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

COBEM-2017-1986
DIMENSIONING OF THE FRONTAL SUSPENSION SYSTEM OF THE DOUBLE-A TYPE THROUGH TOPOLOGICAL ANALYSIS.

Diógenes França
Tiago Ferreira
Thiago Moreira
Gustavo Melchiades
Lucas Ferreira
Instituto Federal de Minas Gerais – Campus Congonhas, Av. Michel Pereira de Souza, 3007 - Campinho, Congonhas - MG, 36415-000
diogenes.sfs@gmail.com
tiago.simao@ifmg.edu.br
thiago.moreira@ifmg.edu.br
gustavommelchiades@gmail.com
lucasadriel@outlook.com

Abstract. *The suspension system of a motor vehicle is always complex to size. In order to do so, it is necessary to take into account several factors such as efficiency, safety and comfort, for example. With the goal of designing a double-A type frontal suspension system that is safe and effective in overcoming obstacles, the initial step is to study which phenomena directly interfere in the suspension. Among the various phenomena, the forces generated by free fall, lateral force during a curve and braking are the ones that most require the whole system. Of the possibilities to improve the damping of these forces, the possibility discussed in the present article is to size the forks of the suspension using the topological method in order to reach the best geometry for the suspension, associated to the lower weight. This work aims to define all the loads and then pass analytical analysis to numerical, where a system of double suspension type A will be simulated by finite elements, with the intention to measure the tension and the resulting deformation of these requests. With the result of the simulation, it is possible to optimize by means of a topological analysis the geometry of the suspension forks, creating an innovative sizing method that guarantees a damping that does not compromise the pilot's health neither the useful life of the vehicle structure.*

Keywords: *Suspension, Finite Elements Methods, Topological Optimization Method*

1. INTRODUCTION

The present study is divided into two parts. The first part consists in the determination of the loads acting on the suspension system of the vehicle at different times: longitudinal, lateral and vertical. Even among these loads, the vertical impact force being the one with the highest value, because the car is subject to all loads at different times, the system geometry will be generated to resist to the three, in an extreme situation where both the braking force, the lateral and impact forces would act at the same time.

The second part is divided in two moments: the first one is the application of the loads acquired in the first part to a generic model of suspension arm that is simulated computationally by the finite element method (FEM), for the acquisition of the voltage of von Mises and the equivalent deflection of the three combined loads. The second moment is the use of the topological optimization method (TOM) to generate a new geometry for the suspension system.

The purpose of TOM is to reduce the mass of the system, which is reduced until a maximum deflection that does not compromise the safety of the vehicle is achieved. The new geometry is simulated by FEM and the results found by the computer simulation ensure that the deflection constraint is not exceeded and that the resulting tension is below the flow limit. Thus, the efficacy of the study is proven, which may be used in future studies.

2. DETERMINATION OF LOADS

With the adoption of the local coordinate system indicated in Fig. 1, the car's movements are divided into longitudinal (Y), lateral (X) and vertical (Z). For each of these coordinates, the main working load will be characterized, so that when the system is optimized, this is done taking into account loads in all directions.

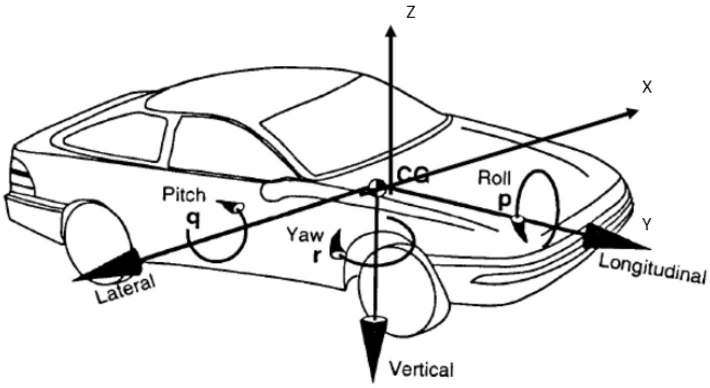

Figure 1: Local coordinates adopted to carry out the study.

2.1 Longitudinal Force

The longitudinal dynamics, according to the definition of Leal et al. (2008), is the vehicular behavior study in the direction (Y). The study of longitudinal dynamics assesses the performance of the vehicle as it accelerates, its ability to overcome ramps and its performance during braking. For the present study, the three aspects of interest during braking are time, distance and maximum deceleration. Tab. 1 gives some relevant information about the vehicle studied that will support the acquisition of the maximum braking force that can be actuated in the car without causing it to overturn.

Table 1. Geometric variables of the vehicle that influence the braking dynamics.

Total mass (kg)	267,98
Front axle mass (kg)	112,59
Rear axle mass (kg)	155,39
Maximum speed (m/s)	13,064
Minimum stopping distance (m)	6
Height of the center of gravity (m)	0,350

The maximum deceleration value (a) is obtained by means of dynamic analysis, through the energy sum, shown by Equation 1.

$$a = \frac{-v^2}{2d_{stationary}} \qquad (1)$$

In order to determine the braking force it is necessary to multiply the total mass (m) of the vehicle by deceleration (a), as shown in Equation 2.

$$F_{braking} = m_{total} * a \qquad (2)$$

2.2 Vertical Force

In the same way as the characterization of the braking force was performed, the force resulting from a free fall will be calculated through a dynamic analysis, using sum of energy, as indicated by Leal et al. (2008). The vertical force resulting from the free fall is shown in Equation 3, where "m" is the total mass, "h" is the height of the free fall, "k" is the spring constant and "Ffall" is the fall force which can still be decomposed into (X) and (Z), due to the inclination of the damper.

$$F_{fall} = \left(\sqrt{\frac{m_{total}.g.h_{fall}}{k}}\right) * (0{,}250 * k) \quad (3)$$

2.3 Lateral Force

In order to conclude the study of the loads acting on the suspension system, it is necessary to know some design variables that are available in Tab. 2, so that we can use Equation 4 and calculate the centripetal force value.

Table 2. Input data for lateral dynamics.

Curvature radius (m)	2,5
Scroll center height (m)	0,220
Angle between CC and CG (º)	16,26
Maximum curve speed (m/s)	5,53

$$F_{centripetal} = \left(\frac{v^2}{R}\right) * m_{total} \quad (4)$$

It is worth emphasizing that the force that is wanted to characterize is not the centripetal force, that acts in the center of mass, but the force that acts in the suspension, that is the lateral force. Fig. 2 represents the vector sum of lateral force components on the front axle, lateral force on the rear axle and centripetal force, when a vehicle is at constant speed, as Canale claims (1989).

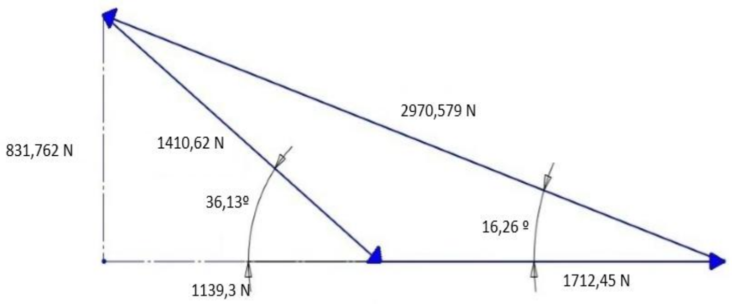

Figure 2: Distribution of the lateral force between the front and rear axles.

3. COMPUTATIONAL ANALYSIS

3.1 Simulation of the Generic Model

In order for the simulation to occur correctly, it is necessary to follow some steps such as: definition of the conditions of static contour and distribution of the support reactions; choice of the type, geometry and size of the mesh for the creation of the discrete model; implementation of the material; inputs of the charges that will act on the generic component, so that it responds to von Mises voltage and deflection. The generic model that will be simulated can be seen in Fig. 3.

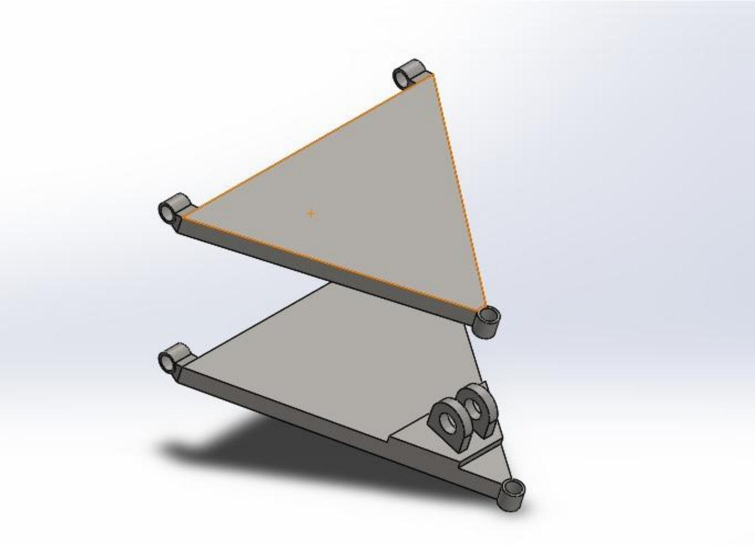

Figure 3: Generic model of the suspension system.

The software used to perform the simulation understands the directions as number, where 1 is translation in (X) and 4 rotation in (X), 2 translation in (Y) and 5 rotation in (Y), 3 translation in (Z) and 6 rotation in (Z). Tab. 3 shows how each of the 22 constraints were scattered.

Table 3. Restriction distribution.

Localization	Restrictions
Top Left Node	12346
Top Right Node	1346
Lower Left Node	12346
Lower Right Node	1346
Cage Damper Node	1346

The finite element simulation process is a numerical process, according to Alves Filho (2008). It is therefore essential to find a way to reduce the divergence of results. For this, a good mesh quality is recommended, according to the author. In this way, the quality of the mesh is defined according to the type of mesh, the geometry of the elements, the size and the aspect of the mesh. Tab. 4 shows the result of the discretization.

Table 4. Mesh Properties.

Minimum size of the element (m)	0,002
Maximum size of the element (m)	0.01
Mesh aspect	3,31
Type of mesh	Tetra
Number of elements	44061

Regarding the material, as can be seen in Tab. 5, the properties are typical of a structural steel. The objective of this simulation is to achieve a voltage value lower than the flow limit, with a safety factor of at least 1,5. If this does not happen, the generic template will need to be redone.

Table 5. Material properties.

Elastic modulus (GPa)	210
Density (kg/m³)	7900
Poisson	0,3
Flow Limit (MPa)	210

Figure 4 shows how the model will look with all its supporting loads and constraints, and Table 6 shows the value of the input loads.

Table 6. Input loads for the simulation value.

Force	Component X	Component Y	Component Z
Braking (N)	0	-3812	1262
Impact (N)	1745	0	2873
Lateral (N)	-705	0	979
Resulting (N)	1040	-3812	5112

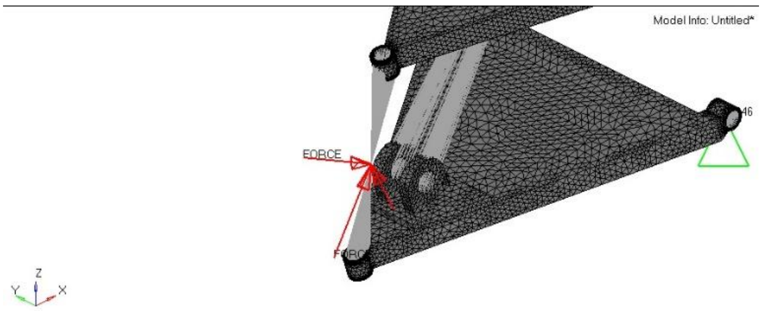

Figure 4: Distribution of loads in the discretized body.

The results can be seen in Fig. 5 and the values in Tab. 7. It should be noted that the maximum voltage was well below the flow limit, even with the adoption of the safety factor, which guarantees the possibility of optimization.

.

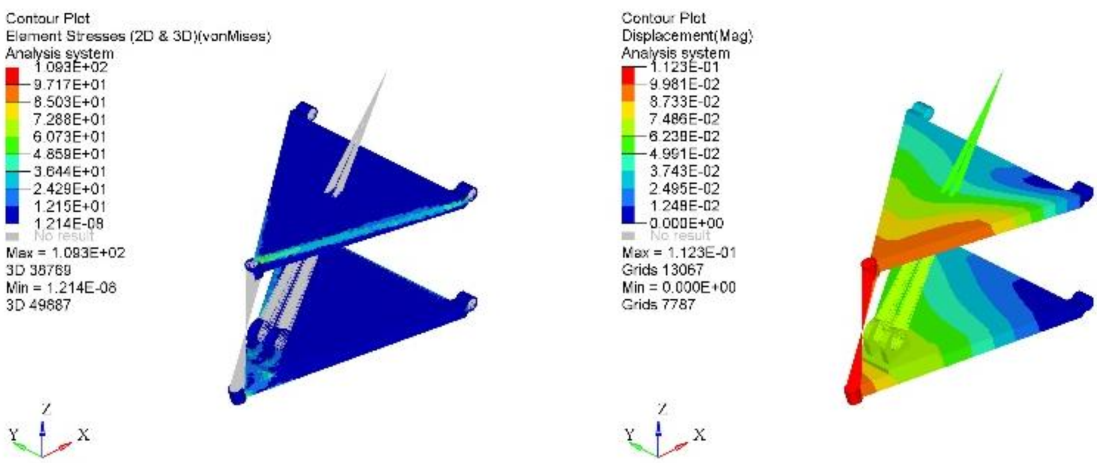

Figure 5: Result of the static simulation. On the left, tension result and on the right, deflection result.

Table 7. Static simulation result.

Maximum Tension (MPa)	Deflection (mm)
109,3	0,1123

3.2 Topological Optimization Method

The topological optimization method (TOM) is a computational method that allows to design the optimal topology of a structure from certain parameters. In short, the TOM distributes material within the volume of a geometry in order to maximize or minimize a given function, which for this study will be mass reduction. The material at each point in the volume can vary from empty (there is no presence of material, represented by the number 0) until full (total presence of material, represented by the number 1) and can assume intermediate densities between 0 and 1 according to the defined model, according to Neto (2015).

Figure 6 shows the division between the two models within the generic model. The part of the model in yellow will be the non-optimizable part, while the part in light blue will be the part that will undergo optimization.

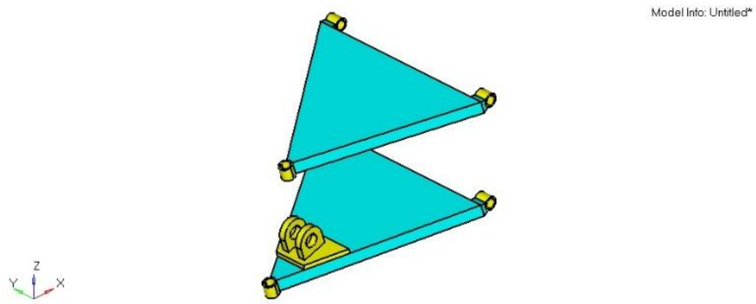

Figure 6: Separation between optimizable model in blue and model not optimizable in yellow.

It was decided to leave in yellow only the points of support of the suspension system with the cage and the axle sleeve, in addition to the support of the shock absorber. The optimization of the optimizable part of the geometry will be done by means of variables. These parameters are divided into two blocks: design variables and state variables. You can define a project variable as any function that can be modified until an optimal value is obtained, as affirmed by Neto (2015). While state variable can be understood as the representation of the expected responses during optimization.

The mass was defined as the objective function, which means that the value of the mass will be dependent on the other project variables and will serve to demonstrate how good the problem is. In order for the optimized model to have at most the same deflection as the generic model, it has been defined that the constraint function will be the deflection function, with a value greater than 0,1 millimeter.

A state variable is a variable that will guide the optimization process. For the current study, the desirable maximum stress in the geometry was stipulated as being 1,5 of the flow limit, that is why the value 140MPa. Another defined state variable was symmetry. The triangle-shaped geometry is expected because it is a double-A type suspension system.

Figure 7 presents the result of the last interaction of the optimization, presenting the optimized geometry, containing visible only elements with density above 0.67.

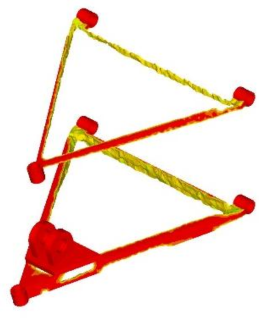

Figure 7: Result of the simulation using the Topological Optimization Method.

4. RESULTS

Although this geometry is already optimized, it is a fully solid geometry. For the manufacture of the suspension system, aiming, in addition to safety, also efficiency, the most suitable is to use a tubular geometry. The purpose of post-processing the simulation is to find a geometry that exhibits the same behavior as the optimized geometry but has the tubular profile, rather than the solid profile. For this, the value of the resistance module of each of the bars was collected, as shown in Tab. 8 and 9.

Table 8. Top arm bar resistance module optimized suspension system.

Arm	Basis (mm)	Height (mm)	Resistance mod. (mm^3)	Ext. Side (mm)	Int. Side (mm)	Thickness (mm)
Right	6,21	15,49	248,52	25,4	24,80	0,59
Left	5,03	17,27	250,27	25,4	24,79	0,60
Transversal	2,34	14,44	81,43	25,4	25,20	0,19

Table 9. Resistance module of lower arm bars optmized suspension system.

Arm	Basis (mm)	Height (mm)	Resistance mod. (mm)	Ext. Side (mm)	Int. Side (mm)	Thickness (mm)
Right	11,69	18,54	669,90	25,4	23,67	1,72
Left	13,32	16,29	589,82	25,4	23,90	1,49
Transversal	1,55	25,36	167,07	25,4	25,00	0,39

As a commercial matter, it was decided to build the suspension system using steel tubes of square profile and thickness of 2mm. The conversion shown in Tab. 8 and 9 already presents the values of the external and external size and from there we arrived at the final geometry, seen in Fig. 8.

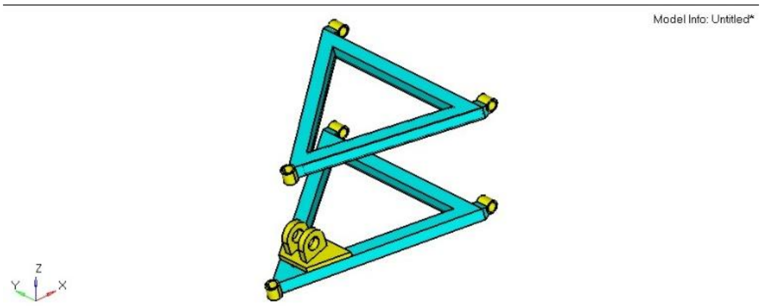

Figure 8: Optimized Geometry.

The result of the simulation can be seen in Fig. 9 and the maximum tension values of von Mises and deflection are shown in detail on Tab. 10.

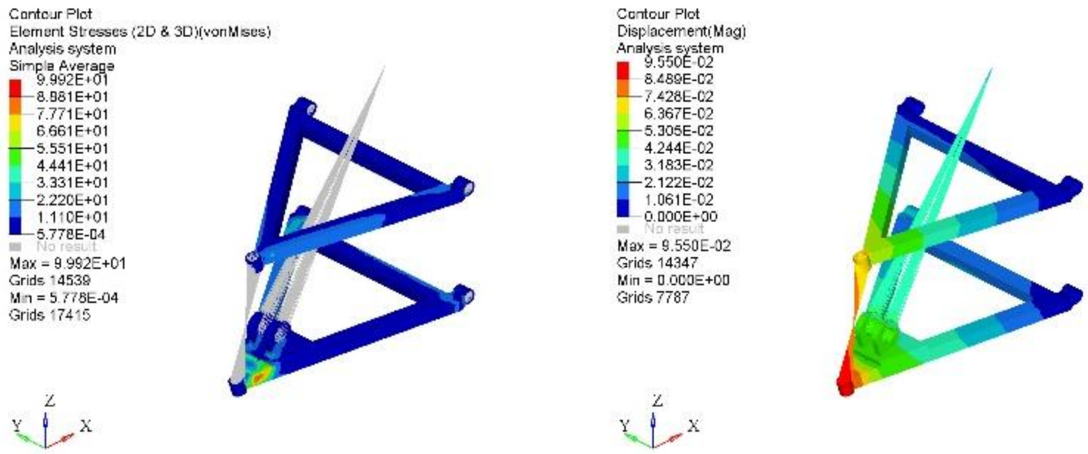

Figure 9: Result of the tension and deflection for the optimized geometry.

Table 10. Maximum tension value of the von Mises and the deflection in the optimized geometry.

Maximum Tension (MPa)	Deflection (mm)
99,2	0,955

5. CONCLUSION

When comparing the results of tension and deflection of the two models, it is possible to affirm that both did not have substantial alteration, and, more importantly: it is perceived that even without great alteration, there was decrease, instead of elevation. The new geometry is therefore as apt to be used as the generic geometry in terms of voltage and deflection.

The possibilities of using the method presented in this work go far beyond the design of the suspension system. After all, whenever it is possible to determine all boundary conditions, loads and material of any mechanical component, it will be possible to topologically optimize any desired mechanical component. The use of such tool presents the possibility of mass reduction, cost reduction, efficiency increase, among others.

6. ACKNOWLEDGMENTS

The authors would like to thanks the IFMG-Congonhas, the Bajeiros Equip and Altair Brasil.

7. REFERENCES

Alves Filho, A., 2008. “Elementos Finitos - A Base da Tecnologia Cae - Análise Não Linear”. 1ªEd. São Paulo: Érica.
Neto, F.R.F., 2015. "Projeto de um braço inferior de suspensão para um veículo off-road utilizando otimização topológica".
Leal, L.C.M., Nicolazzi, L.C. e Rosa, E, 2008. “Uma introdução à modelagem quase estática de veículos automotores de rodas”. Publicação interna do GRANTE – Departamento de Engenharia Mecânica da UFSC.SC, Brasil.
Canale, A.C., 1989. “Automobilística Dinâmica e Desempenho”. 10ªEd. São Paulo: Érica.

8. RESPONSIBILITY NOTICE

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