

# Computer Simulation Helps Engineers Improve Ferrari Formula One Aerodynamics

By Francis T. Makowski, Ferrari S.p.A. Gestione Sportiva, Maranello MO ITALIA / Fluent Inc., Lebanon, NH  
 Luciano Mariella, Ferrari S.p.A. Gestione Sportiva, Maranello MO ITALIA

Computer simulation has helped engineers at Ferrari make significant improvements to the aerodynamics of a Formula One vehicle. The simulation was used to fine-tune the aerodynamics of a new roll-hoop structure required by recent rule changes. Because of the compressed design cycles required in Formula One racing, there wouldn't have been time to

evaluate alternatives if wind tunnel testing had been the primary design tool. Instead, engineers used computational fluid dynamics (CFD) to model the configuration

of the upper rear bodywork and optimize its design. In order to minimize simulation time, a reduced-domain simulation was performed in the vicinity of the new structure, with the conditions on the boundaries of the domain coming from an existing CFD model. During the CFD analysis, several sophisticated modeling methods were used to ensure an accurate solution, including the Reynolds Stress Model (RSM) and non-equilibrium wall functions for the turbulent flow. Subsequent air tunnel testing revealed that panels whose design was based on the simulation met the design objectives; they were essentially invisible aerodynamically.

In 2001, the body governing Formula 1 racing required that, for safety reasons, the roll hoop structure be capable of supporting higher vertical and lateral loads. This requirement forced all F1 teams to revise their chassis structures and bodywork in the vicinity of the inlet to the airbox snorkel. It ultimately became necessary for each team to re-

optimize aerodynamically all of the upper surfaces of the car aft of the cockpit.

FIA (Federation Internationale de l'Automobile)

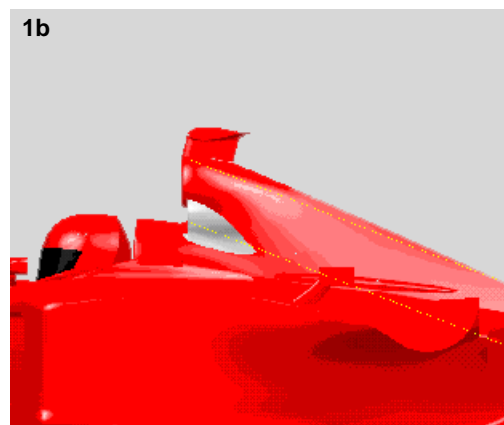
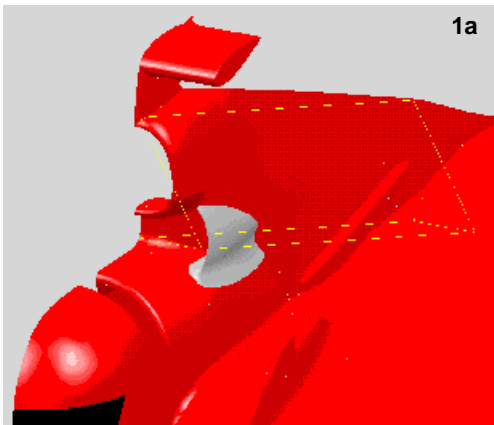


Figure 1a: Isometric view of a wireframe rendition of the tobleron  
 Figure 1b: Side view of a wireframe rendition of the tobleron

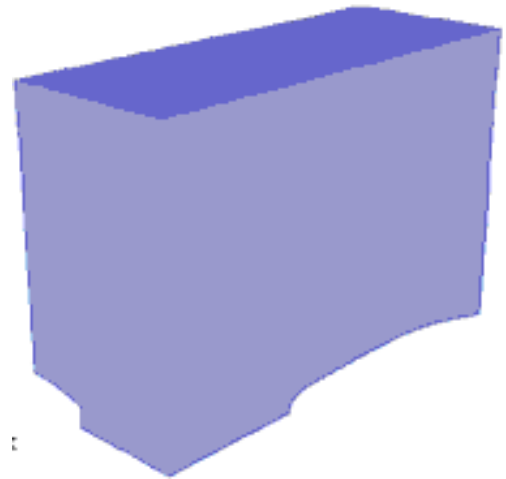
regulations required that an imaginary prism be concealed by the bodywork when the car is viewed from the side only. The location, size and orientation of this prism are illustrated by wireframe representations in Figures 1a and 1b. Ferrari internally refers to this prism as the "tobleron", a name that was inspired by a well-known prismatic bar of candy whose proportions are similar.

Because the prism could be visible from other viewpoints, Ferrari was able to interpret the regulations in a way that led to a distinct performance advantage. A special piece of body work was designed that accommodates the prism but leaves an

open space between the side of the airbox snorkel and itself. Called a toblerone panel by Ferrari engineers, the new body work reduced the car's frontal area while complying with the FIA regulations. Optimizing the aerodynamic surfaces of the toblerone panel was critical in order to avoid having a major negative impact on performance. Indeed, the primary design objective was that the toblerone panel introduce only minimal perturbations to the flow field. The wrong shape could yield a significant loss of downforce from the rear wing assembly.



651. An examination of the flow field from the Model 651 full-vehicle simulation revealed that the velocities outboard of the cockpit at the elevation of the stagnation line on the leading edge of the sidepod lay mostly in a horizontal plane. Therefore, a symmetry condition was prescribed on this horizontal planar boundary. Another symmetry boundary was used at the center plane of the vehicle and driver. The two symmetry boundaries are shown in gray in Figure 2a.



**Figure 2a: (left) and b (right): Domain boundaries colored by boundary condition**

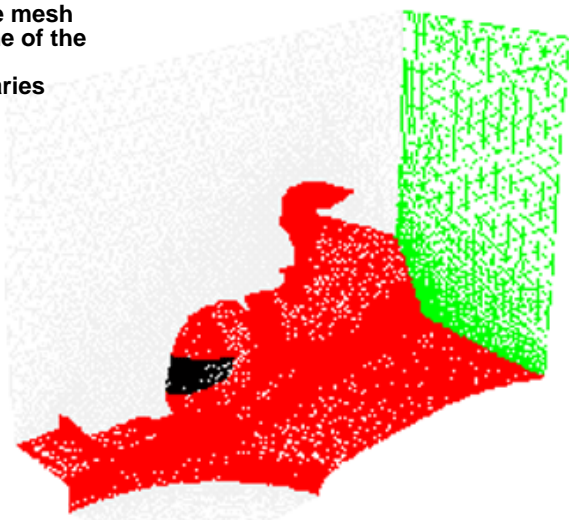
To simulate the design with a reasonable amount of computing resources, a reduced-domain simulation was performed for the vicinity of the toblerone panel (Figures 1a and 1b), with the conditions on the boundaries of the domain coming from a full-car simulation of the previous year's model. This was possible because of the fact that the area around the toblerone panel, including the cockpit, airbox inlet, and upper sidepod, is so constrained by the regulations that they change little from year to year. It should be noted that although the computational domain was reduced relative to the one for the full-car simulation, it was still quite large compared to the dimensions of the toblerone panel. The CFD simulation didn't explicitly model the toblerone panel; instead, it was performed to establish the unperturbed flow field. The optimal toblerone panel was defined by merely post-processing this solution.

The exterior boundary conditions for the simulation were supplied by flow field profiles that were taken from a full-vehicle simulation of the 2000 Ferrari F1 Grand Prix car, also known as the Model

651. Profiles of all three Cartesian components of velocity from the 651 flow field simulation were prescribed as boundary conditions on the front, side and top planar boundaries of the domain, shown in blue in Figure 2b. Like the 651 full-vehicle simulation, the 652 reduced-domain simulation employed the RSM for turbulence modeling. Therefore, profiles of each of the six unique components of the Reynolds stress tensor were prescribed on these boundaries, as were profiles of the turbulence kinetic energy and dissipation rate. To account for aspiration effects, a region inside the airbox snorkel was included where various velocity conditions could be specified to correspond to the typical range in engine aspiration rate. At the rear boundary of the reduced domain, shown in green on Figure 2a, a profile of static pressure from the 651 simulation was prescribed.

The CFD simulation was performed using FLUENT software from Fluent Inc., Lebanon, NH. The model of the 652's reduced domain employed a hybrid mesh of 1.04 million cells. First, a strategically non-uniform triangular mesh (Figure 3)

**Figure 3:**  
Surface mesh  
on some of the  
domain  
boundaries

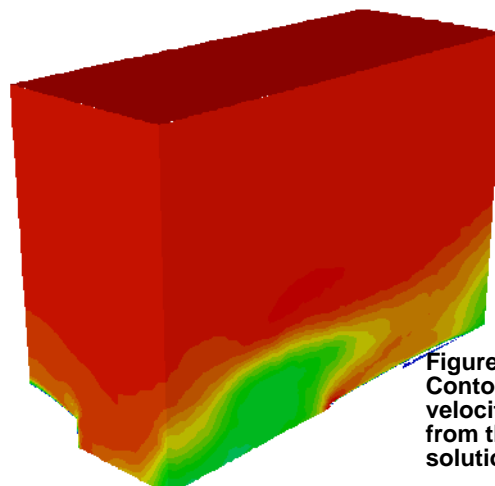


was applied to all boundary zones. From this surface mesh a volume mesh of prism and tetrahedral elements was built using TGrid, a mesh generator from Fluent. The solution was performed with FLUENT V5.4.8. It employed all of the physical models and numerical schemes that are required by FLUENT for accurate predictions of external aerodynamics in a turbulent flow that has possible regions of flow separation. The RSM turbulence model facilitates an accurate prediction of anisotropic turbulence which is especially important near areas of flow separation and re-attachment and in regions of high swirl, such as recirculating wakes. Non-equilibrium wall functions were used to correct the prediction of viscous shear at the wall for the local gradient of static pressure in the direction parallel to the wall, providing more accurate predictions of flow separation.

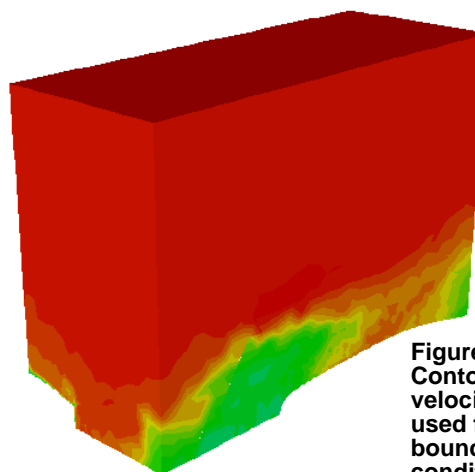
The FLUENT calculations for the 652's reduced domain were performed in parallel on two COMPAQ DS20 two-processor compute nodes. The 651 calculation that provided the boundary condition profiles for the 652's reduced domain was launched only shortly before the 652 calculation. The 652 calculation began with profiles from a 651 solution that was not yet fully converged. After 1500 iterations on the 652 calculation, its boundary condition profiles were replaced by profiles from the well converged 651 solution. After the first 652 calculation, all subsequent ones differed only in the rate of engine aspiration. Over the entire range in aspiration rate, the external flow field was virtually the same everywhere, except near the inlet to the

airbox snorkel. This fact was exploited by arranging the run matrix in the order of increasing engine aspiration rate and by initializing each subsequent calculation with the converged solution from the preceding one. By operating in this way, every 652 calculation after the first one required only approximately 500 iterations to achieve good convergence.

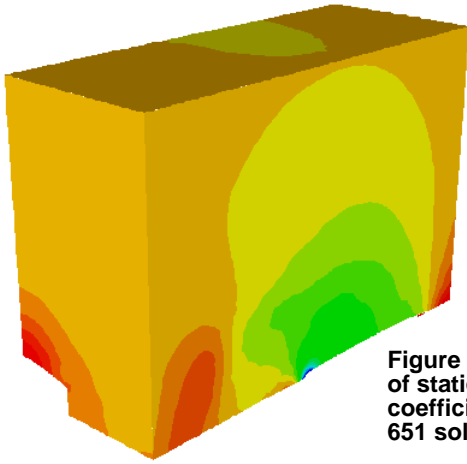
A 652 simulation for zero rate of engine aspiration was performed first so that its computed (i.e., not merely assigned) flow field quantities at the velocity and pressure boundaries could be compared against the flow field quantities for the 651 at the same locations and under identical conditions. Figures 4a and 4b compare velocity magnitude from the 651 solution against velocity magnitude from the profiles on the boundaries of the 652's reduced domain. The ranges are identical. The contours for the 652 represent boundary conditions that were supplied by the 651 solution. The contours in both figures are based on values at nodes that lie in the same planes. While the contours are very similar,



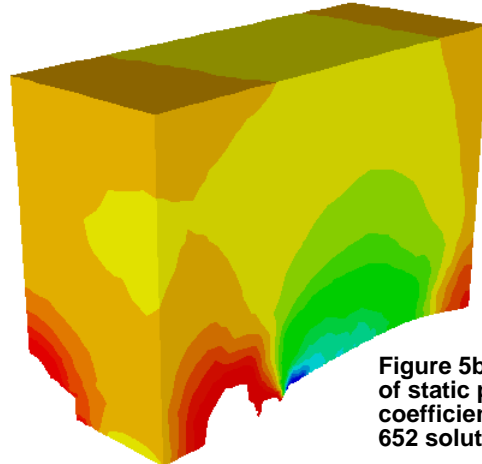
**Figure 4a:**  
Contours of  
velocity magnitude  
from the 651  
solution



**Figure 4b:**  
Contours of  
velocity magnitude  
used for the 652  
boundary  
conditions



**Figure 5a: Contours of static pressure coefficient from the 651 solution**



**Figure 5b: Contours of static pressure coefficient from the 652 solution**

those for the 651 are smoother because more nodes lie on the planes that were constructed to match the boundary locations of the 652. Both simulations used meshes with the same spatial resolution. The planes constructed for the 651 solution, however, randomly intersect the cells in the 651 model, and for each intersection with a cell edge, a node is formed and used in FLUENT's plotting routines. Thus, despite equivalent grid densities, the number of the 651's intersected cells was much greater than the number of the 652's mapped cells, so the 651's contours appear smoother.

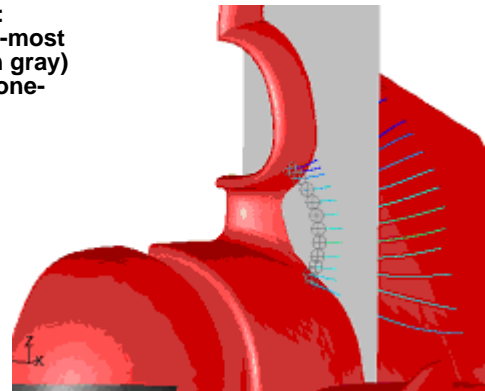
The static pressure coefficient for the two simulations is compared in Figures 5a (651 model) and 5b (652 model). The pressure coefficient is computed, not set as a boundary condition, so it is a more valid measure of comparison between the simulations. The range (using the same scale) and distribution on the planar surfaces is very similar. "Holes" in the 652 contours appearing near the lower edges of the front and side correspond to values that were (only slightly) out of the plotting range. Most of the differences in pressure coefficient are due to the more forward location of the sidepod leading edge on the 652.

The objective of the project was to make the toberone panels aerodynamically invisible. The panels would be thin; the shapes of the left-hand and right-hand side panels would be mirror images, and the panels would be positioned symmetrically about the vertical center plane of the car. Hence, only one of the two toberone panels was unique. It was possible to define it optimally for an appropriate rate of engine aspiration with a single FLUENT simulation of the 652's reduced domain. Due to the high Reynolds number of the flow, the boundary layer on an optimal toberone panel is thin. Hence,

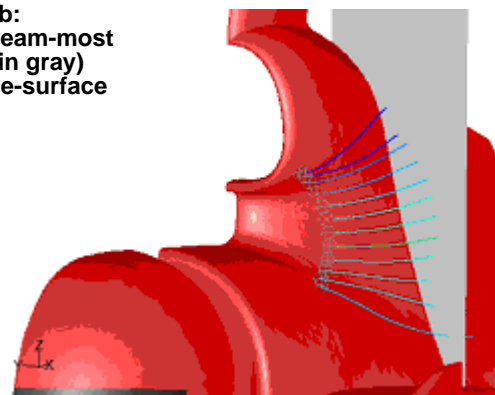
viscous effects from an optimal toberone panel are minimal. This fact allowed all design requirements to be satisfied without modeling the toberone panel in the FLUENT simulation that defined it! Instead, the optimal toberone panel was defined by post-processing the solution for the 652's reduced domain with an appropriate rate of engine aspiration using the standard FLUENT graphical user interface. Figures 6a and 6b illustrate how this was done.

First, a planar (i.e., constant-x) curve representing the leading edge of the toberone panel was defined in keeping with the FIA regulations. Path lines were released from points at regular intervals along this curve. These points are indicated

**Figure 6a: Upstream-most station (in gray) for toberone-surface**



**Figure 6b: Downstream-most station (in gray) toberone-surface**



by the crosshairs in Figures 6a and 6b. At many planar (i.e. constant-x) stations downstream of the release points, the path line trajectories were sampled for their spatial coordinates. The resulting set of coordinates was used to define a "toblerone surface", from which the toblerone panel was trimmed according to the FIA regulations. Wind tunnel tests have verified that the toblerone panels are aerodynamically invisible. Figure 7 shows them at work.

In summary, there was satisfactory agreement in the computed flow field quantities between the 651's full-vehicle simulation and the 652's reduced-domain simulation. This indicated that the 652's domain boundaries were located far enough away from important differences between the 651 and 652 geometries. It demonstrated that flow field profiles at these locations from the 651 full-vehicle simulation served adequately as boundary conditions

for the 652's reduced domain simulation. More specifically, it justified a good level of confidence in the procedure for defining the toblerone panels.

The toblerone panels were quickly designed with a small, reduced-domain FLUENT simulation that didn't explicitly model them. Subsequent windtunnel tests confirmed that the toblerone panels were essentially invisible aerodynamically. These tests validated the simulation and design strategies. The tests also suggested that the mesh for the 652's reduced-domain simulation could serve adequately as part of a 652 full-vehicle mesh. In fact, such a mesh was created later, and it yielded FLUENT results that agreed well with wind tunnel measurements. The success of this project was facilitated by FLUENT's flexibility. It typifies the symbiosis between reduced-domain simulations and full-vehicles simulations that prevails under Ferrari's F1 CFD program.



**Figure 7: The potential performance advantage provided by the toblerone panels is obvious**