

Pressure contours on the surface of the pickup and pathlines, colored by velocity magnitude showing the flow around the vehicle

# Streamlining Pickup Trucks

By Sandeep Sovani, Fluent Inc., and Bipin Lokhande, Fluent India Pvt. Ltd.

For 22 years, America's best-selling vehicle has been the pickup truck. Although pickup trucks together account for just 13% of new vehicles sold in the United States, familiar offerings from Ford, GM, and Chrysler held down the top three spots in overall units sold in 2004 through the first half of the year. This statistic is due in part to the country's love of size and cargo utility. Whether they have short beds or long beds, two doors or four doors, short cabs or long cabs, pickup trucks are in continuous demand by the public.

From an operational point of view, a drawback of pickups is that they consume much more fuel than a standard passenger car. At typical highway cruising speeds, the majority of fuel consumed by an automobile is spent on overcoming aerodynamic drag. Because trucks are aerodynamic bluff bodies, they experience particularly high drag forces, something pickup designers are always interested in reducing.

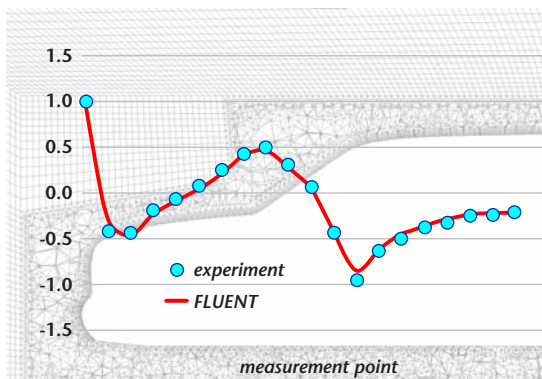
CFD is an effective tool for helping designers understand the flow structure around a truck and optimize its shape. The less-streamlined shape of a truck means that the flow field around it is inherently transient. Unsteady CFD simulations therefore model the flow field more accurately than steady-state computations. In the present study [1], the external aerodynamics of a generic truck body are considered by making use of GAMBIT and FLUENT. The truck being modeled is a 1:12 scale replica of an actual pickup with its side mirrors and underbody components removed. Starting from a triangular surface mesh, a fine prismatic boundary-layer mesh is created on all prominent surfaces of the truck. A layer of tetrahedra is created immediately above the prism layers. The rest of the domain is filled with hexahedra for an overall mesh size of about 3.5 million cells.

Using FLUENT's large-eddy simulation (LES) and RNG  $k-\epsilon$  turbulence models in separate simulations, transient CFD calculations were conducted using a 0.3 ms time step and compared to detailed wind tunnel experiments [2]. These comparisons include time-averaged pressure distribution around the vehicle and wake velocity profiles. Pressure profiles predicted on the vehicle body show excellent agreement with experimental data. A notable feature of the air flow is a large vortex forming behind the truck's cab. Additionally, there is a rapid change of direction behind the tailgate as the flow passing above the vehicle turns downward and mingles with the flow underneath. The pathlines predicted by FLUENT closely follow those measured in the experiments.

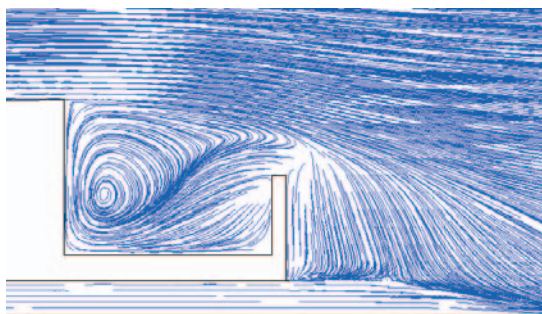
The goal of this study was to generate a set of data that can be used for validating CFD simulations of pickup trucks in the long term. Now that fuel economy is joining safety and versatility as a consideration when buying a truck, designers can be confident that modern CFD techniques will accurately predict the complex bluff-body wake flows that occur around pickups and give them insight into the challenge of drag reduction. ■

## references:

1. B. Lokhande, S. Sovani, and B. Khalighi, "Transient Simulation of the Flow Field Around a Generic Pickup Truck," SAE Transactions: Journal of Passenger Cars – Mechanical Systems, Paper no. 2003-01-1313, p. 1358-1376 (2003).
2. A. Al-Garni, L. Bernal, and B. Khalighi, "Experimental Investigation of the Near Wake of a Pickup Truck," SAE Paper 2003-01-0651 (2003).



Pressure coefficient distribution on the truck's upper body, plotted at the symmetry plane, compared to experiment [2]



Time-averaged pathlines in the truck's wake are in very good agreement with experiment [2]