33RD CONGRESS
OF THE INTERNATIONAL COUNCIL
OF THE AERONAUTICAL SCIENCES
STOCKHOLM, SWEDEN, 4-9 SEPTEMBER, 2022

ICAS 2022 SWEDEN

PRELIMINARY STUDY ON A FLUIDIC DEVICE SUPPORTING AIRCRAFT-FLIGHT CONTROL

Wienczyslaw Stalewski [1]

1 Łukasiewicz Research Network - Institute of Aviation, Al. Krakowska 110/114, 02-256 Warsaw, Poland

Abstract

A novel fluidic device concept designed to control the velocity circulation around an aircraft wing has been developed and pre-investigated based on an advanced computational approach. As assumed, the invented device is to support or even replace traditional control surfaces such as ailerons or rudders. The device operates based on simultaneous blowing and suction through the mini-nozzles located at the trailing edge of any lifting or control surface of aircraft. The mini-nozzles are arranged in two rows, one above the other. Compressed air is supplied to only one of these rows, depending on whether the lift force is to be increased or decreased. The quantity of lift-force growth or drop strongly depends on the level of overpressure of air feeding the blowing nozzles as well as on the level of underpressure generated in non-blowing nozzles. The presented device, currently developed at a low technology readiness level, has been studied based on a computational approach. At this preliminary stage, the investigations focused on a simple segment of a wing. Results of computational investigations have been discussed.

Keywords: fluidic devices, flight control, Coanda effect, wing trailing edge, blowing-suction, circulation control

1. General Introduction

Deflectable control surfaces like ailerons or rudders are classic flight control devices. In general, well-designed deflectable control surfaces do their job well. However, in aeronautical engineering practice, there are areas where the drawbacks of such solutions become apparent. First of all, when increasing the angle of deflection of a classic aileron or rudder, the risk of strong flow separation appearance on a given control surface increases and, consequently, the loss of the controllability of the aircraft. Deflectable control surfaces are quite an unfavourable solution for stealth-type aircraft, too. For such reasons, studies on alternative solutions for aircraft flight control devices are being conducted [2],[8]. The subject of the paper is in line with this direction of scientific and research work. The presented studies have been focused on the analysis of the possibility of supporting modern aircraft flight control systems through the use of the circulation control technique utilizing properly designed and optimized fluidic devices localized at a wing trailing edge. In this way, the idea of the device presented in this work was born. The presented study was conducted based on the author's great experience in the area of circulation control techniques in the aeronautical engineering domain [3],[4],[5],[6],[7].

2. Concept of the F2DCC Device

When starting the preliminary study on the fluidic support of classic ailerons or rudders, several assumptions were made:

- The device has to act bidirectionally, which means, it should both increase and decrease the lift force of an airfoil equally effectively.
- The device should work based on the circulation-control technique. Thus, it should be localized close to a trailing edge of an airfoil, but this must not require introducing any significant modifications in an airfoil shape, in particular thickening of the trailing edge.
- Compared to classic deflectable ailerons and rudders, the designed device should be much more resistant to the risk of losing its controllability due to strong flow separation occurring on the flight-control surface.
- The fluidic device should be as possible effective, which means it should operate at minimum power consumption.

Based on these assumptions, the idea of the F2DCC device (acronym derived from Fluidic 2-Directional Circulation Control) was born. The proposed device, schematically presented in Figure 1 and Figure 2, consists of a system of mini-nozzles located at a trailing edge of an airfoil. The mini-nozzles are arranged in two rows, one above the other. In Figure 1, the upper row of nozzles is shown in red while the lower row is in green. Compressed air is supplied to only one of the two nozzle rows at a time, depending on whether the airfoil lift force is to be increased or decreased. At nozzle outlets, the compressed-air stream is strongly deflected, nearly by 90 degrees, which influences a significant change of the velocity circulation, thus the significant change of lift force acting on the airfoil. The strong deflection of the air stream is obtained by the use of the Coanda effect appearing on specifically shaped surfaces of nozzle outlets. Additionally, the strong deflection of the air stream is enhanced by the generation of a certain level of underpressure in the nozzle row that is not transporting, at a moment, compressed air. This blowing-and-suction device has been specifically designed so as to minimize the power needed to achieve the required increase or decrease in airfoil lift, which characterizes the most efficient fluidic systems.

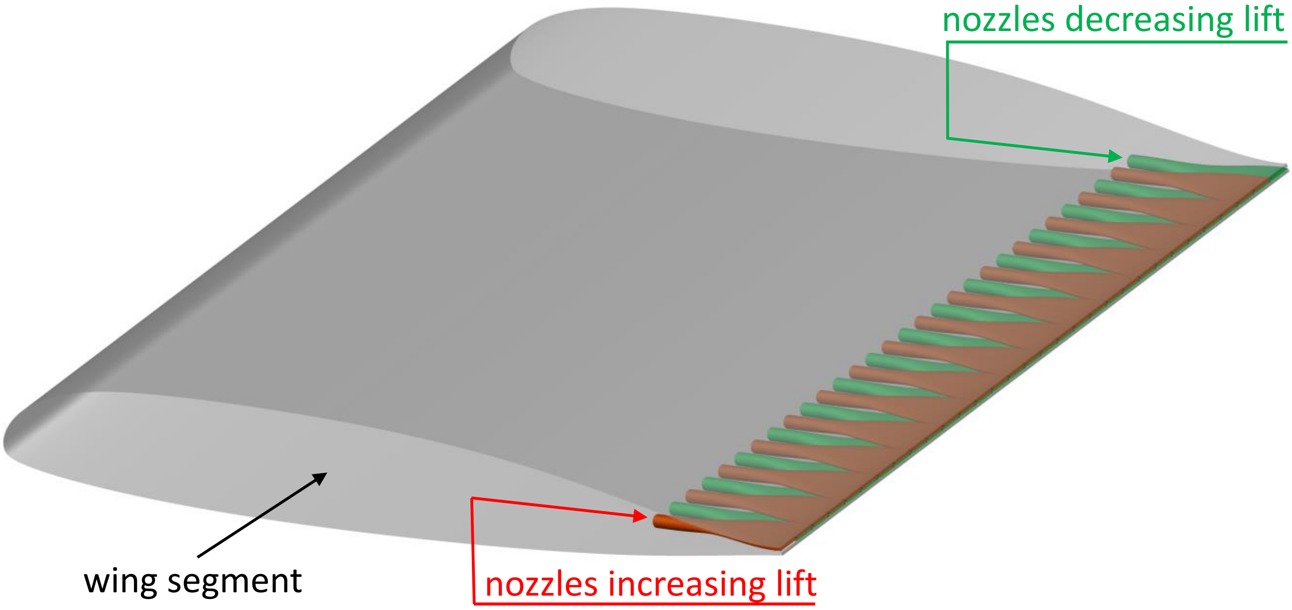

Figure 1 - Wing segment (aileron zone) equipped with a row of F2DCC devices placed at the trailing edge.

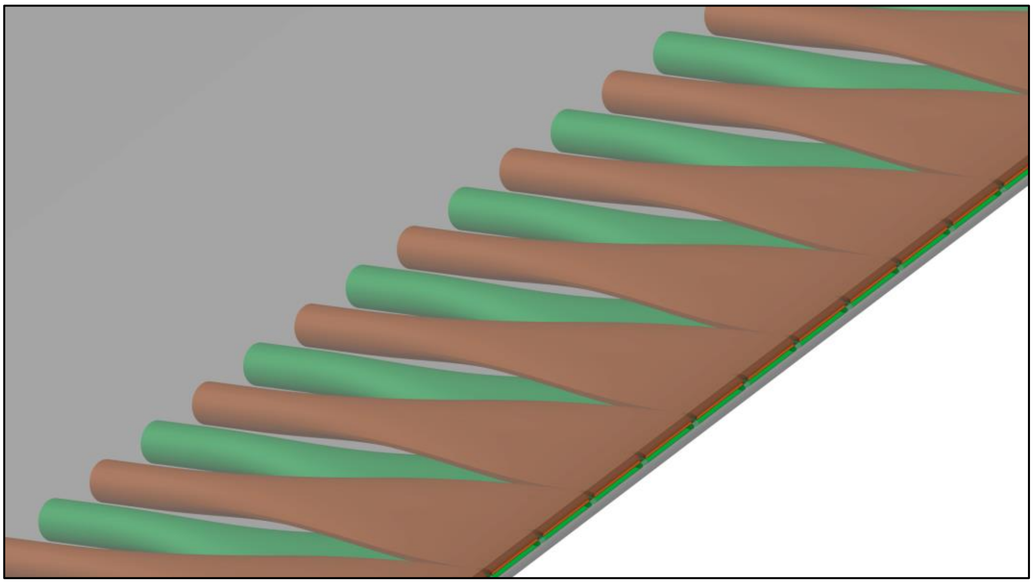

Figure 2 - Wing segment (aileron zone) equipped with a row of F2DCC devices placed at the trailing edge. Zoomed view.

It should be noted that, as shown in Figure 1 and Figure 2, the inlet ducts leading to the nozzles are positioned side-by-side, which favours the installation of the F2DCC device on airfoils with a relatively thin trailing part.

Flow effects caused by the acting F2DCC device are presented in Figure 3. Figure 3a presents the state of the flow around the airfoil trailing edge when the F2DCC device is off. In Figure 3b the device is working to increase lift force. The compressed air is delivered to the upper row of nozzles. In the ducts leading to lower nozzles, a certain level of underpressure is set. As result, the air streams flowing through the upper row of nozzles deflect themselves downward at the wing trailing edge. This effect leads directly to an increase in lift force acting on the airfoil. The reverse situation is shown in Figure 3c. The compressed air is delivered to the lower row of nozzles. In the ducts leading to upper nozzles, a certain level of underpressure is set. The air streams flowing through the lower row of nozzles deflect themselves upward at the wing trailing edge, which leads to a decrease in lift force.

Compared to the well-known DEMON UAV [2],[8], the developed aircraft flight-control system is similar functionally, but different in some very important technical details. According to the available data, the DEMON flight-control system utilizes pure blowing of compressed air while the F2DCC system uses both the air blowing and suction techniques. The F2DCC design is directed primarily to achieve the high efficiency of the developed flight control system while maintaining the thin trailing edge of the aircraft wing, which allows for avoiding of the "drag penalty" phenomenon.

3. Research Methodology

In the presented preliminary stage of the research, a methodology based on a computational-simulation approach was applied. The research focused on a wing segment corresponding to an aileron zone. Figure 1 presents the investigated configuration. Along the whole span of the wing-segment trailing edge, the F2DCC device was mounted. In the developed computational model, in addition to the flow region around the aileron wing zone, also the nozzles and inner channels feeding the nozzles with compressed air, were faithfully reproduced. The external and internal flow simulations were conducted using the ANSYS FLUENT code [1] implemented to solve the 3D URANS (Unsteady Reynolds Averaged Navier-Stokes) equations. The assumed flow model was: 3D, unsteady, compressible, viscous, turbulent with the turbulence model: Shear-Stress-Transport k-ω. The computational mesh was of high quality and density (Y+ $\approx$ 1). The 3D URANS computations also concerned the flow inside the nozzle ducts. At inlets to the ducts leading to the blowing nozzles, the "pressure-inlet" boundary condition was established. At the inlets to the ducts leading to the sucking nozzles, the "pressure-outlet" boundary condition was assumed.

4. Results of Computational Simulations

4.1 Results of Stationary Simulations

Exemplary quasi-steady simulations of the flow around the wing segment were conducted for the Mach and Reynolds numbers: M=0.1, Re=1164800 and angle of attack α = 0 deg. The intensity of blowing of compressed air from the nozzles was expressed in terms of the Blowing Momentum Coefficient (C_μ) defined as:

$$C_\mu = \frac{\dot{m} \cdot V_J}{\varrho_\infty \cdot V_\infty^2 \cdot S} \tag{1}$$

where $\dot{m}$ - mass flow rate, V_J - jet velocity at a nozzle outlet, ϱ_∞- free-stream air density, V_∞- free-stream air velocity, S - reference area of the wing segment. In the presented example, only two, similar values of C_μ were taken into account: C_μ = 0.0248 and C_μ = 0.0247 for the blowing increasing and decreasing the lift force, respectively.

To evaluate the performance of the F2DCC device in controlling aircraft flight, the lift coefficient C_L of the wing segment was selected. This coefficient is defined as follows:

$$C_L = \frac{L}{1/2 \cdot \varrho_\infty \cdot V_\infty^2 \cdot S} \tag{2}$$

where L - lift force. If the considered wing segment we assume as an aileron zone of the aircraft semi-wing, the change in C_L coefficient of this segment is strictly correlated with the rolling-moment-coefficient change of the aircraft, generated by a single "fluidic aileron".

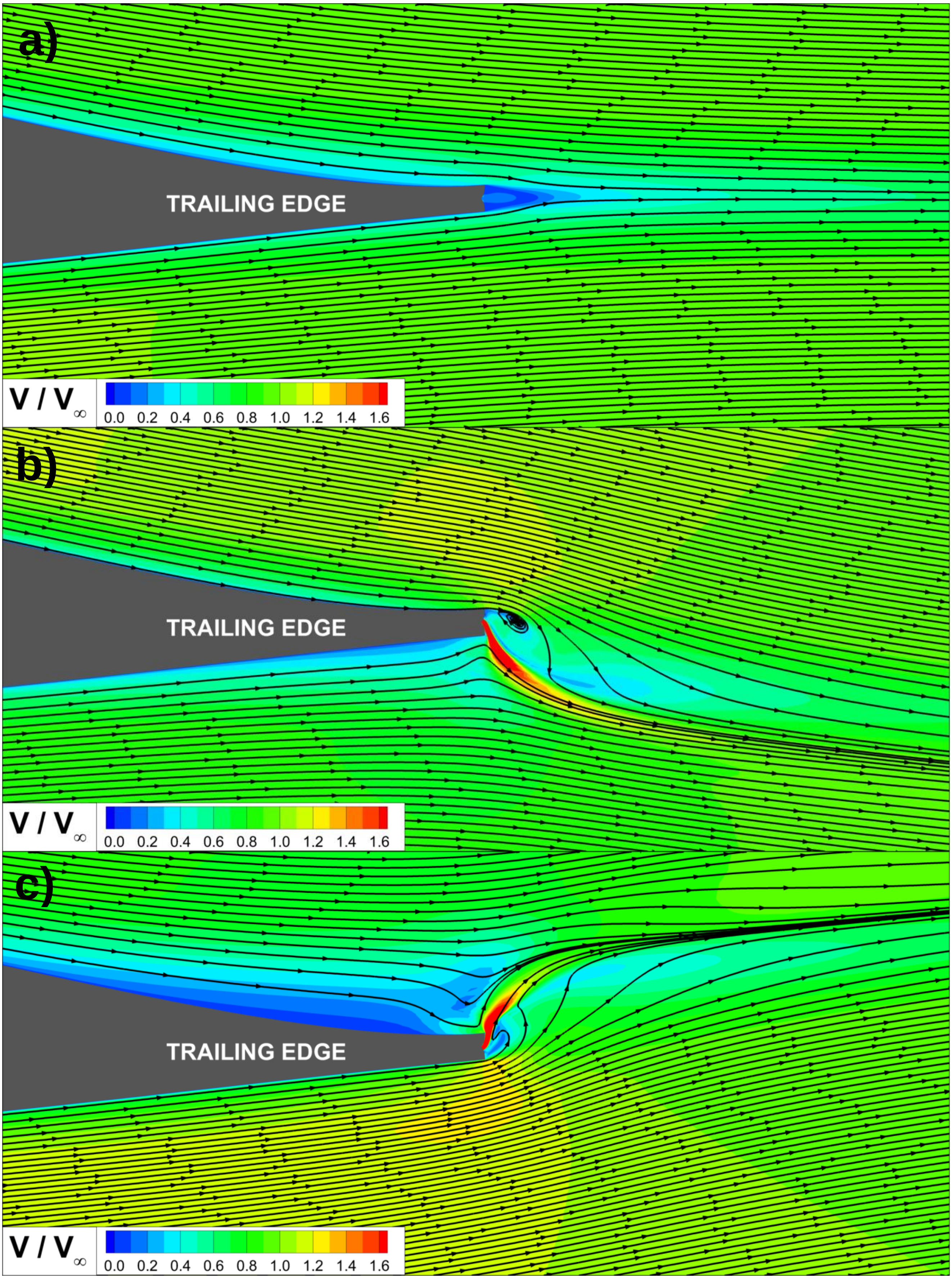

Figure 3 - Flow effects caused by the operating F2DCC device.
The device is a) off, b) increasing lift force, and c) decreasing lift force.

To assess the energetic efficiency of the presented fluidic device, the Blowing Efficiency Coefficient (η) has been defined as follows:

$$\eta = \frac{|\Delta C_L|}{C_\mu} \quad (3)$$

where ΔC_L is the growth or drop in Lift Coefficient (Eq.2), caused by the blowing of the air jet of nondimensional momentum equal C_μ (Eq.1). Table 1 presents correlations between the Blowing Momentum Coefficient (C_μ) characterising the momentum of air blown from the nozzles and wing-segment Lift Coefficient C_L (Eq.2), measured in the conducted flow simulations. The presented results show, that:

- The blowing of intensity C_μ = 0.0248 from the upper row of mini-nozzles, increased the lift coefficient C_L by 0.634,
- The blowing of intensity C_μ = 0.0247 from the lower row of mini-nozzles, decreased the lift coefficient C_L by 0.646.

In the last column of Table 1, the Blowing Efficiency Coefficient η (Eq.3) is presented. In both cases, the efficiency of blowing is similar and relatively high.

Table 1 - Correlations between Blowing Momentum Coefficient (C_μ), Lift Coefficient (C_L) and Blowing Efficiency Coefficient (η) observed in the conducted simulations.

Configuration	C_μ	C_L	η
no blowing	0.0000	0.129	-
blowing from the upper row of the nozzles	0.0248	0.762	25.6
blowing from the lower row of the nozzles	0.0247	-0.517	26.2

Relative-velocity-magnitude contours, in the cross-section of the flow field around the wing aileron zone, for all three computational configurations, are presented in Figure 4, Figure 5, and Figure 6. The figures show, that the F2DCC device while keeping the thin trailing edge of the airfoil, can deflect the flow close to the trailing edge even by 90 degrees, which confirms the high performance of this device.

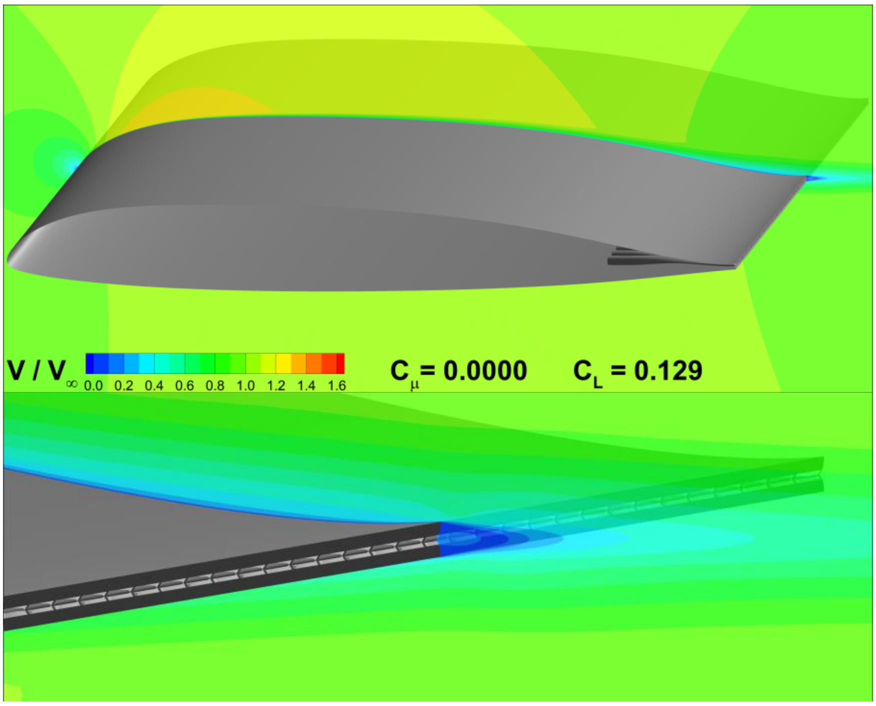

Figure 4 - Velocity-Magnitude contours in a cross-section of the flow field around the wing aileron zone. "No blowing" computational case.

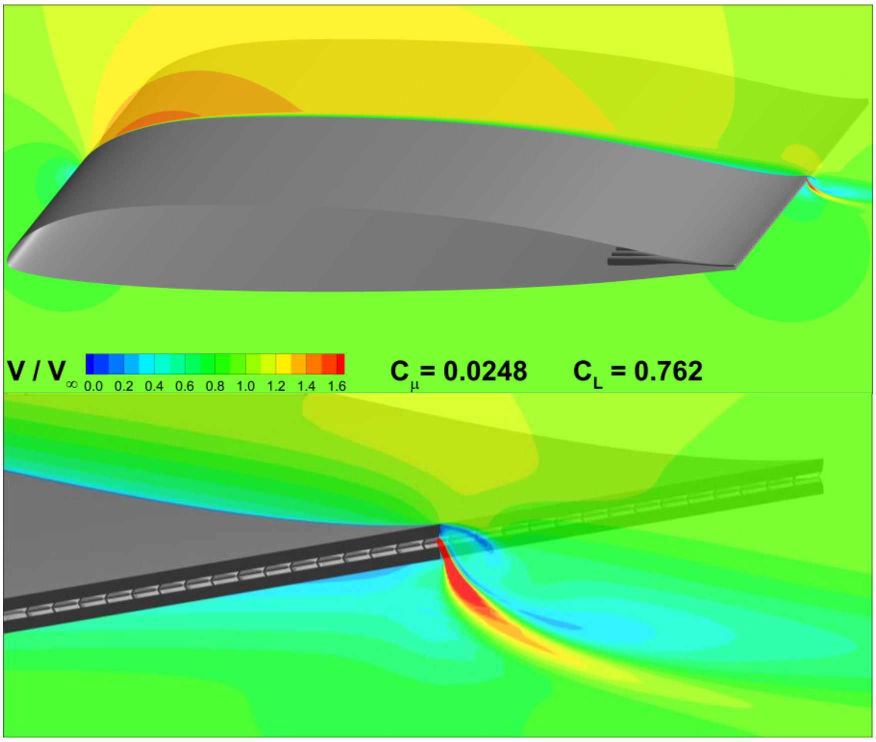

Figure 5 - Velocity-Magnitude contours in a cross-section of the flow field around the wing aileron zone. "Blowing down" computational case.

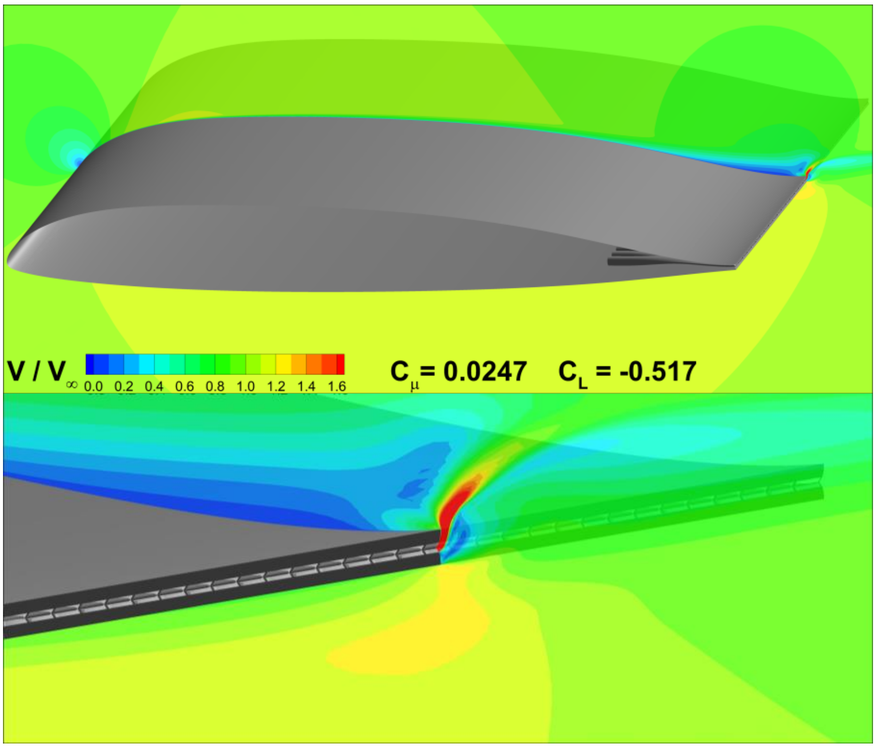

Figure 6 - Velocity-Magnitude contours in a cross-section of the flow field around the wing aileron zone. "Blowing up" computational case.

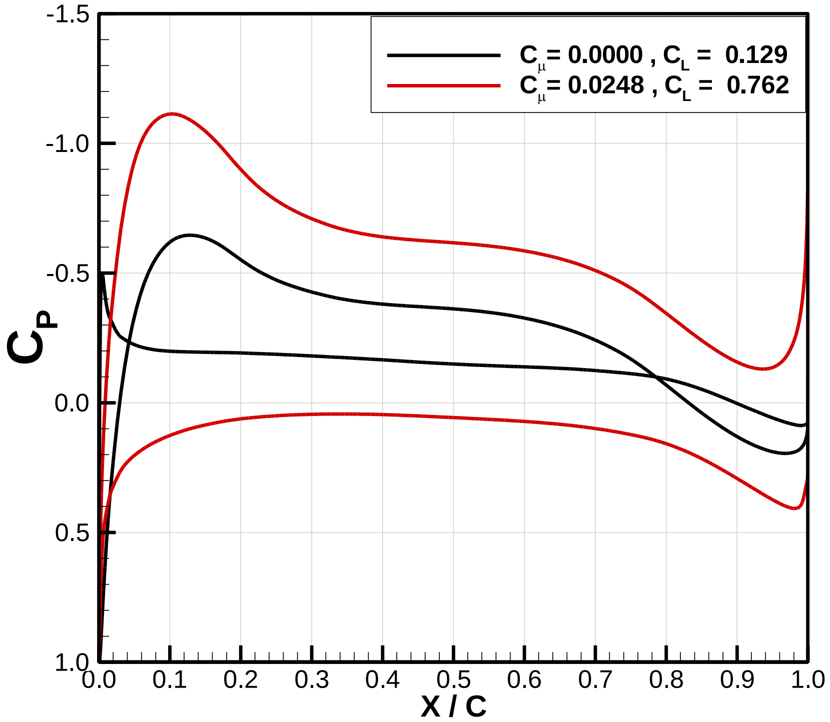

Figure 7 - Comparison of pressure coefficient (C_P) distribution in the aileron-zone cross-section, for two computational cases: "no blowing" and "blowing down".

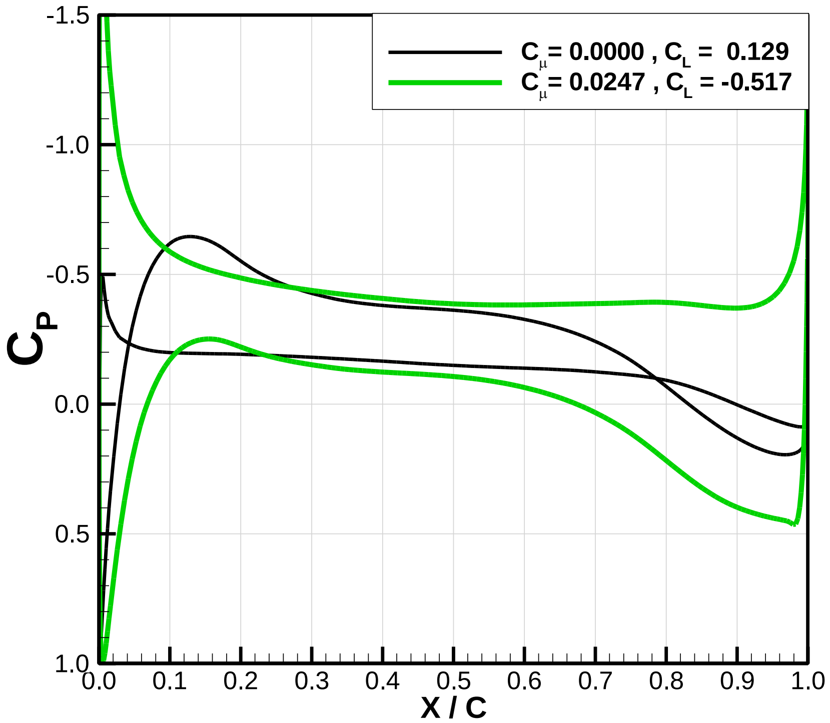

Figure 8 - Comparison of pressure coefficient (C_P) distribution in the aileron-zone cross-section, for two computational cases: "no blowing" and "blowing up".

4.2 Results of Unsteady Simulations

An exemplary fully unsteady simulation of the flow around the aircraft wing aileron zone was conducted for Mach and Reynolds numbers: M=0.1, Re=1164800 and angle of attack α = 2 deg. The simulation was conducted to demonstrate the fluidic generator of an effect of pitching oscillations of classic aileron around its rotation axis. The oscillation frequency was 1 Hz. During the simulation, the pressure at inlets to lower and upper rows of nozzles was changed according to the sinusoidal formula, as shown in Figure 9, where: t - time, T – period of oscillations (= 1sec), Δ_P - undimensional pressure, defined as:

$$\Delta_P = \frac{p - p_\infty}{p_\infty} \quad (4)$$

where: p – static pressure, p_∞ - atmospheric pressure. Established changes of pressure, set at nozzle inlet, caused flow through the nozzle ducts. If overpressure was set, the blowing from the nozzle was observed, while for underpressure certain amount of air was sucked by the nozzle. The intensity of flow through the nozzle outlet, presented in terms of the Blowing Momentum Coefficient (Eq. 1) as a function of undimensional time (t/T) is shown in Figure 10. Positive values of C_μ correspond to blowing while negative values correspond to the suction of air.

Such oscillating actuation of flow through the nozzles caused unsteady, oscillating phenomena if flow around wing aileron zone. Figure 11 shows changes in wing segment lift coefficient (C_L) and pitching moment coefficient (C_m) as functions of undimensional time (t/T).

Figure 12 presents pressure coefficient (C_P) distribution in the wing aileron zone cross-section, in selected moments (t/T). For these moments, current values of the lift coefficient (C_L) are presented too.

Figure 13, Figure 14 and Figure 15 show relative-velocity-magnitude contours around the wing aileron zone trailing edge, for selected moments t/T. Comparing these graphs with Figure 11 it may concluded, that for maximum and minimum lift coefficient (C_L) the air jets blown from the trailing-edge nozzle, are perpendicular to the wing chord.

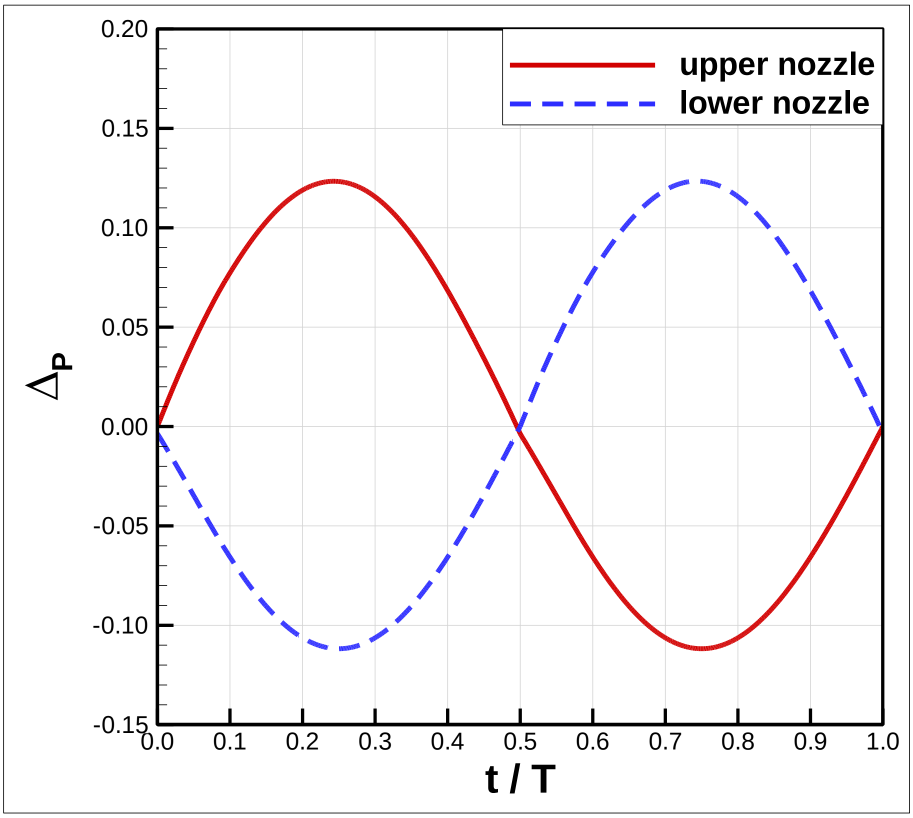

Figure 9 – Changes of static pressure at inlets to lower and upper rows of nozzles, during one period (1 sec) of "oscillations" of the fluidic aileron.

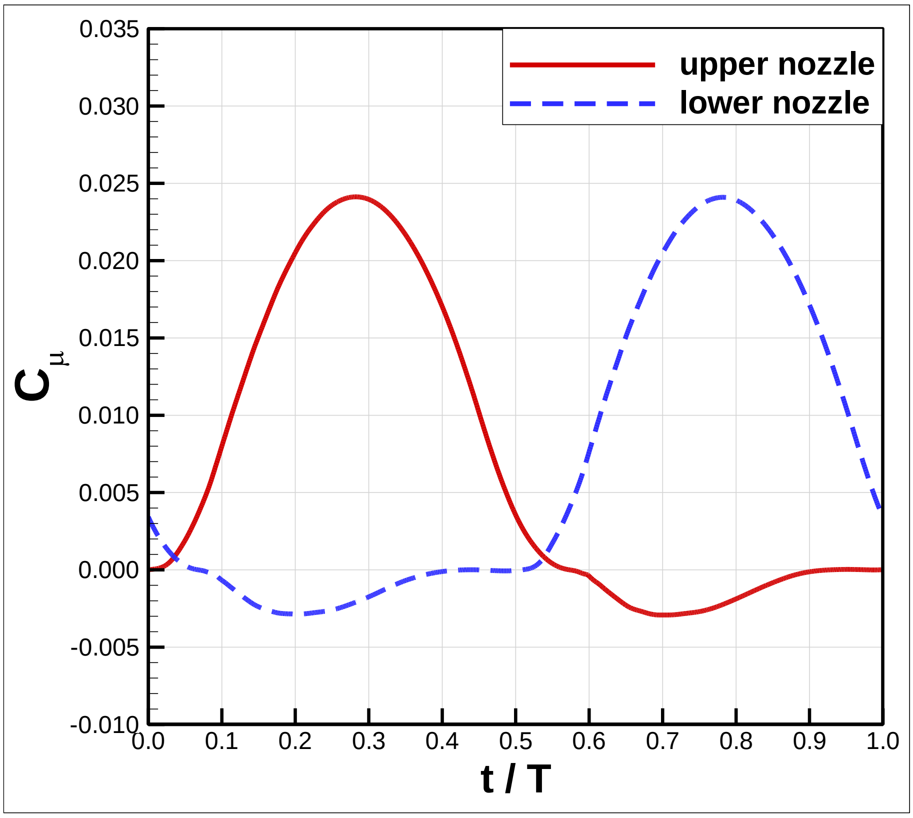

Figure 10 - The Blowing Momentum Coefficient measured at nozzle outlets as a function of undimensional time (t/T).

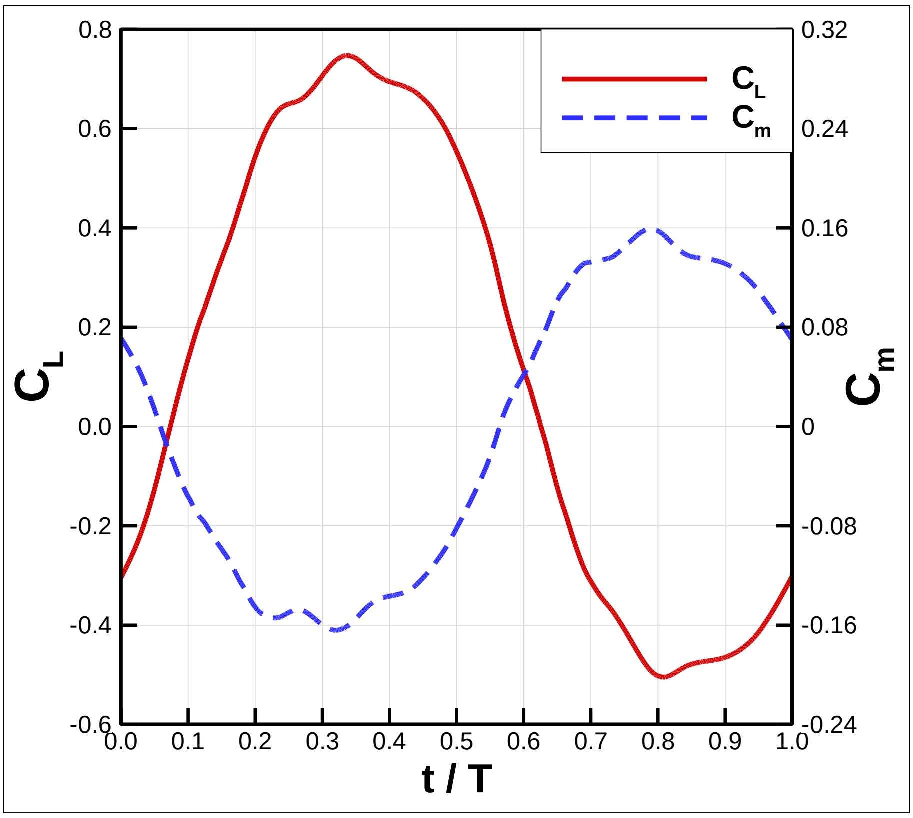

Figure 11 - Values of lift coefficient (C_L) and pitching moment coefficient (C_m) of the wing aileron zone as functions of undimensional time (t/T).

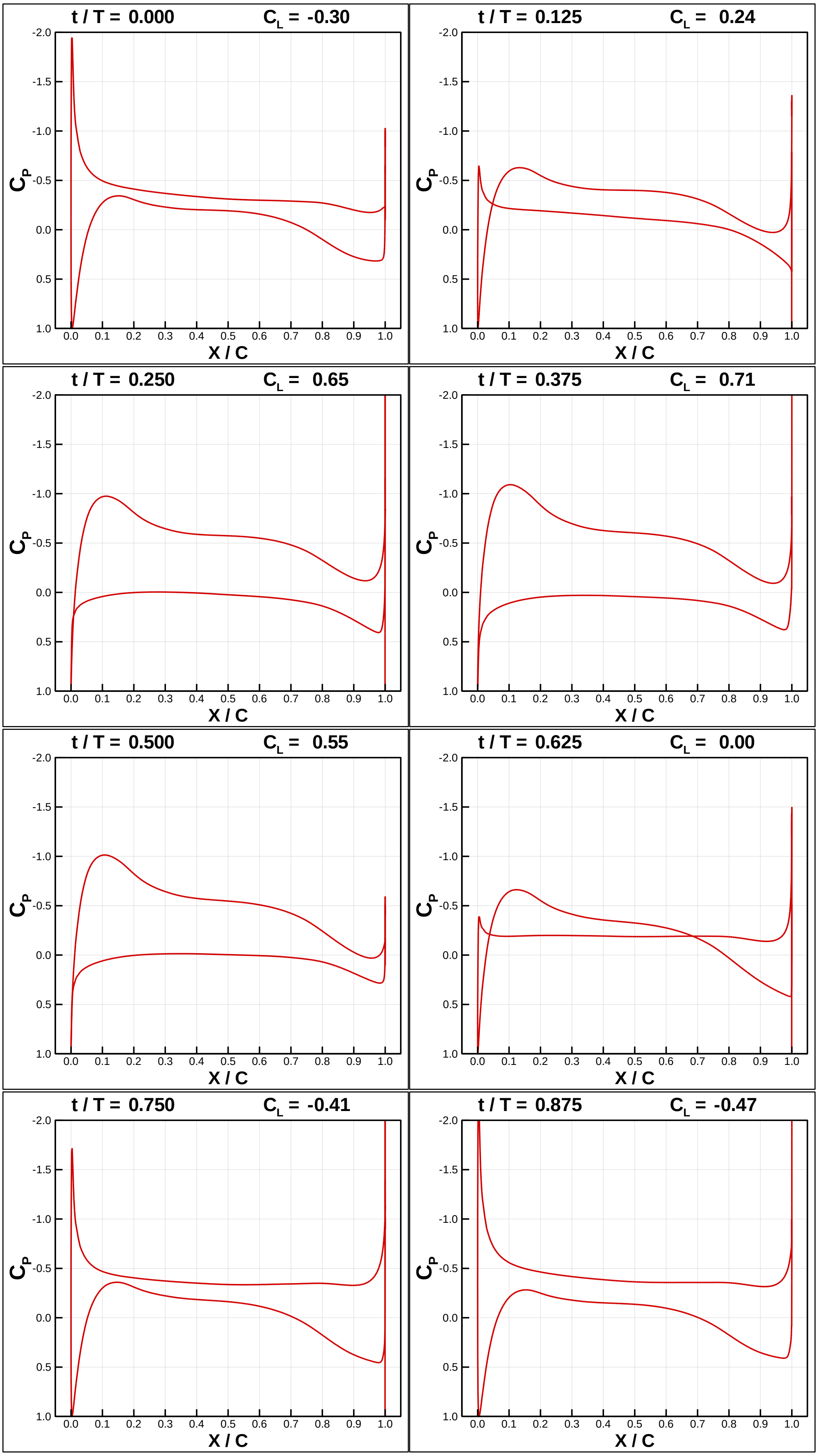

Figure 12 – Pressure coefficient (C_P) distribution in the wing cross-section, in selected moments (t/T).

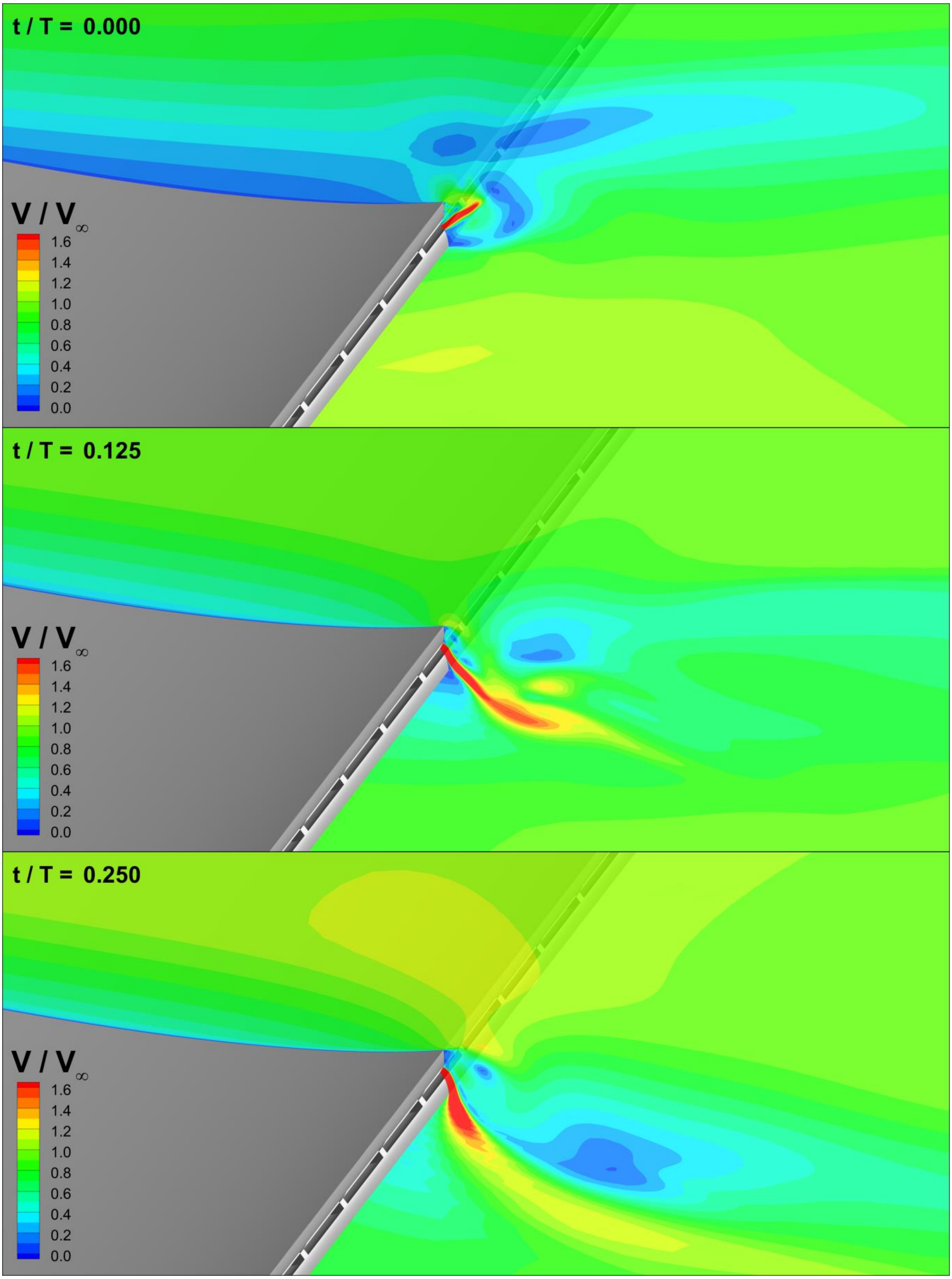

Figure 13 - Relative-velocity-magnitude contours around the wing trailing edge, for selected moments t/T = 0.000, 0.125, 0.250.

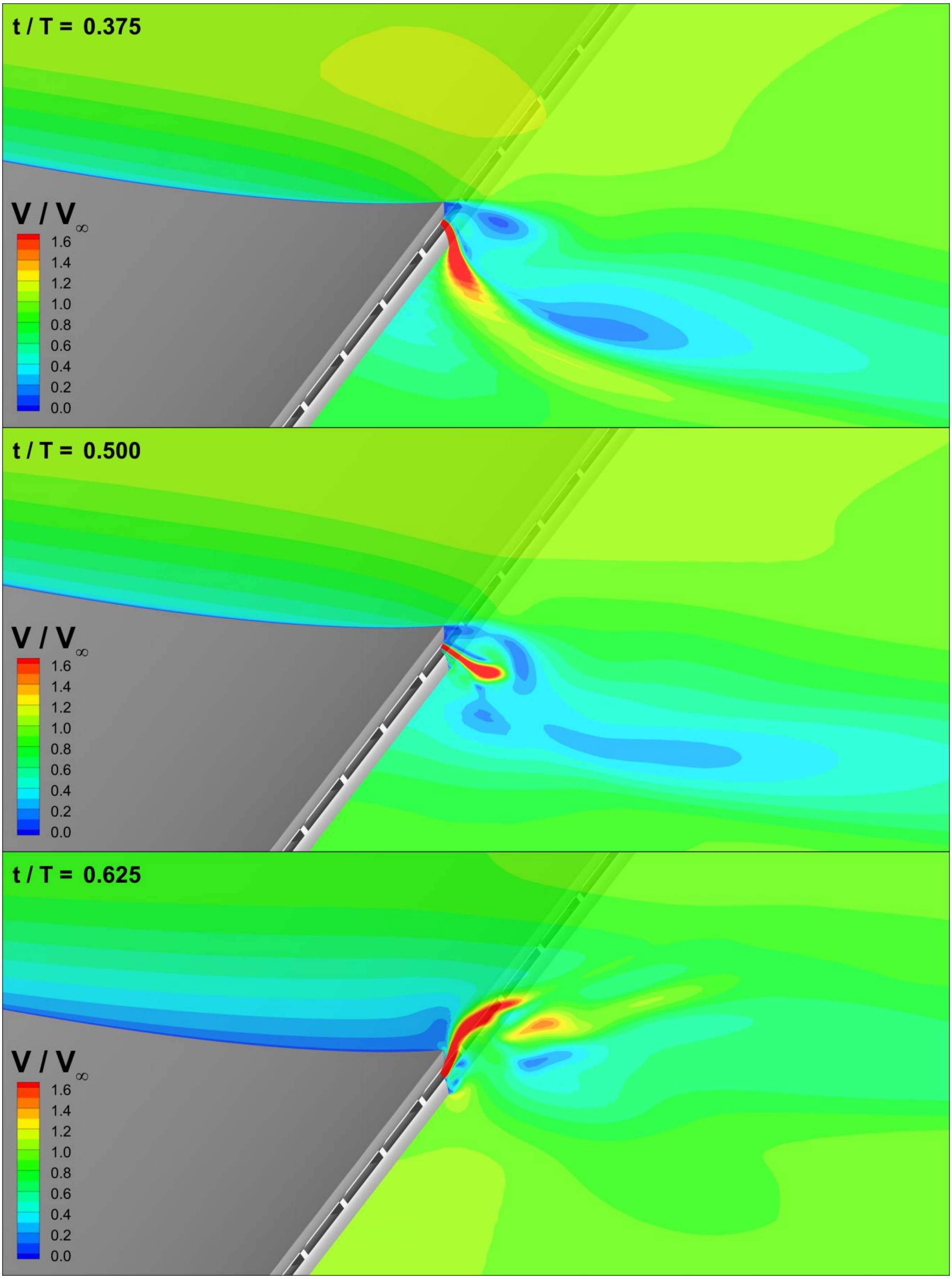

Figure 14 - Relative-velocity-magnitude contours around the wing trailing edge, for selected moments t/T = 0.375, 0.500, 0.625.

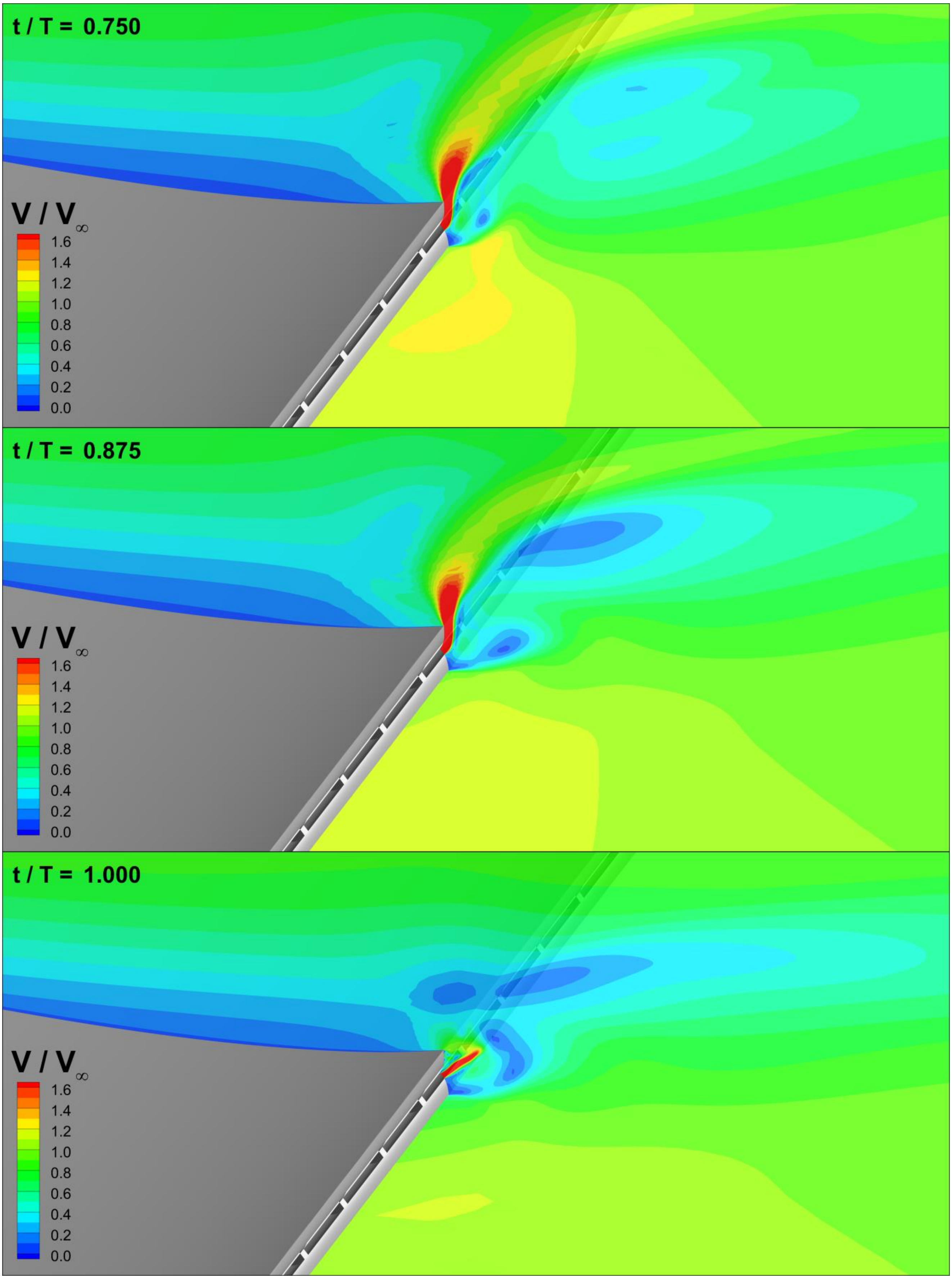

Figure 15 - Relative-velocity-magnitude contours around the wing trailing edge, for selected moments t/T = 0.750, 0.875, 1.000.

5. Conclusions

The innovative fluidic device, named F2DCC, has been developed and preliminarily investigated computationally. The device is intended to stimulate fast changes in the lift force, based on the circulation control technique. Compared to other similar solutions, the presented device is distinguished by: simplicity, no moving parts, applicability on airfoils with a thin trailing edge as well as high performance and efficiency. It is also unique that the F2CDD device uses the technique of simultaneous blowing and sucking of air, which was motivated by the desire to achieve its highest possible aerodynamic performance and energetic efficiency.

The device was developed as a fluidic replacement for classic ailerons and rudders. The results of the preliminary computational simulations confirmed the great potential of the developed device in this range of applications. Other applications of the F2DCC device are also possible. For example, it can be considered for use in wing load control systems as well as in high lift systems.

However, the first planned step of further research is to bring the F2DCC device to a higher level of technological readiness. It is also planned to optimize the device, aimed primarily at maximizing its efficiency so that the expected aerodynamic effects are obtained with the lowest possible power necessary to generate an appropriate amount of compressed air supplying the nozzles. The presented device is a subject of patent-pending.

6. Contact Author Email Address

mailto: wienczyslaw.stalewski@ilot.lukasiewicz.gov.pl

7. Copyright Statement

The authors confirm that they, and/or their company or organization, hold copyright on all of the original material included in this paper. The authors also confirm that they have obtained permission, from the copyright holder of any third party material included in this paper, to publish it as part of their paper. The authors confirm that they give permission, or have obtained permission from the copyright holder of this paper, for the publication and distribution of this paper as part of the ICAS proceedings or as individual off-prints from the proceedings.

References

[1] ANSYS, Inc. (2018). ANSYS FLUENT User's Guide. Release 19.2. Available from: http://www.ansys.com

[2] Fielding J. P., Lawson C. P., Pires R. and Monterzino G., Development of The Demon Technology Demonstrator UAV, *Proceedings ICAS 2010, 27th International Congress of the Aeronautical Sciences*, 19 - 24 September 2010, Nice, France.

[3] Stalewski W., 2013, Computational Design and Investigations of Closed-Loop, Active Flow Control Systems Based on Fluidic Devices, Improving a Performance of Wing High-Lift Systems. *Proceedings of the 4th CEAS Conference* in Linkoping, Sweden.

[4] Stalewski W., Sznajder J., 2014, Wing Load Control via Fluidic Devices, Proceedings of 4th EASN Workshop on Flight Physics and Aircraft Design, 27-29 October, Aachen, Germany.

[5] Stalewski W., 2016, Flow Control on Helicopter-Rotor Blades via Active Gurney Flap, Proceedings of the 30th ICAS Congress of the International Council of the Aeronautical Sciences, ICAS, Daejeon, Korea, 25-30 September. ICAS 2016 CD-ROM Proceedings. ISBN 978-3-932182-85-3.

[6] Stalewski, W., New Concepts of Environmentally Friendly High-Lift Devices and their Application in a Small-Aircraft-Transport Area, *Proceedings of 1st Aerospace Europe Conference 2020*, Bordeaux, France, 25-28 February, 2020.

[7] Stalewski, W., Improvement of Helicopter Performance through Pulse Air Jets Blown From the Trailing Edge of Main Rotor Blades, *Proceedings ICAS 2021, 32nh International Congress of the Aeronautical Sciences*, 6 - 10 September 2021, Shanghai, China.

[8] Yarf-Abbasi, A, Clarke, A, Lawson, C. P. and Fielding, J. P., Demon Demonstrator UAVs, *Proceedings of ICAS 2008, 26th International Congress of the Aeronautical Sciences*, Anchorage, Alaska, September 2008.