TECHNICAL UNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

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

CFD PREDICTIONS REGARDING STABILITIES IN ROOLING OF THE TWO "LOGAN" EDITIONS

Mircea DIUDEA, Sergiu MARCIUC, Radu BALAN, Victor HODOR

Abstract: *Present analyses concerns with the goal of demonstrating our recent skills in manipulating the use of numerical predictions, at the very beginning stage of a new automotive shell designing process. In order to demonstrate without any doubts and as clear as possible our expertise, we choose to evaluate something what has been already evaluated and confirmed by the reality. The Logan 2004 version, versus the Logan 2009 version, could be considered the most appropriated model to reveal the differences in stability given by some shape details –like adding a very small extra "aileron" at the back -shell design. Our CFD numerical investigations one can reveal the difference in terms of dynamic lift extra charge at the back suspension zone, and at the same time offer the possibility to reveal the risks to obtain some kinetic energy turbulences spectrum, in the way of an undesired resonance with the vibrations coming from the back suspension, on some medium running roadway conditions.*

Key words*: CFD numerical prediction, LOGAN face lift, Aerodynamic stability, Kinetic energy spectrum, Suspension*

1. INTRODUCTION

Spoilers gives a dynamically increase of the down force on the vehicle, which helps in keeping the wheels in solid contact with the ground at high. Lift and drag forces have consistent relevancies in running vehicles at different speeds. The overall influences could be revealed by using numerical simulation predictions. The magnitude of lift forces could be predicted by summing the pressure distribution over the specific profiles like spoilers. In the present paper we are concerned only with the effects given by a very small appendices (spoiler) from the Logan2009 edition.

2. OVERVIEW OF THE INTEREST DOMAIN

Spatial discretization 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 up winding 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. The turbulence away from the boundaries was model 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 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 industries, etc. 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. EQUATIONS

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:

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

where p is the static pressure, $\overline{\overline{\tau}}$ 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 $\overline{\overline{\tau}}$ is given by

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

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.
Particular care has been devoted to addressing issues of near-wall accuracy via the use of extended wall functions and zonal models.

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 1.3 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 polyhedral 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 dense grid would be beneficial for obtaining the accuracy of the solution.
Using a height of 6m and adding an opening parameter we have reduced the above wall influence of the virtual aerodynamic tunnel. Boundary conditions and sometimes the initial conditions are giving a particular solution that is obtained by processing equations. Analyzing several inlet conditions, we have selected three different velocities: 70, 100 and 130 km/hour.
In the present paper we present only the 120 km/h rolling conditions results, as it looks to be the most critical conditions –for the stability.

Fig.1 Dacia Logan 2004

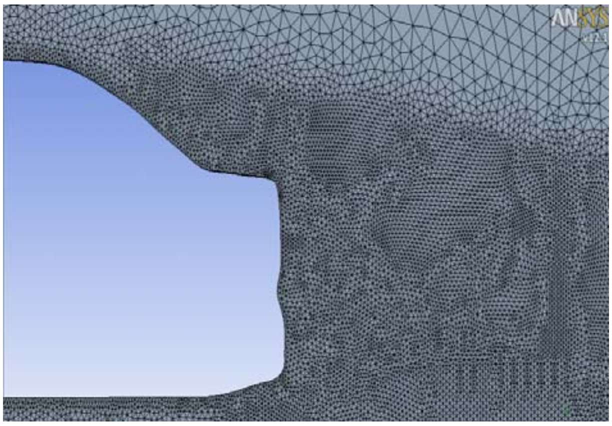

Fig.2 Mesh Logan 2004

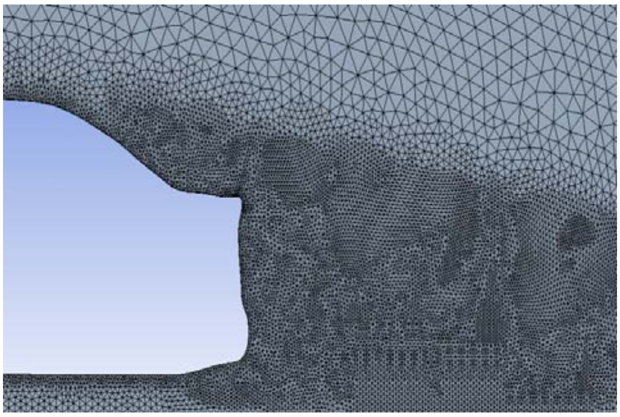

Fig.6 Mesh Logan 2009

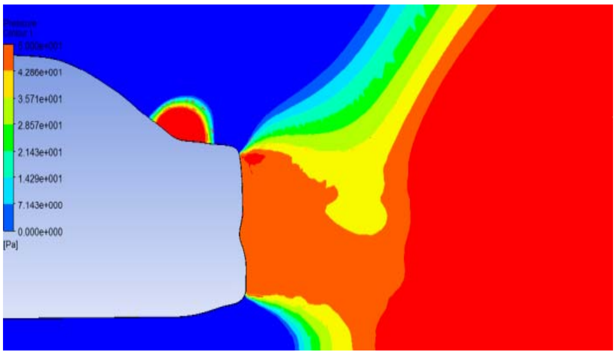

Fig.3 Pressure Logan 2004

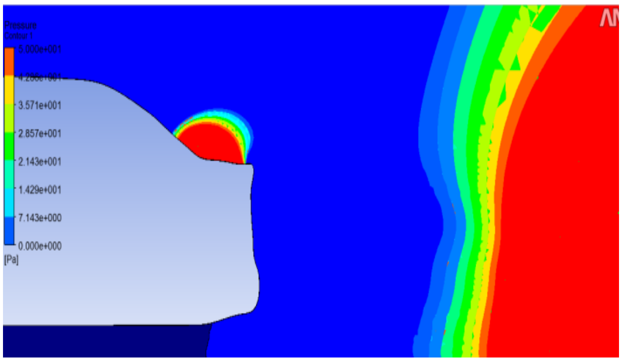

Fig.7 Pressure Logan 2005

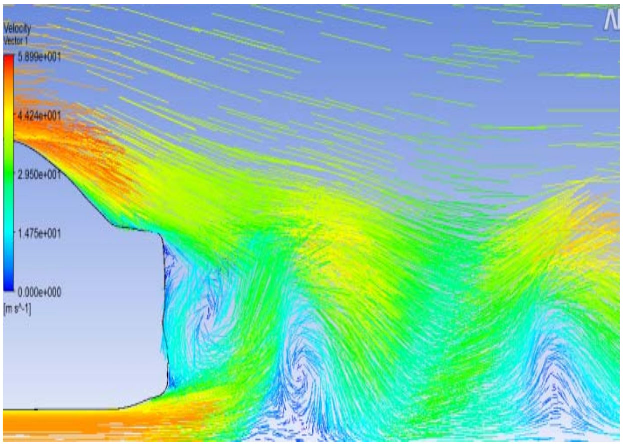

Fig.4 Velocity Logan 2004

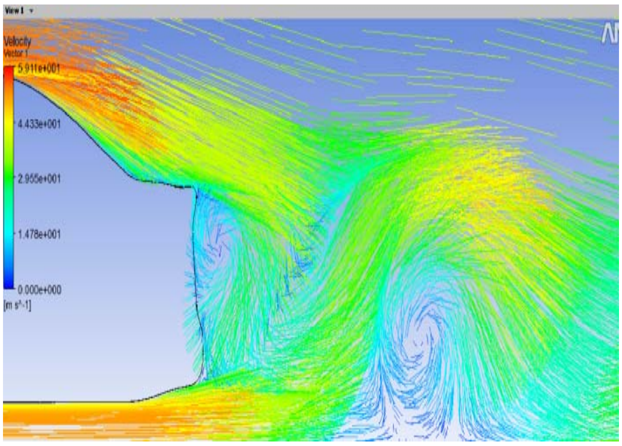

Fig.8 Velocity Logan 2009

Fig.5 .Dacia Logan 2009

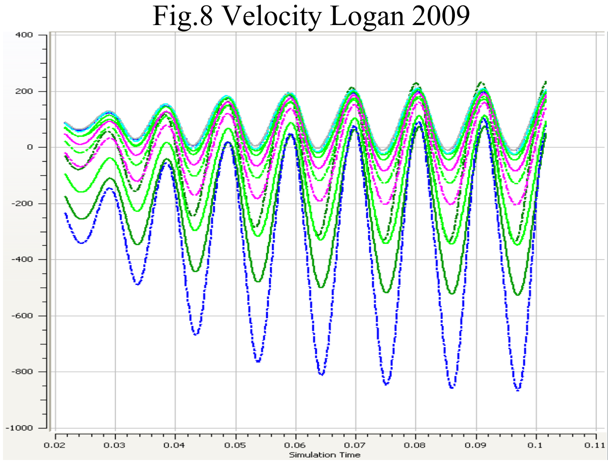

Fig.9 Pressure spectrum of variation Logan2004 at the rolling conditions of 120km/h

5. CONCLUSION

The comparative analysis of the two design, for the three speeds of running conditions, and lifting forces computation revealed that the maximum extra-charge is obtained at 120 km/h. The Kinetic energy with high magnitude that appears in the spoiler area, gives real problems concerning reliability and because of the powerful vibrations that are induced, and these are profitable diminished for the Logan2009 edition.

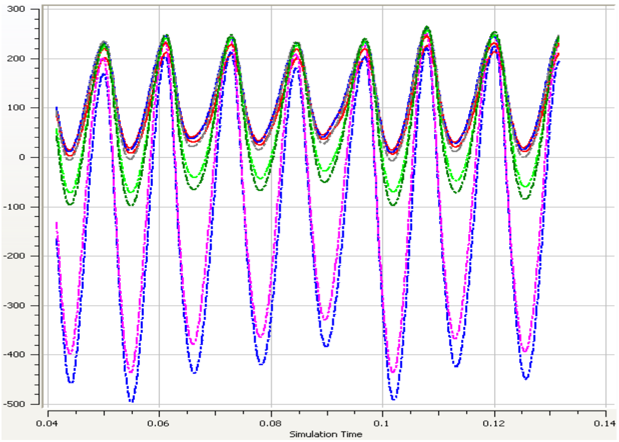

Fig.10 Pressure spectrum of variation Logan2009 at the rolling conditions of 120km/h

At 100 km/h a noticeable depression behind spoiler will increase the lifting effect, but the turbulence induced by the kinetic energy magnitude begins to be relevant. At 120 km/h higher lifting effects are emphasized by the high pressure developed on the upper part of the aerodynamic profile. Higher speed involve high magnitude kinetic energy and areas off stress upon the eleron and also high magnitude turbulences that affects the car performance. These effect are vell diminished for the Logan2009 edition.

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, Nr 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)

PREDICTII PRIN SIMULARE NUMERICA A STABILITATII IN RULAREA CELOR DOUA VERSIUNI DE LOGAN

Rezumat: Analizele efectuate in prezenta lucrare se refera la sistematizarea avantajelor pe care simularea numerica cu element finit, le poate aduce in proiectarea caroseriilor unor vehicule –din punctul de vedere al efectelor termogazodinamice aparute in timpul rularii acestora. Studiile intreprinse in aceasta lucrare, arata avantajele in stabilitate, aduse de versiunea "LOGAN 2009" visavis de versiunea "LOGAN 2004", in rularea la viteze de peste 100km/ora. Se remarca plusul de stabilitate (la vibratii) adus de noua variantă –prin intermediul suplimentarii cu elemental de eleron adaugat suplimentar in forma capotei porbagaj/spate.

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