Study of the Instability due the Different Position of an Aileron

Oana DUMITRESCU[1], Bogdan GHERMAN[1], Valeriu DRĂGAN[2]
CFD department
[1]Romanian Research and Development for Gas Turbine COMOTI
[2]KTH Royal Institute of Technology, Sweden
220D Iuliu Maniu Ave. 061126, Bucharest
Romania
oana.dumitrescu@comoti.ro, bogdan.gherman@comoti.ro, valeriu@kth.se, http://www.comoti.ro/en/

Abstract: - The present paper describes the numerical analysis of the fluid flow around a finite span wing, NACA 0012 profile, provided with an aileron, placed at three static positions, corresponding to three different angles. The numerical simulation has been made with the commercial software ANSYS Fluent. The study is conducted in the case of the unsteady regime, the turbulence model used is SST k-ω, and the filtering of the governing equations is carried out under the DES (Detached Eddy Simulation) approach. The results show that the aileron position and velocity influences the flow and are leading to an aerodynamic instability at the trailing edge of the aileron, in some cases also influencing the wing end.

Key-Words: - aerodynamic instability, aileron, CFD, aileron deflection, wing, Fluent

1 Introduction

Under certain conditions, a wing or a wing – aileron structure may become unstable, because of the interaction of the elastic, inertial and aerodynamic forces acting on the structure. Air forces can damp vibrations of a single degree of freedom. These forces act on the wing or wing-aileron in an arbitrary oscillatory motion, due to the air pressure [4].

The flutter, as one of the aerodynamic instabilities that can appear on a wing, is a kind of self-excited vibration involving the interaction of inertial, elastic and aerodynamic forces [5]. It is an unstable oscillation that can lead to the destruction of the structure. It may appear on fixed surfaces, such as the wing or the elevator, and also on the control surfaces: aileron, stabilizer and rudder.

The pressure distribution around an airplane wing is influenced by the control surfaces that help to stabilize or destabilize an aircraft according to their position and geometry. The design of the control system requires complete knowledge of the aerodynamic forces acting on the wing. The FSI (Fluid / Structure Interaction) method is used in a model to determine the forces and the response of the structures [6][2].

In the paper [3] was developed a mathematical modeling of aileron actuation that uses piezo V-shaped stacks. The purpose of the research was to determine a solution that can counteract flutter and the solution will be tested on an experimental test rig.

In this paper the numerical analysis of an aileron and the influence of its position downstream of the fluid flow field are presented. The position of down aileron deflection can influence not just the downstream field, but also the field near the wing end. The most important goal of this study of flutter phenomenon is to determine if the position of the aileron has a major impact in the appearance of this instability.

2 Problem Setup

The study presented in this paper was carried out using the commercial software ANSYS Fluent. The geometry of the domain is presented in Fig. 2, and consists in a wing with aileron and an exterior domain. The dimension of the wing and domain are presented in Table 1. The extent of the exterior domain is limited by the available computational resources and running time.

Table 1: Geometrical parameters of the domain

	Parameters		Value
Wing	Length	[mm]	1200
	Aileron Length	[mm]	490
	Aileron Width	[mm]	110
Exterior domain	Length	[mm]	2500
	Width	[mm]	800
	Height	[mm]	300

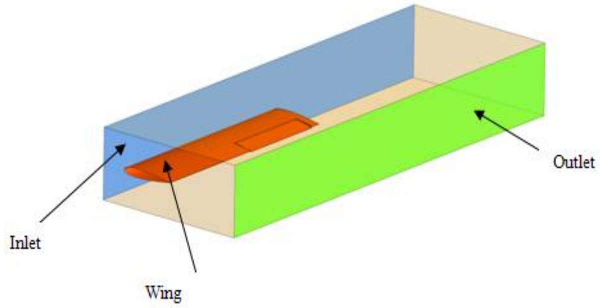
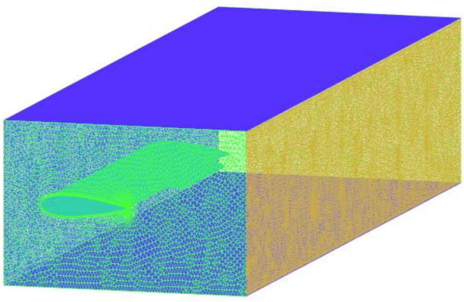
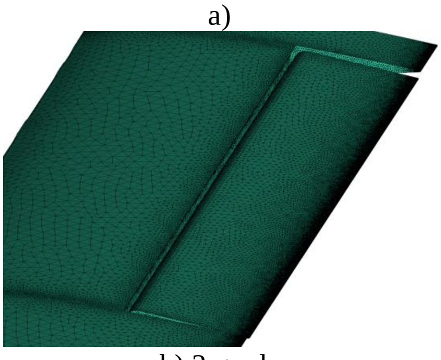


Fig. 2 Boundary conditions

The computational domain was discretised using quadrilateral elements, because of the complex geometry of the wing-aileron structure, Fig.3. At the trailing edge of the aileron, the grid is much denser compared to the rest of the wing in order to properly capture the key flow dynamics in the region.



a)



b) 2 grade

Fig.3 Grid for the computational domain: a) entire domain, b) aileron at 2°

The turbulence model used for this simulation was SST k-ω using a DES methodology. The approach uses the RANS formulation in the regions near solid boundaries, and switches to the LES filtering approach if the turbulent length scale exceeds the grid dimension [7].

3 Results

The following five regions are defined to be used throughout the processing of the results:

- 750 mm – The middle of the wing
- 490 mm – Aileron beginning
- 250 mm – The middle of the aileron
- -0.05 mm – Aileron end
- -54 mm – Wing end

The wing studied in this case will be provided with a piezoelectric actuator and tested in a wind tunnel; the purpose will be to obtain an anti-flutter demonstrator, validated by experiment. To realize those testes it is important to determine the aileron deflection, δ – Fig.4. The angles used in this case are a preliminary test, to determine the influence of different deflection angles.

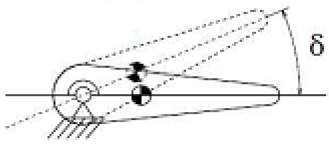
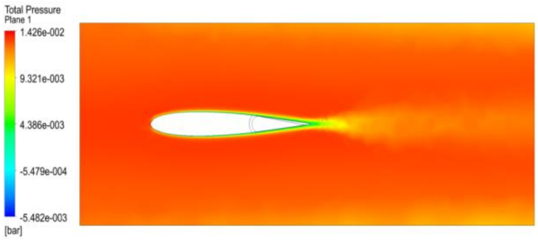


Fig.4 A simple physical model of aileron actuation [3]

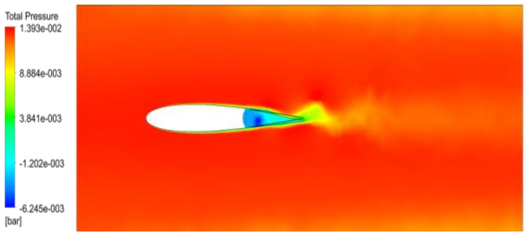
The velocity used for these cases is 45m/s, this value is used due to the fact that experimental measurements carried out by a different research team within same project [8] have shown that 41m/s is the velocity at which the flutter phenomenon appeared.

a) Aileron at 2 degrees - speed 45 m / s

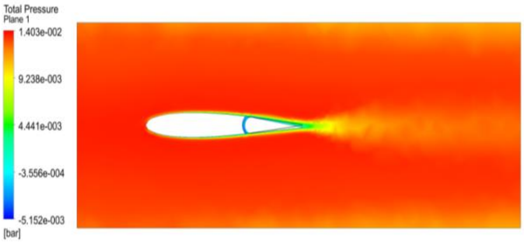
Fig. 5 shows the pressure distribution over the five previously defined regions of the domain. The most significant case is shown in Fig 5.d, where a slight detachment of the flow from the pressure side of the aileron can be seen. In Fig. 5.b, at the beginning of the aileron, instability of the fluid flow appears because of the gap between the two parts.



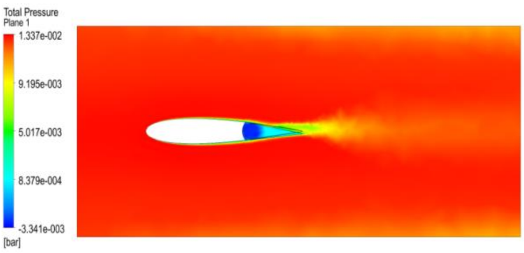
a)750 mm



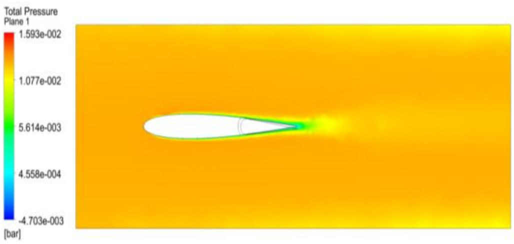
b)490 mm



c)250 mm



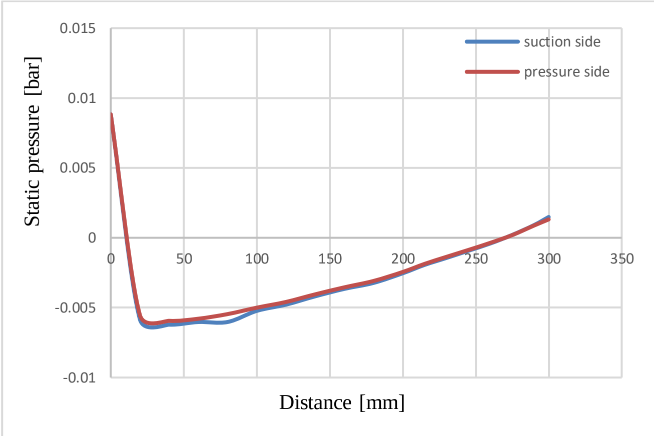
d) -0.05mm



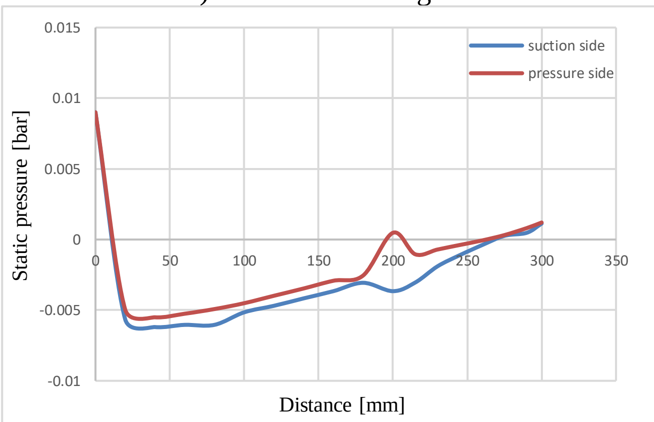
e) -54 mm

Fig.5 Pressure distribution in the 5 regions

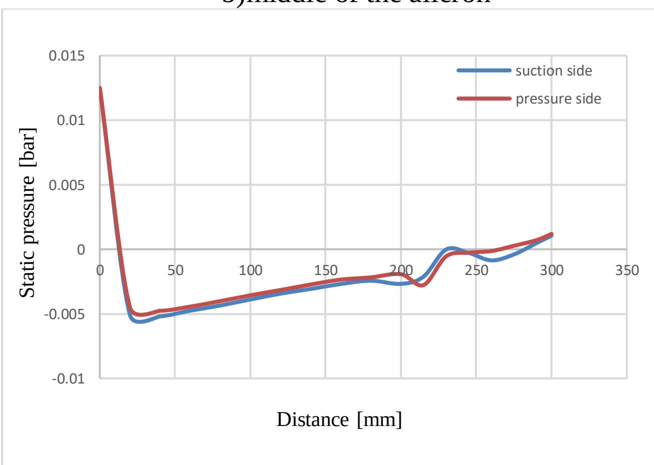
Fig. 6 shows the pressure distribution only on the suction side and pressure side of the wing and aileron. It can be seen that at the middle of the wing and aileron, the value of the static pressure on the suction side is smaller than on the pressure side, the pressure gradient between the two surfaces generating the lift force.



a)middle of the wing



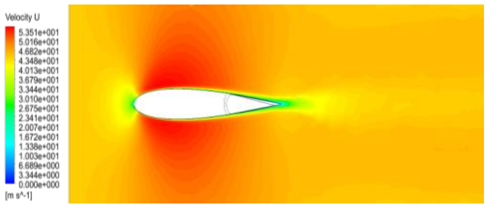
b)middle of the aileron



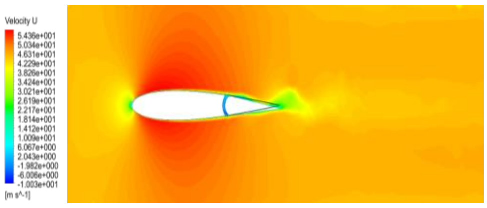
c)end of the aileron

Fig.6 Pressure Distribution relative to the distance from the leading edge

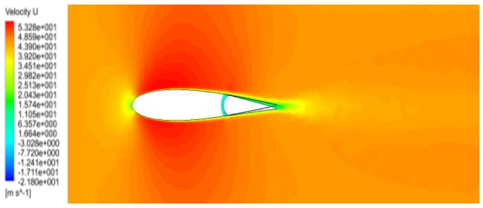
The evolution of velocity in this case is shown in Fig. 7. The velocity distribution on the both surfaces of the profile is the same, due to the symmetry of the profile. In the trailing edge area, the velocity is higher on the suction side, and the detachment of the flow may be observed. A detachment can also be seen at the beginning of the aileron.



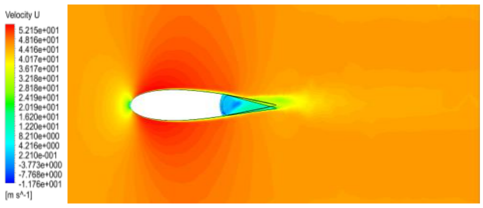
a) 750 mm



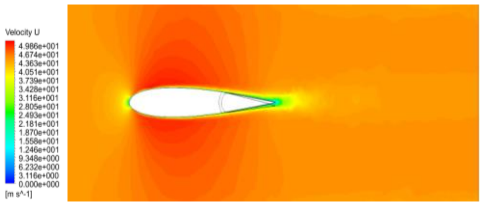
b) 490mm



c)250mm



d)-0.05mm

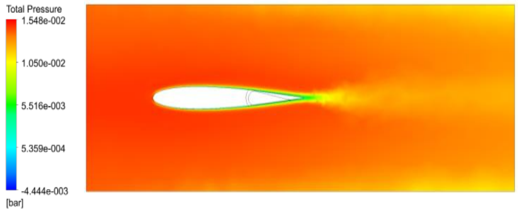


e) -54 mm

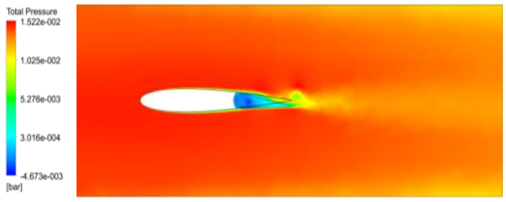
Fig. 7 Velocity variation in the x direction around the profile in the 5 regions

b) Aileron at 5 degree – 45 m/s

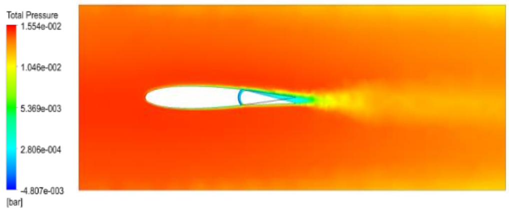
In this case, the position of the aileron was changed to 5 degrees. Upstream of the aileron, the pressure distribution is symmetrical, as seen in Fig.8.a. Further downstream, differences starts to appear, as the position of the aileron influences the flow field. Figs 8.b, 8.c, and 8.d show the evolution of the flow downstream of the aileron, presenting flow detachment on the suction side because of the pressure differences over the two surfaces.



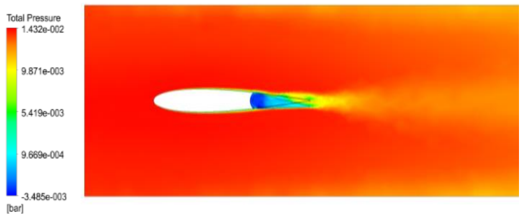
a)750 mm



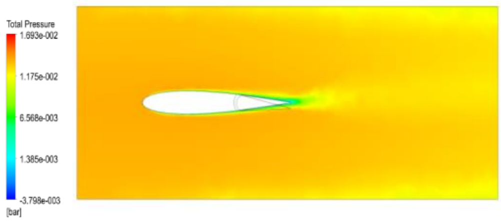
b)490mm



c)250 mm



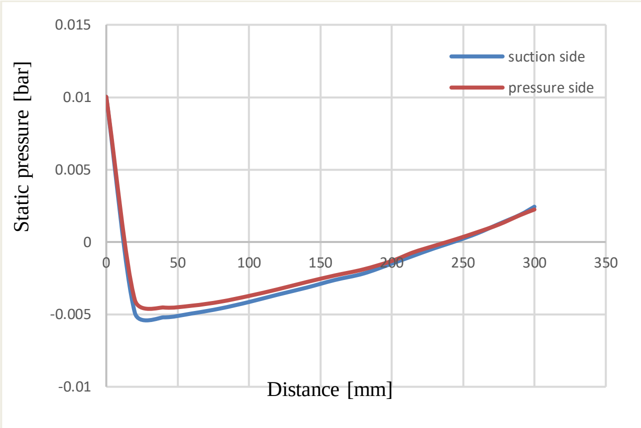
d)-0.05mm



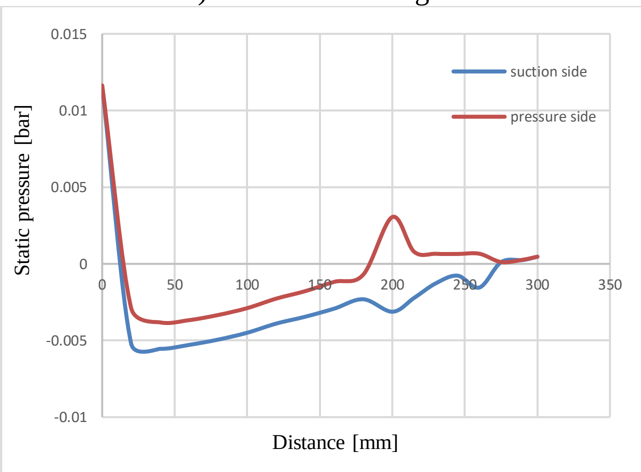
e)-54mm

Fig. 8 Distribution of total pressure around the profile in the five regions

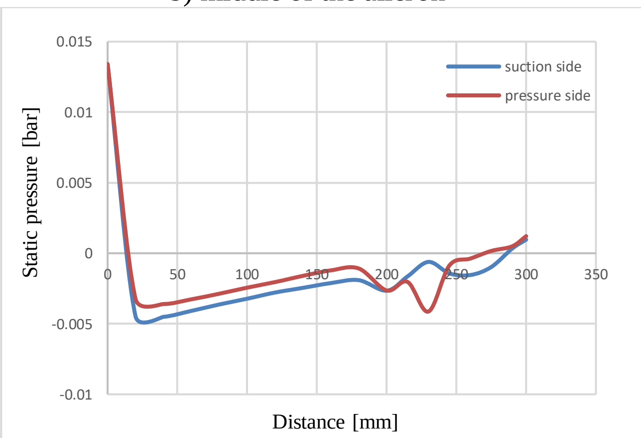
The differences in velocity and the angle position of the aileron can be seen well in the line plots shown in Fig.9. The instability at the trailing edge of the wing can be very clearly seen as the pressure difference between the two surfaces is increasing with the aileron angle.



a)middle of the wing



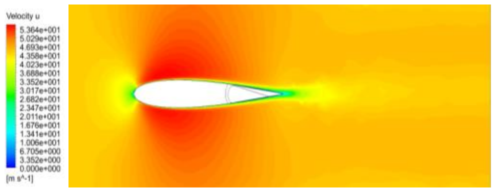
b) middle of the aileron



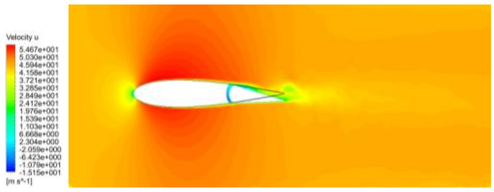
c) end of the aileron

Fig. 9 Pressure Distribution relative to the distance from the leading edge

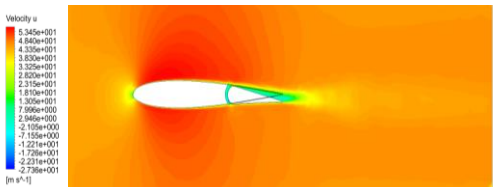
The velocity distribution over the wing is symmetrical on the wing, but instability appears closely downstream of the trailing edge.



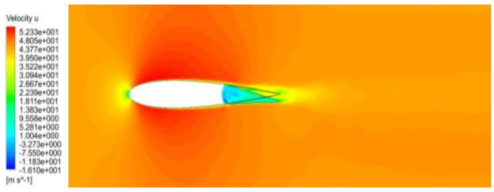
a)750 mm



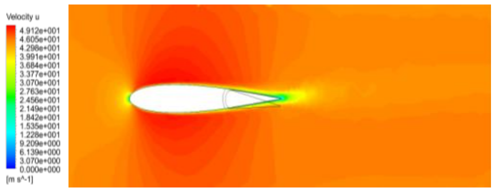
b)490mm



c)250 mm



d)-0.05 mm

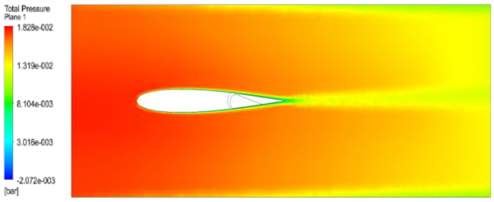


e)-54mm

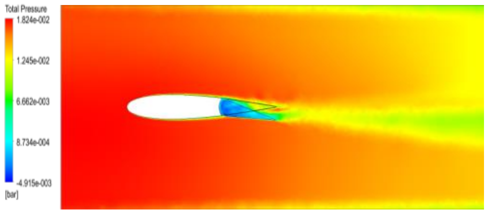
Fig. 10 Velocity variation in the x direction around the profile in the 5 regions

c) Aileron at 10 degree – 45 m/s

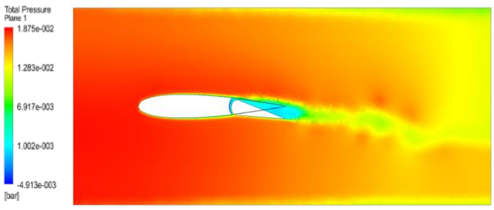
This latter case, illustrates the wing positioned at a zero-degree angle of attack and the aileron turned at 10 degrees. The flow on the wing progresses symmetrically until the aileron area, Fig. 11.a. An aerodynamic instability that spreads downstream of the field occurs further. Fig. 11.b captures the appearance of a vortex, which extends to the end of the wing. In the previous cases, towards the end of the wing the flow became symmetrical again, but this does not happen anymore in this case, as the influence of the aileron position expands.



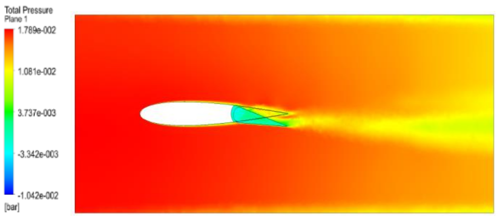
a)750mm



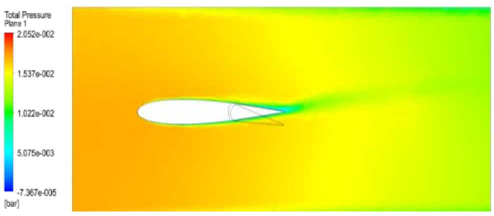
b)490 mm



c)250 mm



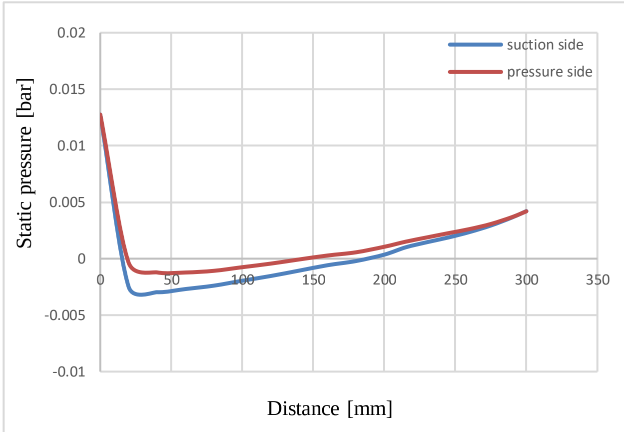
d)-0.05 mm



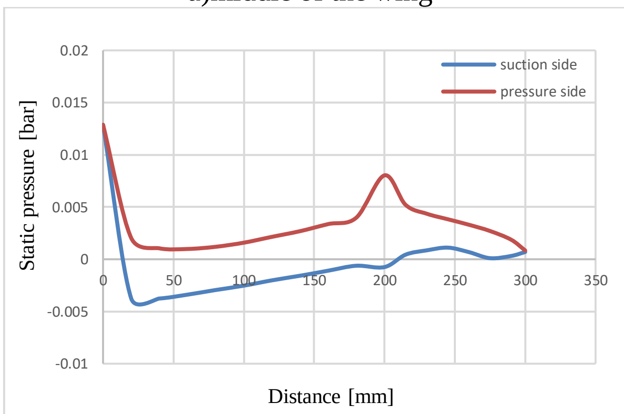
e)-54 mm

Fig. 11 Distribution of total pressure around the profile in the five regions

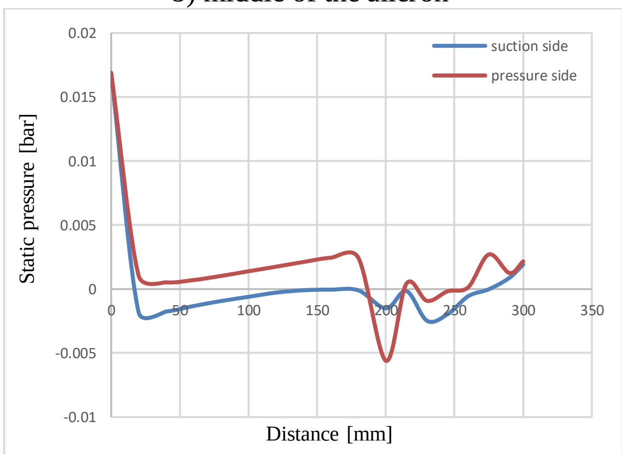
Fig. 12 illustrates how the aileron position influences the flow field. The pressure difference between the two surfaces is best observed in Fig. 12.b, where the pressure on the suction side of the wing is the highest.



a)middle of the wing



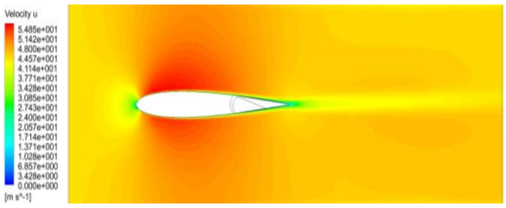
b) middle of the aileron



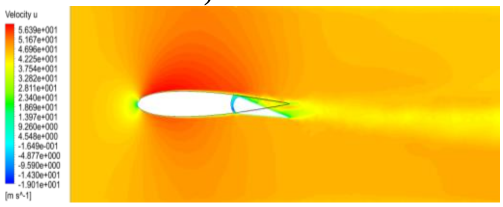
c) end of the aileron

Fig. 12 Pressure Distribution relative to the distance from the leading edge

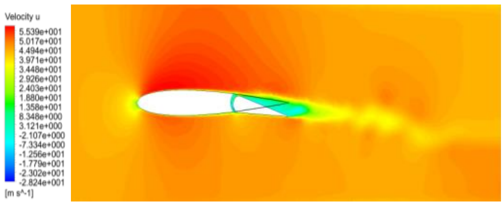
Fig. 13 shows the x - component of velocity. The detachment areas and the occurrence of the vortex are visible in Fig. 13.c. The fact that the position of the aileron influences the flow until the end of the wing is illustrated in Fig.13.e, where the output is not symmetrical, having a trend of inverse evolution to the position of the aileron.



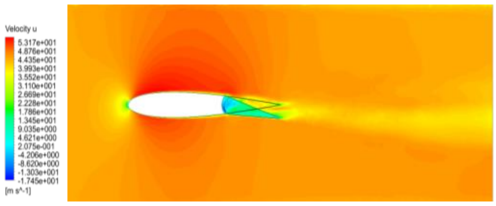
a)750 mm



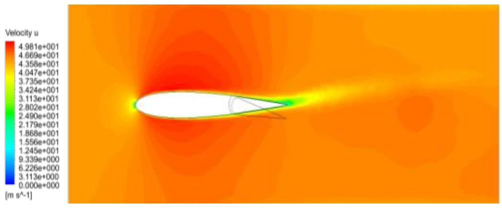
b)250mm



c)250 mm



d)-0.05 mm



e)-54mm

Fig. 13 Velocity variation in the x direction around the profile in the 5 regions

4 Conclusion

This paper presents the numerical analysis of a medium-sized wing, which has an aileron as control device. In this case, three positions of the aileron: 2°, 5° and 10°, were analyzed.

The results obtained from these analyses show that the aileron position and the velocity value have a great impact on the flow around a wing. With the increase of the aileron angle, an aerodynamic instability can be noticed. In the case of the 10° angle, the stream of vortices forming downstream of the wing in the area of the aileron can be observed. The pressure is another important factor to be taken into account in studies of this kind, the value of the pressure on the suction side is lower compared to the pressure side, which is absolutely normal – for generating lift force.

Acknowledgements

The authors gratefully acknowledge the financial support of the National Authority for Scientific Research−ANCS, UEFISCDI, through AFDPA research project, Contract 289/2014.

References:

[1] Kai Liu, Andrea Da Ronch, Daochun Li, Jinwu Xiang, Modeling of Unsteady Aerodynamics for a Falpping wing, *Science Direct*, IFAC – PapersOnLine 48- 28 (2015) 404 -408

[2] Erkut Başkut, Ali Akgül, Development of a Closely Coupled Procedure for Dynamic Aeroelastic Analyses, *Scientific Technical Review, 2012,Vol.62, No.2, pp.30-39*

[3] I. Ursu, D. D. Ion Guta, Daniela Enciu, A. Toader, M. Dan, C. Donciu, V. Turcan, Mathematical modeling of a V-stack piezoelectric aileron actuation, INCAS Bulletin, **8**, 4, 2016, pp. 141-156

[4] Theodore Theodorsen, General Theory of Aerodynamic Instability and the Mechanism of Flutter, Report National Advisory Committee for Aeronautics, December 1979

[5] Torsten H. Fransson, Aeroelasticity in Axial – Flow Turbomachines, CompEdu HPT/S5: Aeroelasticity/B1: Introduction /C5: Dynamic Aeroelasticity

[6] Kyoungsoo Lee, Ziaul Huque, Raghava Kommalapati, Sang-Eul Han, The Evaluation of Aerodynamic Interaction of Wind Blade Using Fluid Structure Interaction Method, *Journal of Clean Energy Technologies, Vol. 3, No. 4, July 2015*

[7] Philippe R. Spalart, Detached-Eddy Simulation, Annu. Rev. Fluid Mech. 2009.41:181-202,RENSSELAER POLYTECHNIC INSTITUTE

[8] Antiflutter demonstrator with piezoelectric actuator, Phase 3, Preliminary tests of AFDPA system, INCAS - National Institute for Aerospace Research "Elie Carafoli" - internal report, Code 16 0004 C, December edition 2016, DT 73/16 12 2016