Improving brake efficiency by optimizing the design of a car brake caliper

M Gaidur, I Pascal, C C Ciobotar, E Rakosi, S Talif, Gh. Manolache and M Nazare
Mechanical and Automotive Engineering Department, “Gheorghe Asachi” Technical University of Iasi, Romania
E-mail: mihai.gaidur@gmail.com

Abstract. The continuous advancement in the field of automotive engineering supports the development of new working techniques regarding the definition of new machine parts. In this study, the topological optimization of a tangentially loaded brake caliper is treated. For the first part of the study, the distribution of forces inside the caliper is explained. In the next stage, the computation stage, the working method approached for topological optimization is described. The results and conclusions are commented in the last part of this paper.

1. Introduction

In the first part of this paper, the relevant forces at the level of the caliper were evaluated. Later, they serve as input data for the topological analysis. For the determination of the forces, the following parameters are established: the value of the pressure inside the hydraulic circuit of the braking system $P = 100$ [bar], the friction coefficient of the friction gaskets $\mu = 0.4$, the diameter of the main brake cylinder $d_1 = 45$ [mm], the diameter of the secondary braking cylinder $d_2 = 40$ [mm], and the diameter of the tertiary braking cylinder $d_3 = 35$ [mm].

As a constructive solution, a fixed six-cylinder caliper was chosen. Usually, a caliper can contain from two to six drive cylinders, the latter being used for very high demands. The different diameters of the cylinders result from the fact that large lateral forces appear inside the caliper and the cylinders from outside are heavier loaded in comparison with the middle one. With this method, a balance of pressure distribution on the friction gasket was ensured, and the braking efficiency was acquired [1].

The following known equation was used to determine the axial forces: $F_{Ai} = P \cdot A_i$ ($i = 1, 2, 3$), where $P$ is the pressure inside the braking circuit [MPa] and $A_i$ is the surface of the brake piston, related to its cylinder. The areas are defined in the following matrix:

$$A_i = \begin{bmatrix} \frac{\pi \cdot 45^2}{4} \\ \frac{\pi \cdot 40^2}{4} \\ \frac{\pi \cdot 35^2}{4} \end{bmatrix}.$$
In the given conditions, the $F_{Ai}$ axial forces, with $i = 1, 2, 3$, are defined in the following matrix:

$$A_i = \begin{bmatrix} 1590 \\ 1256 \\ 962 \end{bmatrix} \text{[mm}^2]\] \quad (2)$$

The next matrix contains the numerical values of the $F_{Ai}$ axial forces:

$$F_{Ai} = \begin{bmatrix} F_{A1} \\ F_{A2} \\ F_{A3} \end{bmatrix} \rightarrow F_{Ai} = \begin{bmatrix} 10 \cdot 1590 \\ 10 \cdot 1256 \\ 10 \cdot 962 \end{bmatrix} \quad (3)$$

Once the forces acting inside the cylinder are determined, the tangential forces on the surface of the friction gasket are also defined. These are calculated with the following equation, where $\mu$ is the friction coefficient between the friction gasket and the brake disc [1]:

$$F_{Ti} = \begin{bmatrix} F_{T1} \\ F_{T2} \\ F_{T3} \end{bmatrix} \rightarrow F_{Ti} = \begin{bmatrix} 0.4 \cdot 10 \cdot \frac{\pi 45^2}{4} \\ 0.4 \cdot 10 \cdot \frac{\pi 40^2}{4} \\ 0.4 \cdot 10 \cdot \frac{\pi 35^2}{4} \end{bmatrix} \quad (5)$$

The values of the tangential forces, for each cylinder, are defined in the following matrix:

$$F_{Ti} = \begin{bmatrix} 6361 \\ 5026 \\ 3848 \end{bmatrix} \text{[N]} \quad (6)$$

Once the numerical values of the axial and tangential forces are determined, they serve as input data for the applicative part of this study.

2. Mathematical approach of the optimization criteria

The theory on which the topological optimization is based is distinct from the standard optimization of the surfaces, where the working area contains only a subset of allowed work surfaces, surfaces that have topological properties with imposed limits, for example, a certain number of holes in them. Topological optimization represents a mathematical approach with the role of perfecting the arrangement of material layers, in a given space, with specific loads and associated constraints so that the final result meets the requirements inflicted by the user. Using this tool, engineers can find the best product concept, a product that meets the required design conditions. For the analysis of the designs, the same techniques are used as in the FEA (Finite Element Analysis) analysis, and for their optimization, genetic algorithms and topological derivatives were used [2].

Figure 1. The sequence of steps that must be followed to perform a topological optimization analysis.
The optimization queries are marked in the next three steps, described through the graph in figure 1.

2.1 Structural model creation
The structural model represents the structure to be optimized and it describes mathematically or numerically the physical behavior of the structure. Optimal problems are mostly nonlinear and contain many restrictions. The algorithms that solve these problems are based on sets of iterative calculations starting from initial design, and the goal is the achievement of vectors with improved design variables.

2.2 Optimization model creation
The optimization model represents the link between the initial (non-optimized) model and the optimization algorithm. It has the role of evaluating the imposed design according to the commanded optimization objective, as well as the constraining conditions of the three-dimensional model. The definitions and transformations of the optimization model are called parameterization. During the optimization process, the nodes that formed the initial mesh will shift their position to form a new mesh. The new mesh will represent the final geometry of the new optimized model.

The design variables expressly define the final form of design.

The design model describes the mathematical architecture between the analysis variables and the design variables.

2.3 Model validation
The convenient method used for validating a topologically optimized design is represented by finite element analysis. The main criteria that are verified are how the optimized part will respond to both the initial loads and the limit loads.

This concept is called "Three Columns Concept". The concept presented in this paper was developed by Eschenauer [2]. He says that a practical and straightforward approach is to break the problem in several stages, which are solved directly and successively. This process is developed to work with mathematical programming algorithms, but can also work with genetic optimization algorithms (figure 2).

![Figure 2. Model validation cycle that was used to design the brake caliper [5].](image-url)
The software used for this analysis is AUTODESK FUSION 360, specifically, the Shape Optimization module. This module is capable of:

- Defining project objectives, defining constraints, establishing materials and can also simulate manufacturing options;
- Generating and exploring alternative solutions for the project. That is, it can quickly create design variants by generating multiple design processes and exploring viable options. Project alternatives are not processed or stored on the local computer, but in the AUTODESK cloud;
- Project optimization and material choice. The program can evaluate the critical aspects of the project after each design decision and even analyze the trade-offs that are essential in terms of the goals initially set;
- Exporting CAD files. After choosing the best solution identified, the respective CAD geometry is brought from the cloud to the local computer. The advantage of this solution is that it offers the possibility of investigating a topological analysis even for users with limited resources [3].

3. Analysis and results interpretation

For the applicative part of this paper, it is necessary to define the input parameters (e.g. forces, moments, material, etc.) together with the restrictions during the operation (e.g. active surfaces of the caliper, surfaces representing interfaces with other parts, etc.). Depending on the constraints imposed on the computational relations, the optimal shape of the analyzed part will be determined. The calculation relationships underlying this process aim to determine the values of the minimum stresses, the minimum deformations, the minimum displacements, the minimum mass as well as the minimum volume of the part. Based on a set of calculations using genetic algorithms, optimal criterion method, level analysis and topological derivatives, in addition to the dedicated software programs, this optimization process also needs a computer with powerful hardware resources.

The geometry of the analyzed caliper can be seen in figure 3.

![Figure 3. The mesh of the braking caliper.](image1)

![Figure 4. The defined loads inside the brake caliper.](image2)

After defining the geometry, the second step consists in adopting the material for manufacturing the piston. It requires a material that can withstand the amplitude of the forces to which it is subjected and also has high machinability. For this case, the Al7075 aluminum alloy was chosen. As the main element of
zinc alloy, this alloy is a resistant one, comparable to many steel alloys; it has excellent fatigue resistance and average workability. In the composition of this alloy there are the following elements: between 5.6% and 6.1% Zn, between 2.1% and 2.5% Mg, between 1.2% and 1.6% Cu and below 0.5% Si, Fe, Ti, Cr. The mechanical properties of this material depend on whether it was subjected to heat treatments or not. The untreated 7075 has a maximum tensile strength of 276 [MPa] and an elasticity limit of 145 [MPa]. The maximum elongation of the material is between 9% and 10%. The 7000 series alloys are mainly used in the automotive, marine and aeronautical industries due to the high resistance and density ratio [4].

The next parameter that is defined is the target mass. We opted for a 15% reduction in mass. Next, it was chosen to maintain the initial rigidity of the caliper. The distribution of forces is done by the following (figure 4):

- the axial forces, $F_{Ax}$, will be applied inside the brake cylinders;
- the tangential force, $F_T$, will be generated by the frictional moment between the brake disc and the friction gasket;

The load path throughout the brake caliper is represented in figure 5.

![Figure 5](image5.png)

**Figure 5.** The load path criticality represented throughout the brake caliper.

![Figure 6](image6.png)

**Figure 6.** Detailed view of a point probe with CRT at maximum value, CRT = 1.

![Figure 7](image7.png)

**Figure 7.** Detailed view of a point probe with CRT at minimum value, CRT = 0.3176
The results of the optimization process can be seen in figure 6, figure 7, figure 8 and figure 9.

**Figure 8.** Isometric view with the topologically optimized caliper.

**Figure 9.** Isometric view from a second plane with the topologically optimized caliper

Figure 6 and figure 7 show the load path criticality values. This parameter will be noted with CRT, and the scale of its value is shown in figure 10.

The load path criticality is a discrete variable that ranges from 0 to 1. A value of 1 represents a region in the model that is critical to resist the applied load. A value of 0 represents a region in the model that is not critical to resist the applied load. When attempting to achieve the target mass, the shape optimization module removes the elements with the lowest load path criticality values first [5].

Following the study and analysis, the values obtained at the level of the optimized caliper are synthetically reproduced using the diagram in figure 11. The 0.5 value represents the limit, from which, any more material removal would diminish the global stiffness and the resistance to the applied load.

**Figure 10.** The load path criticality variation scale

**Figure 11.** The limit values representation of the CRT coefficient.
The 3D printed optimized model is represented in figure 12. A 1:1 scale model was printed in order to validate the new geometry.

![3D printed topologically optimized brake caliper at 1:1 scale](image)

**Figure 12.** 3D printed topologically optimized brake caliper at 1:1 scale

### 4. Conclusions
- The reduction of mass ensures a more accessible part, with lower consumption of material and implicitly lower costs;
- Under these conditions imposed by rigidity, the absolute values of the CRT coefficient, values between 0.3176 and 1, are highlighted. However, even if the value of CRT has decreased, the part is still within the optimal range of values;
- The corresponding rigidity obtained allows the use of different diameter cylinders, which ensure better resistance to larger lateral forces as well as a more even distribution of the pressure on the friction gasket;
- Such an approach, based topological optimization criteria, ensure the development of more complex studies that explore a broader spectrum of influencing factors, highlighting several relevant, useful determinants in the optimization process.

### 5. References

[1] Farias L, Schommer A, Haselein B, Soliman P and Oliveira L 2015 Design of a Brake Caliper using Topology Optimization Integrated with Direct Metal Laser Sintering *24th SAE Brasil International Congress and Display* 2015 pp 539-551
[2] Rozvany G and Lewinski T 2014 Topology Optimization in Structural and Continuum Mechanics *CISM International Centre for Mechanical Sciences Courses and Lectures* Springer 2014 pp 549 ISSN 0254-1971 ISBN 978-3-7091-1643-2
[3] Badut M 2019 Fusion 36 from https://www.ttonline.ro/revista/cad-cam-cae-pdm-plm-erp/fusion-36
[4] Gökhan Ö and Karaslan A 2017 Properties of AA7075 aluminum alloy in aging and retrogression and reaging process *Transactions of Nonferrous Metals Society of China* 27 pp 2357-2362
[5] Autodesk Inc. 2020 Product Documentation Fusion 360 Help Shape optimization results, from http://help.autodesk.com/view/fusion360