TECHNICAL UNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics and Mechanics
Vol. 55, Issue III, 2012

GAS-DYNAMICS INFLUENCES IN RUNNING CONDITIONS OF TWO VEHICLES

Mircea DIUDEA, Sergiu MARCIUC, Radu BALAN, Victor HODOR

Abstract: *This paper is intended as a study vehicle running at speed on the highway, especially the case when a car exceeds a truck running on a side band. Will be studied in terms of pressure fields aerodynamic interactions between the two vehicles in order to determine the variation of pressure forces on the surface on the side of the car at different times of performing overcome*
Key words: *Vehicles, Running, CFD prediction, Velocity field, Pressure field*

1. INTRODUCTION

In order to increase safety in running vehicles, there have been done a large number of studies on the behavior of vehicles subject to the action of air currents that form around the body in different situations depending on the conditions studied (running at high speed, running in tunnels, traffic on bridges, etc.). The study is important in order to maintain a travel path as close to the first one, in the given conditions as, the rolling system from vehicles are flexible, and that react differently to the occurrence of external factors.

2. OVERVIEW OF THE INTEREST

The present analysis has been performed using a commercial CFD program called Ansys. The computational domain has two space dimensions and time. Spatial discreetizating is done using the control volume approach on a staggered, structured, non-uniform grid with convective terms treated using a weighted average of first order accurate unwinding and second order accurate central differencing according to the limits of numerical stability. The method is robust but does introduce numerical errors of a diffusive nature where the flow is strongly convective and skewed to the coordinate direction. Temporal discreetizating is

uniformly spaced and uses an implicit first-order differencing method. Details of the numerical method can be found in Refs. [6,3].The present flow problem is inherently unsteady, not only because of the flow turbulence, but also because of the large scale vortex shedding present in the separated wake behind the truck and another car. The turbulence away from the boundaries was modeled by a constant eddy viscosity of 1000 times the molecular value for air under standard conditions. Near the solid boundaries a log law was used to calculate the wall shear stress from the velocity at the closest node to the boundary. A more complex model of turbulence, the k - e model [7], was also tested on this problem with the result of increasing the number of steps required for convergence several fold and changing aerodynamic drag by only a few percent. Consequently, all solutions presented were calculated using the constant eddy viscosity model of turbulence. More recently formulated versions of the standard k -e model (for example RNG) and Reynolds Stress Models [7] were not available for testing but it should be expected that such methods would make incremental improvements to the turbulent viscosity and k - e model results under the present circumstances.

ANSYS provides comprehensive modeling capabilities for a wide range of incompressible and compressible, laminar and turbulent fluid flow problems. Steady-state or transient analyses can be performed. In ANSYS , a broad range of mathematical models for transport phenomena (like heat transfer and chemical reactions) is combined with the ability to model complex geometries.

Examples of ANSYS applications include laminar non-Newtonian flows in process equipment; conjugate heat transfer in automotive engine components; pulverized coal combustion in utility boilers; external aerodynamics; flow through compressors, pumps, and fans; and multiphase flows in bubble columns and fluidized beds. Turbulence models are a vital component of the ANSYS suite of models. The turbulence models provided have a broad range of applicability, and they include the effects of other physical phenomena, such as buoyancy and compressibility.

3. FUNDAMENTALS

Particular care has been devoted to addressing issues of near-wall accuracy via the use of extended wall functions and zonal models. For all flows, ANSYS solves conservation equations for mass and momentum. The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m$$

Equation is the general form of the mass conservation equation and is valid for incompressible as well as compressible flows. The source Sm is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources. Conservation of momentum in an inertial (non-accelerating) reference frame is described by:

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F}$$

where p is the static pressure, $\bar{\bar{\tau}}$ is the stress tensor (described below), and $\rho \vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively. $\vec{F}$ also contains other model-dependent source terms such as porous-media and user-defined sources.

The stress tensor $\bar{\bar{\tau}}$ is given by

$$\bar{\bar{\tau}} = \mu \left[(\nabla \vec{v} + \nabla \vec{v}^{\,\mathrm{T}}) - \frac{2}{3} \nabla \cdot \vec{v} I \right]$$

where μ is the molecular viscosity, I is the unit tensor, and the second term on the right hand side is the effect of volume dilation.

4. CFD PREDICTION

The geometry was created in the ANSYS-CFX processor in 3D. The geometrical mesh of the domain of interest led to a scale of 1.9 million cells with polyhedral shape. The computations ware performed on Pentium 4 quad-core with 4GB of internal memory.

The grid is a structured one, consisting of tetrahedral elements with a higher density of nodes located in areas where it is expected to have higher velocity and pressure gradients, and where it is assumed that a more density grid would be beneficial for obtaining the accuracy of the solution. These areas are: the boundary layer around the vehicle and the rear area between the vehicle.

Boundary conditions and the initial conditions are giving a particular solution that is obtained by processing equations.

5. CONCLUSION

Comparing the related results on the contour pressure integrals, at three different positions, it could be shown that the most dangerous position would be at the final stage, when the small car is going to end pass the long vehicle.

The turbulences increase, the pressure field rise to more and more twisting effects.

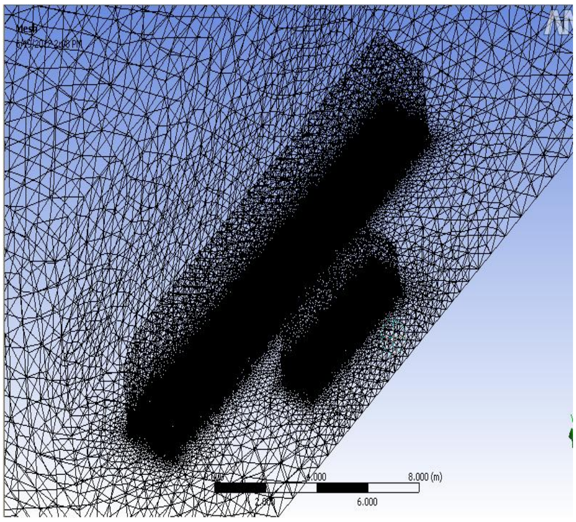

Fig.1 Mesh

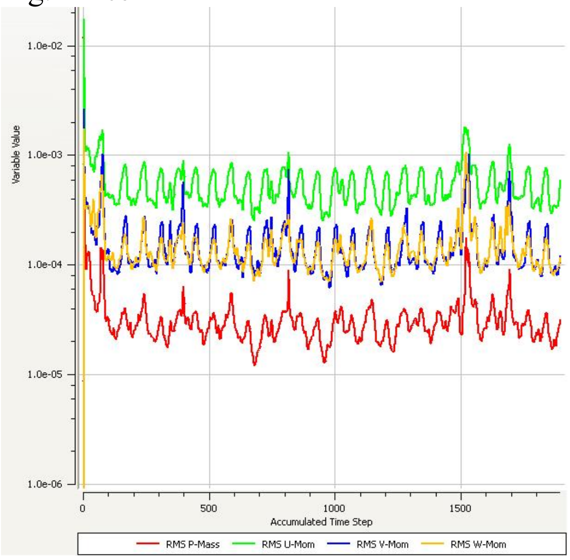

Fig.2 .RMS

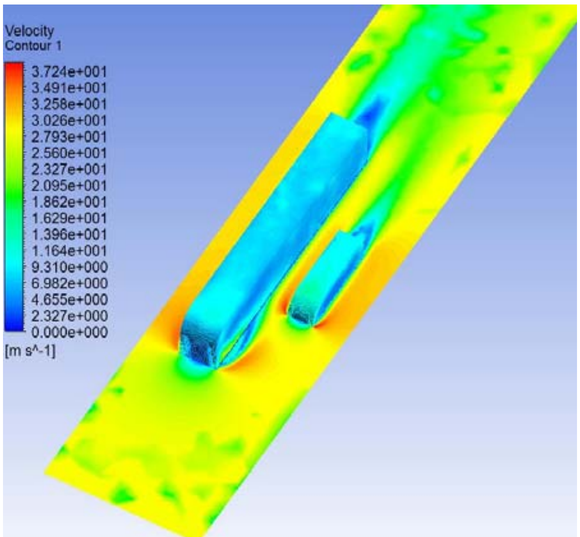

Fig.3 Velocity poz. 1

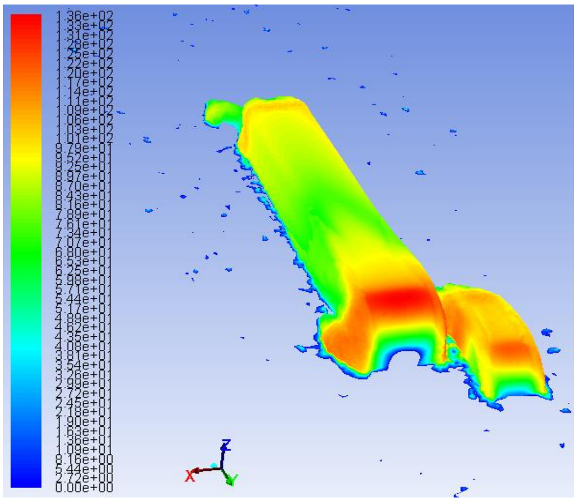
Fig.4 Velocity poz. 2

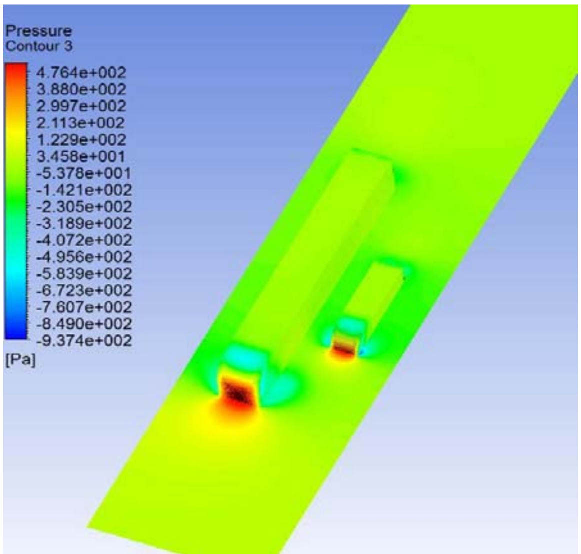

Fig.5 Pressure poz.1

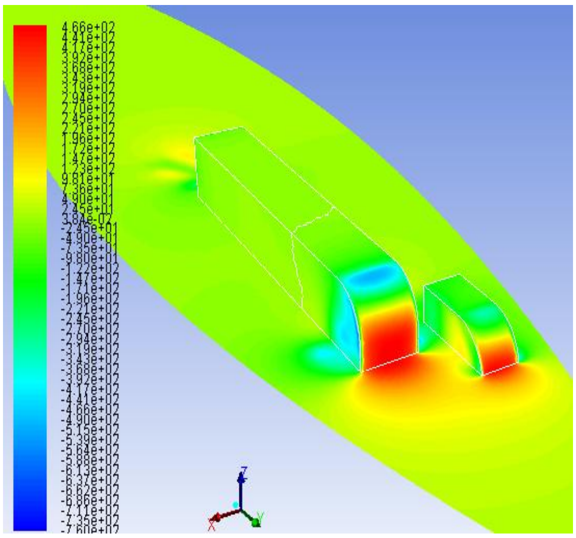
Fig.6 Pressure poz.2

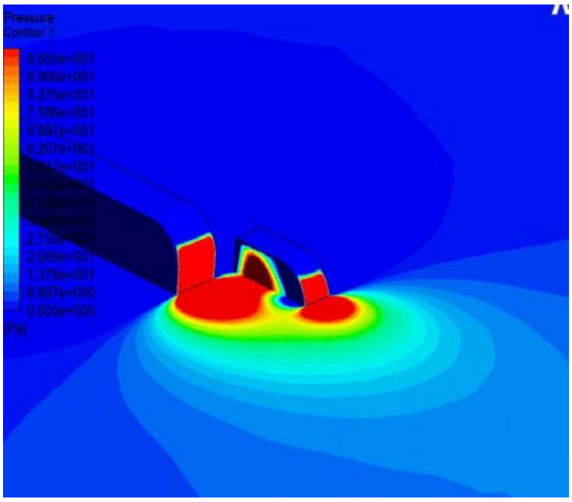

Fig.7 pressure poz.3

6. REFERENCES

[1] Manualul (tutoriale CFD) Fluent, Ansys CFX

[2] Victor Hodor, “Turbo assisted burner, for small boiles (designed by CFD numerical prediction)”, International Conf. “TURBO’98” Vol. I, Edit. “Printech” - 1998

[3] Victor Hodor, Dinamica Gazelor – Ecuatia reunite a combustiei si termogazodinamicii, Ed. Casa Cartii de Stiinta, ISBN-973-9404-54-5, Cluj – 1998

[4] J. Smagorinsky, with the primitive equations, The Basic Experiment, *Monthly Weather Rev.*, **vol.91**, Num.3, pp.99-164 (1963)

[5] T. Nakashima et al., Flow structures above the trunk deck of sedan-type vehicles and their influence on high-speed vehicle stability 2nd report: Numerical investigation on simplified vehicle models using Large-Eddy Simulation, *SAE Int. J.of Passenger Cars – Mechanical Systems*, **vol.2** no.1, pp.157-167 (2009)

INFLUENTE GAZODINAMICE DECURGIND DIN RULAREA A DOUA VEHICULE IN DEPASIRE

Rezumat: Prezenta lucrare se ocupa cu evidentierea predictiei prin simulare numerica prin CFD, a riscului de instabilitate ce decurge la rularea la depasire a unui vehicul de gabarit mare. Studiile surprind efectele aparute prin evidentierea cimpului de presiuni si respectiv a cimpului de energie cinetica turbulent, in cazul in care la o anumita viteza de rulare un vehicul se angajeaza in depasire.

Mircea Diudea, doctorand UTCN, mircea_diudea@yahoo.com, tel. 0740061866
Sergiu Marciuc engineer EMERSON Company
Radu Balan professor, Mecatronic Dep. UTCN, radu.balan@yahoo.com
Victor Hodor professor, Mechanical Engineering Dep. UTCN, victor.hodor@termo.utcluj.ro, tel. 0741181132