

Using LES for Predicting High Performance Car Airbox Flow

F. Brusiani and G. M. Bianchi

DIEM
University of Bologna

T. Baritaud and A. Bianchi d'Espinosa

Ferrari GeS S.p.A.

Copyright © 2009 SAE International

ABSTRACT

Aerodynamic had played a primary role in high performance car since the late 1960s, when introduction of the first inverted wings appeared in some formulas. Race car aerodynamic optimisation is one of the most important reason behind the car performance. Moreover, for high performance car using naturally aspired engine, car aerodynamic has a strong influence also on engine performance by its influence on the engine airbox. To improve engine performance, a detailed fluid dynamic analysis of the car/airbox interaction is highly recommended.

To design an airbox geometry, a wide range of aspects must be considered because its geometry influences both car chassis design and whole car aerodynamic efficiency. To study the unsteady fluid dynamic phenomena inside an airbox, numerical approach could be considered as the best way to reach a complete integration between chassis, car aerodynamic design, and airbox design.

This paper presents the application of *Large Eddy Simulation* (LES) approach to the study of the unsteady flow conditions inside a real geometry of a high performance car airbox. LES is a promising technique to yield a CFD tool able to predict flow unsteadiness since in LES numerical modelling concerns only a small part of the energy spectrum while the large scale motion is directly resolved.

The FLUENT 6.3 code has been used and the *Wall Adaptive Local Eddy-Viscosity* (WALE) *sgs* model has been adopted. A *Bounded Central Differencing* (BCD) second order scheme has been adopted and a

discussion of the kinetic energy conservation attitude of such a scheme performed.

Results obtained by LES simulations have been analysed in terms of mean evolution and *rms* fluctuations of both pressure and velocity components.

INTRODUCTION

Race car performance depends on elements such as the engine, tire suspension, road, and aerodynamics. In recent years, however, the total aerodynamic package of a race car gained increasing attention, mainly due to the utilization of the negative lift principle, yielding several important performance improvements.

Traditionally, the effects of external aerodynamics are summarized in terms of drag, lift, and car stability: basically the efficient design of a chassis is based on a downforce/drag compromise.

However, today the car aerodynamic package could not be optimize only in terms of total downforce acting on the car. To create the fastest vehicle in a specific race category it is necessary to consider the significant aerodynamic interaction among various car body components. The air motion around a moving vehicle affects engine intake flows, cooling flows, tire cooling, and overall external flows.

In high performance cars equipped with a naturally aspired engines, the engine intake flows play a fundamental role in the optimization of engine performance. Generally an airbox is placed between external ambient and engine intake ducts. To maximize the engine volumetric efficiency is necessary to optimize the airbox fluid dynamic performance, and, in particular, its aerodynamic interaction with the car body.

The main goal of airbox design is to increase the air density available to intake runner, and to minimize the unsteady effects induced by the pulsating flows due to the intake phase of the different cylinders, and the air entering into the airbox.

The design of the airbox geometry comes from the balance of different car design targets. Many of them are not necessary correlated with the engine performance, and they rely on chassis design, and aerodynamic of the car. For example, the airbox inlet shape, and its capacity affect the overall car aerodynamic efficiency. Moreover, the airbox geometry has to be defined concurrently with the chassis design because of the strictly correlation between airbox inlet section design, and rollover hoop design.

From the above comments, it clearly appears the need to define the airbox geometry as earliest as possible in the design process of the race car.

Typical design tools such as wind tunnel testing, and track testing, are used from many years to improve race car aerodynamic design. During the last decade, thanks to the increased computational resources, also the Computational Fluid Dynamic (CFD) tools have become useful to address the aerodynamic design of a racing car. In fact, CFD approach represents a unique tool to obtain detailed information on the fluid dynamic efficiency of a system, helping the engineers to take their key decisions in advance with confidence.

In particular, one of the key point in the analysis of a generic aerodynamic profile is the study of its flow unsteadiness that could provide flow detachment, and lift loss. CFD can provide predictions on flow unsteadiness only when the Navier Stokes equations are not time averaged. In fact, adopting a standard Reynolds Averaged Navier Stokes (RANS) approach, the time average of flow fluctuations is zero by definition, thus it becomes impossible to numerically reproduce the real flow unsteadiness. An alternative, and in principle more physically grounded, numerical method to predict flow unsteadiness is Large Eddy Simulation (LES). The basic idea in LES is to resolve large Grid Scale (GS) and to model small SubGrid-Scales (SGS). The limit (cut-off) between GS and SGS is supposed to take place in the inertial subrange (see Figure 1).

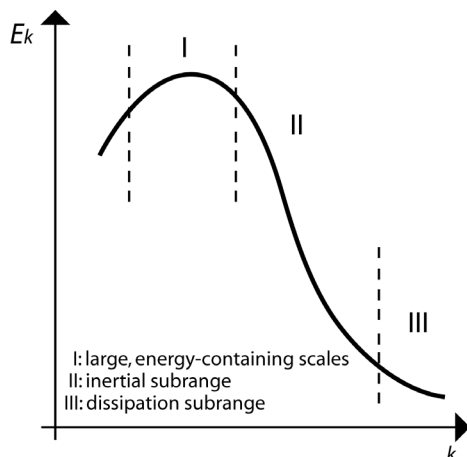


Figure 1: Energy cascade.

LES flow governing equations are spatially averaged (and not time averaged as in the RANS approach) to allow the direct numerical solution of the largest scales of motion (GS). Only a small fraction of the turbulence energy spectrum (SGS) is modeled. LES is expected to bring an improvement in relation to RANS when the rate controlling-processes of momentum, heat and mass transfer are dominated by the large scales of motion [1]. The most critical problems encountered using LES are:

- The need of a grounded and validated methodology;
- The computational cost;

The methodology has to be based on the assessment of the numerical framework attitude in conserving kinetic energy and on the definition of parameters suitable to check LES quality. It is well known that a calculation can be named as LES if a large fraction ($> 80\%$) of the turbulence energy spectrum is directly resolved [1]. This requirement usually results in the need to adopt fine grids and small Courant numbers to avoid energy cut-off at the highest frequency. When the filter and grid are too coarse to resolve the 80% of the flow energy, a Very Large Eddy Simulation (VLES) is performed.

Clearly the computational cost of a LES is quite higher than a RANS simulation because of the finest domain discretization. Moreover, it must be considered that the statistical procedure required to analyze the data of a LES simulation further increases the overall simulation time and post-processing duration.

The aim of this paper is to present the industrial application of LES approach to investigate the fluid dynamic characteristics of a race car airbox. To assure realistic fluid dynamic conditions on the airbox inlet section a part of the car body was considered in the computational domain.

It must be worth noting that detailed steady and unsteady 3D RANS CFD simulations were previously performed on the airbox under investigation. By these simulations, it was clear that:

- Flow unsteadiness must be further analyzed as well as their effect on the energy conversion from dynamic pressure to static pressure;
- The fluid dynamic behavior of the present airbox is not influenced by the cylinder dynamics: it could be assumed almost as a perfect capacity. Consequently, airbox flow unsteadiness can be analyzed by using LES with "fixed" inlet/outlet boundary conditions;

The LES methodology here adopted was previously defined by the authors on different test cases [2,3]. On the basis of the results obtained during the LES methodology setup, the WALE model for the *sgs* scale stresses without wall function was chosen to study the airbox flow behavior.

Numerical results are presented in terms of mean and *rms* pressure and velocity evolution along flow paths defined inside the airbox geometry by positioning more than forty probe points.

LES EQUATIONS

As previously introduced, the goals of LES approach are to explicitly compute the large-scales of motion (not universal and influenced by flow geometry), and to model the smaller scales (which have a universal character). Considering a generic random scalar function $\phi(x)$, by the filter operation it can be decomposed into the sum of two different contributions:

$$\phi(x) = \bar{\phi}(x) + \phi'(x) \quad (1)$$

where:

- $\bar{\phi}(x)$ represents the motion of the large eddies, thus the resolved component (GS);
- $\phi'(x)$ represents the unresolved (SGS) component of the motion;

The general filter operation of the random scalar function $\phi(x)$ can be defined as:

$$\bar{\phi}(x) = \int G(r) \cdot \phi(x-r) dr \quad (2)$$

where the integration is over the entire flow domain, and the specified generic filter function G satisfied the normalization condition:

$$\int G(r) dr = 1 \quad (3)$$

In particular, in this case $G(r)$ represents a uniform and isotropic filter. Generally, the filter may be uniform (i.e., $G(\vec{x}, \vec{r})$ does not depend on $\vec{x}$) or it may be non-uniform. It may be isotropic (i.e., $G(\vec{x}, \vec{r})$ depends on $\vec{r}$ only through $|\vec{r}| = r$) or it may be anisotropic.

Different filter functions can be used. The most common filter functions are: box filter, Gaussian filter, and sharp spectral filter [1]. In finite volume methods box filtering is always used. Box filter filters out fluctuations in space with scales smaller than the filter size Δ :

$$G(r) = \begin{cases} 1/\Delta, & \text{if } r \leq \Delta/2 \\ 0, & \text{otherwise} \end{cases} \quad (4)$$

Filter size is identified with computational cell dimension, thus in 3D the filter size is defined as:

$$\Delta = (\Delta_x \Delta_y \Delta_z)^{1/3} = (V)^{1/3} \quad (5)$$

where V is the generic computational cell volume.

The equations for the evolution of the filtered velocity field are obtained by the Navier-Stokes equations. The filtered Navier-Stokes equations are of the standard form; subgrid-terms appear in the turbulent stress tensor (analogous to the Reynolds-stress tensor of the RANS approach). Closure of the equation set is obtained by modeling the residual-stress tensor by an eddy viscosity model.

Considering a compressible flow, filtering the original flow governing equations, the filtered flow governing equations used in LES are obtained:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho \bar{u}_i)}{\partial x_i} = 0 \quad (6)$$

$$\frac{\partial(\rho \bar{u}_i)}{\partial t} + \frac{\partial(\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = \frac{\partial^2(\mu \bar{u}_i)}{\partial x_j \partial x_j} - \frac{\partial \bar{p}}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} \quad (7)$$

where τ_{ij} is the sub-grid scale stress defined as:

$$\tau_{ij} = -\rho(\overline{u_i u_j} - \bar{u}_i \bar{u}_j) \quad (8)$$

The subgrid-scale stresses are unknown terms that must be modeled. The eddy-viscosity approach [4] is used to approximate the subgrid-scale turbulence stresses:

$$\tau_{ij} - \frac{1}{3} \tau_{kk} \delta_{ij} = \nu_t \bar{S}_{ij} \quad (9)$$

where μ_t is the sub-grid scale turbulent viscosity, δ_{ij} is the Kronecker delta and $\bar{S}_{ij}$ is the rate-of-strain tensor of the resolved scales defined as:

$$\bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (10)$$

WALE SGS STRESS TENSOR MODEL

In the present work the WALE incompressible sgs model was adopted to calculate the sgs stress tensor. This model was developed by Nicoud and Ducros [4]. It is an eddy viscosity based sgs model where the eddy viscosity is defined as:

$$\nu_t = (C_\omega \Delta)^2 \overline{OP}(\vec{x}, t) \quad (11)$$

with

$$\overline{OP} = \frac{(S_{ij}^d S_{ij}^d)^{3/2}}{(\bar{S}_{ij} \bar{S}_{ij})^{5/2} + (S_{ij}^d S_{ij}^d)^{5/4}} \quad (12)$$

where $\bar{S}_{ij}$ is the strain rate of the resolved field while S_{ij}^d is the traceless symmetric part of the square of velocity gradient tensor:

$$S_{ij}^d = \frac{1}{2} (\bar{g}_{ij}^2 - \bar{g}_{ji}^2) - \frac{1}{3} \delta_{ij} \bar{g}_{kk}^2 \quad (13)$$

where

$$\bar{g}_{ij}^2 = \frac{\partial \bar{u}_i}{\partial x_k} \frac{\partial \bar{u}_k}{\partial x_j} \quad (14)$$

This sgs model can detect turbulence with both strain rate and rotation. In the case of pure shear, the considered invariant is zero. This means that it produces no eddy viscosity in laminar flow. Thus, it accounts for



effect of both the strain and the rotation rate to obtain the local eddy viscosity. As a result, it returns the proper y^3 scaling for the eddy viscosity approaching walls.

FLOW SOLVER AND GENERAL NUMERICS

All the airbox LES simulations were performed by using Fluent v6.3 adopting a segregated flow solver together with a fully implicit second-order scheme [5]. Diffusive fluxes were discretized by using central differencing (CD) schemes. Convective fluxes were treated in a special way, using the so called bounded central differencing (BCD) second order scheme [2,3,6], developed for complex industrial flows involving High-Reynolds number. In Fluent [5] the BCD scheme is available instead of the classic second order upwind scheme.

BCD scheme was designed to suppress unphysical wiggles numerically introduced by the pure CD scheme, preserving CD's non-dissipative property. In particular, the BCD scheme detects wave length of twice the filter size or less and, in this case, it blends with a second-order upwind scheme or, in worst case, a first-order upwind scheme. BCD scheme is also suited for hybrid grids as required in engineering applications [7,8].

EVALUATION OF FLUENT NUMERICAL DISSIPATION

In LES the physical dissipation of the kinetic energy (directly or via the sgs tensor modeled) must be provided while the numerical dissipation should be zero or kept at minimum. Numerical scheme usually introduces dissipation, therefore, a check on this attitude must be performed, and it can be considered as mandatory.

As proposed by Moreau et al. [9], the inviscid simulation of a vortex convection in a 2D periodic box was considered for the check. Figure 2 shows the computational domain and the boundary conditions adopted to reproduce the test case.

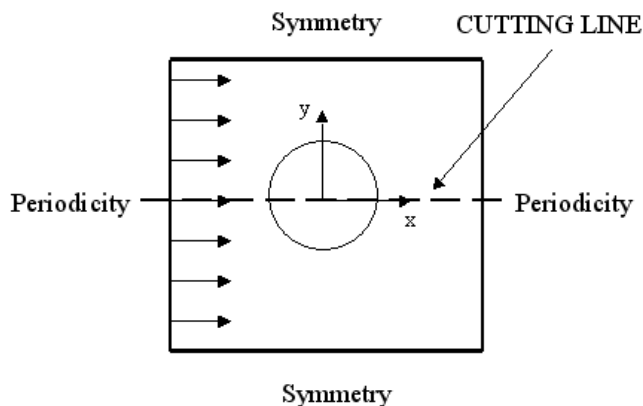


Figure 2: Computational domain and boundary conditions adopted to reproduce the periodic eulerian vortex test case proposed by Moreau et al [9].

A structured 2.5D mesh with 18000 hexahedral cells was used (only 5 cells were placed along the z direction).

The initial spatial fluid dynamics vortex conditions were:

$$u = -\frac{\Gamma}{R_c^2} y e^{-\frac{x^2+y^2}{2R_c^2}} \quad (15)$$

$$v = -\frac{\Gamma}{R_c^2} x e^{-\frac{x^2+y^2}{2R_c^2}} \quad (16)$$

$$p - p_0 = \frac{\rho}{2}(u^2 + v^2) \quad (17)$$

where:

$$\Gamma = 1 \text{ m/s}^2; \quad R_c = 19,45 \cdot 10^{-3} \text{ m}; \quad p_0 = 101325 \text{ Pa}$$

A constant convective velocity of 100m/s along the x direction is added to allow the vortex convective transport.

The numerical setup adopted was the same used in LES 3D calculation and it is here briefly summarized:

- Second-order implicit scheme for the unsteady formulation;
- Second-order discretization schemes for both pressure and momentum equations. In particular, the BCD scheme was adopted for the momentum equation;

Numerical results were compared with the exact solution (i.e. at $t=0$) in terms of relative pressure, and V_y (fluid velocity along y-direction) profiles obtained along a cutting line coincident with X axis. Figures from 3 to 6 show the influence of the convective time and different Courant number (Co) on the numerical pressure and V_y profiles.

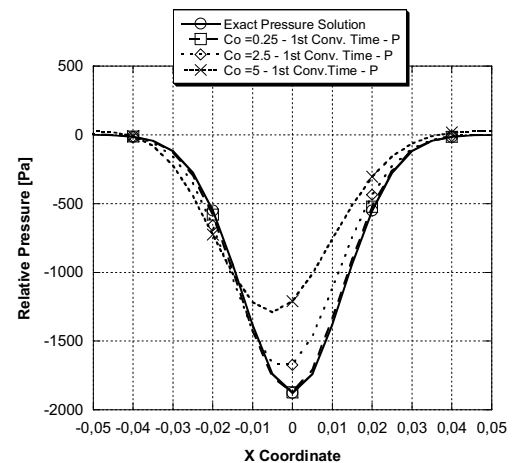


Figure 3: Relative pressure profile obtained after one convective time for different Courant number.

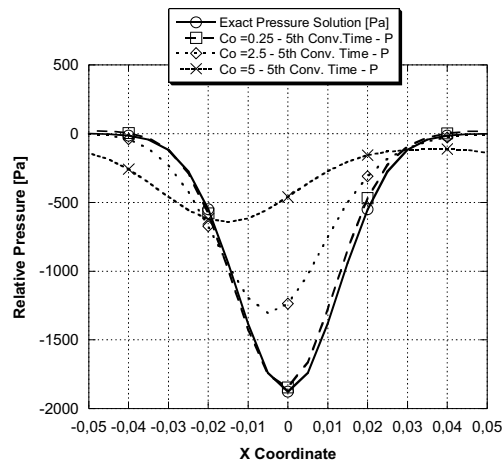


Figure 4: Relative pressure profile obtained after five convective times for different Courant number.

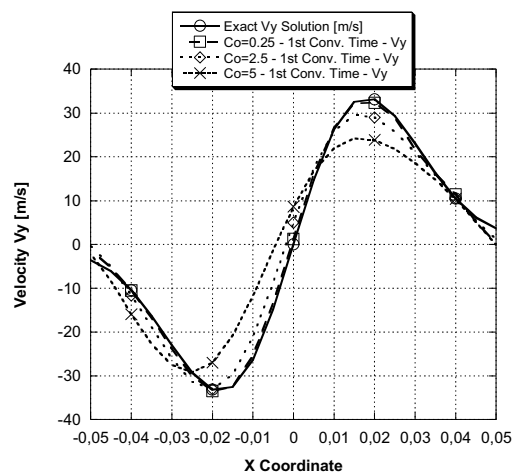


Figure 5: Vy profile obtained after one convective time for different Courant number.

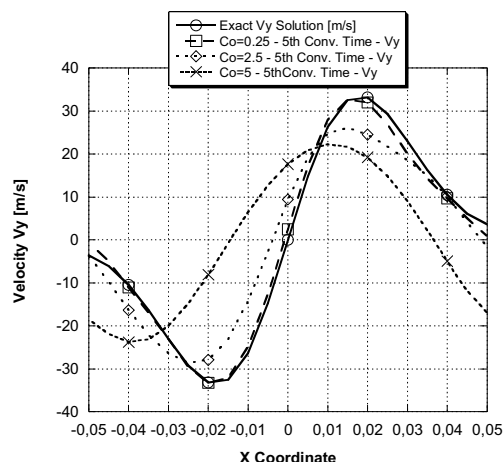


Figure 6: Vy profile obtained after five convective time for different Courant number.

As expected using a second order scheme like the BCD, increasing the maximum simulation Courant number, relative pressure and Vy profiles increase their difference

with respect to exact solution profiles. The differences increase in magnitude and phase also increasing the convective time since cut-off of the energy at higher frequency (i.e., higher wave number) is made.

On the basis of the numerical results obtained on the vortex test case and of the airbox computational mesh defined, the time step value for the airbox simulation was chosen to maintain the Courant number under the value of 1.5 on the whole computational domain. This value was fixed, according to Dugué's [6] investigation, as a compromise between the quality of solution and the computational time.

LES SIMULATION: PRELIMINARY ASSESSMENT

To numerically resolve high Reynolds number aerodynamic flows the required numerical effort is very high: using Direct Numerical Simulation (DNS) approach, the computational effort to resolve such turbulent flow increases with $\approx Re^3$. To overcome this prohibitively high computational cost the LES approach could be used but a fine simulation tuning has to be guaranteed in terms of mesh generation, and numerical set-up to render LES approach usable for industrial application. To reach this goal the LES simulation methodology here presented has been previously defined and tested by the authors on two different test cases:

- Backward Facing Step (BFS) test case [2];
- Steady engine intake flow [3];

In both cases a merit function was adopted to evaluate the influence of the different numerical set-up, and grid resolution tested on the LES solution level.

The BFS geometry is a standard test case useful to evaluate turbulence model characteristics. Performing LES simulations on the BFS geometry, the authors would check the influence of different parameters on numerical solution of the turbulent flow. In particular, the attention was focused on the following aspects:

- recording time window;
- cell type (hexahedral or tetrahedral cells);
- grid resolution (filter size);
- sgs closure model;

The definition of the recording time window is an important aspect about the evaluation of LES simulation results. In fact, adopting LES approach mean and *rms* flow values can be evaluated only performing statistical operation on the stored instantaneous flow values on a defined time window. BFS simulation results underlined the needed to hold at least one convective time before to start the result collection to avoid the influence of initial conditions on the time average procedure [10].

Regarding cell type influence, BFS simulations showed how the predicted mean and *rms* flow velocity profiles were comparable for hexahedral and tetrahedral cells because both types suffered from a comparable truncation errors in the zone where the flow direction was not aligned to the cell directions [11]. The main