Navy funded a project that would extend its code capacity, making it possible to predict performance loss in welded submarine hulls due to residual distortion. The order was beyond the capabilities of the computers at the time, but the developers were still able to program a software that was not as computationally costly and at the same time had an interactive interface, features that are still present in current versions. This was an important part of the software automation process strategy - allowing less experienced users to access and apply finite element software.

The topological optimization tool is included in Abaqus since version 6.11, from 2011. The current version is 2019, following the trend of Dassault Systèmes to release a new version of its services yearly. The student version supports meshes limited to 1000 nodes [16] . The programming language used is Python.

Abaqus presents three optimization approaches [17] : topology, format, and sizing. Topology optimization first analyzes the initial shape, and by analyzing the material properties, it effectively removes elements from it. Shape and size optimizations are model refining tools. The shape modifies the surface of the component to reduce stress concentrators; Sizing modifies plate thickness in components that exhibit plate behavior. Generally this refinement aims to increase stiffness or reduce vibration. All of these optimizations are governed by a set of objectives and constraints.

Abaqus supports two optimization algorithms - the general algorithm, which is more flexible and can be applied to most problems, and the condition-based algorithm,which is more efficient but has limited capacity. Each algorithm has a different approach to determining the optimal solution. By default the program's Optimization Module uses the general algorithm.

The general algorithm used by the program is partially described in [18]: In short, it adjusts the density and stiffness of design variables while trying to satisfy objective function and constraints. Density is the design variable and interpolation penalizes intermediate densities. The method used for optimization uses a displacement based on finite element analysis and the optimality criteria taking into account the density of a material with homogeneous distribution. The number of design cycles is undefined until optimization begins, but is usually between 30 and 45 cycles. Thus, the structure of the algorithm is as follows:

Make an initial design. For the density distribution of this design, calculate by the finite element method the resulting displacements and stresses. Then calculate the compliance of this design. If there is little change in compliance compared to the last cycle, stop the iterations. Otherwise continue, which guarantees the loop of iterations. For more detailed studies, stop when optimality conditions are met.

Finally, interpret the optimal distribution of material in a CAD representation. It is noteworthy that the described algorithm can be implemented in any finite element mesh and in any type of reference domain. This ensures the flexibility of the method in terms of defining boundary conditions and fixed areas. However, in many cases it works with rectangular (2D) or box-like (3D) domains, and with the mesh consisting of squares or cubes.

The condition-based algorithm was developed at the University of Karlsruhe, Germany, and is described by Bakhtiary [19]. This algorithm is more efficient because it uses strain energy and node stresses as Input, and it is not necessary to calculate the local rigidity of design variables. This optimization is more efficient and looks for a solution with a maximum of 15 cycles by default. It is an applied energy equation. This equation says that for elastic systems external work is preserved without loss in the deformed system. Thus the displacements depend on the accumulated stress energy.

The Optimization Module iteratively prepares design variables (such as element densities and surface node positions) updating the Abaqus finite element model and performs an analysis. These iterations or design cycles continue until the maximum number of cycles is reached or the stop conditions are met. Figure 3 shows the interaction of Abaqus and the optimization process.

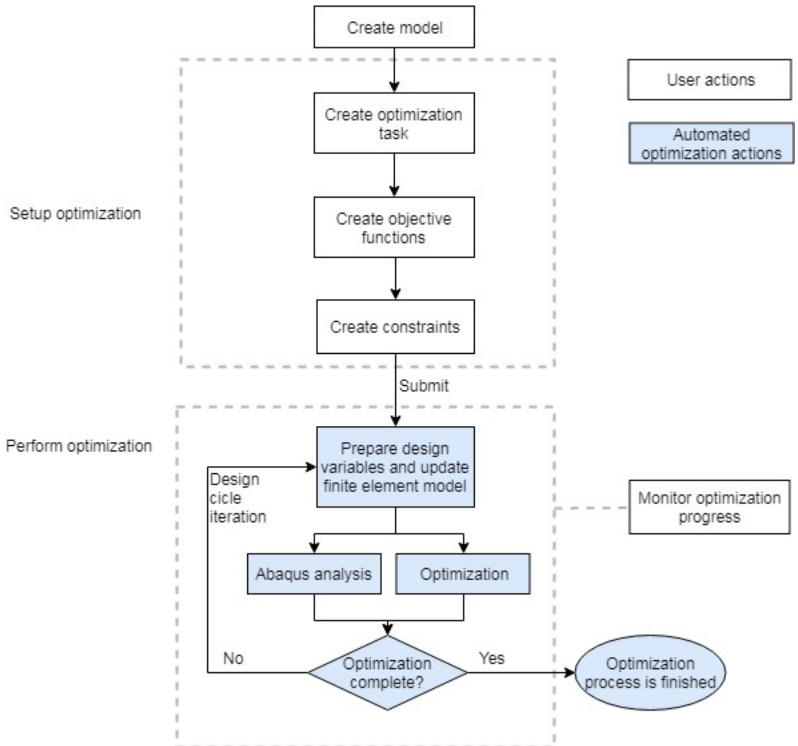


Figure 3. Schematic representation of the topology optimization process on ABAQUS.

As an example, the Abaqus Example Problems Guide (chapter 11.1.1) [20] shows the topological optimization of an automotive suspension arm, as can be seen in Figure 4. The goal was to reduce the volume of this component while maximizing its stiffness. After defining the model and boundary conditions, it was specified that the optimized model should contain 57% of the initial volume of the initial component. This example was governed by the condition-based optimization algorithm. Thus, it was possible to verify the progression of topological optimization through 17 cycles, as seen in the figure:

The software can apply the following objectives for a topological optimization process using the general algorithm: strain energy (a measure of structural stiffness); eigenfrequency; internal forces and reaction forces; weight and volume; center of gravity and moment of inertia. For the condition-based algorithm, the software is capable of applying only the deformation energy and volume objectives.

Abaqus supports add-ons, and because its base language is Python, it benefits a lot of code that complements the software tools. It is worth mentioning, therefore, the work of researchers in creating

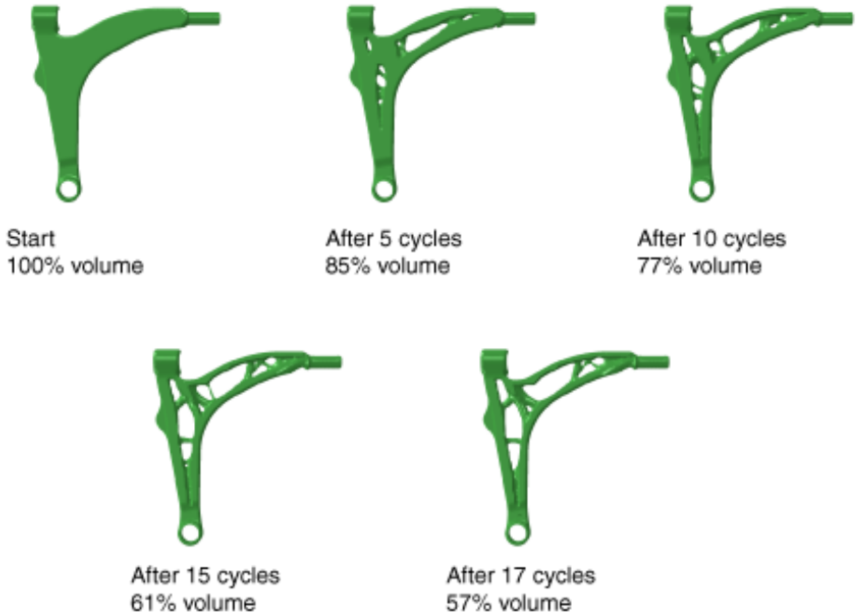


Figure 4. ABAQUS Topological Optimization example.

practical codes, such as Zuo and Xie [21], who developed a 100-line Python code for general 3D topological optimization using the BESO (Bi-directional Evolutionary Structural Optimization) method. This is only possible thanks to a Python language extension, Abaqus Scripting Interface (ASI), which provides a convenient interface for models and results. With the sophisticated modeling capability provided by the software, the code is capable of handling arbitrary shapes of 2D or 3D geometries, not being restricted to simple 2D geometries only.

Codes can be built and adapted to new challenges, with some changes, making use of a convenient platform and practical interface that Abaqus offer. As much as codes like Zuo and Xie are presented for educational purposes, nothing prevents them from being useful to solve engineering design problems by generating more complex conceptual designs.

3.3 ANSYS

ANSYS is an engineering simulation software company. It was founded in 1970 by John A. Swanson, and it was originally named Swanson Analysis Systems, Inc. ANSYS has acquired several companies since 2000, including ICEM CFD Engineering, CADOE, CFX, Century Dynamics, Harvard Thermal, Fluent Inc. (2006) and Ansoft Corporation (2008), thereby, expanding its research and work areas.

The Topology Optimization feature was added on 2018 (ANSYS 18). It was available for both, structural and modal simulations. Since then, this feature has been enhanced and it is also available in the current version (ANSYS 19 R2). The Topological Optimization tool has several advantages, with a wide range of objectives, stopping criteria, which are different for each type of simulation, and manufacturability and/or design constraints, as well as the ease of validating the result obtained.

The ANSYS workbench allows the interaction between the optimization process and a wide range of simulation cases. Thus, opening the possibility of applying this feature to different kind of analyses. The mesh generation is considered to be one of ANSYS main stregths, as well as achieving reasonable good and accurate results with the automatic mesh option. Several mesh parameters are available, permiting different mesh refinements for different regions, for example.

A schematic diagram of how the Topological Optimization is accounted in ANSYS is shown in Figure 5.

User programmable features (UPFs) are available in ANSYS from Mechanical APDL software, a powerful fortran-based scripting language that allows model parametrization and the automatization of

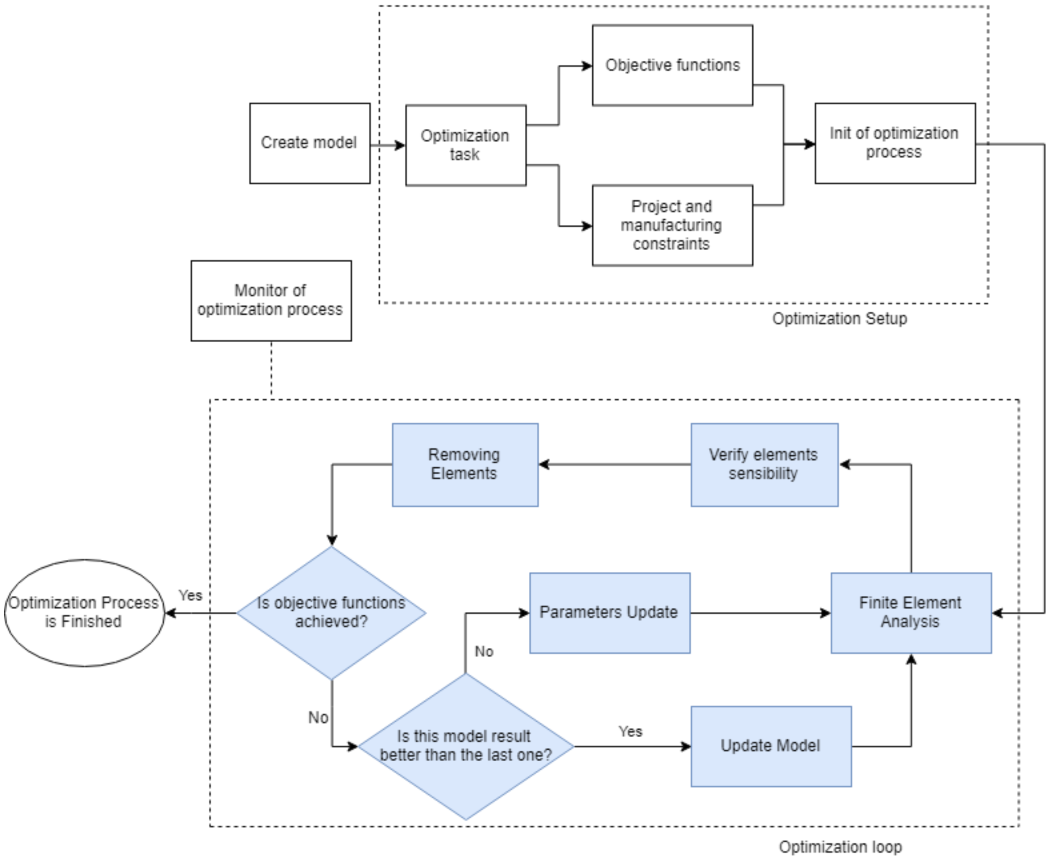


Figure 5. Schematic representation of the topology optimization process on ANSYS.

common tasks. ANSYS scripts can generate data inputs, outputs, perform mathematical calculations and create macros to perform a sequence of tasks.

Figure 6 shows a part optimized from Ansys software, and grayed out its initial shape before the process was performed.

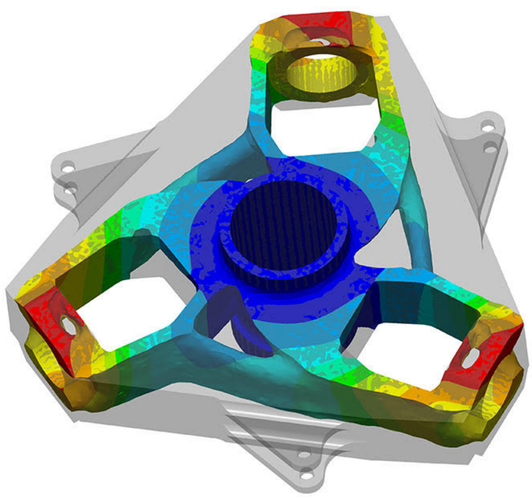


Figure 6. Example of an optimization problem in Ansys. Source: `https://www.ansys.com/products/structures/topology-optimization`

In addition, Ansys Blog allows the user to read about different applications for the software, addressing interesting and specific points around new technologies.

3.4 COSMOS (SolidWorks Simulation)

Cosmos was initially a standalone program from SRAC (Structural Research & Analysis Corp.). It was purchased by Dassault Systèmes in 2001 [22]. In 2006, with the proposal of integrating validation in the design process with a practical and interactive interface, the simulation software was embedded into SolidWorks®, becoming a component of it. With the release of SolidWorks 2015, Cosmos services were discontinued [23] and now operate under the SolidWorks Simulation name. Today, it features three distinct versions - Standard, Professional, and Premium.

All versions are integrated with SolidWorks®, and use the same interface, but are divided by their complexity [24]: the Standard version is suitable for static and fatigue studies. However, this version does not feature the thermal analysis, frequency studies, buckling, pressure vessel, or topology functions that are present only in the Professional and Premium editions. The last version differs from the others because it contains the option of linear dynamics studies and nonlinear analysis.

The topological optimization tool came to SolidWorks Simulations only in 2018, later compared to other software [25]. It requires few user inputs (loads, design space, constraints, boundary conditions and manufacturing methods) and then runs a subtractive algorithm. To achieve this, the program uses the Tosca structural optimization engine, which is the standard finite element solvers in the industry such as Ansys and Abaqus.

3.5 NASTRAN

Its developer is MSC Software, which created the software from a partnership with NASA in the 1960s. The goal was to develop a generic program that engineers could use to more efficiently model and analyze aerospace structures, as the already retired space shuttles. In the 70's NASA launched the commercial version of the software, which was widely used in the automotive industry. To this day, the software is one of the main finite element analysis programs, and several features are implemented with each release.

One of such innovations was the release of the topological optimization feature, which was introduced along with the release of MSC Nastran 2005 [26]. In the current release, the software already features resources that have been incorporated to meet the demands of the industry. Some features such as: smoothing the checkerboard effect; large number of manufacturing constraints, such as cyclic symmetry, extrusion constraint for uniform thickness in the direction of extrusion, or casting constraints to prevent cavities.

4 Results

Two examples are presented in order to show the real application of the topology optimization feature available in the commercial software ANSYS. In the first example, a well-known literature example is shown. A supported beam with a central load is optimized having as the objective function its minimum compliance. Moreover, the optimized piece was smoothed and printed in 3D, showing that the process is straightforward and results are manufacturable. On the second example, a piece from the racing competition Formula SAE is optimized. Results are compared with those obtained from a manual optimization process, obtaining better results when the optimization feature available on ANSYS was used.

4.1 Example 1

A bi-supported beam with a central load is commonly found in the topological optimization literature. Geometry and boundary conditions are considered as presented by Xie and Huang [9]. The main objective is to demonstrate the effectiveness of the topological optimization process by confronting results with the literature. Moreover, the optimized piece was produced through additive manufacturing. The initial example can be seen in Figure 7.

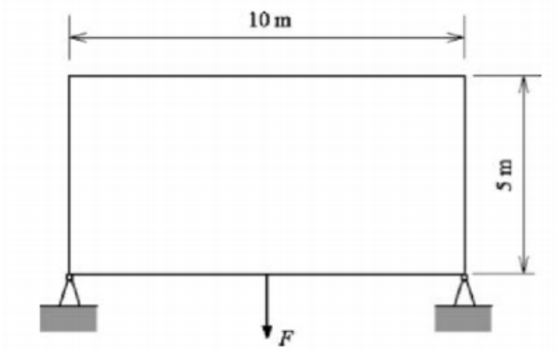


Figure 7. Bi-supported beam, [9]

Dimensions were reduced in order to give a more manufacturable component. Thus, the main purpose of the present example is to show that the optimized piece is feasible to be constructed by additive manufacturing. Original dimensions were 5 x 10 m, while a geometry of 50 x 100 mm was herewith considered.

The objective function was the minimization compliance, having as the stopping criterion a final volume of 10% of the initial volume. The result of the topology optimization can be seen in Figure 8.

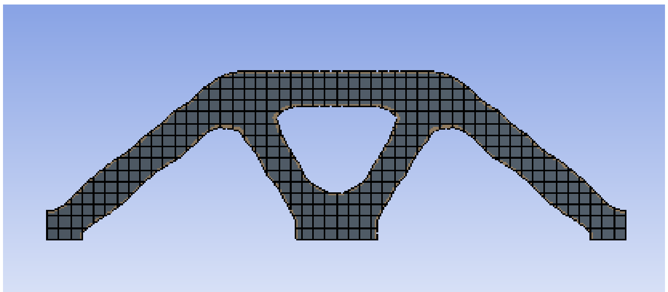
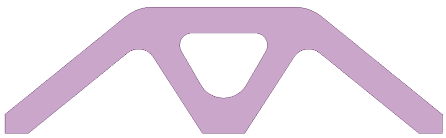


Figure 8. Topology optimization final structure.

The optimized structure given by ANSYS first analysis is not the final result of the process. Further analysis and refinement are still required so as to obtain the final manufacturable model. The final optimized geometry, already refined and validated, can be seen in Table 2.

Table 2. Comparison between the initial geometry and the final manufacturable structure.

	Case 1	Case 2
Description	Start piece without any kind of optimization	Result from the topological optimization in Ansys (smoothed)
Representation		
Volume (mm^3)	50.000	10.956
Maximum stress (MPa)	3	33,2

Some further modifications were accounted for giving the piece the possibility for mechanical testing. Since the piece will be used in a destructive test, it needs a geometrical place to apply the mechanical load. The modified geometry with the point of load application can be seen in Figure 9 and Figure 10.

The 3D printing was achieved using a Stratasys F170 printing machine. Due to its simple geometry, the process took only 27 min to be completed. Figure 9 (a) shows the interface of the Stratasys F170 printing machine, while Figure 9 (b) shows the final printed structure with support material.

(a) (b)

Figure 9. 3D printing process: (a) Stratasys F170 interface, (b) Final printed piece.

Finally, the printed component without the support material is shown in Figure 10. The present main objective was to show that results from topological optimization are feasible to be manufacturable. Next steps will consider mechanical testing of the printed pieces.

Figure 10. Topology optimization final structure.

4.2 Example 2

The main objective of the present example is to show a more applicable design optimization case. The optimization process will be also accounted using the commercial software ANSYS. The piece to be optimized is a front wing fixation component of a racing car prototype made by the Formula UFSM team for a racing competition.

Boundary conditions for the static structural simulation have already been defined by the Formula UFSM team, therefore, they are not modified in the present work. For comparison purposes, the part was manually optimized by a member of the prototype development team. The results obtained, maximum stress and reduced volume, were used as stopping criteria in two topological optimizations cases with objectives for volume minimization and compliance. Since the part to be optimized will be laser cut on a steel plate, a manufacturability constraint must be applied. The restriction used in this case is extrusion, which limits the topological optimization to two axes. An optimization region exclusion was also applied to the faces where the part will be fastened, as these regions are necessary for the correct fixation of the components. It is important to mention that the result of topological optimization is not the final result of the process, further analysis and manual refinement are still required to obtain the final model and in order to validate the results.

Table 3 shows results from the two optimization cases. In both cases the main objective function is the minimum compliance. However, in the first case, optimization (a), the restriction is the final volume (set up to 85% of the initial volume); while, in the second case, optimization (b), the restriction is a maximum stress of 259 MPa.

Table 3. Results for the optimization process.

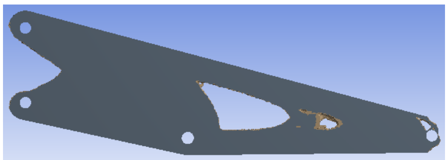
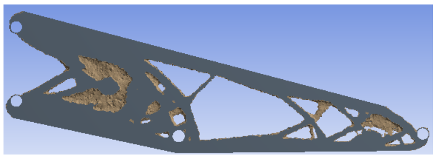
	Optimization (a)	Optimization (b)
Objective	Compliance minimization	Compliance and volume minimization
Stopping criterion	85% of the initial volume	Maximum stress of 259 MPa
Constraints	Manufacturing by extrusion and Fastening holes	
Iteration	15	35
Result		

Table 4. Comparison between manual and ANSYS optimization.

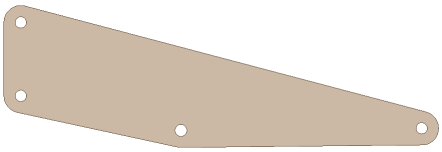
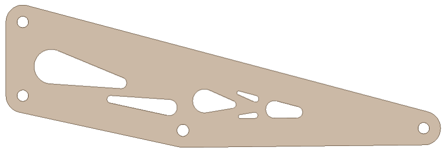
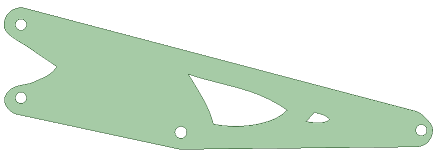
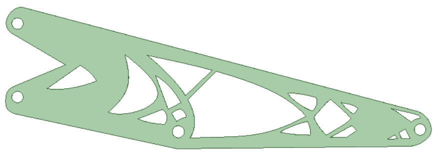
	Case 1	Case2
Description	Start Piece, without any kind of optimization.	Part optimized by prototype development team member.
Result		
Volume (mm^3)	19.588	17.099
Maximum stress (MPa)	216,59	259,06
	Case 3	Case4
Description	Topologically optimized part in Ansys software aiming to achieve the same reduced volume of Case 2.	Topologically optimized part in Ansys software aiming to achieve the same maximum stress of Case 2.
Result		
Volume (mm^3)	16.846	12.290
Maximum stress (MPa)	224,03	244,00

Table 4 show final results from the manual optimization, case 1 and 2; and results obtained from ANSYS optimization (already smoothed), case 3 and 4.

5 Conclusions

The present work presented a brief review of main softwares that have topological optimization as a feature. Each software was described showing its main characteristics, giving the reader a wide range of options. Furthermore, the commercial software ANSYS was used for explaining how the optimization process is accomplished, from its numerical idealization to its real conception. It was shown that the use of commercial softwares for topological optimization is bridging the gap between the theory and real applications, exposing a huge potential for solving real complex problems.

Acknowledgements

The authors would like to thank the Formula UFSM team for providing data for our optimization problem; as well, as to thank Prof. Alexandre Buenos, NAFA director, for gently supplying NAFA facilities and his technical support during the 3D impression.

References

[1] Cheng, G. & Jiang, Z., 1992. Study on topology optimization with stress constraints. *Engineering Optimization*, vol. 20, n. 2, pp. 129–148.

[2] Silva, K. E. S., 2018. Otimização topológica evolucionária de estruturas. Trabalho de Conclusão de Curso (Graduação em Engenharia Civil), Universidade do Estado de Mato Grosso, Mato Grosso, Brazil.

[3] Maxwell, J. C., 1870. I.—on reciprocal figures, frames, and diagrams of forces. *Earth and Environmental Science Transactions of the Royal Society of Edinburgh*, vol. 26, n. 1, pp. 1–40.

[4] Cox, H. L., 1958. The theory of design. *Aeronaut. Res. Council Rep*, vol. 1, n. 19791.

[5] Schmit, L. A., 1960. Structural design by systematic synthesis. In *Proceedings of the Second National Conference on Electronic Computation, ASCE, Sept., 1960*.

[6] Coutinho, K. D., 2006. Método de otimização topológica em estruturas tridimensionais. Master's thesis, Universidade Federal do Rio Grande do Norte.

[7] Fonseca, M., 2007. Otimização de estruturas treliçadas planas e espaciais sob carregamentos estáticos e dinâmicos, usando algoritmos genéticos e redes neurais. Master's thesis, Universidade Federal de Ouro Preto - UFOP, Ouro Preto.

[8] Olivieri, B. P., 2004. Otimização do projeto de pontes protendidas pré-moldadas pelo método dos algoritmos genéticos. Trabalho de Conclusão de Curso, Universidade Federal do Rio de Janeiro, Rio de Janeiro, Brazil.

[9] Huang, X. & Xie, Y., 2010. *Evolutionary Topology Optimization of continuum structures - Methods and Applications*. John Wiley and Sons.

[10] Xie, Y., Steven, G., & of Sydney. Finite Element Analysis Research Centre, U., 1992. *A Simple Evolutionary Procedure for Structural Optimization*. Finite Element Analysis Research Centre, University of Sydney.

[11] Xie, Y. M. & Steven, G. P. A., 1993. A simple evolutionary procedure for structural optimization. *Computer and Structures*, vol. 49, pp. 885 – 896.

[12] Xie, Y. M. & Steven, G. P. A., 1997. *Evolutionary Structural Optimization*. Springer, London.

[13] Huang, X. & Xie, Y. M., 2007. Convergent and mesh-independent solutions for the bidirectional evolutionary structural optimization method. *Finite Elements in Analysis and Design*, vol. 43, pp. 1039 – 1049.

[14] Yang, X. Y., Xie, Y. M., Steven, G. P., & Querin, O. M., 1999. Bidirectional evolutionary method for stiffness optimization. *AIAA Journal*, vol. 37, pp. 1483 – 1488.

[15] Systèmes, D., 2019. Analysis of the origins of abaqus. `https://www.nafems.org/blog/posts/analysis-origins-abaqus/`. Accessed: 2019-07-26.

[16] Systèmes, D., 2017a. *Abaqus 6.14 - Getting Started with Abaqus: Keywords Edition*. Dassault Systèmes.

[17] Systèmes, D., 2017b. *Abaqus 6.14 - Abaqus Analysis User´s Guide*. Dassault Systèmes.

[18] Bendsøe, M. P. & Sigmund, O., 2003. *Topology Optimization: Theory, Methods and Application*. Springer.

[19] Bakhtiary, N., Allinger, P., Friedrich, M., Mulfinger, F., Sauter, J., Muller, O., & Puchinger, J., 1996. A new approach for size, shape and topology optimization. *SAE International Congress and Exposition*, vol. 1, pp. 6 – 7.

[20] Systèmes, D., 2017c. *Abaqus 6.14 - Abaqus Example Problems Guide*. Dassault Systèmes.

[21] Zuo, Z. H. & Xie, Y. M., 2015. Simple and compact python code for complex 3d topology optimization. *Advances in Engineering Software*, vol. 85, pp. 1 – 11.

[22] Staff, D., 2001. Dassault systèmes acquires SRAC. `https://www.designnews.com/automation-motion-control/dassault-systemes-acquires-srac/118613072235993`. Accessed: 2019-07-26.

[23] 2006, B. W., 2006. Dassault systemes : Cosmos 2007's simplicity and ease of use encourages designers to innovate by validating how their designs will perform in actual use. `https://www.marketscreener.com/DASSAULT-SYSTEMES-9005/news/Dassault-Systemes-COSMOS-2007-S-Simplicity-and-Ease-of-Use-Encourages-Designe` Accessed: 2019-07-26.

[24] Briand, C., 2016. Where did COSMOSM / GEOSTAR go? `https://www.javelin-tech.com/blog/2016/04/where-did-cosmosm-geostar-go/`. Accessed: 2019-07-26.

[25] Solidworks, 2019. Solidworks simulation products. `https://www.solidworks.com/pt-br/product/solidworks-simulation`. Accessed: 2019-07-26.

[26] Nastran, 2007. *User's Guide for Topology Optimization*. Nastran.