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

COBEM-2017-2797
TOPOLOGICAL OPTIMIZATION OF AN ENGINE BRACKET

Marco Túlio Batista dos Anjos
Dimas Medeiros Júnior
Fiat Chrysler Automobiles, Betim, MG, Brazil
marco.anjos@fcagroup.com
dimas.medeiros@fcagroup.com

Abstract. *This study aims to present the results of a particular automotive component, called Engine Bracket, subjected to Topology Optimization to find the most suitable mass distribution of material, according to a geometry defined in the project and working conditions for this component.*
Topological Optimization is a technique that aims to find the best mass distribution within a predetermined volume of the component that is subject to a single load or multiple loads.
The Engine Suspension is formed by the powertrain assembly and the anchoring brackets thereof with vehicle body. The powertrain subsystem is the component responsible for powering the wheels of the vehicle. The engines brackets are the elements which the powertrain is installed on the body. They must comply with the assignment of resisting the possible cases of loadings generated by the engine running on them (and consequent fatigue), restrict the movement of the powertrain for not occur contact between any component of the engine compartment, filtering vibrations generated by the engine in its varied operating conditions, among other functions.
In this work, as a goal for the optimization, it was proposed to minimization of mass distributed in the pre-established volume, restricting the resulting Maximum Main Stress, given a most severe case of load in which the assembly was submitted.
The result of the Topological Optimization, showed regions within the volume predicted on the Engine Bracket in which there is a need to fill material, contributing considerably to the design of the final geometry of the component.

Keywords: *Finite Element Method, Topological Optimization, Virtual Simulation, Engine Bracket, Engine Suspension.*

1. INTRODUCTION

The engine suspension of a vehicle, according to (Medeiros Júnior, 2015), is formed by the powertrain and the anchorage supports of these with the vehicle body. The powertrain is responsible for powering the wheels of the vehicle. The components are the engine, gearbox and differential, as well as peripherals. Also according to Medeiros Júnior (2015), the brackets are elements that support the powertrain, within the engine compartment. Its function besides to resists the weight of the powertrain, is endure the cases of loads generated by the operation of the engine, restrict the movement of the powertrain to guarantee there is no contact between components in the engine compartment, filtrate the high frequency vibrations (greater than 25 Hz), and attenuation the level of energy in case of resonance. In addition, the stiffnesses of each support must result in a modal behavior to the powertrain suitable for each working condition.

Currently, for the structural dimensioning of many components in various engineering areas, especially automotive components, the Finite Element Method is used as a computational tool. In his book, (Alves Filho, 2005) defines the Finite Element Method as an approximate method of calculating continuous systems so that the component, structure, is subdivided into a finite number parts, called elements, connected together by discrete points, the nodes. These elements connected by these nodes form the Mathematical Model. The latter has its behavior defined by a finite number of parameters. In a structural analysis, these parameters are the displacements of the nodes, which describe the behavior of the system for internal forces, stresses, strains among others.

Optimization mathematical algorithms applied together with the Finite Element Method result in the Topological Optimization that, according to (ALTAIR 2017; ANSYS, 1998), is a technique that aims to find the best mass distribution within a predetermined volume of the component that is subject to a single load or multiple loads. Thus, the objective function of this type of optimization is to maximize the structural stiffness.

For the performance of its attribution of supporting the powertrain assembly in the body, the engine brackets are designed and sized in order to meet structural objectives, and the Topological Optimization is an important tool that assists the best component design and reduction of its project time and cost.

2. COMPUTATIONAL PROCEDURE

As previously mentioned, the engine brackets have many functions and among them it stands out to resist the various cases of loads imposed by the operation of the engine and its own weight. A side engine bracket is formed mainly by the parts as shown in the Fig.1 below:

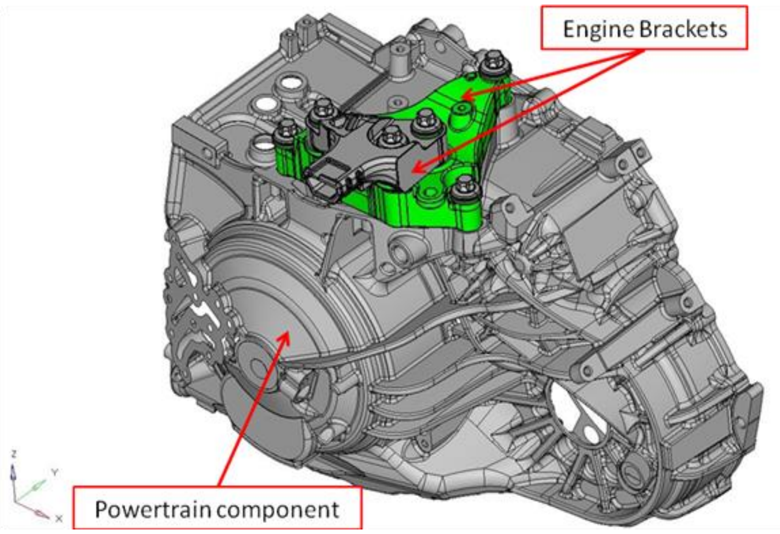

Figure 1: Left Engine Bracket.

Listed the load cases for the structural dimensioning, the mathematical model in Finite Element Method of the assembly above (figure 1) was built, using the software Hypermesh®, as Fig.2 shown:

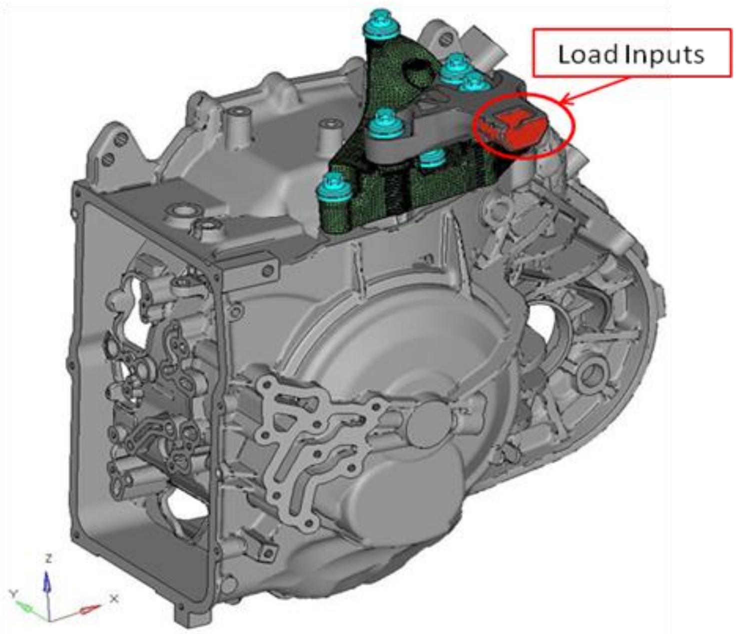

Figure 2: Engine Mount Suspension Assembly in Finite Element Method.

It should be remembered that for reasons of industrial confidentiality, it is not possible to list the load cases and their values in the present work.

The particular component submitted to the Topological Optimization procedure was the support highlighted in green in Fig.1 and Fig.2.

Therefore, in the Finite Element Model, was adopted the geometry as shown in Fig.3, which corresponds to the maximum volume that the bracket can be ported.

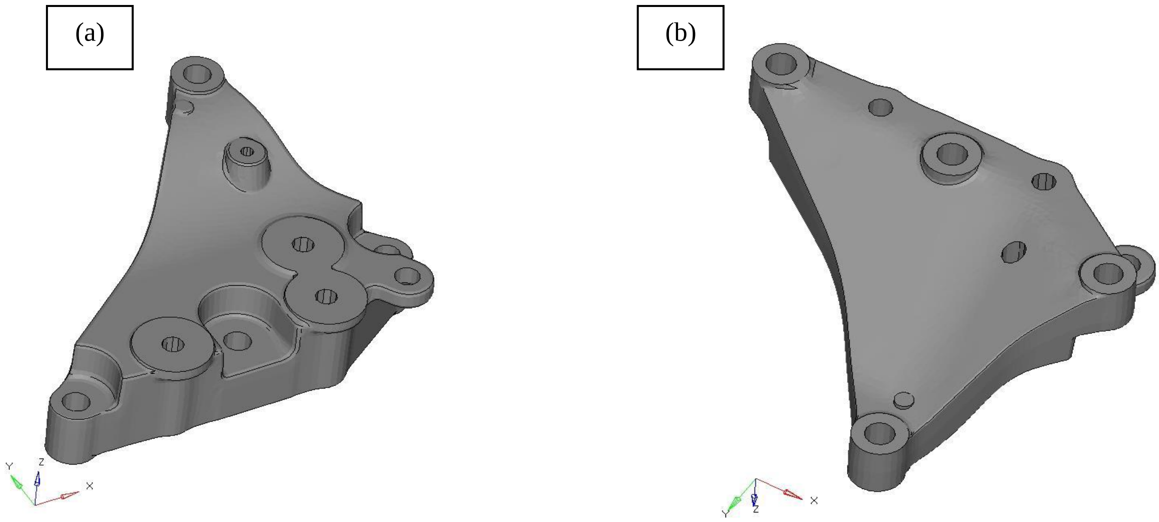

Figure 3: Initial geometry of Engine Bracket. (a) Upper view, (b) Bottom view.

With the Finite Elements Model of the assembly prepared, the Optimization Algorithm was executed, through Optistruct® software, with the objective of minimizing the mass distributed in the initial volume, and as a constraint, the limit of 128.0 MPa of Maximum Main Stress given a severe case of static loading. This value de Maximum Main Stress is a function of the yield stress and fatigue resistance of the Bracket material.

3. RESULTS AND DISCUSSION

The optimization results are shown in the Fig.4 below:

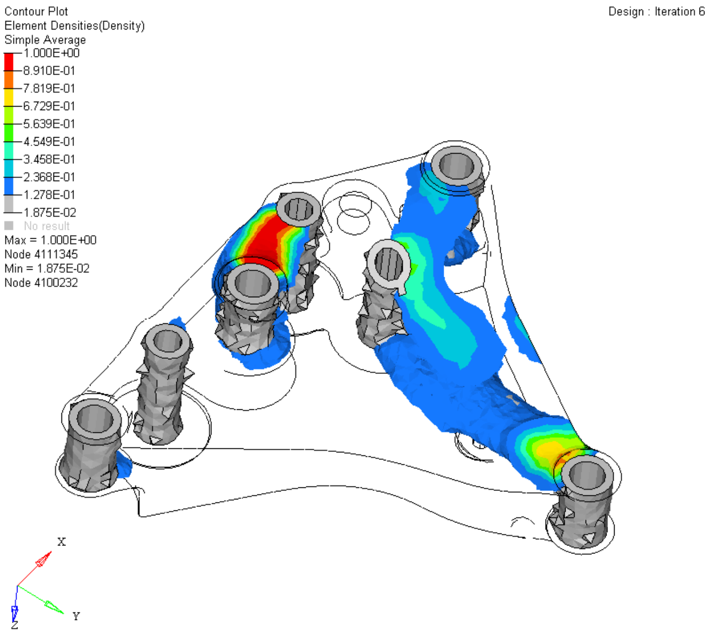

Figure 4: Topological Optimization main result.

In Figure 4 above, the highlighted regions in colors are those where the presence of material is essential for maintaining tensions below the limit of 128.0 MPa. It is highlighted the region in red in which the highest density of elements was found.

Given the result shown above, the final layout of the Bracket was proposed, as shown in the fig.5 below:

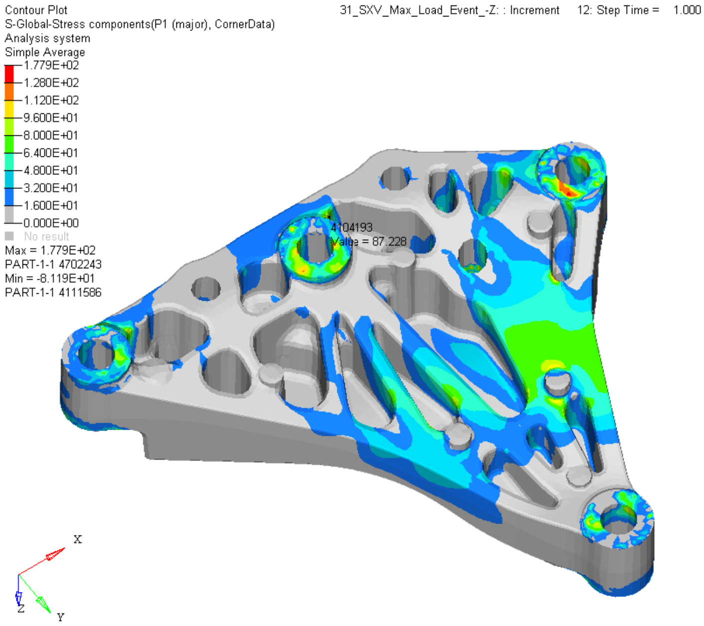

Figure 5: Maximum Main Stress result in new Bracket.

Figure 5 presents the results of Maximum Main Stress, 87.2 MPa, for the same load case applied in optimization procedure, but for the component with the new geometry proposed based on the result of the Topological Optimization.

4. CONCLUSIONS

As the main gains obtained in the topological optimization procedure in which the Engine Bracket was submitted, stands out the considerable economy of material used in the construction of the component, in addition, the decrease in the time of project development.

5. ACKNOWLEDGMENTS

This work was developed with the support of FCA - Fiat Chrysler Automobiles.

6. REFERENCES

ALTAIR UNIVERSITY, 2017. What is Topology Optimization about?. September 01th, 2017. Avalaible in: <https://altairuniversity.com/optimization/topology/>.

Alves Filho, A., 2013. *Elementos Finitos - A Base da Tecnologia CAE*. Editora Érica Ltda., São Paulo, 6nd edition.

ANSYS GUIDES, 1998. What is Topological Optimization?. September 01th, 2017. Avalaible in: <http://www.ansys.stuba.sk/html/guide_55/g-adv/GADV2.htm>.

Medeiros Junior, D., 2015. *Modelamento Matemático Computacional da Suspensão Motopropulsora Veicular para Instalação de um Motor Três Cilindros*. Master Degree Final Dissertation, CENTRO FEDERAL DE EDUCAÇÃO TECNOLÓGICA DE MINAS GERAIS, Brazil.

7. RESPONSIBILITY NOTICE

The authors, Marco Túlio Batista dos Anjos and Dimas Medeiros Junior, are the only responsible for the printed material included in this paper.