Numerical simulation of supersonic flow around a dimpled relief

S.A. Isaev[1], **A.I. Leontiev**[2], **A.G. Sudakov**[1] *and* **P.A. Baranov**[1]

[1] *Saint-Petersburg State University of Civil Aviation, 38 Pilotov Str., Saint-Petersburg, 196210, Russia*
[2] *Baumann Moscow State Technical University, 5 Second Baumann Str., Moscow, 107105, Russia*

Introduction. Over many decades systematic studies on numerical simulation of enhancement of tornado-like heat and mass transfer in flow around convex-shaped surfaces [1,2] have been carried out Up to now it is still actual to analyze the physical mechanism for enhancement of heat transfer from a dimpled surface at a substantially less growth of hydraulic losses. The thing is that controlling processes occur in a narrow wall layer, as a rule, of small sizes, which hampers their experimental investigation. In addition, the considered problem is many-parametric. Besides such initial geometrical sizes as relative values of depth, edge rounding, longitudinal and transverse pitch (in the case of spherical dimples), aerodynamic characteristics and heat transfer are essentially affected by a relative thickness of a boundary layer and a turbulence degree of an incoming flow [3,4].

Based on the developed multiblock-based computational technologies (MCT) (packet VP2/3), numerical simulation of convective heat transfer in a single dimple shows that self-generation of tornado-like vortex structures cardinally depends on their relative depth. For dimples with the ratio of depth to diameter of the order of 0.2 the regime of flow past them occurs spasmodically, i.e. a transition of a two-cell vortex structure in a dimple to a monovortex one is realized. The above change in the flow regime is accompanied by the same spasmodic increase in heat transfer from the dimple to the wake zone behind it [4,5].

In the course of physical and numerical simulation of tornado-like heat transfer in a narrow channel, when one of its walls is provided with a set of relatively deep spherical dimples [6,7], the effect of synchronization of vortex structures is revealed in the dimples. It is shown that near the wall the ordered vortex layer is formed, which is responsible for heat transfer enhancement. As compared to a smooth surface, heat transfer from the wall increases approximately 2.4 times, while the hydraulic resistance – only 1.4 time.

The study of the effect of the depth of concavities on drag and heat transfer [8] has shown that the situation is opposite for small depths of groove and dimple. It appears that at small subsonic speeds of the wall flow it is possible to reduce a drag of a dimpled-made relief and to decrease heat transfer from the wall. This idea is supported experimentally in [9]. It is shown that making dimples over the surface of the model of a high-speed train provides a 15% reduction of its total drag. Computational studies involving numerical simulation of incompressible fluid flow around grooved and dimpled relieves [10] as a whole have supported a tendency for a drag reduction. However, the quantitative predictions of drag reduction have appeared to be somewhat more conservative (of the order of 1-4%). Also, it has been noted [8] that in the wake behind the groove heat transfer decreases more than by 10% at the depths of the order of 0.07.

In succeeding years the Institute of Mechanics at Moscow State University is making experimental investigations of the cooling effect of a curvilinear surface on a wall temperature in super-and hypersonic flow using for this purpose heat images. This work develops this research trend in the way of making numerical experiment with regard to the accumulated experience in simulating heat transfer in the vicinity of wall concavities [10].

Analysis of tornado-like heat transfer in wall flows. Numerical study of super-and hypersonic viscous gas flow around a curvilinear wall is based on the multoblock-based methodology developed for calculation of separated flows of incompressible viscous fluid [11]. It should be noted that being its basis, the concept of splitting into physical processes that has been realized in pressure correction has no widespread use for solving compressible fluid problems. However, from the standpoint of keeping a unified approach to solve various problems and also of using wide experience of calculation of separated flows of incompressible fluid it is of interest to implement a generalized approach to solve the equations of hydromechanics and energy, including the recommendations of the monograph [12]. Besides the methods and research, the present work covers a verification block containing the calculation results of test problems having experimental analogs.

Specific features of a multiblock-based computational algorithm. MCTs realized in the packet VP2/3 (velocity-pressure) as a methodical basis for numerical simulation of convective heat and mass transfer processes are the result of the works of the last years since 1995 when there appeared the necessity to realize calculations on different-scale structured grids with their overlapping [11]. The use of the obtained results has allowed one to correctly solve problems on fluid motion in multi-connected regions and to adopt the advanced low Reynolds turbulence models of the type of Menter's model for shear stress transfer (MSST). These results are also generalized to the solving of 3 D problems and the using of sliding grids.

The computational technology has been developed between 1983 and 2004. Solving the Navier-Stokes and Reynolds equations closed in the last case by the semi-empirical differential model of shear stress transfer (Menter, 1993) is made using a finite-volume implicit procedure (predictor-corrector) of SIMPLEC type with elements for computation control. This procedure is based on the concept of splitting into physical processes. The writing of the original equations in increments of dependent variables, different-order schemes for approximation of RHS and LHS, upwind differences and additional damping in the implicit part, Leonard's scheme QUICK for the convective terms of the explicit part, Rchi-Chou's approach for interpretation of a relationship between pressure and velocity, the method of incomplete matrix factorization – all the above elements – are the characteristic features of the developed methods. The multoblock grids consisting of fragments of orthogonal grids of simple topology are distinguished by computational and connected cells. In the first cells the original equations are being solved, while in the second cells the parameters are determined using linear interpolation.

In the present work special attention is paid to constructing a SIMPLEC version for compressible medium. Also, from the calculations of non-viscous flows with shock waves it is found that from the standpoints of the stability of the computational process it is advisable to approximate the convective terms of the Reynolds equations using Leonard's scheme and the remaining equations – Van Leer's scheme. It is important to emphasize that the generalized approach uses MSST – Menter's turbulence model well approved for calculation of separated flows of incompressible fluid [11]. It should be noted that the attempts to develop such an approach were made earlier [13], but they were as a rule connected with the limitations on the range of determining parameters or following these limitations the obsolete turbulence models were considered.

Testing. Two considered 2D problems fall into the category of generally accepted testing ones: a) interaction of a generated oblique shock wave with a turbulent boundary layer on the flat wall; b) compression angle or super-and hypersonic flow around a wedge on the flat wall.. The results of numerical prediction are compared with Horstman's experimental measurements of dynamic and thermal characteristics on the streamlined wall [14]. A solution to the first problem is the computational domain, whose upper boundary is the symmetry plane spreading up to the wedge, the wedge-shaped section with an inclination angle of 10^o and the section of the flat wall parallel to the symmetry plane up to the outlet boundary. The lower boundary of the domain – flat wall parallel to the upper wall. A vertical size of the outlet boundary is chosen as a characteristic linear scale. A distance from the inlet boundary up to the wedge is assigned equal to 10, and an entire length of the domain is 19.67. A vertical size of the inlet cross-section – 2. Parameters of a uniform flow with Mach and Reynolds numbers 8.2 и 10^6.are predetermined at the inlet. A temperature factor T_w/T_o on the lower wall is chosen equal to 0.27. The computational domain is covered with an oblique grid containing 199×146 cells. A minimal step near the wall is 10^{-4}.

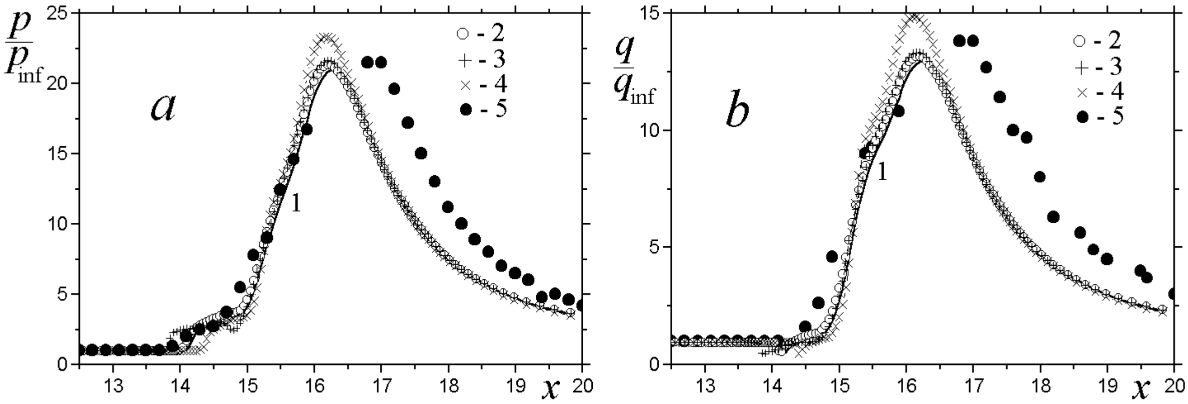

Figure 1. Comparison of the distributions of the relative values of pressure (*a*) and heat fluxes (*b*) in the zone of interaction of a shock wave with a turbulent boundary layer using different temperature factors, discretization schemes and modifications of the turbulence model. 1– T_w/T_o=0.27 on the upper wall; 2 – upper wall is heat-insulated; 3 – as in the first case but with regard to a correction for the streamline curvature; 4 – as in the first case but with the use of the approximation scheme SuperBEE. 5 – [14].

The results of a comparative analysis of the numerical and physical modeling plotted in Figure 1 show that as a whole the computations quite fairly describe the behavior of the characteristics in the flow separation zone in front of an incident shock wave. The agreement of the experimental and calculated data on maximum force and heat loads on the wall is also quite reasonable. Some disagreement with numerical predictions can be attributed to the uncertainty of geometrical sizes of the experimental setup [14] This causes a difference in the initial boundary layer thickness and also in the values of the Reynolds number.

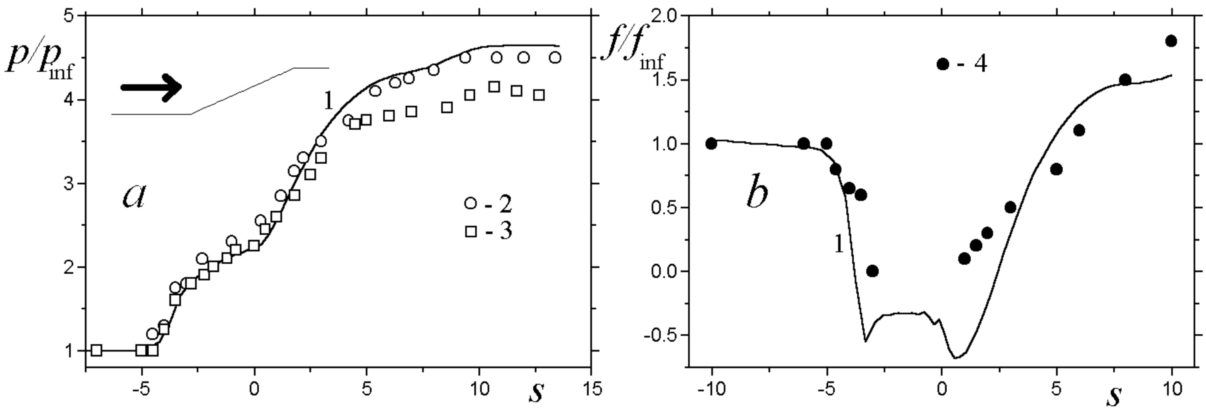

Figure 2. Comparison of calculation (1) and experimental (2-4) distributions of the relative values of pressure *(a)* and friction *(b)* in the vicinity of a 24-degree compression angle at M=2.85. 2-4 – [3].

The compression angle is calculated using the modified program on the modeling of the supersonic channel flow. The upper symmetry plane is replaced by a solid wall and the lower wall – by the symmetry plane which is moved away from the upper wall at a given distance. In principle, this problem has no linear scale, i.e. it must be solved in dimensional quantities. Nevertheless it is assumed that the Reynolds number built as in the previous problem in terms of a characteristic unit size does not exert an essential influence on the solving to be made. In the considered case the wedge angle is assigned equal to 24°, Reynolds and Mach numbers – 10^5 and 2.85, the temperature factor T_w/T_o = 1.06. The flow cross-section at the channel inlet is equal to 8 unit sizes and at the channel outlet – 5. From the outlet boundary up to the compression angle – 10, the length of the section of the upper wall up to the outlet boundary – 14, and the length of the entire computational domain – 30.738. The computational domain is covered with an oblique grind comprising 211×163 cells. A minimal step near the wall is 10^{-4}.

As seen from Figure 2, there is a good agreement between the predicted and experimental data on the compression angle at a moderate Mach number. To some degree, this supports the validity of the assumption made.

As a whole, the calculations of the test problems have strengthened that the chosen turbulence model – MSST can be used for calculating separated flows of compressible gas.

Investigation results. Over several years the experimental studies of the cooling influence of a relief made by convexes and concaves in the supersonic flow using heat images are being carried out at the Institute for Mechanics at Moscow State University under the supervision of Academician of the RAS Leontiev A.I. Currently, the numerical calculations have been made that as a whole have supported the obtained results.

In Figures 3 and 4 comparison is made the calculated data on the local and integral loads on the relief made by ten sequential identical grooves in the supersonic air flow. The results are given for the sixth groove, the depth Δ and the edge rounding radius R are varied.

Depending on the height of bumps or the depth of grooves two flow regimes are seen: with attached shock waves in the case close to non-separated flow around the groove-made relief and with detached shock waves in the developed separated wall flow with periodic recirculation zones (Figure 3). The regimes differ in heat loads, in the second case the local peaks of heat fluxes being much lower. It should be noted that the integral heat loads on a groove are determined as the ratio of the groove length to the streamlined St3 and also to the corresponding flat wall St2.

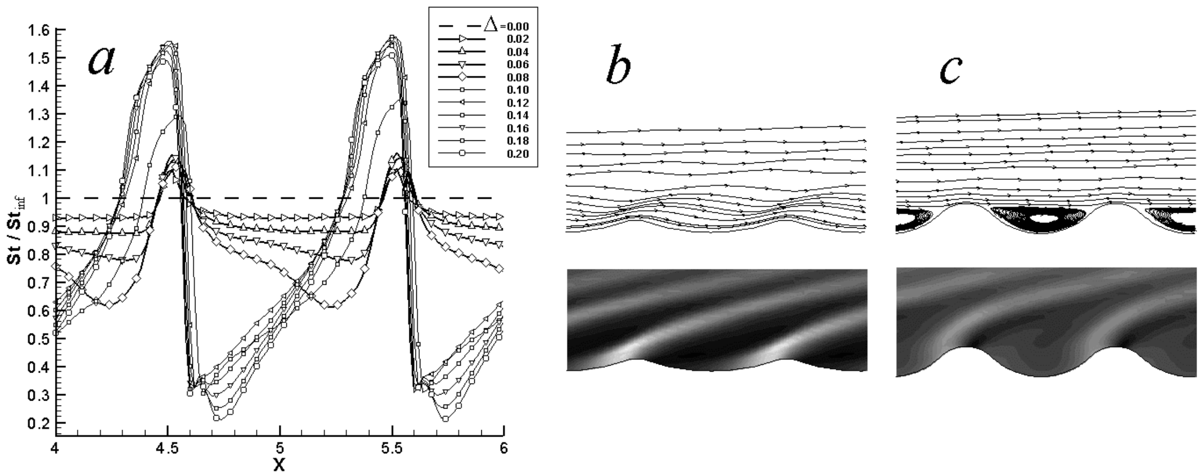

Figure 3. Estimation of the influence of the depth Δ of a periodic groove on the distribution of relative heat transfer (*a*), and comparison of the flow patterns and pressure fields for the grooves with a depth of 0.08 (*b*) and 0.2 (*c*) at R=0.25, M=4 and Re=2×10^5.

Increasing the groove depth (Figure 4a) provides a non-monotonic relation $C_x(\Delta)$ with a maximum at a moderate depth of 0.08. A decrease in the force load on the curvilinear relief is connected with the progressive flow behavior as Δ increases. The drag of a curvilinear relief reaches a plateau but appears to be higher (three times) as against a flat wall. Increasing R (Figure 4b) first affects the decreasing of heat load peaks. At all Δ there occurs the cooling influence of the relief. As follows from Figure 4c, in the hypersonic flow around the relief with a concave the distribution of local heat fluxes over the middle cross-section is practically the same both flat and space flow. Only in the wake behind a dimple there is a higher level of heat loads.

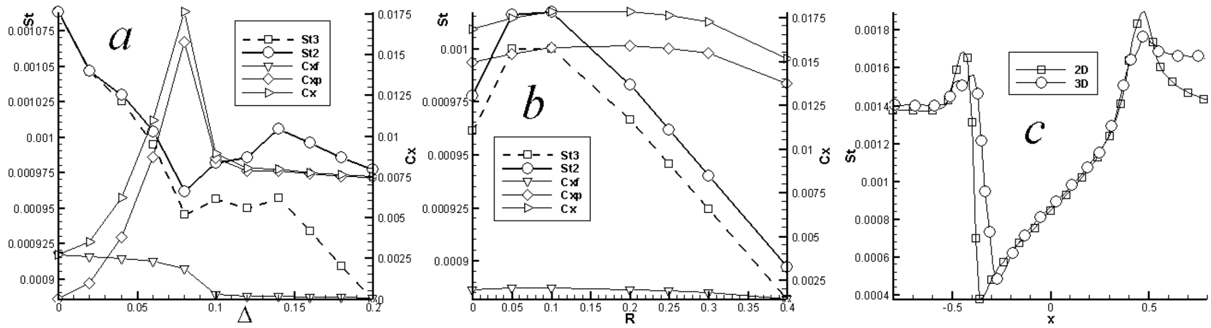

Figure 4. Influence of the depth (*a* - at R=0.25) and the edge rounding radius (*b* – at Δ=0.08) upon the integral force *Cx* and heat loads St on the groove at M=3 and comparison of the distributions of heat fluxes over the middle cross-section of the dimple and the size-equivalent groove (*c* - Δ=0.08, M=4).

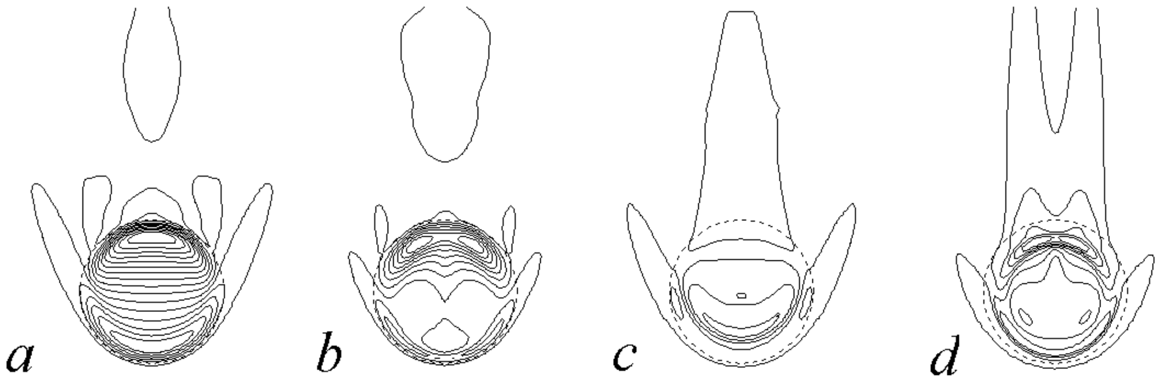

Figure 5. Comparison of isobars (a, b) and the Stanton numbers (c, d) for spherical dimples with depths of 0.1 (a, c) and 0.2 (b, d) over the flat wall at M=4, Re=2×10^5. The isobars are given with a step of 0.005 and the Stanton numbers – with a step from 0.0005 to 0.0025.

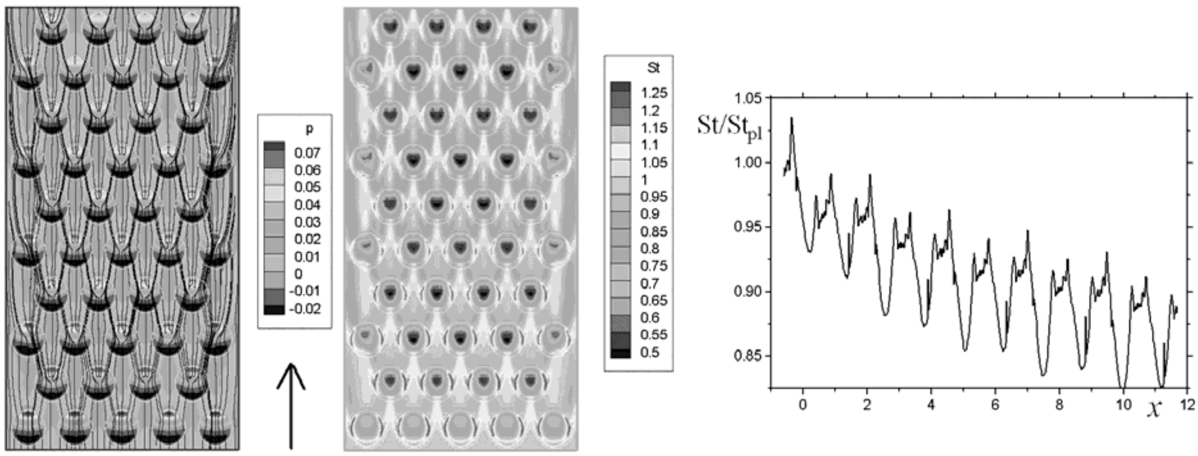

Figure 6. Isobars, Stanton numbers and also longitudinal distributions of relative Stanton numbers on a surface with 45 dimples at Δ=0.08, R=0.25, M=4, Re=2×10^5.

Figure 5 compares the results of numerical simulation of supersonic air flow around dimples of different depth.. It should be noted that unlike the subsonic flow regime, increasing the dimple depth does not transform the vortex structure in the dimple. It is formed and remains a two-cell one. The distributions of local force loads on the dimple of moderate depth (of the order of 0.1) are much higher as against that of large depth, i.e., the tendency observed in the case of flow around grooves is supported. Figure 6 plots the computational data on the flow around the surface with 45 dimples.

Conclusion. Comprehensive testing of multiblock-based algorithms is made on the problems, including those having experimental analogs. The applicability of Menter's model for transfer of shear stresses for calculation of separated wall flows and tornado-like heat transfer is substantiated. The developed algorithm is approved for the problems on interaction of an oblique shock wave with a turbulent boundary layer. The experimentally revealed at IM MSU cooling influence of the surface with concaves at super-and hypersonic flow velocities (Figure 6) is supported numerically. It is shown that a 2.5 – 3-fold (as compared to the flat wall) increase in drag is accompanied by decreasing heat transfer is accompanied by heat transfer reduction (of the order of 8%) for multirow, grooved and dimpled surfaces. The honeycombed-shaped surfaces that possess the better properties in comparison with the spherical dimple characteristics are proposed.

Acknowledgements

The work was performed under financial support of Russian Foundation of Fundamental Research (projects 05-02-16184, 04-02-81005, 05-01-00162).

References

[1] Kiknadze GI, Gachechiladze IA, Oleinikov VV (1994-1995) Tornado-like Technology of Heat and Mass Transfer Intensification at Lagging Increase of Hydraulic Losses (Review of Experimental Results and Intensification Mechanisms). Moscow: Basert Co.Ltd.
[2] Leontiev AI, Olimpiev VV, Dilevskaya EV, Isaev SA (2002) Essence of the mechanism for heat transfer enhancement over the surface with spherical dimples. Izv. RAS. *Energetics,* No., P.117-135.
[3] Isaev SA, Leontiev AI, Baranov PA, Pyshnyi IA (2003) Numerical Analysis of the Influence of the Depth of a Spherical Hole on a Plane Wall on Turbulent Heat Transfer. J. Engineering Physics and Thermophysics. Vol. 76. N1. P.52-59.
[4] Isaev SA, Leontiev AI (2003) Numerical Simulation of Vortex Enhancement of Heat Transfer under Condititions of Turbulent Flow past a Spherical Dimple on the Wall of a Narrow Channel. High Temperature. Vol.41. N5. P.665-679.
[5] Isaev SA, Leontiev AI, Kudryavtsev NA, Pyshnyi IA (2003) The Effect of Rearrangement of the Vortex Structure on Heat Transfer under Condition of Increasing Depth of a Spherical Dimple on the Wall of a Narrow Channel. High Temperature. Vol.41. N2. P.229-232.
[6] Chyu MK, Yu Y, Ding H, Downs JP, Soechting FO (1997) Concavity Enhancement Heat Transfer in an Internal Cooling Passage. *ASME Paper*. 97-GT-437. 8p.
[7] Isaev SA, Leontiev AI, Baranov PA, Pyshnyi IA, Usachov AE (2002) Numerical Analysis of the Vortex Intensification in a Heat Transfer in a Channel with a Set of Deep Spherical Dimples on One of the Walls. Doklady Physics. Vol.47. N10. P.755-757.
[8] Isaev SA, Leontiev AI, Kudryavtsev NA (2005) Numerical Simulation of Hydrodynamics and Heat Transfer under Conditions of Turbulent Transverse Flow past a “Trench” on a Plane Surface. High Temperature. Vol.43. N1. P.89-102.
[9] Wuest C (2004) Dellen im dach. Der Spiegel. N14. P.170-172.
[10] Isaev SA,Leontiev AI, Kiknadze AI, Kudryavtsev NA Decrease in hydraulic losses due to dimpled coatings (numerical simulation). Hydraulics *(science and discipline): Proc. Sci.-Theor. Conf. St. Petersburg: St. Petersburg State Polytechnic University Press,* 2004, c.102-105.
[11] Control of flow around bodies with vortex cell when applied to vehicles of integral arrangement (numerical and physical modeling). Eds. A.V. Ermishin and S.A. Isaev. Moscow: MSU Press, 2003.
[12] Ferziger JH, Peric M (1999) Computational Methods for Fluid Dynamics. Berlin, Heidelberg.
[13] Bobyshev VK, Isaev SA (1998) Numerical Investigation of the Effect of Compressibility on the Mechanism of Decreasing the Motion Drag of a Cylinder with Organized Stall Regions in a Turbulent Flow of Viscous Gas. J. Engineering Physics and Thermophysics. Vol.71. N4. P.600-606.
[14] Horstman CC (1991) Hypersonic Shock-Wave Turbulent Boundary-Layer Interaction Flows - Experiment and Computation. *AIAA Paper.* No. 91-1760. 23 p.