

Investigation of the Flow Unsteadiness of Car Air-Box by Using LES

F. Brusiani, G. M. Bianchi

DIEM

University of Bologna

A. Bianchi d'Espinosa

Ferrari GeS S.p.A.

Copyright © 2009 SAE International

ABSTRACT

Today, high performance race car efficiency is based on a very fine equilibrium between aerodynamic efficiency, engine performance, and chassis behaviour. In particular, from the engine point of view, one way to increase the performance is to increase its volumetric efficiency.

The aim of this paper is to present the application of the Large Eddy Simulation (LES) approach for the fluid dynamic analysis of a high performance race car airbox geometry. For a naturally aspired engine, the fluid dynamic optimisation of the airbox geometry means to optimise the energy conversion (from dynamic to static pressure) inside the airbox itself, therefore to increase the flow energy on the engine trumpet sections.

The LES approach seems to be the best candidate to investigate such a flow since flow unsteadiness are expected to affect airbox efficiency in terms of pressure recovery.

The airbox simulations were performed by using the commercial CFD code Fluent v6.3. The Wall Adaptive Local Eddy-Viscosity (WALE) Sub-Grid Scale (SGS) model was adopted together with a bounded second order central differencing scheme. The LES methodology here adopted was validated by previous works.

In order to guarantee realistic fluid dynamic conditions on the airbox inlet section, a part of the car body was considered in the computational domain.

Results obtained by LES simulations were analysed in terms of mean and *rms* evolutions of both pressure and velocity components.

This study shows that, today, the LES technique is a very promising and proficient way to obtain detailed

information about flow unsteadiness also on industrial cases.

INTRODUCTION

In engineering applications most flows are turbulent, thus they are affected by rotational and 3D phenomena characterized by an high level of random fluctuating vorticity. For this reason, turbulence flow investigation still have a primary role in fluid dynamic analysis

From the previous considerations it clearly appears that the direct solution (DNS) of the time-dependent Navier-Stokes equations seems to be the only available approach to numerically reproduce the turbulence [1,2]. The numerical solution of high Reynolds flows by using DNS approach requires a very high numerical effort because a very fine discretization of the computational domain (from 10 to 100 μm) together with a very small time step (lower than 100 μs) are required.

Today, the computer power is still not enough to allow the application of DNS approach on real industrial cases. Usually, for industrial applications Reynolds Averaged Navier Stokes (RANS) simulation approach is adopted instead of the DNS approach [1,2]. Using the RANS approach the governing equations are time averaged and the turbulence is reproduced by using a numerical model able to guarantee a qualitative reconstruction of turbulence effects.

Actually, RANS approach is successfully applied in many industrial cases but it could not be considered an appropriate numerical solution when the main challenging simulation point is the prediction of the flow unsteadiness. In fact, the time average operation of the flow governing equations implies that the flow

fluctuations become zero by definition. To overcome this problem, Large Eddy Simulation (LES) approach could be considered.

LES simulation method is a physically grounded numerical method able to predict flow unsteadiness [3,4,5]. It is based on spatially filtered flow governing equations that allow the direct numerical solution of the largest scales of motion (GS). The small turbulence scales are modeled by using a Sub-Grid Scale model (SGS). The limit (cut-off) between GS and SGS is supposed to take place in the inertial sub-range (Figure 1). In particular, 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 [3]. Clearly, the LES computational cost is quite higher than a RANS simulation because of the finest domain discretization that is needed to directly solve the large scales of motion. At the same time, the LES computational cost is quite smaller than a DNS simulation because the small turbulence scales are modeled and not directly solved.

As stated by Pope [3], a simulation can be named as LES if at least the 80% of the turbulence energy spectrum is directly solved. When the filter (cell dimension) are too coarse to resolve the 80% of the flow energy, a Very Large Eddy Simulation (VLES) is performed.

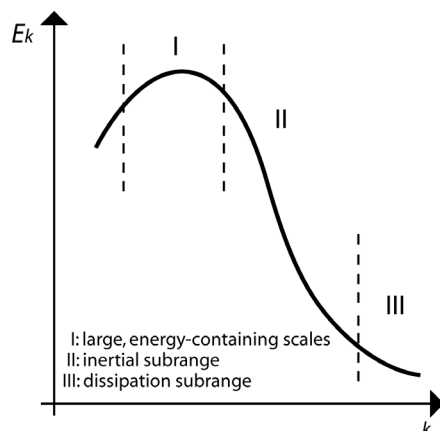


Figure 1: Energy cascade.

In the present paper Large Eddy Simulation (LES) approach was used to numerically reproduce the airbox fluid dynamic behavior. From the engine point of view, the airbox plays a fundamental role because:

- Generally, the airbox is the only link between the ambient external to the car and the engine, therefore its aerodynamic efficiency has a direct influence on engine volumetric efficiency;
- In high performance cars equipped with naturally aspirated engines, the airbox could assure a natural overcharge effect thanks to the conversion of dynamic pressure into static pressure (increase of fluid density on the cylinder trumpet sections);

- The airbox should minimize the unsteady effects induced by the intake process of each cylinder by a proper design;

From the previous considerations, it clearly appears the need to perform a fine airbox tuning. Moreover, this tuning has to be performed concurrently with the car chassis because of the strict correlation between these two elements.

As detailed after, to assure realistic fluid dynamic conditions on the airbox inlet section also a part of the car bodywork was considered in the computational domain.

The LES methodology here adopted was previously validated by the authors on two different test cases [6,7]. On the basis of the obtained results the WALE model for the SGS scales was chosen. No wall function was adopted.

PAPER OUTLINE

This paper was organized in three main areas. In the first part, the LES method is presented. In the second part, the considered computational domain is presented together with the mesh generation, the adopted simulation set-up, and the boundary conditions. In the last part, numerical results are presented. In particular, CFD simulations were performed in two subsequent steps:

- The first step was about a detailed unsteady RANS simulation. The goal of this simulation was to underline the overall characteristics of the mean turbulent flow identifying the domain zones where the fluid dynamic gradients were the highest. It was very useful to address the LES simulation set-up, especially from the mesh generation point of view (filter size distribution). Detailed description about RANS simulation methodology could be found at [1,2];
- The second step was about the LES simulation. The goal of this simulation was the reproduction of the main turbulent structures inside the airbox to evaluate their influence on the energy conversion process (from dynamic to static pressure) inside the airbox itself. As previously underlined it is one of the most important aspects that has to be studied to optimize the engine volumetric efficiency.

RANS and LES numerical results are presented in terms of mean pressure and velocity evolutions along flow paths identified inside the airbox geometry. Moreover, for LES simulation statistical analysis were performed on the recorded data to find out also the *rms* pressure and velocity evolutions along the same airbox flow paths.

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) [3,5]. 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 resolved component of $\phi(x)$ associated with the large eddies (GS);
- $\phi'(x)$ represents the unresolved component of $\phi(x)$ (SGS);

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 anisotropy.

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:

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

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

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

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

The subgrid-scale stresses are unknown terms that must be modeled. The eddy-viscosity approach [3,4,5] 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 ν_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 [8,9]. 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.

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 numerical scheme attitude is mandatory.

At this scope, the test case proposed by Moreau et al. [10], the in viscid 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.

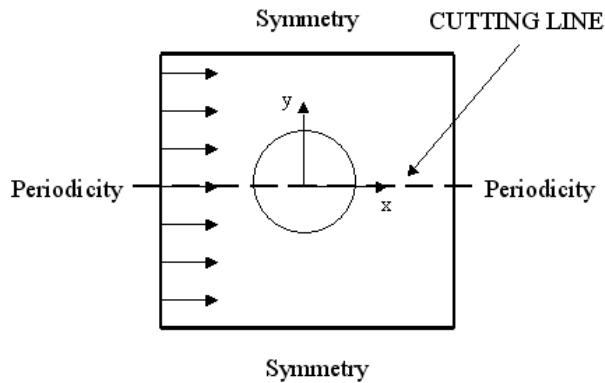


Figure 2: Computational domain and boundary conditions adopted to reproduce the periodic Eulerian vortex test case proposed by Moreau et al [10].

A structured 2.5D mesh with 18000 hexahedral cells was generated (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:

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

A constant convective velocity of 100m/s along the x direction was 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;

Numerical results were compared with the exact solution in terms of relative pressure, and V_y (fluid velocity along y-direction) profiles obtained along the X axis. For example, Figures 3 and 4 show the influence of Courant number (Co) on the numerical pressure and V_y profiles (a detailed presentation of this test case could be found in [11]).

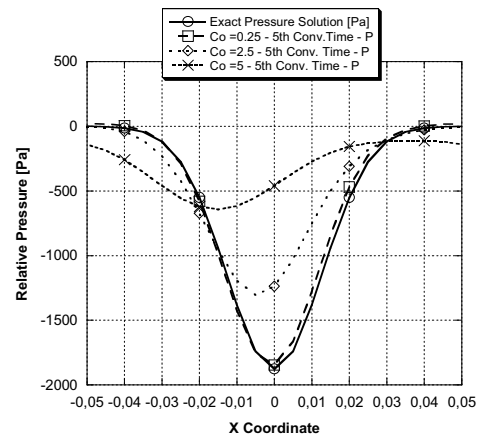


Figure 3: Relative pressure profile obtained after five convective times for different Courant number.

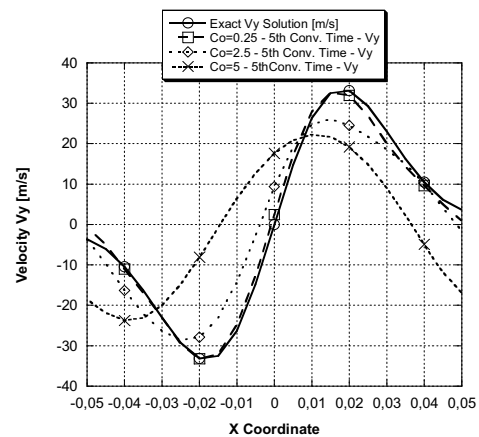


Figure 4: V_y profile obtained after five convective time for different Courant number.

As expected using a second order scheme like the BCD, increasing the maximum simulation Courant number, relative pressure and V_y profiles increase their difference with respect to exact solution profiles.

LES SIMULATION METHODOLOGY

To perform a LES simulation it is necessary to perform a fine tuning of the simulation methodology (mesh generation and numerical set-up). The simulation methodology here adopted was previously defined and tested on two different cases [6,7]:

- Backward Facing Step (BFS) test case;
- Steady engine intake flow test case;

The BFS geometry is a standard test case to evaluate turbulence model performance. In particular, this test case was useful to check the influence of different parameters on LES numerical solution as:

- 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 because mean and *rms* flow values are evaluated performing statistical operations on the stored instantaneous flow values on a defined time window. BFS simulation results underlined the need 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 [12].

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 comparable truncation errors in the zone where the flow direction was not aligned to the cell directions [13]. The main differences observed between the usage of hexahedral and tetrahedral cell type were about the mesh generation time and the computational cost of the simulation. In fact, with respect to a fully structured hexahedral mesh, generally the mesh generation time of a tetrahedral mesh is shorter but its total cell number is greater.

About spatial discretization influence on numerical results, it was found that increasing the number of cell (and so decreasing the filter length) the numerical results became closer to the experimental ones since the fraction of modeled energy was smaller [3]. The assessment of the LES quality in terms of energy resolved fraction was performed according to the procedure defined by Pope [3].

As known, different SGS models can be adopted to model the small turbulent scales. The authors tested two SGS models: WALE model [8,9] (with and without Werner-Wengle wall function [14]) and the One Equation Localized Dynamic Kinetic Energy model (LDKEM) proposed by Kim and Menon [13,15,16]. The mean and *rms* velocity profiles obtained using the LDKEM closure model were closer to experimental profiles than using the WALE closure model. On the other hand, the WALE closure model requires a smaller total number of cells in relation to the LDKEM closure model guaranteeing, at the same time, a reasonable engineering prediction of large flow scales far from walls [6].

After the definition of the LES methodology on the BFS test case, it was used to predict the flow on a steady engine intake flow bench configuration simulated by Thobois et. al. [6,7,17]. Numerical results were compared with Laser Doppler Anemometry (LDA) measurements of the mean and *rms* fluctuation values on both axial and tangential velocity components. Numerical results obtained by using the WALE SGS model without wall function (the same approach adopted in this paper) were in reasonable agreement with experimental and LDKEM results, encouraging the use of this approach also on real industrial cases.

FLOW SOLVER AND NUMERICAL SET-UP

Both RANS and LES simulations were made by Fluent v6.3 CFD code [18].

RANS simulation was performed by adopting a segregated flow solver together with a fully implicit first-order scheme [18]. Convective and diffusive fluxes were discretized by using Central Differencing (CD) scheme [1,2], while energy and turbulence equations were discretized by using Up-Wind scheme [1,2]. As turbulence model, a standard *k-ε* model was adopted [2]. About near wall treatment, standard wall function approach was adopted [2].

LES simulation was performed by adopting a segregated flow solver together with a fully implicit second-order scheme [18]. Diffusive fluxes were discretized by using CD schemes, while convective fluxes were treated by using the so called Bounded Central Differencing (BCD) second order scheme [18,19], developed to simulate complex industrial flows involving High-Reynolds number. In particular, BCD scheme was designed to suppress unphysical wiggles that could be numerically introduced by the pure CD scheme preserving, at the same time, CD's non-dissipative property. BCD scheme is also suited for hybrid grids as required in engineering applications [11,20]. No wall function was adopted for the LES simulation.

The time step value for the airbox simulations was chosen to maintain the Courant number under the value of 1.5 at least inside the airbox volume. This value was fixed, according to Duguè's [21] investigation, as a compromise between the quality of solution and the computational cost.

COMPUTATIONAL DOMAIN

As previously underlined, the main goal of this work was to study the fluid dynamic behavior of a real airbox geometry by LES simulation methodology. Realistic fluid dynamic conditions on the inlet airbox section were guaranteed by considering a portion of the car mockup (Figure 5). The car mockup was placed inside a box that represented the ambient external to the car. Indicating with A^2 the box cross section area, the longitudinal box length was set to $1,5 \cdot A$. The relation between the A dimension of the computational box and the mockup dimension was chosen to guarantee a velocity field on