25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

COB-2019-1142
FINITE ELEMENT ANALYSIS AND TOPOLOGY OPTIMIZATION OF THE BRAKE CALIPER AND THEIR SUPPORTS

Wilber Silva Leonel Júnior
Tobias Anderson Guimarães
Federal University of Triângulo Mineiro, Department of Mechanical Engineering, Av. Randolfo Borges Júnior, 1250, Univerdecidade, Unidade I, Uberaba, MG
wilberjunior@live.com
tobias.guimaraes@uftm.edu.br

Abstract. *The major role of a brake system is to mitigate the kinetic energy from vehicle by the dissipation energy. Hence, the brake systems components, such as, the disk, caliper and their supports should be designed to support the stress and strain produced by the forces during the braking. In this sense, the objective of this work is to analyze the structural integrity of brake system components by using the Finite Element Method (FEM). Initially, the maximum torque applied to the brake system was obtained by using the dynamical forces for the critical situation. Subsequently, a finite elements model of the brake disk, caliper and supports was created and simulated using the boundary conditions and the loads produced by the braking process. In order to define the optimal material distribution of the caliper, the Topological Optimization Technique was also applied to the finite elements model from brake system. The results proved that the FEM is a useful tool to be used in the analysis of the structural integrity from brake system. Furthermore, the topology optimization technique was successfully applied to the geometrical design of the caliper from brake system.*

Keywords: *brake system, caliper, finite element analysis, topology optimization*

1. INTRODUCTION

The major role of the brake systems is to control the velocity of a vehicle. Thus, the brake system components geometry should be designed to support the stress and strain that are produced during the braking. Because the brake system is a critical component of a vehicle, the structural integrity of material of the brake caliper, disk and shoes should be preserved. Furthermore, another important design criteria is the reduction of the brake system weight. Indeed, if the weight of the brake system is very large, the acceleration and the fuel consumption of the vehicle can be inefficient and the performance of the vehicle will be reduced.

In this sense, Budynas and Nisbett (2016) introduce the basic theory and the equations to be applied in the design of the brake system shoes based on the torque and forces that act during the braking. Limpert (2011) also demonstrates the equations to be employed in the brake system design by considering the dynamical loads. Blumberg and Foschini (2010) have applied the finite element method in the study of the stress and strain of a brake caliper model in order to analyze the structural integrity of this system. They used the software ANSYS® in the modeling of brake system for the generation of the meshing, application of loads, contact conditions and boundary conditions. Subsequently, the stress distribution of brake system components, such as, the disk, shoes and caliper was simulated in order to analysis the structural integrity. However, it was not found in the literature any article by studying the reduction of the weight or the optimization of the caliper material distribution for the increasing the performance of the vehicle.

In the context, the objective of this work is to design the caliper material distribution of a brake system using the topology optimization technique. After the optimization, the structural integrity of shoes, brake disk and caliper will also be studied by using the stress distribution of the brake system finite element model. In the following, it will be described a summary of the methodology used for the generation of the results and the new geometry of caliper that was purposed.

2. BRAKE SYSTEM

Figure 1 illustrates a brake system of a vehicle with the brake disk, shoes, caliper and the shaft pivot. During the braking, the brake caliper apply a compression forces on the brake disk. Hence, the shoes that are bonded on the caliper surfaces transmit this effort for the brake disk and the kinetic energy of the vehicle is transformed in thermal energy. During the braking process, a stress distribution is generated in all components due the forces and contact that acting among the subsystems. If the stress level is very high, a fault could be occur in the brake system and the safety of the driver will not be preserved. Therefore, as the material, as the geometry of each component should be carefully specified for the brake system.

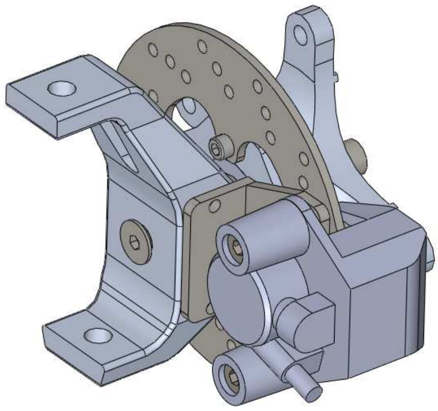

Figure 1. Geometry of the brake system components.

3. ANALYSIS AND TOPOLOGY OPTIMIZATION OF THE BRAKE CALIPER

3.1 Estimation of the force and torque of the brake system

The first step to create the brake system model was to define the geometry of each component: caliper, brake disk, shoes and shaft pivot. Hence, the initial geometry of each component was created based on the intuition and experience of the designer. Subsequently, it was estimated the magnitudes of pressure and force that act during the vehicle braking. The force applied to the master cylinder of the brake system was measured by using the specific instrumentation. The magnitude of this measured force is equals to 800 N.

For the calculation of these parameters, it was used the formulation described by the Limpert (2011). The pressure, p_l, applied by the cylinder to the braking system is defined by the equation (1):

$$p_l = F_p l_p \eta_p / A_c \tag{1}$$

which Fp is the force applied by the driver during the braking, lp is a parameter caused by the moment arm, η is the efficiency due the moment arm (usually equals to the 0.8 or 80%) and the Ac the area from the cross section of the cylinder.

According Limpert (2011), the braking force is evaluated multiplying the pressure by the area of the cross section from the caliper piston and other parameters defined by the equation (2):

$$F_b = 2(p_l - p_o) A_w \eta_p BF(\frac{r}{R}) \tag{2}$$

where po is the pushout pressure of the cylinder, Aw is the area of the cross section from the caliper piston, BF is the braking factor, usually equals to the twice the friction coefficient from shoes, r is the radius of the brake disk and R is the radius of the pneu from the vehicle.

After the calculation of the braking force, the braking torque, T, is obtained by multiplying the braking force by the disk radius:

$$T = F_b r \tag{3}$$

The torque calculated by the equation (3) should be larger than the torque provided by the vehicle engine. For the calculation of this torque, it is necessary to compute the parameter ψ defined by the relation between the load applied on the rear wheel, Fr and the weight of the vehicle, W:

$$\psi = F_r / W \tag{4}$$

In a similar way, the ratio between the height of the gravity center, h, and the distance, L, between the wheels from the vehicle is given by the parameter χ:

$$\chi = h/L \tag{5}$$

The Figure 2 illustrates a vehicle with the dimensions used in the calculation of equations (4) and (5). By using the equations (4) and (5), the breaking dynamical forces applied on the front and rear wheels, F_f and F_r, are respectively:

$$F_f = (1 - \psi + \chi a)w\mu \tag{6}$$

$$F_r = (\psi - \chi a)W\mu \tag{7}$$

where the variable μ represent the friction coefficient pneu/rail and the parameter a is the acceleration from the vehicle during the breaking. The torque provided by the engine is also evaluated by multiplying these forces by the radius of the braking disk.

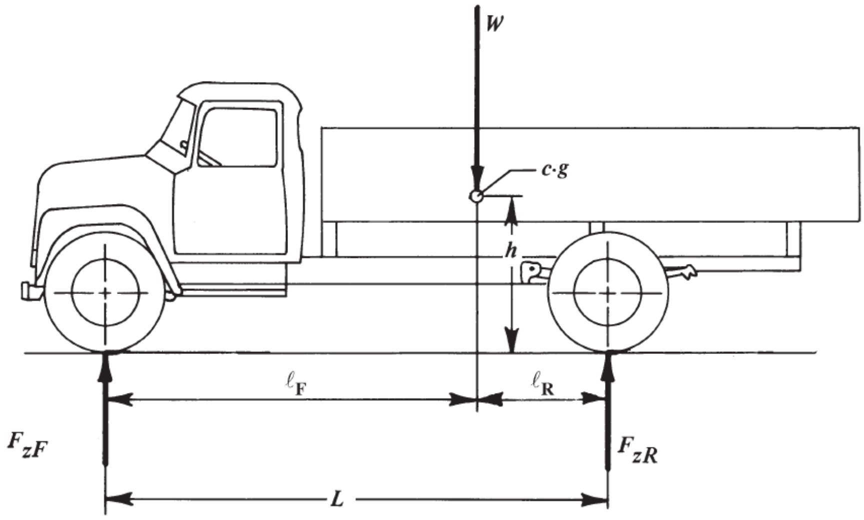

Figure 2. Free body diagram of the vehicle (Limpert, 2011).

The table 1 summarizes the parameter values that were employed in the estimation of the braking torque. By using the parameter magnitude defined in the table 1 and the results obtained by the equations described in the above formulation, it was calculated the torque required by the braking system and the torque to be transmitted by the engine to the front wheels. The braking torque was equals to 150 N.m and the torque to be transmitted by the engine is 125 N.m.

Table 1. Parameter values of the vehicle to be used in the calculation of the brake torque.

Parameter	Magnitude
Weight of the vehicle, W [N]	2557,47
Friction coefficient pneu/rail, μ[]	0,67
Height of the gravity center, h [m]	0,55
Acceleration of the vehicle during the braking, a [g]	-0,91
Parameter due to moment arm, l_p [m]	4,50
Efficiency due to moment arm, η[]	0,80
Breaking factor, BF []	0,70
Pushing pressure, po [Pa]	0,00
Area of the cross section, Aw [cm^2]	7,31
Radius of the braking disk, r [mm]	63,50
Radius of the front pneu, R [mm]	177,80
Radius of the rear pneu, R [mm]	279,40
Distance between wheels, L [m]	1,40

3.2 Model of the finite element of the brake system

After the calculation of the braking torque, the geometrical model of the brake system has created in the software SolidWorks® was exported to the software ANSYS® of the finite elements. Figure 3 shows the mesh of each component from the braking system. During the braking process, the caliper moves in the x direction since it applies a compression force on the brake disk which executes a rotation movement with respect to the axis x. The braking torque of 125 N.m was applied in one of the holes from the braking system disk. The contact between the caliper and the shoe was defined as bonded. In a similar way, the contact between the pins and the disk was also defined as bonded. On the other hand, the contact between the brake disk and the shoes was frictional. For the creation of the finite element model, the material from shoes was defined as ABNT steel 1020. The material of the others components from the brake system was defined as an aluminum alloy 7075-T6.

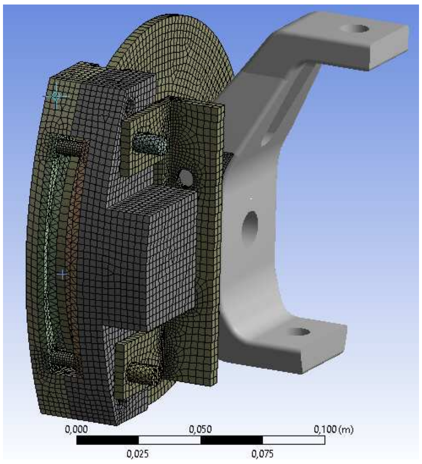

Figure 3. Mesh of the finite element model for the brake system.

3.3 Topology optimization of the brake caliper

The topology optimization technique was applied in this work for reducing the caliper mass and to maximize the stiffness. Hence, it is expected that the displacements from the caliper caused by the torque be reduced by the maximization of the stiffness. Another advantage of this procedure is the increasing of the vehicle performance because of the reduction of mass which could improve the fuel consumption. For the generation of the optimized topology of the caliper, it was simulated the full model of the brake system by using the sub-modelling technique available in the software ANSYS®.

In this work, it was employed the topology optimization tool available in ANSYS® in order to optimize the material distribution or layout of caliper by using a similar procedure as described by Sigmund (2000). The general optimization problem for this situation is:

Minimize: $$C(x) = \sum U^T KU \tag{8}$$

Subject to: $$\frac{V(x)}{V} = f \tag{9}$$

$$KU = F \tag{10}$$

$$0 < x_{min} \leq x \leq 1 \tag{11}$$

where $C(x)$ represents compliance or strain energy of the structure, the parameter K denotes the global stiffness matrix of the finite element model, U is the displacement vector, F the load vector applied to the structure. The design variable, x, of this optimization problem is the relative density from finite elements of the design space.

The initial design space of the optimization problem was the caliper geometry illustrated in the Fig. 1. During the optimization, the volume fraction of the optimum topology was defined as 75%, that is, the optimal topology volume fraction, f, will be equal to 0,75 after the optimization process. The equation (10) represents the equilibrium relation of the finite elements model from caliper and the equation (11) represents a side constraint applied to the relative density of each element of the structure. The parameter x_{min}, usually equals to 0.01, is the minimal value that may be assumed by the design variables. The importance of this constraint is to avoid a singularity in the global stiffness matrix K, during the calculation of the displacement vector, U.

For the updating of the design variables, it was used the Optimality Criteria Method applied to the Lagrangian of the topology optimization problem. In this case, it is necessary to define a material model for the relative density from design space. In the literature, the material models commonly used are the homogenization method and the power law (Bendsoe and Sigmund, 2008). By using power the law, the relationship between the original and modified element stiffness matrix is given by:

$$k = x^p k_o \tag{12}$$

where k_o is the original element stiffness, k represents the modified element stiffness matrix and p is the penalization power usually equals to 3. By using this material model and applying the Optimality Criteria Method to the Lagrangian of the optimization problem, it can be demonstrated that the strain energy density of the structure is constant for the whole design space (Bendsoe and Kikuchi, 2008):

$$B = \sum \frac{p x^{p-1} u^T k_o u}{\lambda \dfrac{\partial V}{\partial x}} \tag{13}$$

where B is the strain energy density, u is nodal displacement vector of the finite elements model and λ is the Lagrangian multiplier has calculated from the active volume constraint (9). Once again, since the design space of the caliper of the braking system is subjected to several loads, it necessary to sum the sensitivities caused by each force acting on the space design. By using this procedure, the topology from caliper is updated until the convergence of the optimization process.

4. ANALYSIS OF THE RESULTS

Figure 4 shows the Von Mises stress distribution of the brake system after the finite element simulation. As can be seen in this figure, the maximum value of the Von Mises stress is near from the pin used for the support of the caliper. The magnitude of the Von Mises stress in this region is 237,83 MPa which is less than the yield stress (about 400 MPa) for the aluminum alloy 7075-T6. Since the hole of the pin represents a discontinuity, it was expected that maximum Von Mises stress could be concentrated in this region. Although this stress level is considerably high, this proves that this caliper design could be manufactured and applied in the practice since that the components of the brake system would be subjected only the elastic strain.

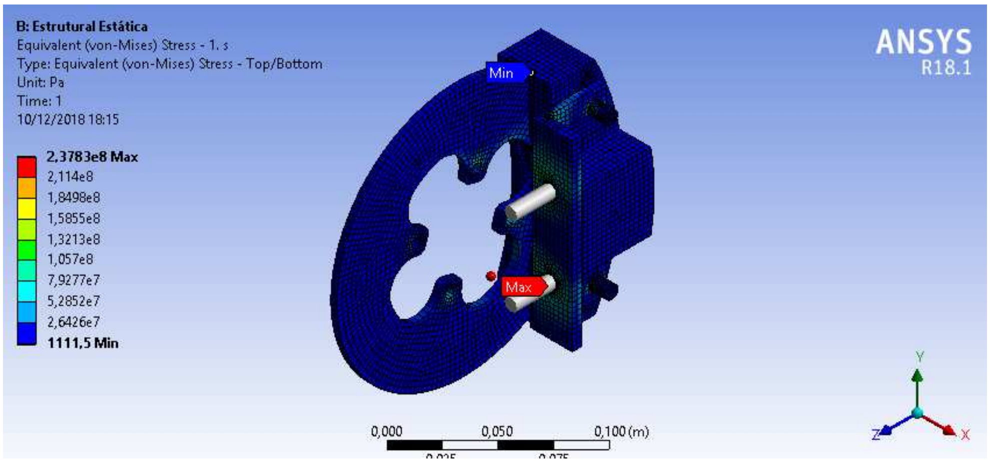

Figure 4. Von Mises stress distribution of the finite element model for the brake system.

The Von Mises stress field for rear part of the caliper is illustrated in the Fig. 5. For this situation, the maximum Von Mises stress is equals to 47,17 MPa which is also less than yield stress from caliper material (about 400 MPa). The factor of safety for the braking system has obtained by the finite elements analysis varies in the range of 1,69 to the 4,63. Since the minimum value of the factor of safety is larger than 1 (one), all subsystems of the brake systems are dimensioned. In this way, it is proved that the structural integrity of brake system has proposed is preserved. Hence, the brake system may operate with torque of 125 N.m and the boundary conditions used in this analysis.

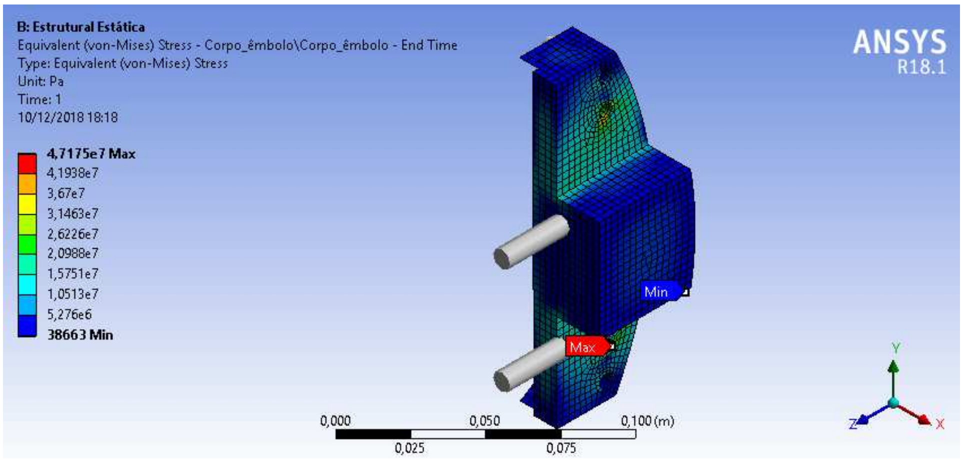

Figure 5. Von Mises stress distribution of the finite element model for the caliper.

The Table 2 describes the results for the mesh convergence test of the caliper finite elements model. In this convergence study, it was used the sub-modelling available in the software ANSYS®. Since the caliper geometry is complex with several discontinuities, it was necessary to check the influence of the caliper meshing about the results. Hence, if the finite element size used in the discretization from caliper model is too large, regions with "artificial" stress concentration may be present in the Von Mises distribution and the results may be wrong. After 7 iterations, the maximum Von Mises stress converged (about 260 MPa) and the meshing does not have influence about the results, according to the Table 2. In this way, it was used the parameter values described in this table as reference for the results.

Table 2. Results of the mesh convergence of the caliper finite elements sub-model.

Parameter	P1 - Face Sizing Element Size [m]	P2 - Body Sizing Element Size [m]	P3 - Edge Sizing Element Size [m]	P4 - Total Deformation Minimum [m]	P5 - Total Deformation Maximum [m]	P6 - Equivalent Stress Maximum [Pa]	P7 - Equivalent Stress Minimum [Pa]	P8 - Safety Factor Minimum []
1	1,0E-03	1,0E-03	1,0E-03	3,09E-05	1,274E-04	1,535E+08	106,593	3,010
2	5,0E-04	5,0E-04	5,0E-04	3,09E-05	1,273E-04	1,910E+08	34,721	2,419
3	4,0E-04	1,0E-03	4,0E-04	3,09E-05	1,273E-04	2,087E+08	106,784	2,214
4	4,0E-04	1,0E-03	3,0E-04	3,09E-05	1,273E-04	2,088E+08	105,115	2,213
5	3,5E-04	1,0E-03	4,0E-04	3,09E-05	1,273E-04	2,146E+08	107,035	2,153
6	3,0E-04	1,0E-03	4,0E-04	3,09E-05	1,273E-04	2,239E+08	106,836	2,063
7	3,0E-04	8,0E-04	3,0E-04	3,09E-05	1,273E-04	2,265E+08	78,896	2,040
8	2,0E-04	8,0E-04	3,0E-04	3,09E-05	1,273E-04	2,595E+08	79,084	1,780
9	2,0E-04	6,0E-04	3,0E-04	3,09E-05	1,273E-04	2,513E+08	45,235	1,839
10	2,0E-04	4,0E-04	3,0E-04	3,09E-05	1,273E-04	2,543E+08	23,123	1,817
11	1,5E-04	4,0E-04	3,0E-04	3,09E-05	1,273E-04	2,750E+08	23,090	1,680

Figure 6 shows the optimal topology from brake caliper has obtained by the software ANSYS. Since the volume fraction selected for the design space of the optimal topology is high (75%), the optimization procedure removed a small amount of material near from the corners of initial design. Hence, the optimized caliper will be able to absorb the elastic strain energy due to the braking pressure and its structural integrity will be preserved.

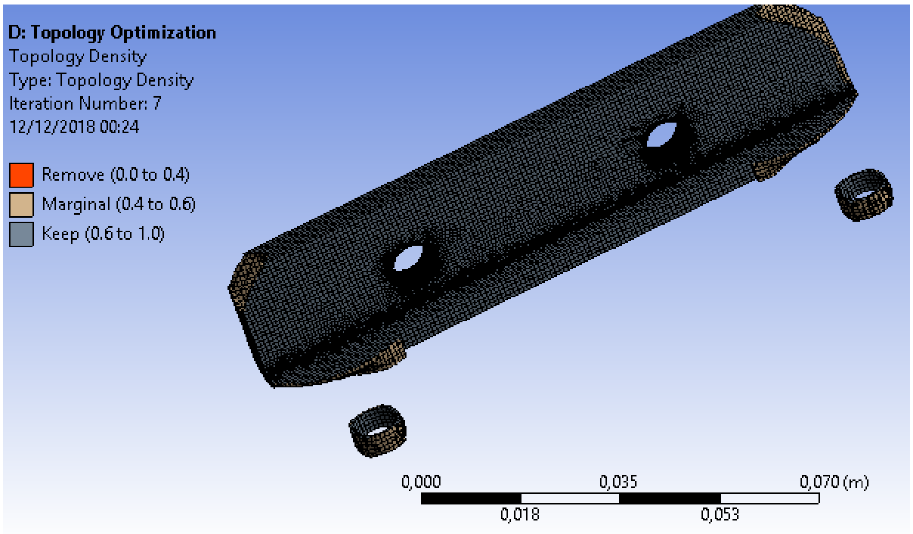

Figure 6. Optimal topology of the brake caliper.

5. CONCLUSIONS

In this work, it was purposed a methodology for the finite elements analysis of a brake system and the design of the brake caliper material distribution using the topology optimization technique. The first step of the modelling was the estimation of the pressure to be applied on the brake disk and the selection of the initial design space of the brake subsystems. The braking torque was calculated using the equations of the dynamical forces and compared with the torque to be transmitted by the powertrain to the front wheels. Subsequently, it was created the finite elements model of the brake system subjected to the braking pressure, contacts and boundary conditions by using the software ANSYS®. Furthermore, it was also applied the topological optimization technique for the reduction of the weight of the braking caliper.

By using the results of this analysis, it was possible to check that the maximum Von Mises stress magnitude is less than the yield stress of the materials used in the manufacture of the brake system. Another interesting result was the caliper geometry produced by the topology optimization process. The results prove that the purposed methodology can aid the designer to improve the initial geometry of the brake system. Since that caliper weight was reduced and its stiffness was maximized, it is expected that the elastic strain caused by the pressure be minimized. This same methodology may also be applied to others brake systems models by modifying the pressure, contacts, boundary conditions and initial geometry of the subsystems.

6. ACKNOWLEDGEMENTS

The authors would like to acknowledge the FAPEMIG (Fundação de Amparo à Pesquisa do Estado de Minas Gerais) and the Department of Mechanical Engineering from UFTM for provide the software ANSYS® of the finite elements necessary for this work.

7. REFERENCES

Blumberg, R. and Foschini, N. N., 2010. "Design of high performance automotive brake caliper using ANSYS". Mai. 2018 https://docplayer.net/67933701-Design-of-high-performance-automotive-brake-caliper-using-ansys-rafael-blumberg-author-nicolino-foschini-neto-co-author.html.

Budynas, R. and Nisbett, J. K., 2016. *Elementos de Máquinas de Shigley: Projeto em Engenharia Mecânica.* AMGH, Porto Alegre, Brazil, 10th edition.

Limpert, R., 2011. *Brake Design and Safety*. SAE International, Warrendale, 3rd edition.

Sigmund, O., 2001. "A 99 line topology optimization code written in Matlab". *Structural Optimization*. Vol. 21, pp. 120 – 121.

8. RESPONSIBILITY NOTICE

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