TRANSACTIONS ON AEROSPACE RESEARCH
eISSN 2545-2835, VOL. 274, NO. 1/2024, 19-28
DOI: 10.2478/tar-2024-0002

VARIOUS BLOWING-SUCTION SCHEMES FOR MANIPULATING TURBULENT BOUNDARY LAYERS

Yevhenii Shkvar
Key Laboratory of Urban Rail Transit Intelligent Operation and Maintenance Technology & Equipment of Zhejiang Province, College of Engineering, Zhejiang Normal University, 688 Yingbin Road, Jinhua, 321005, China
Institute of Hydromechanics of the National Academy of Sciences of Ukraine, Department of Hydrobionics and Flow Control, 8/4 Marii Kapnist Street, Kyiv, 03057, Ukraine
ORCID 0000-0001-9589-5860

Shiju E
Key Laboratory of Urban Rail Transit Intelligent Operation and Maintenance Technology & Equipment of Zhejiang Province, College of Engineering, Zhejiang Normal University, 688 Yingbin Road, Jinhua, 321005, China
ORCID 0000-0002-9362-4159

Andrii Kryzhanovskyi*
Key Laboratory of Urban Rail Transit Intelligent Operation and Maintenance Technology & Equipment of Zhejiang Province, College of Engineering, Zhejiang Normal University, 688 Yingbin Road, Jinhua, 321005, China
ORCID 0009-0004-2010-0428

Dmytro Redchyts
Institute of Transport Systems and Technologies of the National Academy of Sciences of Ukraine, Department of Dynamics and Strength of New Types of Transport, 5 Pisarzhevsky Street, Dnipro, 49005, Ukraine
ORCID 0000-0001-8538-6026

ABSTRACT
The methodology of efficiency analysis of turbulent flow modification by making permeable sections in the streamlined wing surface with the aim to reduce aerodynamic drag is the principal subject of the presented research. The numerical analysis of the effect of laterally and longitudinally located permeable sections on boundary-layer properties showed the following flow features: (1) the most effective place for permeable surface is an upwind side of the wing; (2) multi-sectional blowing can simply be organised as non-uniform (especially in the case of laterally arranged permeable sections) that brings additional flexibility to change the blowing

* Corresponding Author. Email address: akrizhanovsky@gmail.com
Received 04.02.2022 Revised 21.12.2023 Accepted 08.01.2024

intensity depending on flight mode and, first of all, on angle of attack; and (3) arrays of longitudinal permeable sections allow to intensify turbulent vortical structures exchange in the lateral direction and improve flow stability to stall. Moreover, due to creating the regular anisotropy of the boundary layer in the lateral direction, this modified blowing technique can potentially have some synergistic properties, which can give the additional benefit. All these effects are too delicate and their experimental study cannot be performed with the use of directed measurements of aerodynamic forces. The comparison of the obtained flow properties with the corresponding experimental data demonstrates an appropriate level of agreement.

Keywords
combined flow control; drag reduction; blowing; suction; numerical flow modelling; RANS; experimental data analysis.

1. INTRODUCTION

Pressure and friction drag components contribute the most to the whole drag of different high-speed vehicles [1,2]. Drag reduction is one of the most popular directions of modern research in the field of fluid-mechanics, because it allows to reduce power consumption and amount of air pollutant emissions and make airplanes and other high-speed vehicles more efficient. Existing methods of friction reduction can purposefully influence a very complicated energy exchange mechanism in the near-wall vortical structure, with a wide range of scales and disturbances, localised in the wall neighbourhood. These methods are effective for streamlined bodies, elongated in the direction of flow development. Pressure drag reduction is rather associated with streamlined body shape optimisation. These two different techniques are not independent, and their interaction for some modes of streamline can be significantly non-linear. Sometimes, even quite a strong positive reduction of the friction drag component can be neutralised by its concomitant negative influence on pressure distribution and, as a result, some promising drag reduction technologies cannot be effectively implemented in engineering practice.

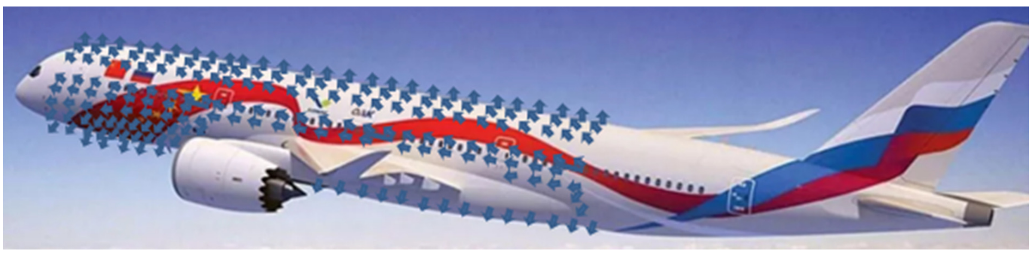

Fig. 1. The principal idea of the microblowing application to the aircraft streamlined surface.

This paper deals with the results of the research conducted by the authors and is associated with one of the most effective methods of skin friction reduction, which is the distributed exchange of mass through the streamlined surface in the form of microblowing or/and suction (Fig. 1).

This technology is not new, and its mechanism has been known for a long time. However, about 30 years ago, the method of micro-injection through huge arrays of very small holes got a new impetus in Hwang's research activity [3] and was named by him as microblowing. One of the objective reasons for this unexpected growth of interest in this promising technology is the significant progress made in the creation and fabrication of materials with the required permeability and reasonable cost. Different effective improvements of this technology and, in particular, microblowing through several permeable sections, placed laterally to the flow direction (Fig. 2A), have been studied by Kornilov and Boiko [4].

The goal of this paper is to analyse this technique's efficiency as well propose and study the multi-slot microblowing through longitudinally placed permeable sections (Fig. 2B).

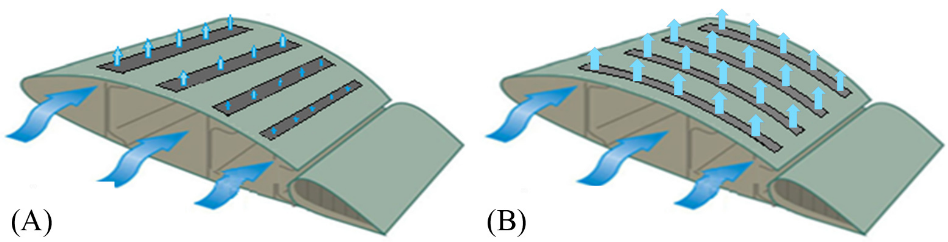

Fig. 2. Schematics of microblowing through the array of lateral slots (A) and longitudinally placed slots (B).

In addition, we propose an improved methodology for the experimentally based indirect drag coefficient estimation with the use of the wake flow parameters and by taking into account the possible presence of blowing/suction through some locally placed separable permeable sections.

2. GENERAL METHODOLOGY OF NUMERICAL MODELLING OF FLOW DEVELOPMENT WITH BLOWING/SUCTION

This section represents the results of the numerical modelling of the influence of mass-exchange through the permeable sections of a streamlined surface of a rectangular wing section on the developing flow properties. The formalised problem statement consists in the numerical prediction of the most-relevant distributed characteristics of the flow around the rectangular wing section with an NACA0012 airfoil under the following conditions (reproducing the experiments performed by Kornilov [5]): chord c = 0.5 m, span L = 0.95 m, free stream air velocity U_∞ = 21 m/s, Re = $7 \cdot 10^5$, angle of attack $\alpha = 0 - 12°$, average mass transfer speed v_n = 0 – 0.013 m/s, corresponding to the flux of secondary air flow of 0 – 500 l/min through the permeable sections, installed at x/c = 0.625 – 0.776. For simplicity, here we will consider this problem on the 2D level and replace the permeable sections with the corresponding airfoil arcs, characterised by the given non-zero v_n.

2.1. Computational domain

The computational domain ABCDEF (Fig. 3) had a C-topology, and its dimensions were determined as follows: the radius of the arc BCD and the lengths AB and DE are 6- and 8-times more than a chord c, respectively. The width EF corresponds to the span L. Inside the ABCDEF, a structured mesh was built, the detailing of which, when approaching the wing section G, was chosen based on ensuring the condition of the height of the wall cells in the wall law coordinates $y_1^+ \le 1$, which was achieved with the number of nodes 12.3×10^6 and the use of large bias factors (1,000–1,500). In addition, due to the interest in the wake flow properties and to ensure high resolution when calculating the parameters of viscous flow behind the airfoil, the mesh was refined in the vicinity of the trailing edge and behind it, as well as in the near wake area. The results of mesh verification in the process of preliminary testing showed that a further increase in the number of nodes and a decrease of the size of near-wall cells do not lead to any significant changes in the results of numerical calculations.

2.2. Governing equations

To simulate the 3D flow around a high-speed vehicle, the following system of Reynolds-averaged Navier–Stokes (RANS) governing equations has been solved under the assumptions of stationary incompressible and predominantly turbulent air flow, completely corresponding to the formulated above-mentioned working conditions and with the use of the ANSYS Fluent 17.1:

$$\begin{cases} \nabla \cdot \bar{V} \\ \left(\bar{V} \cdot \nabla\right)\bar{V} = -\nabla p \big/ \rho + \nabla \cdot \left(\nu \nabla \bar{V} + \bar{\sigma}_{t_{ij}}\right) \end{cases} \tag{1}$$

Here $\bar{V}$ – flow velocity, p – pressure, ρ – density, v – kinematic viscosity coefficient, $\bar{\sigma}_{t_{ij}} = -\overline{u_i' u_j'}$ – additional Reynolds stresses that are the result of turbulent exchange dissipative mechanism. The order of accuracy of the finite-volume discretisation was chosen as second, and due to the slowly convergent iteration process, the residuals for all computational variables were determined as $\varepsilon = 3 \times 10^{-5}$.

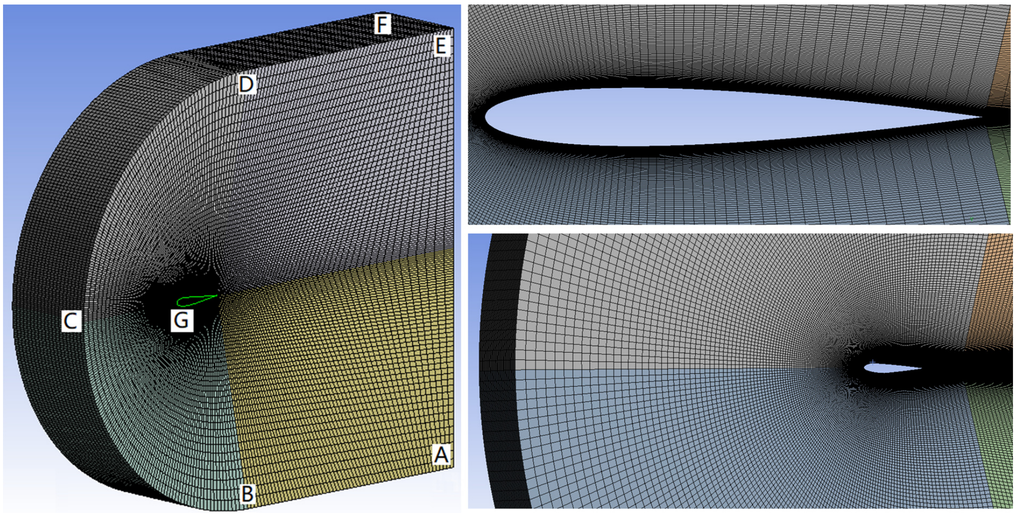

Fig. 3. Computational domain, mesh and its fragments.

2.3. Boundary conditions

The boundary conditions were established by taking into account the motion of ground surface relative to the train body according to the ANSYS Fluent formalism and given as follows: external surface of the wing section G – velocity magnitude 0 m/s (wall); inlet faces (horizontal AB, DE, forward arc BCD) – velocity magnitude V_∞ = 21 m/s in projections on the longitudinal x and normal y axes $V_x = V_\infty \cos(\alpha)$, $V_y = V_\infty \sin(\alpha)$ (velocity-inlet); lateral faces – symmetry; outlet face AEF – gauge pressure is 0 Pa (pressure-outlet).

2.4. Turbulence model

According to the Boussinesq approach, the Reynolds stresses can be directly connected with the strain rate tensor components similar to the laminar case, but by the use of additional viscosity (so-called eddy- or turbulent viscosity v_t) that reflects the dissipative mechanism of the turbulent vertical system:

$$\bar{\sigma}_{t_{ij}} = -\overline{u_i' u_j'} = v_t \bar{S}_{ij}, v_t = \mu_t / \rho. \quad (2)$$

The turbulent viscosity v_t must be modelled with the use of an additional semi-empirical model of turbulence according to the assumptions of the RANS approach. Within the framework of ANSYS software, it is impossible to completely take into account the influence of microblowing on the level of any of the turbulence models available therein, and this is the reason for adding some computational code. As it follows from the obtained results, the one-equation Spalart–Allmaras turbulence model in the strain/vorticity-based modification and with a curvature correction allows to get one of the most realistic results that is very close to the averaged value of experimentally determined airfoil drag coefficient and at the same time, this model is the fastest in comparison with other turbulent models. Due to these reasons, the Spalart–Allmaras turbulence model was taken as the basis for further computations.

2.5. Mass-exchange through the streamlined surface

The mass transfer factor through the streamlined surface was taken into account by modifying the sources values in governing equations of continuity, momentum and turbulence model using specially developed user defined functions (UDF) that determine the permeable sections' geometry and other parameters of mass-exchange through the streamlined surface. The obtained numerical solutions were additionally tested to implement the integral mass balance at the outer boundary of the computational domain and the wing streamlined surface.

3. NUMERICAL RESULTS FOR UNIFORM AND INTERMITTENT MICROBLOWING THROUGH A FLAT PLATE

The first step of verification of the proposed modelling methodology was done for the simplest flow, developing over the flat plate with the permeable insert (x_b = [1.17; 1.58]) that was studied experimentally by Kornilov and Boiko [6]. The elaborated approach to correct the turbulent viscosity distribution in the near-wall region, described in Refs. [7–10], was applied here in the frames of the developed modelling methodology. The corresponding numerical results are presented in Fig. 4 for uniform (A) and intermittent multi-slot (B) microblowing realisations.

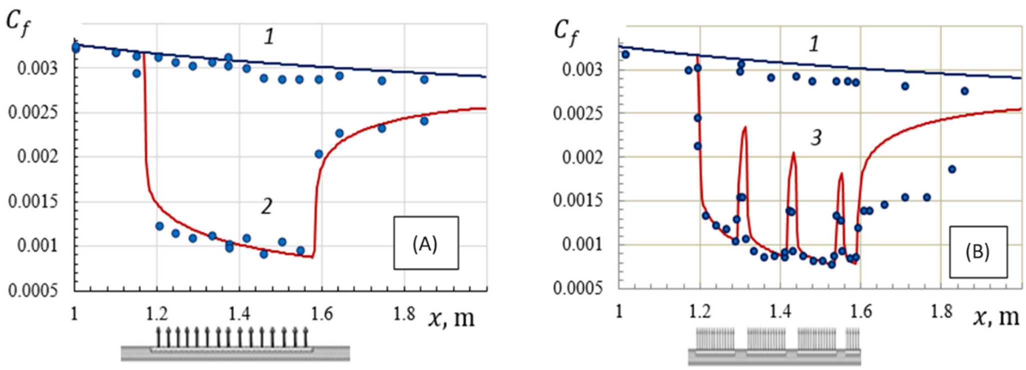

Fig. 4. Local skin friction coefficient C_f distribution along the longitudinal coordinate x of flow development around flat plate without (1) and with microblowing (2, 3): circles – Kornilov-Boiko experiments [6]; lines – Shkvar's numerical predictions. Cases (A) and (B) correspond to the uniform and intermittent microblowing, respectively, with blowing intensity $C_b = V_y / V_\infty = 0.00277$.

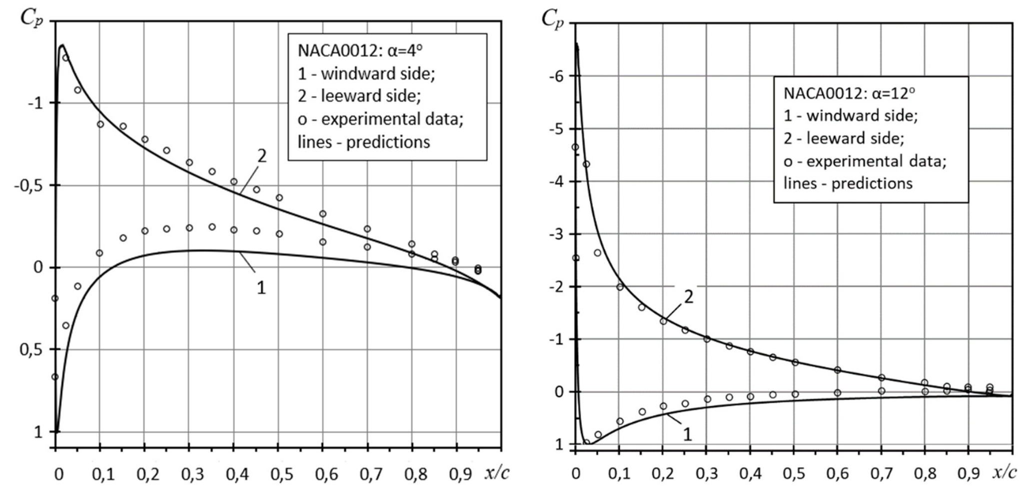

Fig. 5. The pressure coefficient distribution along the NACA0012 airfoil chord $C_p(x/c)$. For α = 4° (left) and α = 12° (right) in the reference configuration (mass transfer through the streamlined surface is absent).

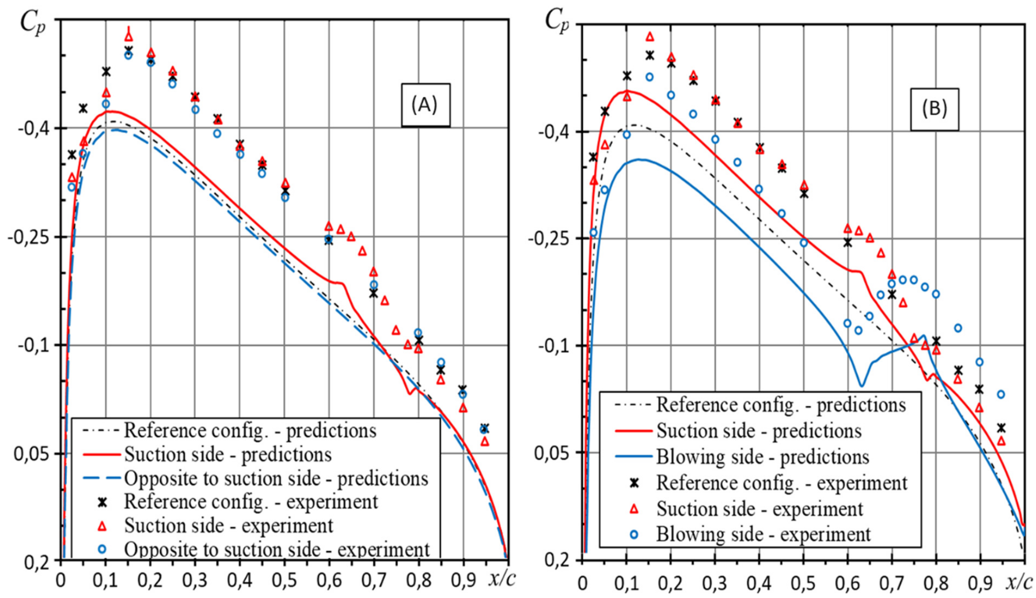

Fig. 6. The pressure coefficient distribution along the NACA0012 airfoil chord in the configuration α = 0° for suction influence through one of the airfoil sides with v_n = – 0.00687 U$_\infty$ (flux 263 l/min) – (A); and for the same suction influence, combined with blowing through the windward side with v_n = 0.013 U$_\infty$ (flux 500 l/min) – (B).

The results of $C_p(x/c)$ modelling in the reference configuration, i.e. without mass-exchange through the streamlined surface, are shown in Fig. 5 for two angles of attack (for α = 4° and α = 12°). The result of $C_p(x/c)$ modelling with suction (v_n = – 0.00687 U$_\infty$) through one of airfoil sides for α = 0° is illustrated in Fig. 6A, and $C_p(x/c)$ modelling with the same suction intensity through one of wing sides for α = 0° in combination with blowing through the opposite airfoil side (v_n = 0.013 U_∞) is presented in Fig. 6B.

As it follows from the presented comparisons with the experimental data [5], the elaborated formalised numerical procedure is able to reproduce the principal aspects of flow development behaviour in the case of laterally placed permeable slots (Fig. 2A). The systematic discrepancies between numerical predictions and experimental data (Fig. 6) can be explained by the different conditions of numerical modelling (unbounded free-stream) and experiment (influence of the walls of the wind tunnel 1 × 1 m^2 testing section and presence of turbulators near the leading edge of the wing). At the same time, the numerical prediction does not allow to simulate the slow rate of relaxation of $C_p(x/c)$ in the flow zone behind the end of blowing section like it was demonstrated in the experiments [5].

5. NUMERICAL RESULTS FOR THE LONGITUDINALLY PLACED SLOTS FOR BLOWING/SUCTION ON A WING SECTION

The next step of study of the properties of mass-exchange through the streamlined surface is associated with arrays of longitudinally arranged permeable slots (Fig. 2B). We suppose that the arrays of longitudinal permeable slots allow to intensify turbulent vortical structures exchange in the lateral direction and, as a result, improve flow stability to stall. Moreover, due to creating the regular anisotropy of the boundary layer in the lateral direction, this modified blowing technique can potentially have some synergistic properties, which can give the additional benefits. Here, we shall demonstrate only the first attempts of this idea realisation. The span of wing section was separated into n subsections with permeable slots, occupying a half of the subsection length along the lateral coordinate. The developed UDF allows to change the slot number n, slot location, length along chord and width along span as well as mass-exchange intensity. The obtained results of an uneven, but at the same time regular distribution of the isolines of the longitudinal velocity component in the vicinity of the streamlined surface of a flat plate in the transversal z direction with respect to the incoming flow in the case of 11 longitudinal slots, uniformly installed along its span are illustrated in Fig. 7. In the absence of blowing through these longitudinal slots, the isolines should be strictly parallel to the wall, whereas the presence of blowing naturally leads to local displacement of the flow and its velocity isolines above the slots in the normal to the wall direction y, which is adequately reproduced by the developed numerical model.

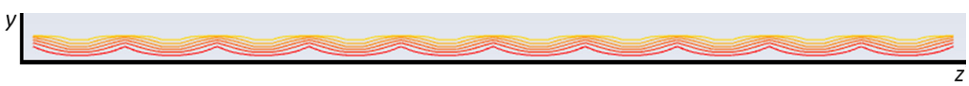

Fig. 7. TVelocity magnitude isolines in the boundary layer along wing span (*z*-coordinate).

The obtained regularity of the velocity isolines along the span of the streamlined surface (wing, etc.) creates favourable conditions for the formation of transverse inhomogeneity of the boundary layer in the near-wall region, which allows generating an appropriate regular vortex structure of longitudinal vortices with the desired frequency along span, which will actively influence the process of formation and the further development of turbulence. In addition, the three-dimensional wall structure of longitudinal vortices increases the level of spatiality of the boundary layer near the streamlined surface, which should also increase the resistance of the boundary layer to separation, which is also very important, since the blowing itself acts in the opposite way. All these factors make it possible to consider this flow control method as promising both for its stationary and non-stationary implementations, for example, for generating transverse travelling waves, which requires further research and will be the subject of further efforts application by the authors.

7. CONCLUSIONS

1. The obtained results demonstrate a quite good level of correlation between the behaviour of numerical predictions of pressure coefficient distribution along chord and the corresponding experimental data both for the reference configuration and in the cases of separate suction and combined suction-blowing.
2. Even a fairly intensive blowing from one side of airfoil (v_n = 0.013 U_∞), combined with a suction on the opposite side with twice smaller intensity (v_n = – 0.00687 U_∞), does not allow achieving the drag-reduction effect, despite a strong local drag reduction on the blowing section due to $C_p(x/c)$ redistribution and growth of pressure component of the drag coefficient.
3. Various complex combinations of blowing and suction intensity, slots orientation and placement can be promising drag-reduction techniques for different streamlined surfaces of aircraft. In addition, this technology has great potential for effective application in other high-speed vehicles such as submarines, bullet trains, etc.

ACKNOWLEDGEMENTS

This research was supported by the Ministry of Science and Technology of the People's Republic of China (Intergovernmental Chinese-Ukrainian project of scientific and technological exchanges N30815).

REFERENCES

[1] Kornilov V. Current State and Prospects of Researches on the Control of Turbulent Boundary Layer by Air Blowing. Prog Aerosp Sci. 2015;76(2015):1-23.

[2] Orellano A, Sperling S. The Aerodynamics of Heavy Vehicles II Trucks, Buses and Trains. Lecture Notes in Applied and Computational Mechanics. In: Browand F, McCallen R, editors. Aerodynamic Improvements and Associated Energy Demand Reduction of Trains [Internet]. Berlin, Heidelberg: Springer; 2009. p. 219-31. Available from: https://doi.org/10.1007/978-3-540-85070-0_19

[3] Hwang D. An Experimental Study of Turbulent Skin Friction Reduction in Supersonic Flow Using A Micro-Blowing Technique. AIAA. 2000:No. 2000-0545.

[4] Kornilov V, Boiko A. Towards Improving the Efficiency of Blowing Through A Permeable Wall and Prospects of its Use for A Flow Control. Perm National Res Polytech Univ Aerosp Eng Bull [Internet]. 2016;2016(45). Available from: https://doi.org/10.15593/2224-9982/2016.45.03

[5] Kornilov V. Steady Blowing/Suction into Turbulent Boundary Layer of a Symmetrical Airfoil Section. Sib J Phys [Internet]. 2018;13(1):33-44. Available from: https://doi.org/10.25205/2541-9447-2018-13-1-33-44

[6] Kornilov V, Boiko A. Turbulent Flow Control by Microblowing Through Microhole Perforated Wall. Proc 10th Int Symp Exp Comput Aerothermodyn Intern Flows 4 7 July 2011 Bruss Belg. 2011:ISAIF10-046.

[7] Shkvar E, Jamea A, Shi-ju E, Cai J, Kryzhanovskyi S. Effectiveness of Blowing for Improving the High-Speed Trains Aerodynamics. Thermophys Aeromechanics [Internet]. 2018 Dec 26;25(2018):675-86. Available from: https://doi.org/10.1134/S0869864318050049

[8] Shkvar E, Shi-ju E, Kryzhanovskyi A. Mathematical Modeling of Turbulent Boundary Layers, Modified by Wall-Localized Drag Reduction Techniques. Aerosp Sci Technol. 2019;93(2019):53-9.

[9] Shkvar E, Zinchenko D, Trotsenko D, Jamea A. Airplane Friction Drag Reduction by Means of Microblowing through Permeable Wing Surface Sections. Mech Gyroscopic Syst NTUU KPI [Internet]. 2016;32(2016):108-19. Available from: https://doi.org/10.20535/0203-377132201695789

[10] Kornilov V, Shkvar E. Computational and Experimental Study of the Control Efficiency of the Flow Around an Airfoil by Means of Distributed Mass Transfer. Thermophys Aeromechanics [Internet]. 2021;28(2):187-206. Available from: https://doi.org/10.1134/S0869864321020025