U.P.B. Sci. Bull., Series D, Vol. 76, Iss. 4, 2014 ISSN 1454-2358

AUTOMOTIVE WING WITH ACTIVE CONTROL OF FLOW

Angel HUMINIC[1], Gabriela HUMINIC[1]

In this paper, is studied the aerodynamic behavior of an automotive wing having an active control of the flow on the lower face, using the Coanda effect, which represents a new approach in the field of the auxiliary devices of cars, used to generate down force.

The airfoil of the aileron studied is Eppler E423. It was designed for high lift and based on experiments operates well down to Re of 200k. For the analyses of wing with Coanda effect, a curved slot of 0.7 mm width was considered on the lower side of the wing, placed at 0.6 of airfoil chord. The influence of initial velocity of the Coandă jet was studied.

Keywords: automotive wing, aerodynamics, Coanda effect

1. Introduction

Known also as boundary layer attachment, Coandă effect represents the tendency of a stream of fluid to stay attached to a convex surface, rather than follow a straight line in its original direction. The principle was named after Romanian discoverer Henri Coandă, who observed it for the first time during experiments with his Coandă-1910 aircraft, the first one which used a motorjet, an early type of jet engine. Coandă understood the practical importance of this phenomenon for aircraft development and in 1934 he obtained a patent in France for a „Method and apparatus for deviation of a fluid into another fluid". Other significant patents are „Lenticular Aerodyne", „Device used for improving of the internal combustion engine efficiency", „Airbrake with recoil for fire guns", a.o. During its entire scientific work, Henry Coandă obtained 215 patents for devices assisted by Coandă effect.

Concerning road vehicle aerodynamics, one of the applications of the Coandă effect is dealing with improvement of the aerodynamics performances of medium and large vehicles, mainly with hatchback [1, 2]. In this way, in the last decade, starting from a previously developed and flight-tested pneumatic (blown) aircraft technology, the researchers of Georgia Tech Research Institute (GTRI) USA developed and improved an aerodynamic device used to control the flow behind heavy trucks and SUVs. For vehicles equipped with such devices, their

[1] Aerodynamics Laboratory, Transilvania University of Brasov, Romania, e-mail: angel.h@unitbv.ro

studies show significant drag reductions, download increases and also, the ability to control aerodynamic moments of vehicles about all 3 axes without any moving control surfaces, as adjustable spoilers, which recently became very complex systems for air guiding around vehicles [3]. Fig. 1 shows such of device with variable surface and variable arrangement on the vehicle body concerning vertical position and angle of attack, according with speed of vehicle. Its motion is performed by complex mechanisms.

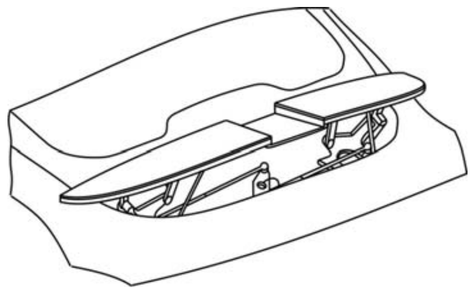

Fig. 1. Adjustable automotive aileron (after Ref. [3])

New results concerning the aerodynamic characteristics of an aileron assisted by Coandă effect are presented in this paper, benefiting by experience of previous work [4, 5] in the field of automotive wings. This type of active control of flows represents a new approach in the field of the auxiliary devices of the cars, used to generate download. In this sense, such automotive aileron takes advantages of both types of usually used fixed ailerons of cars, without mechanical parts in motion, and respectively, the adjustable ailerons, mechanically controlled, used to generate variable download.
By using of Coandă ejection to control the flow on lower surface of spoiler, the negative effects of trailing edge separation can be avoided [4], as shown in Figure 2.

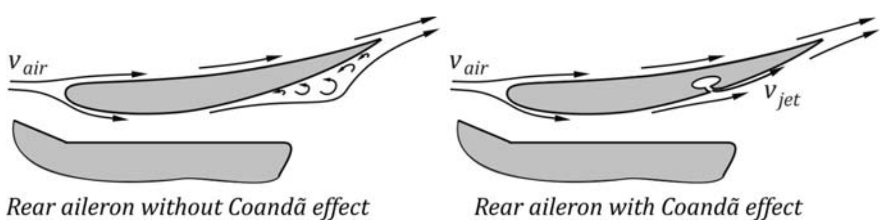

Figure 2: Principle of automotive aileron with Coandă effect

In this paper are presented the results of 3D simulations of flow around a wing, without and with Coandă effect. The airfoil of the aileron studied is E423, with a maximum thickness 12.5% at 23.7% chord. It was designed by R. Eppler for high lift [6] and based on experiments [7] operates well down to Re of 200k, when $c_{L\,max}$ is nearly 2.0. For the analyses of wing with Coanda effect, a curved slot of 0.7 mm width was considered on the lower side of the wing, placed at 0.6 of airfoil chord. The influence of the initial velocity of the Coandă jet was studied with the aid of the ANSYS CFX, finite volume CFD code.

2. CFD Methodology

The flow field around a vehicle is physically very complex. In consequence, the efficiency of an aerodynamic CFD simulation depends on many factors. Creation of the model geometry and its integration in a physical domain, grid generation and choice of a suitable numerical computing scheme are significant factors that can influence the success level of the simulation process. The main steps of the simulation processes of this study are described in the following paragraphs.

2.1 CAD Model

For current study, a 3D wing was considered, having the following characteristics, as usually for an automotive aileron (see Figure 3):

- chord of airfoil $c = 160\ mm$;
- span $s = 9.725 \cdot c$;
- angle of attack $\alpha = -6°$, for the highest value of lift to drag ratio;

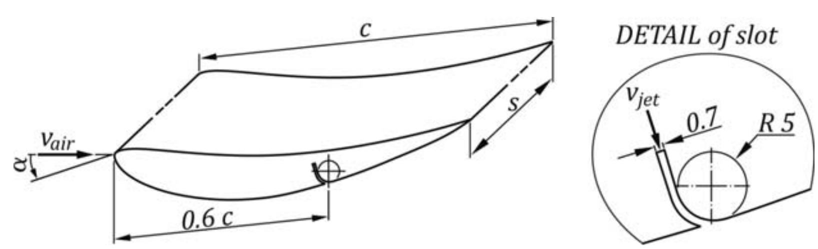

Fig. 3. CAD model - dimensions are in mm

For the analyses with Coandă effect, a curved slot of 0.7 mm width was considered on the lower side of the wing at $0.6 \cdot c$. The surfaces of the bodies were drawn as parameterized CAD data and integrated into computational domain, rectangular enclosure, with symmetry plane (zOx), as shown in Figure 4. According to previously established procedures the chose dimensions ensure that the adverse pressure effects between bodies and walls are negligible, the assigned blockage being 1.125%. In order to avoid the possible blockage effect, the cross-section size was set to be relatively larger than usually as for a wind tunnel facility.

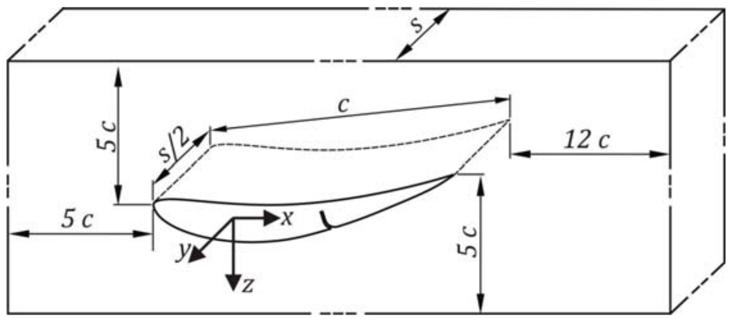

Fig. 4. Computational domain

2.2 Grid Resolution and Boundary Conditions

The grids were generated using a multi-block scheme, with hexahedral cells. In order to accurately solve the flow nearest to the surfaces of the wing, a resolution of the boundary layer of 30 points was assigned and the maximum distance from solid surfaces of the vehicle to the first layer of grid points was set to fulfill the condition of $y^+ \leq 100$, for computations where a wall function is used. For half of the model, the dimensions of the computational grids were more than:

- 4,600,000 grid points for entire computational domain, of which
- 50,000 grid points for surfaces of wing.

Boundary conditions for the computation were applied based on wind tunnel tests described by Selig et al. [7], used for validation of CFD procedures:

- an uniform and constant velocity $v_x = v_\infty$ (velocity of the free stream) and $v_y = v_z = 0$ were imposed at the inlet of domain;
- an uniform and constant velocity of Coandă jet, v_{jet}, perpendicular on inlet of slot;
- at the outlet boundary, a zero pressure condition was imposed: $p = 0$;
- no slip conditions on surfaces of the wing: $v_x = v_y = v_z = 0$;
- for the rest of the surfaces, free slip conditions were assigned.

2.3 Conditions of Simulations and Turbulence Model

For the computation of flow variables, the used code solves the full RANS (Reynolds-Averaged Navier-Stokes): mass (continuity) equation, momentum equation and energy equation, if necessary. The analyses were performed in steady state, adiabatic, turbulent conditions (0.1 %), for a reference pressure and temperature of the air $p_{\infty} = 1\,At$, $T_{\infty} = 288.15\,\text{K}$. These were used for computation of the rest of the air parameters, as density and viscosity. The velocity of the free stream, $v_{\infty} = 27.4\,m/s$, was imposed to meet a Reynolds number of $\text{Re} = 300k$. Initial velocity of Coandă jet was the variable parameter.

The Shear-Stress-Transport (SST) closure model of Menter, 1994, [8], was used to solve the simulation processes. This is a two-equation eddy-viscosity model, based on $k-\omega$ model of Wilkox, 1986, [9]. The SST model was developed to effectively blend 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 (Launder and Spalding, 1974, [10]) in the far field. This model uses a blending function to switch from $k-\omega$ to $k-\varepsilon$ in the wake region to prevent the model from being sensitive to free stream conditions. The definition of the turbulent viscosity is modified also to account for the transport of the turbulent shear stress. These features result in a major improvement in terms of flow separation predictions, and the performances of this model have been demonstrated in a large number of validation studies.

2.4 Validation of Computational Model and CFD Procedure

In order to evaluate the accuracy of the established CFD procedure and also to examine the independence of solutions with respect to the computational domain, six 2D analyses were performed, for which experimental data were available in very well known conditions. The parameter was the angle of attack with the following values: $\alpha = 0°, 2°, 4°, 6°, 8°, 10°$. The main used convergence criteria are as follows:

- decreasing of the residuals at least under 10^{-5} for all variables;
- a continuous, realistic distribution of the variables on the entire computational domain;
- variations of the computed aerodynamic coefficients smaller than 10^{-4} for the final iterations, in order to discriminate changes in drag as small as $\Delta c_D = \pm 0.001$ [11];
- a value of y^{+} of the first grid point above the surface of the wing, between 10 and 100, since a wall function was used.

The results of these CFD analyses are presented in Figure 5, together with the experimental results, provided by Selig et al. [7].

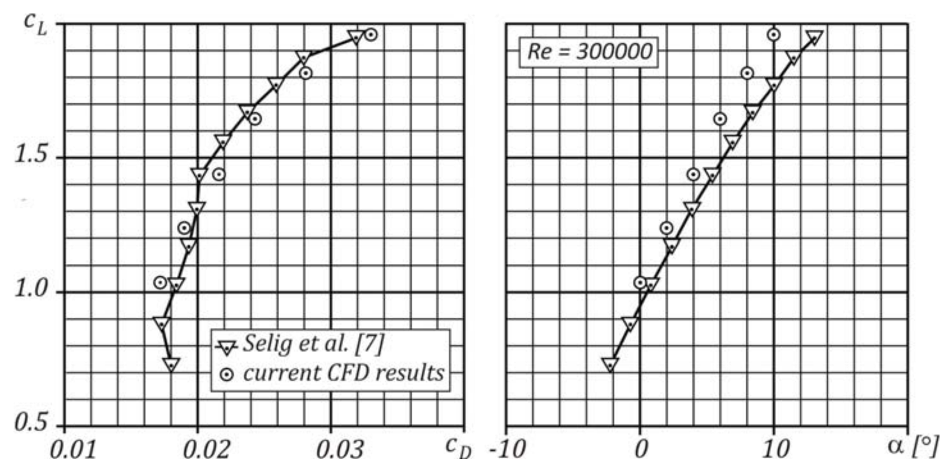

Fig. 5. Current CFD results versus experimental results of Selig et al. [7] for E423 airfoil

As one can observe, the accuracy of the adopted CFD procedure with respect to aerodynamic coefficients prediction is very good for this case, the deviations (lower than 15%) being the result of differences between theoretical airfoil and real tested one, mainly in the trailing edge area [7].

3. Results and Discussions

The results concerning the aerodynamic coefficients, C_L and C_D, of the wing assisted by Coandă effect are shown in table below.

Table 1

Aerodynamic coefficients of wing with Coandă effect

v_{jet} [m/s]	0	25	30	35	40	45	50
C_L [-]	1.366	1.245	1.306	1.367	1.422	1.471	1.511
C_D [-]	0.0755	0.0692	0.0727	0.0770	0.0813	0.0856	0.0896

For the studied cases, results confirm that, generally speaking, there is an improvement of the aerodynamic characteristics of the aileron, when the flow is controlled on the lower side, using Coandă effect. This improvement is emphasised by the c_P variation on the airfoil chord along the span (Figures 6 and

7) and by generated download: increasing of C_L with $\Delta C_L = 0.105$ for $v_{jet} = 45\, m/s$, accompanied by a small penalty on drag coefficient, $\Delta C_D = 0.01$.

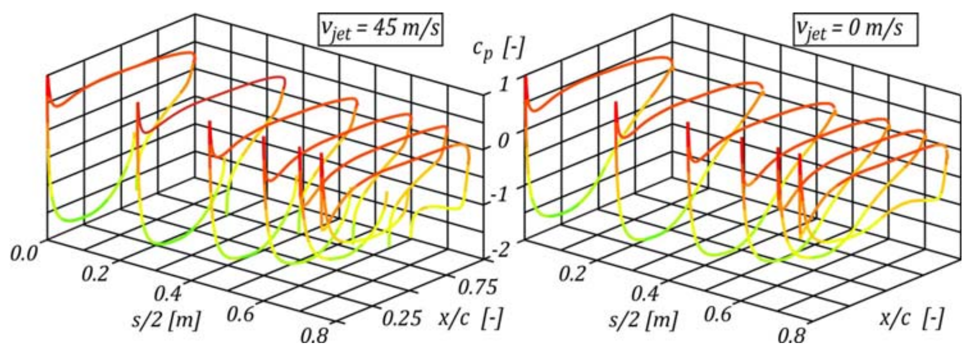

Fig. 6. Cp variations on wing

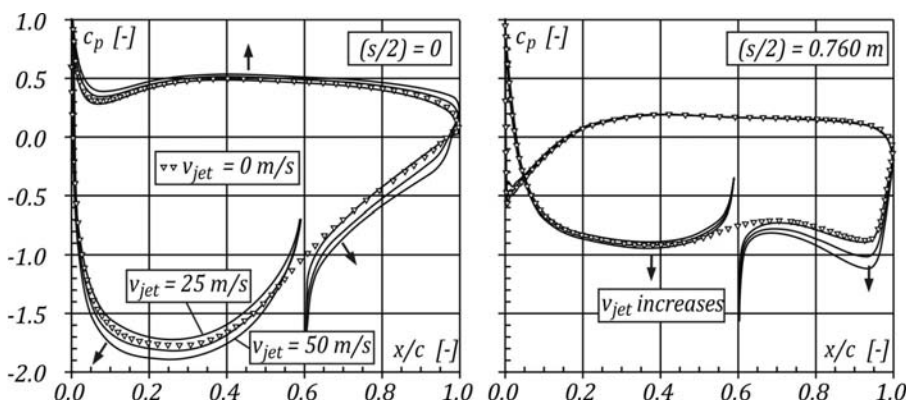

Fig. 7. Cp variations on wing, v_{jet} = 0-50 [m/s

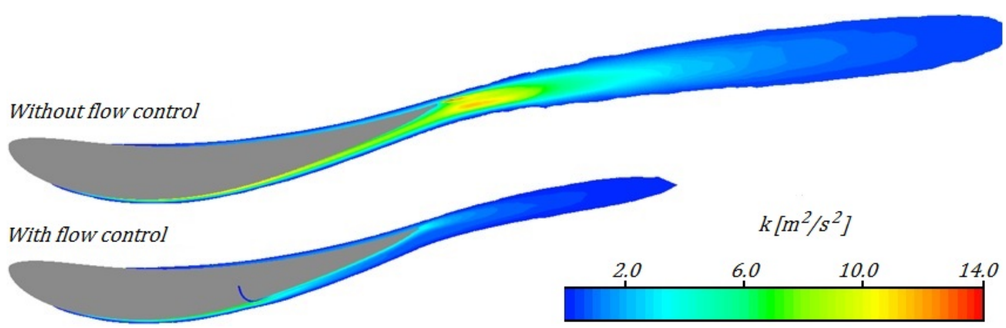

Fig. 6. Turbulence kinetic energy contours

From a qualitative point of view, by controlling of flow on the airfoil surface, using Coandă ejection, the negative effect of trailing edge separation can be avoided, as shown in Figures 8.

4. Summary and Conclusions

In this paper, the authors investigate numerically the 3D flow around an automotive wing, which generates download, taking into account also the effect of an active control of flow on the lower side, using Coandă ejection. Thus, a curved slot of 0.7 mm width was considered on the lower side of the aileron placed at 0.6 of airfoil chord. The influence of the initial velocity of the Coandă jet was studied.

The results show that Coandă effect can be used to reduce trailing edge separation, in order to improve the aerodynamic characteristics of the ailerons, and later to increase the aerodynamic behavior of the vehicle concerning the aerodynamic loads, drag and lift, and, consequently, stability and handling.

Also, such ailerons combine the advantages of fixed spoilers, without mechanical part in motion, and adjustable ones.

R E F E R E N C E S

[1]. *R. J. Englar,* “Advanced Aerodynamic Devices to Improve the Performance, Economics, Handling and Safety of Heavy Vehicles”, SAE Technical Paper 2001-01-2072, 2001, doi:10.4271/2001-01-2072.

[2]. *R. J. Englar,* “Improved Pneumatic Aerodynamics for Drag Reduction, Fuel Economy, Safety and Stability Increase for Heavy Vehicles”, SAE Technical Paper 2005-01-3627, 2005, doi:10.4271/2005-01-3627.

[3]. *M. Froeschle, M. Schulzki*, “Air Guiding System for a Vehicle”, Patent US2007236046, 2007.

[4]. *A. Huminic, G. Huminic,* "Study of the Automotive Wings with Coanda Effect", FISITA 2010 World Automotive Congress, ISBN 978-1-61782-507-1, vol. 5. pp. 4134-4143.

[5]. *A. Huminic, G. Huminic,* "Automotive Wing with Coanda Effect", International Congress of Romanian Society Automotive Engineers, 2010, ISSN 2069-0401, vol. 2, pp.171-178.

[6]. *R. Eppler*, "Airfoil Design and Data", ISBN 3-540-52505-X, Springer-Verlag, Berlin, 1990.

[7]. *M. Selig, C. Lyon, P. Giguere, C. Ninham, J. Guglielmo,* "Summary of Low-Speed Airfoil Data", vol. 2, ISBN 0-9646747-2-6, SoarTech Publications, Virginia USA, 1996.

[8]. *F. Menter*, "Two-equation eddy-viscosity turbulence models for engineering applications", AIAA Journal, vol. 32(8), 1994, pp.1598–1605.

[9]. *D.C. Wilcox*, "Multiscale model for turbulent flows", AIAA 24th Aerospace Sciences Meeting, American Institute of Aeronautics and Astronautics, Reno, Nevada, USA, 1986, paper 86-0029

[10]. *B. Launder, D. Spalding,* "The numerical computation of turbulent flows", Computer Methods in Applied Mechanics and Engineering, vol. 3(2), 1974, pp. 269–289.

[11]. ***, "Aerodynamic Testing of Road Vehicle - Testing Methods and Procedures", SAE Information Report SAE J2084 JAN93, 1993.