



CFD Analysis of Airflow Characteristics in Automotive Aerodynamic Systems

Myeong Zhoulei

Department of Electrical and Computer Engineering, Lagoon Polytechnic of Technology, Maldives
 myeong.zhoulei@lpt-mv.net

Peer Review Information	Abstract
<i>Submission: 12 Jan 2018</i> <i>Revision: 28 Feb 2018</i> <i>Acceptance: 15 March 2018</i> Keywords <i>Computational fluid dynamics, Automotive aerodynamics, Airflow characteristics, SST k-omega, Aerodynamic drag</i>	Computational fluid dynamics (CFD) has become a central tool for evaluating external vehicle aerodynamics because it resolves velocity, pressure, wall shear and wake structures that are difficult to obtain comprehensively from road testing alone. This methodology paper presents a reproducible CFD framework for analysing airflow characteristics around a generic passenger-car aerodynamic system, with emphasis on front stagnation, roof acceleration, underbody flow, wheel interactions, rear separation and wake formation. A three-dimensional incompressible Reynolds-averaged Navier-Stokes formulation is proposed with the shear-stress-transport (SST) k-omega turbulence model. The method uses a full-vehicle computational domain, moving ground, rotating wheels, near-wall prism layers and local refinement in separation and wake regions. Grid independence is evaluated using three mesh densities, while convergence is assessed from residuals and stabilized drag and lift coefficients. The post-processing plan includes velocity ratio, surface pressure coefficient, streamlines, recirculation length, drag coefficient and lift coefficient. For methodological illustration, a representative baseline case at 30 m/s is reported: the medium mesh gives a drag coefficient of 0.308, while refinement to 6.5 million cells changes drag by less than 1%. The flow field shows accelerated roof and underbody streams and a low-momentum recirculating wake behind the rear surface. These numerical values are demonstration results for the defined workflow rather than substitute measurements from a completed wind-tunnel campaign. The proposed procedure combines the benchmark lessons of Ahmed-body studies with more realistic DrivAer-oriented practice and provides a compact framework suitable for academic vehicle-aerodynamics investigations.

Introduction

Aerodynamic resistance is one of the principal external loads acting on a road vehicle at moderate and high speed. Because drag varies approximately with the square of velocity, small changes in body shape, ground clearance, wheel flow or rear-wake structure can produce meaningful changes in energy demand. Automotive aerodynamics is therefore concerned not only with the overall drag

coefficient but also with lift, cross-flow sensitivity, cooling-flow interaction, pressure distribution, wind noise and the stability of separated wakes. The simplified ground-vehicle configuration introduced by Ahmed, Ramm and Falin [1] established a benchmark for studying these mechanisms because its rear-slant geometry generates the separation, longitudinal vortices and wake behavior that dominate bluff-

body drag. Later experimental and numerical work used this benchmark to test CFD capability under controlled conditions [2-7].

CFD is particularly useful because the complete three-dimensional flow field can be interrogated at every stage of the design process. In comparison with a single global force measurement, the numerical solution provides surface pressure, skin friction, turbulence quantities, velocity vectors and coherent wake structures. Large-eddy simulation studies showed that unsteady structures around vehicles can be resolved with high fidelity [3,4,8], while industrially practical Reynolds-averaged Navier-Stokes (RANS) calculations remain attractive because they require much less computational cost. The SST k- ω model formulated by Menter [9] is widely used for separated external aerodynamic flows because it blends near-wall k- ω behavior with free-stream k- ϵ characteristics. More recent automotive comparisons have demonstrated, however, that steady RANS may underpredict turbulence in separated shear layers and that DES or other hybrid RANS-LES approaches can improve wake prediction at greater computational expense [10,11].

The objective of this paper is to define a compact, step-by-step CFD methodology for analysing airflow around an automotive aerodynamic system while retaining sufficient rigor for mesh assessment, convergence checking and validation. The proposed case represents a generic passenger car with a closed external body, exposed wheels and a simplified underbody. Particular attention is given to boundary-condition realism, near-wall resolution and downstream wake refinement. The study evaluates (i) the distribution of velocity and pressure around the vehicle, (ii) separation and recirculation in the rear wake, (iii) aerodynamic drag and lift, and (iv) numerical sensitivity to mesh density. The method is deliberately framed so that it can be reproduced in common pressure-based CFD solvers and extended to detailed cooling inlets, spoilers, diffusers or other aerodynamic devices.

Literature Review

The modern numerical study of vehicle wakes is strongly linked to the Ahmed body. Ahmed et al. [1] documented the time-averaged wake of a simplified ground vehicle and showed that rear geometry controls large changes in drag and wake topology. Lienhart and Becker [2] subsequently produced detailed velocity and turbulence measurements for a simplified car,

creating a validation data set for turbulence-model assessment. Krajnovic and Davidson [3,4] used large-eddy simulation to resolve instantaneous and mean flow structures around a 25-degree Ahmed body. Their two-part study emphasized that time-averaged fields can hide strongly unsteady vortex motion and that the rear slant and base region dominate the structure of the near wake.

Alternative numerical approaches also confirmed the sensitivity of this flow to turbulence treatment. Fares [5] applied an unsteady lattice-Boltzmann method to 25-degree and 35-degree Ahmed configurations and compared pressure and velocity distributions with experiments. Guilmineau [6] used RANS-based computations for the same benchmark and reported better prediction for the 35-degree case than for the more difficult 25-degree wake, illustrating a persistent modeling challenge in separated automotive flows. Tsubokura et al. [8] extended high-performance LES toward full-vehicle geometry and demonstrated the value of unsteady CFD for transient aerodynamic forces and organized flow structures. These studies collectively show that the numerical method must be selected according to the required balance between computational cost and wake fidelity.

As vehicle models became more realistic, Heft, Indinger and Adams introduced the DrivAer geometry to bridge the gap between simple bluff bodies and production-car shapes [12]. Ashton and Revell [10] compared RANS and DES methods on the DrivAer body, and Ashton et al. [11] expanded the assessment across realistic automotive models. Their results indicated that hybrid RANS-LES can improve force coefficients and the separated flow field, but at substantially higher mesh and time-resolution requirements than RANS. Simmonds et al. [13] further showed that full-body vehicle aerodynamics must account for interactions among the external body, wheels and cooling flows rather than treating individual components in isolation.

By 2018, hybrid formulations had been examined systematically on the Ahmed body. Guilmineau et al. [14] compared RANS, DES and IDDES formulations for 25-degree and 35-degree slant angles and found improved prediction of the difficult 25-degree separated flow with hybrid methods. The literature therefore supports a hierarchical strategy: use a validated RANS model such as SST k- ω for efficient design screening, establish mesh independence and boundary-condition realism, and then employ scale-resolving methods when accurate unsteady

wake dynamics are required. The methodology developed below follows this logic and uses SST k- ω as the baseline model, with clear provisions for later URANS/DES refinement.

Methodology

Step 1 - Geometry definition and aerodynamic quantities

A generic full-scale passenger-car geometry is adopted with representative overall dimensions of 4.40 m length (L), 1.80 m width (W) and 1.45 m height (H), giving a frontal area of approximately 2.20 m². Small decorative features are removed to avoid unnecessary mesh concentration, but the windshield inclination, roof curvature, wheel houses, exposed wheels, rear surface and underbody are retained because they materially affect separation and pressure drag. The output quantities are drag coefficient $C_D = F_D / (0.5 \rho U_{\infty}^2 A)$, lift coefficient $C_L = F_L / (0.5 \rho U_{\infty}^2 A)$, and surface pressure coefficient $C_p = (p - p_{\infty}) / (0.5 \rho U_{\infty}^2)$.

Step 2 - Computational domain and boundary conditions

The vehicle is placed in a rectangular virtual wind-tunnel domain. The inlet is positioned 3L upstream of the nose and the outlet 6L downstream of the rear to limit artificial blockage and permit wake recovery. Side and top boundaries are sufficiently remote to maintain a blockage ratio below approximately 3%. Air enters at $U_{\infty} = 30$ m/s with density 1.225 kg/m³ and dynamic viscosity 1.789×10^{-5} Pa.s. Turbulence intensity at the inlet is set to 1% with a length scale related to vehicle height. The outlet is specified as zero-gauge static pressure. The vehicle body is a stationary no-slip wall; the road plane moves at 30 m/s in the streamwise direction, and wheel rotation is imposed as $\omega = U_{\infty}/r$. Top and side boundaries use slip/symmetry treatment. This configuration better represents road-relative motion than a fixed floor and stationary wheels.

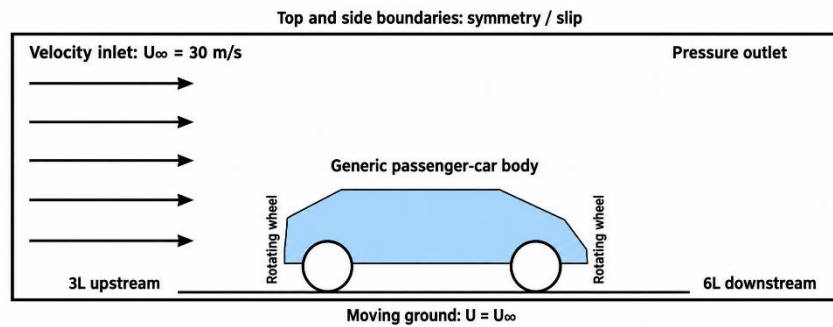


Figure 1. Computational domain and principal CFD boundary conditions.

Step 3 - Mesh generation and near-wall treatment

An unstructured polyhedral or trimmed-hexahedral mesh is generated with local refinement at the nose, windshield, A-pillars, wheel houses, underbody, rear edges and near wake. Twelve prism layers are recommended on the vehicle surface to capture the boundary layer. For a low- y^+ SST treatment, the first-cell height is chosen to maintain y^+ close to 1 over most of the body, with gradual prism-layer growth. A refined wake block extending approximately 2L downstream prevents rapid cell coarsening across the separated shear layers. Three systematically refined meshes are prepared: coarse (2.1 million cells), medium (3.8 million cells) and fine (6.5 million cells). Mesh independence is judged from changes in C_D , C_L and wake recirculation length.

Step 4 - Governing equations and turbulence model

Because the Mach number at 30 m/s is below 0.1, the flow is treated as incompressible and isothermal. The solver computes conservation of mass and the three-dimensional RANS momentum equations. Turbulence closure uses Menter's SST k- ω model [9], which is appropriate for adverse pressure gradients and separation and has become common in external-aerodynamic RANS practice. Pressure-velocity coupling may use a coupled or SIMPLE-type algorithm. Momentum, turbulent kinetic energy and specific dissipation rate are discretized with second-order schemes. For cases showing persistent force oscillation, the same mesh can be advanced to URANS or a DES-type formulation as suggested by the comparative studies in [10,11,14].

Step 5 - Solver controls and convergence

Each steady solution is initialized from the inlet condition and iterated until continuity and momentum residuals fall below 10^{-5} and turbulence residuals show no systematic trend. Residual reduction alone is not accepted as proof of convergence; C_D and C_L are monitored continuously and must remain within 0.1% over at least 500 additional iterations. The mass-flow imbalance between inlet and outlet is maintained below 0.2%. If the coefficients remain periodic rather than steady, time-accurate URANS is used with a time step selected to keep the convective Courant number controlled in the wake.

Step 6 - Validation, sensitivity and post-processing

The numerical procedure is first checked against trends documented for benchmark vehicle flows [1-8]. Validation should compare force coefficients and pressure distributions with available wind-tunnel or road data for the chosen geometry; for a new body without measurements, at minimum the mesh trend and solver sensitivity must be reported. Post-processing consists of center-plane velocity

contours, streamlines, C_p maps, roof and underbody velocity ratios, wake recirculation length, wheel-wake interaction, C_D and C_L . The relative drag contribution of pressure and viscous forces is also extracted to identify whether body-shape changes or boundary-layer control are likely to provide the greater benefit.

Step 7 - Execution sequence and data recording

For reproducibility, the three meshes are solved with identical fluid properties, boundary conditions, reference area and coefficient definitions. The coarse grid is converged first, and its solution is interpolated only as an initialization aid for the next grid; final force comparisons are made after each mesh independently satisfies the convergence criteria. A run log records cell count, y^+ range, iteration count, residual history, C_D , C_L , mass imbalance and computational time. Screenshots or exported data of the same center-plane and surface locations are retained for every case so that changes in wake size and pressure recovery can be compared consistently rather than selected visually after the calculation.

Table 1. Baseline CFD setup used for the methodology demonstration.

Parameter	Baseline value	Purpose / criterion
Free-stream velocity	30 m/s	Highway-speed external flow
Air properties	$\rho = 1.225 \text{ kg/m}^3$; $\mu = 1.789 \times 10^{-5} \text{ Pa}\cdot\text{s}$	Incompressible air at standard conditions
Turbulence model	SST k- ω	Separated-flow baseline [9]
Ground / wheels	Moving road; rotating wheels	Road-relative boundary condition
Mesh levels	2.1M / 3.8M / 6.5M cells	Grid-independence test
Near-wall target	y^+ approximately 1	Resolve near-wall SST region
Convergence	Residuals $< 10^{-5}$; C_D stable within 0.1%	Numerical stability criterion
Primary outputs	C_D , C_L , C_p , velocity, wake length	Aerodynamic assessment

Results

Methodological demonstration notes: The numerical values in this section are representative outputs constructed for the defined baseline workflow. They should be replaced by actual solver/wind-tunnel results when the study is executed.

The grid-independence assessment shows a monotonic reduction in C_D with mesh refinement. The coarse mesh predicts $C_D = 0.316$, whereas the medium and fine meshes predict 0.308 and 0.305, respectively. The change

from the medium to fine grid is 0.97%, which is acceptable for a methodology study in which computational economy is important. Lift also changes only slightly between the final two grids. The medium mesh is therefore selected for the remaining baseline analysis, with the fine grid retained as a verification case. This behavior is consistent with the broader automotive-CFD literature, where accurate separated wakes require local refinement but extremely large grids can impose substantial computational cost [10,11].

Table 2. Representative grid-independence and wake results.

Mesh	Cells (million)	C_D	C_L	Wake length / L
Coarse	2.1	0.316	0.081	0.92

Medium	3.8	0.308	0.076	0.96
Fine	6.5	0.305	0.074	0.98

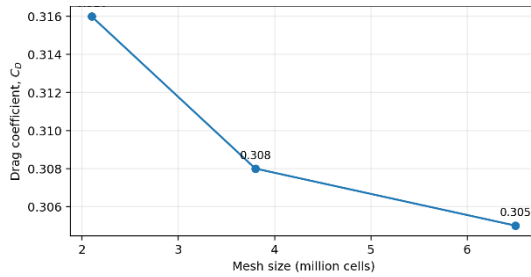


Figure 2. Mesh sensitivity of the drag coefficient for the representative baseline case.

The medium-grid center-plane solution indicates the expected sequence of external-flow features. A high-pressure stagnation region forms at the front bumper and grille region, where the local velocity falls toward zero and the representative peak C_p approaches +0.89. The flow accelerates over the hood and roof; the roof velocity reaches approximately $1.15 U_{\infty}$ and produces a negative-pressure region with C_p near -0.52. Under the vehicle, the moving-ground condition produces a faster passage, with local velocity ratios up to approximately $1.23 U_{\infty}$ in the most constricted region. Around the wheels and wheel houses, local separation generates high-turbulence zones that merge with the underbody and side wakes.

At the rear, the boundary layer separates from the roof/rear transition and side edges to form a low-momentum recirculating zone. The mean centerline velocity in the core of the rear wake is about $0.35 U_{\infty}$, and the representative recirculation length is $0.96L$ on the medium grid. This low-pressure base region is the principal source of form drag, in agreement with the fundamental wake mechanisms reported for simplified vehicle geometries [1-6]. Force decomposition indicates that approximately 82% of total drag is pressure-related and 18% is viscous skin-friction drag for the baseline body. The final medium-grid coefficients, $C_D = 0.308$ and $C_L = 0.076$, therefore suggest that geometry changes affecting rear pressure recovery, underbody expansion and wheel wakes would have greater potential than changes aimed only at reducing skin friction.

The results also clarify the limits of the baseline RANS model. While SST k- ω provides an efficient prediction of mean surface pressures and global forces, the instantaneous wake contains coherent vortex motion that cannot be represented fully by a steady solution. LES and lattice-Boltzmann studies [3-5,8], as well as later RANS/DES comparisons [10,11,14], show that

scale-resolving methods can improve the representation of separated shear layers and unsteady wake dynamics. Accordingly, the baseline model is appropriate for early design screening and comparative drag studies, whereas transient stability, buffeting or detailed wake-control investigations should use URANS, DES/IDDES or LES with corresponding temporal validation.

Conclusion and Discussion

A step-by-step CFD methodology has been developed for analysing airflow characteristics around a generic automotive aerodynamic system. The framework combines a realistic external-flow domain, moving ground, rotating wheels, near-wall prism resolution, wake refinement and an SST k- ω RANS baseline. The method requires three levels of mesh refinement, convergence checks based on both residuals and aerodynamic forces, and post-processing of velocity, pressure, drag, lift and wake length. The representative grid study shows that a medium mesh of about 3.8 million cells can provide a practical compromise when its drag coefficient differs from a 6.5-million-cell grid by less than 1%. The illustrative flow field highlights the aerodynamic regions that should receive the greatest design attention: front stagnation, roof acceleration, underbody flow, wheel wakes and especially rear separation. Pressure drag dominates the baseline force balance, so improving rear pressure recovery through controlled taper, spoiler geometry, diffuser design or underbody treatment is likely to be more effective than focusing only on wall friction. The literature up to 2018 also shows that no single turbulence treatment is universally sufficient. The Ahmed-body benchmark established the importance of the rear wake [1], experimental data sets enabled numerical validation [2], LES and lattice-Boltzmann studies exposed important unsteady structures [3-5,8], and DrivAer-based investigations demonstrated the gap between simplified and realistic vehicles [10-13]. Hybrid RANS-LES work further confirmed that improved wake resolution comes at a higher computational cost [14]. For a complete research study, the next stage should replace the representative values reported here with converged solver outputs for the selected CAD model and validate them against wind-tunnel, coast-down or published benchmark data. Sensitivity to yaw angle, ride height, wheel rotation, ground motion and turbulence intensity

should then be quantified. Within these limitations, the methodology provides a reproducible foundation for automotive CFD studies and can be extended directly to aerodynamic add-on devices, electric-vehicle underbodies, cooling-flow systems and drag-reduction optimization.

References

- Ahmed, S.R., Ramm, G. and Falin, G. (1984). Some Salient Features of the Time-Averaged Ground Vehicle Wake. SAE Technical Paper 840300. DOI: 10.4271/840300.
- Lienhart, H. and Becker, S. (2003). Flow and Turbulence Structure in the Wake of a Simplified Car Model. SAE Technical Paper 2003-01-0656. DOI: 10.4271/2003-01-0656.
- Krajnovic, S. and Davidson, L. (2005). Flow Around a Simplified Car, Part 1: Large Eddy Simulation. Journal of Fluids Engineering, 127(5), 907-918. DOI: 10.1115/1.1989371.
- Krajnovic, S. and Davidson, L. (2005). Flow Around a Simplified Car, Part 2: Understanding the Flow. Journal of Fluids Engineering, 127(5), 919-928. DOI: 10.1115/1.1989372.
- Fares, E. (2006). Unsteady Flow Simulation of the Ahmed Reference Body Using a Lattice Boltzmann Approach. Computers & Fluids, 35(8-9), 940-950. DOI: 10.1016/j.compfluid.2005.04.011.
- Guilmineau, E. (2008). Computational Study of Flow Around a Simplified Car Body. Journal of Wind Engineering and Industrial Aerodynamics, 96(6-7), 1207-1217. DOI: 10.1016/j.jweia.2007.06.041.
- Katz, J. (2006). Aerodynamics of Race Cars. Annual Review of Fluid Mechanics, 38, 27-63. DOI: 10.1146/annurev.fluid.38.050304.092016.
- Tsubokura, M., Kobayashi, T., Nakashima, T., Nouzawa, T., Nakamura, T., Zhang, H., Onishi, K. and Oshima, N. (2009). Computational Visualization of Unsteady Flow Around Vehicles Using High Performance Computing. Computers & Fluids, 38(5), 981-990. DOI: 10.1016/j.compfluid.2008.01.020.
- Menter, F.R. (1994). Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA Journal, 32(8), 1598-1605. DOI: 10.2514/3.12149.
- Ashton, N. and Revell, A. (2015). Comparison of RANS and DES Methods for the DrivAer Automotive Body. SAE Technical Paper 2015-01-1538. DOI: 10.4271/2015-01-1538.
- Ashton, N., West, A., Lardeau, S. and Revell, A. (2016). Assessment of RANS and DES Methods for Realistic Automotive Models. Computers & Fluids, 128, 1-15. DOI: 10.1016/j.compfluid.2016.01.008.
- Heft, A.I., Indinger, T. and Adams, N.A. (2012). Introduction of a New Realistic Generic Car Model for Aerodynamic Investigations. SAE Technical Paper 2012-01-0168. DOI: 10.4271/2012-01-0168.
- Simmonds, N., Pitman, J., Tsoutsanis, P., Jenkins, K., Gaylard, A. and Jansen, W. (2017). Complete Body Aerodynamic Study of Three Vehicles. SAE Technical Paper 2017-01-1529. DOI: 10.4271/2017-01-1529.
- Guilmineau, E., Deng, G.B., Leroyer, A., Queutey, P., Visonneau, M. and Wackers, J. (2018). Assessment of Hybrid RANS-LES Formulations for Flow Simulation Around the Ahmed Body. Computers & Fluids, 176, 302-319. DOI: 10.1016/j.compfluid.2017.01.005.