



COB-2021-1720

BAJA SAE VEHICLE BRAKE DISC DESIGN THROUGH A THERMO-STRUCTURAL ANALYSIS USING THE FINITE ELEMENT METHOD

Mateus Feitosa Camboim

Ludmila Martins de Araújo

Universidade Federal da Paraíba, Campus Universitário I, Dpto. Eng. Mecânica, João Pessoa/Pb CEP: 58051-900

ludmila.martins@academico.ufpb.br

mateusfcam10@gmail.com

Nathalya Rebeca Melo da Cunha

Universidade Federal da Paraíba, Campus Universitário I, Dpto. Eng. Mecânica, João Pessoa/Pb CEP: 58051-900

nathalyarebeca@gmail.com

Rodolfo Vieira Sarmento

Universidade Federal da Paraíba, Campus Universitário I, Dpto. Eng. Mecânica, João Pessoa/Pb CEP: 58051-900

rodolfosarmento01@gmail.com

Abstract. *The present work aims to describe a Baja SAE vehicle brake disc design, with the aid of reputable tools in the development of mechanical projects such as ANSYS INC, simulating the brake disc working conditions in order to obtain the working temperature and stresses induced by temperature and mechanical requests through a thermostructural analysis and, from the results, define the geometry and material most suitable for this application best performance. The activity that will be described here was developed by the Federal University of Paraíba Baja SAE team (UFPBaja) which, located in the academic scope, seeks to disseminate and assist the importance of understanding finite element method (FEM) use and its tools for the development of new products using as an example the team's brake disc project. It was possible to determine the geometric parameters (radius, thickness, shape pattern, mass relief) as well as the ideal material to be used to meet the established goals, allowing to achieve more than 50% mass reduction compared to the model used previously and, in this case, eliminating prototyping costs.*

Keywords: *brake disc, finite element method, thermo-structural, steel, baja sae, thermal simulation.*

1. INTRODUCTION

Over the years and with constant technological advancement, industry had to adapt to launching new products in ever shorter intervals. The demand for better, cheaper and more technological products has required companies to be qualified to meet and keep up with market change speed due also to rivalry. In contrast, technological advancement in the automotive field also helps industries evolution through engineering software emergence that made a new product development much faster and cheaper. Likewise, academic projects such as Formula SAE and Baja SAE have benefited from engineering tools power, since simulation software using finite element method dramatically reduces project definition time and prototyping costs, helping to improve academic engineering and, consequently, competitions level.

It is in this context, this work seeks to detail the development of one of the pieces produced by the Baja SAE team at Federal University of Paraíba (UFPBaja), seeking the optimization, reliability and continuous improvement of the prototypes, thus adding to the engineering achieved level in the academic environment. In Figure 1, the most recent UFPBaja's prototype is shown.



Figure 1. UFPBaja's Indominável prothotype during 2020 National Championship.

The activity described here was motivated by putting into practice a continuous improvement plan and, with this, compete at a high level since UFPBaja is inserted in the academic scope where every year are held two competitions: first with their respective region teams, and later the national competition in which all Baja SAE Brazilian teams face each other.

The competitions in question are conceived by the international Society of Automotive Engineers (SAE), where each university teams take their own Baja SAE off-road vehicle to have tested the projects conceived by them during the year. In this scenario, the goal is to improve the prototypes each year, being one of the proposals the development and manufacture of the largest number of parts and components possible by the team itself, as required by the competition regulation, and using the infrastructure and equipment available in the university.

Several components used in the vehicle can be developed and manufactured by the team itself or purchased, but there is a problem tied to the second one, since the vast majority of options found in the market are specific products for commercial vehicles. Thus, these parts are often oversized for applications in the Baja SAE setup, with the main consequence an increase in prototype's mass, a factor that directly interferes in its performance.

As part of the efforts to work around the problem and being the object of this study, a new brake disc was developed to support the vehicle mechanical requests and with the objective of reducing the new part mass, compared to that previously used, thus avoiding oversizing, but offering the same performance and reliability found in Original Equipment Manufacturer (OEM) parts.

For the brake disc sizing was used the finite element method (FEM), more specifically a thermostructural simulation, which consists of an analysis performed through the ANSYS finite element software, in which a digital model of the part was used to simulate the working conditions the component is submitted. Hence, it was viable to obtain the stresses and deformations caused by mechanical requests during the use and, through these results, it was possible to define the geometric aspects (thickness, radius, pattern, etc.) as well as the material with ideal properties to be used and, so, reach the desired results.

2. FINITE ELEMENTS METHOD (FEM)

2.1 3D Modeling

First, it was necessary to define a 3D digital model of the part to be designed. For this, the computer-assisted drawing (CAD) software SOLIDWORKS was used. The disk model commonly used by UFPBaja is a wave pattern, named after the disk outer edges pattern that resemble waveforms, making this model lighter compared to the more common formats such as solid disks. In addition, the current design was modeled to maximize mass reduction and, with this, cutout patterns were added to the disc, without affecting the area swept by brake pads that is covered during a complete revolution. Thus, the resistive forces responsible for braking, caused by the brake pads friction, were not affected, what also helped to reduce the mass even further.

Another reason for adding mass reliefs was improving heat dissipation, one of the most critical factors in this type of application as high temperatures can be extremely impaired to the brake correct operation or even cause complete failure. The increase in temperature causes a decrease in the coefficient of friction, which can origin a fault known as fading (Limpert, 2011), considerably affecting braking efficiency, what should be avoided. Figure 2 shows the proposed final disk model after taking into account all the factors mentioned.



Figure 2. UFPBaja's brake disc final 3D model.

2.2 Overview in Finite Element Method Analysis

As can be seen in Figure 2, the disk proposed geometry is composed of several details that make it complex. Elements and components physical behaviors are described by the theory of solid mechanics (or strength of materials) and by the theory of elasticity for deformable objects in general (Riley *et al.*, 2003). Stresses, deformations, moments and efforts are described through equilibrium equations, departing from differential equations that describe bodies physical behavior. This complex mathematical approach, allows to obtain the exact solution of mechanical problems in general, but it requires a lot of time to generate this solution and they can be applied only for objects with simple and basic shapes. For intricate and abstract forms, as in UFPBaja's brake disc, the precise results become extremely difficult to obtain, thus needing FEM to solve problems as the one in analysis.

In the classical form of mathematical treatment, differential equations govern the physical behavior along infinite points that constitute the object of study; by the FEM method the approach consists of discretizing the object in question in a finite number of elements.

Alves Filho (2018) describes that the structure, the mechanical component or, in general, the continuous body is subdivided into a finite number of parts (elements), connected to each other through discrete points (nodes). The author also states that the elements assembly, which constitutes the mathematical model, has its behavior specified by a finite number of parameters. Specifically in the structural scope, which corresponds to the case presented here, the parameters mentioned refer to element nodes displacements, as from the displacements of the nodes it is possible to find the resulting forces or stresses within the element. This becomes possible because, according to the principle of balance found in the theory of mechanics, if a body is in equilibrium, then all the constituent parts of that body are also in equilibrium. This principle supports the possibility of discretizing an object into several elements.

With the state of equilibrium guaranteed, from the study of a free body diagram of the elements it is possible to assemble the constituent relation between displacements and forces (or stresses). With the displacements of each node, it is possible to obtain the internal forces active, condition that is intrinsic to the behavior of materials. When in the linear regime, this relationship is described by Hooke's law through Eq. 1:

$$\{f\} = [k] \cdot \{d\}, \quad (1)$$

Where $\{f\}$, $[k]$ and $\{d\}$ are the matrices of force, stiffness and displacement, respectively.

The previous relation is described in the matrix form since Alves Filho (2018) says that the most efficient way to store this information and process it is through matrices, which are widely used in the procedures of MEF.

Since the stiffness matrix represents the stiffness or deformation resistance of a given element, its shape is defined by the number of nodes and the amount of behaviors the element is able to simulate. In Alves Filho (2018) the spring element, which is the simplest of the library of elements and has two nodes, one at each end, is used to demonstrate the principle of its formulation. In addition, the spring element is capable of transmitting only efforts and axial displacements, thus having two displacement components and a dimensional matrix 2×2 .

Generally speaking, for any element with "n" offset components, its matrix will be of the $n \times n$ order. For the spring, its stiffness matrix $[k]$ is represented in Eq. 2.

$$[k] = \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix}, \quad (2)$$

Where k_{11} is the force that appears on node 1 due to the offset caused on node 1, while node 2 was locked, k_{21} is the force that appears on node 2 due to the offset caused on node 1, while node 2 was blocked, k_{12} is the force that appears on node 1 due to the offset caused on node 2, while node 1 was blocked, and k_{22} is the force that appears on node 2 due to the offset caused on node 2, while node 1 was blocked.

The coefficients present in the stiffness matrix of any element need to be defined. The procedure for this is to analyze each node alone, and movement constraints should be applied, i.e., to apply offset equal to zero on a constrained node ($\{d\} = 0$) in the displacements matrix and a unit offset to the other ($\{d\} = 1$) in Eq. 1. In the case of the spring element that has only two nodes, the offset matrix is a column type one with two rows and a coefficient for each node, that is, 2×1 . The details of the procedure to obtain these parameters will not be described, as it is not the focus of this paper. Thus, it is observed through Eq. 2 that each coefficient of the stiffness matrix has a well-defined physical meaning, where each term represents the force required to cause a unit displacement at one node, keeping the other node fixed.

With this, the behavior of the spring element can be obtained. This physical definition can be generalized for any finite element whose matrix $n \times n$ also transmits any behavior of mechanics, even simultaneously as torsion, bending and shear.

This is one of the key points of the FEM method analysis, because the elements are connected to each other by the nodes, i.e., where the forces are transmitted, and thus the response of the discretized object can be obtained. Thus, with the addition of working conditions (restrictions of freedom degrees, external forces, types of connections, etc.) through the boundary conditions, with the appropriate choice of considerations by a prior study of the component in question use,

an answer is obtained from the approximate physical behavior of reality to any variable of interest, in any application and with any complexity such as UFPBaja's brake disc.

3. THERMOSTRUCTURAL ANALYSIS FOR SIZING A BRAKE DISC

As stated in the previous topic, it was necessary a study on the processes involved in the application in question through the boundary conditions, because the applied considerations should be as close as possible to the part actual working conditions, so the results obtained would be consistent with the application physical processes. The process variables should be known since the input data for the analysis must be obtained in advance. The procedure of acquiring the thermostructural analysis data described here will be detailed in next.

3.1 Thermal Analysis

Thermal analysis was crucial in this type of application, as the working temperature directly influences the efficiency of the braking system. To avoid possible problems and even failure by fading, it was important to predict which temperatures the system reaches and adopt corrections or changes in the design when necessary. With this, to perform the thermal analysis, some parameters based on fundamental principles of the heat transfer theory had to be obtained previously.

Considering what the second law of thermodynamics says, for two systems with different amounts of thermal energy that share borders, there will be a flow of thermal energy between the borders with the direction of flow from the highest to the lowest energy until both reach the equilibrium (Çengel *et al.*, 2006). However, for there to be the opposite direction of the flow, it is necessary that there is a device that performs work on the environment, as is the example of refrigerators in general.

The thermodynamic equations describe the energy transfer rates, but do not describe the type of process through which they occur, but using the properties of heat transfer theory it was possible to identify by which processes the transport takes place. The first mode of energy transfer required to perform thermal analysis is conduction, more specifically the heat flow, between the brake pads and the disc and the external medium. Bergman *et al.* (2011) defines conduction as the transport of energy in a medium due to a temperature gradient, being governed by Fourier's law (Eq. 2):

$$q_x^n = -k \frac{dT}{dx} \quad (2)$$

Where q_x^n is the thermal flow in the "x" direction perpendicular to the transfer direction, k is thermal conductivity of the material and dT/dx is the temperature variation or gradient in that direction.

However, Limpert (2011) proposes a different and specific approach to obtaining the heat flow already applied to the purpose in question, which can be seen through the equations arranged in Table 1.

Table 1. Heat flow obtaining through the disc swept area.

HEAT TRANSFER	
HEAT FLOW, W/m ²	MAXIMUM DISK POWER, W
$F_c = \frac{P_m}{A_v}$	$P_m = \frac{D_m \cdot V_{lm} \cdot M_v \cdot 0,5}{2}$
P_m – Maximum disc power, W	D_m – Maximum deceleration, m/s ²
A_v – Swept disk area, m ²	V_{lm} – Maximum linear speed, m/s
	M_v – Vehicle mass, kg

The second mode of heat transport required to perform the analysis was convection calculation, more specifically the convection coefficient. Bergman *et al.* (2011) says that the convection process occurs with energy transfer between a wall and a moving fluid and, due to random molecular movement (diffusion), energy is also transferred through the global, or macroscopic, fluid movement. Convection process can be classified according to its nature: forced, when the flow is caused by external means; or free, when the fluid flow occurs naturally due to the buoyancy forces caused by the density differences arising from the difference in fluid temperatures. Being free convection the case that fits here, Limpert (2011) proposes to obtain the convection coefficient by Eq. 3:

$$h_{conv} = 0,4 \left(\frac{k_{ar}}{D} \right) Re_w^{0,8} \quad (3)$$

Where h_{conv} is the convection coefficient, k_{ar} is the air thermal conductivity, D is the brake disc diameter and Re_w is the rotational Reynolds number.

Finally, it was required to define the radiation process through the brake disc walls. For the radiation addition in the analysis, it was necessary to define only some parameters such as ambient temperature and emissivity of the material (ϵ).

With this, all the need parameters for thermal analysis were obtained. Its application and finite element mesh can be seen in Figure 3.

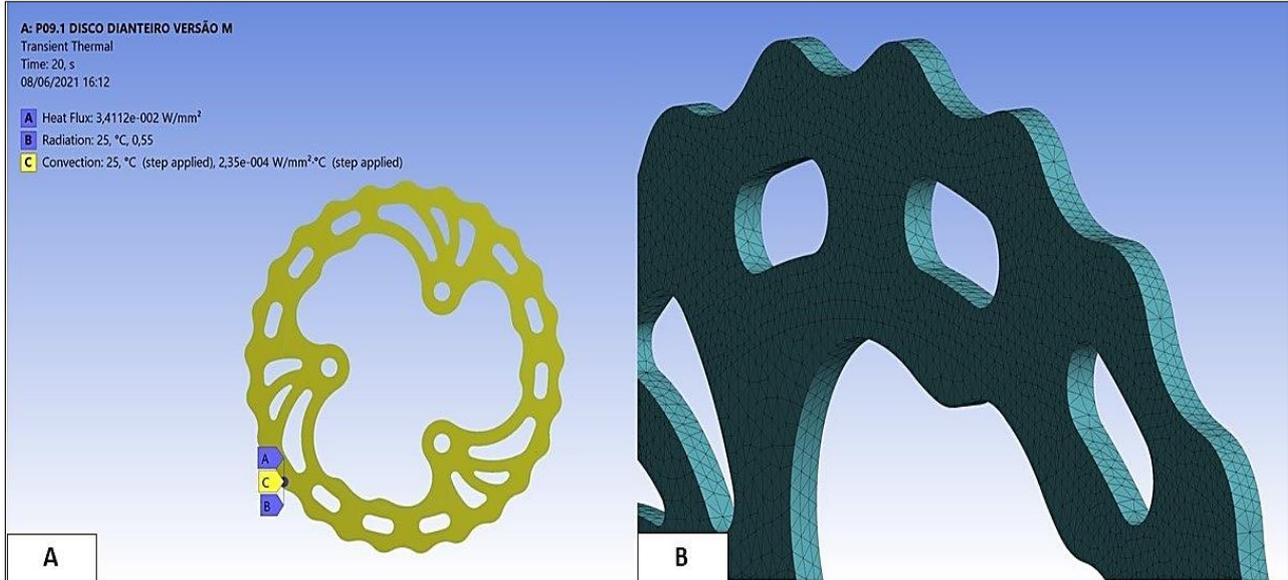


Figure 3. Image A shows the boundary conditions of thermal analysis; image B displays finite element mesh.

3.2 Structural Static Analysis

After finishing the thermal analysis, the subsequent step was to import the results for a structural analysis, which is possible due to the ANSYS software's Workbench tool allowing simulations of several distinct physics (fluid dynamics, electromagnetic, dynamic, static, etc.) to be performed in the same environment.

In addition, the tool enabled multiphysical simulations that interacted with each other in the case described here, which was configured by a thermostructural analysis simulating a more consistent reality and results. This was possible due to ANSYS' Workbench capability of adding stresses caused by the mechanical requests obtained in the structural analysis, besides the working temperature and stresses induced inside the part due to high temperatures, as well as probable deformations. Through this, possible failures were identified and corrected. This is another benefit of the finite element method, as it has drastically reduced the expenditure of time and resources with prototyping.

To perform the structural analysis, it was necessary to find the pressure on the brake line that activates the brake caliper plungers, being responsible for pressing the brake pads against the disc. Table 2 shows the obtaining of brake line pressure.

Table 2. Brake system pressure acquisition.

Calculation of system pressure, (MPa)	
$P = \frac{R_b}{A_e}$	$R_b = F_{in} \cdot A_m$
P – Line pressure, MPa	F_{in} – Pedal entry force, N
R_b – Cylinder entry force, N	A_m – Mechanical pedal amplification
A_e – Embolus area, m ²	

After finding the pressure, it was possible to obtain through Eq. 4 the force applied by the brake caliper due to pressure.

$$F_s = n \cdot A \cdot P, \quad (4)$$

Where F_s is the output force, n is the number of brake calipers, A is the area of the plungers and P is the pressure.

Therefore, the data needed to perform the numerical simulation were complete and, to perform it, an assembly was made with the brake system components for the correct application of the boundary conditions, that is, to simulate more closely to reality the parts working conditions, as shown in Figure 4.

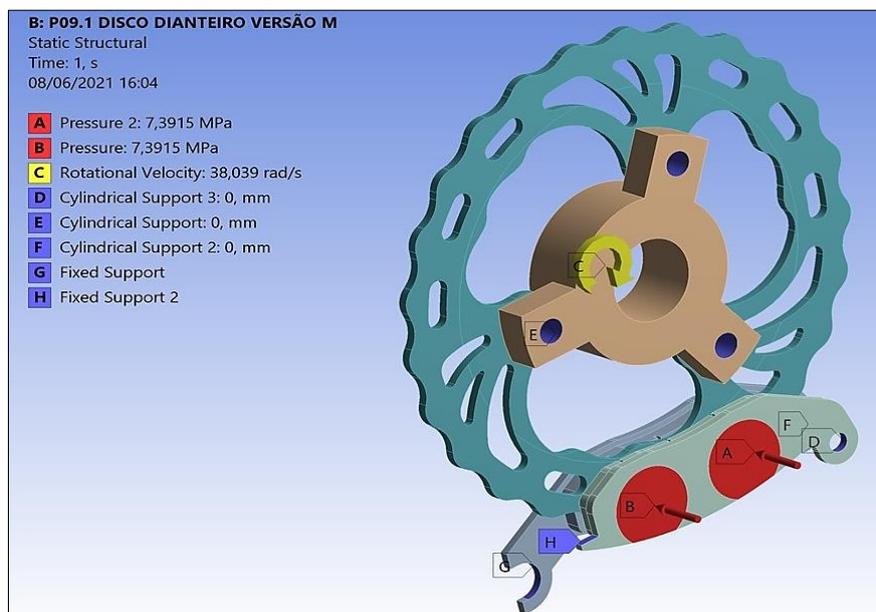


Figure 4. Structural analysis contour conditions.

4. RESULTS

At this point, it is important to emphasize that to find the final results it was first necessary to achieve mesh convergence through the correct choice of finite element type, since this affects directly convergence.

For the mesh of the brake components the quadratic tetrahedral type element was selected, which has additional nodes in the middle of each edge, making the precision of the element higher compared to the normal tetrahedral and helping to achieve faster mesh convergence, in addition to also better fit the different types of geometry. To reach convergence, a results iteration process was necessary, where the amount of total elements present in the mesh was changed to the point where the addition of more elements was no longer necessary because the results did not change.

After getting the convergence, the analyses were replicated in different pre-selected and commonly used materials for this type of application. Table 3 shows the results of the temperatures reached for each pre-selected material.

Table 3. Maximum working temperatures achieved for each material.

Material	Temperature, (°C)
CARBON STEEL SAE 1020	122,68
CARBON STEEL SAE 1045	122,96
STAINLESS STEEL AISI 304	130,41
STAINLESS STEEL AISI 420	127,48
GRAY CAST IRON	125,78

The stresses of structural analysis for each material, already with the temperatures in action, can be seen in Table 4, where it is possible to verify that, among steels, the variation of stresses is not high, but between steels and cast iron there is a considerable variation.

Table 4. Stresses obtained by thermostructural analysis for each pre-selected material.

Material	Stress, (MPa)
CARBON STEEL SAE 1020	160,89
CARBON STEEL SAE 1045	167,52
STAINLESS STEEL AISI 304	170,55
STAINLESS STEEL AISI 420	171,72
GRAY CAST IRON	135,34

This fact occurs by the difference in the magnitude of the resistive properties between them, being the most evident difference in the elasticity module (Eq. 5), which is the proportionality relation between stress and strain of the linear regime of the stress-strain curve governed by Hooke's law, where, as explained in Beer *et al.* (2011), is said that the higher the elasticity module, the greater the force required to cause deformation.

$$E = \frac{\sigma}{\varepsilon}, \quad (5)$$

Where, E is the elasticity module, σ is the uniaxial stress and ε is the deformation.

It is important to highlight that the deformations were not added to the results, since their magnitude is of 10^{-2} mm and can be disregarded because they do not cause significant effects.

Another aspect that is important to be mentioned, and is evidenced in Figure 5, is the results difference between a thermostructural analysis that considers temperature and a structural analysis that does not consider it. It is possible to observe the high contribution of this in the results and the importance of multiphysical analyses, because the fact that the variables of interest interact with each other makes it possible to achieve more accurate results that represent reality more intelligently with extremely low percentages of error.

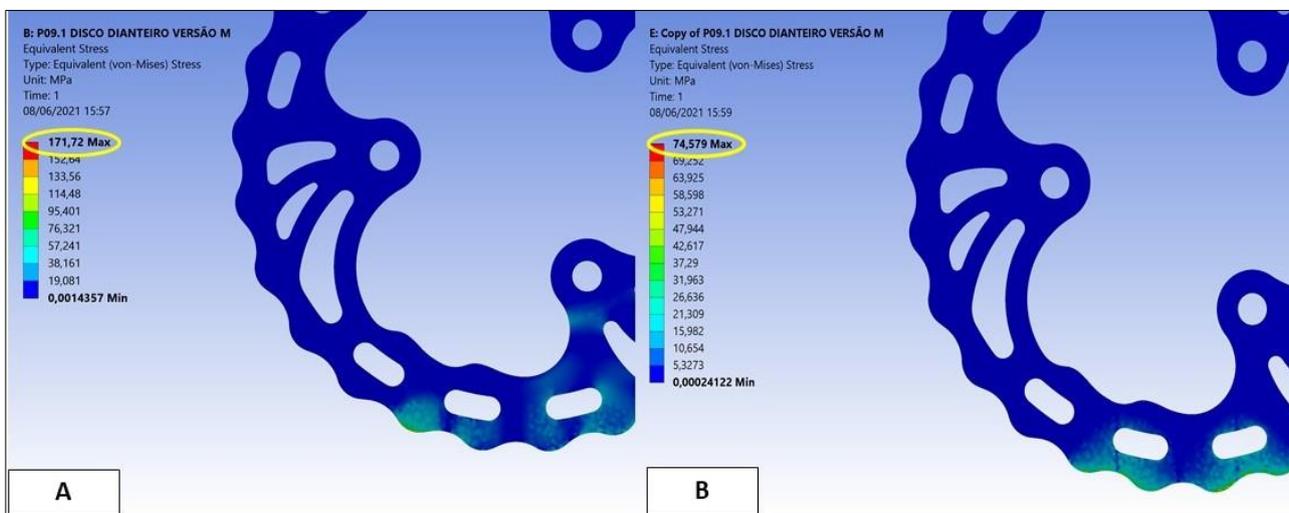


Figure 5. Image A shows the maximum stress result of one of the multiphysics analyses and image B, the result of the analysis without considering the temperature.

4.1 MATERIAL SELECTION

For the ideal material selection, criteria were defined according to the objectives established for the new UFPBaja's prototype, aiming at reducing overall mass to improve performance and ensure pilot safety.

As the materials are consisted of different alloying elements, they have different permissible working temperatures, however, none of the pre-selected materials presented temperatures close to the permissible working temperature. This fact alone does not allow selecting a material, but makes it clear that there is no risk of failure by fading into any material.

Another criterion was established by Design Failure Mode and Effect Analysis (DFMEA). According to the FMEA Ford Handbook (2011), DFMEA is a tool used to help to analyze possible flaws and their causes during the design of a component so, these possible defects are treated and corrected by reformulating the project. In addition, through the analysis of failures and their causes together with its severity, DFMEA allows to establish a minimum safety coefficient (SC) value to be adopted in each application.

The safety coefficient is the relationship between the material yield limit and the stresses in the part (obtained through the analyses) and is used as a measure to ensure that the projected part can withstand the stresses both in the working conditions, as well as presenting a margin to resist, without irreversibly deforming, unwanted events that could not be predicted.

After analyzing the possible failures and their causes, the DFMEA method provided the minimum safety coefficient of 1.7 for this application, i.e., the material must have a yield limit of 1.7 times the active stress. Table 5 shows the flow limit (FL) values of each material, safety coefficient (SC) and model mass with each material compared to the Original Equipment Manufacturer (OEM) model.

Table 5. Yield limit and safety coefficient values for each material.

MODEL	FL, (MPa)	STRESS, (MPa)	MASS, (g)	SC
OEM (STAINLESS STEEL AISI 304)	207	135,2	400	1,5311
CARBON STEEL SAE 1020	350	160,89	195,08	2,1753
CARBON STEEL SAE 1045	530	167,52	194,59	3,1639
STAINLESS STEEL AISI 304	215	170,55	198,31	1,2607
STAINLESS STEEL AISI 420	345	171,72	195	2,0091
GRAY CAST IRON	265	135,344	178,48	1,9566

From the data in Table 5, it was possible to notice that stainless steel AISI 304 should be discarded because it does not meet the 1.7 minimum value of the safety coefficient, unlike the others. But it is not always ideal to choose the material that has the highest SC value, as this can cause problems such as oversizing and high mass. As the compared materials densities are very similar, this problem is solved, since there would be no relevant differences in the piece final mass. With this, it was necessary to analyze other aspects for the selection. Among these aspects, the availability and supply in the local market as well as the price of each material were observed.

Carbon steel SAE 1045 has the highest SC, but its price is quite high compared to other materials that also meet the DFMEA criteria. Thus, because it has a similar mass and a lower price, carbon steel SAE 1020 was identified as being ideal for this application. With this choice and with the geometric patterns defined, it was possible to achieve a mass reduction of more than 50% compared to the model used previously. As can be seen in Figure 6, it is possible to perceive the difference between the mass reliefs of the disc proposed in this work and the original disc. In order not to affect the resistive braking forces, the greatest reliefs were made outside the area swept by the insert and, as previously stated. Together with this, there was a reduction in thickness compared to the original part, from 3.5 mm to 3.0 mm, which was possible after reaching the results described in Table 5 that meet the stipulated SC, thus defining the final geometry for this part.

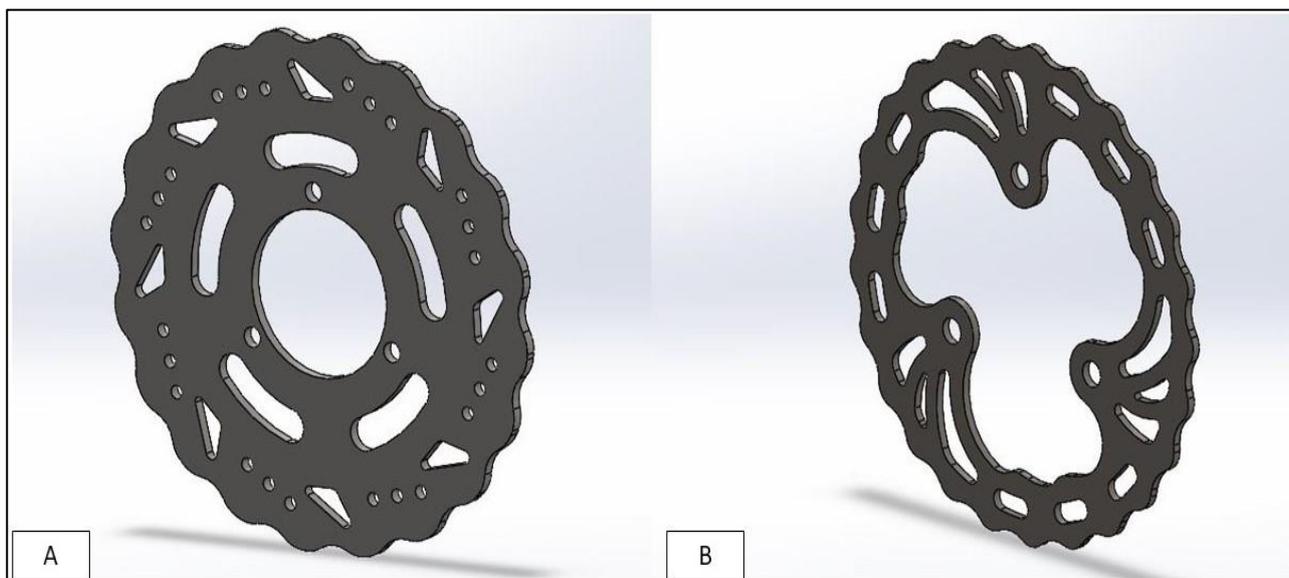


Figure 6. Image A shows the commercial disc and image B the proposed model.

5. CONCLUSION

Through the methodology described in this paper, putting into practice UFPBaja team the continuous improvement plan, it was possible to achieve the desired goals and objectives for this application. As seen in the results section, the strategies adopted provided the dimensioning of a new piece with just over 50% mass reduction compared to the commercial brake disc, where it was aimed not only reducing the mass, but also helping in heat dissipation, a crucial factor to ensure the efficiency of the system.

These results were possible thanks to the finite element method capability to combine the classical methodology of mathematical treatment with problems of complex geometries, discretizing the continuous bodies into finite elements, with low percentiles of error in the results, of course with the contour conditions applied in the correct way. This fact can be proven by comparing the results of the thermal analysis of the commercial brake disc with the image, shown in Figure 7, of a thermal camera made after a battery of tests using the part:

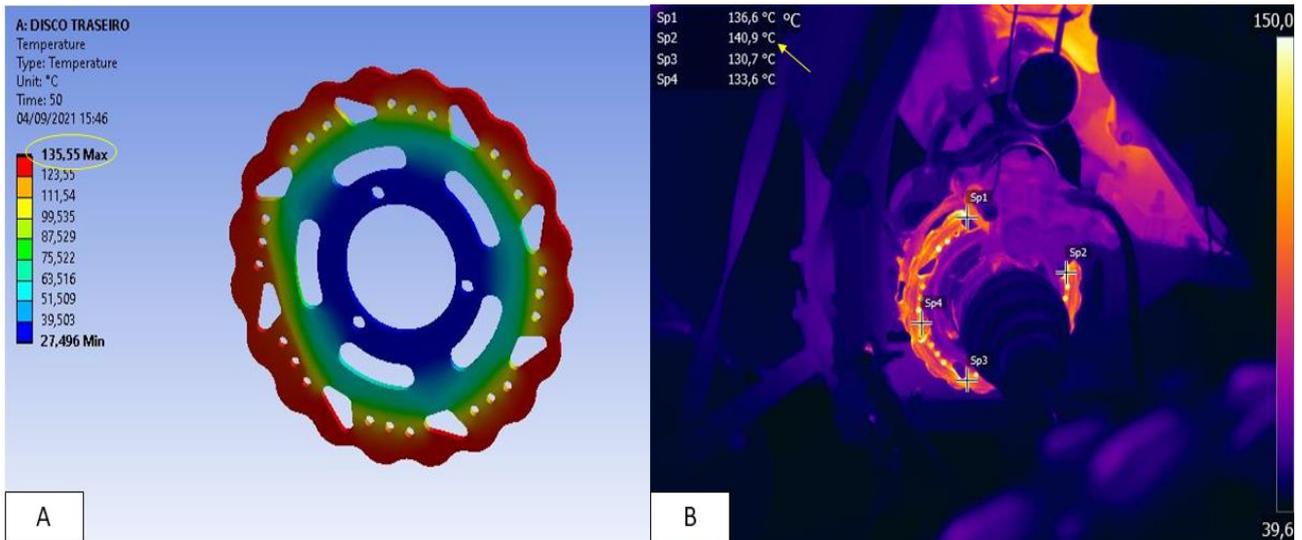


Figure 7. In image A: numerical simulation, in image B: temperatures obtained by the thermal camera.

It is possible to perceive through the above comparison that the difference between the maximum temperatures of the simulation and the thermal camera is only 3.79%, proving the efficiency of the simulation method and model. With this, it is evident the importance of using the FEM approach as part of the sizing procedure of new components and products, with extremely accurate results, significant prototyping costs reduction and thus decreasing the time required for the project completion. All of this is responsible for ensuring reliability and safety, a crucial point in the dynamics of the current market with the interval of new products launching increasingly smaller.

6. REFERENCES

- Alves Filho, A., 2018. Finite Elements: the basis of CAE Technology (in Portuguese). Saraiva Educação SA, São Paulo.
- Beer, F. P., Johnston, E. R., Dewolf, J. T., and Mazurek, D. F., 2011. Statics and mechanics of materials. McGraw-Hill Companies, Inc.
- Bergman, T. L., Incropera, F. P., DeWitt, D. P., and Lavine, A. S., 2011. Fundamentals of heat and mass transfer. John Wiley & Sons.
- Ford Design Institute. 2011. FMEA Ford Handbook Version 4.2. Dearborn.
- Limpert, R., 2011. Brake design and safety. SAE. Warrendale.
- Çengel, Y. A., Boles, M. A., and Buesa, I. A., 2006. Thermodynamics (in Portuguese). São Paulo: McGraw-Hill.
- Riley, W. F., Sturges, L. D., and Morris, D. H. 2003. Mechanics of Materials (in Portuguese). 5. ed. Rio de Janeiro. LTC.

7. RESPONSIBILITY NOTICE

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