

The Effects of Detailed Tire Geometry on Automobile Aerodynamics – a CFD Correlation Study in Static Conditions

Todd H. Lounsberry and Mark E. Gleason
Chrysler LLC

Satheesh Kandasamy, Khaled Sbeih, Richa Mann and Bradley D. Duncan
Exa Corporation

Copyright © 2009 SAE International

ABSTRACT

A correlation study was performed between static wind tunnel testing and computational fluid dynamics (CFD) for a small hatchback vehicle, with the intent of evaluating a variety of different wheel and tire designs for aerodynamic forces. This was the first step of a broader study to develop a tool for assessing wheel and tire designs with real world (rolling road) conditions. It was discovered that better correlation could be achieved when actual tire scan data was used versus traditional smooth (CAD) tire geometry.

This paper details the process involved in achieving the best correlation of the CFD prediction with experimental results, and describes the steps taken to include the most accurate geometry possible, including photogrammetry scans of an actual tire that was tested, and the level of meshing detail utilized to capture the fluid effects of the tire detail. The effects of this scanned tire geometry were significant, and improved the results for drag, lift, and the cooling drag for the vehicle relative to experimental full scale testing.

INTRODUCTION

The end objective is to be able to evaluate a production vehicle wheel design for aerodynamic drag. A predictive tool for developing a good wheel design for aerodynamics is the goal.

Since very little literature or documentation exists on the subject, especially for production vehicles, past experiences have led to only general “rules of thumb” in the aerodynamic community, and a more in-depth scientific study and a solidly correlated development tool is needed.

However, the task is complex, as interactions with the vehicle body, tires, and ground plane result in flow field dynamics which are difficult to document and substantiate. In the interest of time and resources, a simple study for investigating the primary parameters of wheel-only effects on aerodynamic drag is proposed.

The study involved using computational fluid dynamics (CFD), with experimental data for correlation (to build confidence in the results).

In order to focus only on wheel parameters which lead to drag, a single vehicle body as well as one tire type was used in all of the iterations. Certainly the possibility exists for the selection of bodystyle to affect the results; however, for the purpose of correlating a predictive tool, the only requirement is that the vehicle body be consistent between the sets of results.

Selecting the wheel designs to be tested was more complicated. A “shot gun” approach of using various available production wheels was not preferred – too many variables. However, in order to complete this

study in a timely manner and also because it was felt that the focus should be on “sorting the large boulders” first (large increments, which should easily be seen experimentally as well as computationally), therefore it was decided to use 3 or 4 different wheel configurations that were readily available.

The following discussion outlines both the experimental and computational setups for this study, as well as a discussion of the results and findings.

DISCUSSION

Abundant data was available on several production vehicles, but in this case a small hatchback (2007 Dodge Caliber SXT) was chosen to begin the analysis.

EXPERIMENTAL TESTING

The CFD results are compared to experimental (wind tunnel test) results for the following cases:

1. Baseline (Al wheels), with open grille
2. Baseline with closed cooling openings
3. With convex wheel covers (ideal)
4. Steel wheels only
5. Steel wheels with improved wheel cover



Figure 1: Baseline Wheels (5-spoke, cast Al)

These five cases were chosen in order to give a wide spread of results to compare. The open-versus-closed grille increment (cooling drag) is a standard quality check, to ensure that correlation results are matching both with and without cooling airflow (closed grille takes this factor out, but is unrealistic for real world operation).

The convex wheel covers should be the best-case scenario, essentially a zero drag wheel increment. Plain steel wheels were expected to be poor aerodynamically, and essentially simulate a deep dished style wheel. Also, this allows us the flexibility to try many different wheel covers, to find both good and bad “wheels” without having to exchange entire wheel sets each time.

For all cases, the same tire size and brand were used, varying only the wheels. The tires used in this study were production Firestone Affinity P215/60 R17 with standard tire pressures.

In regards to setup, the vehicle was set each time to design attitude, e.g. EPA “2 Pass” loading (2 x 68 kg and full fuel). For this vehicle, body attitude was maintained at 742mm in front and 744mm in rear (ground to wheel lips at centerline of spindle), within +/- 2mm.

Initially, efforts were made to also “clock” the baseline wheels, or ensure that the spokes were all set in the same direction, and that for the CFD simulations as well. However, it was found that there was little difference for these wheels on this vehicle for clocking (within test repeatability, although on another vehicle that may or may not be the case), and thus for later runs the wheels were not clocked.

Aerodynamic forces and moments were measured via the balance of the Chrysler Aero Acoustic Wind Tunnel, which is a weigh-beam type [1].

The aerodynamic drag results were corrected for wind tunnel interference effects using a modified Mercker-Wiedemann adjustment factor [2-4]; however, this adjustment is typically small, and not significant for this study.

Other aspects of the testing were also investigated, such as body lift at speed, flow visualization of the wake from the airdam, etc. Each of these was deemed insignificant or satisfactory, so no further discussion is warranted.

CFD TESTING

The simulation setup for the computational testing was intended to be identical to the experimental, for correlation purposes. Vehicle specifications, ride attitude, and wind tunnel test conditions were all replicated in the CFD model. These will be briefly described in more detail below.

Simulation geometry

The geometry used was initially compiled from CATIA version 4, and converted to be cleaned and surface meshed in ANSA. Great efforts were taken to ensure that all components were accurately represented, even

to the degree of having windshield wipers, emblems, and body cutlines, etc. It took several attempts at gathering all the underhood components, as sometimes some things would not translate properly or were simply missed during the first go-around. All details were included to eliminate the risk of interactions, that might not be obvious, influencing the results.

The vehicle geometry and surface mesh are shown below:

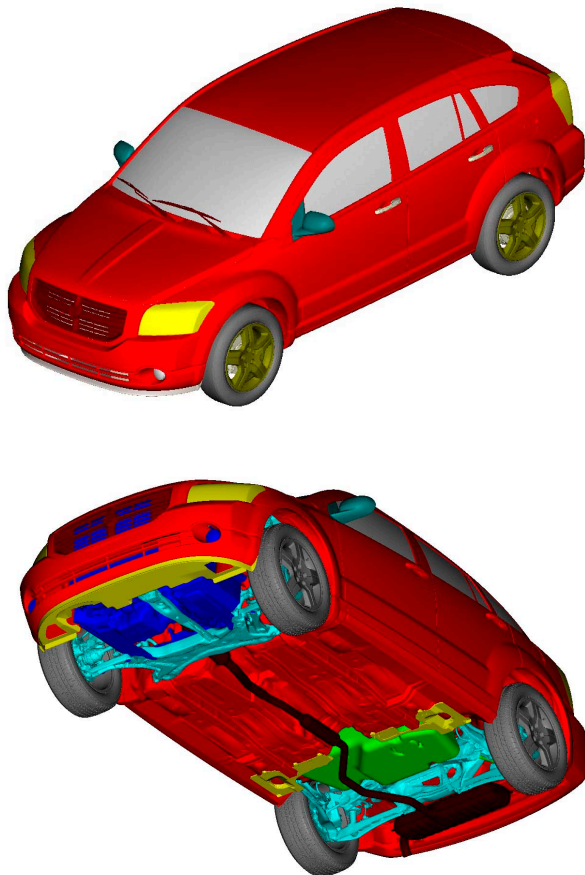


Figure 2: Simulation Geometry

The wheels and tire geometry were (initially) taken from the CAD geometry, and although the file for the tires was supposedly an actual profile provided by the tire manufacturer, it is to be noted that some differences to the actual tire, such as lack of tread detail, sidewall lettering, etc. existed; however, this level of detail was typically used for the CFD simulations in the past and gave fairly reasonable results.

Simulation Boundary Conditions

As mentioned above, the cases were all run with the same test conditions as the experiments. For all cases, in order to correlate to wind tunnel results (AAWT), static ground and static wheels were used.

Ride attitude was matched to the experiment, which was run at design loading (EPA 2 Pass).

Wind speed was 31.3 m/s (70 mph), and efforts were made to match the boundary layer profile (and thickness at front wheels of 30mm) of the Chrysler Aero Acoustic Wind Tunnel [1].

Porous media resistances for simulating the cooling module heat exchangers were taken from internal Chrysler measurements. The fan model was not represented here, since the experimental setup did not have powered fans (they were allowed to free wheel).

Simulation Code and Mesh Resolution

The commercial CFD software code Exa PowerFLOW™ v4.0d was used in all cases of this study.

The resolution used resulted in a total of 47 million cubic volume cells (voxels), which required approximately 26 GB of memory. The simulations were run on an IBM Linux cluster of 64 CPUs.

The sizes of voxels in the simulation volume are determined by nested variable resolution (VR) regions. The setup consisted of:

- VR6 and VR7: body fitted, extruded to ground
- VR8 (5mm cell size): upper grille, lower grille, cooling module, underhood, fog lamps, leading edge of hood, front closeout and airdam, A-pillars, cowl, wipers, and rear spoiler
- VR9 (2.5mm cell size): upper grille, front closeout (from front fascia to radiator support) and airdam, fog lamps, tire rib (to be discussed further)

The digital wind tunnel simulation used a velocity inlet condition and static pressure outlet ($C_p=0$ reference), and the size of the “wind tunnel” was set so that blockage ratio of the vehicle was less than 0.5%, so no “blockage” corrections to the data were necessary.

Again, the ground condition was given as a fixed wall (static), with the same for vehicle body and wheels/tires as well.

FIRST ATTEMPT, AERODYNAMICS SUMMARY

After the first results from the simulation were completed, the experimental versus computational results were compared. In addition to drag coefficient, the correlation of front and rear lift coefficients as well as cooling drag and cooling airflow were also of primary

interest. These would be used to establish the baseline, and provide confidence in the results.

In short, the results were okay, but it was hoped that the results would be better for this correlation study:

- Drag was under predicted for the open grille case (by about 11 counts C_D)
- Front lift was under predicted for the open grille case (by about 31 counts C_{LF})
- Cooling drag increment was under predicted (by almost half), and cooling airflow was also low

Table 1: Comparison of Experimental vs. CFD Results (CAD tires)

Open Grille	C_D	C_{LF}	C_{LR}
Experiment (baseline)	0.383	0.013	0.083
CAD tires	0.372	-0.018	0.097
deltas (experimental vs. CAD)	-0.011	-0.031	0.014

Closed Grille	C_D	C_{LF}	C_{LR}
Experiment (closed grille)	0.359	-0.072	0.129
CAD tires	0.358	-0.121	0.160
deltas (experimental vs. CAD)	-0.001	-0.049	0.031

Cooling Drag Increment	ΔC_D	ΔC_{LF}	ΔC_{LR}
delta Cooling Drag - Experiment	-0.024	-0.085	0.046
delta Cooling Drag - CFD (CAD)	-0.014	-0.103	0.063

Probable Causes for the Discrepancies

The first step in investigating why the computational results did not match the experimental was to go back over the vehicle geometry, and ensure all components were represented. Possible geometry issues, especially for the underhood, fan shroud, and tires (which had a small rib on the parting line between tread and sidewall, known to raise drag slightly) were all suspect. It was discovered that there were a couple of missing underhood components, as well as a degenerative fan shroud surface mesh (which had some webbing between the spokes, caused by “wrapping” the geometry with too low resolution mesh. However, when these were re-run, the results still did not improve significantly.

Additionally, cooling module modeling using porous media to simulate the heat exchangers was also investigated as a possible source of error. However, it was determined that the resistance parameters used were correct and appropriate, and even if these were varied by a fairly significant amount (>10%) it did not affect the aerodynamic forces much. Therefore, it was concluded that it must be something else.

Wind tunnel effects were considered. Blockage correction used in the experiment (but not in the computation, due to low blockage), static pressure gradients, etc. were all considered and dismissed, because they were deemed non-issues for the wind tunnel and digital simulations.

Lastly, the resolution used in the CFD simulation was further investigated. Chrysler had developed its own internal best practices for grid resolution, based on tradeoffs of speed vs. accuracy. Exa also had its own best practice guidelines [7], which were more conservative than ours. When the baseline case was re-run with Exa best practices, it did somewhat improve the drag correlation (by about +0.007), but did not address the differences in the front lift nor cooling drag increment. Therefore, something else must have been causing the discrepancy.

While investigating all these other factors which could have played a role, it was attempted to further refine the tire geometry. As it was known that there was a small rib (approx. 2 x 3.3mm in size; see below) on the tire which was not represented in the CFD simulations, and that when this was trimmed off once during wind tunnel testing and resulted in a 4 count C_D delta, it was concluded that that tire/wheel simulation should be improved.

This “tire rib” was in a critical area of the shoulder, and was found via flow visualization (plus objective measurements as mentioned above) to cause flow separation off the tire sidewall, thus increasing drag. So it would have at least some small effect, but it was not felt to be the entire cause of the discrepancy. Thus, it was investigated further.



Figure 3: Rub Strip (Rib) on Firestone Tires