AERODYNAMIC STUDY OF AIRFOIL WITH ROTARY SLAT

Frolov Vladimir

Samara State Aerospace University

Russia, Samara,
Moskovskoe shosse, 34

frolov_va_ssau@mail.ru

Klementjev Vasilij

Samara State Aerospace University

Russia, Samara,
Moskovskoe shosse,34

Kurkin Evgenij

Samara State Aerospace University

Russia, Samara,
Moskovskoe shosse, 34

eugene.kurkin@mail.ru

Lyaskin Anton

Samara State Aerospace University

Russia, Samara,
Moskovskoe shosse, 34

alyaskin@mail.ru

Shakhov Valentin

Samara State Aerospace University

Russia, Samara,
Moskovskoe shosse, 34

shakhov@ssau.ru

Abstract. Aerodynamic study of an airfoil with rotary slat conducted in Samara State Aerospace University (Russia) is presented. "Rotary slat" is a term for the rotating cylinder installed at the nose of the airfoil. Two types of the airfoil were investigated, both sharing the same circular nose but with different aft parts: one was wedge-shaped, the outline of the other was made of two circular arcs. Both CFD modelling using Star-CD and ANSYS CFX, and experimental study in a low-speed subsonic wind-tunnel were conducted. Experimental study consisted of measurements for pressure distribution at the airfoil surface and flow visualization using smoking wire and laser sheet. Variable parameter were the angle of attack α and relative rotational velocity of the slat $\overline{U} = U/V_{\infty}$, where U – linear velocity of the cylinder surface; V_{∞} – flow velocity. Angle of attack α varied from 0 to 10 degrees, and relative rotational velocity $\overline{U}$ - from 0 to 5. Different flow velocities were considered - 7 *m/s* and 31.5 *m/s* for the experiment and 10 *m/s* for CFD modelling. CFD models used in Star-CD simulation were 3D, and featured a stationary diaphragm bisecting the rotating cylinder in the spanwise direction (in the experiments it was used for pressure orifice location). RNG *k-ε* turbulence model was used together with non-equilibrium wall function accounting for streamwise pressure gradient. CFD models used in ANSYS CFX were 2D. Two different turbulence modelswere compared: standart *k-ε* and *k-ω* SST. CFD simulations and experiments resulted in the estimation of key aerodynamic properties (lift and drag coefficients, lift-to-drag ratio) as functions of kinematic parameters α and $\overline{U}$. It was found that rotary slat can increase lift-to-drag ratio comparing to $\overline{U} = 0$ case. Flow visualization have shown that it can prevent airfoil stall for the angle of attack up to $\alpha \approx 90$ degrees.

Keywords. Airfoil, rotary slat, experiment, Star-CD, ANSYS CFX, lift coefficient, drag coefficient, lift-to-drag ratio, pressure distribution, visualization.

1 Introduction

The idea to use rotating cylinders in order to increase kinetic energy of the boundary layerwas proposed by Prandtl. In 1906 he studied the case of isolated rotating cylinder [1]. He also proposed to use rotating cylinder located at the nose of the airfoil in order to suppress stall (Fig. 1) [1].

Figure 1: Airfoil with the rotating cylinder at the nose, proposed by Prandtl [1]

Measurements made by Wolf in Netherlands have shown that the wing with an integrated rotor can provide maximum lift coefficient $C_{L\,max}$ = 2.43 at the angle of attack α = 41.7º [1]. In [1] one can also find flow maps for isolated cylinders rotating at different relative velocities. In early 1920s Yuriev in Russia [2] proposed the design of the wing with a moving surface, in order to increase aerodynamic performance. In 1938 this concept was tested by A. Favre [3]. Favre's model had three rotors supporting a wide moving band at the upper side of the airfoil. The airfoil relative thickness was about 23% and aspect ratio of the model wing was about 2.96. The experiments have shown that stall performance is sensitive to relative velocity of the moving band $\bar{U} = u/V_\infty$, where u – velocity of the band; V_∞ – airspeed. Favre was able to achieve maximum lift coefficient $C_{L\,max} =$ 3.5 at $\bar{U} = 2{,}64$ and $\alpha = 55°$, while for original airfoil without moving surface $C_{L\,max}$ < 0.75.

Different reviews concerning boundary layer control with by means of moving surfaces can be found in [4-6]. Studies of boundary layer control for the combinations of rotating cylinder and flap for STOL aircraft were presented in [7]. Efficiency of separation control with a rotating cylinder in the nose part of the airfoil was investigated in [8, 9]. A big contribution to the study of boundary layer control by means of rotating cylinders was made by the team of researches leaded by V.J. Modi [10-19]. One of the latest works dedicated to this problem was [20], which presents the results for $\bar{U}[0; 4]$ and $\alpha[0; 40]$.

Though there were a lot of studies concerning the problem of boundary layer control by rotating cylinders, the details of this topic are still not clear enough.

This paper presents the results of aerodynamic studies of an airfoil with rotary slat made in Samara State Aerospace University (Russia). These studies included both CFD simulations with CD-Adapco Star-CD and ANSYS CFX codes and experiments in low-speed subsonic wind-tunnel.

2 Object of the study and CFD model

The object of the present study was a rotating cylinder with a wedge-shaped streamer behind it. This shape resulted in the airfoil with a leading edge formed by circular arc about 200º span and straight upper and lower surfaces. The similarity between experimental ("physical") and CFD ("virtual") models was maintained as close as possible. General view of the model is shown at Fig.2.

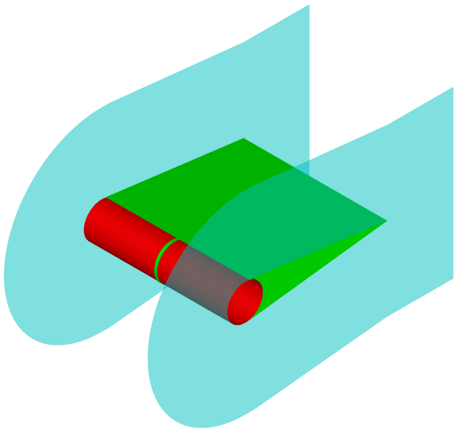

Figure 2: Model of the airfoil with rotary slat

Diameter of the cylinder was 50 *mm*. Model chord b was 225 *mm*, span 203 *mm*. Aerodynamic plates were installed at the tips to compensate for low aspect. Rotating cylinder was split into two parts in the spanwise direction, with stationary 3.3 *mm* wide diaphragm between them. This diaphragm was necessary in order to locate pressure orifices. Radial gap between the cylinder surface and the streamer was 0.5 *mm*, axial gap between the cylinder parts and the diaphragm was also 0.5 *mm*. Uniform airflow velocity V_∞=10 *m/s* and 20 *m/s* was specified at inlet. Angular velocities of the rotating cylinder and corresponding relative velocities of its surface are given in Table 1.

Table 1 – **Angular velocities of the rotating cylinder and corresponding relative velocities of its surface**

Ω, *RPM*	0	3750	7500	11250	15000	19100
$\bar{U}$	0	1	2	3	4	5

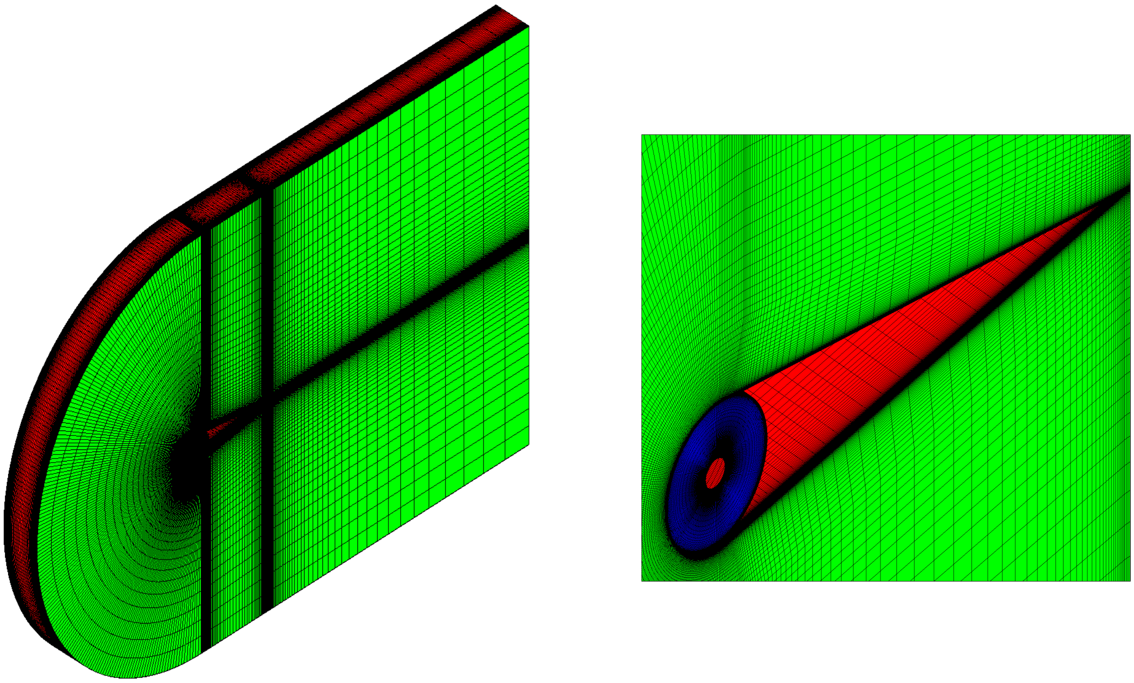

Figure 3: Computational mesh for CFD simulations

All simulations were fully turbulent, with *k-ε* turbulence model with non-equilibrium wall function, accounting for pressure gradient. Values of forces were used for convergence monitoring.

3 Results of CFD analysis with CD-Adapco Star-CD

Fig.4 and Fig.5 present the plots of lift and drag coefficient vs. relative velocity for $\alpha = 0°$. Fig. 6 presents the details of flow field near the rotating cylinder at maximum relative velocity. Zero values of velocity identify two recirculation regions – one in front-bottom of the cylinder and the other (big one) at the lower surface of the streamer.

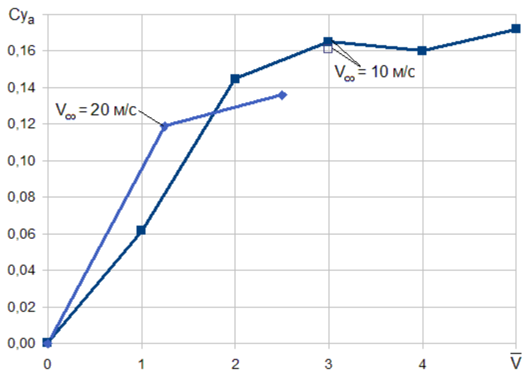

Figure 4: Lift coefficient vs. relative velocity

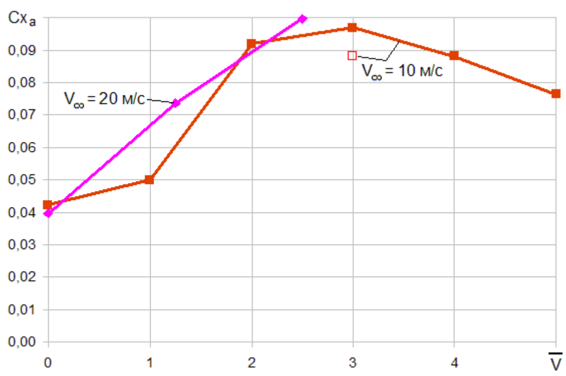

Figure 5: Drag coefficient vs. relative velocity

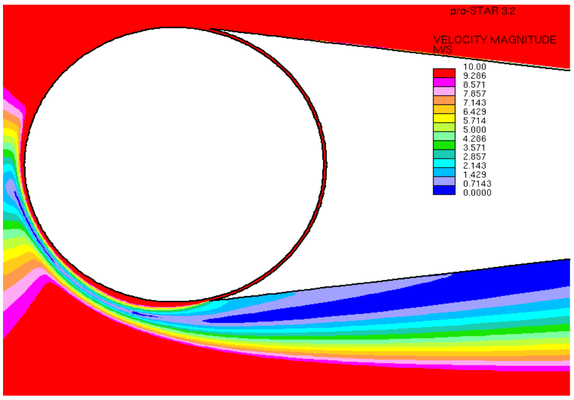

Figure 6: Velocity magnitude, $\bar{U} = 5$

4 Results of CFD analysis with ANSYS CFX

While simulations with STAR-CD were made for full 3D model, simulations with ANSYS CFX were made for 2D.

Boundary conditions were the same as for STAR-CD simulations. Angle of attack α varied between 0^0 and 10^0. The influence of the radial gap between the rotating cylinder surface and the streamer was studied. Velocity magnitude plots are shown at Fig. 7. Lift and drag coefficients are shown at Fig.8.

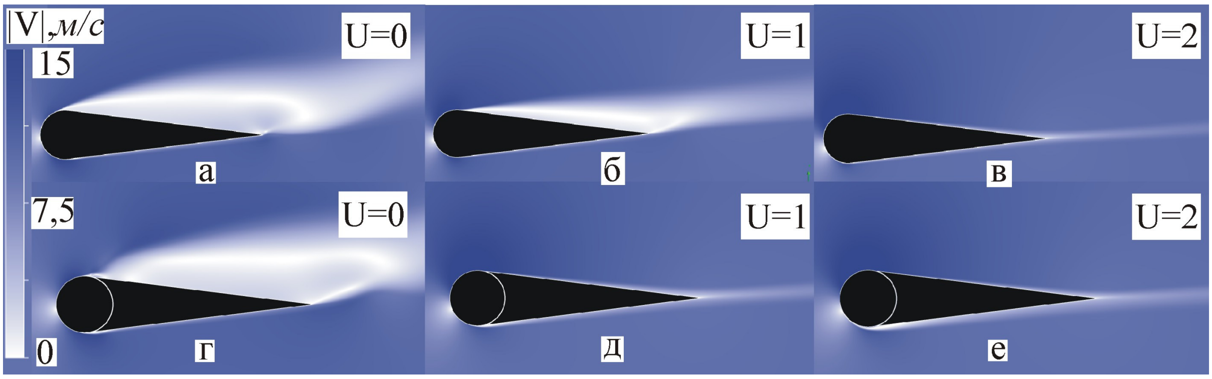

Figure 7: Velocity magnitude plots for α=10^0: upper row – no radial gap between the cylinder and the streamer; lower row – radial gap equal to 0.5 *mm*

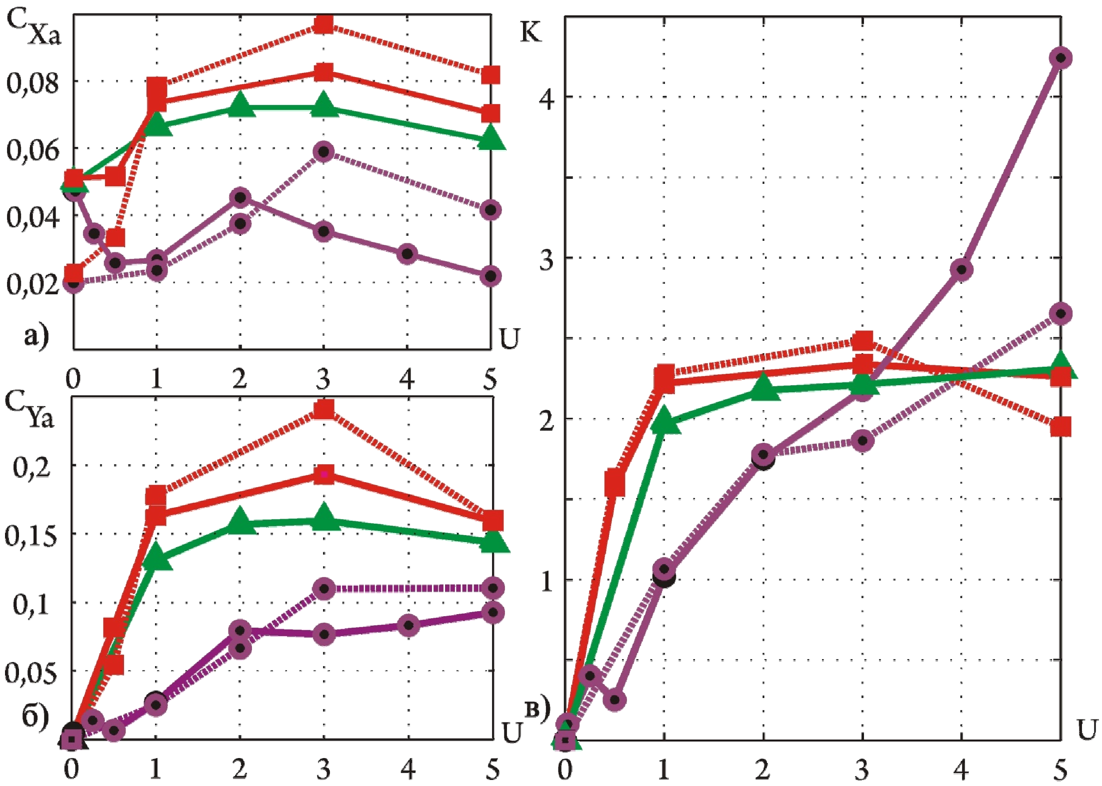

Figure 8: Drag (C_{Xa}), lift (C_{Ya}) coefficients and lift-to-drag ratio (K) for different radial gaps (circles for 0, triangles for 0.25 *mm*, squares for 0,5 *mm*) at α=0° and different turbulence models: *k-ω* SST (solid lines) and *k-ε* (dotted lines)

5 Comparison between CFD and experimental results

Fig.9 shows comparison between pressure distributions obtained in the experiment and calculated with ANSYS CFX. It can be seen that the results are in good agreement, so the results of the CFD simulations for the airfoil with rotary slat are correct.

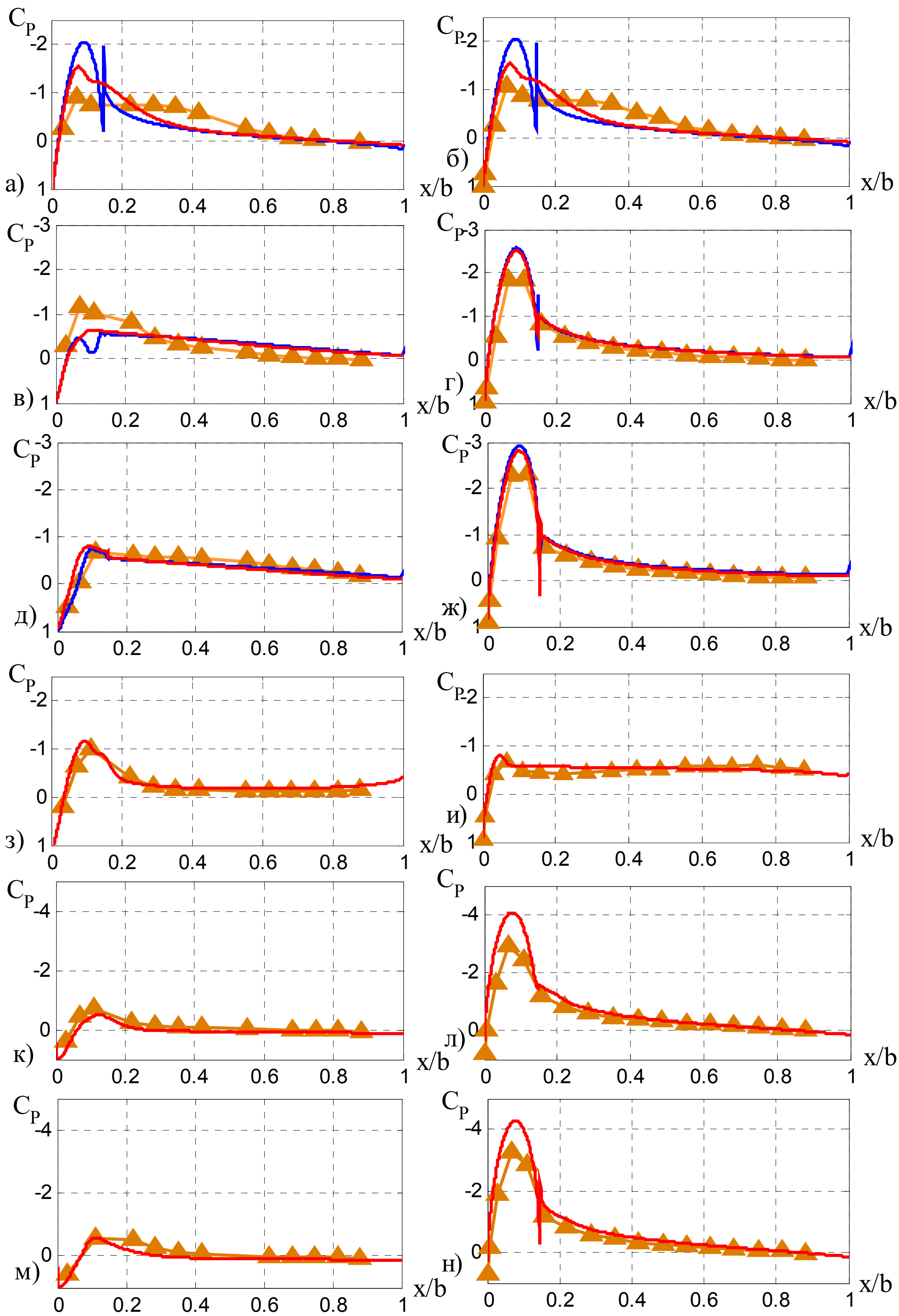

Figure 9: Comparison between CFD and experimental results, V=10 *m/s*. Orange line with triangles – experiment, red line – CFD with *k*-*ω* SST turbulence model; blue line – CFD with $k-\varepsilon$ turbulence model (left column – lower surface, right column – upper surface; upper 6 charts – $\alpha = 0°$, lower 6 charts – $\alpha = 10°$; within the groups of 6 charts for one α: first row – U=0, second row – U=1, third row – U=2)

6 Flow visualization for airfoil with rotary slat

Experimental studies included not only measurements of pressure distribution, but also flow visualization. Visualization was made by smoking wire at low airspeed, about 1 *m/s*. Fig.10 and Fig.11 present the examples of visualized “streamlines”.

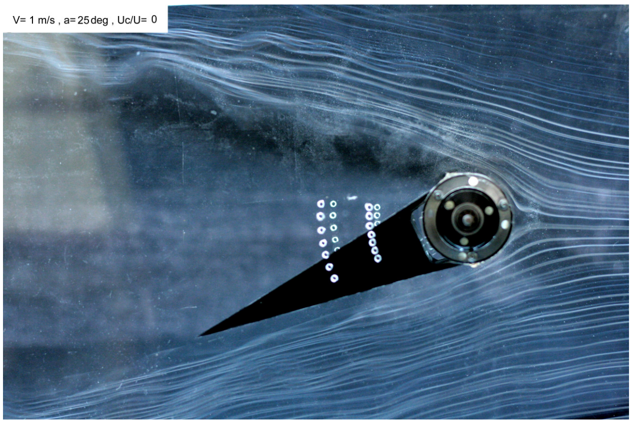

Figure 10: Cylinder is not rotating, $\alpha = 25°$

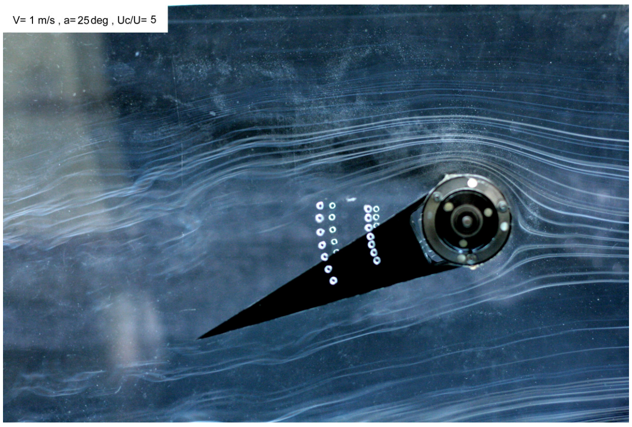

Figure 11: Cylinder rotating at $\bar{U} = 5$, $\alpha = 25°$

Comparing Fig.10 and Fig.11 one can see that rotating cylinder greatly reduces separation and stall on the upper surface.

7 Conclusion

The study has shown the possibility of CFD simulations for airfoil with rotary slat with state-of-the-art commercial CFD codes. CFD and experimental results for pressure distribution are in good agreement. It was found that increase of rotating speed can prevent separation and stall. The proposed mechanism for boundary layer control has low sensitivity to Reynolds number of the flow – twofold increase of the airspeed didn’t yield significant changes in the aerodynamic performance of the airfoil with rotary slat. On the other hand it was sensitive to the size of the radial gap between the rotating cylinder and the other part of the airfoil.

Acknowledgment

This work was supported by Federal Agency of Education within the framework of “Program for development of scientific potential of higher educational institutions” (2009-2010), work registration number 1.7.08 (State Registration № 01200803232).

References

[1] L. Prandtl, O. G. Tietjens. Applied Hydro- and Aeromechanics, McGraw-Hill, N.Y., 1934.

[2] Martynov, A. Eksperimentalnaya aerodinamika (Experimental Aerodynamics), Moscow, 1950 (in Russian).

[3] Favre, A. "Contribution a L'etude Experimentale des Mouvement Hydrodynamiques a Deux Dimensions", Thesis, University of Paris, Paris, 1938.

[4] Lachmann, G. V., Boundary Layer and Flow Control, Vols. I and II, Pergamon Press, Elmsford, NY, 1961.

[5] Schlichting, H., *Boundary Layer Theory*, McGraw-Hill, N.Y., 1968.

[6] Chang, P.K., *Separation of Flow*, Pergamon Press, Elmsford, NY, 1970.

[7] Alberto Alvarez Calderón, Rotating Cylinder Flaps for V/S.T.O.L. Aircraft, Aircraft Engineering, 1964. – P.304-309.

[8] Johnson W.S., Tennant J.S., Stamps R.E. Leading-Edge Rotating Cylinder for Boundary-Layer Control on Lifting Surfaces, Journal of Hydronautics, Vol.9, No.2, April 1975, pp.76-78.

[9] Tennant J.S., Johnson W.S., Krothapalli A. Rotating Cylinder for Circulation Control on a Airfoil, Journal of Hydronautics, Vol.10, No.3, July 1976, pp.102-105.

[10]Modi V.J., Sun J.L.C., Akutsu T., Lake P., McMillan K., Swinton P.G., Mullins D. Moving-Surface Boundary-Layer Control for Aircraft Operation at High Incidence, AIAA Atmospheric Flight Mechanics Conference, Danvers, Mass., USA, Collect. Techn. Pap. 1980, pp.515-522.

[11]Modi V.J., Sun J.L.C., Akutsu T., Lake P., McMillan K., Swinton P.G., Mullins D. Moving-Surface Boundary-Layer Control for Aircraft Operation at High Incidence, J. Aircraft, Vol.18, No. 11, 1981, pp.963-968.

[12]Mohtarian F., Modi V.J. Fluid Dynamics of Airfoils with Circulation Control for V/STOL Application, AIAA 11th Atmospheric Flight Mechanics Conference, Seattle, Washington, USA, August 1984, Paper No. AIAA-84-2090.

[13] Modi V.J., Mohtarian F. Joukowsky Airfoil with Circulation Control, AIAA 12th Atmospheric Flight Mechanics Conference, Snowmass, New York, USA, Collect. Techn. Pap. 1985", Paper No. AIAA-85-1772, pp.50-60.

[14] Mohtarian F., Modi V.J. Fluid Dynamics of Airfoils with Moving Surface Boundary-Layer Control, J. Aircraft, Vol. 25, No.2, 1988, pp.163-169.

[15] Mohtarian F., Modi V.J., Yokomizo T. Rotating air scoop as airfoil boundary-layer control, J. Aircraft, Vol. 25, No. 10, 1988, pp.973-975.

[16] Modi V.J., Mohtarian F., Fernando M.S.U.K., Yokomizo T. Moving Surface Boundary-Layer Control as Applied to Two-Dimensional Airfoils, J. Aircraft, Vol. 27, No. 1, 1990, pp.42-50.

[17] Modi V.J., Mohtarian F., Yokomizo T. Effect of Moving Surfaces on the Airfoil Boundary-Layer Control, J. Aircraft, Vol.28, No.1, 1991, pp.104-112.

[18] Modi V.J., Fernando M.S.U.K., Yokomizo T. Moving Surface Boundary-Layer Control: Studies with Bluff Bodies and Application, AIAA Journal, Vol. 29, No. 9, 1991, pp.1400-1406.

[19] Modi V.J., Munshi S.R., Bandyopadhyay G., Yokomizo T. High-Performance Airfoil with Moving Surface Boundary-Layer Control, J. Aircraft, Vol.35, No.4, 1998, pp.544-553.

[20] Al-Garni Ahmed Z., Al-Garni Abdullah M., Sahin Ahmet Z. Flow Control for an Airfoil with Leading-Edge Rotation: An Experimental Study, J. Aircraft, Vol.37, No. 4, 2000, pp.617-622.