

NUMERICAL SIMULATION OF THE FLOW AROUND THE AHMED VEHICLE MODEL

G. FRANCK^{†‡}, N. NIGRO[‡], M. STORTI[‡] and J. D'ELÍA[‡]

[†] *Universidad Tecnológica Nacional (UTN), Facultad Regional Santa Fe (FRSF)
Lavaise 610, 3000-Santa Fe, Argentina, gfranck@intec.unl.edu.ar, http://www.utn.edu.ar*

[‡] *Centro Internacional de Métodos Computacionales en Ingeniería (CIMEC)
Instituto de Desarrollo Tecnológico para la Industria Química (INTEC)
Universidad Nacional del Litoral - CONICET, Güemes 3450, 3000-Santa Fe, ARGENTINA
ph.: (54342)-4511 594/5, fx: (54342)-4511 169
(nnigro, mstorti, jdelia)@intec.unl.edu.ar http://www.cimec.org.ar*

Abstract— The unsteady flow around the Ahmed vehicle model is numerically solved for a Reynolds number of 4.25 million based on the model length. A viscous and incompressible fluid flow of Newtonian type governed by the Navier-Stokes equations is assumed. A Large Eddy Simulation (LES) technique is applied together with the Smagorinsky model as Subgrid Scale Modeling (SGM) and a slightly modified van Driest near-wall damping. A monolithic computational code based on the finite element method is used, with linear basis functions for both pressure and velocity fields, stabilized by means of the Streamline Upwind Petrov-Galerkin (SUPG) scheme combined with the Pressure Stabilizing Petrov-Galerkin (PSPG) one. Parallel computing on a Beowulf cluster with a domain decomposition technique for solving the algebraic system is used. The flow analysis is focused on the near-wake region, where the coherent macro structures are estimated through the second invariant of the velocity gradient (or Q-criterion) applied on the time-average flow. It is verified that the topological features of the time-average flow are independent of the averaging time T and grid-size

Keywords— Ahmed vehicle, bluff aerodynamics, incompressible viscous fluid, large eddy simulation (LES), time-average flow, finite element method, fluid mechanics.

I. INTRODUCTION

A. The Ahmed vehicle model

Ground vehicles can be classified as bluff-bodies that move close to the road surface and are fully submerged in the fluid. In general, for the usual velocities of commercial passenger cars, buses and trucks, compressible effects can be neglected and an incompressible viscous fluid model can be assumed. As the Reynolds numbers based on the body length are usually too high, the flow regimes are fully turbulent. In addition, a key feature of the flow field around a ground vehicle is the presence of several separated flow regions, while the net aerodynamic force is the result of complicated interactions among them. Even simple basic vehicle configurations with smooth surfaces, free from appendages and wheels, generate a variety of quasi two dimensional and fully

three dimensional regions of separated flows where the largest one is the wake. In a time-averaged sense, the separated flow regions exhibit complicated kinematic macro structures and those present in the wake determine mostly the body drag. Nowadays, numerical simulations, wind tunnel and road tests are jointly used in the automotive industry for the aerodynamic study from several perspectives.

The Ahmed vehicle model is a very simplified bluff-body which is frequently employed as a benchmark in vehicle aerodynamics. It has been used in several experiments (Ahmed *et al.*, 1984; Sims-Williams, 1998; Bayraktar *et al.*, 2001; Spohn and Gilliéron, 2002; Lienhart and Becker, 2003) and numerical studies (Han, 1989; Basara *et al.*, 2001; Basara, 1999; Basara and Alajbegovic, 1998; Gilliéron and Chometon, 1999; Kapadia *et al.*, 2003). A slightly modified version was also studied by Duell and George (1999); Barlow *et al.* (1999); Krajnović and Davidson (2003).

The shape of this body is free from all accessories and wheels but it still retains the primary behavior of the vehicle aerodynamics, as seen in Fig. 1. Special attention is focused on the time-averaged flow in the near-wake and the dependence of drag on the slant angle α at the top rear end. The rear end is a simplification of a so-called fastback one such as on a Volkswagen Golf I. This model was tested in open jet wind tunnels and no velocity distributions were reported in the inlet and outlet boundaries (Ahmed *et al.*, 1984). Previous numerical simulations had assumed the incoming flow to be laminar and steady (Han, 1989; Krajnović and Davidson, 2004; Basara and Alajbegovic, 1998). Wind tunnel experiments on this model show two critical slant angles: $\alpha_m \approx 12.5^\circ$ and $\alpha_M \approx 30.0^\circ$, called first and second critical angles, respectively, where the main topological structure of the time-averaged flow in the near-wake changes significantly as summarized in Table 1 (Ahmed *et al.*, 1984; Sims-Williams, 1998; Bayraktar *et al.*, 2001; Spohn and Gilliéron, 2002; Lienhart and Becker, 2003).

Among the experimental tests, Janssen and Hucho (1975) showed the dependence of the flow on the slant angle at the rear end for an industrial vehicle model, while Morel (1978) performed tests on the Morel body which is previous to the Ahmed one. Both models rep-