CONEM 2016
CONGRESSO NACIONAL DE ENGENHARIA MECÂNICA
21 - 25 AGOSTO DE 2016
FORTALEZA - CEARÁ

BRAKE CALIPER STRUCTURAL OPTIMIZATION OF A FORMULA SAE VEHICLE

Lucas Travi Farias, farias.formula@gmail.com[1]
Bruno Ziegler Haselein, brunozh.formula@gmail.com[1]
Carlos Eduardo de Souza, carlos.souza@ufsm.br[1]

[1]Universidade Federal de Santa Maria, Av. Roraima, 1000 – Camobi, Santa Maria – RS, 97105-900

Abstract: *In competition vehicles, brake calipers must meet a number of design conditions. They should be rigid for a minor pedal stroke and to obtain a quick response, light to collaborate with the overall performance of the vehicle, capable of withstanding high temperatures and dissipate the heat. In order to improve the brake system of the previous years prototype, a Formula Student team developed a study of structural optimization in the previously used brake caliper body in order to get a new lighter and more rigid structure. To conduct such a study, the understanding of boundary conditions has great importance. Thus, it was necessary an experimental analysis to determinate the maximum pressure in the brake system with pressure gauges and the axial displacement in the brake calipers through the use of Strain Gauges. This axial displacement data were compared with the finite element analysis to get an initial simulation model. This model, with loads and boundary conditions was transferred to the optimization software. After optimization, a new geometry was created with mass reduction from 210 grams of the initial model to 166 grams in the final model. The reduction in the maximum axial displacement was also observed in the finite element simulation, where in the initial model the maximum displacement was 0,46mm and went up to 0,31mm in the final model. These results represent an approximately 21% of reduction in weight and 32.6% of reduction in the maximum axial displacement, i.e., the structural optimization method can be extremely useful if properly known the boundary conditions, objectives and constraints of the analysis.*

Keywords: *Formula Student, Structural Optimization, Topology, Brake Caliper, Brake system.*

1. INTRODUCTION

Over the years, the development of new products is becoming an increasingly complex task due to tight budgets, contraction of deadlines, in addition to the rate of diffusion of new technologies constantly increasing. At the same time, new tools such as the Topology Optimization are designed to assist the design process by reducing development time and overall costs, allowing improvements in product performance.

These tools have reached the academic level through university competitions such as Formula SAE/ Student. This event evaluates the performance of an automobile single seater prototype in a series of static and dynamic tests, awarding projects that best meet the requirements of the rules developed by SAE. In these events students seek to compete with increasingly lighter cars and a greater number of parts with its own developed projects, in order not to create dependency on purchased market items, minimize costs and gain more points in the design events. However, to develop parts that require a high degree of reliability as the brake calipers, which are critical components for the safety of a vehicle, an analysis on levels of displacement and stress in the critical loading conditions is mandatory.

Finite element methods and topology optimization have been used by Sergent, N., Tirovic M., Voveris J. (2014) to design an opposed piston brake caliper in which the aim were to optimize the caliper for minimum mass, for a given stiffness. These methods are also used by Farias, L., et al. (2015) to develop a design optimization of a monoblock brake caliper for a Formula Student team, whose main goal was better heat dissipation as well as mass reduction.

In this work, the analysis will be performed experimentally by comparing the new optimized caliper with a reference model. Among the objectives of the new product are the mass reduction and the maintence of the stiffness of the baseline model. To say that such a small part can become even lighter and, at the same time, more resistant, only to a rearrangement in the distribution of material is enough to whet the curiosity by this study.

1.1. The brake system of the formula SAE prototype

The prototype Silver of the Formula SAE team Formula UFSM can be seen in Fig. 1a. This vehicle has a braking system with double acting hydraulic circuit and parallel configuration (front/rear), i.e., a circuit acting on the brakes of the front axle and the other controlling the brakes of the rear axle. Each of these hydraulic circuits contains a simple master cylinder (15.875 mm in diameter) capable of pressurizing the DOT 4 fluid in the system. The fluid contacts each of the two pistons (25.4 mm diameter) of the fixed brake calipers, which are present on all four wheels of the car. Once pressurized, the pistons transmit this pressure to the brake pads moving until they reach the brake disc and thus, causing the vehicle to slow down by the friction generated in the contact. However, it is important to highlight that the prototype has no Anti-lock Breaking System (ABS).

The details of the brake system of the prototype are shown in Fig. 1b. The brake lines in blue represent the flexible lines on the vehicle and the brake lines in orange represent the rigid lines. Between the lines there are some connections in golden color. Positioned on the right panel, there is a blue dial with red central adhesive. This button is responsible for setting the brake bias front/rear. Just behind the pedal there are reservoirs that store the brake fluid and are connected to the master cylinder. In the wheels are the discs and brake callipers, the last connected by screws to the prototype`s upright.

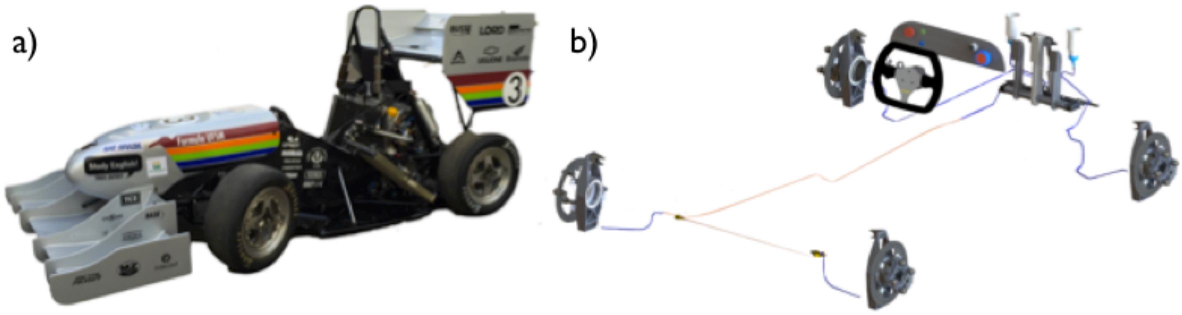

Figure 1. a) Silver Prototype - Formula UFSM team. b) Brake system of the Silver prototype.

In this pedalbox configuration, the master cylinders are tilted at an angle to the vertical. For this type of pedal, the pedal ratio (geometric parameter) varies with the angle of application, but this variation is negligible because the angle variation is not wide (pedal ratio varies from 4,2:1 to 4,3:1), in other words, the car has a small pedal stroke.

2. LOADS DETERMINATION

2.1. Maximum pressure in the brake system

According to Limpert (1999, p. 180), the hydraulic pressure in the brake lines p_l produced by the pedal force F_p is:

$$p_l = \frac{F_p l_p \eta_p}{A_{mc}} \tag{1}$$

Where,
A_{mc} = Master cylinder cross-sectional area, mm^2
F_p = Pedal force, N
l_p = Pedal lever ratio
η_p = Pedal lever efficiency

Knowing that the cross-sectional area of the master cylinder is $197{,}93\ mm^2$, adopting the pedal ratio equal to $4{,}3{:}\,1$ and the pedal ratio efficiency equal to 1, the maximum pressure in the hydraulic lines of the prototype can be obtained. For calculation purposes, it is considered two possibilites:

In the first possibility, according to Eq (1), considering the brake pedal force of 800N - according to Limpert (1999), the maximum brake pedal force for male drivers is 823N - and the balance of 55-45, lead to a pressure in the front line of $97{,}47 kgf/cm^2$. The maximum hydraulic pressure on the rear line is approximately $79{,}75 kgf/cm^2$.

In the second situation, according to Eq. (1), starting with 1250N of brake pedal force – accordign to Puhn (1985), the maximum brake pedal force for a big man can reach 1335N - and balance 55-45, lead to a maximum pressure on the front line of $152{,}30\ kgf/cm^2$ while the maximum hydraulic pressure in the rear line is approximately of $124{,}61\ kgf/cm^2$.

For the third situation, according to Eq. (1) and more one time considering a pedal force of 800N, but now a balance of 70-30, leads a maximum pressure to the front line of $124{,}06 kgf/cm^2$. The maximum hydraulic pressure on the rear line is approximately $53{,}17 kgf/cm^2$.

In order to validate these calculations, a test was made with the installation of pressure gauges on the front lines (maximum range of $300\ kgf/cm^2$) and rear lines (maximum range of $140\ kgf/cm^2$) of the car. This same test dealt with the maximum displacement in the brake caliper, however, for organizational purposes, this second part of the test will be treated in section 2.3.

After calibration, the pressure gauges were installed in the hydraulic system of the prototype. The two gauges were positioned next to one another to record in just one camera and avoid time synchronization problems.

The test was performed on an asphalt track plane, ambient temperature of 26 degrees Celsius and dry conditions. The test consisted of two braking situations. At first, the driver exercised the force normally used to brake the car. In the second brake situation, the driver was instructed to apply an extremely greater force than usual, the goal was to obtain a maximum displacement in the brake calipers. The results of the maximum pressure found in the first and second braking can be seen in Fig. 2a and Fig. 2b:

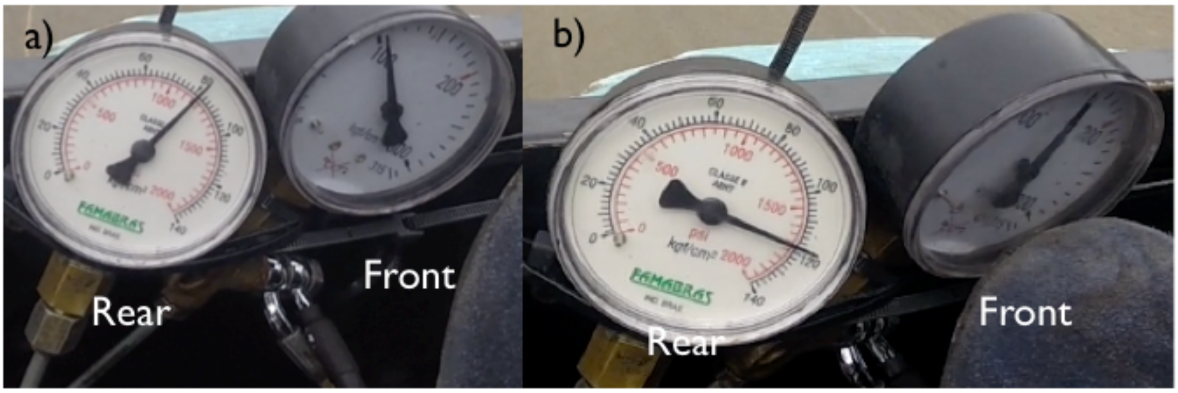

Figure 2. a) Moment of maximum pressure in the first braking, with pedal force normally applied in a hard stop. b) Moment of maximum pressure in the second braking, with pedal force above the usual applied by the driver on a hard stop (maximum pedal force).

The braking balance was setted in 55-45. For the first braking, the maximum pressure found in the rear circuit was approximately $80 kgf/cm^2$ while the pressure found in the front circuit was about $100 kgf/cm^2$, confirming the calculations performed for the first possibility, where the results found were $97{,}47 kgf/cm^2$for the front line and $79{,}75 kgf/cm^2$ for the rear line.

For the second braking situation, the maximum pressure found in the rear lines was approximately $120\ kgf/cm^2$ while the pressure found in the front line was approximately $150\ kgf/cm^2$, confirming the calculations performed in the second possibility where the maximum pressure found in the rear line was $124{,}61\ kgf/cm^2$ and the maximum pressure in the front line was $152{,}30\ kgf/cm^2$.

The pressure gauge representing the pressure of the front brake circuit (the right in Fig. 2) was partially tilted when the driver applied full force on the brake pedal. However, it can be still noticed the marking of $150\ kgf/cm^2$, where the line indication is longer than the other lines. Due the fact that the gauge is inclined, it is possible to have a parallax error of the angle of the displayed image. For avoidance of doubt as reading, confirmation of the gauge information in the front circuit can be determined by the value found in the gauge of the rear circuit. Since the braking balance was set as 55-45, a simple proportion rule can be used. If $120\ kgf/cm^2$ represent 45%, 55% is equivalent to $147\ kgf/cm^2$, value close to $150\ kgf/cm^2$.

The third possibility is calculated considering the braking balance 70-30, never used in competition, by leaving the brake excessively frontal, is just to show that with a force normally applied to the brake pedal, pressure on the front line not exceed the $124{,}06 kgf/cm^2$, or the pressure is less than the $150\ kgf/cm^2$ found in the test and the calculation of the second possibility.

Once checked the maximum pressure in the lines, the next step is to understand the loads applied to the brake caliper. As the brake caliper will be the same for the front and rear axles of the prototype, it will be considered the greater pressure for the new brake caliper design, in this case of $150\ kgf/cm^2$.

2.2. Forces in the brake caliper

According to Farias, L., et al. (2015), there are two types of loads on the brake caliper, axial (Fig. 3a), which is a response to the compression forces between the brake pads and disk and tangential (Fig. 3a and Fig. 3b), which is a loading due to the tendency of the discs to "drag" the brake pads, rubbing these pads against the abutment surfaces of the brake calipers body.

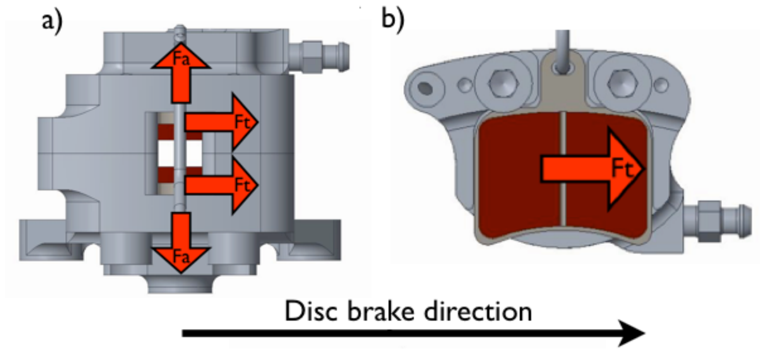

Figure 3. a) Axial and tangential forces in the brake caliper. b) Tangential forces in the brake caliper.

The axial load can be calculated by:

$$F_a = p_l A \tag{2}$$

Where,
A = Cross section of each cylinder, mm^2
F_a = Axial force, N
p_l = Line pressure, N/mm^2
The cylinder cross section is a circle, easily calculated by its radius by the formula $A = \pi r^2$. This area corresponds to the caliper body surface that is in contact with the piston. The top view of Fig. 3a assists in understanding it. When the system is activated, the caliper body that is in contact wiht the piston receives hydraulic pressure and moves "opening up" the caliper.

Using the Eq. (2) and knowing that the area is $506{,}71mm^2$, and the maximum pressure found in tests is approximately $150kgf/cm^2$, it was possible to obtain $F_a = 760kgf$ or approximately $7455N$.

And the tangential load can be calculated by:

$$F_t = p_l A \mu_p = F_a \mu_p \tag{3}$$

Where,
A = Cross section of each cylinder, mm^2
F_t = Tangential force, N
p_l = Line pressure, N/mm^2
μ_p = Pad/disc average coefficient of friction (obtained from the manufacturer), N/N
Using the Eq. (3), and given that the average coefficient of friction between the pad and the brake disc provided by the manufacturer is $0{,}41N/N$, was possible to obtain $F_t = 312kgf$ or about $3060N$.

2.3. Maximum axial displacement test

The test performed to measure the maximum pressure in item 2.1 is the same in that one of the calipers of the front brake prototype was instrumented and will be treated in this section.

During the braking process, the temperature of the brake calipers will increase, therefore, the strain gage configuration with two active and two passive gages was chosen. two full bridges are installed, one in the upper end of the caliper which will be called the upper bridge and other in the lower end of the caliper, which will be referred to as lower bridge (Fig. 4a).

For calibration, the axial force due to the hydraulic pressure and the tangential force, which is generated by braking torque, were applied. The axial force generated by hydraulic pressure was applied in stages through a ratchet strap fixed on the brake pedal. The ratchet strap stages were secured by pressing the brake pedal and generating pressure in the lines. The pressure was controlled through the installed pressure gauges by visual mode.

To simulate the braking torque, a 1 meter lever was manufactured with the addition of weights controlled at its end as the pressure applied to the brake lines (Fig. 4b). A 45° angle has been left to the vertical so that the cathetus opposite to the angle represented 0,71m to the mass point of application. So to know the torque that should be applied, it had to be mounted a pre-calibration table with the desired pressure and required torques (Tab. 1).

According to Puhn (1985, p.99), the torque on the brake disc can be calculated by the Eq. (4):

$$T_{Disco} = p_l . \mu_p . n . A . r_{efetivo} \tag{4}$$

Where,
A = Cross section of each cylinder, mm^2
μ_p = Pad/disc average coefficient of friction (obtained from the manufacturer), N/N
p_l = Line pressure, N/mm^2
$r_{efetivo}$ = Effective radius of the disc (central point of contact: pads/disc), mm
n = Number of cylinders

The effective radius of the used brake disc is $97{,}25mm$, the pressure on the front lines ranges from 10 to $50kgf/cm^2$, the average friction coefficient is $0{,}41N/N$ (provided by the manufacturer), the number of cylinders per caliper is equals to 2, and the contact area of each cylinder is $506{,}71mm^2$.

With these values, the torques for each pressure can be calculated and the table for calibration mass settings can be mounted. As the lever is $0{,}71m$, the mass needed to reach the computed torque must also be calculated.

Table 1. Table for defining the calibration mass for a given pressure.

Front brake line pressure $[kgf/cm^2]$	Front brake line pressure $[N/mm^2]$	Front disc torque $[kN/mm]$	Mass to 0,71m leverage $[kg]$	Mass applied in calibration $[kg]$
10	0,98	39,63	5,69	5
20	1,96	79,25	11,38	10
30	2,94	118,88	17,07	15
40	3,92	158,51	22,76	20
50	4,90	198,13	28,46	25

Dial gauges were positioned in specific points on the brake caliper to obtain the displacement with increasing pressure in the lines and torque on the axle (Fig. 4b and Fig. 4c).

Points on the outer surface of the caliper were adopted (surface facing the outside of the car) because it is the region where the axial displacements are higher. The opposed surface or inner surface (surface facing the central plane of the car) is close to the bolted conections with the cars vehicle upright, meaning that the axial displacements in this region are present in minor values.

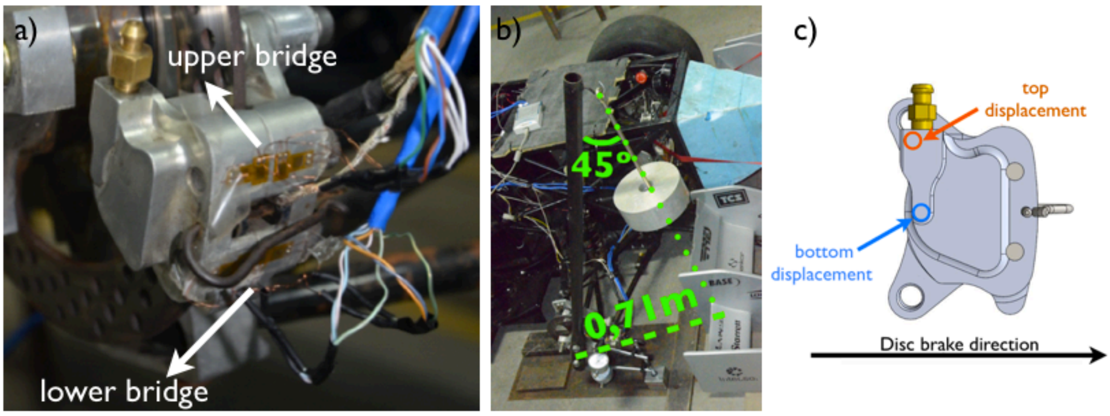

Figure 4. a) Strain gages installed in the right front brake caliper of the prototype. b) Installation of the brake calipers calibration experiment. c) Dial gauges positioned to measure the displacement in the brake caliper.

Based on Tab. 1, the calibration was executed and the values of pressure, displacement and strain were recorded in the Tab. 2:

Table 2. Calibration data.

Front brake line pressure $[kgf/cm^2]$	Torque $[kgf.m]$	Top displacement $[mm]$	Top strain $[\mu m/m]$	Bottom displacement $[mm]$	Bottom strain $[\mu m/m]$
0	0	0	8,30E-06	0	2,16E-06
10	4,04	0,015	2,16E-06	0,020	-8,56E-05
20	8,08	0,035	-1,57E-04	0,040	-1,53E-04
30	12,12	0,050	-1,93E-04	0,050	-1,90E-04
40	16,16	0,075	-2,75E-04	0,075	-2,80E-04
50	20,20	0,105	-3,39E-04	0,090	-3,62E-04

From Tab. 2, charts were created with the points of displacement vs strain of the upper and lower strain gages and a trend line was added with a linear function and its characteristic equation that describes the behavior of the points found in the experiment. (Fig. 5a and Fig. 5b).

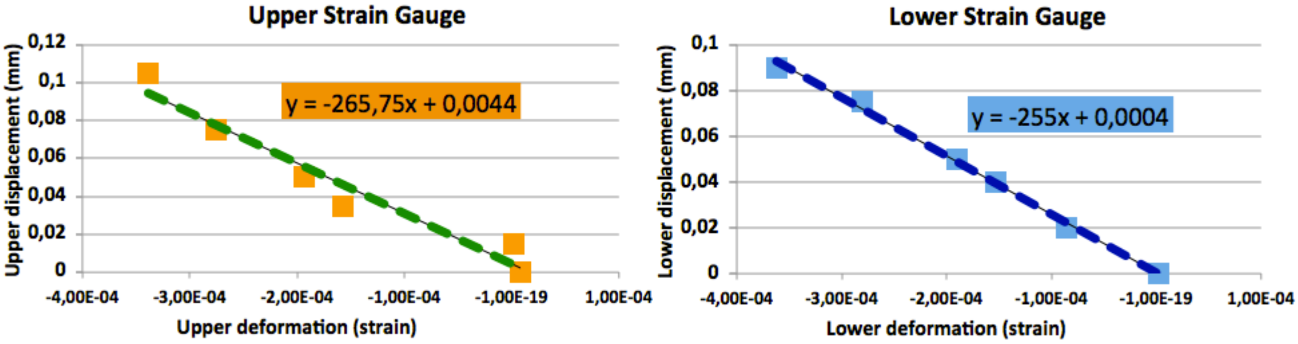

Figure 5. Calibration graphs and trend lines with linear behavior and characteristic equations. a) Upper bridge. b) Lower bridge.

These functions were used to determine the maximum displacement at the selected points to measure the brake caliper during a critical braking. In the second braking process, the prototype reached about 90 km/h before braking in a straight line until the complete stop of the vehicle. For data storage, a data logger National Instruments model 9237 was used, connected to a computer running the software LabView Signal Express 2013. The results of the strain vs runtime of the test are shown in Fig. 6.

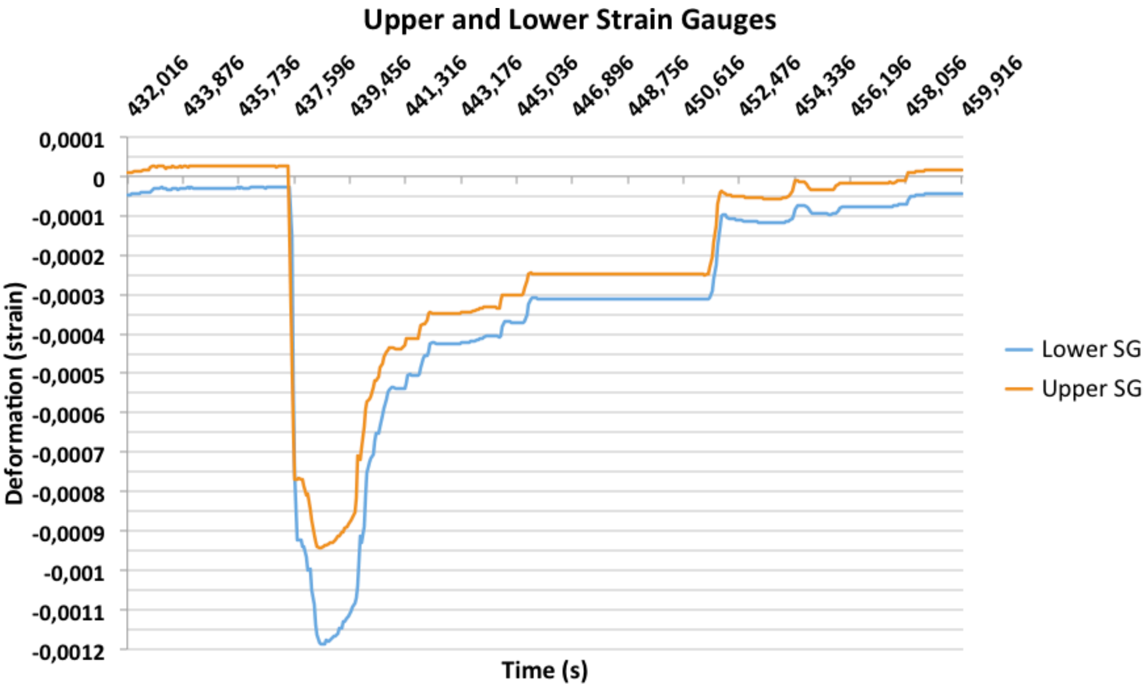

Figure 6. Deformation found during the test - Strain gage upper and lower vs. time in seconds.

The maximum strain value to the upper bridge was $-9{,}42\times10^{-4}\mu m/m$ while for the lower bridge this strain was $-1{,}19\times10^{-3}\mu m/m$. The equations found to calibrate the upper and lower bridges were $y = -265{,}75x + 0{,}0044$ and $y = -255x + 0{,}0004$, respectively. Substituting the values of "x" in these equations, the deformations obtained for the braking pressure in the front lines of $150 kgf/cm^2$, the maximum displacement in the upper and lower point can be obtained. This maximum displacement in the top spot was $0{,}255\ mm$. To the point below the maximum displacement was found to be $0{,}303\ mm$.

3. NUMERICAL ANALYSIS PROCEDURE

The procedure for numerical analysis is to obtain the baseline model in SolidWorks CAD software, transfer this geometry for finite element analysis and verification of the results found in Abaqus. If these results are not consistent with the experimental part performed in item 2.3 or the elastic limit of the material, the configuration of the simulated model should be checked, particularly the contacts between components and the boundary conditions. But if the results in the simulation are in accordance with the experiments of item 2.3, a return to the CAD to build the design and non design spaces should be done.

The geometry of these spaces will be transferred in the STEP file type to HyperMesh. In this software a preparation of the model analogously to Abaqus (with the same boundary conditions and application areas) was made. The Opticstruct is a module in HyperMesh which also requires the creation of the objective function (value to be reduced or increased) and the restriction conditions (design conditions to be met).

The brake caliper objective function under study is to reduce the volume and the restrictions are the displacement control of the nodes located in the lower region of the outside of both caliper cylinders toward the "z" axis so that it does not "open up" and does not cause excessive travel on the brake pedal during braking process. This control of the displacement of the nodes of the cylinders should be based on the results of displacement found in the previous simulation of Abaqus.

If the results of optimization are in equilibrium with the expected, the next step is the construction of the final CAD model. If the optimization results are different from the expected, the designer should return to the optimization model and check the boundary conditions, the objective function and constraints, among other parameters. Each case is very specific but it is known that the result depends directly on the input data determined by the engineer.

The construction of the final model in CAD should take into account the expected changes. To the case in study, is desired to change the body of two parts for monoblock. This construction should also evaluate the process by which the final piece will be manufactured. After this step, it should again lead to ask a finite element software and evaluate the stresses and strains encountered in order to identify and quantify improvements or not. If improvements are found and stresses are within the normal, optimization was completed successfully. Otherwise, it must return to the CAD and rework the final model. Fig. 7 is a flowchart summarizing all above described steps:

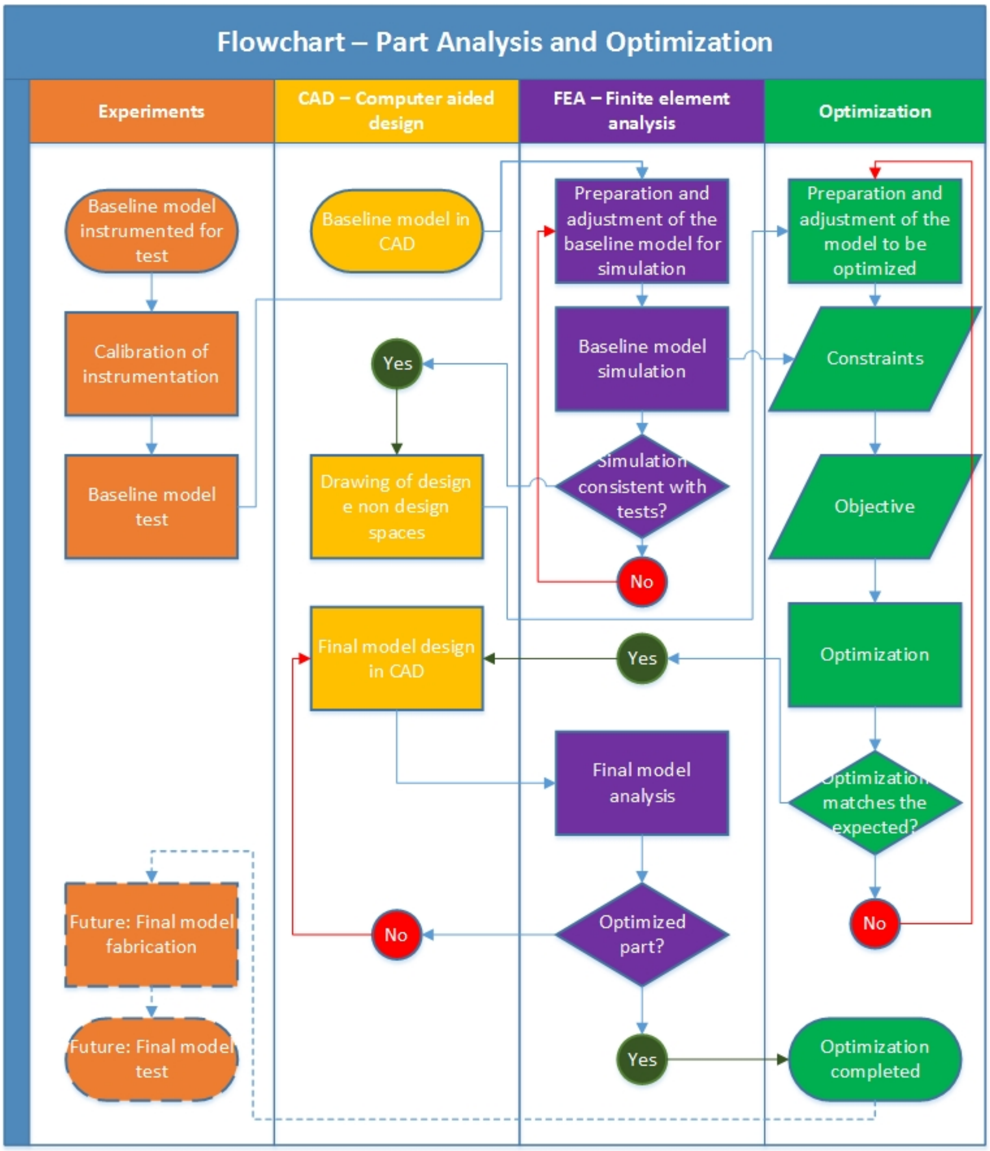

Figure 7. Model flowchart for analysis and part optimization.

After conclusion of the optimization, it should be worked in manufacturing the final model and testing for validation of the entire process. These last steps are not scope of this work and will be suggested for future work, so the arrow indicating these processes is dotted in the flowchart.

3.1. Baseline model simulation

The brake caliper used on the prototype was adopted as the baseline model, where data obtained from real tests will be compared with the results obtained in simulation to verify the accuracy of the simulated offset values. This reference model will still be compared to the new model developed in this work (final model).

All simulations were performed on a notebook with Core i5 processor (i5-540M) and 4GB of RAM. It was simulated the baseline model with the Abaqus software.

The maximum axial displacement was observed for the two points previously measured in item 2.3. For the top point, the maximum measured displacement was $0{,}255\ mm$ and the one found in the simulation was $0{,}301\ mm$ (red dot in Fig. 8a). The error is approximately $15{,}28\%$ for this situation.

For the lower point, the measured maximum displacement was $0{,}303mm$ and the found in the simulation was $0{,}370mm$ (red dot in Fig. 8b). The error was approximately $18{,}11\%$.

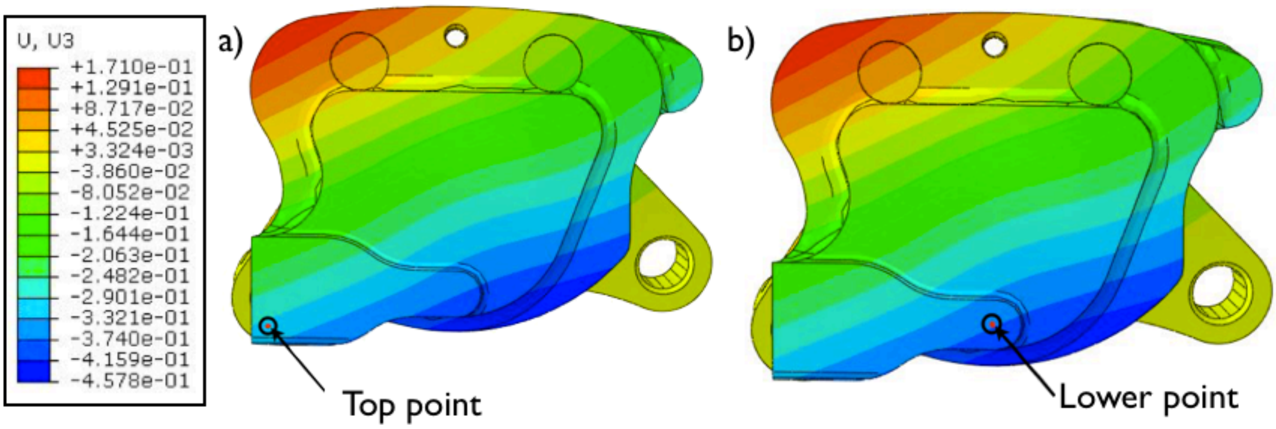

Figure 8. Axial displacement (simulation of the baseline model). a) Top point. b) Lower point.

These errors are related, mainly to difficult modeling the contact of the screws with the caliper body where a pre charge was not added by convergence problems in the results. Furthermore, the measured values have a variation and it is known that if the experiment was done again, the same values would be hardly found because the pressure was visually identified in the manometer, causing inaccuracy. Another factor that contributed in increasing the error was the difficulty to control the brake pedal modulation, where there had been an inadequate sensitivity.

Comparing the simulation results obtained in practice, even after the verification of possible sources of error, there was obtained an acceptable accuracy in simulated model. This model, with the same forces and boundary conditions served as a reference for the construction of the model to be optimized.

3.2. Optimization

The new design of the brake caliper will mantain the same pistons, bleed screws, pads and sealing rings of the previous caliper, so the body of the caliper is the only part that is actually modified. This piece will be no more composed of two parts joined by screws, becoming a body with monoblock configuration.

In order to accelerate the process in the conceptual stage, the structural optimization method was adopted. According to Altair (2014), in the conventional design process, the designer must rely on their own experience and expertise to present proposals. The analysis tool is then used to evaluate each proposal causing the designer to measure the test results and decide what is the "best". Using the design optimization as part of CAE, both activities can be performed simultaneously, that is, the designer tdefines the restrictions and leaves with the optimization tool to check the proposals. The optimizer uses the analysis tool to decide how to change the initial design to get to a better design.

The results that the software will produce are completely depended on the input information that the engineer provides. The "best" project will also be defined by this engineer, however, aiming to know what is the "optimal" in a project, objectives and constraints are needed, quantitative parameters used for the evaluation of this project.

For the case in study, the project area or design space (Fig. 9a) is the limit space where the brake caliper may be, respecting the region where the brake disc will pass until the wheel inside subtracting from this the wheel hub. In this volume the solver can remove material deemed unnecessary.

The non design space (Fig. 9b) is the volume where optimization is not performed. This volume will remain unchanged throughout the optimization process and, therefore, consists of the parts that are not part of the caliper body.

These parts are the fixings of the caliper with the upright, the hydraulic connection of brake fluid, abutment areas and a cover around the regions where are located the cylinders to receive the hydraulic pressure and transfer it to the body.

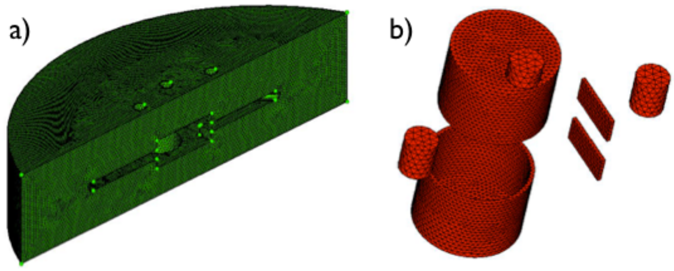

Figure 9. a) Design space in green.b) Non design space in red.

The model to be optimized was set and after 51 iterations this model converged. For the constraint of displacement control of the lower rear wall portion nodes that are on the side of attachment to the prototypes upright (region with smaller axial displacements) a maximum of $0{,}06mm$ of displacement was selected, i.e. the displacement in "z" direction of these nodes could not be greater than $0{,}06mm$ (rounded value of $0{,}04525mm$ found in orange region of the graph of Fig. 8). For the nodes of the lower rear wall portion opposite to the fixed part in the prototypes upright, a lower limit of $-0{,}470mm$ was adopted, that is, the amount of axial displacement of the nodes of that part can not be less than $-0{,}470mm$ (rounded value of $-0{,}4578mm$ found in Fig. 8 legend).

An image of the optimization with the plotting of the isodensities above 0.3 is shown in Fig. 10, with the project area highlighted in light gray. It is noticed that the material is concentrated in the central region, near the fixings and applying loads.

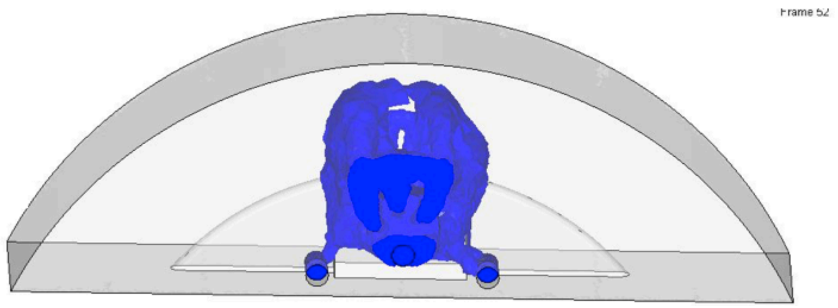

Figure 10. Isodensities plot and design domain view.

4. RESULTS

4.1. Final model simulation

The CAD optimized model or final model is a monoblock brake caliper without the need for screws to connect the two parts of the caliper body. This final model was adapted for the same pads and accessories of the baseline model.

It is noticed that the part has a certain similarity with the baseline model, however, a deeper analysis reveals mass reductions in the central part of the "bridge" with material required only for the positioning of the pads. A reinforcement was added to the tangential load support wall, increasing the thickness by 1 mm in all these walls. The arches found in the optimization were also added with different thicknesses. In Fig. 11 is an isometric view of the optimized caliper.

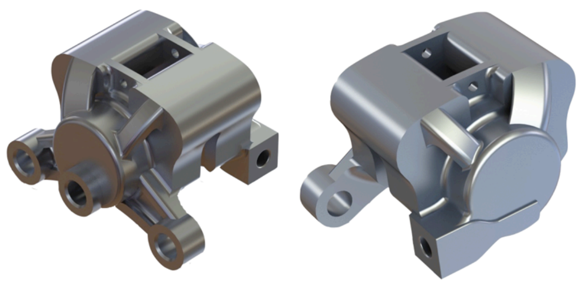

Figure 11. Final model in CAD.

4.2. Final model simulation

Analogously to the baseline model, a simulation was configured for the final model in Abaqus software. The same loads and pressures were applied to the same areas and the overall result of displacements can be seen in Fig. 12:

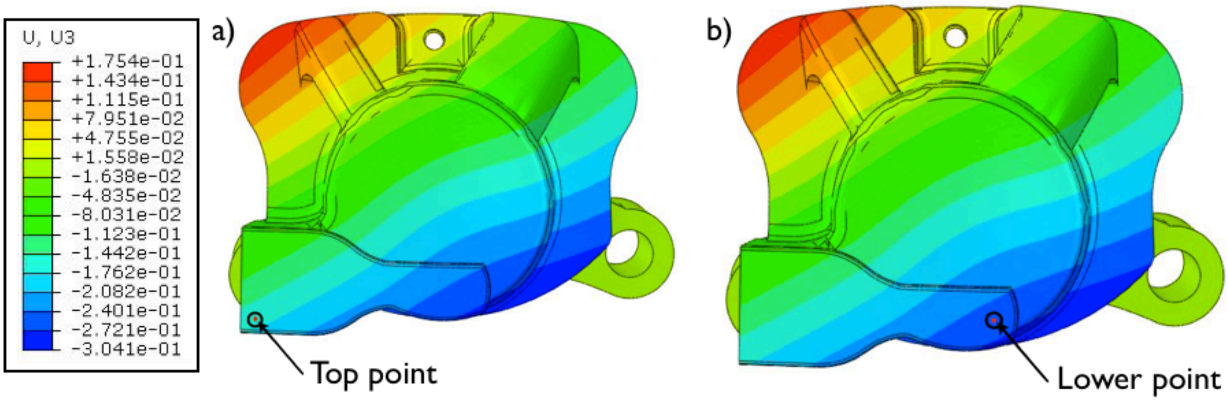

Figure 12. Axial displacement (simulation of the final model). a) Top point. b) Lower point.

For the top point, the maximum displacement found in the simulation was $0{,}164mm$ (red dot in Fig. 12a). For the lower point, the maximum displacement found in the simulation was $0{,}252mm$ (red dot in Fig. 12b).

5. CONCLUSION

After optimization, a new geometry was created with mass reduction from 210 grams of the initial model to 166 grams in the final model, that means 21% in weight reduction.

The reduction in the maximum axial displacement was also observed in the finite element simulation, where in the initial model the maximum displacement was 0,46mm and went up to 0,31mm in the final model. These results represent an approximately 32.6% of reduction in the maximum axial displacement, i.e., the structural optimization method can be extremely useful if properly known the boundary conditions, objectives and constraints of the analysis.

This paper presents a specific methodology to perform a topology optimization of a brake caliper body. The methodology present in numerical analysis section can be extrapolated to any part that need to be optimized. Although the analysis considers simplifications (Between the items not covered by this analysis are: Pad wear not uniform, influence of temperature), the proposed procedure is shown to be consistent and presented positive results.

6. REFERENCES

Abaqus, 2011, "ABAQUS Documentation", Dassault Systèmes, Providence, RI, USA.

Altair, 2014, "Practical Aspects of Structural Optimization", A Study Guide. Ebook. Altair.

Dassault Systèmes - SolidWorks Corporation. 2010. "Manual do Instrutor de Ensino do Software SolidWorks®", Concord, Massachusetts, EUA. Avaliable in http://www.solidworks.com/sw/docs/instructor_wb_2011_ptb.pdf.

Farias, L., 2016, "Projeto e Otimização Topológica de uma Pinça de Freio fixa de dois Pistões para um Protótipo de Formula SAE". Trabalho de Conclusão de Curso (Graduação em Engenharia Mecânica) – Universidade Federal de Santa Maria, Santa Maria, RS.

Farias, L., Schommer, A., Haselein, B., Soliman, P., de Oliveira, L., 2015, "Design of a Brake Caliper using Topology Optimization Integrated with Direct Metal Laser Sintering", SAE Technical Paper 2015-36-0539, doi:10.4271/2015-36-0539.

Limpert, R., 1992, "Brake Design and Safety", Society of Automotive Engineers, ISBN: 1-56091-261-8. 2nd edition. Warrendale, PA.

National Instruments Corp., 2004, "Getting Started with SignalExpress". Avaliable in http://www.ni.com/pdf/manuals/373873a.pdf.

National Instruments Corp., 2009, "NI9237 Operating Instructions and Specifications". Avaliable in http://www.ni.com/pdf/manuals/374186e.pdf.

Puhn, F. , 1985, "Brake Handbook". 2nd Ed. Tucson: HPBooks.

Sergent, N., Tirovic M., Voveris J., 2014, "Design Optimization of an Opposed Piston Brake Caliper". Engineering Optimization, 46:11, 1520-1537, DOI: 10.1080/0305215X.2013.846337.

7. COPYRIGHT RESPONSIBILITY

The authors are the only responsible for the content of this work.