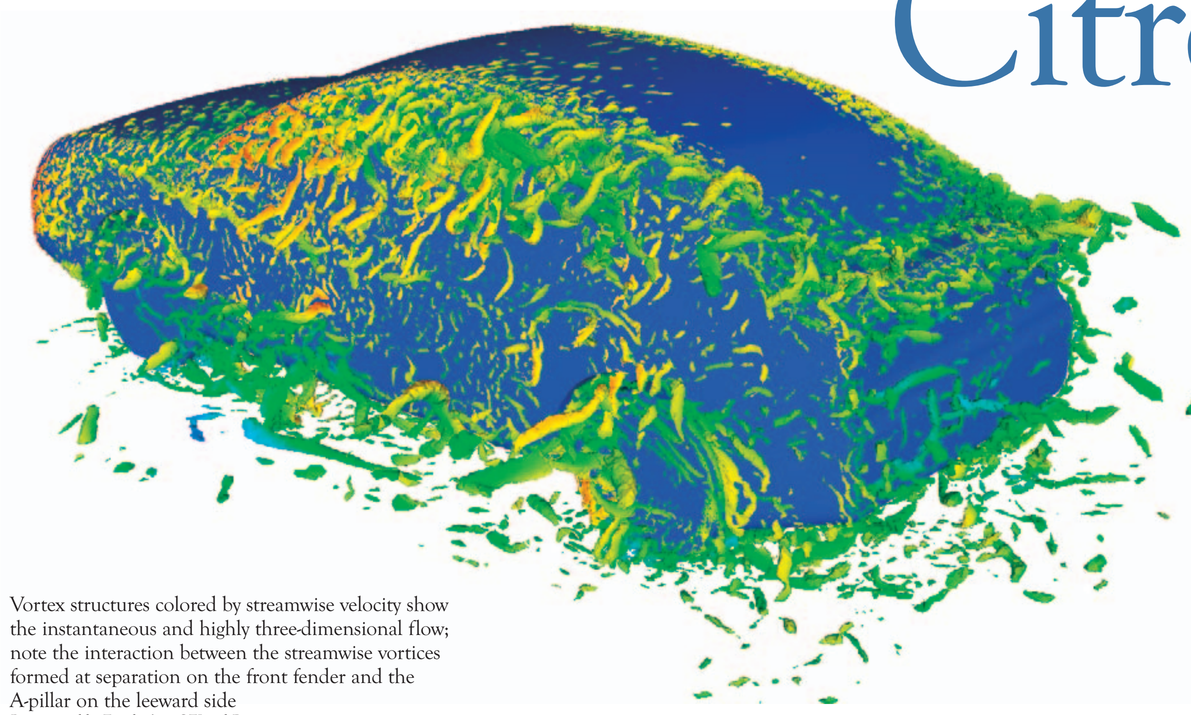
Citroën



Vortex structures colored by streamwise velocity show the instantaneous and highly three-dimensional flow; note the interaction between the streamwise vortices formed at separation on the front fender and the A-pillar on the leeward side
Postprocessed by Ensight from CEI and Distene

As PASSENGER VEHICLES BECOME LARGER, their stability and safety characteristics become more sensitive to aerodynamic forces. Such effects are particularly important for vehicles exposed to a crosswind [1]. For instance, all drivers are familiar with the effect of cross-wind gusts, like those that may occur when a vehicle emerges from a tunnel or during overtaking. CFD can provide detailed information for understanding the changes in car behavior, handling, and performance under these conditions.

Simulations are routinely performed at PSA using engineering approaches that are based on the solution of the Reynolds-averaged Navier-Stokes (RANS) equations. While this approach is often adequate for steady attached flows with no recirculation regions, numerous studies have shown the limitations of the RANS approach to accurately predict flows with massive separations and that are fundamentally unsteady.

Large eddy simulation (LES) can have greater success than RANS methods in predicting separation and large scale unsteady flows, particularly in the wake and on the leeward side of vehicles under crosswind conditions. In recent years, this type of unsteady calculation has been increasingly used in industry and FLUENT has made a special effort to develop industrial-strength validation examples for LES modeling techniques [2,3].

PSA Peugeot Citroën has developed a strong expertise in CFD for external aerodynamics, cabin modeling, underhood thermal management, and more. In-house methodologies have been developed and are regularly improved, in order to fulfill new project requirements, which are getting more and more demanding, both in terms of accuracy and turn-over time. Computations are done in conjunction with wind tunnel experiments, either on full scale prototypes or, as in the case presented here, on a 1:5 scale model. Different measurement techniques are used, such as particle-image velocimitry (PIV), pressure taps, and tomography. The different methodologies also benefit from a collaboration between PSA and Fluent, and this was the basis of a recent joint project to address the potential benefit and cost of LES compared to the current RANS approach for semi-realistic configurations. Fujitsu/Siemens provided access to a large-scale high performance computing (HPC) facility for this project.

	Model	Exp	SST k-ω	LES WALE (35M cells)	LES WALE (65M cells)	RSM	v^2-f
Forces	**Drag** (SCx)	**0.70**	0.66	0.69	0.68	0.71	0.73
	Side (SCy)	**2.22**	2.00	2.19	2.18	2.30	2.10
	Lift (SCz)	**1.40**	1.66	1.27	1.30	1.82	1.77
Moments	**Yawing** (SCn)	**-0.64**	-0.60	-0.57	-0.59	-0.47	-0.47
	Rolling (SCl)	**-0.42**	-0.36	-0.49	-0.49	-0.46	-0.41
	Pitching (SCm)	**0.12**	0.10	0.21	0.23	0.03	0.07

Detailed force and torque components acting on the vehicle; under crosswind conditions, the side force (SCy) and yaw moment (SCn) are the most influential for vehicle stability

C5 in a Crosswind

By Sylvain Lardeau, PSA Peugeot Citroën Automobile, Vélizy-Villacoublay, FRANCE, and Fabrice Mathey and Nicolas Vallée, Fluent France

The RANS and LES simulations were performed on a 1:5 scale simplified model of a Citroën C5. The cross-wind effect was produced with the model placed at a yaw angle of 20°. The Reynolds number of the flow, based on the incoming velocity and car height, H, was 6.19×10^5. The surface mesh (around 730,000 elements) was generated with ANSA™ and was locally refined close to the edges and on the leeward faces of the vehicle. The boundary layers were generated with the preprocessor TGrid, using 5 layers of prisms. The remaining part of the domain consisted of tetrahedral cells and was generated with the preprocessor GAMBIT. Sizing functions were used to control the growth rate and the cell size of the mesh in the region of separation on the leeward side of the car. The minimum resolution required to resolve the relevant turbulent length scales was estimated from prior RANS simulations. Two grids (of 35 million and 65 million cells) were created to address the sensitivity of the results to the mesh resolution. The body was placed in an open channel with a cross section of 14.5H x 7H. The channel inlet was located 5H from the front face of the body, and the channel outlet was located 8.5H from the rear. The boundary layers were modeled with the Werner-Wengle wall-function approach. The average y^+ for the first cell off the wall was less than 50.

The simulation benefited from the recent enhancements in numerics and sub-grid scale (SGS) modeling in FLUENT 6.2. The bounded central differencing scheme was used to prevent unphysical wiggles numerically introduced by the pure central differencing scheme. The non-iterative time advancement, or NITA algorithm significantly increased the speed of the transient calculation. The wall-adapting local eddy-viscosity (WALE) SGS model, which returns the correct wall asymptotic variation of the turbulent viscosity in a cost-effective manner, was used as well. The simulations were performed on a 96 CPU Fujistu cluster. For the 35M case, statistics were gathered during 16,000 iterations for an elapsed time of 70 hours. Statistics were gathered during 6,000 iterations for the 65M case, for an elapsed time of 50 hours.

The full mean aerodynamic forces and torques were compared with RANS methods and experimental data. Under crosswind conditions, the side force and yaw moment are the most influential for vehicle stability. The LES results showed very good agreement with the experimental data for these quantities. The difference between measurements and calculations was 2% for the side force coefficient (SCy), and was in the range of 10% for the yawing moment (SCn). The LES runs also gave consistent results for the other aerodynamic force components; the drag force (SCx) was predicted with a 1% error, for instance. It is noteworthy that even if some RANS simulations are able to predict side force and yawing moment with reasonable accuracy, none of the turbulence models could predict consistent results for all of the force components and moments. Surface pressure coefficients also confirm the differences between the different computational methods used. On the A-pillar on the leeward side, especially, the pressure drop is too strong with the RANS model. The two LES runs gave comparable results, indicating that reasonable mesh independence was reached with the 35 M grid, as far as global force coefficients are concerned.

The visualization of the mean velocity streamlines indicates the formation of two streamwise vortices on the leeward side of the vehicle. The first one is formed on the front fender and the second is formed on the A-pillar and along the side window. Animation and instantaneous flow visualization demonstrate the highly three-dimensional, unsteady nature of these vortices. The predominant peak frequencies calculated from a Fourier transform of the pressure signal at several locations on the leeward side is equal to 110 Hz, a value very close to the experimental peak frequency of 100 Hz.

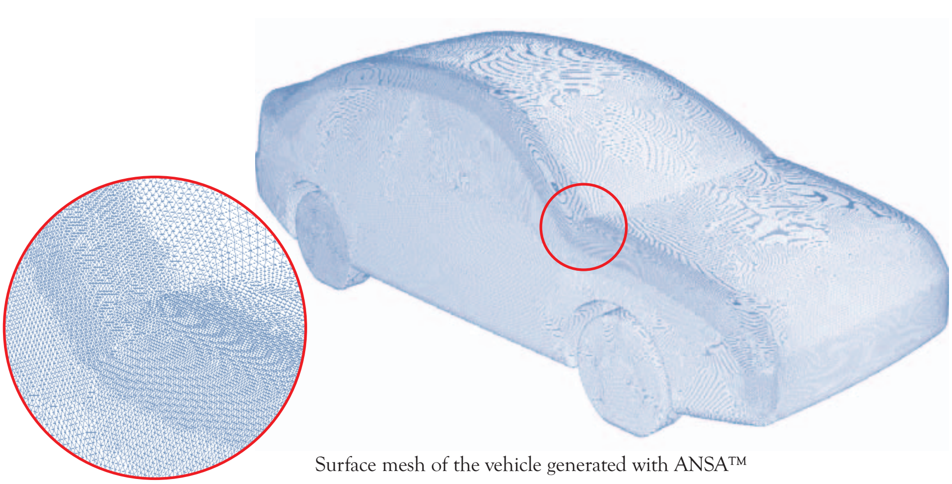
These simulations demonstrate how RANS and LES modeling approaches can complement each other to study car aerodynamics. Current RANS-based methodologies are able to correctly predict drag force components for attached flow and mild separation on realistic car geometries under uniform wind conditions. LES is a better tool for studying vehicle aerodynamics in crosswind conditions, since it provides a more accurate drag force component and pressure distribution along the vehicle. In addition, LES gave insight into the unsteady behavior of the flow, an important feature for both vehicle stability studies and aeroacoustic applications. ■

References:

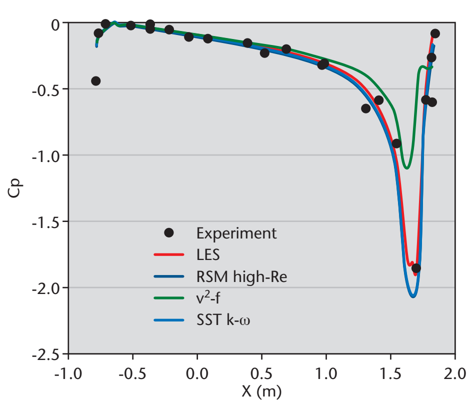
1 Ryan, A. and Dominy, R.G.: The Aerodynamic Forces Induced on a Passenger Vehicle in Response to a Transient Cross-wind Gust at a Relative Incidence of 30 Degrees. *SAE Paper* 980392, 1998.

2 Kim, S.E.: Large Eddy Simulation using Unstructured Meshes and Dynamic Subgrid-scale Turbulence Models. AIAA Paper no. 2004-2548, 2004.

3 Mathey, F. and Cokljat, D.: Zonal Multi-domain RANS/LES Simulation of Air Flow over the Ahmed Body. In: Engineering Turbulence Modelling and Experiments, 6, pp. 647-656, Elsevier Ltd., 2005.



Surface mesh of the vehicle generated with ANSA™



Surface pressure distribution on the windward side on the "eye plane", z=1.250 m