

Aerodynamics of a Pickup Truck: Combined CFD and Experimental Study

Scott Holloway, James H. Leylek and William D. York
Clemson University

Bahram Khalighi
General Motors R&D Center

Copyright © 2009 SAE International

ABSTRACT

This paper describes a computational and experimental effort to document the detailed flow field around a pickup truck. The major objective was to benchmark several different computational approaches through a series of validation simulations performed at Clemson University (CU) and overseen by those performing the experiments at the GM R&D Center. Consequently, no experimental results were shared until after the simulations were completed. This flow represented an excellent test case for turbulence modeling capabilities developed at CU.

Computationally, three different turbulence models were employed. One steady simulation used the realizable $k-\varepsilon$ model. The second approach was an unsteady RANS simulation, which included a turbulence closure model developed in-house. This simulation captured the unsteady shear layer rollup and breakdown over the front of the hood that was expected and seen in the experiments but unattainable with other off-the-shelf turbulence models. Details of the high performance computing effort required to produce these results are discussed. The third simulation employed a new closure model designed to include the effects of small-scale unsteadiness on the mean flow without actually performing time-accurate simulations. As expected, this reduces CPU time, disk storage, and data I/O times significantly. In this case, CPU time was reduced by 95%. An approach of this nature would greatly reduce design time and make CFD a more feasible option.

INTRODUCTION

In an effort to understand the lift and drag forces acting on vehicles, many researchers have applied both experimental and/or computational methods to document the flow field surrounding the vehicle. These vehicles have ranged from simplified Ahmed bodies to racecars. The current study focuses on a pickup truck.

Though pickup trucks make up a significant percentage of the overall auto market, few studies exist in the open literature that focus on pickup truck aerodynamics. An early example is Butz et al. [1]. This trend is changing due to increasing fuel prices and relatively larger drag coefficients exhibited by pickup trucks, which typically have drag coefficients that are 15% higher than those for SUVs and 25% higher than those for sedans [2].

Of particular interest to designers for this blunt body is the flow field in the bed and behind the tailgate. These wake regions exhibit re-circulating flow and unsteady vortex shedding that are challenges to measure experimentally and resolve computationally. This is in addition to features seen on most automobiles, such as flow in the underbody region and flow over the front of the truck that can exhibit boundary layer transition and separated shear layer rollup and breakdown.

Experimental methods used to document this type of flow includes: Laser-Doppler Velocimetry (LDV), hot-wire anemometry, and Particle-Image Velocimetry (PIV). For

cases exhibiting periodic reversed flow, it is important to choose an appropriate method. In the current work, PIV has been utilized to document the flow field in the wake regions of the pickup truck. This method has been documented previously [3,4]. Further details of the experimental methodology are given in the next section.

The area of computational fluid dynamics (CFD) that is most commonly utilized in a design environment is Reynolds-Averaged Navier Stokes (RANS). Though Direct Numerical Simulations (DNS) and Large-Eddy Simulations (LES) are theoretically more physically realistic, the computer requirements to use these methods for wall-bounded flows with realistic Reynolds numbers is still out of reach for years to come in a design environment [5]. By Reynolds-averaging the governing equations, a new set of equations involving the mean flow field and the fluctuating flow field is obtained. However, the non-linearity of the momentum equations leads to the introduction of a new term, the Reynolds stress tensor (RST), that must be modeled to bring closure to the set of equations.

Much effort has been devoted to modeling of the RST, and many reviews exist, such as Wilcox [6]. Turbulence models can typically be divided into:

- algebraic mixing length models;
- eddy-viscosity models (EVMs);
- full differential Reynolds stress models (RSMs).

Eddy-viscosity models are generally implemented in industry since they are more general than algebraic models and more consistent to run than the numerically stiff RSMs. However, the traditional use of EVMs eliminates the ability to deal with the anisotropy of the RST and introduces an insensitivity of the turbulence field to streamline curvature. To accurately model flows with strong streamline curvature, such as around the hood, top and bottom of the windshield, and around pillars, it is necessary to model this effect in an eddy-viscosity setting [7,8].

Studies using off-the-shelf turbulence models for flow over a pickup truck have been documented in the open literature for both unsteady and steady frameworks. There is an inherent assumption in the RANS approach that there is a distinct separation for the time-scales of the turbulent fluctuations and the mean flow fluctuations (unsteadiness). Thus, the implementation of an unsteady RANS (URANS) approach should be scrutinized thoroughly, particularly in the regions of small-time scale unsteadiness [9-11]. An example would be the shear layer rollup and breakdown over the truck hood. This effect would be less pronounced, but present in the wake region in the bed and rear of the truck. This can lead to the conclusion that traditional RANS turbulence models are incapable of resolving some types of unsteadiness.

An RSM model with non-equilibrium wall functions was used in Zhu and Yang [12] in an unsteady framework

with a time step size of 0.001 s. The characteristic surface mesh had a size between 15 and 25 mm with a total cell count of 2.4 million finite volumes. Reasonable comparisons were seen for the pickup's surface pressure distribution and the velocity field in the wake regions using two different pressure strain models. A transient solution using the renormalization group (RNG) $k-\varepsilon$ model in addition to a very Large-Eddy Simulation (VLES) was discussed in Lokhande et al. [13]. The authors used a time step size of 0.0002 s. The computational mesh was designed to utilize non-equilibrium wall functions and consisted of 3.5 million finite volumes with a characteristic surface mesh size of 3.5 mm. For the mesh resolution utilized, the authors found that the RANS method produced similar levels of agreement to the VLES. A steady simulation utilizing the realizable $k-\varepsilon$ (RKE) model of Shih et al. [14] was used in Yang and Khalighi [2]. The mesh characteristics were similar to Lokhande et al. [13]. Similar to Zhu and Yang [12], reasonable agreement was achieved for the basic flow structure even though a steady approach was implemented.

Another area of CFD that has shown promise is the use of the Lattice-Boltzmann method (LBM). It relies on a different formulation (Boltzmann equation) of the governing equations to simulate the flow with the use of collision models. Thus, LBM models the propagation and collision of particles over a discrete mesh or lattice. This inherently unsteady method has shown great promise, in particular for wake flows. A comparison of the relative performance of traditional RANS methods [13] to LBM for flow over a pickup truck is given in Khalighi [15]. The authors found that both methods performed well for some measures and not as well for others. It should be noted that all turbulence is assumed to be modeled in the LBM approach though the grid resolution is more on the VLES scale. Incorporating (by modeling or resolving) all scales of turbulence in the LBM setting is a key item in making this method an even better predictive tool.

EXPERIMENTAL METHODOLOGY

PICKUP TRUCK MODEL - The generic extended cab pickup truck model used in the previous studies [3,4,13,15] was used for the present investigation. The model represents a 1:12 scale of the full-scale pickup truck. The model was designed with a smooth underbody, enclosed wheel-wells and without openings for cooling airflow. Schematics of the pickup truck model with the pertinent dimensions are depicted in Figure 1. A coordinate system is attached to the front tip of the vehicle in the symmetry plane as shown in Figure 1.

WIND TUNNEL GEOMETRY - Figure 2 shows schematics of the side view of the wind tunnel test section. It has a 0.14 m fillet at the bottom corners to suppress corner flows. For the present investigation a 2-m long ground board (mounted approximately 0.1 m from the top wall of the test section) was used to simulate the ground effects [3]. The location of the model

in the wind tunnel is also depicted in Figure 2 schematically. The front bumper is located approximately 0.4 m from the leading edge of the ground board.

PRESSURE MEASUREMENT INSTRUMENTATION - Mean pressures were measured at all the locations shown in Figure 1. In addition, the total pressure and the static pressure along the length of the test section at the center of the fillets were measured. A 32-manometer rake was used for the mean pressure measurements. The pressure lines, 1-mm Teflon tubing, were installed inside the model and through the access holes in the wheels and the wind tunnel wall to minimize flow disturbances. The data were recorded by photographing the manometer rake using a digital camera. For each flow condition a total of 10 pictures acquired over a time period of 10 minutes (steady state conditions) and stored in a computer. The manometer rake pictures were processed using a commercial image processing software package (Adobe Photoshop) to determine the mean pressure at each pressure port. The measured mean pressures were used to determine the pressure coefficient (c_p):

$$c_p = \frac{p - p_{ref}}{q}$$

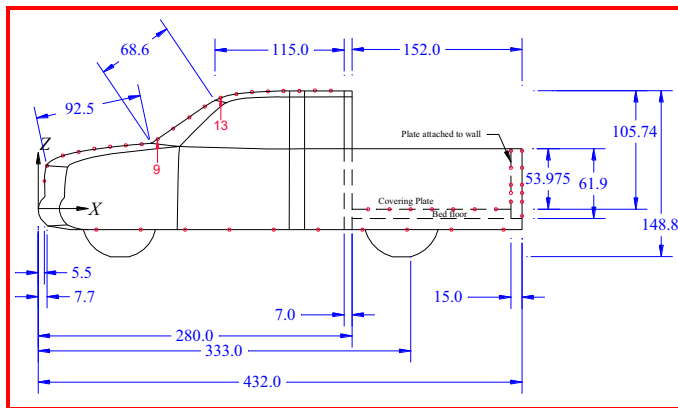
where p_{ref} is the reference pressure and q is the wind tunnel dynamic pressure. For the present measurements the reference pressure is the static pressure measured at the pressure port on the bottom fillet located approximately at the same downstream location as the cab base. The accuracy of the mean pressure coefficient measurements is 0.3%.

PARTICLE IMAGE VELOCIMETRY (PIV) - The velocity field on planes normal to the free stream (see Figure 3) was measured using a two-frame digital PIV system. The flow was illuminated using two Nd-YAG lasers. Both lasers were operated at approximately 115 mJ per pulse. The output beams were merged and directed into the test section by a set of mirrors. The resulting beams are shaped into a light sheet normal to the free-stream by a combination of 1000 mm focal-length spherical lens and a -6.35 mm focal-length cylindrical lens. The optical setup produced a laser sheet 0.5 mm thick by 400 mm wide. The whole optical setup was mounted in an external rig which is fixed to the floor of the wind tunnel lab. The lasers were synchronized to illuminate the flow twice with a short time delay between the two exposures. Because the velocity normal to the sheet is very large, time delays in these experiments were set to 4 μ s. The flow was seeded with small oil particles generated by a smoke generator located upstream of the settling chamber. A typical particle size was around 30 μ m. The smoke was directed into a 'smoke box' which is equipped with a fan. The fan scatters smoke particles in order to produce a uniform smoke stream of 70 $\times$ 70 cm. The cross-sectional area of the smoke stream in the test section was about 20 (H) $\times$ 20 (W) cm.

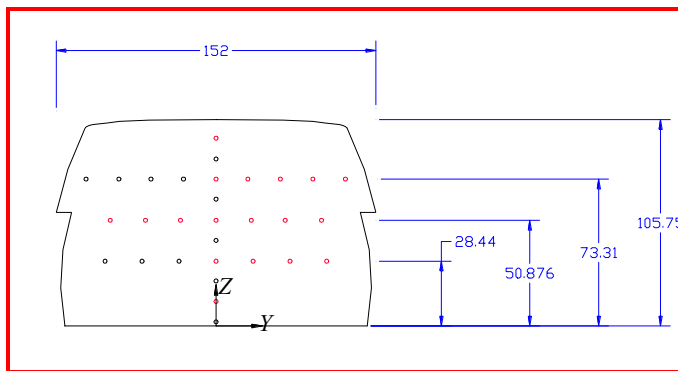
In order to photograph the flow on planes normal to the free-stream, a 3 $\times$ 4 inch mirror is attached to the trailing edge of the ground board. The mirror is rotated 45 degrees relative to the flow direction, and placed 1350 mm downstream of the pickup truck model. This location is chosen to minimize flow interference. The camera is positioned outside the tunnel normal to the flow direction, and records the images of the cross section planes through the 45 degree mirror. This arrangement is designed to position the CCD array of the camera parallel to the light sheet. In order to assess the PIV measurement noise and to correct for any deviation in the angle of the camera, measurements were taken without the model in the test section. The velocity fields reported here were obtained by subtracting the mean velocity without the model from each realization of the flow with the model present.

The light scattered by the seeding particles was recorded with a high-resolution dual-frame digital PIV camera (1280 $\times$ 1024 pixels). The field of view of PIV images was around 200 $\times$ 160 mm that results in a resolution of 6.5 pixels/mm. The particle displacements are limited by the velocity component normal to the sheet and the thickness of the sheet. In these measurements, the particle displacements were approximately 0.3 pixels. The images were sampled at a rate of 1 Hz limited by the transfer time of the digital images to the data acquisition computer. A total of 360 PIV images were recorded at each plane. The PIV images were processed using an image shifting algorithm consisting of two steps. In the first step the images are processed using a standard cross correlation algorithm on un-shifted interrogation windows to determine the particle displacement at each grid point. The interrogation window size is large enough to accommodate the expected particle displacement. In the second step cross correlation analysis between shifted interrogation windows is used. The window shift at each grid point is the particle displacement obtained in the first step. The main advantage of this technique is that the measurement resolution is decoupled from the magnitude of the particle displacement and consequently the noise level of the PIV measurement is improved. The interrogation window used in the second step determines the spatial resolution of the measurement. For the present measurements the interrogation window used was 64 $\times$ 64 pixels and the grid spacing was 16 pixels, which correspond to a spatial resolution in the flow of 10 $\times$ 10 mm and 2.5 mm grid spacing.

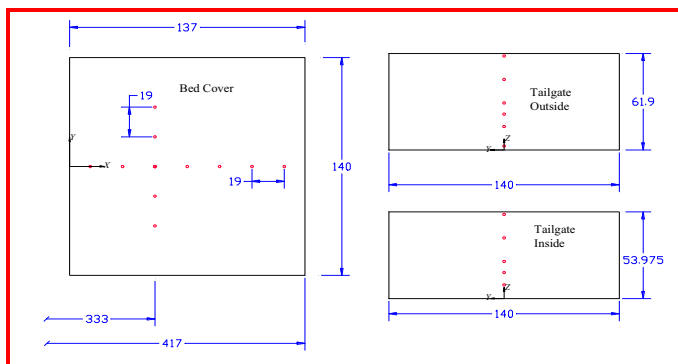
The PIV data were processed to obtain mean and turbulence properties in the wake of the pickup truck. Grids used were 74 $\times$ 24 points for the flow above the bed at x = 375 and 410 mm and 74 $\times$ 58 points for the flow behind the tailgate at x = 482 and 550 mm. In the analysis of turbulent data, each PIV velocity field is treated as an independent realization of the flow since the time between each sample is long compared to the turbulence time scales. Ensemble averages of 300 instantaneous fields are reported. The uncertainty of the mean velocities is estimated to be less than 2.6%.



(a) Side view



(b) Cab rear view



(c) Bed cover and tailgate view

Figure 1. Dimensional details of generic pickup truck model including location of pressure ports.

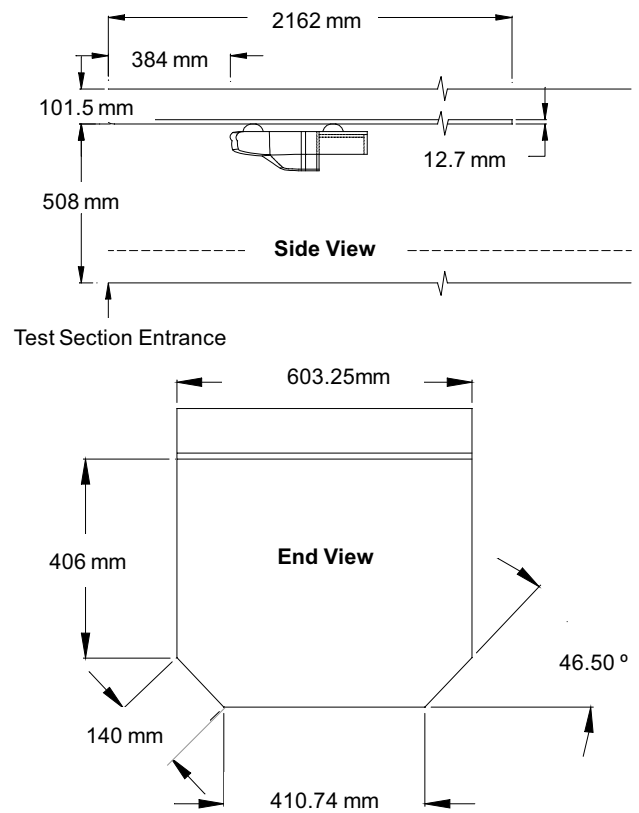


Figure 2. Schematic of wind tunnel and test section.

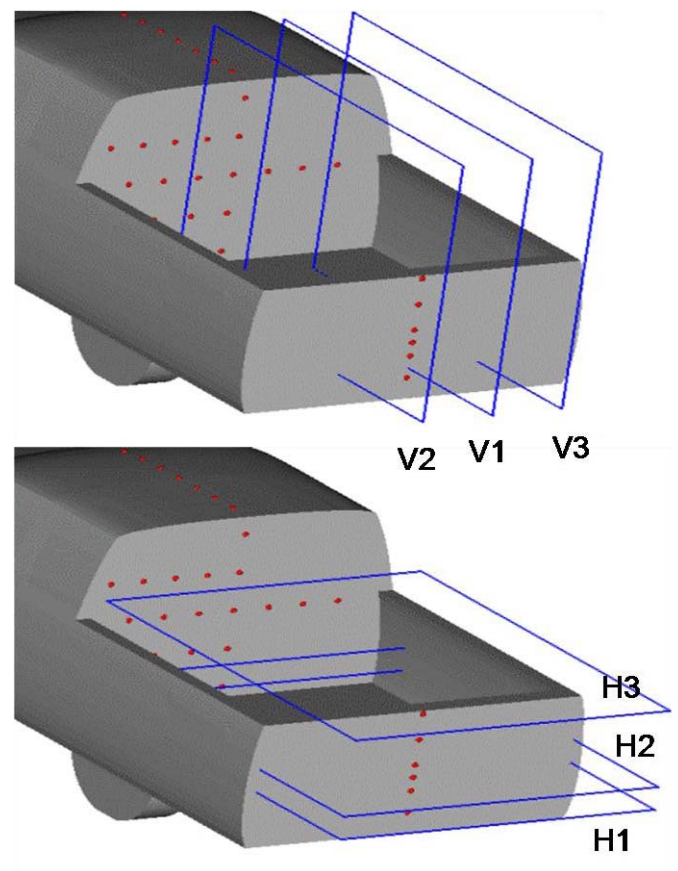


Figure 3. Location of planes for velocity comparisons.