CFD modeling of the flow around the Ahmed vehicle model

Gerardo Franck[1,2] and Jorge D'Elía[2]

1: *Aula CIMNE-FICH, UNL, Ruta Nac. 168, km 472, 3000-Santa Fe*
2: *Centro Internacional de Métodos Computacionales en Ingeniería (CIMEC)*
INTEC (UNL-CONICET), PTLC, 3000-Santa Fe, Argentina
http://www.cimec.org.ar, e-mail: cimec@ceride.gov.ar,
ph: 54-342-4511594, fx: 54-342-4511595

SUMMARY. Current vehicle development needs a strong background in aerodynamics to improve flow control by means of active or passive control devices. The complexity involved in the automobile design specially due to the great amount of accessories and devices that form its geometry makes the validation tasks unaffordable. The Ahmed model is a simple geometric body that retains the main flow features, specially the vortex wake flow where most part of the drag is concentrated and it is a good candidate to be used as a benchmark test. In this work a large eddy simulation turbulence model is employed. We compare our results about the detailed flow patterns with those previously published by Ahmed and coworkers. We have used GiD for post-processing the 3D flow patterns. Flow visualizations through streamlines, isosurfaces and isolines are presented on the forebody part, rear end and neighboring wake. We focus specially on the distinctive features of the flow around bluff bodies.

KEY WORDS. Vehicle aerodynamics, bluff bodies, vortex wake flow, finite element solution, parallel computing, fluid mechanics

INTRODUCTION

In order to shorten lead time, lower experimental work and lower the costs associated with vehicle design work, aerodynamics specialists are constantly seeking new ideas and solutions capable of providing a fast and accurate answer to the design targets. One way to obtain this goal is combining numerical simulation with experimental measurements in wind tunnel tests. However, the current state of the art in the computational fluid dynamics shows that over the last years accurate results for the automotive aerodynamics expectations have appeared. One of the key points in the development of computational codes is its validation with experimental results.

The aerodynamics forces on road vehicles are the result of complex interactions between flow separations and the dynamic behavior of the released vortex wake. This work presents a large eddy simulations (LES) of the Ahmed reference model. LES allows a much greater depth of analysis than most other turbulent simulations methods. The body geometry is analysed at a given slant angle of 12.5^o using the PETSc-FEM code platform. We be used a GiD as a pre and post-processing results, previously transform files as to be read for this code.

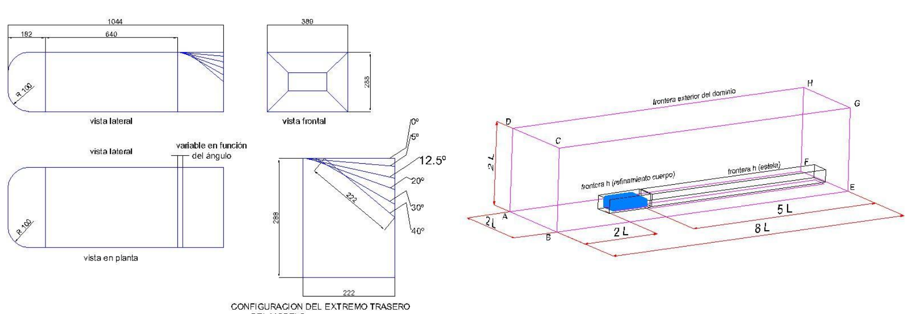

Figure 1: Left: geometrical dimensions of the Ahmed model. Right: computacional flow domain.

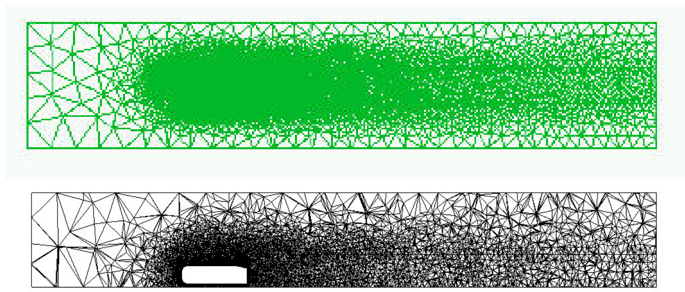

Figure 2: Top: meshing on the floor surface. Down: a longitudinal cut with some refined mesh zones.

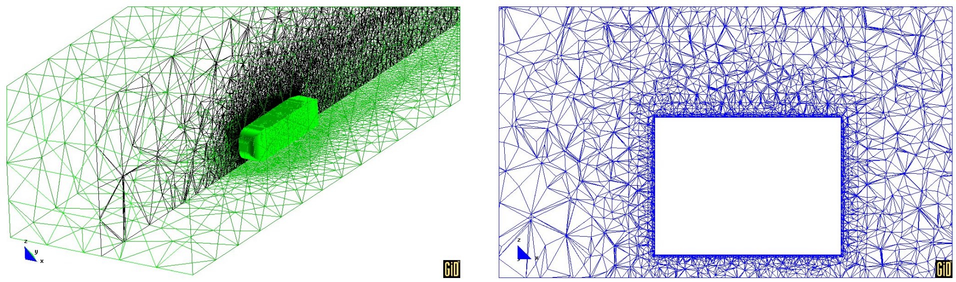

Figure 3: Top: a longitudinal cut with the boundary domain. Down: a transversal cut of the flow domain.

GEOMETRICAL DESCRIPTION AND NUMERICAL SCHEME

The Ahmed reference model is a generic car type bluff body shape which is a enough simple for accurate flow simulation but retains some important practical relevant features of automobile bodies. The body geometry is shown in Fig.1. The flow domain chosen is one in which the body of length L is suspended to $0.05m$ to the ground in a domain of $10L \times 2L \times 1.5L$ in the streamwise (y), spanwise (x) and stream-normal (z) directions. The boundary conditions for the problem are uniform flow at the inlet, slip at both sides, no-slip for the surface of the body, Dirichlet at the floor with velocity equal to inlet and imposed pressure is used as the outflow boundary condition.

In this paper the non-structures tetrahedral grid approach is applied to the same geometry at $Re = 4.25 \times 10^6$. This grid presents wedge elements for simulations of boundary layers. The accuracy of using wedge elements in the very thin boundary layer portion of the mesh, will capture and develop a flow field around the bluff body. A special companion program is used to attach three layers of wedge elements to the tetrahedral element mesh so as to construct the boundary layers of wedge elements into tetrahedral elements to create a mesh composed entirely of tetrahedral elements. The wedge element is defined as an element composed of six nodes as in Fig.4. Because we are using meshes with both tetrahedral and wedge element in the domain, special consideration is needed for the parallel implementation. The surface mesh for our model contains 25.000 nodes and 51.000 triangular elements. The automatic mesh generator of GiD created a 3D mesh with 87.000 nodes and 450.000 tetrahedral elements. The final mesh contains 162.000 nodes and 903.000 tetrahedral elements equivalent. The figures 2and 3 shows this mesh.

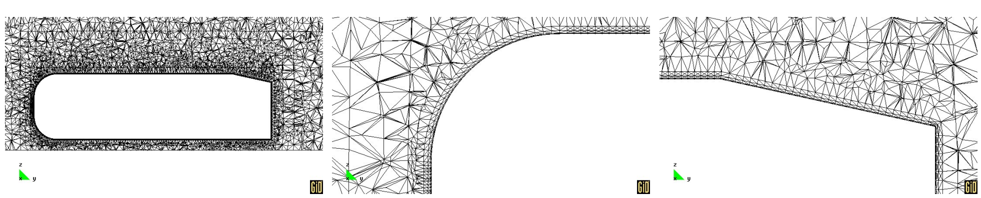

Figure 4: Details on the structured layer: a longitudinal cut (left); on the body front (center) and the body rear (right).

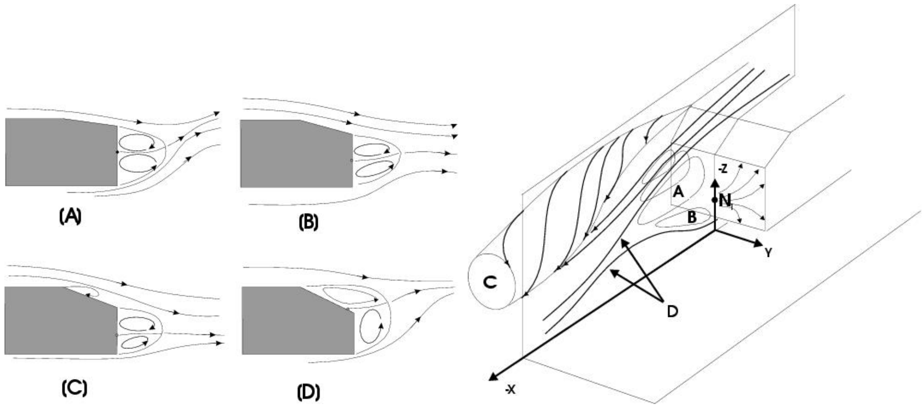

Figure 5: Left: flow behind the rear body as a function of the inclination angle. Right: flow behind the rear body for an inclination angle of 12.5°.

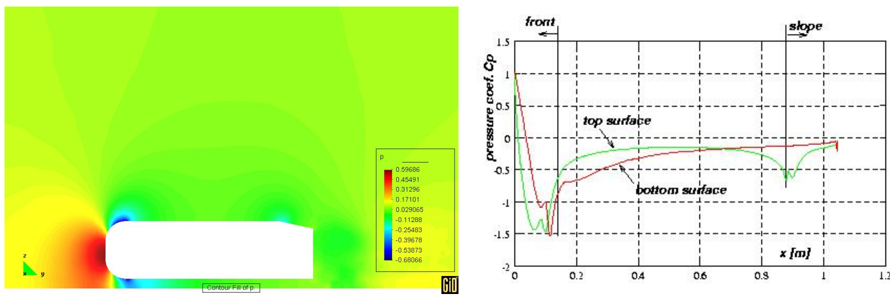

Figure 6: Left: contour fill field pressure Right: pressure coeficient over Ahmed body.

This flow was solved using an incompressible Navier-Stokes formulations typical of CFD area named SUPG-PSPG from Tezduyar, *et al.* [2](1992). To do this we used our code called PETSc-FEM developed in C++ and based on MPI and PETSc library routines.

DRAG AND PRESSURE MEASUREMENTS

See below table 1 calculate the main drag values for different slant angles and for different parts of the Ahmed body.

Here we can observe that the contribution of the front part of the body in very small compared with the total pressure drag. The interaction between the fore body and the rear end part is weak as a consequence of the long distance between these two parts in terms of flow features. As the flow is subsonic interactions might be present if the mid section is shortened. Therefore, the major contributions to the pressure drag comes from the slant and vertical base rear part. We only analyze the pressure drag contributions it should be mentioned that the viscous effects add some drag mostly at the lateral parts of the body.

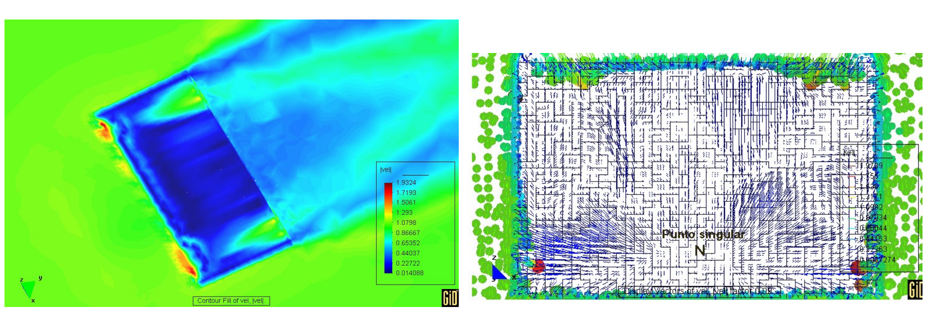

Figure 7: Left: contour fill for the speed (practically 2D) over the slant surface of the Ahmed model. Right: velocity over the basis and location of the singular point N.

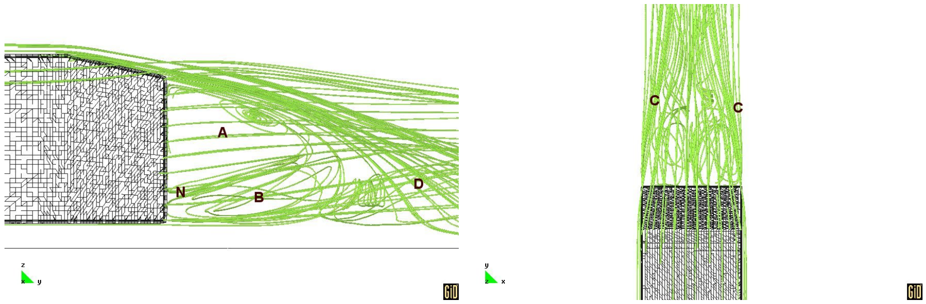

Figure 8: Streamlines in the near wake. Left: longitudinal plane. Right: plant plane.

Drag coefficients: Ahmed vs. numerical				
α	$\mathbf{C}_D$	$\mathbf{C}_{Df}$	$\mathbf{C}_{Ds}$	$\mathbf{C}_{Db}$
Ahmed 12.5°	0.2300	0.0160	0.0370	0.1220
Numerical 12.5° $slant = 176mm$	0.2346	0.0230	0.0385	0.1658
% Difference	+2 %	+43.75 %	+4.05 %	+35.90 %

Table 1: Drag force descomposition for $\alpha = 12.5°$.

In relation to pressure measurements in the unfolded rear end we observe: the presence of vortices at side edges of the slant surface; the flow on the slant appears to be two dimensional with parallel isobars running across the surface. The Fig.6 shows a contour fill and contour lines plot of the pressure field and right shows the variations of the pressure coefficient on the top and bottom surface of the body.

WAKE STRUCTURE

The physical features of the flow are discussed in two fundamental experiment works (Morel and Ahmed *et al*[1]) Due to the inherent turbulence the flow is completely three dimensional and unsteady. Using a time average flow we may put in evidence the existence of some sort of macrostructure that appears to govern the pressure drag created ay the rear end. Moreover, there are some particular situations where the vortices are organized in such manner that a two dimensional flow pattern may be found. The Fig. left 7 shows contour fill a flow field velocity in a slant surface. The slant and the vertical base of the rear end have edges that allow the formation of vortices by a rolled up of the shear layer. The side edges create a longitudinal vortex indicated in figure as C and the top and bottom edges of the vertical base create two recirculatory flows A and B situated one over the other. Experimental evidence does not indicate that these two flow regions end on the base surface; therefore these two recirculatory flow may seem to have been generated through two horseshoes vortices located one over the other in the separation bubble indicated as D. This vortices determine a singular point N over the base such as show Fig. right 7. The Fig.8 shows the streamlines in the wake structure.

CONCLUSIONS

Computed results obtained on the Ahmed reference model using PETCs-FEM code are in agreement with experimental results. In particular, computation successfully reproduces chages in vortex wake airflows and aerodynamics drag coefficients during transitions in critical angle of 12.5^o.Our results show numerical simulations to be a highly promising technique to research on physical phenomena in automotive aerodynamics.

References

[1] S.R. Ahmed, G. Ramm, and G. Faltin. Some salient features of the times-averaged ground vehicle wake. *SAE Society of Automotive Eng., Inc*, 1(840300):1–31, 1984.

[2] T. Tezduyar, S. Mittal, S. Ray, and R. Shih. Incompressible flow computations with stabilized bilinear and linear equal order interpolation velocity-pressure elements. *Comp. Meth. Applied Mechanics and Engineering*, 95(95):221–242, 1992.