

Aerodynamic Investigation of Cooling Drag of a Production Sedan Part 2: CFD Results

Levon Larson, Ronald Gin, and Robert Lietz
 Ford Motor Company

ABSTRACT

Cooling drag is a metric that measures the influence of air flow travelling through the open grille of a ground vehicle on overall vehicle drag, both internally (engine air flow) and externally (interference air flow). With the interference effects considered, a vehicles cooling drag can be influenced by various air flow fields around the vehicle, not just the air flow directly entering or leaving the engine bay. For this reason, computational fluid dynamics (CFD) simulations are particularly difficult. With insights gained from a previously conducted set of experimental studies, a CFD validation effort was undergone to understand which air flow field characteristics contribute to CFD/test discrepancies. A Lattice-Boltzmann Large Eddy Simulation (LES) method was used to validate several test points. Comparison using integral force values, surface pressures, and cooling pack air mass flows was presented. The test points include cooling pack configuration changes, underbody shield changes and vehicle attitude changes. Select test points were compared in both static and moving ground wind tunnel conditions. This paper represents the second of a two-part effort, with experimental and numerical foci respectively.

CITATION: Larson, L., Gin, R., and Lietz, R., "Aerodynamic Investigation of Cooling Drag of a Production Sedan Part 2: CFD Results," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(1):2017, doi:10.4271/2017-01-1528.

INTRODUCTION

The current age of automotive production is bringing previously unseen combined requirements of high performance, high fuel economy, low emissions and high safety. To remain competitive, vehicles must achieve a harmony of these requirements while cost and weight are both minimized. Furthermore, development and production costs must be kept to a minimum while regulatory costs are at an all-time high. Clearly, timely and effective delivery of these requirements under the aforementioned pressures would be near impossible without the aid of computer aided engineering (CAE).

Of the challenging requirements listed above, fuel economy is one that can be characterized by the reduction of vehicle energy losses, whether mechanical or chemical. One of the prominent mechanical losses is aerodynamic drag, which increases exponentially with vehicle speed. The aerodynamic design of vehicles has been a widely covered topic and the general design guidelines have been covered extensively [1]. One particular sub-topic of automotive aerodynamics that remains difficult to design for is cooling drag. Cooling drag refers to the difference in drag coefficient (C_d) between the cases where a vehicle has the grille open and closed, as shown in [equation \(1\)](#).

$$C_{d_{cooling}} = C_{d_{open\ grille}} - C_{d_{closed\ grille}} \quad (1)$$

Cooling drag remains an elusive topic for several reasons. Flow patterns through the grille, heat exchangers, fan module and engine bay are all complex and typically, drag reduction is not the primary objective for components in these regions. As cooling drag is defined as the difference between two vehicle conditions, it describes the difference between two significantly different air flow fields. Because air flow that would have otherwise entered the grille must be diverted elsewhere in the closed grille condition, the external surfaces play a large role in the cooling drag term, this phenomena is called interference drag. Interference drag, depending on the vehicle, can occur on any of the external vehicle surfaces and can interact with external air flow patterns [2].

Kuthada et.al studied the interference effects and their effect on rear end shapes with and without moving ground conditions [3–4]. Wickern et.al investigated the interference effects on spoiler inclination angle with and without moving ground conditions [5]. Lai et.al presented numerous drag and lift trends between cooling drag and rear diffuser angle, ride height, cooling exit air flow inclination angle and exit location on a simplified body [6]. Tesch et.al conducted experiments with the cooling exit either somewhere on the hood, underbody or in the front wheel wells and showed that the interference effects were significantly different depending on the configuration [7].

Several methods (not considering interference drag) to mitigate cooling drag have been discussed in the literature, these include: grille opening optimization, cooling fan/shroud optimization, active grille shutters (AGS), and heat exchanger ducting [8, 9, 10, 11, 12, 13, 14]. These strategies rarely only affect cooling drag, uniformity of heat exchanger air flow or component thermal management may also be objectives.

The vehicle used in this study contains the AGS technology, herein AGS drag will be defined as shown in equation (2).

$$C_{d_{AGS}} = C_{d_{open\ vanes}} - C_{d_{closed\ vanes}} \quad (2)$$

Historically, due to the difficulty of predicting underhood air flow patterns and interference drag, attempts at modelling were limited to momentum balances through the engine bay [15, 16, 17]. Tesch et.al formulated a momentum equation that separates the cooling drag into four components: momentum flux difference through the control volume (engine bay), pressure forces at inlet and outlet surfaces and interference drag, though this method requires detailed pressure field information [7].

With CFD's potential ability to model air flow gradients around various complex geometries of various length and time scales, and with ever-increasing computational efficiencies, CFD has become the foremost tool used in the automotive community for cooling drag analysis. Alajbegovic et.al used a coupled aero-thermal approach to predict the top tank temperature of a Land Rover LR3 and Ford Mondeo [18–19]. Baeder et.al. used CFD to study the influence of heat exchanger resistance and cooling exit configuration on cooling drag for the SAE-Body and a quarter scale production representative vehicle with an interchangeable rear-end [20]. Christoffersen et.al determined that engine bay packaging density had an effect on external aerodynamics through a CFD and wind tunnel study on a scale model of the Volvo S60 [21]. Kubohara et.al used CFD to optimize the air flow uniformity approaching the radiator [13]. Kuthada et.al. CFD in conjunction with experiments to investigate the influence of road simulation, rear end shape and cooling exit location on cooling drag [3–4, 22]. Martini et.al. compared CFD and experimental results for top tank temperature and cooling drag under yaw conditions for a Volvo FH 4x2 truck [23]. Sengupta et.al. compared CFD and experimental results of cooling drag and grille air mass flow for a half-scale Volvo VNL and a full scale Class 8 Tractor Trailer [24]. Simmonds et.al. used CFD to study the effect of various heat exchanger resistances on cooling drag on a Jaguar XJ [25]. Zhang et.al. optimized for radiator air flow uniformity and cooling drag simultaneously by varying upstream ducting geometry and cooling grille openings with CFD [26].

EXPERIMENTAL/NUMERICAL SETUP

Model Details

The modelled vehicle is the 2013 Lincoln MKZ described in [27] and shown in Figure 1. Care was taken to model as many physical details of the test vehicle as possible, several component scans were required. The vehicle was modified such that several front end and underbody components could be altered for ease of model change while the vehicle was being tested. The vehicle was also fitted with measurement equipment for the purposes of gaining deeper insights into wind tunnel condition air flow field behaviour and for numerical validation. The effects of AGS sealing, AGS vane position, vehicle attitude, cooling pack configuration, engine bay blockage, cooling air flow exit paths and underbody shields on cooling and AGS drag were studied. In addition to force and moment coefficients, local pressures, heat exchanger air flow rates and underbody velocity magnitudes were measured for select cases. The vehicle was primarily tested in stationary ground conditions though select points were also tested in moving ground conditions as well.



Figure 1. 2013 Lincoln MKZ test vehicle, shown here in a moving ground test condition.

Numerical Details

All reported simulations were conducted with the commercial Lattice-Boltzmann code EXA PowerFLOW. Several validation reports exist, including those already mentioned [28]. The Lattice-Boltzmann method formulates conservation theorem for any quantity through collision of particles, for this study, mass and momentum conservation was used to describe the incompressible Navier-Stokes equations (energy conservation is also possible). Sub-grid scale energy dissipation is determined through a RNG k-ε model and a wall model is used to avoid computationally restrictive near-wall discretization.

All simulations were solved transiently over a physical time of 3.46 to 4.61s with cases of approximately 330 to 390 million volume elements (voxels). The smallest element size used was 1.25mm, a sample cut-section ($y = 0m$) is shown in Figure 2. The mesh strategy used in this study featured extensive resolution in known critical areas of cooling air flow. The vehicle was simulated under open-road conditions with distant far field boundary conditions. Wind tunnel interference/blockage effects were not taken into account for either of

the stationary or moving ground comparisons, for a description of the wind tunnels used to determine the experimental results refer to [27]. Non-deformed floor-intersecting tires were used for all simulations presented and a rotating wall boundary condition was used for both the wheels and tires for the moving ground simulations. The air flow resistance from the heat exchangers was modelled with Darcy's law with experimentally determined inertial and viscous resistance coefficients. The cooling module fans were assumed stationary for all simulations (in the experimental testing campaign, no appreciable difference in aerodynamic coefficients or local Cp values were seen if the fans were held stationary).

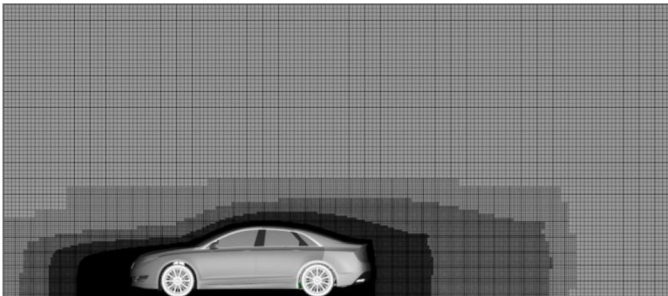


Figure 2. Y = 0m cut-section of mesh for a generic case.

RESULTS

All experiments and simulations were carried out under assumed adiabatic air flow conditions with a Reynolds number of 1.06×10^7 on a stationary ground unless otherwise mentioned. All AGS experiments refer to comparisons between fully open and fully closed AGS positions, intermediate AGS increments were not studied. Herein the baseline configuration refers to cases with the production level AGS system, production shields and the attitude set to the curb ride heights.

Vehicle Attitude

It was shown in a previous work that the vehicle attitude can have a significant effect on cooling and AGS drag [27]. Furthermore, it is possible that the wind load from the wind tunnel can cause the test vehicle to pitch up or down depending on the lift being generated. The test vehicle used for this study had its production suspension replaced with a rigid suspension in order to avoid this issue.

Figure 3 shows the comparison between experimental and CFD values for cases where the attitude was adjusted from the curb position. The largest test-CFD deviation for cooling drag was 0.005 Cd and 0.008 Cd for AGS drag and the trends were not predicted correctly with either. Aside from the final configuration, the CFD predicted cooling and AGS Cd deltas were insensitive to vehicle attitude changes, no change over 0.001 Cd was seen until the final configuration. It should be noted that the final configuration was a large enough attitude shift such that the pitch of the vehicle was reversed from positive to negative.

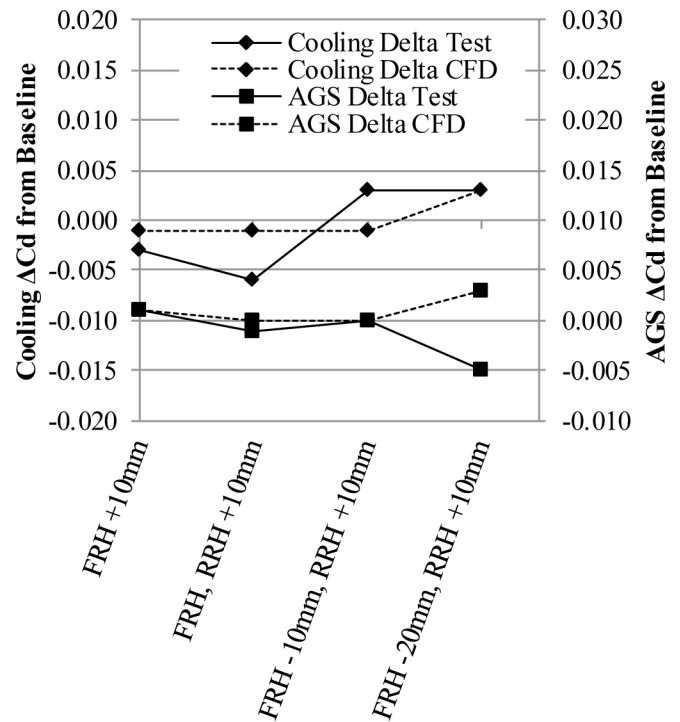


Figure 3. Cooling and AGS Cd for both test and experiment for the vehicle attitude study, shown as a delta from the baseline configuration (positive values refer to increases in drag). Front Ride Height (FRH), Rear Ride Height (RRH).

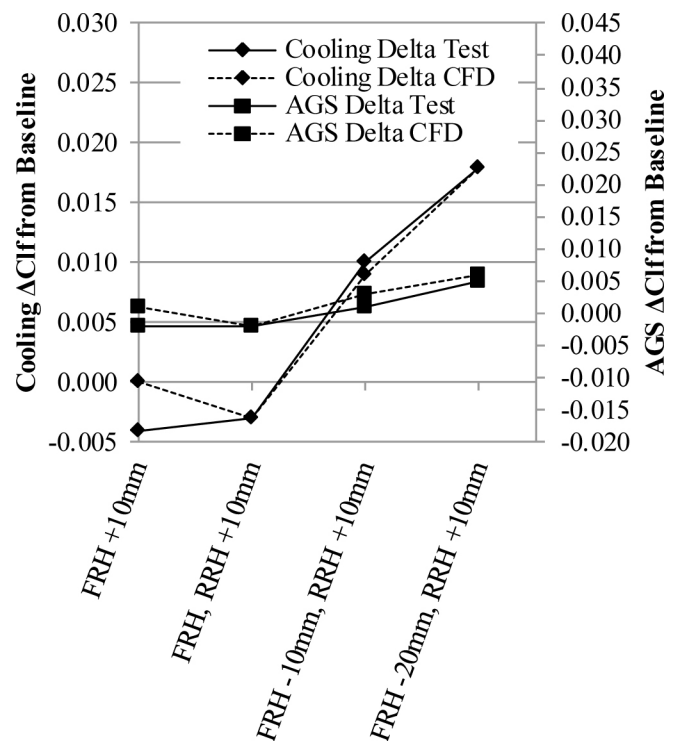


Figure 4. Cooling and AGS Clf for both test and experiment for the vehicle attitude study, shown as a delta from the baseline configuration (positive values refer to increases in lift). Front Ride Height (FRH), Rear Ride Height (RRH).