UDC 519.63: 697.957

NUMERICAL SIMULATION OF THE AIRFLOW WITH WEAR PARTICLES IN THE AERODYNAMIC CHAMBER

Uthara Lal, master student
Gdańsk University of Technology, Poland
Y. Tsybrii, PhD, assistant professor
Gdańsk University of Technology, Poland

Key words: airborne wear particle, aerodynamic chamber, particles emission, disc brake, pin-on-disc, airflow.

The primary purpose of car disc brakes is to slow down a vehicle by converting kinetic energy into frictional heat. Nowadays, most modern cars are equipped with disc brakes that are not sealed off from ambient air, during braking both the rotor and pads in the disc brakes are worn, and this wear process gives rise to wear particles. Some of them are deposited on the hardware while others may become airborne.

Many studies have shown the adverse effects of disk brake wear particles on the health conditions of living beings [1]. Studies have shown that these airborne particles penetrate the human body continuously through breathing, drinking, and skin contact. Some of the major health effects caused by these particles include asthma, lung cancer, cardiovascular diseases, and skin diseases.

To study the phenomenon of airborne wear particles emission different types of experimental stands and approaches are used, e.g.: pin-on-disc tribometer, brake dynamometer, car with closed brake system connected to the measuring equipment, etc. [2]. The experimental process of braking and particles emission occurs in closed volume, called aerodynamic chamber. The filtered airflow is supplied to its inlet and the air with emitted from friction couple particles is transported to the measuring equipment through the aerodynamic chamber’s outlet channel. The perfect aerodynamic chamber is one with rapid response, this means that the particle generated and particle measured in the perfect case is when the delay time is zero.

The present study aims to evaluate the efficiency of a typical aerodynamic chamber used for airborne wear particle measurements in dependence on the particle properties and airflow regime.

Computational Fluid Dynamics (CFD) is a simulation-based tool for examining particle behavior, dispersion, and depositions by investigating complex flow systems. This approach is known as a dependable tool for various scientific issues. In the past, CFD models have been widely used to analyze and optimize disc brake cooling, flow, and heat transfer through the brake system, brake disc contamination, and brake dust particles emission. Therefore, simplified pin-on-disc model was used in this study. The CFD approach, applying ANSYS Fluent software package, was selected to simulate the airborne wear particles generation between the friction zone of the pin and the rotational disc and their transportation with airflow to the aerodynamic chamber's outlet channel. The simulation scheme is shown in the fig. 1.

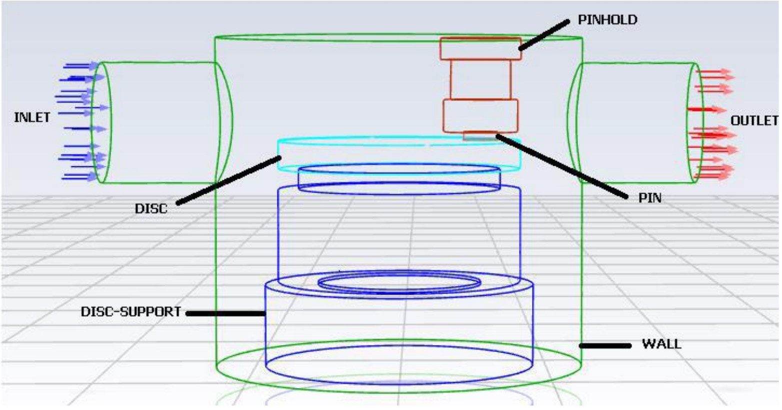

Figure 1 – Simulation scheme of the airflow in the aerodynamic chamber

Four influential factors namely airflow velocity (0.1-0.3m/s), angular velocity (100-300rpm), particle diameter (0.1-10μm), and particle density (500-2000 kg/m^3) were examined. Typical result of the simulation is in the fig. 2.

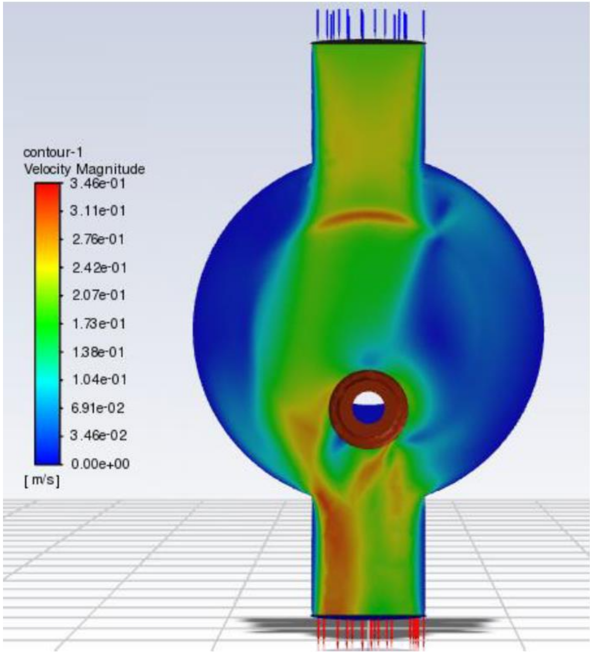

Figure 2 – Airflow pattern in the aerodynamic chamber at 100rpm (inlet velocity 0.2m/s, particle diameter of 1μm, particle density of 1000 kg/m^3)

The simulated results of the research showed that increasing the airflow velocity also increases the particle sampling ratio i.e. more particles leave the chamber. It was also seen that heavier particles had longer response time and particle path lengths.

References

1. E.R. Shell, The Role of Air Pollution, Scientific American, 322 (5) (2020) 42–47.

2. J. Kukutschová, P. Filip, Review of brake wear emissions: A review of brake emission measurement studies: Identification of gaps and future needs, in: F. Amato (Ed.), Non-exhaust emissions, Academic Press, 2018, 123–146.