Mitteilung

Fachgruppe: Aerodynamik bodengebundener Fahrzeuge

The influence of upstream wind variations on the aerodynamic drag of a model cargo train

K. Weinman*, T.S. Müller, U. Fey, K. Ehrenfried
DLR Göttingen, Institute of Aerodynamics and Flow Technology
Bunsenstr. 10, 37073 Göttingen, Germany
*keith.weinman@dlr.de

DLR NGT-Cargo [1] is a logistics concept using rail as the central mode of freight transport. The concept requires the development of competitive transport systems with respect to operational and associated ecological costs. Within this framework an accurate assessment of the aerodynamic drag under realistic vehicle operating conditions is essential. The air flow about a train is characterized by a large range of energetically significant flow scales which challenge accurate numerical simulation, however CFD methods for certification of trains are now accepted with some restrictions by the transport industry. For example, the EN 14067-6 [2] standard permits evaluation of aerodynamic forces by means of computational fluid dynamics (CFD) simulation for full-scale or reduced model geometries under constant cross-wind conditions. At the present time there are no acknowledged international standards concerned with CFD drag prediction under unsteady on-flow conditions for rail vehicles. It is therefore useful to develop sufficiently accurate CFD models to assist in the study of these flows, particularly with respect to operational cost and safety requirements.

The current work extends an earlier result [3] which compared numerical estimates of drag against wind tunnel measurements. Two approaches are used for the CFD model. Approach D (illustrated in Figure 1) is discussed in [3]. This CFD model uses a computational geometry which closely matches the geometry of the experimental facility. Accurate geometrical descriptions of the wind tunnel nozzle, test section and diffusor sections are used, as well as the active and passive suction devices used in the experiments. A complete description of the 3D geometry of the moving belt, including rollers, is also included in the model. Gaps between the test section environment and the laboratory space, required to correctly manage pressure build-up with an operational moving belt, are also included. Approach C is illustrated in Figure 2. This approach removes many of the modeled components required for Approach D. Only the internal surfaces of the wind-tunnel nozzle and test section are needed. A slip wall boundary condition, due to [4], is applied upstream of the model to mimic the influence of both passive and active suction on the boundary layer approaching the model. The moving belt is modeled as a wind tunnel wall surface patch, under the model, with moving wall boundary conditions. The boundary patches used to mimic the influence of boundary layer suction as well as the moving belt are shown in Figure 2. For both approaches a moving flap system, located approximately in the middle between passive and active suction upstream of the model, is used to control the unsteadiness of the on-flow approaching the model.

After discussion of the CFD validation procedures used in this work, computed aerodynamic drag forces acting on the model under steady and unsteady on-flow conditions are evaluated for both approaches and compared against the experimental measurements. The upstream flap frequencies are varied while holding the bulk wind tunnel velocity constant. RANS approaches are compared against hybrid RANS-LES methods. The meshes used in the current work are designed to properly resolve unsteady velocity fields for these hybrid methods. The influence of the numeric schemes chosen on the solution quality are then reviewed, which allows us to provide a number of provisional recommendations regarding a modeling approach fulfilling a set of relevant objectives.

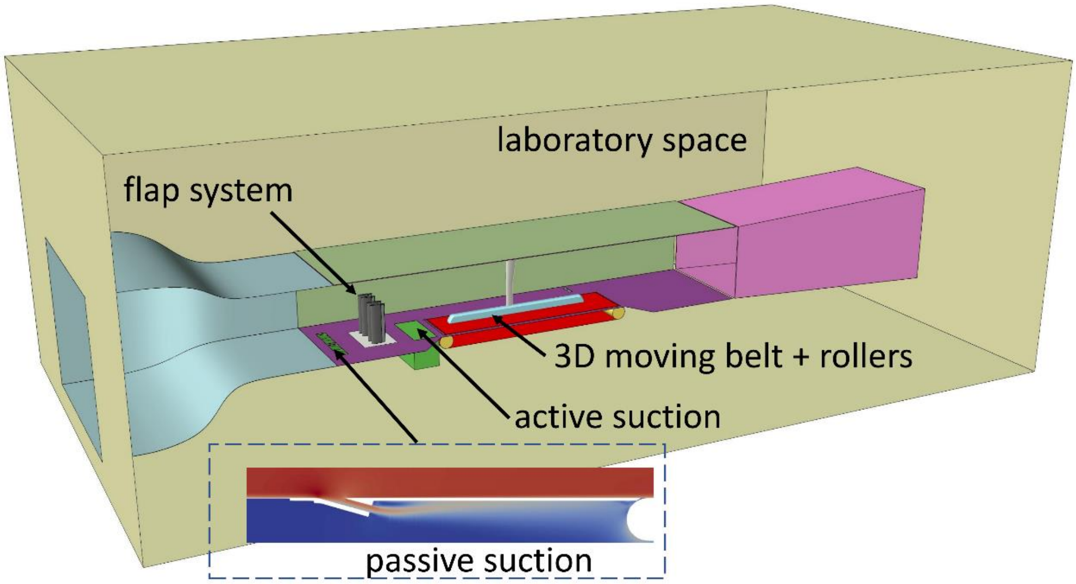

Figure 1: Approach D models the wind tunnel nozzle test section and diffuser embedded inside the laboratory space. The active suction outflow is prescribed as a constant mass flow outlet, while passive suction is provided by a duct system discharging directly into the laboratory space. The outflow from the passive suction system is illustrated in the subfigure above

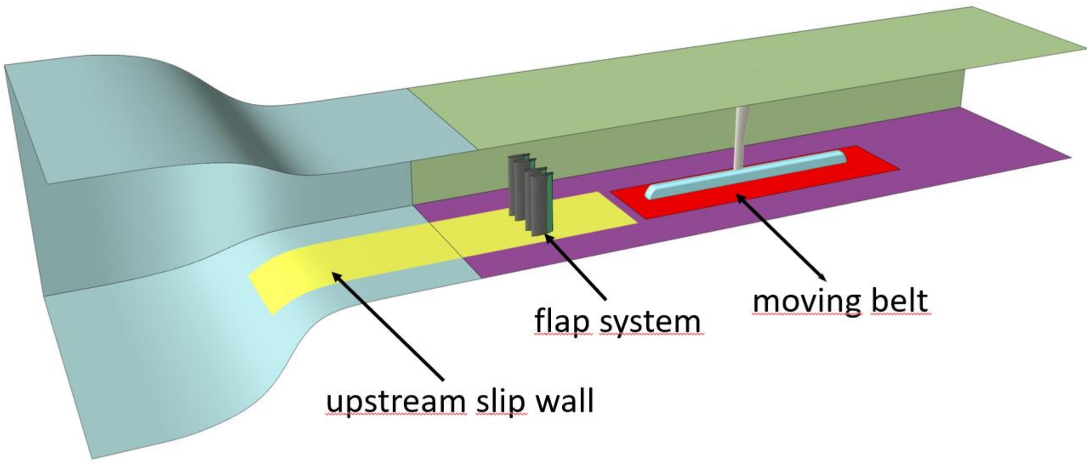

Figure 2: Approach C uses a simplified CFD model including only the wind tunnel nozzle and test section, as well as a simplified moving belt. The effect of suction devices seen in D is mimicked by defining a partial area of the nozzle flow as a slip wall (yellow area). Note that the model and the mounting sword are identical for both approaches

[1] https://verkehrsforschung.dlr.de/en/projects/ngt-cargo,

[2] DIN EN 14067-6:2018, Railway applications – Aerodynamics - Part 6: Requirements and test procedures for cross wind assessment, September 2018

[3] K.A. Weinman and Ehrenfried, K. (2022) *Unsteady inflow effects on model train drag.* In: New Results in Numerical and Experimental Fluid Mechanics XIV, Contributions to the 23rd STAB/DGLR Symposium Berlin, Germany 2022. A. Dillmann, G. Heller, E. Krämer, C. Wagner and J. Weiss (Ed.), Springer, 2024

[4] Wallin, Stefan: Kungliga Tekniska Hoegskolan (private communication)