Mitteilung

Projektgruppe/Fachkreis: Technische Strömungen

An Assessment of the Aerodynamic Characterization of a Compact Car Driving behind a Heavy Vehicle using Computational Fluid Dynamics

K. A. Weinman[1*], H. Wilhelmi[1], J. R. Bell[1], D. Heine[1] and C. Wagner[1,2]
[1]DLR Göttingen, Institute of Aerodynamics and Flow Technology
Bunsenstr. 10, 37073 Göttingen, Germany
[2]Technical University of Ilmenau, Institute of Thermodynamics and Fluid Mechanics
Helmholtzring 1, 98693 Ilmenau, Germany
*keith.weinman@dlr.de

In collaboration with the University of Stuttgart and the Research Association of Automotive Technology (FAT), a series of on-road [1] and wind tunnel experiments [2] were conducted in order to characterize the on-flow conditions for a vehicle driving behind another vehicle under real-world conditions. In conjunction with these experiments a CFD study was undertaken for both the on-road and wind tunnel measurements. This study, the subject of the paper, assesses the ability of two different CFD approaches to reproduce both the observed on-flow conditions as well as the aerodynamic drag of the vehicle. Both sets of CFD calculations are performed within the framework of the OpenFOAM [3] computational library.

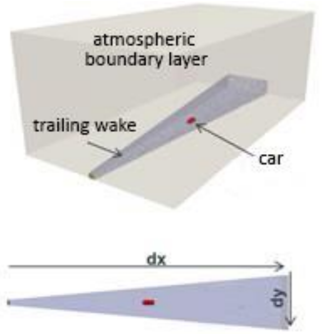

Figure 1: The computational domain used in approach A is illustrated in this figure. A highly refined wake core is used.

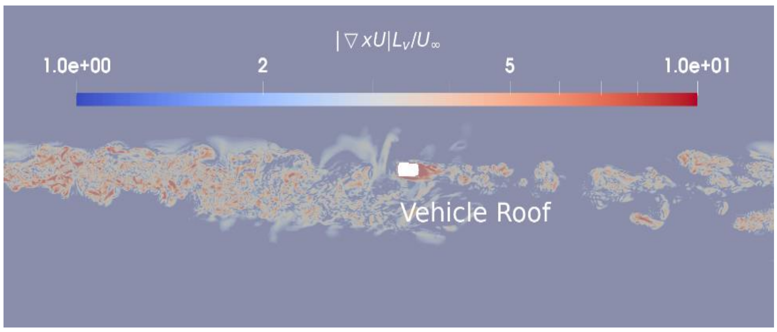

Figure 2: The figure shows the vorticity magnitude (normalized with the flow convective time scale) that has been generated using approach A. The flow is highly unsteady in the trailing wake region. Note that in this study the ABL turbulence is not finely resolved.

The first approach (A) views the flow problem as an inner wake core region embedded within the atmospheric boundary layer (ABL). This concept is illustrated in Figure 1. Radial basis function interpolation methods [4] are used to reconstruct on-flow velocity fields for the inner wake and ABL regions using on-road velocity measurements made with an array of static five-hole-probes. A typical distribution of the computed vorticity magnitude (normalized using the flow convective time scale), obtained using this approach, is shown in Figure 2. The flow, traveling from left to right, contains a significant fraction of the unsteady character embedded in the real-world flow. The second approach (B) uses OpenFOAM moving mesh algorithms to simulate motion of a set of flaps oscillating about the vertical axis of the wind tunnel in order to perturb the on-flow. The computational geometry is shown in Figure 3. The flaps are mounted upstream of a 1:4 scale model (of the vehicle used in the on-road tests) inside a full-scale model of the Göttingen SWG facility. The amplitude and frequency for the flap motion are selected to match the Strouhal number of the scaled wind tunnel model with that of the full-scale vehicle at on-road conditions. The distribution of the normalized vorticity magnitude generated by this method is illustrated by Figure 4. The on-flow velocity is dominated by a single length and time scale and contains significantly less complexity than that seen in Figure 2. The reduced on-flow for this approach enhances the computational efficiency of the problem in comparison to the method A.

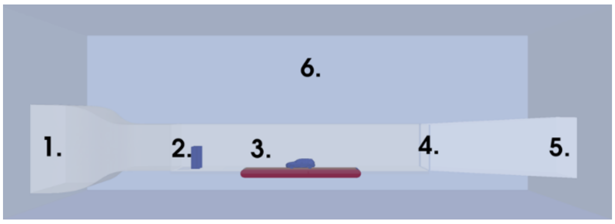

Figure 3: The computational domain used in the CFD study for the approach B. 1: wind tunnel inflow, 2: moving flaps, 3: moving belt, 4: pressure recovery gap, 5: outflow, and 6: laboratory plenum.

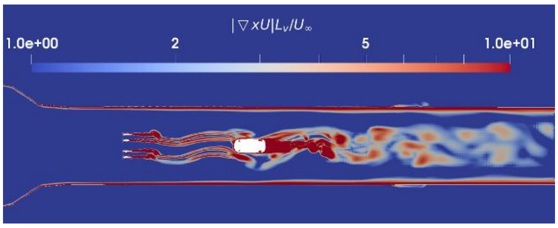

Figure 4: The figure shows the vorticity magnitude (normalized with the flow-convective time scale) that has been generated using approach B.

For both approaches the hybrid scale-resolving turbulence model (Delayed Detached Eddy Simulation) satisfactorily predicts the vehicle/model surface pressure coefficient distribution and integral drag coefficient. Approach A slightly underpredicts the inflow velocity magnitudes due to the inability of the reconstruction method to account for all of the velocity scales present in the real-world flow. Numerical dissipation also plays a role. Both approaches can reproduce a significant fraction of the unsteady dynamics observed in real-world conditions. This is seen in Figures 5 and 6 which show a good agreement for the computed and measured flow frequency content.

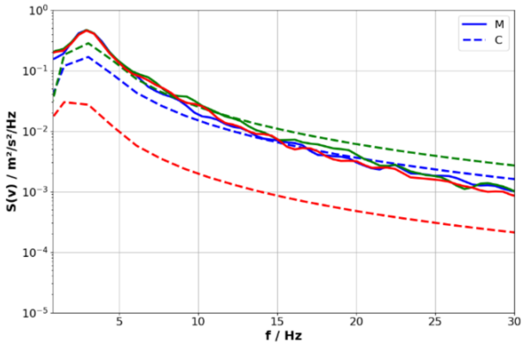

Figure 5: Using approach A, the computed (C) and measured (M) spectra for the v component of velocity match in terms of the frequency distribution. However, the computation underpredicts the magnitude of the spectral energy.

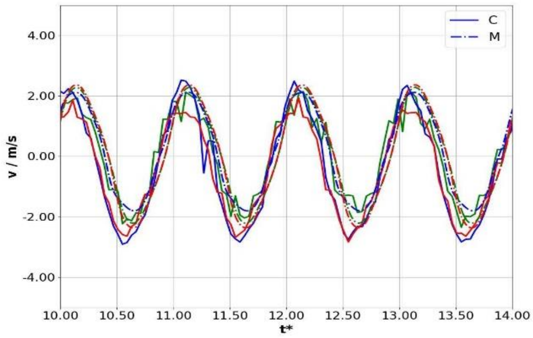

Figure 6: Using approach B, the computed C) and measured (M) v components match very well both in magnitude and in frequency.

In conclusion the CFD studies show that both approaches offer potential as tools to further the understanding of complex flow interaction about moving vehicles. The paper will overview the methodology implemented in this study and will discuss the advantages/disadvantages of both methods in detail.

[1] Wilhelmi, H. et al. (2020), "Aerodynamic Characterisation of a Compact Car Overtaking a Heavy Vehicle" in: Dillmann, A., Heller, G., Krämer, E., Wagner, C. et al. (eds.), New Results in Numerical and Experimental Fluid Mechanics XII, Notes on Numerical Fluid Mechanics and Multidisciplinary Design, Springer International Publishing, Cham, ISBN 978-3-030-25252-6:794–804

[2] Wilhelmi, H. et al. (2020), Simulation of Transient On-Road Conditions in a Closed Test Section Wind Tunnel using a Wing System with Active Flaps. SAE International Journal of Passenger Cars - Mechanical Systems (0688), Seiten 1-13. Society of Automotive Engineers. DOI: 10.4271/2020-01-0688 ISSN 1946-3995

[3] H. G. Weller, G. Tabor, H. Jasak, C. Fureby, "A tensorial approach to computational continuum mechanics using object-oriented techniques", COMPUTERS IN PHYSICS, VOL. 12, NO. 6, NOV/DEC 1998.

[4] T.C.S Rendall et al. (2010),"Evaluation of Radial Basis Functions for CFD Volume Data Interpolation", AIAA Paper 2010-1414, January 2010