Aerodynamic and Flexible Trucks for Next Generation of Long Distance Road Transport

EUROPEAN COMMISSION
Horizon 2020 | GV-09-2017 | Aerodynamic and Flexible Trucks
GA - 769658

Deliverable No.	AEROFLEX D3.2	
Deliverable Title	CFD simulations	
Deliverable Date	2020/06/30	
Deliverable Type	REPORT	
Dissemination level	CONFIDENTIAL	
Written By	Kamran Noghabai (SCANIA)	
Checked by	Per Elofsson (SCANIA) Date Rentema (DAF) Roy Veldhuizen (WABCO)	22.06.2020
Approved by	Ben Kraaijenhagen - Coordinator	23.06.2020
Status	Final	23.06.2020

Document information

Additional author(s) and contributing partners

Name	Organisation
Luca Miretti	CRF
Torbjörn Larsson & Petter Ekman	CREO
Roy Veldhuizen	WABCO
Onno Bartels	NLR
Per Elofsson	SCANIA

Document Change Log

Name	Date	Comments
2020/05/14		1st draft sent to WP3 partners
2020/06/04		2nd draft sent to AEROFLEX reviewers
2020/06/16		Final draft sent to Coordinator

Publishable Executive Summary

This document represents Deliverable D3.2 of the AEROFLEX project. It describes the CFD simulations performed to demonstrate the effect of various concepts, nominated in D3.1 (ref [1]), on aerodynamic drag for two types of heavy truck vehicle combinations, namely a Tractor-semitrailer and a Truck -dolly -semitrailer EMS1 (25.25m) combination. The results of the simulations are used as a basis for selection of suitable combinations of concepts to fulfil the prescribed Key Performance Indicators (KPIs).

The CFD simulation campaign included simulations of the two Baseline models, used as bases for the concept investigations, representing Tractor-semi-trailer and EMS 1 vehicles. These simulations were performed by all the partners participating in the CFD work, each using their own respective best-practice CFD method. Despite the different methods applied, the results showed generally good agreement in terms of the overall flow structure and most of the key features in the flow field. A comparison of change in drag when the boat-tail is removed, revealed reasonable spread between the results from different partners (Table 1-1).

Table 1-1 ΔC_D predictions by different partners, with different methods, for the CFD Baseline model with and without Boat-Tail at yaw -5˚

Partner	CFD SoftWare	Method	ΔC_D (cts)
Scania	PowerFLOW	Lattice Boltzmann (transient)	40
CRF	Helyx OpenFOAM 3.0.0	Finite Volume (transient)	40
NLR	OpenFOAM	Finite Volume (steady state)	29
WABCO	PowerFLOW	Lattice Boltzmann (transient)	36
Creo	OpenFOAM v1606	Finite Volume (steady state)	34

The different partners continued with simulation of the designated concepts, as individual measures on the applicable vehicle type, to evaluate the change in drag compared to the corresponding Baseline.
The simulated results for the geometry related measures, showed gains which were generally less than predicted in ref[1]. The reason can to some extent be attributed to that most of the concepts were not optimised for best performance. Still, the simulations showed that the investigated concepts constitute considerable potential for drag reduction on both vehicle configurations.
Simulations of the active flow control related concepts, however, didn't show any encouraging results. This is partly due to the difficulties related to simulating such devices, and partly because of the extensive iteration scheme required for each concept to arrive at an optimal solution, which was unfeasible to perform within the frame of this work package. During this work, a few number of new concepts, which are not listed in ref[1], were also developed and investigated, which add to the potential for improvement.

Since different concepts were investigated by different partners and thereby different CFD methods, the most promising candidates were simulated again, as established in advance, now using a uniform CFD method. These confirmation simulations showed the same trends and generally good agreement in levels of improvement, as the initial CFD simulations indicated. Many of the concepts were also optimised further during this process, adding to the potential gain by those concepts.

To demonstrate validity of the CFD results, extensive analyses were conducted, comparing CFD simulation results with wind tunnel measurements (ref[3]). This was done, quantitatively, in terms of drag prediction and more importantly, change in drag due to geometrical modifications, and also qualitative assessments of how well the flow field captured in the simulations matched measured data.
In addition to simulations of the full scale model in Open road condition, the validation analyses included new simulations with the 1:3 scale model in the FCA wind tunnel domain, and modelling minor geometrical discrepancies, to as far as possible replicate the measurement setup. Also, due to limitations in the wind tunnel, the applied speed for the scale model simulations differ from the full scale simulations.
The results of the simulations with different scales were then compared to each other, as well as to the equivalent wind tunnel measurements. The analyses showed in general satisfactory agreement between CFD simulation results of the 1:3 scale model in wind tunnel and the experimental data, especially in terms of ΔC_D trends and predictions, with very few exceptions. The discrepancies observed when comparing with the full scale

simulations (Figure 1-1), could mainly be traced to geometrical differences between the models and not having identical similarity parameters (Reynolds number) due to the limitation in wind speed.

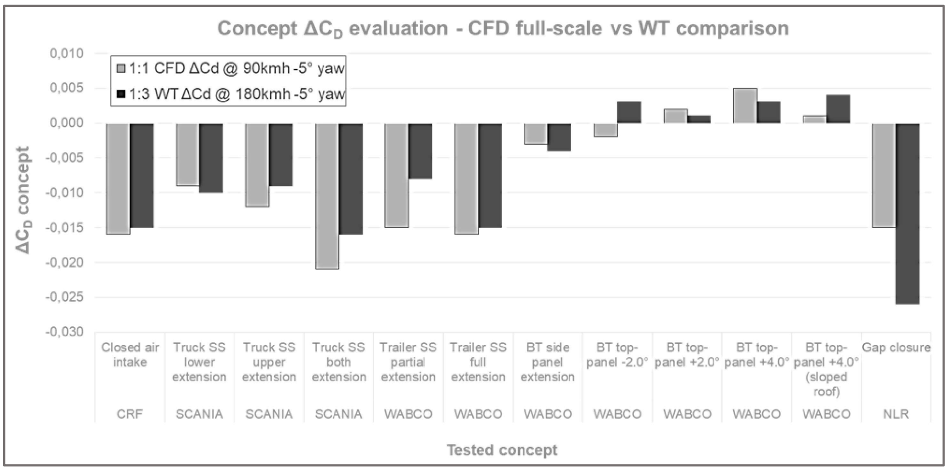

Figure 1-1 Comparison of ΔC_D due to concepts predicted by CFD full scale and measured data in wind tunnel

Additional simulations with the CFD scale model setup, to verify the effect of the tested concepts in combinations and individually, also showed good correlations between numerical and experimental data. Furthermore, acceptable correlations in the general flow field were observed between the CFD simulation results and experimental data, for various conditions and configurations.
Based on these observations, it is concluded that CFD simulation is a reliable tool for development and optimization of drag reduction concepts, as practiced in this work package.

The robustness of the CFD results was tested through series of sensitivity simulations, consisting both of variations in simulation strategies and various geometrical discrepancies. The analyses show variations of ΔC_D due to the applied changes, which were generally small in comparison to the predicted gain for the investigated concept, and more importantly, did not show a different trend. The comparison between transient and steady-state methods with the wind tunnel model showed the largest differences. However, most of the concepts investigated initially by transient analysis, were confirmed by the steady-state re-runs, as mentioned above, indicating good confidence in the applied process. The analyses are therefore considered to indicate satisfactory robustness and validity of the results, both in terms of the applied methodology and the choice of the concepts.

As the last step in this investigation, two sets of concept combinations were defined and simulated for each vehicle type, in order to demonstrate fulfilment of the KPIs for the baseline models stipulated in ref [2]. The concepts included in the combinations are those developed to give the largest improvement.

The configurations were simulated for four different yaw angles, for more correctly estimated, wind averaged C_D x A values. The resulting drag reductions for the different vehicle types, compared to corresponding Reference models, are presented in Table 1-2, showing improvements which by far exceed the stipulated target values. The margin to the target is noticeably higher for the EMS vehicle, due to the additional concepts which were not considered in the first evaluation presented in D3.1.

Table 1-2 Calculated wind averaged drag reduction times Area, compared to the Reference models, for the considered vehicle combinations

Case	Calculated ΔC_{DWA} A $[m^2]$	Calculated ΔC_{DWA} A [%]	Targeted ΔC_{DWA}A [%]
Tractor semi-trailer, Realistic	2.09	42	25
Tractor semi-trailer, Maximum	2.37	48	25
EMS 25.25m, Realistic	2.35	40	17
EMS 25.25m, Maximum	2.62	44	17

A good portion of the improvements presented in the table above, is of course due to the extended front of the truck, removal of side view mirrors and added boat-tail on the Baseline models.
A similar comparison, using the AEROFLEX Baseline models (with extended front and Boat-tail, and without rear view mirrors) as reference (Table 1-3), reveals the level of improvements purely due to the investigated concepts. As can be noted, the drag reductions are still considerable, and the improvements in % are virtually the same for both vehicle configurations.

Table 1-3 Calculated wind averaged drag reduction times Area, compared to the Baseline models, for the considered vehicle combinations

Case	Calculated ΔC_{DWA} A [m^2]	Calculated ΔC_{DWA} A [%]
Tractor semi-trailer, Realistic	0.867	23
Tractor semi-trailer, Maximum	1.152	31
EMS 25.25m, Realistic	1.071	23
EMS 25.25m, Maximum	1.336	29

In summary, it is concluded that the extensive work performed within WP3 of the AEROFLEX project clearly demonstrates CFD simulations to be a reliable tool for investigation and assessment of drag reduction measures for heavy trucks. Furthermore, the simulations of different concepts provide considerable reduction of drag fulfilling the prescribed target values for the vehicle types considered in this work package.