

Embedded Large Eddy Simulation of Flow around the Ahmed Body

Domenico Caridi and Davor Cokljat
ANSYS UK

Jochen Schuetze and Richard Lechner
ANSYS Inc.

ABSTRACT

Airflow over the Ahmed body is simulated by means of the Embedded LES technique. This is a zonal approach which allows the LES turbulence model to be used in a sub-domain while the rest of the domain is solved using a RANS turbulence model, i.e. the $k\omega$ -SST model. This allows the LES zone to be restricted to the rear end of the vehicle where the unsteadiness of the flow must be accurately predicted. A comparison with full RANS modeling and experiments is reported.

CITATION: Caridi, D., Cokljat, D., Schuetze, J. and Lechner, R., "Embedded Large Eddy Simulation of Flow around the Ahmed Body," *SAE Int. J. Commer. Veh.* 5(1):2012, doi:10.4271/2012-01-0587.

INTRODUCTION

The Ahmed body is now accepted as a test-case representative of a typical modern passenger car, which possesses all the typical aerodynamic features found in the flow around a vehicle, a bluff body with a separated boundary layer generating a wake resulting in pressure drag. Several types of vortical structures arise in the wake, which are quasi-stationary and highly dynamic in nature.

Despite the fact that RANS simulations are successful in predicting many parts of the flow around vehicles, they usually fail to predict the effects of separation and the unsteady wake on the body. Large Eddy Simulation can have greater success than RANS in predicting the pressure and velocity field at the rear of vehicles. The main obstacle facing the LES simulations, however, is that the near wall resolution required to represent the structure of the flow in the boundary layer, where most of the turbulence energy is produced and dissipated, can be far too expensive.

The flow field above the slant and in the wake of the Ahmed reference model at a sub-critical angle is considered to be complex due to the presence of a three-dimensional turbulent boundary layer and a recirculation that consists of longitudinal contra-rotating vortices. The pressure drag in the wake is the major component of total drag acting on the vehicle, and is directly dependent on the rear slant angle. A detailed study on the dependence of the drag coefficient on the rear slant angle has been performed both experimentally

[1], [2] and numerically by many authors [3], 4, 5, [6]. The critical rear slant angle is found to be 30 degrees, above which there is a sudden drop in the drag coefficient as the flow fully separates over the rear slant. In the case of slant angles between 15 and 30 degrees, however, there are strong contra-rotating vortices and the flow separates from the top of the sloping surface before reattaching lower down. The wake flow is highly unsteady and unstable and this is probably why RANS simulations perform poorly for these angles.

The present study focuses on the 25 degree slant Ahmed body case which has been the subject of different LES studies in the past. For example, Hinterberger et al. (2004), [6], performed a LES study using 18.5×10^6 computational cells whereby wall functions were used at the walls. The results in the lower part of the slant were in poor agreement with the experiments of Lienhart and Becker (2003), [2]. Furthermore, Krajnovic and Davidson (2004) performed a LES simulation at a lower Reynolds number, arguing that the flow becomes Reynolds independent, [5]. This hypothesis, and the fact that separation is induced by the geometry, allowed a better resolution of the boundary layer. This might not be suitable for today's realistic car geometries, however, that may involve rounded edges. In that case, a separation might be dependent on the Reynolds quantities, e.g. the size of the boundary layer.

To overcome this difficulty, the goal of the current study is to demonstrate the application of a multi-domain RANS-

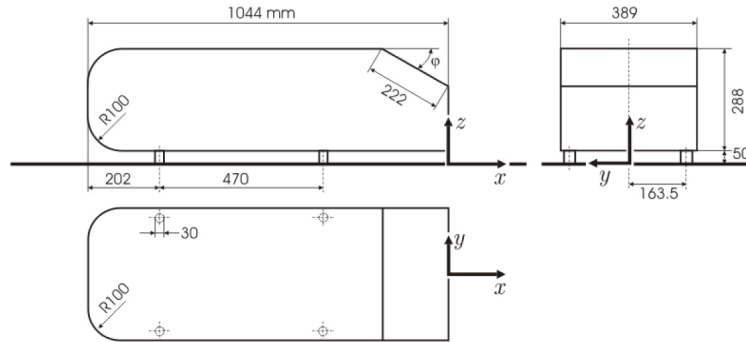


Figure 1. Ahmed body geometry.

LES approach to the simulation of the Ahmed body. A LES simulation that includes the whole Ahmed body at a realistic Reynolds number would require prohibitively large computational resources. Restricting the LES zone to the rear end of the Ahmed body could substantially reduce the computational resources required as the rest of the domain will be solved using a much cheaper RANS simulation. This approach is known as an Embedded LES Simulation, ELES, and requires the formulation and implementation of interface conditions which provide a physically correct transfer of information between the models involved on both sides of the interface. The method presented in this paper is based on the vortex method proposed by Sergeant et al. (2002) for LES-RANS coupling, [7], which was adapted to general flow situations in Mathey et al. (2003), [8]. Full methodology used in this paper is fully described in Cokljat et al [9].

Simulations were performed with the commercial code ANSYS FLUENT 13.0.

THE TEST CASE

The Ahmed body, Figure 1, was first defined and its characteristics described in the experimental work of Ahmed [1]. The experiments to which this paper will refer are the detailed LDA Measurements performed by Becker and Lienhart [2] in the LSTM low-speed wind-tunnel with a cross-section of $1.87 \times 1.4 \text{ m}^2$ (width $\times$ height) with a bulk velocity of 40 m/s.

The test-section of the wind-tunnel was 3/4 open (only ground plate present). The distance between the body and the plate representing the ground is 50 mm. The height of the body is 288 mm and the Reynolds number based on the height of the body is 768,000.

NUMERICAL METHOD

RANS

The turbulence model used in this study is the Shear Stress Transport (SST) $k-\omega$ model developed by Menter (1994), which is designed to overcome some of the limitations of the original $k-\omega$ model by Wilcox (1988). The following list compares the strengths and weaknesses of the $k-\omega$ and $k-\varepsilon$ models:

- The original 1988 standard $k-\omega$ model is very sensitive to the free stream value of ω (a non-zero and non-physical boundary condition for the ω equation is required), while the $k-\varepsilon$ model is not prone to such problems.
- The $k-\omega$ model has many good attributes and performs much better than $k-\varepsilon$ models for boundary layer flows.
- Most two-equation models, including $k-\varepsilon$ models, over-predict turbulent stresses in the wake (velocity-defect) region, which leads to poor performance of the models for boundary layers under adverse pressure gradient and separated flows.

Menter's SST model effectively blends the robust and accurate formulation of the $k-\omega$ model in the near-wall region with the free-stream independence of the $k-\varepsilon$ model in the far field, [10].

The $k-\omega$ model equations for the inner layer are taken from the original Wilcox $k-\omega$ model with some constants modified. The turbulent viscosity μ_t is then computed according to the following formula:

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{\max \left[\frac{1}{\alpha^*}, \frac{SF_2}{a_1 \omega} \right]} \quad (1)$$

where S is the strain rate magnitude, F_2 is a blending function, a_1 is a constant, and α^* damps the turbulent viscosity causing a low-Reynolds-number correction and is expressed by

$$\alpha^* = \alpha^*_{\infty} \left(\frac{\alpha^*_0 + Re_t/R_k}{1 + Re_t/R_k} \right) \quad (2)$$

where α^*_{∞} , α^*_0 , R_k are constant and $Re_t = (\rho k / \mu \omega)$

The $k-\omega$ model equations for the outer layer are obtained by transforming the standard $k-\varepsilon$ equations via change of variable. The turbulent viscosity is then computed as

$$\mu_t = \frac{\rho k}{\omega} \quad (3)$$

The two sets of equations and the model constants for the inner and outer layer are blended in such a way that the resulting equation set transitions smoothly from one equation to another.

LES

In the LES approach, the turbulent scales are decomposed into large resolved scales, which are likely to be anisotropic, and subgrid unresolved scales (which may or may not be isotropic) by applying a spatial filter to the Navier-Stokes equations. The subgrid-scale stresses (SGS) resulting from the filtering operation are unknown, and require modeling. The subgrid-scale turbulence models in ANSYS FLUENT employ the Boussinesq hypothesis as in the RANS models, computing subgrid-scale turbulent stresses from

$$\tau_{ij} - \frac{1}{3}\tau_{kk}\delta_{ij} = -2\mu_t(\bar{S}_{ij}) \quad (4)$$

In the present work the WALE model (Wall Adapting Local Eddy Viscosity) is used. It contains a near-wall modification in the expression for the SGS viscosity which adapts to the local near-wall flow structure and accounts for wall damping effects without using a damping function. Thus the eddy viscosity is modeled by

$$\mu_t = \rho(C_s\bar{\Delta})^2 \frac{(S_{ij}^d S_{ij}^d)^{\frac{3}{2}}}{(\bar{S}_{ij}\bar{S}_{ij})^{\frac{5}{2}} + (S_{ij}^d S_{ij}^d)^{\frac{5}{4}}} \quad (5)$$

where C_s is the Smagorinsky constant, $\bar{\Delta}$ is the local grid scale and $\bar{S}_{ij}$ is the rate-of-strain tensor for the resolved scale.

RANS-LES INTERFACE

In order to generate the small scale fluctuations at the RANS-LES interface (i.e. flow entering LES region from RANS region), a random vortex method (VM hereafter) is considered [8]. With this approach, a perturbation is added to a specified mean velocity profile via a fluctuating two-dimensional vorticity field (two-dimensional in the plane normal to the streamwise direction). The vortex method is based on the Lagrangian form of the 2D evolution equation of the vorticity and the Biot-Savart law. A particle discretization is used in order to solve this equation. These particles or "vortex points" are convected randomly and carry information about the vorticity field. If N is the number of vortex points and S the area of the RANS-LES interface, the amount of vorticity carried by a given particle i is represented by the circulation Γ_i and an assumed spatial distribution η :

$$\Gamma_i(x, y) = 4 \sqrt{\frac{\pi S k(x, y)}{3N(2 \ln(3) - 3 \ln(2))}} \quad (6)$$

$$\eta(\vec{x}) = \frac{1}{2\pi\sigma^2} \left(2e^{-\frac{|\vec{x}|^2}{2\sigma^2}} - 1 \right) 2e^{-\frac{|\vec{x}|^2}{2\sigma^2}} \quad (7)$$

where k is turbulence kinetic energy. The parameter σ provides control over the size of a vortex particle. The resulting discretization for the velocity field is thus given by:

$$\vec{u}(\vec{x}) = \frac{1}{2\pi} \sum_{i=1}^N \Gamma_i \frac{((\vec{x}_i - \vec{x}) \times \vec{z})(1 - e^{-\frac{|\vec{x} - \vec{x}_i|^2}{2\sigma^2}})}{|\vec{x} - \vec{x}_i|^2} \quad (8)$$

LES-RANS INTERFACE

A pre-cursor RANS steady state simulation was performed initially and used thereafter as a starting field for unsteady simulation, which is a common practice used for majority of unsteady simulations. RANS turbulence quantities obtained in such a way were used to compute turbulent quantities, required at the LES-RANS interface (i.e. flow exiting LES region into RANS region). It has been proven in the past (e.g. [9]) that this concept works reasonably well on variety of different flows.

SOLVER SETTINGS

The Pressure Based solver is used. The spatial discretization of the convective terms in the momentum equation is computed via a Bounded Central Difference scheme which ensures very low numerical diffusion while a Second Order Upwind scheme is used for the other equations. Pressure-velocity coupling is achieved by using SIMPLEC which is a variant of the basic SIMPLE algorithm. This can speed up the convergence within each time-step as the under-relaxation factor can be set to high values, 0.9-1.0. The Green Gauss Node Based method is used for the gradient calculation. The transient formulation is Iterative Time-Advancement Second Order Implicit.

GRID AND SIMULATION SET-UP

The fluid domain dimensions are: 9.5, 3.2, 2.0 m. The Ahmed body is placed at a distance of about 20% of the domain length from the inlet. The simulations were carried out on a 4.8 million cell hybrid grid, Figure 2 shows a close up of the body. The RANS mesh consists of 15 prism layers and a tetrahedral volume mesh totaling 1.6 million elements, while in the LES domain a 3.2 million hexahedral grid was generated, Figure 2, 3. It is important that a correct and resolved boundary layer profile is provided as input for the LES zone at the RANS-LES interface. The first two prism