Driving Vehicle Performance

Simulation helps to reduce drag and improve acoustic performance of cars, including an electric vehicle.

By ANSYS Advantage *Staff*

CFD was used extensively to develop the front-end cooling airflow characteristics of the Volt.

A major automobile manufacturer used a combination of a wind tunnel and fluid flow simulation to reduce the vehicle drag of a new pluggable electrical vehicle. The auto company found that computational fluid dynamics (CFD) provides airflow analysis results faster, delivering the ability to use more geometric detail than is practical with a clay model and enhanced flow visualization capacity around the vehicle for diagnostic purposes. The combination of wind tunnel testing and CFD made it feasible to achieve aggressive range and fuel economy targets.

The aerodynamics team starts with concepts and ideas provided by the design studio. The process begins with a considerable amount of interaction between the studio and the aerodynamicists. The designers work to preserve the theme of the vehicle, and the aerodynamicists provide feedback on shape changes and modifications that could be used to reduce the vehicle's drag. The bulk of the wind tunnel testing is performed on a one-third–scale clay model that is continually modified to evaluate various design alternatives.

"While the wind tunnel plays the primary role in the aerodynamic design process, we find ourselves in situations in which wind tunnel testing does not make sense, either because there is not enough time to perform tests or because we can't build a clay model of the geometry to the level of detail that we need," said Ken Karbon, staff engineer for General Motors. "In these situations and others, we use ANSYS FLUENT fluid dynamics software to simulate aerodynamic performance." Karbon added that his team can easily set up an automatic batch run to evaluate several hundred different front-end airflow configurations over a weekend — unattended. Fluid dynamics simulation also provides the ability to model the underhood geometry to a much higher level of detail and within a more reasonable period of time than is possible with a physical model.

To reduce the time required to analyze the large number of iterations needed to optimize a design, an efficient simulation process is used to automate a considerable portion of the model setup process. This makes it possible to semi-automatically run many test points without user interaction. For example, engineers can wrap a workflow toolkit around the fluid dynamics software that guides the user through pre-processing, meshing, case preparation, job submission, post-processing and reporting. The models generally are submitted to a high-performance computing cluster built around IBM POWER6™ blades. Engineers utilize these capabilities with design of
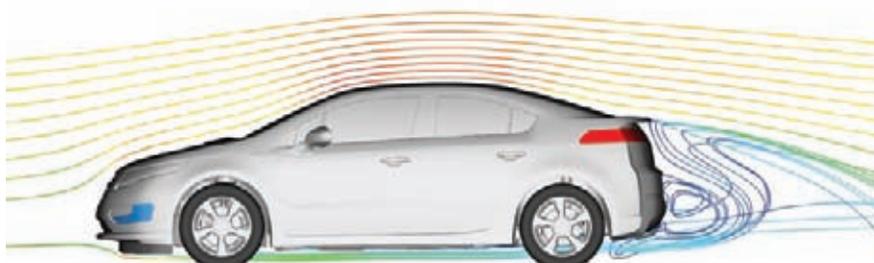
Volt external aerodynamics simulation using CFD
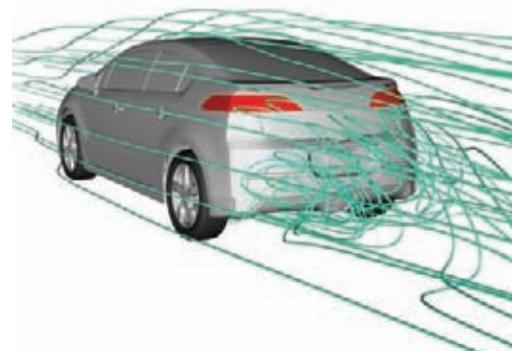
experiments (DOE) or design for six sigma to explore large design spaces and identify the global optimum design.

In the case of the Chevy Volt, aerodynamics engineers obtained CAD data from the design studio in Siemens NX™ format. The team used the fluid dynamics software's pre-processor capabilities to build a tetrahedral or hex-core mesh of the volume surrounding the vehicle's outer surface with a prism layer next to the body to capture boundary-layer gradients. Volumetric refinement was used to capture gradients in other areas of interest, such as the wake and stagnation points. Boundary conditions were defined to match the wind tunnel.

"For the styling surfaces, we applied classic automotive aero-dynamics best practices by ensuring smooth, continuous airflow around the vehicle," Karbon said. "We entrained smoke in the wind tunnel to provide a high-level macroscopic picture of flow direction. Fluid flow simulation allows us to see much more than could ever be obtained from physical testing, such as pressure differentials fore and aft of underbody components. In general, we pay close attention to large separation points or vortex structures coming off the vehicle, because they create a significant drag penalty. We also target very clean separation from the back of the vehicle."

Fluid dynamics analysis played the primary role in designing the grille, radiator and related components to obtain the right amount of airflow through the grille to direct the airflow where it is needed and to minimize cooling drag. "We looked at the pressure and velocity profiles to understand how the air comes through the grille openings. We needed to make sure we were getting enough volume of air to meet the most

Before

After
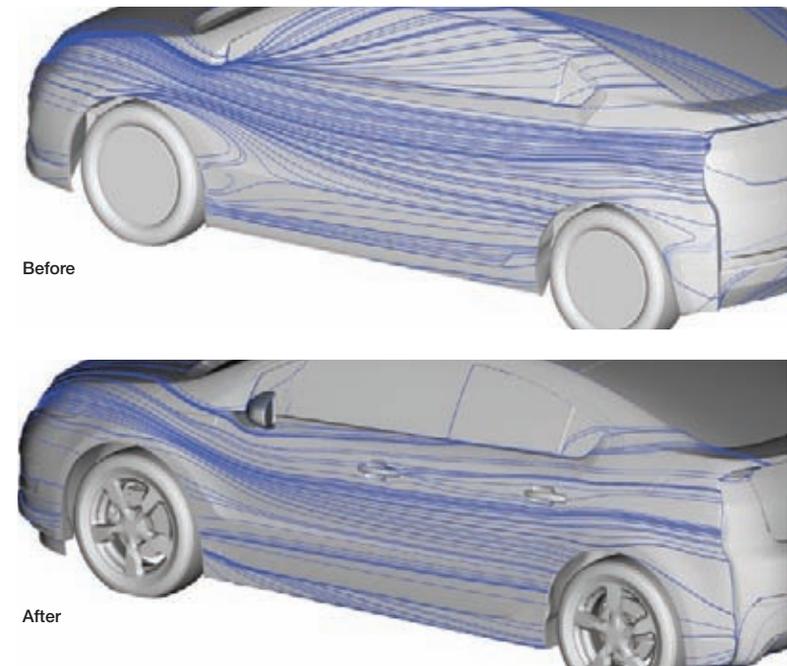
Surface restrict particle traces indicate how the Volt's body sides and wheel arches were streamlined, resulting in lower aerodynamic drag while maintaining the design theme.

demanding engine cooling load cases, such as driving the car through Death Valley desert," Karbon said.

To solve specific problems, engineers created large arrays consisting of hundreds of design variations and utilized parallel batch, scheme and journal processing to quickly analyze all of them. This made it possible to quickly model various design details such as grille openings, grille texture, air dams, heat exchanger size and position, and sealing and baffling underneath the hood.

The engineering team also used fluid dynamics simulation to mitigate and optimize wind noise associated with the wiper blades on the Cadillac CTS, CTS Wagon and SRX. They first determined pressure and flow velocity generated by the wipers during the concept design phase. This information was used as input to software that predicts acoustic performance. The ability to evaluate wind noise at an early stage of the design process helped avoid

later-stage problems that would have been costly to correct.

Fluid dynamics analysis also predicts wind loads on vehicle components such as sunroofs and windows, especially under crosswind conditions. This information is fed into structural analysis programs used to ensure that all components meet high-speed requirements. CFD also estimates the temperature and quality of underhood airflow to the power-train induction system.

"CFD plays a vital role in aero-dynamic and acoustic design by highlighting the areas we need to work on and by providing diagnostic information that we cannot obtain in the wind tunnel," Karbon concluded. "The technology makes it possible to quickly evaluate hundreds of designs in batch processes to explore the complete design space so that we know we have the best possible design. For example, this process made it possible to reduce drag from start to finish." ■