https://doi.org/10.3311/PPtr.36938 | 1

Periodica Polytechnica Transportation Engineering, 53(1), pp. 1–6, 2025

Thermal Analysis of a Simplified Railway Brake Model with Numerical Simulation

Betti Bolló[1*], Ferenc Sarka[2], Katalin Voith[1], Csaba Felhő[3]

[1] Institute of Energy Engineering and Chemical Machinery, Faculty of Mechanical Engineering and Informatics, University of Miskolc, Egyetemváros, H-3515 Miskolc, Hungary
[2] Institute of Machine and Product Design, Faculty of Mechanical Engineering and Informatics, University of Miskolc, Egyetemváros, H-3515 Miskolc, Hungary
[3] Institute of Manufacturing Science, Faculty of Mechanical Engineering and Informatics, University of Miskolc, Egyetemváros, H-3515 Miskolc, Hungary
[*] Corresponding author, e-mail: betti.bollo@uni-miskolc.hu

Received: 22 March 2024, Accepted: 26 October 2024, Published online: 11 November 2024

Abstract

The braking system is the most important safety device in all vehicles. When braking, the kinetic energy of the vehicle is converted into thermal energy. The use of friction brakes can cause problems in the long term, as the brake can heat up, reducing its coefficient of friction. Heating and cooling have a harmful effect on all frictional elements, causing material fatigue. As a result of the negative effects, the hot brake cannot reduce the velocity of the vehicle at the correct rate. In this paper, the propagation of frictional heat is analysed in a test specimen made of cast iron material using different numerical simulations. The computational results are validated with the results obtained during measurement. With proper validation, the brake discs are able to be modelled at different designs and materials without having to manufacture a test specimen for each case. In the study, it is important to point physical quantities to carry out the simulation.

Keywords

FEA, friction heat, railway brake block, thermal analysis

1 Introduction

Braking systems are an important part of any vehicle (Limpert, 2011). Their functionality and effectiveness in rail traffic is perhaps even more important than in road traffic (Belhocine and Abdullah, 2020). Rail transport involves the joint movement of huge masses, which, due to their kinetic energy, can cause great damage in the event of an accident, both in terms of human life and material (Zeng et al., 2022). Because of this importance of railway brakes, the tests presented in the article were initiated jointly with a Belgian partner (Falex Tribology NV) within the framework of a project funded by the European Union. One of the important tasks of the project is to set up a simplified numerical simulation model that can be used to examine heat conduction in the braking system. In the literature, many researchers have investigated the heat generated in the brake disc using thermal mechanical simulation (Belhocine, 2015; Han et al., 2020; Menyhártné Baracskai, 2015) and the numerical modelling of a ventilated brake disc (Belhocine and Abdullah, 2020; Liu et al., 2024). To start the simulation, the needed initial data were created through experiments and measurements. Currently, the railway uses two types of material for brake blocks, one is cast iron, and the other is a composite material type. In Germany, only freight cars are equipped with so-called silent brake blocks (Ficzere, 2024; Tulipánt, 2007). This publication deals only with cast iron, however our research will be continued for other materials and samples made with additive manufacturing technology. To be able to draw conclusions regarding the reasons of the damage the surface was examined. The samples manufactured out of the used brake blocks were examined in a test device to measure the temperature. Using the parameters of the tests a simulation model was created that gives correlating results.

2 Examination of the friction surface of a brake block

The brake blocks press directly against the tread of the wheels of the cargo railroad cars during braking.

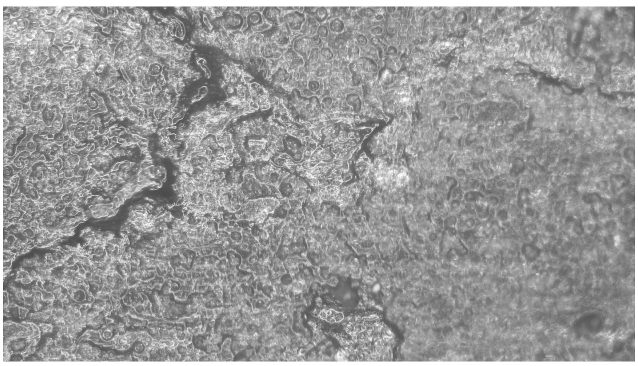
Cite this article as: Bolló, B., Sarka, F., Voith, K., Felhő, C. (2025) "Thermal Analysis of a Simplified Railway Brake Model with Numerical Simulation", Periodica Polytechnica Transportation Engineering, 53(1), pp. 1–6. https://doi.org/10.3311/PPtr.36938

The braking torque is created by the generated frictional force (Szabó and Zobory, 1998). The brake blocks and the wheel (load) of the cargo railroad cars are both made of cast iron. Several worn, used brake blocks were provided by RailCargo Hungária Ltd. Microscopic examinations were performed on the received brake blocks. The friction surface of one of the brake blocks is shown in Fig. 1.

It can be seen in Fig. 1 that the central parts of the friction surface were affected by discoloration, i.e., they were loaded by the higher temperature. The two pictures in Fig. 2 present the external part (not discolored) and central part of the friction surface of the brake block at 100x magnification (Zeiss Discovery V14). The image taken from the central part shows signs of surface melting and adhesion, while the image from the edge (external parts) shows a predominance of grooved surface areas (the grooves run parallel to the direction of movement, the cracks are perpendicular to the direction of movement).

Fig. 1 Frictional surface of a railway brake pad, with clearly visible signs of overheating (circled)

(a)



(b)

Fig. 2 Microscopic image of the friction surface of the brake block, a) external part, b) central part

These pictures also confirm that the central parts of the friction surfaces heat up to a great extent. Traces of plastic deformation are visible on the central surface parts. The pattern of some areas of the surfaces is close to a tribological oxidative wear. This kind of duality leads to the conclusion that adhesive wear and tribological oxidative wear also develops between the contacting surfaces. The thermal shock caused by the resulting high temperature is responsible for the formation of cracks. Of course, the damage caused by heat affects not only the brake blocks, but also the wheels of the wagons. Repairing them is very expensive and time-consuming. One of the causes of the resulting damage is poor heat dissipation.

The striped wear pattern visible on the outer parts of the brake block clearly shows that the wear is caused by abrasive contaminants. The abrasive contaminant can be airborne dust (SiO_2) or wear products from the brake pad itself (Affonso, 2007; Staffelini, 2015; Williams, 2005).

By avoiding the described processes, brake blocks and wagon wheels can have a longer service life. Today's economic trend is to convert the linear economic form to the circular economy form. One of the tools of the 10R principle is increasing the service life (renew/rethink) of structures and components (Borsodi and Takács 2023). According to this trend, the examination of the brake blocks and the determination of the required parameters for the numerical simulation were started.

Besides the optical microscopic examination, surface roughness tests were performed to measure the difference between the external and central parts of the surfaces. A 4 mm × 4 mm surface on both examined surface parts were chosen (Fig. 3), since they represent the best the surface structure.

On the designated surface measurements were performed in two directions. One of the directions is the longitudinal direction of the brake block (y), which is the same as the direction of the movement between the block and the wheel. The other direction is perpendicular to the previous one (x).

The structural difference between the two surfaces is also clearly visible in Fig. 3. The values of the measured surface roughness are summarized in Table 1, where R_a is arithmetic average of profile height deviations from the mean line, and R_q is root mean square average of profile height deviations from the mean line.

The values in the table also show that the surface roughness of the central part of the brake block changes

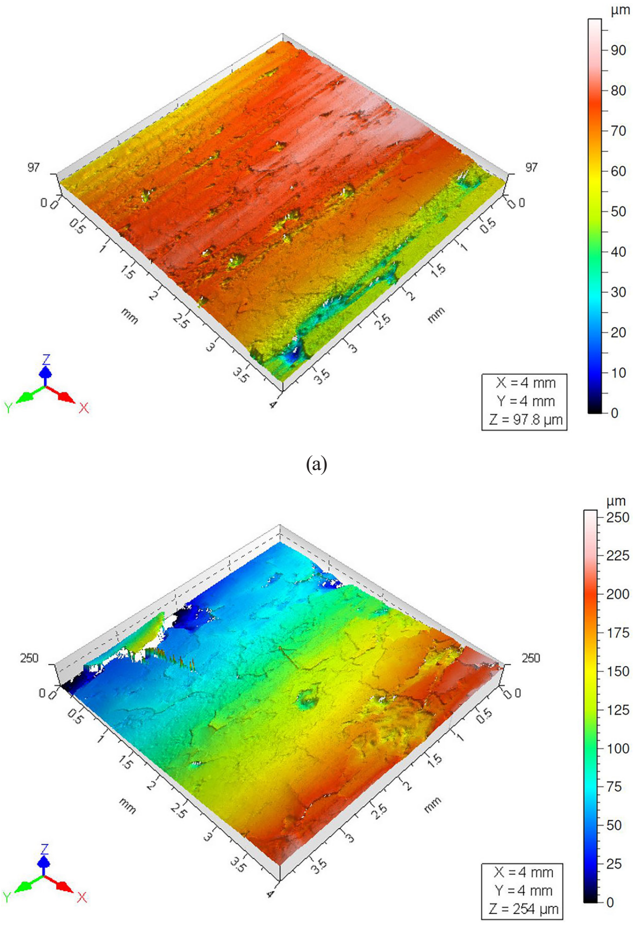


Fig. 3 The image of the selected surfaces is coloured according to height, a) the external part of the brake pad, b) the central part of the brake pad

Table 1 The measured surface roughness (R_a, R_q)

	direction *y*	direction *x*	change
External part	$R_a = 0.59$	$R_a = 1.18$	200%
	$R_q = 0.88$	$R_q = 1.68$	190%
Central part	$R_a = 2.59$	$R_a = 3.84$	148%
	$R_q = 3.8$	$R_q = 4.93$	129%

to a much lesser extent as a result of the change in the examination direction. This fact also allows a conclusion, namely that the friction and wear conditions of the middle parts differ from each other. While abrasive wear occurs on the edge, adhesive wear occurs on the inner parts. As the temperature increases, the friction coefficient of steel-iron alloys decreases (rapidly up to 200 °C, less from then on, it stabilizes above 600 °C) (Straffelini, 2015). The decreasing friction factor can be offset by increasing the clamping force to create the required frictional force. This process moves in the direction of ever-increasing temperature. By keeping the surface temperature of the brake blocks low (below 200 °C), the process can be avoided, and this kind of damage to the brake blocks is also reduced.

3 Heat conduction experiments

Test specimens were manufactured out of the used cargo railroad block received from the supporting company in order to test them in the test machine of Falex Tribology NV. The test specimens were made in two sizes and from two different materials. In this publication, only the results of the experiments carried out with cast-iron, circular specimens are briefly described. The test specimens are shown in Fig. 4.

The cylindrical specimens – called disk from hereon – have a diameter of 54 mm and a height of 15 mm. Fig. 5 shows the test equipment where the disks are tested (in this picture the disk is made of copper). The device consists of two cylindrical components clamped together with a constant load (10 lbf) while the upper element rotates at a constant speed.

The contact takes place along the circular ring formed on the front surface of the rotating upper element. The width of the ring is 4.5 mm. During the experiments, the material of the specimen changes, while the rotating counterpart is always the same (steel). The friction between the rotating and stable part generates the heat, and the temperature

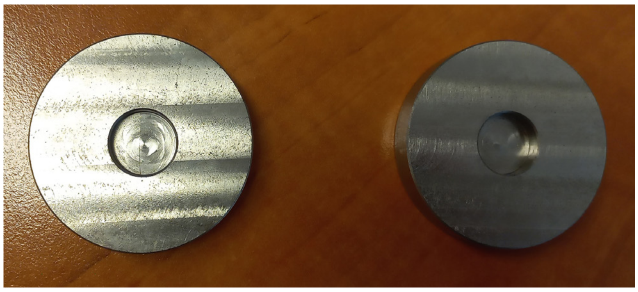


Fig. 4 The created cast-iron test specimens

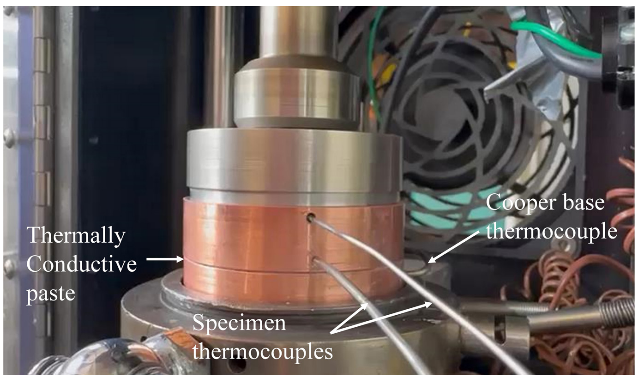


Fig. 5 Design of the test equipment, test run with a copper test specimen at Falex Tribology NV

values are measured at the top and the bottom of the specimen, and under the specimen in the base element.

One of the results can be seen in Fig. 6. The time is given on the *x*-axis in secundum. The rotation speed values are marked on the right *y*-axis, and the temperature in °C is given on the left *y*-axis.

Knowing the test results, our goal is to create a numerical simulation in which the test conditions can be modelled, especially the heat conduction behaviour of the various materials. The results achieved so far in the Ansys program (Ansys, 2023a) are presented in the next chapter.

4 Set up of the numerical simulation model

Ansys 2023 R1 version was used for the numerical simulation. In the program, several options are available for testing heat conduction. In addition to the classical Thermal (Steady State or Transient) analysis, the Mechanical environment and the few years ago introduced Coupled Field module. All three have advantages and disadvantages compared to each other. The best one would be the Coupled Field module, since by simulating the movement of the rotating element, the heat generated by the friction will be the source of the heat (Szabó and Várkuliné Szarka, 2019; Szűcs et al., 2010).

As a first step, two rings are used and the temperature distribution needs to be determined in the lower, fixed test piece. The rotor is above the specimen, the rotating speed is 600 rpm and the rings are pressed together with a force of 45 N. The simulation ran for 1 s to test the program, that the upper part really rotated, and the heat transfer took place. Fig. 7 shows the result, where a temperature of 22 °C was prescribed for the external environment and the maximum heating was 26 °C due to friction.

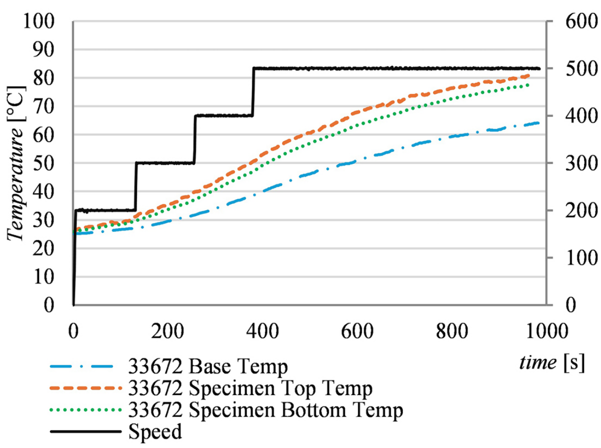


Fig. 6 The result of a test run on a cast alloy test specimen (identification number:33672)

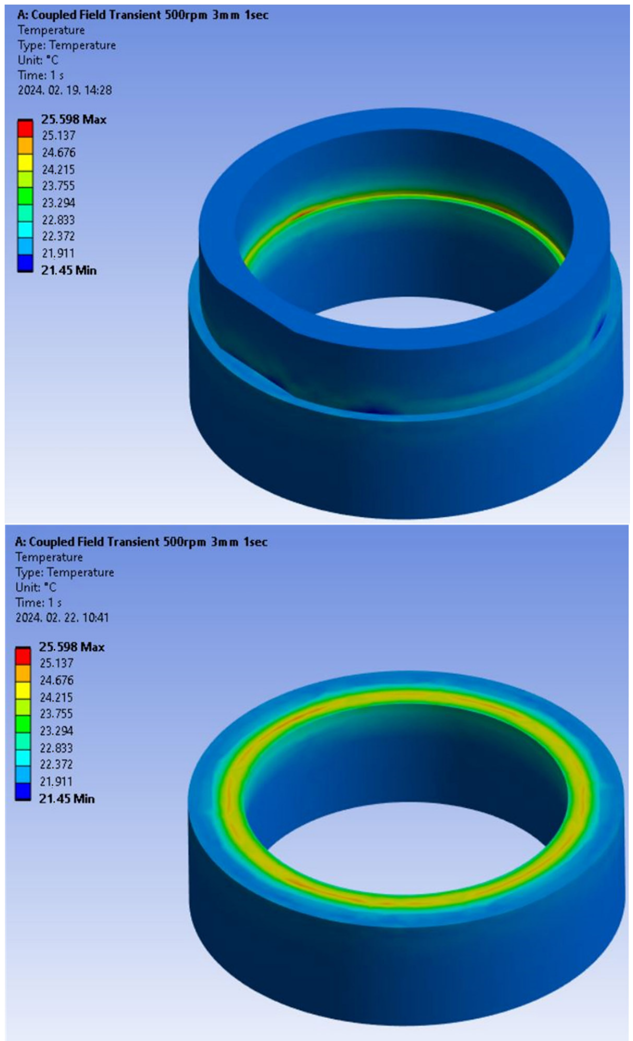


Fig. 7 The small temperature rise in Coupled Field module

The experiments started with a rotation speed of 200 rpm and increased the rotation speed by 100 rpm every 200 s until they reached 1000 s. Since our calculations has to be compared to the experimental results, we started a 100 s run. However unfortunately our computer couldn't cope with this set up, so another solution had to be found. Two different ways were decided: one of them is the thermal analysis in the Mechanical module and the other is the flow and heat transfer around the specimen, which is task of the Fluent software (Ansys, 2023a).

4.1 Thermal analysis in Ansys mechanical

A new CAD model was created for the thermal analysis. At the top of the specimen a ring shaped surface *w* as created. The measured temperatures from the experiment were applied. No simulation is totally equal with the experiment. The origan experiment lasted 1000 s (almost 17 minutes). A 14 s long period in the FEA program (Ansys, 2023b) gives almost the same thermal distribution as the experiment. In Fig. 8 you can see the new CAD model, where at the ring shaped surface (red) the constant (measured) temperature is prescribed (80.8 °C). At the bottom of the model a fixed support is used, and the temperature of the environment was 24.9 °C (this was the started temperature in the experiment).

The temperature distribution of the new model can be seen in Fig. 9. The bottom temperature of the element is about 66 °C. The small holes on the side of the body represent the place of the thermocouples in the experiment. At the bottom hole we get about 63 °C. 14 s to the simulation time was applied.

4.2 Thermal analysis in Ansys Fluent

An air domain with a radius of 40 mm was placed around the test specimen in Fig. 8. A velocity inlet for the bottom of the outer surface, and a pressure outlet boundary condition for the other surfaces was prescribed. The temperature of the ambience environment was given as 22 °C. During the measurement, the temperature of the friction surface increases continuously, so a curve was fitted to the measured values. For the numerical simulation, the friction surface was not given as a constant value, but it was prescribed by a linear function.

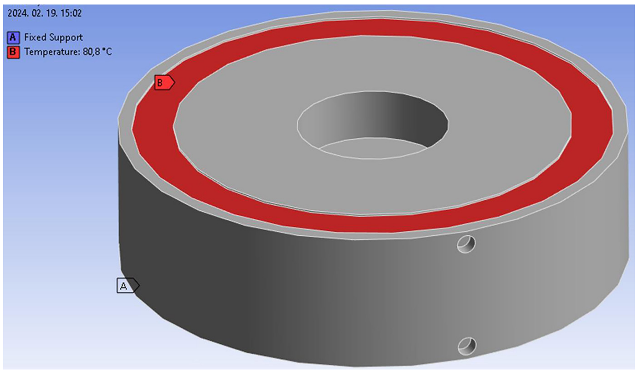


Fig. 8 The new CAD model

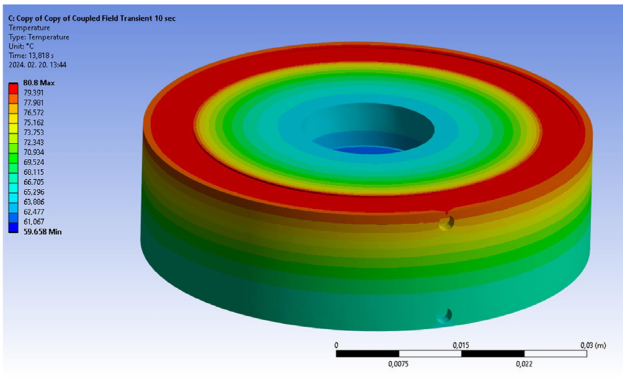


Fig. 9 The temperature distribution after 14 s

For the mechanical modelling the finest mesh consisted of approximately 200000 elements and the skewness parameter of the cells remained below 0.86. For the flow simulation a much finer mesh was needed. On the one hand the external air domain was taken into account, and on the other hand the boundary layer between the solid and the air had to be more finely meshed due to the more intense temperature changes. Therefore the mesh consisted of 1.4 million hexa and tetra elements and the skewness parameter of the cells is 0.91, which is still considered adequate.

4.3 Comparison the result of the simulation and the experiment

The results of the two simulations and the measurements are summarised in Table 2. The difference between the experimental data and the numerical simulations is shown.

It can be seen that the difference between the top temperature is insignificant. The significant difference in the bottom temperature value of the FEA is coming from the fact that there is a constant 22 °C air temperature in the model while in closed environment of the measurement, the ambient temperature is continuously increasing. The difference in the base temperature between the experiment and the Fluent is caused by the different environment: in the experiment it is metal parts at the bottom the assembly while in Fluent it is air since the goal was to create a simplified model.

The achieved results are acceptable as the initial results of the research work. Further adjustment of the finite element model is necessary so that different macrostructural changes of the specimen (printed versions) can also be investigated. Currently, the biggest limitation is the computing capacity, that hopefully will be solved successfully in the next period of the project.

Table 2 The comparison of the measurement and the two type of simulation

	Measurement [°C]	Ansys FEA [°C]	difference	Ansys Fluent [°C]	difference
Top temperature	80.8	80.8°C	0%	78.6	–3%
Bottom temperature	77.7	66 °C	–15%	77.4	–1%
Base temperature	64.1	63°C	–2%	77.3	+20%

5 Conclusions

It can be concluded that one of the most significant reasons for the failure of railway brake blocks is their overheating. Surfaces with different temperatures can be clearly separated on the friction surface of the brake blocks. The wear characteristics are also different. The microscopic and surface roughness tests also supported this theory. The heat conduction of the brake blocks was investigated both experimentally and by simulation. Several possibilities were examined and found conditions that showed a good approximation in terms of heat distribution between the experiment and the simulation.

Acknowledgement

This project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 958174.

The project No. 2019-2.1.7-ERA-NET-2022-00042 was realized in financial help of the National Research Development and Innovation Fund of the Ministry of Culture and Innovation and Hungarian Organizations successfully participating in the joint international calls of the Era-Net Cofund and EJP Cofund programs.

The authors would like to say thanks to Rail Cargo Hungaria Ltd. and TS Hungaria Ltd. for providing used and new railway brake blocks, which enabled the successfully completion of the presented research.

References

Affonso, L.O.A. (2007) "Machinery failure analysis handbook", Gulf Publishing, ISBN 978-1-933762-08-1 https://doi.org/10.1016/C2013-0-15516-X

ANSYS Inc. (2023a) "Ansys®Fluent 2023 R1 Theory Guide (2023 R1)", In ANSYS Inc., USA. https://www.ansys.com/webinars/ansys-2023-r1-ansys-fluent-whats-new [Accessed: 25 October 2024]

ANSYS Inc. (2023b) "Ansys®Mechanical 2023 R1 Theory Guide (2023 R1)", In ANSYS Inc., USA. https://www.ansys.com/webinars/ansys-2023-r1-mechanical-update [Accessed: 25 October 2024]

Belhocine, A. (2015) "Numerical investigation of a three-dimensional disc-pad model with and without thermal effects", Thermal Science, 19(6) pp. 2195–2204. https://doi.org/10.2298/TSCI141130072B

Belhocine, A., Abdullah, O. I. (2020) "Finite element analysis (FEA) of frictional contact phenomenon on vehicle braking system", Mechanics Based Design of Structures and Machines, 50(9), pp. 2961–2996. https://doi.org/10.1080/15397734.2020.1787843

Borsodi, E., Takács, Á. (2023) "Development of a design method for circular manufacturing", [pdf] GÉP, Miskolc, Hungary. Available at: https://real-j.mtak.hu/25414/8/gep_2023_74_4.pdf [Accessed: 25 October 2024]

Ficzere, P. (2024) "Analysis of the causes and effects of noise from rail transport", International Journal of Engineering and Management Sciences, 9(1), pp. 116–130. https://doi.org/10.21791/IJEMS.2024.009

Han, J., Ning, K., Han, M., Zhang, N., Du, J., Lang, Z., Liu, Z., Liu, G., Ji, Z., Jia, C. (2020) "The effect of rotation speed on the temperature and stress field of iron-based friction pairs", presented at Journal of Physics: Conference Series, (ICMSE2020), Guiyang, Guizhou, China, Aug, 14–15. https://doi.org/10.1088/1742-6596/1637/1/012039

Limpert, R. (2011) "Brake design and safety", Society of Automotive Engineers Inc. ISBN 9780768057898 [online] Available at: https://books.google.hu/books?id=HXd0EAAAQBAJ&dq=Limpert,+R.+(1999)+%22Brake+design+and+safety%22,&lr=&hl=hu&source=gbs_navlinks_s [Accessed: 25 October 2024]

Liu, N., Hong, C., Su, X., Jin, X., Jiang, C., Shi, Y., Wang, B. (2024) "Numerical and experimental analysis of the aerodynamic torque for axle-mounted train brake discs", Fluid Dynamics & Materials Processing, 20(8), pp. 1867–1882. https://doi.org/10.32604/fdmp.2024.047427

Menyhártné Baracskai, M. (2015) "Thermodynamical examination of the oily disc brake of an agricultural motor vehicle", Periodica Polytechnica Transportation Engineering, 43(1), pp. 1–8. https://doi.org/10.3311/PPtr.7255

Straffelini, G. (2015) "Friction and wear", Springer, ISBN 978-3-319-05893-1 https://doi.org/10.1007/978-3-319-05894-8

Szabó, A., Zobory, I. (1998) "On deterministic and stochastic simulation of wheel and rail profile wear process", Periodica Polytechnica Transportation Engineering, 26(1–2), pp. 3–17.

Szabó, F. J., Várkuliné Szarka, Á. (2019) "Finite element study of rotating elements of a ventilator", International Review of Mechanical Engineering, 13(6), pp. 326–331. https://doi.org/10.15866/ireme.v13i6.17145

Szűcs, J., Váradi, K., Zobory, I., Szabó, A., (2010) "Wear behaviour of a brake block-wheel connection", GÉP, 11, pp. 28–32. Tulipánt, G. (2007) "Comparison of specific values of noise emission caused by road, railway and combined cargo transport process", Periodica Polytechnica Transportation Engineering, 35(1–2), pp. 85–100. https://pp.bme.hu/tr/article/view/1892

Williams, J. (2005) "Engineering tribology", Cambridge University Press, ISBN 9780511805905 https://doi.org/10.1017/CBO9780511805905

Zeng, C., Huang, J., Wang, H., Xie, J., Huang, S. (2022) "Rail break prediction and cause analysis using imbalanced in-service train data", IEEE Transactions on Instrumentation and Measurement, 71, pp. 1–14. https://doi.org/10.1109/TIM.2022.3214494