Airflow simulation when braking with a disc brake

Y Fomina¹, V Pavelčík¹

¹University of Zilina, Faculty of Mechanical Engineering, Department of Transport and Handling Machines, Univerzitna 1, 010 26 Zilina, Slovak Republic

Corresponding author Email address: yuliia.fomina@fstroj.uniza.sk

Abstract. With an increase in train motion speed, the issue of improving braking systems becomes more and more important, since braking systems have a direct impact on safety of traffic. At high travel speeds, the friction elements heat up strongly, so heat dissipation from the brake discs is essential. Therefore, an important task is to study the aerodynamics of brake discs. To achieve these goals, experimental measurements on a special test bench, or simulation in specialized software packages can be used. The article discusses methods for studying air flow when braking with a disc brake. The simulation methods, turbulence models and software packages used for simulation are considered. The influence of the internal geometry of the discs on ventilation and heat dissipation is considered. Also, the geometry of the disc is analysed using simulation in the specialized software. Based on the simulation results, a conclusion about the influence of the internal disc geometry on the air flow through it is drawn.

1. Introduction

The main purpose of the braking system is to slow down or completely stop the vehicle at a certain distance. Every moving vehicle receives kinetic energy from its movement. The brake converts most of this energy into heat through friction. If this heat is not dissipated from the brakes, it leads to deformation, cracking, vibration, brake fade and increased wear [1-4].

The braking efficiency required for a rail vehicle is one of the most important criteria when choosing the type of braking system. The two most common types of friction brakes currently used in rail vehicles are shoe brakes and disc brakes.

With the increasing speed of passenger rail vehicles and the amount of energy dissipated during braking, and finally as the requirements for braking performance become stricter, the shoe brake has been replaced by a disc brake in modern vehicles [5-7].

The most important advantage of a disc brake over a shoe brake is that it allows high braking power to be performed without creating thermal stresses in the wheels. It has a beneficial effect on the operational safety and service life of the wheelsets. In addition, vehicles with shoe brakes tend to generate rolling noise due to wheel with shoe-rail contact [8-10].

In a typical rail vehicle with disc brake, the brake disc may be axle-mounted or wheel-mounted, depending on the free space in the bogie. Wheel-mounted brake discs are often installed on locomotives and motorized bogies of multiple-unit trains, while axle-mounted brake discs are installed on passenger cars and non-motor bogies of multiple-unit trains.
The disc brake used on high-speed passenger cars, where the main requirement is safety, must have high reliability. The location of the disc on the axle of the wheelset imposes a requirement on the disc that is the durability comparable to the structural life of the wheelset.

An important task is to study the aerodynamics of the brake disc during rotation. High quality of brake discs depends on their aerodynamic efficiency, which affects the characteristics of the train movement, as well as the efficiency of the train braking system.

The brake discs can be ventilated with various geometry or solid (figure 1). Modern high-speed vehicles require more heat dissipation. To increase heat dissipation from the brakes, the discs have internal vanes that increase heat dissipation. Solid discs can absorb heat well but dissipate it poorly. Since the braking process is very fast, the brake disc must be able to store energy and quickly dissipate it during braking. The ventilated disc allows air to pass through, but it is equally difficult to achieve a balance between airflow and convective heat dissipation. The heat dissipation of a ventilated brake disc is highly dependent on the aerodynamic characteristics of the flow through the disc channels. All three heat dissipation mechanisms (heat conduction, convection and radiation) contribute to the cooling of the disc. The aerothermodynamics of a ventilated brake disc is complex and largely dependent on its geometry and environment.

![Figure 1. Wheelset with solid (a) and ventilated (b) discs.](image)

Most of the previous researchers measured the airflow profile using the Particle Image Velocimetry (PIV) method, which belongs to a class of non-contact methods for measuring flow velocity. The use of experimental methods makes it possible to obtain information about the dynamics of structures, their range, calculation of differential characteristics, spatial and space-time correlations, as well as statistical characteristics of the flow [11, 12].

Anemometers, which are devices for determining the motion speed of flows of air, gas or wind, are also used to measure the speed and direction of fluid movement. Hot-wire anemometers (HWA), which, in addition, have a heat meter, are also used for measuring the movement of air flows [13, 14].

Since full scale dynamometric testing is time-consuming and expensive, it seems appropriate to apply simulation methods and use bench tests only to validate the design before proceeding to the prototype stage. With the advent of modern high-performance computers, CFD (Computational Fluid Dynamics) methods began to be used in vehicle aerodynamics.

The use of CFD becomes attractive when experimental research is inapplicable or overly complex. In general, experimental methods require a significant investment of time and financial resources. CFD tools are used to analyze the brake disc aerodynamics while rotating.

The article [15] investigates the thermal analysis of a wheel brake disc for a locomotive under emergency braking. The internal geometry of the disc influenced the change in temperature.

The analysis of heat dissipation by ventilated discs of a high-speed train during emergency braking using the conjugate heat transfer method is described in [16]. Decreasing the wall thickness has been shown to be effective in improving the vented disc heat dissipation characteristics.
The authors [17] studied three types of brake discs in order to analyse the air temperature inside the ventilation channels. Analysis has shown that heat dissipation in disc brakes depends on the internal geometry of the disc.

In a paper [18] the results of a study to determine the ventilation loss of a railway brake disc are described. Authors have optimized the internal geometry of the disc in order to reduce ventilation losses.

The decision of the specific design of the brake disc depends on the operating conditions. If the use of brakes is less frequent and has a higher power (for example, high-speed trains), a higher heat capacity is desirable [19-22]. In contrast, when braking occurs frequently and at a lower starting speed (for example, subway trains), then the cooling efficiency plays a decisive role.

2. Basic equations of aerodynamic calculations

To solve most of problems, flow fluid dynamics uses rigorous mathematical methods of integrating the basic differential equations with an established system of boundary and initial conditions or other mathematically equivalent methods. To obtain the summary characteristics, general theorems of mechanics (the conservation of momentum and moment of momentum theorems, energy conservation theorem, etc.) are used. However, the great complexity and insufficient information about many phenomena leads to the fact that the mechanics of fluids does not apply only strict methods of theoretical mechanics and mathematical physics, but also widely uses all kinds of empirical methods and so-called "semi-empirical" theories, in the development of which individual experimental facts play an important role [23].

According to the property of fluids to change their volume in a varying degree under the influence of pressure or temperature, the fluids are divided into incompressible (i.e., those whose compressibility can be neglected) and compressible or elastic [9].

As a first approximation, when solving many problems, air can be considered as an incompressible fluid. However, at high speeds of movement, air can no longer be considered as an incompressible fluid and it is necessary to take into account its compressibility.

As for the study of the fluid particle motion, the main relation in the mathematical model for its description is the relation of Newton's second law. However, due to the deformability of the medium with the possibility of changing the relative distances between the particles of the environment in the process of motion, it is necessary to supplement Newton's second law with other conservation laws in order to create a system of equations of the mathematical model. First of all, these are conservation laws common to all branches of mechanics, i.e. the energy conservation law, the momentum conservation law and the law of conservation of angular momentum. Conservation laws, which are written in terms and concepts used in the construction of a mathematical model of the phenomenon, often make it possible to formulate important conclusions regarding the system motion.

First of all, it is necessary to point out an important relationship that exists in all models of continuum mechanics, that is, the continuity equation, which can be considered as a specific form of writing the mass conservation law.

The continuity equation in Euler's variables takes the form

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{v} = \frac{\partial \rho}{\partial t} + \frac{\partial \left( \rho \vec{v} \right)}{\partial x_i} = 0$$

(1)

where $\rho$ is fluid density [kg.m$^{-3}$], $t$ is time [s], $\vec{v}$ is the flow velocity vector field, $\vec{v}$ is the flow velocity of the fluid [m.s$^{-1}$].

The first term describes the nonstationarity of the flow, and the second term represents convective transfer.

In the case of an incompressible fluid ($\rho$=const), the continuity equation transforms into the incompressibility equation:
The Navier-Stokes equations for an incompressible fluid are designed to calculate the velocity and pressure fields in a moving viscous fluid. The equations of motion are based on Newton's second law, which is formulated for hydrodynamics in the form of momentum balance equations.

The momentum conservation law states: the rate of the momentum change of the fluid volume is determined by the mass and surface forces.

System of three equations of motion given in stresses is:

\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i
\]

where \(\rho u\) is the specific impulse, or the momentum of a volume unit [kg.m\(^{-2}\).s\(^{-2}\)], \(\tau_{ij}\) is the Reynolds stress tensor [m\(^{2}\).s\(^{-2}\)], \(g_i\) is a gravitational acceleration [m.s\(^{-2}\)].

The first term describes the nonstationarity of the flow, the second one characterizes convective transfer, the third and fourth terms represent surface forces (pressure gradient and molecular diffusion), the fifth one describes mass forces (gravity).

This system is called Navier equations. It is supplemented by the continuity equation for a compressible or incompressible fluid. Obviously, the Navier system of equations is not determined, since the internal stresses \(\tau_{ij}\) are not defined.

Stokes' law of friction is used for determining. For Newtonian fluids, the stress tensor is determined by the Stokes hypothesis.

Navier-Stokes equations for an incompressible fluid are:

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \rho}{\partial x_i} + \frac{\mu}{\rho} \frac{\partial^2 u_i}{\partial x_j \partial x_j} + g_i
\]

where \(\mu\) is a dynamic molecular viscosity coefficient [kg.m\(^{-1}\).s\(^{-1}\)].

To compose an energy equation in a moving fluid, we use the general energy conservation law, which, when applied to a moving individual volume, can be formulated as follows: the change in the total energy of a volume of fluid for an infinitely small period of time is equal to the sum of the elementary work of external mass and surface forces, applied to the allocated volume and its surface, added to an elementary amount of heat supplied from the outside to the volume for the same period of time.

Differential energy conservation equation is:

\[
\rho \frac{\partial c_p T}{\partial t} + \rho c_p u_j \frac{\partial T}{\partial x_j} = \frac{\partial q_i}{\partial x_i} + \frac{\partial \tau_{ij} u_i}{\partial x_j} + S
\]

The first term describes the nonstationarity of the flow, the second one describes convective transfer, the third one represents the transfer due to heat conduction and is determined by the Fourier law for energy transfer, the fourth one characterizes energy dissipation, the fifth one describes the inflow or loss of energy due to chemical reactions, radiation, etc.

There are two types of fluid movement: laminar and turbulent. The flow state can be characterized using the Reynolds number. With an increase in the Reynolds number, the flow loses its stability and there is a transition from the laminar flow regime to the turbulent one.
3. Simulation of turbulent flows

The motion laws of a fluid that has lost its stability are very complex. At the moment, a large number of various models have been created for calculating turbulent flows. They differ from each other in the complexity of the solution and the accuracy of the flow description.

The classification of existing approaches to the numerical modeling of turbulent flows is based on the degree of detail in the solution of turbulent pulsations and their energy spectrum. Taking this feature into account, there are three main approaches (DNS - Direct numerical simulation, LES - Large eddy simulation and RANS - Reynolds-averaged Navier-Stokes) [24].

DNS and LES provide information on unstable turbulent flow effects, RANS works with average values. DNS and LES are complex and more accurately describe the flow. But in practice, they are rarely used due to the limited capabilities of computer technology. There are also hybrid approaches that combine the computing techniques of different approaches.

For technical use, a method based on the solution of the Reynolds-averaged Navier-Stokes (RANS) equations is more suitable.

The Reynolds-averaged system of equations is not determined, and turbulent stresses must be determined to solve this problem. This goal is achieved using one or another turbulence model. Turbulence models are based on the concept of viscosity and turbulent diffusion.

Models using the Boussinesq assumption (the concept of turbulent viscosity) are called first order models. The concept of turbulent viscosity has proven to be extremely useful in the construction of approximate turbulence models, and it is still widely used in numerical simulations. However, the application of the concepts of turbulent viscosity can lead to large errors in the near-wall regions, in the case of flow separation, or in swirling flows.

The simplest models that determine the turbulent viscosity are algebraic models, in which the solution of differential equations is not required to calculate the turbulent viscosity. The advantages of algebraic models include: speed of calculations, ease of calibration and adjustment, taking into account the specifics of the considered flows. However, it is also obvious that these models have a narrow specialization.

Models with one differential equation allow one to take into account the influence of non-local effects (transfer effects). To determine the kinetic energy of turbulence $k$, the transport equation is solved.

More universal models in engineering calculations of turbulent flows are models with two differential equations. They are characterized by the simultaneous use of two transport equations for turbulent characteristics that generate a whole group of models.

The most common model in this class is the $k-\varepsilon$ model. The simplicity, good convergence and good accuracy of the $k-\varepsilon$ model make it the most widely used model for simulating a wide range of turbulent flows. Standard $k-\varepsilon$, $k-\omega$ realizable, and RNG (Renormalization Group Method) $k-\varepsilon$ versions are known, which are used for different types of flow.

There is another class of models where, instead of the transfer equation for $\varepsilon$, the equation for $\omega=\varepsilon/k$ (turbulence frequency is the reciprocal of the lifetime of large eddies, which leads to better agreement with experiment for this class of flows if the separation zone is small) is used.

Turbulence models of the $k-\varepsilon$ type better describe the properties of shear flows, and models of the $k-\omega$ type have advantages in modeling functions near the wall.

On the basis of this, a model, which combines the advantages of the $k-\varepsilon$ and $k-\omega$ models has been proposed. This approach, called shear stress transport (SST), was widely used in other two-equation turbulence models. The SST model has proven itself well in calculating separated flows with a small separation zone.

One interesting model is the Durbin $\nu^2/f$ model, which is often used to calculate flows in gas turbine engine turbines, diffusers, and flow separation configurations. Instead of kinetic energy $k$ it uses the mean-square velocity fluctuation $\nu^2$ along the normal to the streamline.

Nonlinear turbulent viscosity models sometimes allow reproducing flow details that are not described by traditional turbulent viscosity models. However, in engineering practice, models of this class are rarely used. Nonlinear models have a significant disadvantage that is a large number of model constants.
4. Software for flow simulation

The subject of computational fluid dynamics is the solution of complex boundary value problems in fluid mechanics and heat transfer using the methods of computational mathematics and computer simulation.

The computer simulation provides not only the numerical solution of mathematical problems, but also the mesh generation and a graphical presentation of the results (visualization of the results of numerical modeling).

Recently, computational fluid dynamics has been developing rapidly all over the world. Universal commercial CFD software systems have been created, which are intended for industrial operation in companies and design departments of various specialization - from the aerospace and automotive industries (FLUENT, Open FOAM, STAR-CD, PHOENICS, etc.) to oil companies (ROXAR, Schlumberger, STARS and etc.).

In the field of computational fluid dynamics, the main goal is the final creation of a numerical model of the process under study. This model makes it possible to simulate the fluid flow on a computer with various initial data and do research.

The use of computational fluid dynamics tools in the design of processes and equipment has a number of obvious advantages over experimental methods. Here are the main ones:

• Reduced testing time and costs. There is no need to create expensive full-scale models. So energy and human resources are saved. Training of specialists can take place regularly.

• Increase of information content. Many velocity, pressure and temperature fields are simply impossible to obtain experimentally (high temperatures, the impossibility of installing sensors, very small or large spatial and time scales). Various external conditions and process modes are easily simulated by means of CFD by introducing boundary conditions, but they cannot be realized experimentally.

• Improvement of mathematical models and methodologies. In the course of a computational experiment, the parameters of the model are selected, functional connections are refined, and new theories are created. For example, the creation of most semi-empirical turbulence models cannot be imagined without CFD tools.

Any CFD simulation involves solving the three equations described earlier that represent the balance of mass, momentum, and heat in the system.

One of the complex highly specialized programs for solving CFD problems is the ANSYS software complex, which also contains preprocessors for preparing the model, that are processing geometry and generation of mesh necessary for solving fluid dynamics.

ANSYS offers several packages and licensed add-ins for solving CFD problems. It includes ANSYS CFX and ANSYS Fluent, which are often used to calculate brake disc aerodynamics. ANSYS Fluent and ANSYS CFX contain a wide range of different turbulence models, there are time-proved RANS models, as well as modern large and detached eddies methods LES and DES, as well as hybrid models [25].

ANSYS Fluent provides comprehensive simulation capabilities for a wide range of incompressible and compressible, laminar and turbulent fluid problems. Steady or transient analysis can be performed. ANSYS Fluent combines a wide range of mathematical models of transport phenomena (such as heat transfer and chemical reactions) with the ability to model complex geometry.

For all flows, ANSYS Fluent solves the mass and momentum conservation equations. For flows involving heat transfer or compressibility, an additional energy conservation equation is solved. For flows involving mixing of components or reactions, the equation of components conservation is solved or, if a combustion model without premixing is used, the conservation equations are solved for the fraction of the mixture and its dispersion. Additional transport equations are also solved for turbulent flow.
5. Analysis of the airflows
This article considers discs with radial vanes having the same dimensions, but a different number of vanes (10, 24 and 36). The diameter of the disc is 610 mm and a thickness is 110 mm. Models have been created in SolidWorks software and analysis has been performed in ANSYS Fluent software. The airflow field model as well as the rotational region were created by applying the Boolean operation on the disc model. Then a finite element model of the whole computational domain was obtained.

Table 1 shows the number of elements and nodes for each analysed disc model.

|                | 10 vanes | 24 vanes | 36 vanes |
|----------------|----------|----------|----------|
| No. nodes      | 132466   | 256821   | 609846   |
| No. elements   | 352885   | 713782   | 1531819  |

The analysis was carried out under the following boundary conditions:
- The rotational speed of the disc is set equal to 600 rpm. The sliding mesh method was used to realize the wall movement,
- The initial speed of the “velocity inlet” is set equal to the speed of the vehicle. It has a value of 29 m.s\(^{-1}\) when a wheel radius is 950 mm,
- Ambient temperature is 25 C,
- Ambient pressure is 101325 Pa,
- Air density is 1.225 kg. m\(^{-3}\),

The boundary conditions for each disc design are the same. The applied turbulence model is k-\(\varepsilon\) realizable with standard wall function. This turbulence model provides reliability, economy, and reasonable accuracy for a wide range of turbulent flows.

The simulated time is 10 s.

The air velocity through the disc is shown in Figure 2.

![Figure 2](image)

**Figure 2.** Air flow velocity for disc with 10 (a), 24 (b) and 36 (c) vanes

The air flow velocity is shown in the form of vectors. The arrows show the direction of air movement through the disc interior. The direction of air movement changes also can be seen. And it is because of these changes that turbulence occurs. The radial flow through the disc channels depends on vanes individual circumferential position within the wheel. In designs with a large number of vanes, the speed of the passing air increases, and the value of turbulence also increases (figure 3).
**Figure 3.** Air flow turbulences for disc with 10 (a), 24 (b) and 36 (c) vanes

**Conclusion**

The paper discusses methods for studying aerodynamics, which can also be applied in the analysis of braking with a disc brake. The basic equations of aerodynamics are considered, as well as turbulence models and software that can be used to implement the calculation are described.

Finite element models of the ventilated disc designs and the air flow field have been built. Three discs with the same dimensions but different numbers of radial vanes were considered. As a result of simulation, the influence of the number of radial vanes on the speed of air flow through the disc, as well as on the occurrence of turbulence during its rotation, is shown. In this case, the speed and turbulence increased with an increase in the net of the blades.

The design of the brake disc has a great influence on the heat dissipation in the air. Attempts to achieve energy savings must take into account the reduction of pumping losses, these requirements must be complemented by considerations of surface temperature as well as studies of pad and disc life. Finding the right balance between high cooling performance and low aerodynamic losses remains a relevant objective and could be the subject of further research in this area.

**References**

[1] Gerlici J, Lack T and Harušinec J 2013 Development of test stand prototype for rail vehicles brake components testing Communications: Scientific letters of the University of Žilina 16 (3A) 27-32 ISSN 1335-4205

[2] Gerlici J, Gorgunov M, Kravchenko K, Kostyukevich A, Nozhenko O and Lack T 2016 Experimental rigs for wheel/rail contact research Manufacturing technology: journal for science, research and production 16 (5) 909-916 ISSN 1213-2489

[3] Gerlici J, Gorbunov M, Kravchenko K, Prosvirova O and Lack T 2017 The innovative design of rolling stock brake elements Communications: scientific letters of the University of Žilina 19 (2A) 23-28 ISSN 1335-4205

[4] Strážovec P, Gerlici J, Lack T and Harušinec J 2019 Innovative solution for experimental research of phenomena resulting from the wheel and rail rolling TRANSCOM 2019 Amsterdam: Elsevier Science 906-911 ISSN 2352-1465

[5] Kurčík P, Gerlici J, Lack T, Suchánek A and Harušinec J 2019 Innovative solution for test equipment for the experimental investigation of friction properties of brake components of brake systems TRANSCOM 2019: conf. proc. Amsterdam: Elsevier Science 759-766 ISSN 2352-1465

[6] Gerlici J and Lack T 2011 Railway wheel and rail head profiles development based on the geometric characteristics shapes WEAR 271 (1-2) Special Issue SI, 246-258 DOI: 10.1016/j.wea.2010.10.052, Published: MAY 18 2011

[7] Chudzikiewicz A, Gerlici J, Sowińska M., Stelmach A and Wawrzyński W 2020 Modeling and simulation of a control system of wheels of wheelset Archives of Transport 55 (3) 73–83

[8] Lack T and Gerlici J 2014 A modified strip method to speed up the tangential stress between wheel and rail calculation Applied mechanics and materials 486 371-378 ISSN 1660-9336
Gerlici J and Lack T 2014 Modified HHT method for vehicle vibration analysis in time domain utilisation  

Applied Mechanics and Materials 486 396-405, DOI: 10.4028/www.scientific.net/AMM.486.396

Lack T and Gerlici J 2015 Railbcot test stand dynamic properties analysis  

Proc. Of 22nd Int. Conf. on Current Problems in Rail Vehicles part I 159-170

Atkins M D, Kienhofer F W and Kim T 2019 Flow behavior in radial vane disc brake rotors at low rotational speeds  

ASME Journal of Fluids Engineering 141 doi:10.1115/1.4042470

Johnson D A, Sperandei B A and Gilbert R 2003 Analysis of the flow through a vented automotive brake rotor  

ASME Journal of Fluids Engineering 12 979-989

Parish D and Macmanus D G 2005 Aerodynamic investigations of ventilated brake discs  

Proc IMechE Part D: J Automobile Engineering 219 (7) 471-486

Barigozzi G, Cossali G E, Perdicchizzi A, Boden A and Pacchiana P 2002 Experimental Investigation of the Mean and Turbulent Flow Characteristics at the Exit of Automotive Vented Brake Discs  

SAE Technical Paper Series doi:10.4271/2002-01-2590

Ghadimi B, Kowsary F and Khorami M 2012 Thermal analysis of locomotive wheel-mounted brake disc  

Applied Thermal Engineering 51 (2013) 948-952

Luo Z and Zuo J 2014 Conjugate heat transfer study on a ventilated disc of high-speed trains during braking  

Journal of Mechanical Science and Technology 28 1887-1897

Garcia-Leon R A and Rojas E P 2017 Analysis of the Amount of Heat Flow Between Cooling Channels in Three Vented Brake Discs. Ing. Y Univ. 21 71-96

Grive U, Deržič D and Muhič S 2019 Numerical optimisation and experimental validation of divided rail freight brake disc crown  

J. Mod. Transp 27 1-10

Gerlici J, Lack T and Harušinec J 2013 Realistic simulation of railway operation on the Railbcot test stand.  

Applied Mechanics and materials 486 387-395

Lack T and Gerlici J 201. Rail vehicles brake components test bench utilisation  

Applied mechanics and materials 486 379-386 ISSN 1660-9336

Gerlici J Lack T and Harušinec J 2013 The test stand load modulus implementation for the realistic railway operation in the laboratory conditions  

Manufacturing technology: journal for science, research and production 13 (4) 444-449 ISSN 1213-2489

Gerlici J, Lack T and Harušinec J 2014 Rail vehicles wheels and brake blocks wear laboratory test stand utilization  

Prace Naukowe. Transport: Analiza i ocena elementów systemów transportowych 101 21-32 ISSN 1230-9265

Loitsyanskii L G 1950 Mechanics of fluids and gases (in Russian) (Moscow: State Publishing House of Technical and Theoretical Literature) p 698

Wilcox D C 2004 Turbulence Modeling for CFD (California: DCW Industries Inc.) p 477

ANSYS Inc 2013 ANSYS Fluent 15.0. Theory Guide ANSYS Inc.

Acknowledgement

This research was supported by the Slovak Research and Development Agency of the Ministry of Education, Science, Research and Sport of the Slovak Republic in Educational Grant Agency of the Ministry of Education of the Slovak Republic in the project No. VEGA 1/0558/18: Research of the interaction of a braked railway wheelset and track in simulated operational conditions of a vehicle running in a track on the test bench” and No. KEGA 036ŽU-4/2021: “Implementation of modern methods of computer and experimental analysis of the properties of vehicle components in the education of future vehicle designers.”