

A Comparative Study of Airflow Around a Vehicle During On-Road Driving and on a Chassis Dynamometer

Daisuke Kawano ¹⁾ Souma Iiboshi ²⁾ Kazuki Aoyama ¹⁾ Toshimichi Takahashi ³⁾

¹⁾ Osaka Sangyo University

3-1-1, Nakagaito, Daito, Osaka, 574-8530, Japan (E-mail: kawano@mech.osaka-sandai.ac.jp)

²⁾ Osaka Sangyo University Graduate School (Current Daihatsu Motor Co., Ltd.)

3-1-1, Nakagaito, Daito, Osaka, 574-8530, Japan

³⁾ Meidensha Corporation

2-1-1 Osaki, Shinagawa-ku, Tokyo, 141-6029, Japan

KEY WORDS: Heat•fluid, Vehicle body, Computational fluid dynamics, On-road driving, Chassis dynamometer [D1]

In electric vehicles with batteries mounted on the underside, airflow along the underside of the vehicle significantly affects its thermal management system, potentially leading to discrepancies between on-road conditions and driving range. Furthermore, even in ICE vehicles, the temperature of the engine and exhaust catalytic converter system changes due to airflow along the front and underside of the vehicle, potentially preventing accurate reproduction of fuel efficiency and exhaust gas performance during on-road driving. Therefore, a review of the chassis dynamometer (CHDY) testing method is necessary to reproduce airflow similar to that experienced during on-road driving. Based on the above, this study aims to identify the requirements for vehicle cooling fans that can reproduce the airflow along the front and underside of the vehicle during on-road driving. This report compared airflow during on-road driving and CHDY testing by simultaneously measuring airflow velocity at multiple points around the vehicle, and analyzed the factors contributing to these differences using CFD.

To understand the flow velocity at the front and bottom of a vehicle with a typical shape, a standard passenger car (hybrid vehicle) was used in this study. The flow velocity around the vehicle was measured during steady-state driving at 19.4 m/s (70 km/h). Fig.1 shows the measurement locations. There were four locations: in front of the bumper (Ch1), below the bumper (Ch2), the front of the bottom (Ch3), and the rear of the bottom (Ch4). The flow velocity at each location was measured simultaneously for 60 seconds, and the average value was calculated. In the numerical analysis, the set flow velocity was 19.4 m/s, the same as the experimental conditions, and steady-state simulations were performed using the standard $k-\varepsilon$ model as the turbulence model. The domain size was set to 9.0 m wide, 6.0 m high, and 13 m long, the same as CHDY room, and a model of a vehicle cooling fan was placed in front of the vehicle during the numerical simulation simulating CHDY. When performing numerical simulations reproducing on-road driving, the model of the vehicle cooling fan was removed, and the airflow was set to flow from the entire front of the domain, thereby simulating wind tunnel testing.

Fig.2 shows the experimental and simulation results for the flow velocity at each location of the vehicle during CHDY and on-road driving. At the front of the vehicle (Ch1), the flow velocity was significantly higher in CHDY compared to on-road driving, and a similar trend was obtained in the numerical simulations. Numerical simulations of the velocity distribution at the front of the vehicle show that in CHDY, the air discharged from the vehicle's cooling fan maintains its flow velocity as it collides with the front bumper, forming a stagnation point. After the collision, the flow direction changes to follow the bumper surface, exhibiting the characteristics of a wall-impinged jet flowing rapidly to the left and right of the bumper. In contrast, simulations reproducing on-road conditions show a linear decrease in flow velocity before impact with the bumper, and the flow along the bumper surface remains slow after the collision.

On the other hand, at the underside of the vehicle (Ch3, 4), the flow velocity was significantly lower in CHDY compared to on-road driving, the opposite of Ch1, and the trends in experiments and numerical simulations were consistent here as well. Numerical simulations of the velocity distribution at the underside of the vehicle show that in CHDY, the air flowing at the underside of the vehicle is only a portion of the flow after impact with the front bumper. Because the airflow into the underside of the vehicle is small, it cannot maintain the flow velocity directly below the front bumper, resulting in rapid deceleration. In contrast, in on-road driving, air flows across the entire underside of the vehicle, so significant deceleration is suppressed in areas other than behind the tires.

From the above, it was found that the airflow around the vehicle in CHDY differs significantly from that in on-road driving, and that the specifications and position of the vehicle cooling fan need to be optimized in order to reproduce airflow equivalent to that in on-road driving in CHDY.

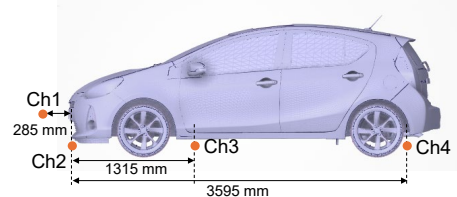


Fig.1 Locations of airflow velocity measurement

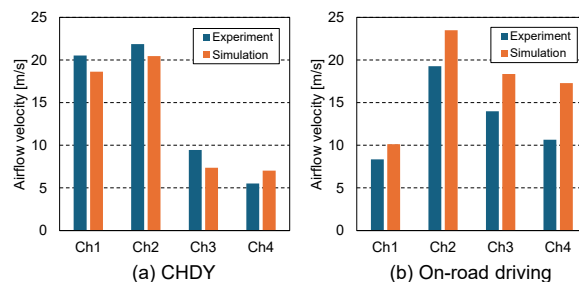


Fig.2 Experimental and simulation results regarding airflow velocity at various points around the vehicle in CHDY and on-road driving