TECHNICAL UNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics, Mechanics, and Engineering
Vol. 68, Issue III, September, 2025

THERMAL TREATMENT OF THE BRAKING SYSTEM: SIMULATION AND MODELING IN ABAQUS

Mihaela Violeta MUNTEANU, Alexandru ȚILEA

Abstract: *This paper investigates the influence of heat treatments on the performance of the braking system, using numerical simulation methods performed in Abaqus. Through a coupled thermomechanical analysis, the temperature and stress distributions in the brake components subjected to intense operating conditions were studied. The developed model allowed the evaluation of the behavior of the heat-treated materials, revealing a reduction in plastic deformations and a more efficient thermal dissipation. The study contributes to the optimization of the geometry and materials used in the braking system components, proposing solutions for increasing safety and durability in automotive and industrial applications.*
Key words: *braking system; heat treatment; Abaqus; thermomechanical analysis; finite element simulation.*

1. INTRODUCTION

Vehicle braking systems are subject to extreme stress, especially under conditions of intense or repeated braking, when kinetic energy is rapidly converted into heat by friction between the brake disc and the brake pads. This conversion generates a sudden increase in temperature in the contact area, directly influencing the mechanical and the thermal behavior of components involved. The brake discs, usually made of cast iron or special alloys, experience rapid thermal variations, and the cooling process by convection and radiation does not always manage to eliminate residual thermal stress. These stress, accumulated over time, can lead to plastic deformations, cracks and, ultimately, to compromising operational safety. At the same time, brake pads made of composite materials must withstand high temperatures, mechanical wear and thermal degradation phenomena, while maintaining a constant friction coefficient [1, 2]. Components, such as calipers or drums, involved in the generation and transmission of braking force, can also suffer mechanical and thermal degradation in the absence of optimized design and appropriate heat treatments [3 - 6].

Due to increasingly stringent requirements for road safety, performance and durability, the analysis of the thermomechanical behavior of brake systems has become an essential direction in engineering research. Heat treatments applied to components can significantly improve mechanical properties, dimensional stability and wear resistance, contributing to the prevention of structural defects and increasing the life of the system [4, 7 - 9].

In this context, numerical simulation methods based on the finite element method, such as those implemented in Abaqus, allow the investigation of the mechanical and the thermal behavior of the brake under realistic operating conditions, reducing the dependence on experimental testing.

The present work aims to evaluate the influence of heat treatment on the performance of the braking system, through a detailed numerical analysis. Through thermomechanical modeling, the behavior of materials and the distribution of stress during heating and cooling processes are analyzed, with the aim of highlighting effective solutions to increase the reliability and safety of the braking system in operation.

Received: 30.07.25; **Similarities**: 01.09.25: **Reviewed**: 04.09./27.08.25: **Accepted**:10.09.25.

2. THEORETICAL CONTEXT

With the evolution of the automotive industry and the increase in safety standards, the optimization of braking systems and the thermal treatments applied to them has become essential. Finite element simulation tools, such as Abaqus, allow engineers to analyze and improve the performance of braking systems virtually [10 – 12]. These tools provide detailed information about thermal and mechanical behavior under various operating conditions, reducing the need for expensive experimental tests.

The efficient operation of a brake system involves the rapid transformation of kinetic energy into thermal energy, generated by friction between the brake pads and the disc [5, 9]. This conversion induces intense thermal stress, which can compromise the integrity of the components if not carefully managed. Brake discs, most made of cast iron, must withstand repeated thermal cycles and stresses generated by temperature differences between the external and internal surfaces. Without adequate heat treatment, these stresses can lead to cracks, permanent deformations or even failure of the part.

The thermophysical properties of the material, such as thermal conductivity, specific heat capacity and thermal expansion, play an essential role in the behavior of components during braking [13]. Also, the coefficient of friction between the contacting materials directly influences the amount of heat generated. In real processes, this coefficient is not constant, but varies depending on temperature, pressure and relative speed [14 – 19].

Heat treatments such as quenching, tempering, normalizing or annealing are essential to improve the mechanical and structural properties of components. These processes modify the internal structure of the material, giving it greater resistance to mechanical and thermal stresses, as well as superior dimensional stability [7, 20]. The choice of the right treatment depends on the operating conditions and the desired thermal behavior for each part.

The analysis of the behavior of materials subjected to thermomechanical stress is based on concepts such as thermoelasticity and plasticity. The constitutive models used in the simulation must correctly reflect the interaction between temperature, stress and deformation. In particular, the material hardening curve and the type of plastic behavior (isotropic, kinematic or combined) influence how the material reacts under rapid heating and cooling conditions. At the same time, yield criteria, such as Von Mises, are used to determine when the material enters the plastic domain [21 – 24].

In numerical simulations, heat transfer is modeled by conduction, convection and radiation, each of which has a distinct role in overall thermal behavior. The correct definition of the contact conditions and the heat flux generated by friction is essential to obtain realistic results [25, 26]. Simulation models thus become an essential tool for the analysis and optimization of braking systems, allowing the evaluation of thermal and mechanical behavior under various scenarios, without the need for extensive physical testing [27 – 30].

3. METHODOLOGY. DESCRIPTION OF THE SIMULATION MODEL

For the analysis of the thermomechanical behavior of the braking system, the Abaqus software, based on the finite element method (FEM), was used [20 – 22, 31]. The purpose of the simulation was to investigate the temperature and stress distribution during the braking process, as well as the influence of heat treatment on the behavior of the materials. The numerical modeling included the geometry of the main components which are involved in the process of brake: the disc, the brake pads, the caliper and the mounting bracket, fig. 1 [32 – 34]. The materials were defined with specific properties, of which, for the disc, temperature-dependent values were introduced for the thermal conductivity, the heat capacity and the linear expansion. For the other parts, constant values, relevant to the simulated working conditions, were used [21, 35].

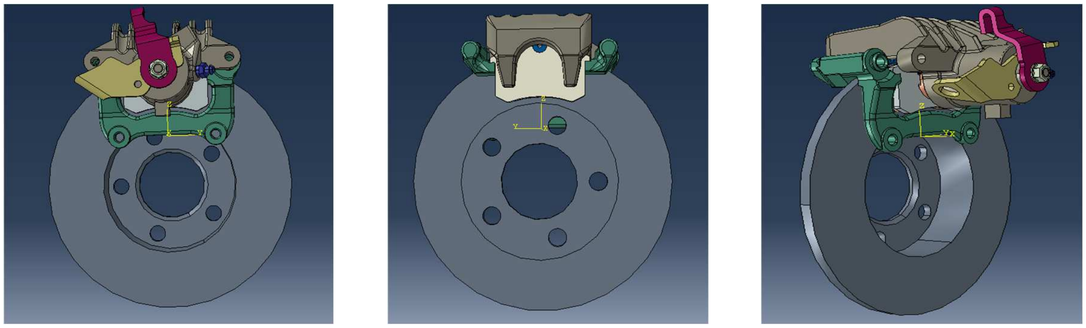

Fig. 1. Brake disc - caliper assembly, configuration and structural geometry

The contact conditions between the pads and the disc were modeled with temperature-dependent friction, which allowed a more faithful representation of the heat generation phenomenon. Thus, the following were established:

- **for the brake pad**: configuration of the thermal conductivity of the material, set to an isotropic value of 32 W/mK; density of the material for the simulation, set to a uniform value of 7850 kg/m³; Configuration of the elastic properties, including Young's modulus of 206 GPa and Poisson's ratio of 0.3; Definition of the isotropic thermal expansion coefficient at 1.1e-5 1/K; configuration of the specific heat of the material "back pad - material" with a value of 595 J/(kg·K), defined at constant volume.

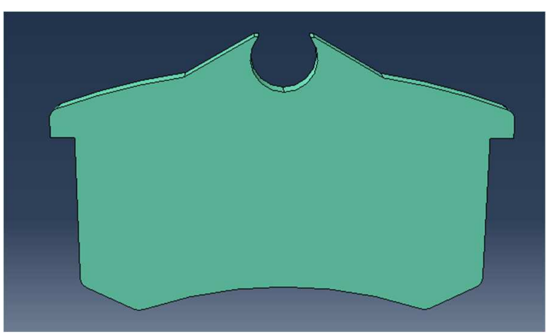

Fig. 2. CAD model of the brake pad in visual section

Fig. 2. illustrates a brake pad, prepared for thermal and mechanical simulation. The geometric contour is used to analyze the heat distribution and stress during operation.

- **For the disc**, fig. 3: Setting the thermal conductivity of the material "Cast Iron UNI 5330 Gh190" with temperature-dependent values; Setting the uniform density of 7200 kg/m³ for the disc material; Setting the Young's modulus and ratio; Defining the thermal expansion coefficient for the disc material (9.67e-7 1/K); Setting the temperature-dependent specific heat, between 470 and 775 J/(kg·K).

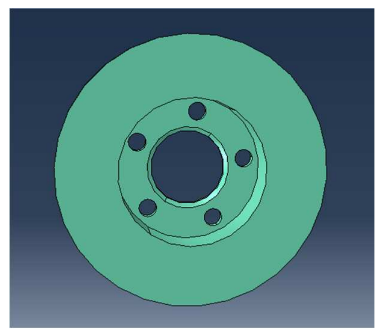

Fig. 3. Brake disc with assigned material highlighted (Disc-Material)

- **For pad material**: Set thermal conductivity for the "Pad-Material," with an isotropic value of 2.06 W/mK; Set uniform density of 26.20 kg/m³ for the pad material; Set Young's modulus (40 GPa) and Poisson's ratio (0.29) for the pad material; Set thermal expansion coefficient for the pad material, with a value of 1.43e-5 1/K; Define specific heat at constant volume for the pad material, with a value of 595 J/(kg·K), see fig. 4.

The heat produced by friction was applied in the form of surface heat flux, and the thermal dissipation was modeled through conduction, convection and radiation mechanisms.

In order to capture the behavior of materials subjected to repeated thermal stress, plasticity models were introduced that include isotropic, kinematic and combined hardening. The constitutive equations were implemented by correlating Hooke's law with the von Mises yield criterion, ensuring a correct description of the transition from the elastic to the plastic domain.

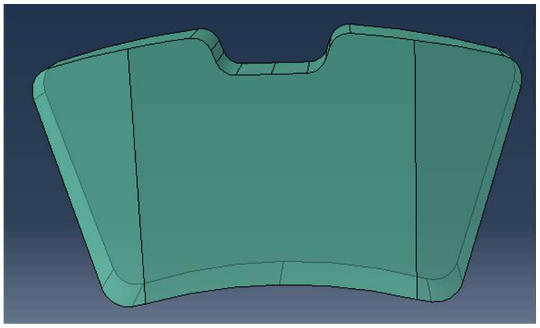

Fig. 4. Brake pad with assigned material and discretized in a finite element mesh

The simulation was carried out in a transient regime, analyzing the evolution of temperature and stresses during a braking cycle. The domain discretization was performed through a finite element mesh adapted to the geometric complexity, aiming at a balance between the accuracy of the results and the calculation time.

This approach allowed the evaluation of the real behavior of components subjected to complex stresses, allowing virtual testing of different operating scenarios and the effects of heat treatment, without the need for physical prototypes or destructive testing.

4. RESULTS

The analysis consists of a nonlinear static simulation sequence, followed by a complex eigenvalue evaluation, used to identify dynamic instabilities associated with the phenomenon of unwanted noise in the braking system. These instabilities can be influenced by factors such as assembly forces, non-uniform distribution of contact pressure between the brake pads and the disc, variation of the friction coefficient as a function of speed, temperature and pressure, as well as the damping generated by friction and the progressive wear of the friction material.

The operating principle of a disc brake system consists in slowing down the vehicle by pressing the brake pads against the rotating wheel discs. The friction between the pad linings and the disc dissipates the kinetic energy of the vehicle, resulting in deceleration. However, during brake operation, the friction between the lining and the disc, in the system, can induce dynamic instabilities, which can generate "brake squeal", the unwanted noise.

The elastic stresses are higher on the inner face of the rotor (154 MPa), in the case of the ventilated disc model for the rear axle, compared to the outer face (99.5 MPa). However, the von Mises elastic stresses that have the maximum values are found in the area of the rotor neck (273 MPa) and near the inner radius of the long blade thread (442 MPa). These stress concentrations are generated by the restrictions imposed on the free expansion of the rotor friction surfaces [2].

Thus, it is important to develop a more realistic inelastic material model that integrates distinct yield surfaces for tension and compression states. Such a model would allow a more accurate assessment of the brake disc behavior under thermal and mechanical loading conditions, contributing to the improvement of its design and performance.

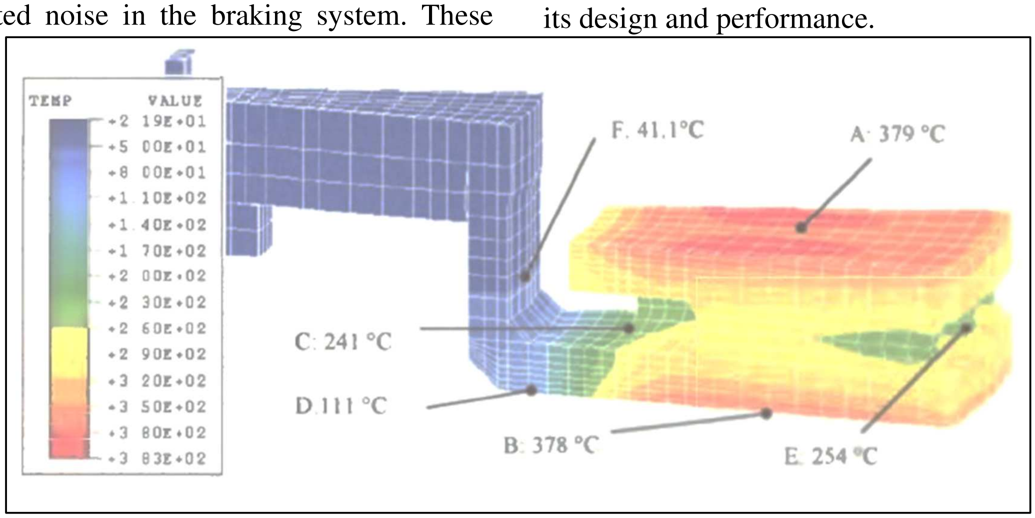

Fig. 5. Temperature for the rear ventilated disc at the end of brake application.

4.1 Temperature distribution in brake system components

The numerical simulations performed revealed a significant temperature variation within the brake system components during an intense braking cycle. The brake disc recorded the highest temperatures in the contact area with the pads, where friction is maximum. This region concentrates most of the thermal energy generated.

In the case of brake discs, circumferential stresses are predominant and play an important role in plastic deformation. Detailed analysis of plastic deformations in this direction is essential to prevent premature failure and improve component durability. This approach contributes to optimizing the design and increasing the reliability of brake systems under intensive use conditions.

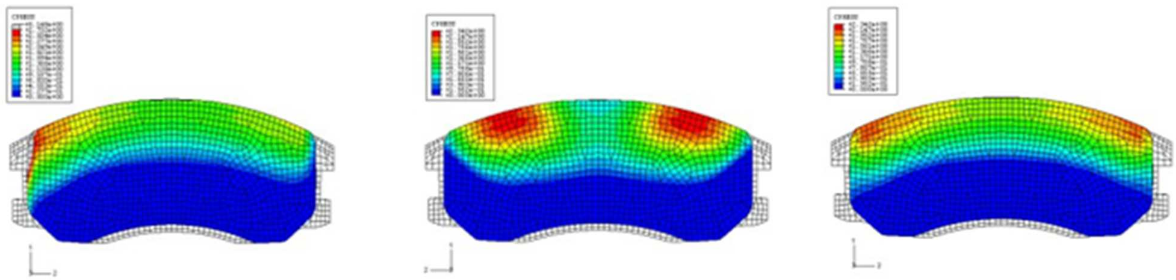

Fig. 6. Temperature distribution on the brake pad surface under different operating conditions

The temperature distribution is not uniform throughout the disc volume. The outer layers heat up quickly, while the deeper areas have a delayed thermal response, which leads to the appearance of significant thermal gradients. These temperature differences can generate internal stresses and contribute to the degradation of the material over time.

The brake pads, in direct contact with the disc, take up part of the heat generated, but their thermal dissipation capacity is limited by the properties of the composites used. A progressive increase in temperature is observed inside them, with an accumulation of heat in the layers close to the friction zone.

Auxiliary metal components, such as the caliper and the mounting bracket, register lower temperatures due to their distance from the active braking zone, but are still affected by thermal transfer by conduction. If cooling is not effective, these components can accumulate heat over time, affecting the durability and stability of the assembly.

4.2 Stress and strain distribution

Thermomechanical analysis performed in Abaqus allowed observing how internal stresses and strains appear and evolve during an intense braking cycle.

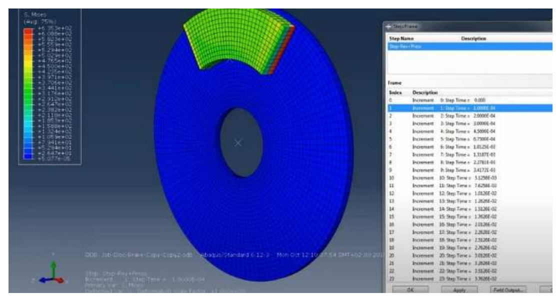

Fig. 7. Simulation results, illustrating the stress distribution in the brake disc

In fig. 7. is presented the history of element centroid temperatures during the first cooling period for the rear ventilated disc is presented.

The highest stress values were located in the contact area between the brake disc and the brake pads, where the stresses are maximum due to the combination of mechanical pressure and rapid heating (Fig. 8.). Thermal distribution in the disc with the highest temperature.

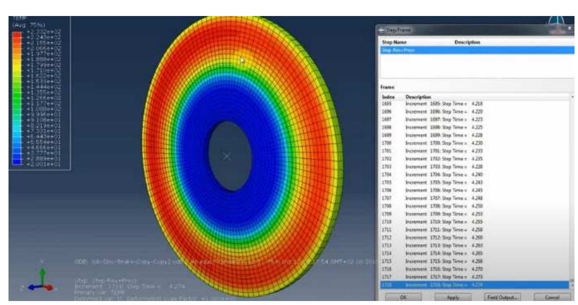

Fig. 8. Temperature distribution in the brake disc under thermal loads

The distribution of von Mises stresses indicates the existence of concentrations in the peripheral area of the disc, especially in regions where the thermal gradient is accentuated. These stresses can lead, in the absence of heat treatment, to the appearance of fatigue cracks or irreversible plastic deformations.

The total deformations were analyzed over the entire geometry of the disc. The largest displacements occur near the outer edge, where heating is faster and the stiffness decreases locally due to the high temperature. If not controlled, these deformations can affect the contact between the pads and the disc, reducing the efficiency of the braking system.

The distributions highlighted by simulation emphasize the importance of heat treatment not only for wear resistance, but also for limiting permanent deformations and maintaining the structural stability of the components under cyclic thermal stress conditions, fig. 9.

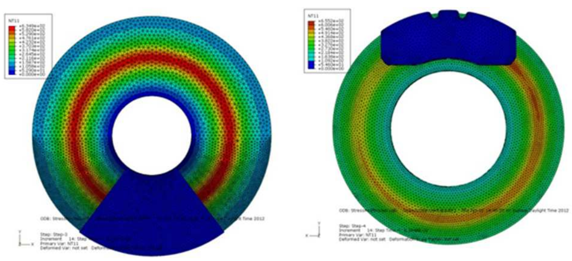

Fig. 9. Temperature distribution in the brake disc

In Fig. 10 a stress distribution is presented, in a circumferential direction:

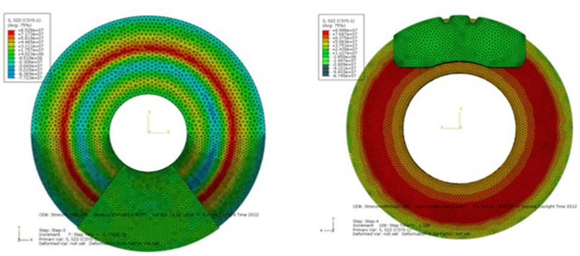

Fig. 10. Stress distribution in the brake disc

The stress from circumferential direction in a circular geometry is significantly higher than the stress from the other two directions. This causes a strong influence of circumferential stress on elastic behaviors in the other directions. However, in the project, the analysis of elastic behavior is not a major priority.

The principal plastic strains do not have a direct relationship with the corresponding principal stresses, since the plastic strain is the cumulative result of all stresses acting simultaneously. For the fatigue analysis, the objective is to reach a stabilized plastic stage. If this stage is reached, the irregularities in the graphs can be neglected.

Circumferential stress is dominant with respect to the plastic deformation in a disc-type geometry. According to previous studies and experience, the failure of the brake disc is usually initiated in the radial direction, which indicates that it is caused by the circumferential stress. Therefore, our analysis can focus on the stress and strain states in this direction.

The results obtained from the thermal stress and strain analysis are reliable and correct, respecting the principles of elasticity and plasticity. Thus, we can continue to apply the same approach used for the reference model to the real model.

5. DISCUSSIONS AND CONCLUSIONS

The results obtained from the thermomechanical simulation of the braking system confirm the importance of heat treatment in increasing the durability and performance of components subjected to intense stress. The temperature and stress distributions highlight the fact that, in the absence of appropriate treatment, the brake disc is prone to residual stress accumulations, which can lead to plastic deformations, the appearance of cracks and, ultimately, to material failure.

The heat treatments applied to the disc contribute to stabilizing the mechanical behavior in the cyclic heating-cooling regime, by homogenizing the internal structure and reducing the sensitivity to thermal gradients. Compared to untreated materials, the heat-treated variants showed a visible reduction in maximum stress and a more uniform distribution of the deformation field.

Although the simulation was performed without complementary experimental data, the qualitative analysis of the obtained figures indicates good coherence of the model. This suggests that the proposed method can be used as a basis for further developments, such as optimizing the disc geometry, testing other materials, or analyzing the behavior under repeated braking. In conclusion, heat treatment is an essential factor in the design of reliable and safe braking systems. Numerical modeling by the finite element method, applied rigorously, can be a valuable tool for evaluating system performance and guiding engineering decisions in the design stage.

6. REFERENCES

[1]. Frățilă Gh, Mărculescu Gh, *Sistemele de frânare ale autovehiculelor* 1986 (Bucharest: Technical Publishing House)

[2]. Saiprasit Koctniyom, *Thermal Stress Analysis of Automotive Disc Brakes,* 2000, The University of Leeds School of Mechanical Engineering

[3]. R. Ş. Ionescu, A. Chiru, A. Roșescu, *Sistemele de frânare ale automobilelor. Situația actuală și perspective*, Buletinul AGIR nr. 1/2014, ianuarie-martie,pp.19-24, http://www.agir.ro/buletine/1953.pdf

[4]. A.L.Crăciun, *Evolution of materials for motor vehicles brake discs*, International Journal of Engineering, Tom XIII, Fas.3, p. 149-154, 2015

[5]. Șt. Voloancă, Ghe. Frățilă, *Cercetări privind solicitările termice ale mecanismului de frânare și influența lor asupra siguranței circulației*, http://www.agir.ro/buletine/2080.pdf

[6]. Sadd M., *Elasticity Theory, Applications and Numerics*, Elsevier Butterworth-Heinemann, 2005

[7]. Grober H., Erk S., Grigull U., 1961, *Fundamentals of Heat Transfer*, 3rd Ed., McGraw-Hill, New York, NY

[8]. Marcel Roşca, Alin Casian Blaga, *Termotehnică*, - Oradea : Editura Universităţii din Oradea, 2008 Bibliogr. ISBN 978-973-759-479-2

[9]. MOLEA Andreia, MERIE Violeta Valentina, SUCIU Ramona Crina, *Materiale utilizate în construcția și întreținerea autovehiculelor* UTPRESS Cluj-Napoca, 2023 ISBN 978-606-737-681-4

[10]. Gh.Badarau, *Proprietatile materialelor metalice*, 2007, Note de curs

[11]. Ștefan VOLOACĂ1, Gheorghe FRĂȚILĂ2, *Cercetări Privind Solicitările Termice Ale Mecanismului De Frânare Şi Influenţa Lor Asupra Siguranței Circulației,* Universitatea „Politehnica" din București

[12]. VALERIA SUCIU, MARCEL VALERIU SUCIU, *Studiul Materialelor* Editura FAIR PARTNERS Bucuresti

[13]. MIRCEA RADEŞ - *Tensiuni Termice*, Editura Printech 2010

[14]. Ion Grindei, *Termoelasticitate*, Editura Didactică și Pedagocică, 1967

[15]. Incropera, F.P., DeWitt, D.P., Bergman, T.L., Lavine, A.S. *Fundamentals of Heat and Mass Transfer*. 7th ed., Wiley, 2011.

[16]. Norton, R.L. *Machine Design: An Integrated Approach*. 4th ed., Prentice Hall, 2010.

[17]. Ozisik, M.N. *Heat Conduction*. 2nd ed., John Wiley & Sons, 1993.

[18]. Talati, F., Jalalifar, S. *"Analysis of Heat Generation in a Disc Brake System."* Heat and Mass Transfer, vol. 42, no. 5, 2006, pp. 365–373. DOI: 10.1007/s00231-005-0651-y.

[19]. Yang, F., Liu, Y., and Jin, J. "*Thermo-mechanical Coupling Analysis of Brake Discs Based on Finite Element Method."* Applied Thermal Engineering, vol. 80, 2015, pp. 140–148. DOI: 10.1016/j.applthermaleng.2015.01.067.

[20]. Smith, J. "*Thermal Analysis of Brake Disc Using Abaqus."* Simulation & Modeling in Mechanical Engineering, 2020, www.mechanicalengineeringjournal.com/article/thermal-analysis-abaqus.

[21]. *Abaqus Documentation, Dassault Systèmes. "Abaqus Analysis User's Manual*." Version 6.14, 2014. Available at: https://help.3ds.com

[22]. *Dassault Systèmes. Abaqus 2021 Documentation. 2021*. Available at: https://www.3ds.com/products-services/simulia/products/abaqus/

[23]. MATLAB & Simulink, MathWorks. "*Modeling Heat Transfer in a Brake System*." Available at: https://www.mathworks.com/help/physmod/simscape/ug/brake-system-heat-transfer.html

[24]. Filippone, M., Di Bartolomeo, O., and Polonara, F. "*Thermo-Mechanical Analysis of a Brake Disc Using Finite Element Method*." Proceedings of the ASME International Mechanical Engineering Congress & Exposition, 2014.

[25]. Popescu, R. "*Modeling of Thermo-Mechanical Behavior of Brake Discs Using Finite Element Method."* PhD Thesis, Technical University of Bucharest, 2018.

[26]. N. Strömberg (2011), *An Eulerian Approach For Simulating Frictional Heating Disc-Pad Systems,* European Journal of Mechanics A/Solids, 30 673-683

[27]. David L. McDowell, Rod Ellis (1993), *Advances In Multiaxial Fatigue,* Ann Arbor

[28]. I. St Doltsinis (2000), Elements Of Plasticity: Theory And Computation, WIT press

[29]. G. T. Houlsby, A. M. Puzrin (2006), *Principles Of Hyper-plasticity: An Approach To Plasticity Theory Based On Thermodynamic Principles,* Springer

[30]. H.D. Solomon, G.R. Halford, L.R. Kaisand (1988), *Low Cycle Fatigue*, ASTM STP 942, ASTM, Philadelphia, pp. 209–235.

[31]. Sidney Yip (2005), *Handbook of Materials Modeling,* Part A, Springer

[32]. O. A. Bauchau, James I. Craig (2009), *Structural Analysis: With Applications To Aerospace*, Springer

[33]. N. Strömberg (2011), *Nonlinear FEA and Design Optimization for Mechanical Engineers*, Jönköping University

[34]. ABAQUS VERSION 6.9 (2012), *Documentation*

[35]. Brian Derby, David A. Hills, Carlos Ruiz (1992), *Materials For engineering: A Fundamental Design Approach*, Longmann

[36]. R. W. Lewis, Ken Morgan, H. R. Thomas (1996), *The Finite Element Method in Heat Transfer Analysis,* Chichester: Willey

TRATAMENTUL TERMIC AL SISTEMULUI DE FRÂNARE: SIMULARE ȘI MODELARE ÎN ABAQUS

Rezumat: Lucrarea de față investighează influența tratamentelor termice asupra performanței sistemului de frânare, utilizând metode de simulare numerică realizate în Abaqus. Printr-o analiză termomecanică cuplată, s-au studiat distribuțiile de temperatură și tensiuni în componentele de frână supuse unor condiții de funcționare intense. Modelul dezvoltat a permis evaluarea comportamentului materialelor tratate termic, relevând o reducere a deformărilor plastice și o disipare termică mai eficientă. Studiul contribuie la optimizarea geometriei și a materialelor utilizate în componentele sistemului de frânare, propunând soluții pentru creșterea siguranței și durabilității în aplicații auto și industriale.

Mihaela Violeta MUNTEANU, Assoc. Prof., TRANSILVANIA University of Brasov, Department of Mechanical Engineering, mail v.munteanu@unitbv.ro

Alexandru ȚILEA, Eng., TRANSILVANIA University of Brasov, Department of Mechanical Engineering