Sergiy YERSHOV, Viktor YAKOVLEV, Anton DEREVYANKO
Podgorny Institute for Mechanical Engineering Problems of NAS of Ukraine,
yershov@ipmach.kharkov.ua
Maria GRYZUN, Dmytro KOZYRETS
National Technical University "KhPI"

THE DEVELOPMENT OF NEW CFD SOLVER FOR 3D TURBOMACHINERY FLOW COMPUTATIONS

The concept of the new CFD solver for the 3D turbomachinery flow simulation using the RANS equations is considered. The governing equations are supplemented with the *k*-ω SST turbulence model. The realisability constraints and the special boundary conditions for adverse pressure gradient flows are shown to be important. An improved numerical technique is suggested to increase efficiency and robustness of the computational procedure. Numerical results for selected test cases are presented. Screenshots of the developed software are shown.

Key words: viscous compressible flow, turbomachinery, CFD solver

1. INTRODUCTION

Currently, many CFD solvers, such as ANSYS CFX, ANSYS Fluent, Fine Turbo Numeca, Star CD, etc., are used to calculate 3D viscous flows. In the middle nineties at the Podgorny Institute for Mechanical Engineering Problems of National Academy of Sciences of Ukraine, under the leadership of the first author of the present paper, a similar software package, named FlowER [16], was developed. It was one of the first CFD solvers for the Reynolds-Averaged Navier-Stokes equations in Eastern Europe and it was widely used by a number of institutions in Ukraine, Russia and Poland. Using this solver, it was possible to simulate a 3D flow through turbines and compressors to improve their design. However, a number of FlowER disadvantages limit its usage today.

It is very difficult and sometimes impossible to calculate the flow with large adverse pressure gradients using the FlowER code. Particularly, the flow in highly loaded compressor cascades, different diffusers and turbine flowpaths at off-design conditions are extremely hard to simulate. In many cases, we noticed the simulated flow separations, which are typical under this flow conditions, appear to be larger than in reality.

Finite-difference approximation implemented in FlowER is second-order accurate if used on uniform grids; however, accuracy deteriorates to the first order otherwise. Moreover, the solution becomes very sensitive if kinks in mesh-lines and skewed cells are present, and convergence rate is reduced.

The kernel of the software package was written more than 15 years ago. Graphic user interface was created for the DOS operating system and it does not work under modern Windows Vista and Windows 7 environments any longer. Additionally, the code does not support 64-bit architecture. RAM limitations that existed at the time of the solver creation resulted in cumbersome, dense and extremely difficult-to-modify code. Thus, a complete rewrite was preferred over modifications of existing code.

In conclusion, it was necessary to develop a new CFD solver without mentioned above disadvantages. In this present paper we describe the main principles assumed as the basis for the new code.

2. FLOW MODELS

At present, different mathematical models are used for a numerical simulation of viscous compressible turbomachinery flows. The system of the Euler equations describes the flow of an inviscid gas. On the other hand, the RANS equations are generally used to simulate a statistically averaged turbulent flow. Moreover, unsteady behaviour of large scale turbulence can be calculated using the Large-Eddy Simulation (LES) technique. The Direct Numerical Simulation (DNS) approach, which is based on the Navier-Stokes equations, allows calculating the entire range of turbulent scales. Finally, the hybrid RANS/LES models are capable of simulating very large turbulence vortices at the scale comparable to the body size.

According to up-to-date estimations, rigorous computations of turbomachinery flows at high Reynolds numbers using DNS and LES models will be scarcely possible in the nearest future. Therefore, the RANS modelling is the basic approach for the industrial flow simulation nowadays [5].

Hybrid models require computational resources of 2-3 orders of magnitude greater than RANS models. Although, it is substantially smaller than the cost of LES or DNS computations, such approach could be used solely for research, not industrial, computations.

The developed CFD solver is intended for both engineering computations and investigations of a turbulent flow through turbomachinery cascades. Hence, we included both RANS and hybrid RANS/LES turbulence models.

3. TURBULENCE MODELLING

There exists a variety of turbulence models for the RANS equations. Nevertheless, none ensure a good quantitative agreement with experimental data for all flow types possible. Therefore, the choice of a turbulence model is very important to guarantee an adequate simulation of different phenomena in a turbulent flow.

Three basic classes of turbulence models for the RANS equations are considered: algebraic models, differential models, and Reynolds-stress transport models. The latter, in spite of their large potential, are rarely used, because their description of a near-wall flow is not sufficient. Algebraic turbulence viscosity models have reached their limitation and yield satisfactory results for simple flow types only. Meanwhile, differential turbulent viscosity models continue to develop. To the large extent, the development takes place due to new turbulence data utilization that is obtained through DNS and LES simulations. Differential models of turbulent viscosity allow adequate numerical simulations, and in many cases they are preferred over the others.

Spalart-Allmaras [11] and k-ω SST [9] turbulence models, together with some of their modifications, are proven to be adequate and reliable models. Therefore, it is reasonable to implement them in the new CFD solver. To avoid the generation of unrealizable nonphysical Reynolds stresses, we use the realisability constraints [7] which guarantee non-positive normal stresses and Cauchy inequality for the stress tensor. We also have implemented the low-Reynolds modification of the k-ω SST turbulence model with the production term modifier [4] in the new code to simulate the laminar-turbulent transition. It allows an increased accuracy of the energy losses calculation for a cascade flow and an improved heat transfer simulation for cooled turbine blades.

Hybrid RANS/LES flow models could be constructed in such a way that the resolvable vortices are calculated directly, given the mesh/turbulence scale ratio in local flow domain is significantly small. Unresolvable (subgrid) vortices, however, are modelled with various adaptive turbulence models that are based on turbulent viscosity models.

4. BOUNDARY CONDITIONS

Formulation of boundary conditions at permeable and impermeable boundaries for the RANS equations is sufficiently studied in the existing literature (see e.g. [3]). Therefore, we skip detailed description of the boundary condition formulation, and focus on the implementation procedure. The set of equations at boundaries consists of predefined boundary conditions and additional relations that link flow parameters at boundaries with those in nearest cells. We use linearised relations relative to increments of flow parameters at boundaries. As a result we obtain a system of linear equations that could be solved efficiently [15].

Characteristic relations at permeable boundaries lead to appearance of a spurious reverse flow in case of a large pressure gradient near that boundary. Hence, in this case flow calculations may be impossible. In the present paper, we do not use characteristic relations at boundary points where diffusive effects predominate over convective ones and the reverse flow can appear. Instead, we extrapolate one of flow parameters, the axial velocity or the pressure, from the nearest cell to the boundary face. Another parameter, the pressure or the axial velocity, is fixed at the exit boundary according to the subsonic outflow boundary conditions. Specifying the axial velocity at the exit, instead of the pressure, the mass-flow-rate or the Riemann invariant, ensures the solution stability in case of the flow with large adverse pressure gradients, in particular, for high-loaded compressors.

5. NUMERICAL APPROACH AND ALGORITHMS

Second- and third-order accurate difference schemes are usually sufficient for the RANS and hybrid RANS/LES computations [5]. These schemes generally are more efficient for three-dimensional problems than their analogues of a higher accuracy. Numerical methods of forth and above order of accuracy could be beneficial for computing turbulent vortices with LES and DNS flow models; however, they do not improve agreement between RANS solutions and experimental data. Moreover, it is sufficient to approximate convective terms of turbulence model equations with the first order only [1, 5].

We implement the second-order accurate ENO scheme [13] and the third-order accurate TVD scheme [17] as the key numerical method in the new CFD code. Taking into account difference between directions of the reconstruction and the extrapolation it is possible to make corrections that improve the numerical accuracy on highly irregular meshes [12].

In the initial version of the FlowER solver, diffusive terms were approximated using derivatives obtained from the ENO reconstruction. Since these are minimized derivatives, diffusive terms were significantly underpredicted. This behaviour generates an unstable velocity profile in boundary layers. To improve the diffusive term computation in the posterior version of the FlowER solver, we used the ENO derivatives for tangential stresses and two-point differences for normal stresses approximation. We suggest using the second-order central differences for diffusive terms in the new CFD code.

Implemented in the FlowER solver the implicit Beam-Warming scheme [2] is based on linearisation, factorisation and diagonalisation of the finite difference approximation of the governing differential equations. Such an approach decreases the accuracy and the stability of the solution, if the time step is sufficiently large. In the present paper, we suggest using the iterative implicit scheme constructed using the modification of the Newton method [14]. To increase the stability the implicit residual smoothing (second-order artificial viscosity) is applied [6].

Due to the computational error some positive-defined quantities, such as turbulence kinetic energy and specific dissipative rate, can become negative. It is possible to avoid this by transforming the increments of positive defined quantities into the increments of their logarithms [8], or by using a non-linear increment correction [12], which ensures similar results.

6. SOFTWARE IMPLEMENTATION

The new CFD software package, known under the draft name F, is a successor of the previous FlowER software package. The preprocessor and the postprocessor of the F solver are written using the Fortran-95 programming language for a Windows operating system. The graphical user interface uses standardised facilities included in Windows. We tested the code under the following operating systems: Windows XP, Windows 7, and Windows Server 2003, as well as under Wine emulator included in Ubuntu Linux.

The F software package realises the following new possibilities:

– work with the distributed database;
– multiple simultaneous executions of the same code on a single PC;
– solver remote startup;
– export of computational results to the external post-processor (Paraview, Tecplot and others);
– simplified access to project data;
– work with meshes that contain over 10 million cells in a single blade-to-blade channel, etc.

The graphical user interface of the F software package is shown in Fig. 1-4.

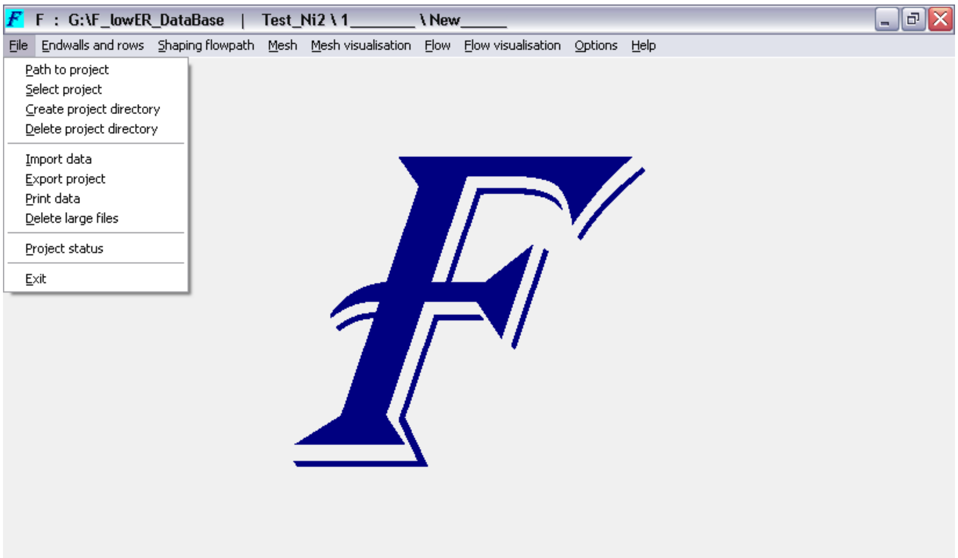

Fig. 1. Graphical user interface of F software package

7. TEST FLOW COMPUTATIONS

We consider the problem of the shock wave / boundary layer interaction (SWBLI) as the first test case. This problem is a classical example of the flow with a large adverse pressure gradient. When the oblique shock wave impinges on the surface with a boundary layer along that, the separation appears near the impinging point. The separation starts upstream the impinging point, whereas the reattachment is located downstream. The reflection shock wave originates near the separation point. The separation zone generates rarefaction waves behind the reflected shock and compression waves further downstream. Such a structure of the flow is predicted well qualitatively and satisfactory approximated quantitatively are predicted by the numerical computation performed with the new F solver; see Fig. 5. These results are obtained using the RANS equations that describe the averaged turbulent flow of viscous compressible gas, Menter's *k*-ω SST turbulence model with the realisability constraints, and modified exit boundary conditions.

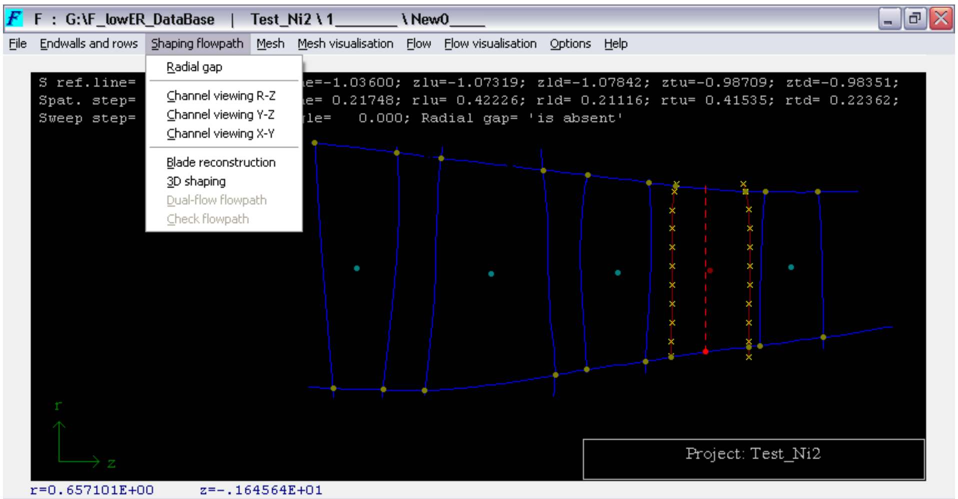

Fig. 2. Flowpath of 2½ stage compressor (preprocessor of F software package)

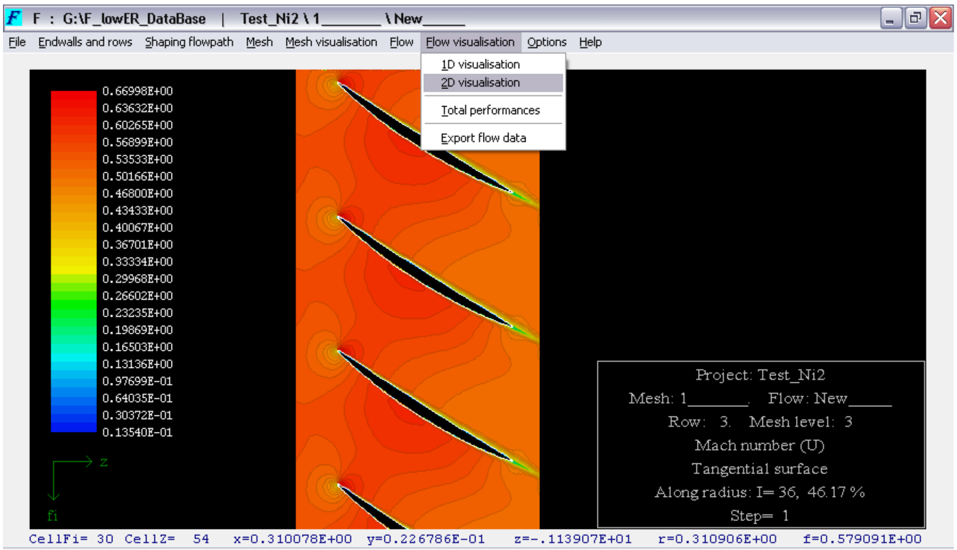

Fig. 3. Mach number contours in compressor cascade
(postprocessor of F software package)

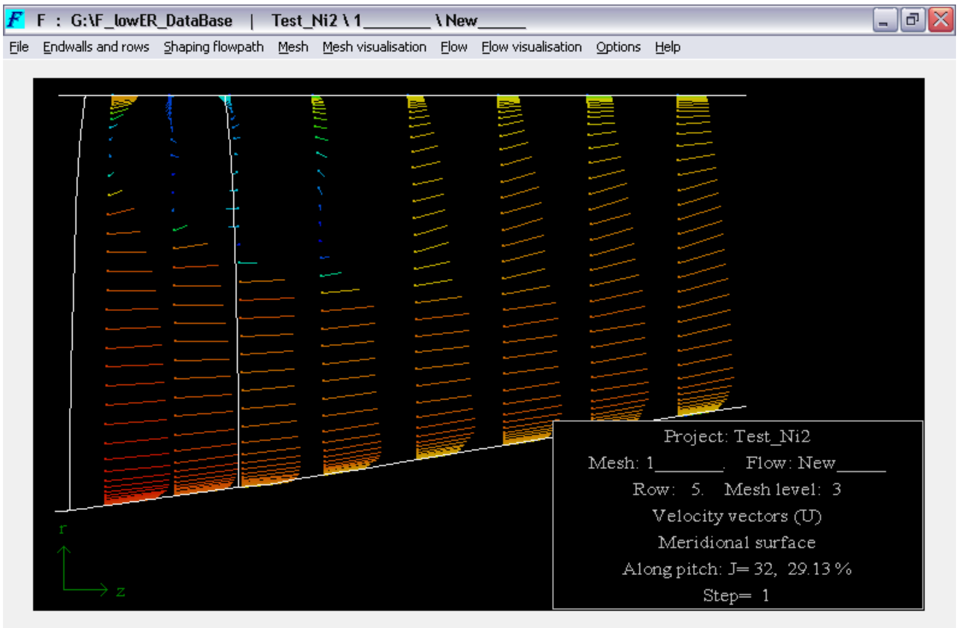

Fig. 4. Velocity vectors in meridional section of last row of 2½ stage compressor (postprocessor of F software package)

Solving the SWBLI problem with the FlowER solver, which does not implement the realisability constraints, we obtain a wrong laminar-like flow structure with two reflected shock waves and pressure oscillations in the separation region. If the pressure rise in the flow direction occurs in the boundary layer at the exit boundary during the convergence process then it is very difficult or even impossible to obtain the numerical solution using the original FlowER solver. Shown in Fig. 5 is the static pressure distribution along the plate in the interaction region. Numerical results are in the acceptable agreement with experimental data [10].

Flows with the large adverse pressure gradient are typical for compressors of jet engines and stationary turbo-installations. Therefore, computations of viscous flow through multistage compressors often cause difficulties. If using the FlowER solver, such computations were usually performed with the initial conditions, which were precomputed using the same model at a slightly larger mass-flow-rate. Unfortunately, even in this case, the flow could not always be calculated at all conditions on the compressor characteristic. This behaviour is due to numerical artefacts: separations originated during the convergence process tend to grow unrestrictedly. Hence, this could result in the program failure. So, for the five-row compressor, shown in Fig. 2, the FlowER solver is able to calculate flow only in the nearest neighbourhood of the flow choking condition, and it fails otherwise.

The inclusion of the realisability constraints in the turbulence model and the modification of exit boundary conditions allow us to perform computations of the flow through the compressor for any flow conditions from a sufficiently large set of the initial flow field. Multistage compressor flow computations are possible for both design and off-design (significantly separated) conditions using the described approach. Figure 6 presents the compressor characteristic obtained numerically using the new F solver. Velocity vectors on the meridional surface of the last stator row at off-design conditions are shown in Fig. 4. The observed separation extends up to 60 percent of the channel circumferential size. Nevertheless, the separation region has constant size and location, and it does not tend to propagate to the exit or the inlet of the computational domain as it occurs when we use the original version of the FlowER solver.

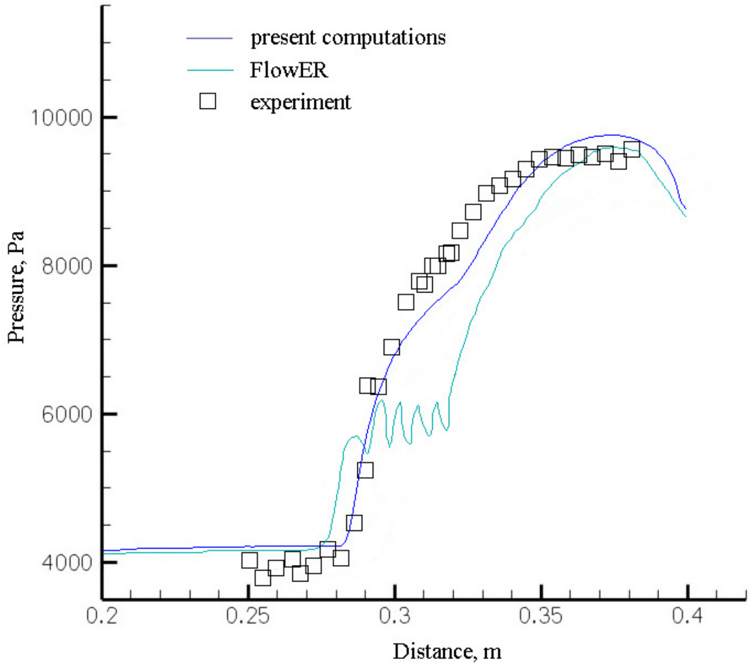

Fig. 5. Pressure distribution along plate in interaction zone

8. CONCLUSION

The concept of the new CFD solver, named F, for the 3D turbomachinery flow simulation using the RANS equations is suggested. RANS and hybrid RANS/LES flow models are considered as the state-of-the-art procedure for industrial and research oriented turbomachinery flow simulations. Spalart-Allmaras and Menter's k-ω SST turbulence models complete the system of the governing equations. The numerical approach is based on non-oscillatory TVD and ENO schemes of the second and third order of accuracy. The numerical simulation of the separated flow induced by the shock/boundary layer interaction as well as flow in multi-stage compressor cascades shows the importance of the realisability constraints and the special boundary conditions at exit boundaries for adverse pressure gradient flows.

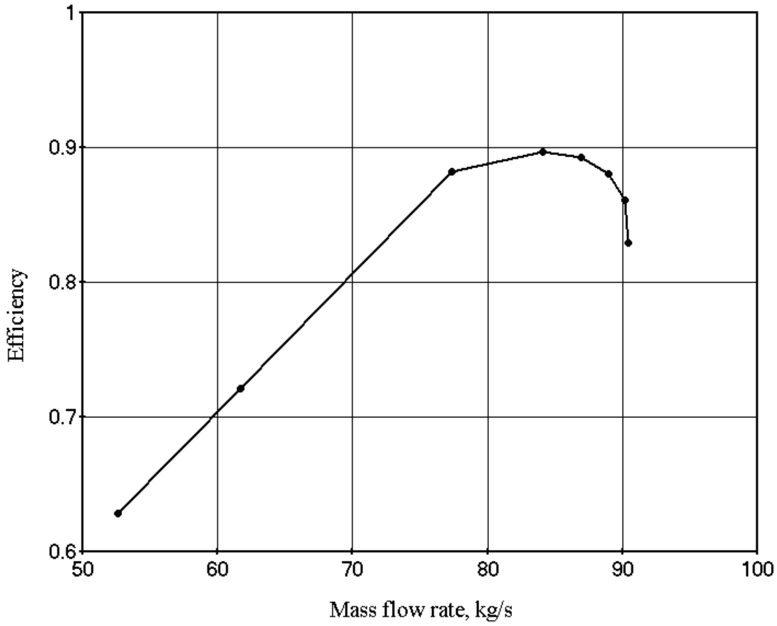

Fig. 6. Computed compressor performance

REFERENCES

[1] Bardina, J.E., Huang, P.G., and Coacley, T.J., 1997, *Turbulence modeling validation, testing, and development*, NASA TM–110446, 98 p.

[2] Beam, R.M., and Warming, R.F., 1977, *An implicit factored scheme for the compressible Navier–Stokes equations*, Proc. AIAA 3rd Comput. Fluid Dyn. Conf, Albuquerque, pp. 645-649.

[3] Blažek, J., 2001, *Computational fluid dynamics: principles and applications*, Elsevier, 470 p.

[4] Denissen, N.A., Yorden, D.A., and Georgiadis, N.J., 2008, *Implementation and Validation of a Laminar-to-Turbulent Transition Model in the Wind-US Code,* NASA TM 215451, 36 p.

[5] Hirsch, Ch., and Tartinville, B., 2009, *Reynolds-Averaged Navier-Stokes modelling for industrial applications and some challenging issues,* Int. J. Computational Fluid Dynamics, Vol. 23, No 4, pp. 295-303.

[6] Jameson, A., and Baker, T. J., 1983, Solution of the Euler Equations for Complex Configurations, AIAA Paper 83-1929, pp. 293-302.

[7] Lumley, J.L., 1978, *Computational Modeling of Turbulent Flows,*/ Adv. Appl. Mech., Vol. 18, pp. 123-176.

[8] Lund, E., Møller, H., and Jakobsen, L.A., 2002, *Shape Optimization of Fluid–Structure Interaction Problems Using Two-Equation Turbulence Models*, AIAA Paper, No 2002–1478, 11 p.

[9] Menter, F.R., 1994, *Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications*, AIAA J., Vol. 32, No 8, pp. 1598 – 1605.

[10] Polivanov, P., Sidorenko, A. and Maslov, A., 2009, *Experimental Study of Unsteady Effects in Shock Wave/Turbulent Boundary Layer Interaction,* AIAA Paper, No. 2009-409, 7 p.

[11] Spalart, P.R., and Allmaras, S.R., 1994, *A One-Equation Turbulence Model for Aerodynamic Flows,* Recherche Aerospatiale, No 1, pp. 5-21.

[12] Tillyayeva, N.I., 1986, *Generalisation of Modified Godunov's Scheme for Arbitrary Irregular Meshes,* Proceedings, Central Aerohydrodynamical Institute, Vol. 17, No 2, pp. 18–26 (in Russian).

[13] Yershov, S.V., 1994, *Quasi-monotonous ENO scheme of high accuracy for Euler and Navier-Stokes equations*, Matematicheskoye Modelirovaniye, Vol. 6, No 11, pp.58–64 (in Russian).

[14] Yershov, S.V., 2007, *Solution of gasdynamic equations with iterative implicit scheme*, Bulletin of Karazin Kharkov National University, "Mathematical simulation. Information technologies. Computer-aided systems", No 775, iss. 7, pp. 159–173 (in Russian).

[15] Yershov, S.V., 2011, *Numerical Implementation of Boundary Conditions at Permeable Boundaries for Gas-Dynamic Equations*, Problems of Mechanical Engineering, Vol. 14, No 4, pp. 25-37 (in Russian).

[16] Yershov, S.V., and Rusanov, A.V., 1996, *The Application Package FlowER For The Calculation of 3D Viscous Flows Through Multi Stage Turbomachines*, Certificate of State Registration of Copyright, Ukrainian State Agency of Copyright and Related Rights, ПА No 77 (in Ukrainian).

[17] Zijlema, M., 1994, *On the construction of a third-order accurate TVD scheme using Leonard's normalized variable diagram with application to turbulent flows in general domains*, Delft University of Technology, Technical Report DUT-TWI-94-104, 25 p.