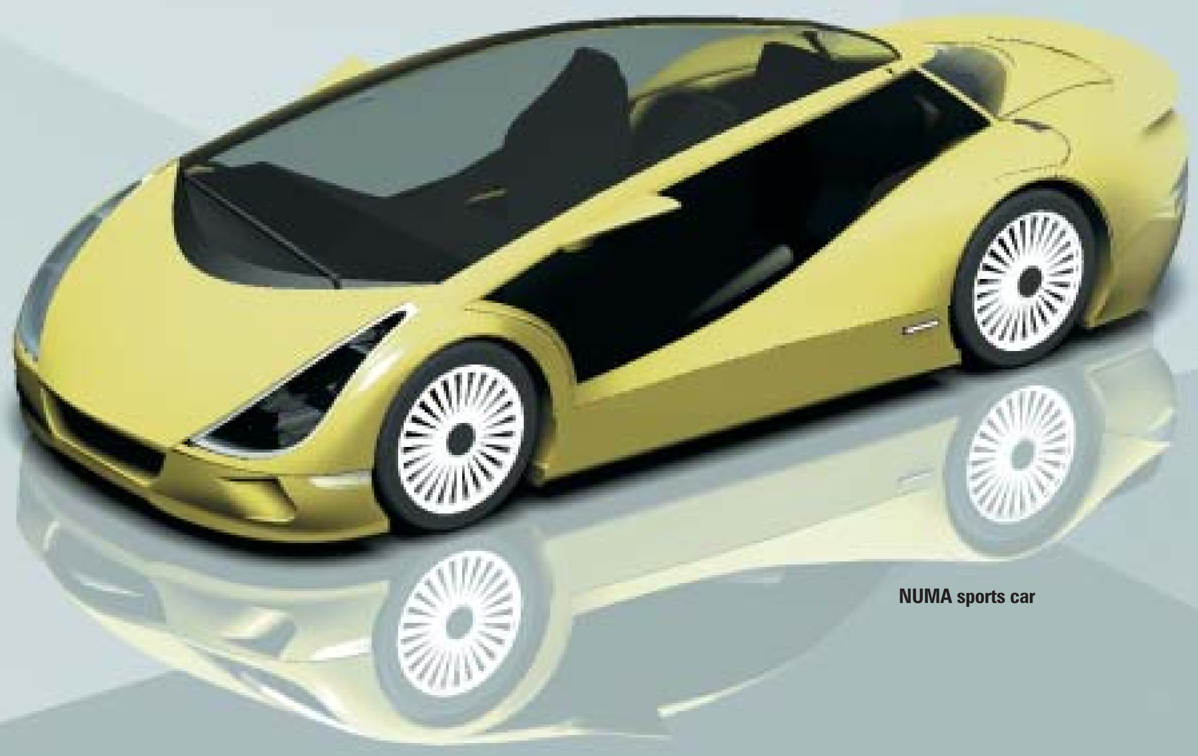
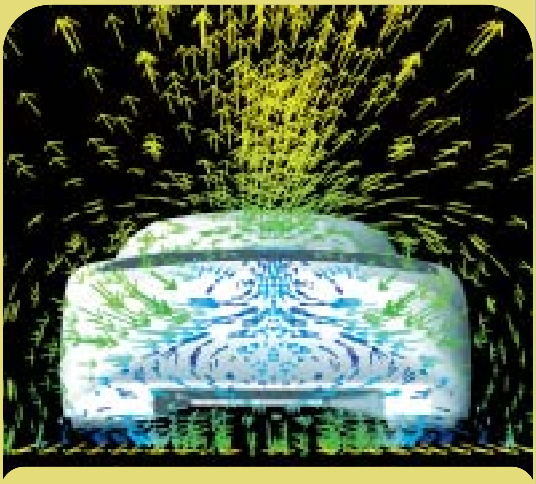
Designing a Better Sports Car Through Engineering Simulation

Aerodynamic analysis with ANSYS CFX fine-tunes the vehicle shape.



NUMA sports car

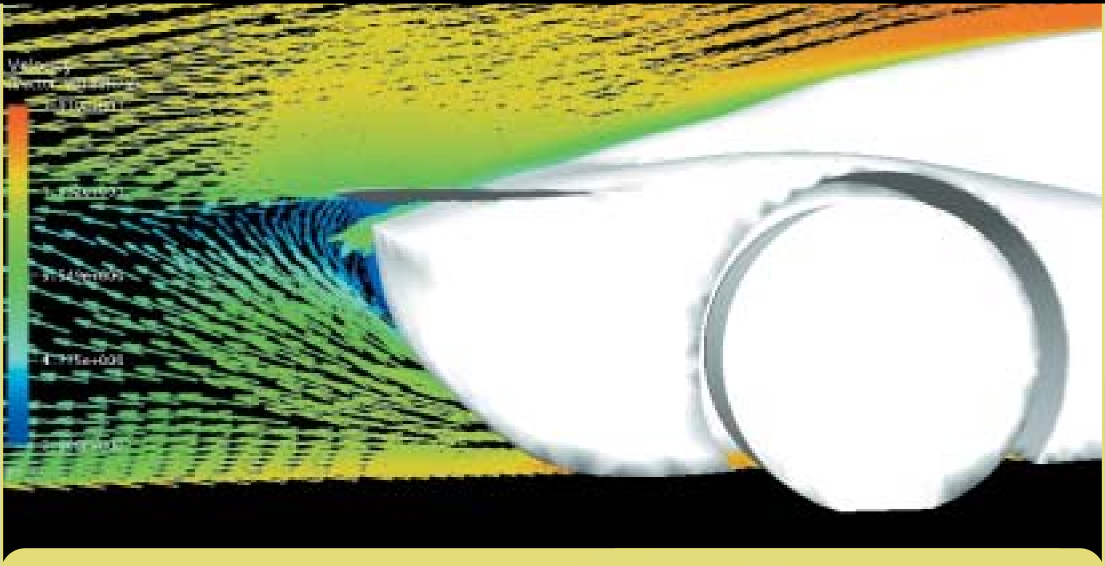
By Nuno Ricardo Rosmaninho
and António Gameiro Lopes
Department of Mechanical Engineering
University of Coimbra, Portugal



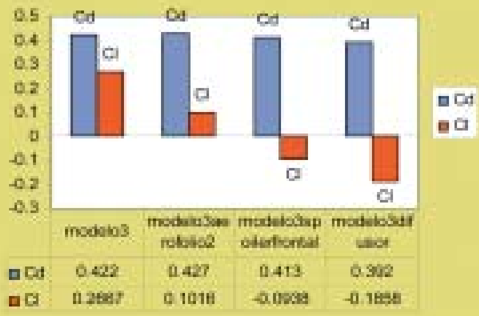
Velocity vector field at the rear of the car

The study of aerodynamics in the automotive industry is important to improve fuel economy as well as vehicle comfort and safety. Since the 1980s, automobile industries have relied more and more on numerical methods for vehicle design in order to reduce expensive experimental tests traditionally required for aerodynamic studies.

In the Department of Mechanical Engineering at the University of Coimbra, Portugal, engineers have been working on designing an innovative vehicle that combines a sporty design with standard transportation capability, the NUMA car. In order to fine-tune the shape of the car, an aerodynamic analysis was performed with ANSYS CFX computational fluid dynamics software.



Airflow past the rear airfoil



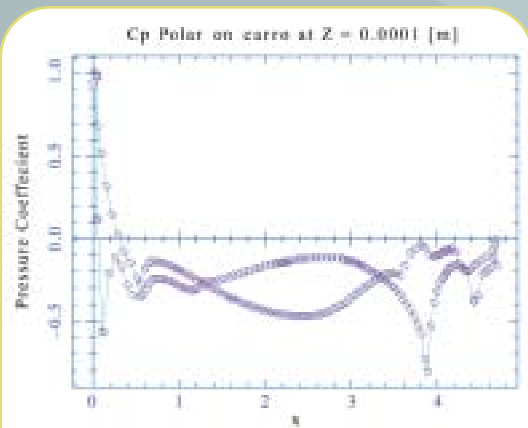
Evolution of C_D and C_L for different configurations of the car geometry

An inflated boundary of prismatic elements was used near the car surface to improve spatial resolution and gain a better understanding of boundary-layer phenomena. An unstructured mesh with tetrahedral elements was used for volume meshing. Simulations were carried out with the SST turbulence model, coupled with a blend factor of 0.5 for the advection scheme.

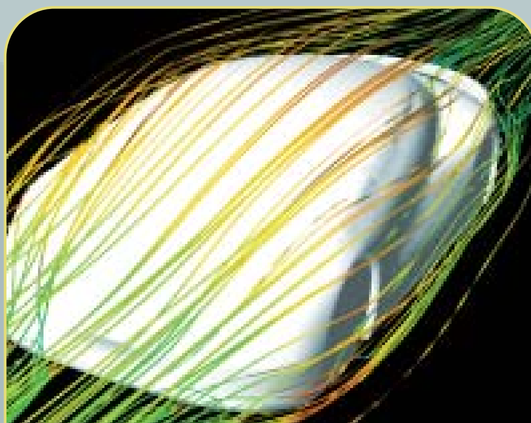
The research started with three models with characteristics of a sports car. The models differed in the design of the rear end. Aerodynamic optimization was used to determine the best configuration of airfoils, spoilers and diffusers. Because standard solutions for optimizing aerodynamics have been well developed over the years, we simply studied the best choices to apply. And, since the car was to be designed for sporty performance, study of both drag and lift were important. Little details on shape and position of spoilers and airfoils played an important role in the compromise between lift and drag.

The ANSYS CFX analyses provided information on flow separation, pressure and velocity fields, vortices and forces interacting with the vehicle. This information allowed engineers to make modifications in the car shape, producing better results. In the final design, airfoils were positioned based on the visualization of the flow field in the rear of vehicle and on the analysis of the corresponding effect in terms of drag and lift. Pressure coefficient charts helped with the definition of the overall geometry and gave insight on the distribution of the applied forces.

Thanks to ANSYS CFX software, the university could reduce the positive lift and still bring the drag to 95 percent of the initial value. ANSYS CFX proved to be an excellent tool for automotive aerodynamics. ■



Pressure coefficient results on the car symmetry plane



ANSYS CFX visualization of flow streamlines