Reproducing the Airflow around Railway Vehicles

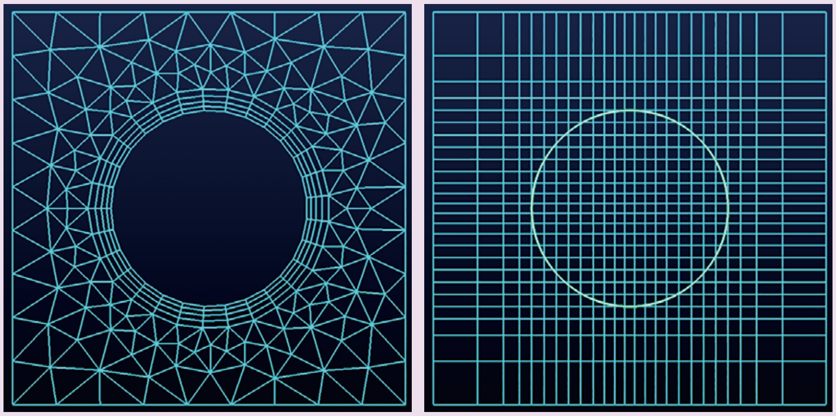
Dr. Koji Nakade
Chief Researcher
Computational Mechanics

Research methods for elucidating airflow include measurements on real targets, tests using wind tunnels, and numerical analysis using a computer. The numerical analysis provides detailed information about the airflow around railway vehicles. This paper outlines the features of the Airflow Simulator, which is our numerical analysis tool. As examples of reproducing the airflow around railway vehicles with the Airflow Simulator, the paper presents numerical simulations of (i) airflow around vehicles in cross winds, (ii) train set airflow around vehicles, and (iii) airflow outside and inside commuter vehicles (in vehicle ventilation during running with windows open).

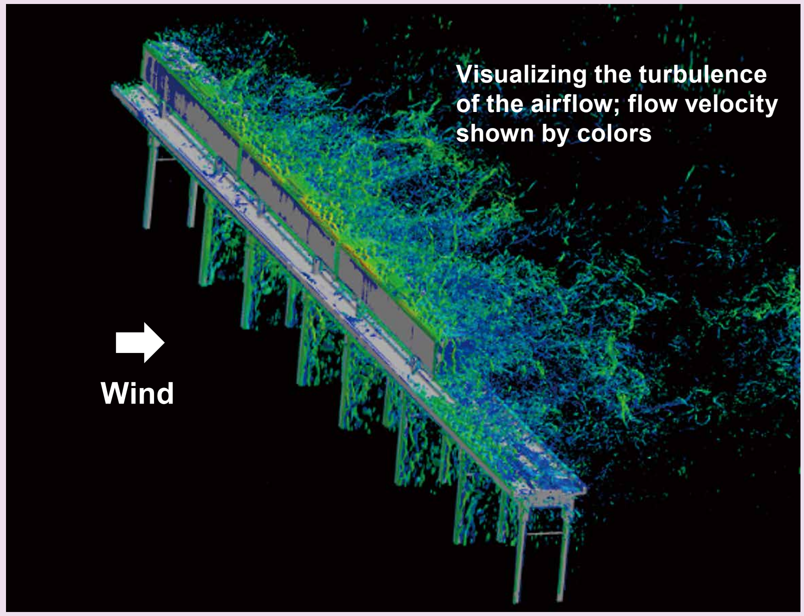
About Airflow Numerical Analysis

Airflow can be expressed by the Navier Stokes equations, which were derived about 180 years ago. They are nonlinear partial differential equations, and instead of finding an exact solution analytically, we choose to find an approximate solution by numerical analysis using a computer.

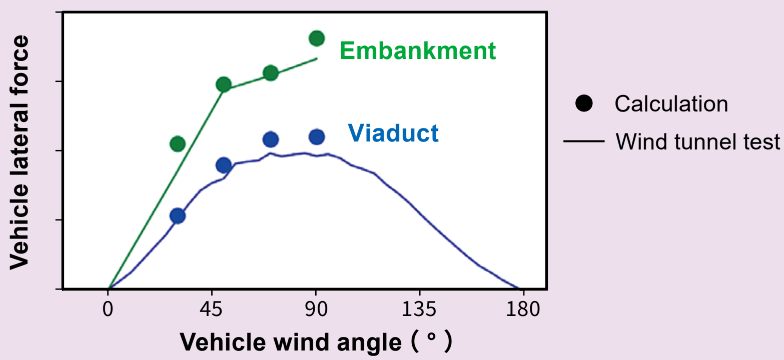
When numerical analysis is applied to the airflow around railway vehicles, that flow is reproduced by the computer within the range of calculation accuracy. That is, we can observe how the airflow changes with time and location. This is a major feature of the numerical analysis method, which allows us to "watch the airflow" in detail. It allows us to understand not only the causes and effects of phenomena but also their progress (i.e. mechanisms). In addition, with the numerical analysis, we can adjust the computation conditions and examine ideal conditions and virtual conditions as well as the actual conditions, thereby allowing us to predict the airflow for various situations. Furthermore, airflow numerical analysis is becoming used as a means to obtain railway vehicle shapes with excellent aerodynamics, which are difficult to obtain from empirical prediction, by fusing it with information science and combining optimization calculation (automatically computing the solution for the desired condition with a computer) with airflow numerical analysis.



An example of computational grids for calculating the airflow around a cylinder



(a) Airflow in the viaduct condition



(b) Comparing the vehicle lateral force with the wind tunnel test

Numerical simulation of the airflow around the vehicles in cross winds

Airflow Simulator to analyze the airflow of complex shape

In the airflow numerical analysis, the Navier Stokes equations are solved for limited points by dividing the space into smaller sections, called computational grids. In conventional airflow numerical analysis, computational grids (boundary adaptive grids) along the object shape are used to compute the airflow near the object with high accuracy (An example of computational grids for calculating the airflow around a cylinder (a)). However, computing a complex shape this way is very costly in computer processing time. Therefore, we focused on Cartesian grids (An example of computational grids for calculating the airflow around a cylinder (b)), which can almost automate the grid generation, and adopted the Cartesian grid method in the Airflow Simulator. The Cartesian grid method is also characterized by its high parallel computing efficiency (i.e. ability to perform multiple operations in parallel), which is essential for large scale computations, and may become a powerful fluid analysis method in the future ultra large scale parallel computer environment. Thus, detailed computational grids in large scale analysis can be expected to ensure calculation accuracy.

Airflow around the Vehicles In cross winds

To realize safe and stable transportation of railway vehicles in strong winds, it is important to know the aerodynamics of railway vehicles in cross winds. To this end, we have been evaluating cross wind aerodynamics by wind tunnel tests using scale models. However, in the evaluation, many parameters must be included, such as the vehicle shape, aboveground structure, and wind characteristics, and it may be difficult to verify all the necessary conditions only by a wind tunnel test. Thus, it is expected that airflow numerical analysis will be useful as a tool for formulating an efficient test plan by narrowing down the test conditions. This section shows an example of a computation that was conducted to verify the prediction accuracy of cross wind aerodynamics by an airflow simulator.

We conducted a numerical analysis using the Airflow Simulator regarding two typical conditions (the first with a vehicle on a viaduct and then on an embankment) of the previously conducted cross wind tunnel tests. With the Airflow Simulator, natural winds were simulated as in the wind tunnel tests, and the airflow around vehicles was subjected to airflow numerical analysis (Numerical simulation of the airflow around the vehicles in cross winds (a)). Comparison of the calculation results with those of the wind tunnel tests demonstrated that the Airflow Simulator can reproduce a wind tunnel test well (Numerical simulation of the airflow around the vehicles in cross winds (b)). The "lateral force" in the figure indicates the lateral force on the vehicle, and the "angle between the vehicle and wind" is defined as 90° when the wind is from the side and as 0° when it is from the front. In future, we will continue to apply the Airflow Simulator to vehicles and aboveground structures with different shapes, increase the number of demonstrated calculation examples, and further study the prediction accuracy.

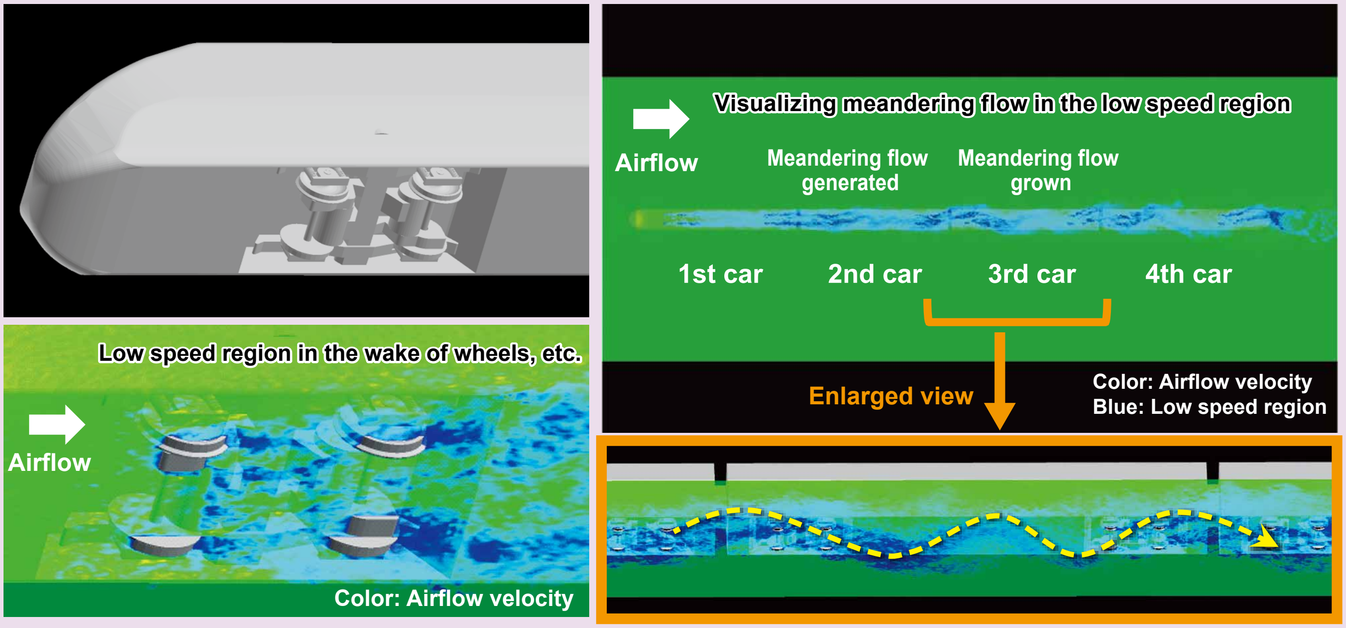
Airflow around Train Set Vehicles

The airflow under the vehicle floor is related to many railway aerodynamic issues such as ballast surface wind speed, vehicle air resistance, snow accretion near bogies, aerodynamic noise generated from the bottom of the vehicles, and car body vibration in the tunnel. We tried to clarify the vehicle underfloor flow by using an airflow numerical analysis.

We conducted an airflow numerical analysis using the Airflow Simulator for the vehicles in a four car train set. The computation results show that, from the flow behavior near the first bogie, a low speed region exists in each wake of a wheel, as shown in Numerical simulation of the airflow around the train set vehicles (a). In addition, by observing the flow of the entire train set vehicles, a meandering flow was found under the vehicle floor, as shown in Numerical simulation of the airflow around the train set vehicles (b). This meandering flow was verified through a wind tunnel test, demonstrating that it was also valid experimentally. Furthermore, the airflow during running through the tunnel was also investigated, which showed that the vehicle underfloor meandering flow seen in the open section spread to the side of each car body (i.e. side near the tunnel wall). It was suggested that the pressure fluctuation caused by this meandering flow acts on the side surface of each car body and that a lateral variable aerodynamic force is generated on the vehicles. We succeeded in demonstrating that the frequency of the fluctuating aerodynamic force predicted herein corresponds to the car body vibration of approx. 2 Hz, measured on the Shinkansen in the past. We presumed that, in conclusion, the primary cause of car body vibration in the tunnel is the meandering flow formed around the vehicles. In this way, we were able to advance the elucidation of the airflow phenomenon around the train set vehicles by using the airflow numerical analysis.

Airflow outside and inside Commuter Vehicles (In Vehicle Ventilation during Running with Windows Open)

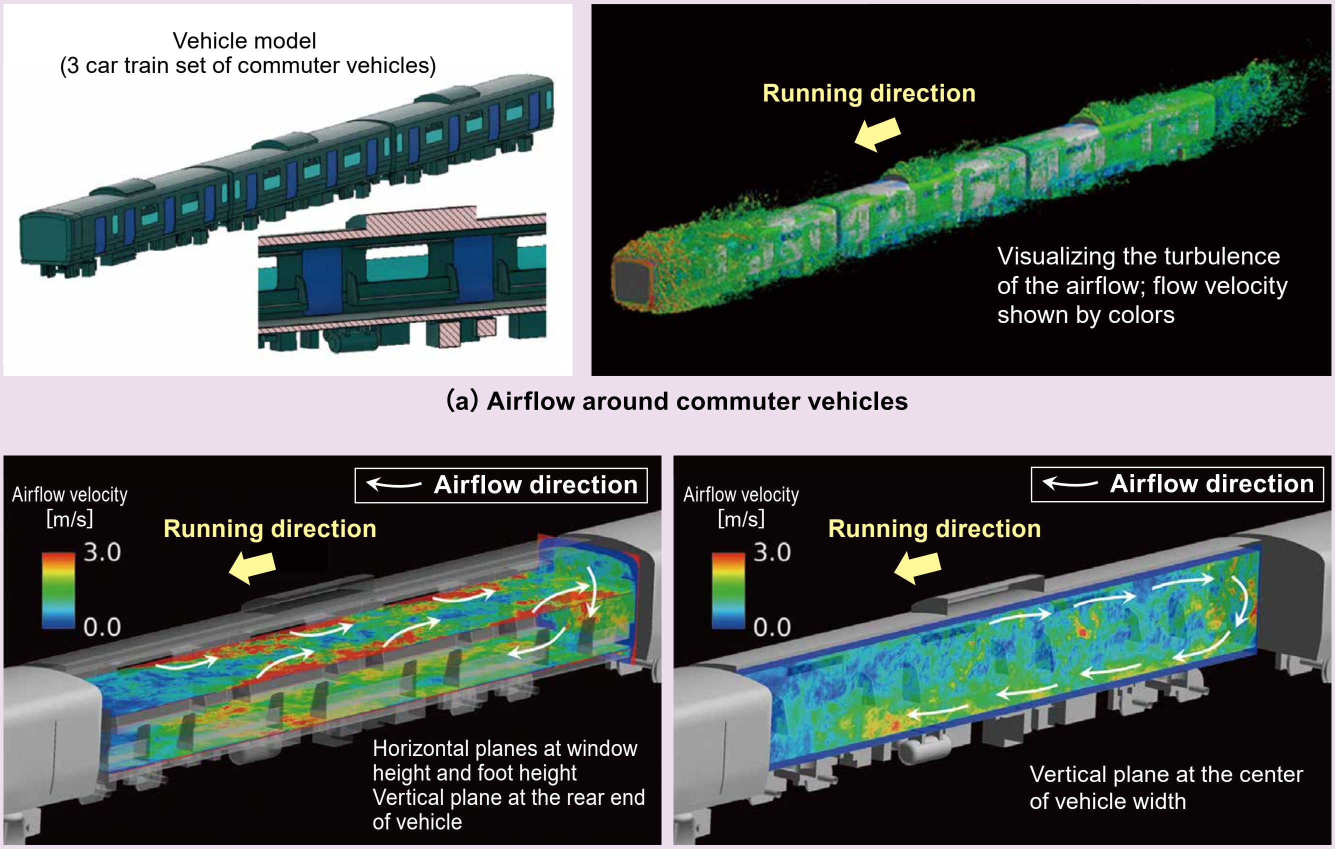
Among the measures against spreading infectious diseases in commuter vehicles, one is to avoid Sealed Space in vehicle ventilation by opening windows. We know empirically that opening the windows improves ventilation in the car.; However, since there are few findings regarding the ventilation volume of running railway vehicles there is a need to quantitatively evaluate the in vehicle ventilation volume.



(a) Airflow near the first bogie

(b) Underbody flow of train set vehicles

Numerical simulation of the airflow around the train set vehicles



(a) Airflow around commuter vehicles

(b) In vehicle airflow (vehicle speed 72 km/h, empty, no air conditioner)

Numerical simulation of in vehicle ventilation during running with windows open

We evaluated the in vehicle ventilation volume during running with the windows open by simultaneously computing the airflow outside and inside vehicles using the Airflow Simulator.

Numerical simulation of in vehicle ventilation during running with windows open (a) shows the airflow around the vehicles, and Numerical simulation of in vehicle ventilation during running with windows open (b) shows the airflow inside the vehicles. By applying the numerical simulation to various conditions such as the window opening amount, the vehicle speed, the occupancy rate, the air conditioner airflow, and the seat layout, we found the following about the in vehicle ventilation volume resulting from opening windows during running: the ventilation volume is proportional to the window opening area and vehicle speed, and the effects are small from the occupancy rate, air conditioner airflow, and seat layout. Finally, we organized these results and proposed a simple prediction formula for the ventilation volume. We verified that the ventilation volume obtained by the simulation roughly matched the measured data.

Conclusion

The main goal is to expand the capabilities of airflow numerical analysis. This R&D has demonstrated three new applications of the method. There are now many cases where the method can be used compared to 10 years ago. We should continue the R&D to expand this range. Effective use of numerical simulation backed by actual measurements is important in research to elucidate phenomena.