



24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

24th COBEM - 2017

COBEM-2017-2883

A TOPOLOGY OPTIMIZATION SOLVER APPLIED TO 3D COMPLIANT MECHANISM DESIGN

Marcos Antônio Silva dos Reis

Cícero Ribeiro de Lima

Federal University of ABC, School of Engineering, Modeling, and Applied Social Sciences, São Bernardo do Campo - SP, Brazil
e-mail: marcos.reis@ufabc.edu.br¹; cicero.lima@ufabc.edu.br²

Abstract. *Compliant mechanisms are monolithic structures, in which the deformation property is explored as source of its motion. These mechanisms have wide application in devices that involve precision mechanics, such as MEMS actuators. In this work, the topology optimization method (TOM) powered by an integration routine between MATLAB™ and ANSYS™ software are implemented to design 3D compliant mechanisms. TOM is a powerful optimization algorithm to distribute material inside a design domain in order to obtain better structural layout to the compliant mechanism. The easy communication between both softwares (MATLAB and ANSYS) provides an efficient alternative to hand hundreds of thousands of design variables and solve large FE equation systems necessary to design 3D mechanical structures. The well-know SIMP (Solid Isotropic Material with Penalization) and the Optimality Criteria method (OC) have been applied respectively as material model and numerical optimization solver for TOM. One of results are shown to illustrate the potential of the implemented algorithm.*

Keywords: *compliant mechanism, topology optimization, three-dimensional structure*

1. INTRODUCTION

Compliant mechanisms can be defined as a new class of mechanisms where the transmission of force and movement occurs due to the deformation of their own body (Howell, 2001). Among some advantages of the compliant mechanism is the reduction of components. (Howell et al., 2013). In many cases, only one piece is necessary for this device to perform the task associated with the desired movement, making it easier to manufacture devices with small dimensions. TOM is a methodology that has been studied for decades so that it can extract the best distribution of material in a certain domain considering mass reduction and other constraints. Among the various techniques available, the SIMP has been gaining popularity due to its relative simplicity. In this approach, the design variables are considered virtual densities (Bensøe and Sigmund, 2004).

In most of the studies published for this topic the 2D domain is applied, because of the high computational cost of 3D systems, however, modeling from the flat system does not always allow real analysis of the behavior of the mechanism. Thus, it is reasonable to consider the use of three-dimensional (3D) numerical simulations to predict the behavior of these mechanisms. However, 3D computer simulations increase program execution time and require more processing hardware. Therefore, options to speed up the execution of the routine performed by MOT are necessary.

Few works in the literature have addressed this topic. Tovar and Liu (2014) developed a 3D routine using the Matlab program to generate compliant mechanisms, where the finite element stage is calculated using hexahedral elements, which allows good simulations for regular domains. Gain et al. (2015) used the Virtual Element Method (VEM) approach, where the mesh was constructed with polyhedral elements intended to reduce computational cost and perform simulations on large scale problems. Zegard and Paulino (2016) used the TOPslicer program to inspect the result generated by the optimization and generate the file in the appropriate format to manufacture the part by additive manufacture.

Companies are investing devices with new features that enable better performance for products. This new market can consume products of relatively large size in three dimensions. MOT can be applied from the product design stage, determining optimal initial layout, generating new functions for old products and reducing manufacturing cost. Building compliant mechanisms is not such an intuitive task, and fine-tuning the design tools can help significantly. Considering this scenario, the authors believe that research on systematic methods to construct compliant mechanisms can bring

contributions, taking advantage mainly of scientific advances and the vast availability of methods of analysis and optimization. The aim of this work is to analyze the potential of the shared routine between ANSYS™ and MATLAB™, by applying it in simulations involving 3D compliant mechanisms. Using the ANSYS framework for the finite element calculation can be a great advantage, since this software is well established in the market and has several algorithms (solvers) to solve large scale equation systems of the finite element analysis, in addition to its easy communication with other software. MATLAB is a software that has become popular in the academic field due to its ability to handle numerical analysis, allowing to represent the problems mathematically, even using a programming structure equivalent to other languages, such as C and Fortran.

2. SYNTHESIS OF COMPLIANT MECHANISM

The topology optimization method (TOM) is the most general approach for the synthesis and design of compliant mechanisms, where a computational algorithm systematically attempts to find the best material distribution or connectivity pattern in a predetermined region of the design domain (Bensøe and Sigmund, 2004). This technique is very useful when one does not have in mind the format of the mechanism to be generated, but one is aware of the load and the boundary conditions to be applied. Using the TOM to create these mechanisms, the designer can obtain very different and original solutions, if compared to the result obtained by another method (Howell et al., 2013). At each iteration, the TOM algorithm finds an improved solution, changing the value of the problem variables (densities), which change between zero (without material) and one (with material) at each point of the design domain, discretized by finite elements.

The steps used by the method to obtain the solution of the optimization of compliant mechanisms will be presented here in a summarized sequence: (1) define the design domain (volume for the 3D case); (2) indicate the point of application of the load); (3) indicate the point where the workpiece displacement will occur when the load is applied (output); (4) indicate the minimum amount of material that the final topology should have (constraint); and finally (5) execute the algorithm of the method to find the topology of the desired mechanism. At each iteration, the TOM algorithm finds an improved solution (Bensøe and Sigmund, 2004). Figure 1 show basic representation this process.

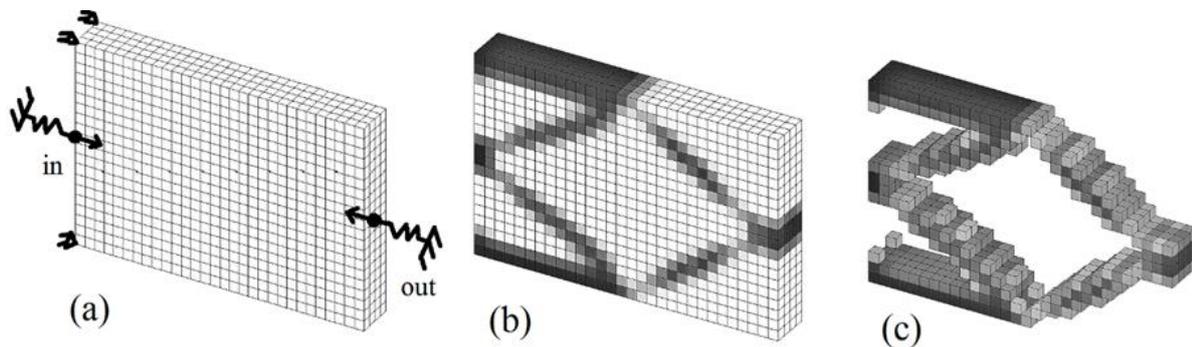


Figure 1 - Inverter mechanism, (a) design domain, (b) result obtained by the algorithm, (c) final topology configuration of the compliant mechanism.

Some theoretical aspects are considered for the use of MOT, so the next sections briefly describe the concepts employed.

2.1. Material model

The density method, also known as SIMP (Solid Isotropic Material with Penalisation), was introduced to MOT by Bensøe and Kikuchi (1989). The SIMP consists of a mathematical equation that defines the value of the design variables (densities) in each point of the domain. In the SIMP the relationship between the design variables and the property of the material is given by the Eq. (1).

$$C(x) = \rho(x)^p C_0, \quad 0 \leq \rho(x) \leq 1, \quad x \in \Omega \quad (1)$$

Where $\rho(x)$ is considered to be a continuous function of density distribution, such that the density can vary from zero (void) to one (solid) along the domain (Ω). In the SIMP, C_0 is an isotropic tensor that depends on the Young's modulus of the material and the Poisson's ration (ν). $C(x)$ is the effective property of the material at the calculated point, thus the Young's modulus at each point of the domain (Ω) varies with the density $\rho(x)$. An improved SIMP approach, known as modified SIMP, establishes a minimum value for $C(x)$, thus avoiding a possible singularity in the numerical computation. The modified SIMP is given by Eq. (2) and has been used in this work.

$$C(x) = C_{\min} + \rho(x)^p (C_0 - C_{\min}), \quad 0 \leq \rho(x) \leq 1 \quad e \quad x \in \Omega \quad (2)$$

2.2 Optimization problem formulation

The topology optimization problem to be solved in the compliant mechanism design is to find an optimal distribution of material that maximizes the output displacement at a certain point, subject to material constraint. Thus, this problem can be expressed as Eq. (3).

$$\begin{aligned} & \text{maximize} \quad u_{\text{out}} = \mathbf{L}^T \mathbf{U}(\rho) \\ & \text{subject to} \quad \sum_{e=1}^{nel} \rho_e V_e \leq V^* \\ & \quad 0 \leq \rho_e \leq 1, \quad e = 1, \dots, nel \end{aligned} \quad (3)$$

Where the vector \mathbf{L}^T has value one only at the output point, in other positions this vector has zero value. The vector \mathbf{U} is the result of the finite element analysis. V_e represents the volume of material in each element and V^* represents the maximum value of material in the domain after the optimization process. Finally, ρ_e represents the density of each element and nel the total number of elements (Sigmund, 1997).

2.3 Sensitivity analysis

The derivative of the objective function is obtained using the adjoint method and can be expressed by Eq. (4) (Liu and Tovar, 2014).

$$\frac{\partial u_{\text{out}}}{\partial \rho_e} = \mathbf{u}_1^T [(\rho_e^{p-1} (C_0 - C_{\min})) \mathbf{K}_e^0] \mathbf{u}_2 \quad (4)$$

where \mathbf{K}_e^0 is the stiffness matrix of each element, \mathbf{u}_1 represents the displacement generated by a dummy load and \mathbf{u}_2 represents the displacement of the input load. For calculation of sensitivity analysis in this work, Ansys used two ways. In the first case Ansys provided stiffness matrix and displacements in each element. In this case, derivative was calculated by Eq. (4). For second case the derivative was represented by Eq. (6), where the terms \mathbf{K}_e^0 and \mathbf{u}_1^T are replaced by \mathbf{F}_1^T considering Eq. (5).

$$\mathbf{K}_e \mathbf{U}_e = \mathbf{F}_e \quad (5)$$

$$\frac{\partial u_{\text{out}}}{\partial \rho_e} = \mathbf{F}_1^T [(\rho_e^{p-1} (C_0 - C_{\min}))] \mathbf{u}_2 \quad (6)$$

2.3 Filtering

The density-based approaches, as well as the SIMP, are subject to instabilities such as mesh dependency, checkerboard pattern, and local minimum. Thus, some filtering schemes can be used to overcome this problem, such as the density filter used in this work, where the density of each element is replaced by a weighted average of the densities of the neighborhood elements, as shown at Eq. (7).

$$\rho_e = \frac{\sum_{j \in N_i} H_{ij} V_j \rho_j}{\sum_{j \in N_i} H_{ij} V_j} \quad (7)$$

where N_i represents the neighborhood of the element, V_j represents the volume. H_{ij} is the weight factor, which can be calculated from the Eq. (8).

$$H_{ij} = \max(0, R_{\min} - \text{dist}(i,j)) \quad (8)$$

where R_{\min} is the radius that determines the size of the neighborhood. The second part of Eq. (8) considers the distance between the centroid of the specific element and its neighbors (Liu and Tovar, 2014).

3. COMPUTATIONAL PROCEDURE

The implemented topology optimization routine was assembled based on the work of Liu and Tovar (2014), which is modified to allow the construction of the mesh using other elements (not only the hexahedron), thus allowing meshes for domains with sloping or curved faces. The SOLID45 element of the ANSYS library is used in this work which is hexahedral with eight nodes. The other element used was SOLID92, which is tetrahedral with ten nodes (Madenci et al., 2015).

The optimization process is carried out by the MATLAB, which provides mathematical programming tools in an easy-to-manipulate interface. However, some changes are considered to allow the exchange of information with the commercial finite element (FE) software via the ANSYS Parametric Design Language (APDL). Ansys and Matlab have commands for writing and reading vectors and matrices in files. Thus, the exchange of information and between the softwares was performed through text files in .txt format recorded and read in the same directory as the Matlab routine. The algorithm used in this work is the OC (optimality criteria) method, which is robust and relatively easy to implement, as seen in Liu and Tovar (2014).

4. RESULTS AND DISCUSSION

The tests have been performed in two cases. In the first case, the structure proposed by Liu and Tovar (2014) is maintained, where a specific discretization for 8-node hexahedral elements is defined. Thus, the ANSYS provided the stiffness matrix and the displacements of the element. In the second case, the routine is further modified to allow the use of other elements contained in the ANSYS library, allowing the initial mesh to have non-regular geometries. The visualization of an image of the element densities is also implemented in the ANSYS, as well as the visualization of the deformation of the mechanism at each iteration. The flowchart in Fig. 2 show basic steps of routine implemented.

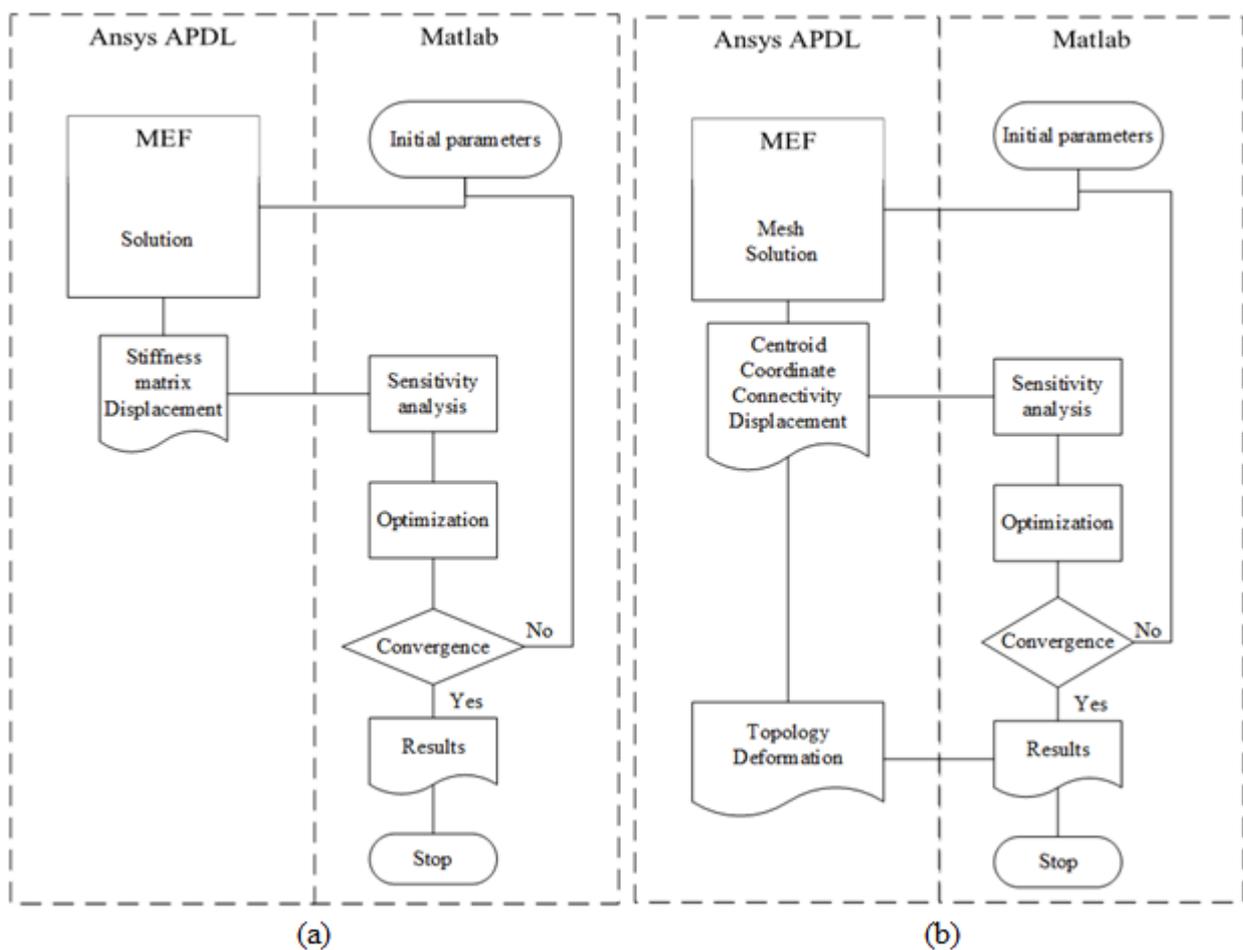


Figure 2: Topology optimization flowchart using integrate routine between Ansys and Matlab, (a) first case, (b) second case.

4.1 First case

In the Liu and Tovar (2014) program (MATLAB code), the displacements are calculated from a global matrix, assembled by a scheme that avoids the use of loops, making this step fast and efficient. The system solution is calculated using the MATLAB backslash. At this point a first change is implemented, where a mesh is created by using ANSYS with the same sequence of elements used in the Tovar program, as well as the respective boundary conditions. The element density values and other necessary parameters are also included in FE modelling created in ANSYS software. Thus, the appropriate Young's modulus of each element is also inserted before performing the displacements calculation. ANSYS also provided the stiffness matrix for use in calculation of the energy of each element. The topology final is assembled in Matlab and showing in tests performed using a force inverter mechanism as Fig. 3, 4 and 5.

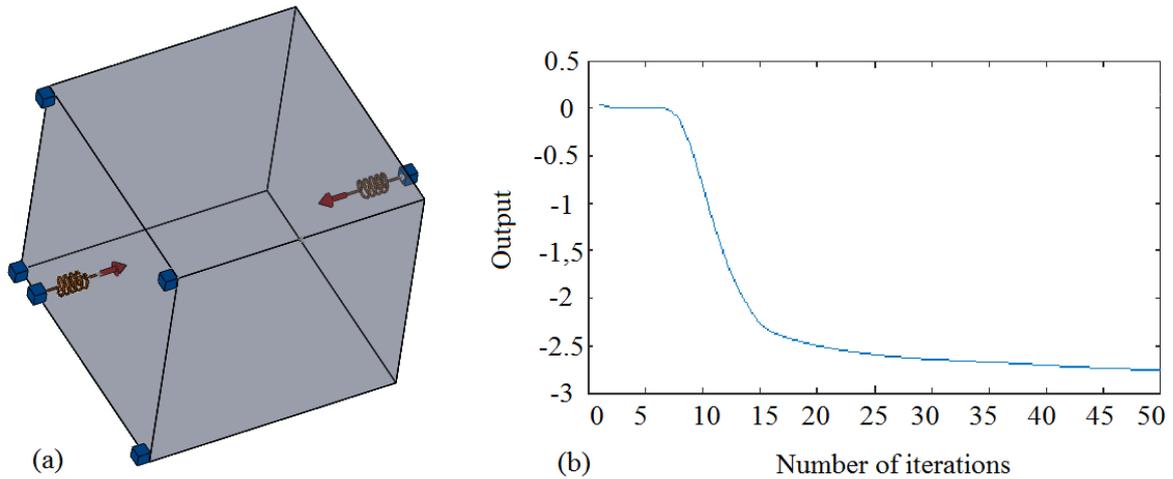


Figure 3: Inverter Mechanism with 40x40x40 dimension, (a) initial design domain discretized by 64,000 hexahedral elements, (b) convergence history using hexahedral elements.

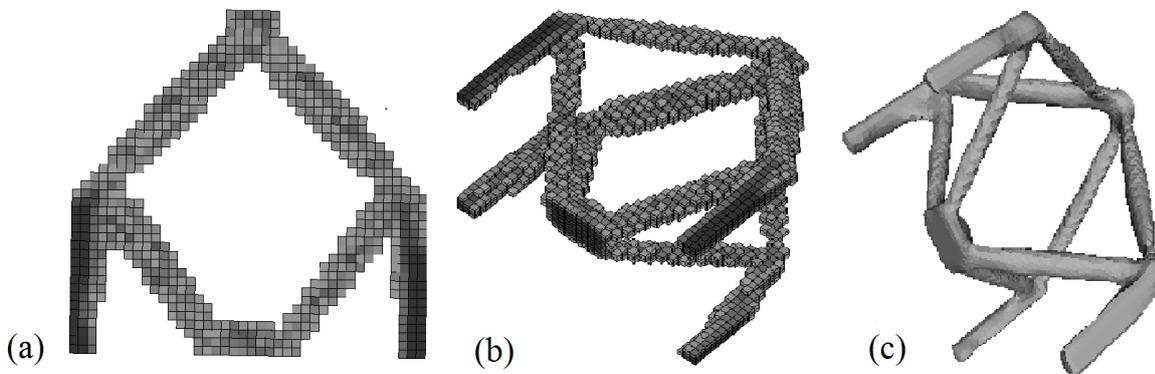


Figure 4: Inverter mechanism obtained with volume fraction of 5%, (a) front view, (b) perspective view, (c) post-processed view generated by TOPSlice (Zegard and Paulino, 2015).

It is noticed in Figure 5 that the mechanism with larger quantity of material tends to create regions with concentrated flexibility. Otherwise, a mechanism with distributed flexibility can be obtained by using smaller quantity of material (Fig. 4).

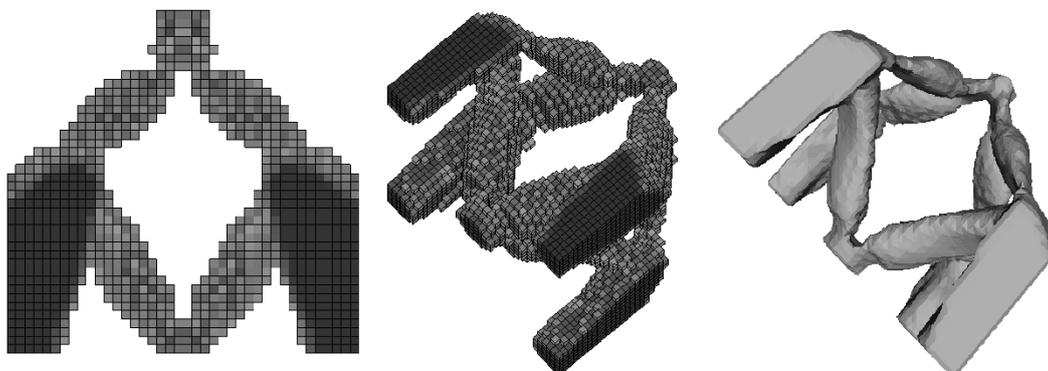


Figure 5: Inverter mechanism with 64,000 hexahedral elements and volume fraction of 15%, (a) front view, (b) perspective view, (c) post-processed by TOPSlice (Zegard and Paulino, 2015).

With tests on a personal computer with Dual Core (processor E8500) 3.16 GHz and 8GB RAM, Liu and Tovar routine cannot work with mesh size equal to or larger than 30x30x30 (twenty-seven thousand elements). Nevertheless, the new

implemented routine allowed the construction of a mesh of 50x50x40 (one hundred thousand elements). Table 1 show some dates.

Table 1. Tests with RAM memory

Mesh	RAM memory (GB)	
	Tovar	Ansys-Matlab
64,000 elements	14	4
100,000 elements	38	8
512,000 elements	-	26

4.2 Second case

As in the first case, the ANSYS is also used to calculate the displacements. However, in this second case the mesh was not assembled in the same way as the Liu and Tovar (2014) program. Here, geometry of the domain is supplied to the ANSYS, which assembles the mesh itself as optimally as possible based on its internal logic. The way to calculate the value of the energy used in the derivatives has also been changed, making it possible to use the same logic for any finite element available in ANSYS software. Once the mesh is assembled, ANSYS provides node coordinate information and connectivity between the elements for plotting the final result. The position of the centroid of each element is also provided to aid in the assembly of the density filter. In this routine is possible to use any elements, thus Figure 6 show inverter mechanism generated from tetrahedral elements.

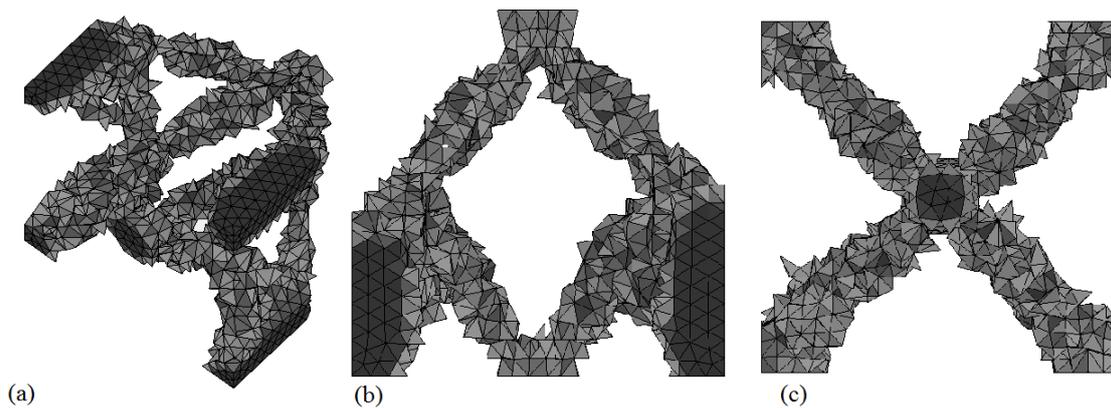


Figure 6: Inverter mecanism, using 64,440 10-node tetrahedral elements and volume fraction of 10%, (a) perspective view, (b) front view, (c) bottom view.

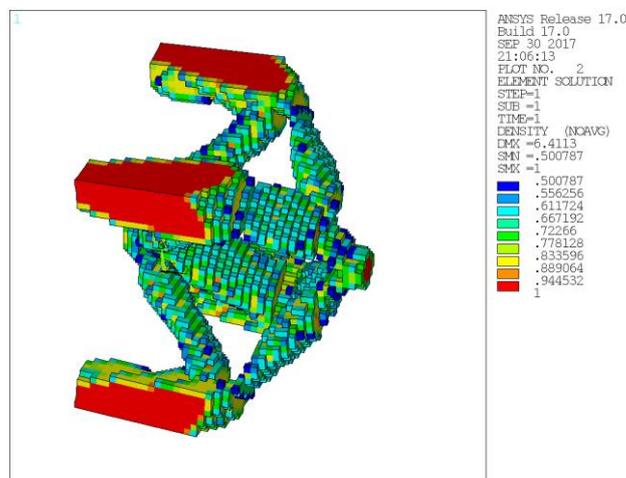


Figure 7: Inverter mecanism result with 15% of volume fraction, showing all elements with density between 0.5 and 1 (image generated by Ansys).

The Matlab prompt may be slow to building the topology of a mechanism with many elements. Thus, Ansys provides an alternative to generate the image of the mechanism at each iteration without increasing the computational cost. In this case, the image of the resulting mechanism is saved in a file and later loaded by Matlab. After generated from images generated by Ansys and loaded into Matlab. Figure 7 and 8 show results, after calculating the displacements, ANSYS can to save image file with analysis of the topology obtained, showing displacement obtained or density of each element in final topology. Thus, at the end of each iteration, MATLAB loads and displays one or more images with the partial results of the simulation.

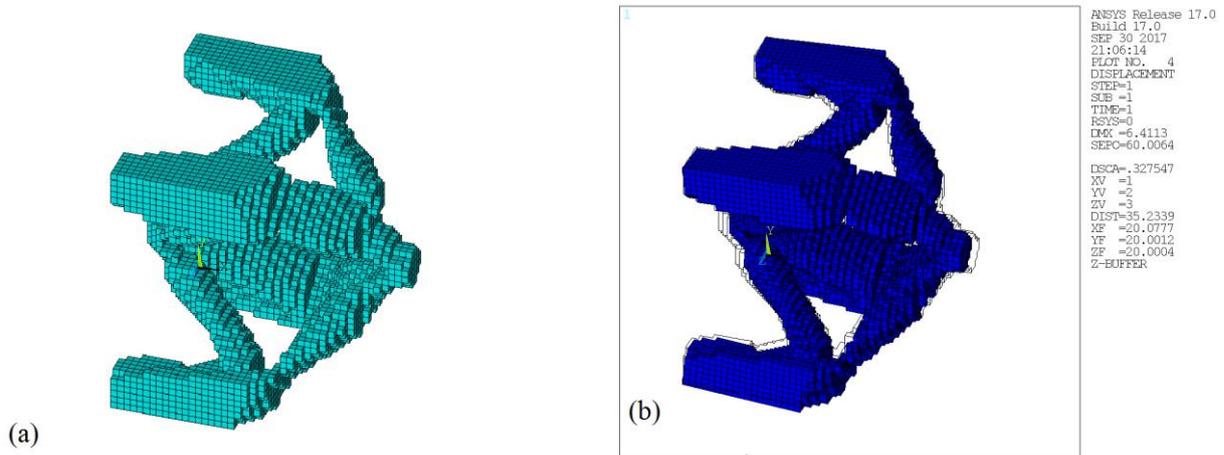


Figure 8: Inverter mechanism FE model, (a) final configuration with elements of density between 0.5 and 1 (b) deformation obtained in the final configuration.

Certainly more discretized the mesh, better the definition of the mechanism topology. Thus, Ansys contributed significantly to obtain results better to visualize than the ones using coarse meshes, since it allows to calculate the displacements using smaller amount of RAM in relation to the program that uses only Matlab.

To show the potential of the implemented algorithm, Fig. 8 presents a result obtained by using a mesh discretized with 512,000 hexahedral elements. With this discretization, the mechanism result acquires a well-defined form, even without apply post-processing.



Figure 9: Inverter mechanism with 5% of volume fraction and higher number (512,000) of elements, using 8-node hexahedral elements.

5. CONCLUSION

The implemented routine integration between ANSYS and MATLAB made the difference, mainly when the tests were performed with larger quantities of elements. The Liu and Tovar (2014) program is fast and can be manipulated with facility, however due to the assembly of the global stiffness matrix, the consumption of RAM memory is very high, limiting the its routine to few amounts of elements.

The new suggested routine uses the optimized ANSYS solvers, which stores some of the information on file and the degrees of freedom of the nodes are treated in an order that reduces the amount of information manipulated at the same time, which allows a faster solution and consumes less RAM memory. Thus, it has been possible to generate a mechanism with a higher discretized mesh of 512,000 finite elements. The implemented routine also allows a quick visualization of the result at each iteration. Thus, the verification of the displacement obtained by the mechanism is straightforward. Moreover, an extension of this research to 3D mechanism with two or more material is intended. Tests for synthesis of compliant mechanism using two or more material are in progress.

6. ACKNOWLEDGEMENTS

The authors thank the Fundação Coordenação de Aperfeiçoamento de Pessoal de Nível Superior (CAPES) and the National Council for Research and Development (CNPq).

7. REFERENCES

- B. M. Philip e S. Ole, *Topology Optimization*, Springer-Verlag, Berlin, 2004.
- Gain, Arun L. et al. *Topology optimization using polytopes*. *Computer Methods in Applied Mechanics and Engineering*, v. 293, p. 411-430, 2015.
- Howell L., *Compliant mechanisms*, John Wiley & Sons, New York; 2001.
- Howell L., Larry L., Magleby, Spencer P.; Olsen, Brian M. (ED.). *Handbook of compliant mechanisms*. John Wiley & Sons, New York; 2013.
- Kai Liu, Andrés Tovar. *An efficient 3D topology optimization code written in Matlab*, *Structural and Multidisciplinary Optimization*, 2014, DOI: 10.1007/s00158-014-1107-x
- Madenci, Erdogan, and Ibrahim Guven. *The finite element method and applications in engineering using ANSYS®*. Springer, 2015.
- O. Sigmund, *On the Design of Compliant Mechanisms Using Topology Optimization*, *Mech Struct Mach* 25(4) (1997) 493-524.
- Sigmund, Ole; Maute, Kurt. *Topology optimization approaches*. *Structural and Multidisciplinary Optimization*, v. 48, n. 6, p. 1031-1055, 2013.
- Zegard, T., Paulino, G.H. *Struct Multidisc Optim* (2016) 53: 175. <https://doi.org/10.1007/s00158-015-1274-4>

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.