6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11 – 15 June 2018, Glasgow, UK

NUMERICAL AND EXPERIMENTAL RESEARCH OF NEW METHODS FOR WALL INTERFERENCE REDUCTION IN WIND TUNNELS OF TRANSONIC AND LOW SUPERSONIC VELOCITIES

SERGEY L. CHERNYSHEV[1], ALEXANDER I. IVANOV[2], EVGENY V.STRELTSOV [3] AND ANASTASIA O. VOLKOVA[4]

[1-4]Central Aerohydrodynamic Institute (TsAGI), 1 Zhukovsky St., Zhukovsky, Moscow Reg., 140180, Russian Federation, [2]ivanov_a_i@list.ru, [3]evg.streltsov@gmail.com, [4]a.volkova.mipt@gmail.com

Key words: Computational Fluid Dynamics (CFD), Wind Tunnel, Wall Interference, Boundary Layer.

Abstract. The process of designing and creating a new generation of transonic and supersonic wind tunnels is closely related to the development of efficient methods of the wall interference reduction. Along with the application of the permeable (perforated and slotted) walls, the possibilities of using alternative types of boundaries are investigated. One of the most promising approaches is generation of the homogeneous jet-boundary conditions on the solid walls of the wind tunnel test section using the near-wall jets or controlled boundary layer. Numerical simulation of a flow around the models with various test section boundaries has made it possible to substantially reduce the volume of the wind tunnel tests and to select optimal parameters of the boundary condition.

1 INTRODUCTION

The problem of the wall interference reduction is primarily the boundary problem, and it remains the topical one in the wind tunnel testing technique, especially at high subsonic and transonic flow velocities. At these regimes, the use of solid walls is unacceptable due to the phenomenon of the test section blockage, which is connected with the fact that in the narrowest section of flow, usually at the location of the model, the sonic velocity is achieved. Permeable walls can remove this disadvantage, allowing extra gas to penetrate through the walls to the plenum chamber and then return back. From the interference point of view, perforated and slotted walls demonstrate different behaviour and both have their own advantages and drawbacks. Slotted walls generate large-scale vortex disturbances and form rather thick near-wall layer with non-uniform flow, and make the formulation of the uniform boundary condition very problematic. Classic boundary condition on the perforated walls does not allow remove all types of wall interference.

One of the most promising approaches for the wall interference reduction is the formation of homogeneous boundary conditions on the solid walls of the wind tunnel test section by arranging the near-wall jets [1]. The relevant character of jet boundaries can be illustrated by the fact that an unbounded flow field over a model can also be represented as a flow in boundaries formed by jets with an infinite thickness and the dynamic pressure equal to that in the main flow. The use of the optimization algorithm for jets parameters made it possible to practically eliminate the influence of boundaries. However due to technological difficulties in

the implementation of jet boundaries, this type of boundary conditions has not been widely used. To simplify the experimental procedure, the effective approach based on the boundary layer control on the smooth solid walls of the test section was proposed [1]. The controlled boundary layer proved to be rather simple modification of jet boundaries. Standard models testing at subsonic and transonic velocities demonstrated the ability of new wall boundaries to eliminate the wall interference. In this case, the balance characteristics of the model should be the same as in the unbounded flow conditions. However sometimes the tested models are oversized and the acceptable thickness of the boundary layer is insufficient to remove the interference perturbations. To overcome this difficulty, the further development of this method was elaborated. Some new tests showed that the combination of a controlled boundary layer and perforated walls of the wind tunnel test section allows eliminating to a considerable extent the shortcomings of both perforated and jet boundaries.

Wall interference problem for low supersonic regimes has some specific features. To eliminate the wall influence usually means to damp the reflected wave disturbances. New experimental and CFD approaches also can help to solve this problem.

Nowadays a very high level of CFD capabilities has been achieved. Thus numerical modelling of flow around 2D and 3D models in different boundaries allows substantially diminish the necessary volume of experimental studies. Within the framework of this paper, the results obtained during experimental research in the TsAGI transonic wind tunnel will be considered, as well as numerical simulation using the ANSYS CFX and the Electronic Wind Tunnel (EWT-TsAGI) software [2].

2 FLOW SOLVER

A modern, integrated approach to an aerophysical research includes the wind tunnel testing and numerical simulation. Nowadays numerical investigation becomes indispensable part of the research because of its informativity and the ability to simulate a wide range of tasks. Numerical methods are especially relevant due to the possibility of modeling the wind tunnel experiment that is extremely important in confirming the effectiveness of promising techniques in experimental aerodynamics. However, it should not be forgotten that the results of numerical calculations require validation and verification.

In this research, we aimed to confirm the effectiveness of the controlled boundary layer concept for wall interference reduction and to optimize boundary layer parameters using well-known ANSYS CFX and EWT-TsAGI codes. Moreover, the “unbounded-flow” characteristics of the tested models were also obtained and compared with the wind tunnel results.

The flow solver used in the present work is the EWT-TsAGI code that has been in use at TsAGI since 1996. The method is well described in Ref. [3, 4]. It permits to solve the stationary (RANS) and non-stationary (URANS) Navier-Stokes equations using Reynolds-averaging. All the results presented here are based on RANS approach. The implicit smoother method is developed to accelerate the Godunov-type TVD scheme for approximation of convective fluxes (MUSCL) [5, 6]. The method used is based on the delayed correction procedure [7] and applies the Gauss–Seidel block method with the cell renumbering to solve the system of linear equations. For the approximation of SST turbulence model [8] source terms, the analysis of eigenvalues of the Jacobi matrix is used. The local choice of an explicit or implicit scheme and the manner of time step implementation (global or local), depending on the relationship between

the specified global time step and the local stability condition of the explicit scheme is the main specific feature of the numerical method.

3 NACA 0012 AIRFOIL

Flow around the NACA 0012 symmetrical airfoil is a classic two-dimensional test case. Flow simulation parameters coincide with the experimental parameters [9] (M=0.7, α=1.49°, chord length c=1 m, Re=$9\cdot10^6$). At the first stage, simulations were performed in "unbounded" flow on structured grids with 16640, 66560 and 2662400 cells. The distance from the airfoil surface to the outer boundary of the computational domain for all grids was 20 chords. Grids were concentrated near the leading and trailing edges and along the normal to airfoil surface. Based on preliminary calculations, at least 32 cells were placed into the boundary layer; the spacing near the wall was 1E-6, which is enough to guarantee y+ < 1 (Fig. 1).

Calculated C_P distributions are in good agreement (Fig. 2). The slight difference can be seen only in the region of shock wave location - on the coarse grid, the shock wave is "blurred". CFD predictions are also in good agreement with the experimental data (Fig. 2) and calculations of other authors [10] (Table 1). At the second stage of the numerical investigation, influence of solid walls and solid walls with the controlled boundary layer was estimated using NACA 0012 airfoil. For this purpose test section of transonic wind tunnel (cross-section blockage of NACA 0012 airfoil was 4.08%) was simulated with boundary layer on top and bottom walls. In the first case, boundary layer grew naturally, and in the second case, its integral parameters were significantly increased by setting longitudinal velocity profile at the entrance to the computational domain.

The obvious result of simulation in solid walls was a significant distortion of flow around airfoil, which resulted in considerable change in pressure distribution (Fig. 3). This result illustrates well influence of solid walls of test section at transonic speeds. To simulate controlled boundary layer at the test section entrance, we set the profile of longitudinal velocity component:

$$u(y) = tanh\left(\left(y+\frac{h}{2}\right)\cdot k\right)\cdot\left(-tanh\left(y+\frac{h}{2}\right)\cdot k - h\cdot k\right)\cdot u_\infty \qquad (1)$$

where h is the test section height, u_∞ is the free stream velocity and k is the dimensionless coefficient that determines velocity profile near the wall.

A set of calculations with different velocity profiles at the test section entrance was performed. The least level of wall interference was obtained by setting the velocity profile with k=12 ratio. Simulation of the controlled boundary layer by setting the velocity profile allowed eliminating practically the wall influence. C_P distributions are in a good agreement with "unbounded" flow results (Fig. 3) even though presence of a solid wall. The values of C_D and C_L coefficients are also close to the values obtained in experiment [9] and simulations in "unbounded" flow (Table 1).

Table 1: Comparison of C_D and C_L coefficients. Experimental data and CFD ("unbounded" flow and solid walls with controlled boundary layer)

Research	C_D	C_L
Experiment (T.L. Holst, [9])	0.00830	0.2540

Bosniakov et al., [10] (Grid 640000 elements)	0.00836	0.2407
Current research («unbounded» flow)	0.00850	0.2366
Current research (solid walls with controlled boundary layer *k*=12)	0.00825	0.2581

For the best result with the controlled boundary layer (*k*=12) on a distance of one chord in front of the airfoil, integral parameters of the boundary layer were determined (Fig. 4). Displacement thickness was 39.8 mm (2.65% of the test section half-height).

4 WIND TUNNEL TEST & EXPERIMENTAL RESULTS

As part of the development of the concept of controlled boundary layer, a series of experiments to assess its operability and effectiveness was carried out. Test facility chosen for these experiments was the transonic TsAGI T-112 wind tunnel that was previously often used for solution of the test technique problems and as a pilot facility for the new concepts implementation.

Boundary layer control on the top and bottom wall was realized with the help of the wedge-formed spoiler grids mounted at the test section entrance and aimed at creating the additional drag in the near-wall region. The spoiler height might be changed manually from 0 to 32 mm. In the model location zone the boundary layer parameters were measured using the Pitot tubes rake. Boundary layer characteristics were changed in wide range. In particular, the displacement thickness varied from 6 mm to 36 mm, i.e. from 2% to 12% of the test section half-height.

In the experiments, the geometrically-similar schematized models of civil aircraft (without empennage) were tested at Mach number 0.8. Their dimensions are given in Table 2. Testing of the geometrically-similar models is the well-known way to estimate the boundary interference. It is considered that the proper choice of boundaries can lead to coincidence of the aerodynamic characteristics for all such models. Although this method is debatable, it helps when there is no more reliable criterion, such as, e.g., the unbounded-flow results. Half model is shown in Fig. 5.

All models were tested in different boundaries: in solid walls and in the 2-side perforation. In order to reduce the wall interference, a combination of the jet-perforated boundaries (perforated walls and controlled boundary layer) was used.

Table 2:Dimensions of the transport airplane schematized models

Model	Length [m]	Wingspan[m]	Wing area [m^2]	b_a [m]	Wing area/ Test section area
8-203-3	0,5681	0,495	0,035	0,0698	0.097
8-203-4	0,4584	0,4	0,0228	0,0564	0.063

Data analysis of application of combined jet-perforated boundaries during the tests of schematized models of civil aircraft were quite encouraging. Compared with the solid-wall data (Fig. 6) and the results, obtained in standard perforated walls (Fig. 7), much better coincidence of aerodynamic curves was obtained in the combined boundaries (Fig 8). It should be noted that

both lift coefficient curves and pitch moment curves demonstrate good agreement even in the present tests with the evidently oversized models.

5 NUMERICAL FLOW SIMULATION

The only real confirmation of the boundary interference approach correctness is a direct comparison of the results obtained in the wind tunnel with the model characteristics in the free flight. These “unbounded-flow” characteristics may be obtained in the wind tunnel with larger dimensions, as it was done in [11], when T-128 results for the transonic reference model (blockage 0.37%) were verified in the AEDC 16T facility (blockage 0.07%). In the case of models with relatively simple geometry the unbounded-flow characteristics may be also obtained numerically. The flow around reference model 12299-2 was simulated in a large computational domain (20×20×20 m^3) at different angles of attack (0°, 4°, 8°, 12°, 14°, 15°, 16°) at Mach number M = 0.8 on the 16 million cells mesh using the turbulence model SST.

Comparison of the "unbounded" curves with the experimental results (Fig. 9) shows that combined jet-perforated boundaries may be considered as the most effective variant of low-interference test section walls arrangement.

6 REDUCTION OF SHOCK WAVE REFLECTION

One of the most important methodological problems is the reduction of the wave perturbations influences on test model characteristics in the wind tunnel. This problem has appeared in the experiments at low supersonic velocities ($1<M<1.5$). The main source of the flow field perturbations is the model itself generating shock waves and rarefaction waves. These wave perturbations interact with flow boundaries at small supersonic velocities and reflect to the tested model. Due to short extension of the characteristic rhomb these wave perturbations distort the flow field near the model rear part. As a result, significant errors occur in measuring the balance characteristics of the model, especially its moment characteristics.

At low supersonic velocities, the elimination of boundary influence consists, mainly, in damping of shock waves and rarefaction waves. Well-known way to solve this problem is the application of perforated-wall test section of the wind tunnel. Investigations in this field have shown that the use of the optimal permeability of perforated walls makes it possible to minimize reflected perturbations at low supersonic velocities [12]. It should be noted that disturbances reflecting from the solid wall have the same sign (shock waves reflect as shock waves, rarefaction waves as rarefaction waves), and the reflection from free boundary (perforation holes) changes the sign to the opposite one. So the reflected disturbances weaken each other and diminish the interference effect. Successful choice of the open-area ratio substantially improves the flow field near a model. However, the discrete character of boundary condition can influence the final result. Some authors also outline that the use of a constant perforation can degrade the interference properties of the wind tunnel walls [13]. The development of new approaches to this problem may contribute to the test technique improvement.

To study the interaction of perturbations generated by a model in the facility with solid walls, as well as with a controlled boundary layer, numerical simulation of the 2D flow around a thin cone-cylindrical model was carried out. The cone generatrix angle of the model was 4°. Dimensions of the square cross-section were 600×600 mm and a length of the simulated test section was 1100 mm; test section blockage was about 1%. In order to save the calculation time,

bearing in mind the symmetry of the calculation area, only half of the test section with the model was considered. The calculation was carried out using the Electronic Wind Tunnel (EWT TsAGI package) and ANSYS CFX software.

To analyze the efficiency of the controlled boundary layer concept to solution of the problem of reflected perturbation waves, we considered different coefficients of the boundary layer thickness, k=10, 20, 30, 40. The table 3 shows the correspondence between the coefficient k from equation (1), the displacement thickness δ^* and the momentum thickness θ of the boundary layer.

Table 3: Matching of the boundary layer thickness coefficient and integral characteristics

k	δ^*	θ
10	0.069391	0.030584
20	0.034824	0.015176
30	0.023356	0.009978
40	0.017664	0.007336

In order to determine the optimal coefficient k, we also calculated the flow around the tested model with an unbounded flow, since such flow can be considered a reference for solving the problem of boundary interference. And also for comparison, calculations were carried out in solid walls on which the boundary layer increased according to the traditional law.

The results obtained comparing the distribution of the Mach number in the center of the calculated region, as well as the pressure distribution over the surface of the cylinder, showed that the optimal coefficient of thickness of the boundary layer on a rigid wall is k = 30. (Fig. 10)

Eventually, application of numerical methods demonstrated that the use of a controlled boundary layer in the case of a two-dimensional problem makes it possible to reduce greatly the level of perturbations reflected from the wind tunnel walls. Thus, the distortion of the model aerodynamic characteristics measured in the wind tunnel test will also be significantly reduced.

7 CONCLUSIONS

- Numerical simulation of flow around the models with various test section boundaries, such as solid wall with controlled boundary layer has made it possible to substantially reduce the volume of the wind tunnel tests and to select optimal parameters of the boundary condition for subsonic and transonic flow regimes.
- A numerical investigation has shown that the application of a controlled boundary layer can significantly reduce wall interference and the level of the reflected wave disturbances. Thus, it can improve the accuracy and reliability of the wind tunnel test results at low supersonic velocities.

ACKNOWLEDGEMENT

The reported study was funded by RFBR according to the research project № 18-38-00735.

REFERENCES

[1] Ivanov A.I., Streltsov E.V. Controlled boundary layer on the solid walls of wind tunnels: new approach to the boundary interference problem // International council of aeronautical sciences congress. 2014.

[2] Neyland V., Bosniakov S., Glazkov S., Ivanov A., Matyash S., Mikhailov S., Vlasenko V. Conception of electronic wind tunnel and first results of its implementation // Progress in Aerospace Sciences. 2001.

[3] Kazhan E.V., Stability improvement of Godunov–Kolgan–Rodionov TVD scheme by a local implicit smoother. TsAGI Sci. J. 2012.

[4] Bosnyakov S, Kursakov I, Lysenkov A, Matyash S, Mikhailov S, Vlasenko V, Quest J. Computational tools for supporting the testing of civil aircraft configurations in wind tunnels. Prog. Aerosp. Sci 2008.

[5] Godunov S. K. A Finite Difference Method for the Computation of Discontinuous Solutions of the Equations of Fluid Dynamics. Mat. Sb. 1959.

[6] van Leer B. Towards the ultimate conservative difference scheme. V. A second-order sequel to Godunov's method. Journal of Computational Physics. 1979.

[7] Ivanov M.Ya., Krupa V.G., Nigmatullin R.Z. Implicit scheme of S. K. Godunov of increased accuracy for numerical integration of Euler equations. Zhur. vych. I Mat. i Mat. Fiz. 1989.

[8] Menter F.R., Improved Two-Equation k-omega Turbulence Models for Aerodynamic Flows. NASA TM 103975. 1992.

[9] Holst T.L. Journal of Aircraft 25(12). 1988.

[10] S.M. Bosnyakov, V.V. Vlasenko, M.F. Engulatova, E.V. Kazhan, S.V. Matyash, A.I. Troshin. Proceedings of TsAGI (2735). 2014.

[11] Glazkov S.A., Gorbushin A.R., Ivanov A.I. and Semenov A.V. Recent experience in improving the accuracy of wall interference corrections in TsAGI T-128 wind tunnel. Progress in Aerospace Sciences. 2001.

[12] Neyland, V.M., Asymptotic transonic theory and optimal permeability of walls of the wind tunnel at M≥1 //TsAGI's Scientific Notes.1990.

[13] Renato Paciorri, Filippo Sabetta, Aldo Rona. Wave reflection on porous walls: numerical modelling and application to transonic wind tunnels // 40th American Institute Aeronautics and Astronautics Aerospace Sciences Meeting and Exhibit. 2002

[14] Collins D J and Krupp J A. Experimental and theoretical investigation in two-dimensional transonic flow// American Institute of Aeronautics and Astronautics Journal. 1974.

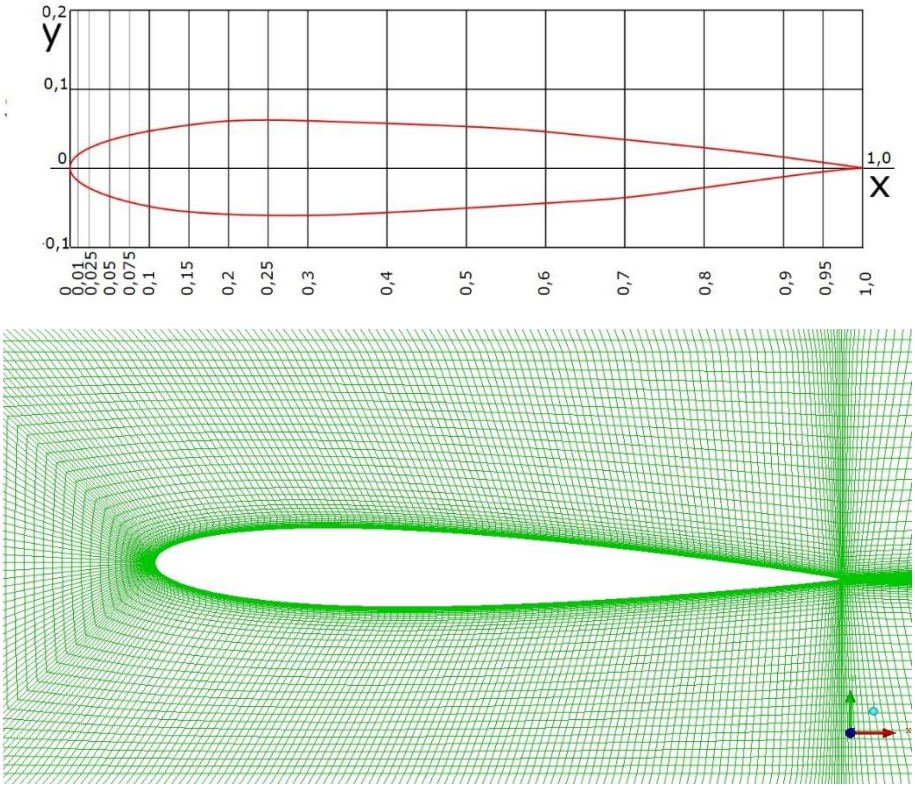

Figure 1: NACA 0012 geometry and computational grid

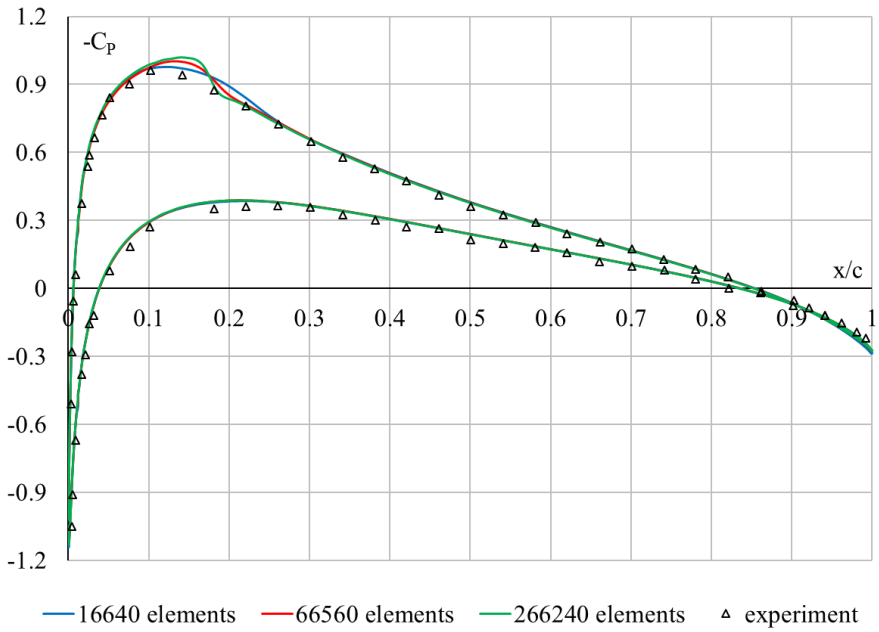

Figure 2: Comparison of experimental and predicted $-C_P(x/c)$ curves, NACA 0012 airfoil, M=0.7, α=1.49°

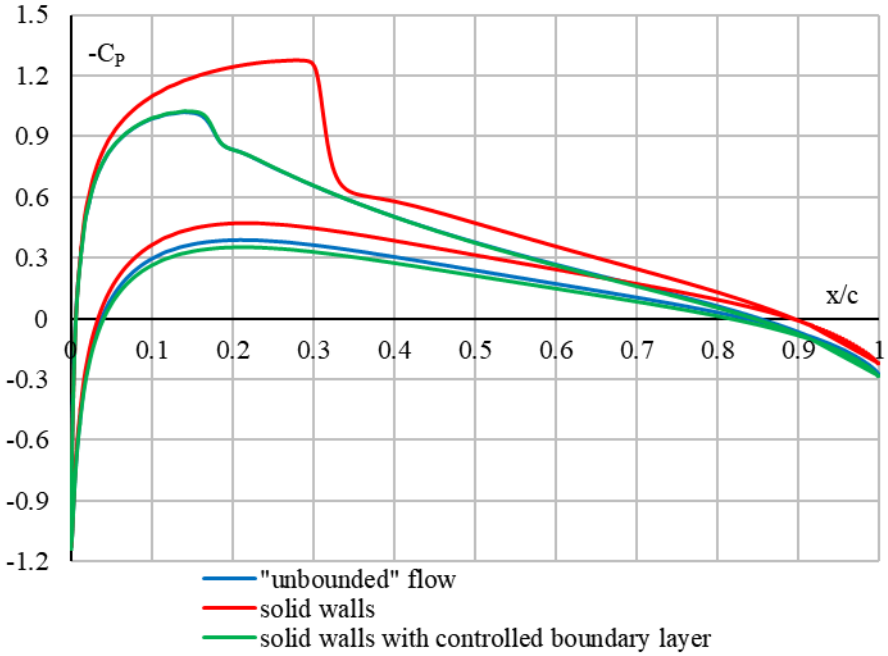

Figure 3: NACA 0012 $-C_P(x/c)$ distribution in different boundaries, M=0.7, α=1.49°

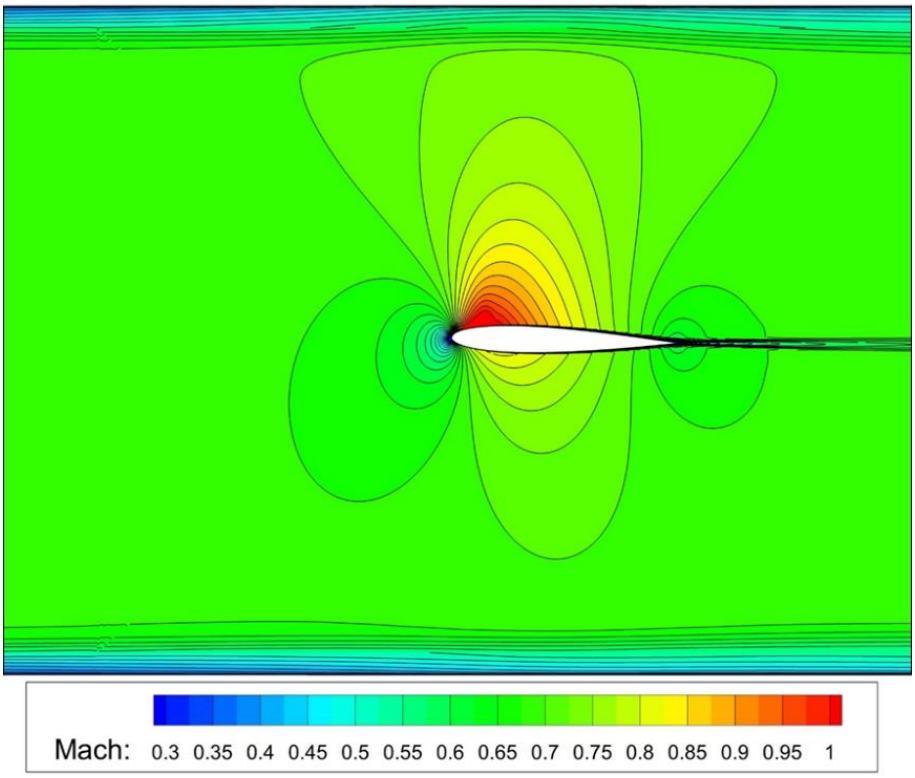

Figure 4: Simulation of NACA 0012 airfoil with the controlled boundary layer, using EWT-TsAGI code, M=0.7, α=1.49°

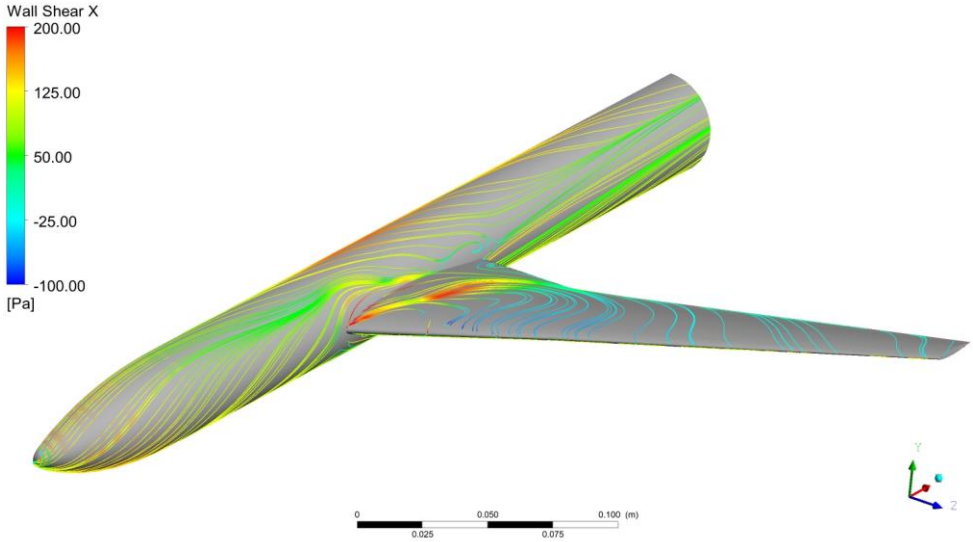

Figure 5: Surface streamlines at geometrically similar schematized model of civil aircraft, M=0.806 α=12.202°

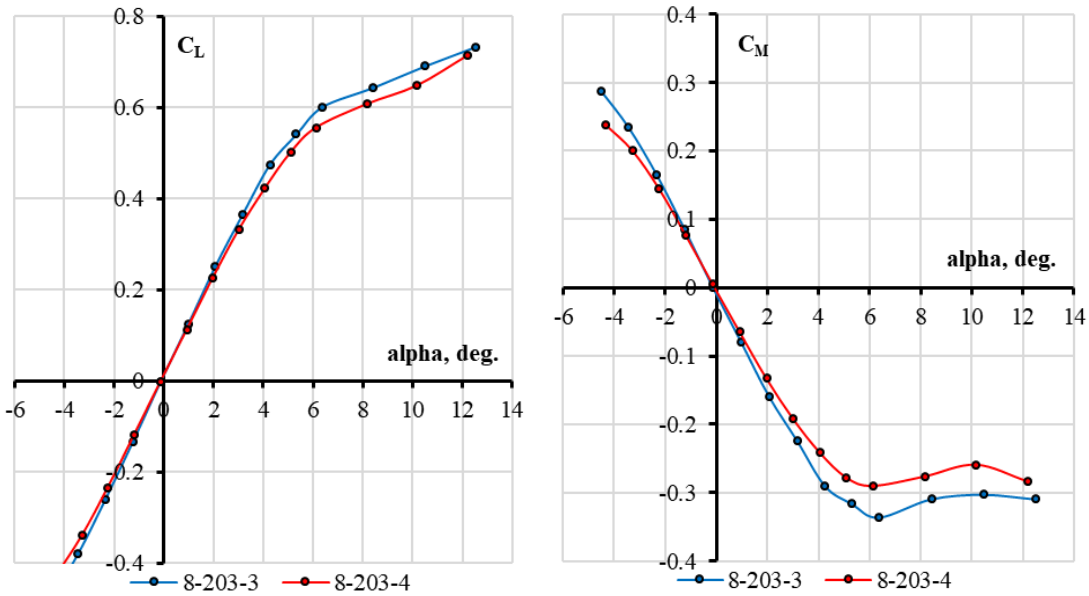

Figure 6: $C_L(\alpha)$ and $C_M(\alpha)$ curves for tests in solid walls. Geometrically-similar schematized models of civil aircraft. M=0.8

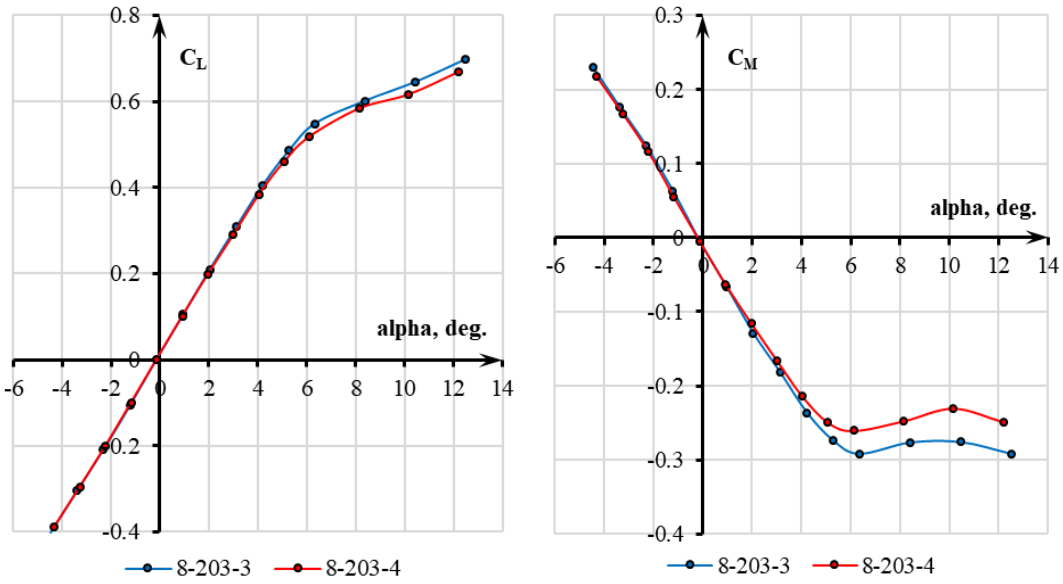

Figure 7: $C_L(\alpha)$ and $C_M(\alpha)$ curves for tests in perforated boundaries. Geometrically-similar schematized models of civil aircraft. M=0.8

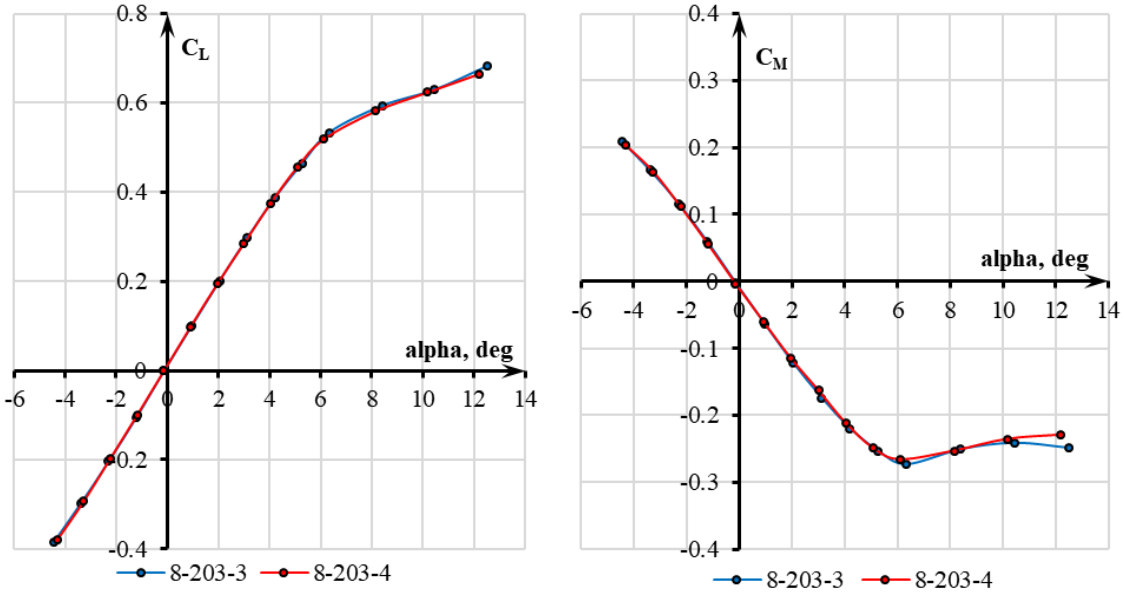

Figure 8: $C_L(\alpha)$ and $C_M(\alpha)$ curves for tests in jet-perforated boundaries. Geometrically-similar schematized models of civil aircraft. M=0.8

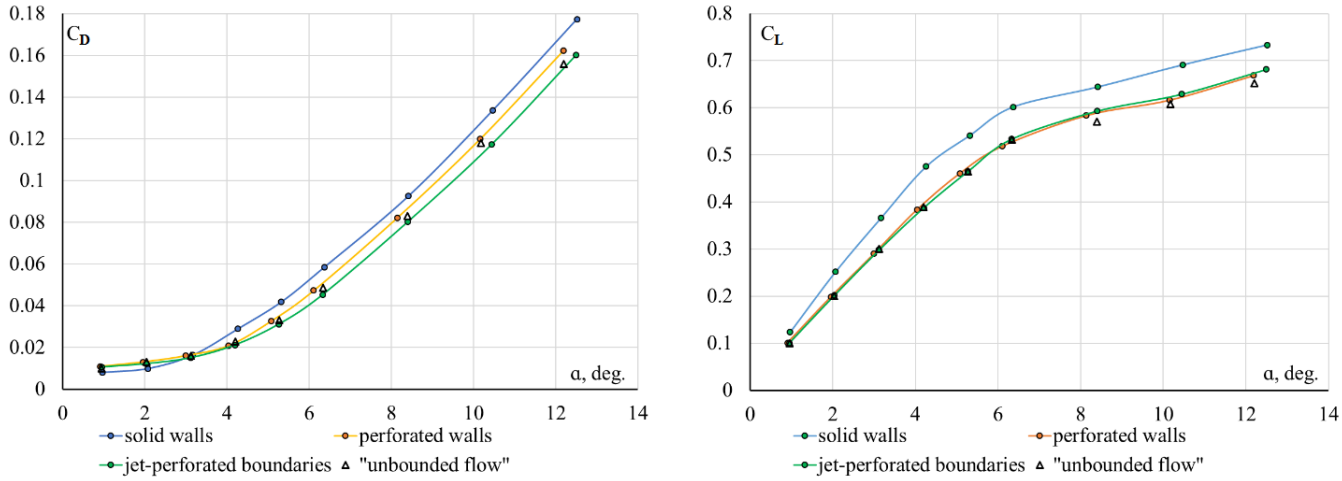

Figure 9: $C_D(\alpha)$ and $C_L(\alpha)$ curves for tests in solid walls, perforated walls, jet-perforated boundaries and "unbounded flow". Geometrically-similar schematized models of civil aircraft. M=0.8

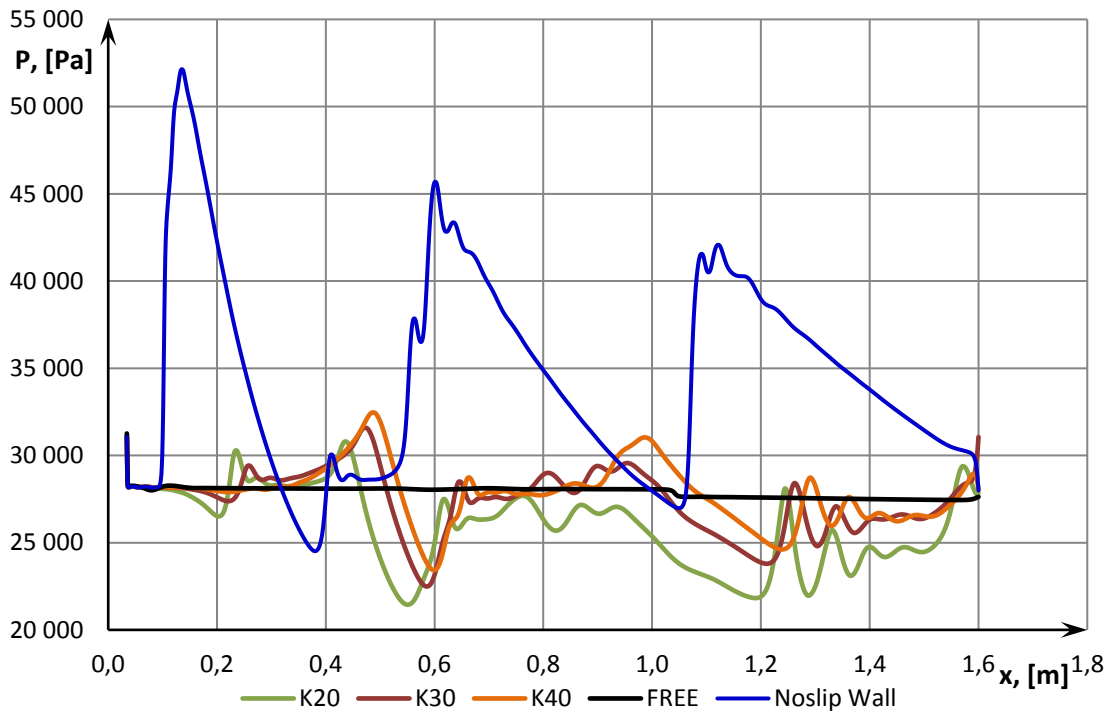

Figure 10: Pressure distribution along cylinder at Mach number 1.5