

Development of an Aerodynamic Analysis Methodology for Tractor-Trailer Class Heavy Commercial Vehicles

Kerem Anbarci and Berkay Acikgoz
AES (Aero Engineering Solutions)

Rustem A. Aslan
Istanbul Technical Univ.

Ozgur Arslan and R. Omur Icke
Ford Otosan AS

ABSTRACT

An aerodynamic analysis methodology which makes efficient use of ANSA and FLUENT software's in the aerodynamic design of tractor-trailer class heavy commercial road vehicles is presented. The aerodynamic drag coefficient of the truck is used as the main control parameter to evaluate the performance of the methodology. Analysis methodology development activities include determining optimal FLUENT software analysis parameters for the defined problem (RANS based turbulence models, wall boundary layer models, solution schemes) and the necessary ANSA mesh generation parameters (boundary layer number and growth rate, wall surface mesh resolution, total mesh resolution).

Proposed methodology is first constructed based on CFD simulations for the zero-degree yaw angle case of the 1/8 sized GCM geometry. The present results are within 1% of the experimental data. The know-how and experience gained, then, used to define an optimal analysis methodology for GCM test model with the optimal analysis and mesh generation parameters. Finally, the optimal analysis methodology of the GCM model is adapted to a recent Ford heavy vehicle geometry. The FORD heavy road vehicle CFD result is within 1% of the vehicle test result. Detailed calculations are obtained within 2 days using the FORD HPC system. Somewhat less accurate results may be obtained within few hours' time using the same HPC resources.

CITATION: Anbarci, K., Acikgoz, B., Aslan, R., Arslan, O. et al., "Development of an Aerodynamic Analysis Methodology for Tractor-Trailer Class Heavy Commercial Vehicles," *SAE Int. J. Commer. Veh.* 6(2):2013, doi:10.4271/2013-01-2413.

INTRODUCTION

Road vehicle drag coefficient (C_D) is one of the most important vehicle design parameter. It represents the aerodynamic effects encountered by the vehicle during its trip [1]. C_D increases with the square of the vehicle's travel speed. Therefore, reducing the drag value is of great importance as the vehicle's speed increases. This is directly related to the fuel consumption. Therefore, even a very small reduction in the drag coefficient of the road vehicle translates to a great savings regarding its cumulative fuel economy. In this respect, Reynolds Averaged Navier-Stokes (RANS) based CFD methods offer an affordable means to reduce the testing

efforts for the prediction of the vehicle's drag coefficient [2, 3].

In this study, a general aerodynamic analysis methodology based on commercial software ANSA [4] and FLUENT [5], used within the Ford Otosan Company, is developed to be used in the efficient design process of Tractor-Trailer Class Heavy Commercial Road Vehicle's. For the performance evaluations of the developed methodology, vehicle drag coefficient is selected as the basic control parameter. During the course of the developments, extensively wind tunnel tested and documented Generic Conventional Model (GCM) generic truck model [6] is selected as the test case to obtain optimal analysis parameters with high quality numerical predictions using FLUENT

employing ANSA generated meshes. The optimal parameters determined are those related to the mesh (such as the number of mesh layers and the expansion ratio within the boundary layer, surface mesh resolution, total mesh resolution), the most suitable turbulence model, the most suitable wall boundary layer modeling approach, solution algorithm and the computational accuracy level. Moreover, optimizing the overall mesh resolution is of great importance for the optimization of the computational time.

The flow field obtained around the 1/8 scale GCM at zero degree yaw angle using FLUENT based CFD analysis is compared with the available experimental data [6]. Using the experience obtained and evaluations carried out based on these predictions and comparisons, an optimal aerodynamic analysis methodology, describing optimal analysis and mesh generation parameters, for GCM model at zero degree yaw angle is finalized. The difference between the wind tunnel tests and predicted results obtained using the developed methodology is less than 1%.

The analysis methodology developed based on the GCM is adapted to a real, full-scale heavy vehicle, namely Ford Cargo [7], developed by Ford. The results obtained for Ford Cargo are compared with “coast down” test data. The test procedure and related accuracy is explained in [8, 9]. As a result of this additional work, an optimal analysis methodology for the Ford Tractor-Trailer Class Heavy Commercial Road Vehicle's has been developed. The methodology developed for the full-scale vehicle differs from that of the GCM model in such a way that all the geometric details present in the underbody and on the sides of the vehicle are taken into account and the rotation of the wheels and relative moving of the ground is also modeled. As a result of this realistic modeling, the number of mesh elements and the computation time is increased, considerably. When the HPC system present in Ford is employed, converged results are obtained within 2-3 calendar days. The difference in results between the real vehicle methodology and the “coast down” [8] test with the actual vehicle is less than 1%. For a vehicle speed of 80kmph, CFD Predictions within 4% of the coast down test are required [8]. If faster results are needed, coarser boundary layer sections with non-moving wheels and ground may be modeled to obtain approximate predictions. However, for best test matching predictions all the realistic effects must be modeled.

PARAMETRIC ANALYSIS OF GCM

General Information on the GCM Generic Model

NASA-Ames Research Center developed the 1/8 scale GCM geometry in order to represent the conventional heavy vehicle-trucks in the wind tunnel experiments [6]. In Figure 1 the GCM geometry in the 12-Ft pressurized wind tunnel is shown.

This geometry is used in many CFD simulation studies for validation. In order to increase precision of simulation results of tractor-trailer type heavy commercial vehicles, optimal parameters of mesh and solver algorithms must be defined. This can be done by detailed comparison of CFD results with experimental results. In this way the CFD methods are validated to be used for different geometries.



Figure 1. 1/8 scaled GCM geometry in the 12-Ft pressurized wind tunnel [6]

In this study, experimental results obtained in the NASA Ames 7×10 wind tunnel are used for validation purposes. The Reynolds number calculation is based on the trailer width of the model (0.324m) and is calculated to have a value of 1.15 million. The free flow Mach number is 0.15. The GCM geometry is positioned 13.33 cm away from the test section inlet. The boundary layer at the tunnel test section inlet had a thickness of 5.3 cm. The velocity distribution of the boundary layer is given in the test data. The turbulence intensity at the inlet is measured as 0.25%. The 1/8 scale model is approximately 2.5 m long, has a width of 0.3 m and a height of 0.5 m. The technical drawing of the GCM is shown in Figure 2. The dimensions are in “cm”.



Figure 2. GCM geometry from three different views

Parametric Analysis

Various analysis parameters and mesh configurations, given in Table 1, are systematically tested and evaluated (see App. I).

Table 1. Analysis parameters

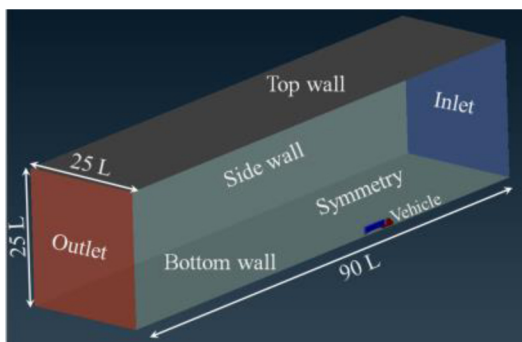
Parameter	Value	Description
Mesh resolution	Coarse(Low resolution), Medium, Fine (High resolution)	The main variation is the mesh resolution on the geometry surface. Near wall volume mesh is also changed for some cases.
Turbulence Model	k-ε, Realizable k-ε, RNG k-ε, k-ω SST, Spalart Allmaras	Selected in order to compare the Standard CFD turbulence models between themselves.
Wall boundary layer modeling	Standard Wall Function, Enhanced Wall Treatment	In order to compare these two models, the wall y^+ distribution must be suitable for each model. The first cell height and the layer number are changed for this purpose.
Discretization method	First-order, Second-order	For the sake of completeness, the effect of the discretization method on the results is investigated.
Solver	SIMPLE (Segregated) Coupled	For the sake of completeness, Pressure-Velocity Coupling effect on the result is investigated.

Computational Domain and Grid Details

Applied boundary conditions are given in [Figure 3](#). The domain boundaries are 5 vehicle lengths to the side and 10 vehicle lengths to the rear. The front distance of the domain is defined depending on the experimental test boundary layer formation.

In order to generate mesh around the GCM geometry, the fluid region is separated into 12 blocks. The model is analyzed with only one symmetrical half of the truck.

It is very important to define the first element height properly in order to get accurate results. This value must be selected according to the turbulence model used in the simulation. The first element height is calculated using the following formula.

**Figure 3. Boundary conditions of the analysis domain**

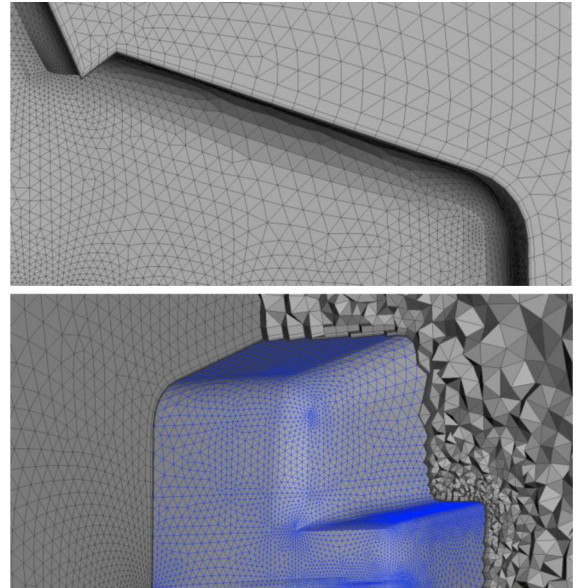
First element height Δy and boundary layer thickness δ [5];

$$\Delta y = L \Delta y^+ \sqrt{74} Re_L^{-13/14}$$

$$\delta = 0.035 L Re_L^{-1/7}$$

where L is the characteristic length. In this study L is equal to the width of the trailer (0.324 m). The boundary layer and

volume mesh structure close to vehicle surface are shown in [Figure 4](#).

**Figure 4. Boundary layer and volume mesh structure close to nose section of the vehicle**

In order to evaluate the mesh resolution effect on results, three different levels of mesh resolution are used; coarse, medium and fine. The near wall mesh elements on the vehicle surface are constructed with mixed mesh (structured and unstructured) and the outer volume elements are constructed with structured mesh elements. The surface mesh of the wheels and the truck has been selected as triangular due to highly curved geometries. The long trailer surfaces are meshed with quad elements. Details of different mesh resolution levels are given in [Table 2](#). Detailed comparison of “Enhanced Wall Treatment” and “Standard Wall Function” meshes is shown in [Table 3](#).

Table 2. Details of mesh resolution levels

			coarse	medium	fine
Surface Elements		Element Type	Element Number / min-max Element Length [mm]		
	Tractor	tria	38593/1-16	42010/1-9	146583/0.7-5
	Trailer	tria/quad	10786/4-21	23629/2-9	207224/1.5-4.5
	Wheel	tria/quad	14870/3-13	13042/4-8	47972/2-4
	Total		64249	78681	401779
Volume elements	Outer Region	hexa	289641/45-1100	1996516/15-700	2056816/15-700
	Vehicle Neighborhood	penta/tetra/pyramid	1537839/1-45	3348993/1-15	9893415/1-15
	Total		1827480	5345509	11950231
Boundary Layer	Element Type		penta/hexa		
	Element Number		642490	786810	4017790
	Number of Layers		10	10	10

Table 3. Details of wall treatment methods

Enhanced Wall Treatment vs. Standard Wall Function Mesh Characteristics			
		Enhanced Wall Treatment	Standard Wall Function
Surface Elements	Element Type	Element Number / min-max Element Length [mm]	
	Tractor	42010 / 1-9	
	Trailer	23629 / 2-9	
	Wheel	13042 / 4-8	
	Total	78681	
Volume elements	Element Type	Hexa	1996516 / 15-700
	Outer Region	Hexa	1996516 / 15-700
	Vehicle Neighborhood	penta/tetra / pyramid	3348993 / 1-15
	Total	5345509	4855578
Boundary Layer	Element Type	penta/hexa	penta/hexa
	Element Number	786810	237081
	Growth Rate	1,15	1,15
	Number of Layers	10	3

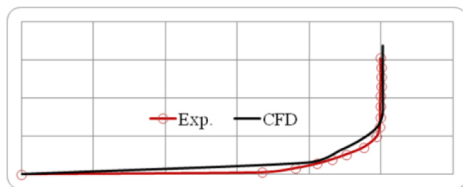
GCM Results and Discussions

Three Dimensional (3D) viscous flow simulations of 18 different cases given in App I carried out with steady, incompressible assumptions. The SIMPLE (segregated) solver is used in order to obtain the results after a minimum of 5000 iterations where the convergence criteria is selected as both the RMS values and the drag coefficient (0.1% tolerance of oscillation is allowed).

Table 4. Flow conditions

Reynolds number	1,15x10 ⁶
Mach number	0,15
Boundary layer thickness	5,3cm
Inlet turbulence intensity	%0,25
Inlet-Outlet static pressure value	1 atm

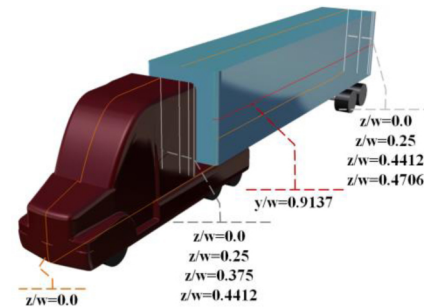
In order to make a consistent comparison between experimental data and the CFD results the inlet flow is modeled to be identical to the wind tunnel test. The flow conditions of the analysis are shown in Table 4. The wind tunnel boundary layer data [6] and the CFD data of the exact same location is compared in Figure 5.

**Figure 5. Experimental and analysis boundary layer velocity distribution comparison**

Experimental data

The experimental results documented in [6] are used for validation of numerical predictions. The dimensionless value of the CFD drag coefficient is calculated in the same way as the experimental values. The vehicle sectional area is 0.1554

m2. The sections where the CFD pressure coefficients are extracted are shown in Figure 6. The Predicted Cp values for the cases given in Table 9 compare well with the data [6] (see Appendix II).

**Figure 6. Pressure measured sections of numerical solution**

Effect of mesh resolution and turbulence model

The base configuration #1, and the related configurations #5 to #18 given in the analysis matrix (App. I) are simulated and the results are compared with each other in different combinations. The calculated drag coefficients comparison is given in Table 5. The percentage values indicate the percentage error of numerical predictions with experimental data [6].

Effect of wall boundary layer modeling

The comparison of configurations #1 and #3 points out the effect of “Enhanced Wall Treatment” and “Standard Wall Function” boundary layer models on analysis results. The “Standard Wall Function” modeling approach could be used for the “medium” mesh resolution case after some modifications are done. The comparison of calculated drag coefficients are given in Table 6. The Percentage error is defined as (exp: test, calc: CFD prediction)

$$\%Error = 100 * (C_{Dexp} - C_{Dcalc}) / C_{Dexp}$$

Table 5. Effect of mesh resolution and turbulence model on C_D prediction

Turbulence Model	Mesh resolution		
	Coarse (#14,15,16,17,18)	Medium (#1,5,6,7,8)	Fine (#9,10,11,12,13)
%Error of Calculated (C _D) Values			
Spalart – Allmaras (#18,8,13)	(18,35%)	(15,03%)	(12,93%)
Standard k-ε (#15,5,10)	(20,57%)	(16,70%)	(15,28%)
RNG k-ε (#16,6,11)	(8,22%)	(6,65%)	(4,00%)
Realizable k-ε (#14,1,9)	(9,84%)	(4,84%)	(-0,84%)
k-ω SST (#17,7,12)	(13,46%)	(9,05%)	(5,97%)