Numerical Simulation of Evolution of Turbulent Wake Subjected to Strong Adverse Pressure Gradient with the use of Zonal RANS-LES approach[*]

E.K. Guseva, M.L. Shur, M.Kh. Strelets, A.K. Travin

Peter the Great Saint-Petersburg Polytechnic University (SPbPU)

Results are presented of high-fidelity scale-resolving unsteady 3D RANS-IDDES simulations of the wake of the flat plate subjected to adverse pressure gradient created by two pairs of thin liner foils at conditions corresponding to the concurrent experimental studies being conducted at the Technical University of Braunschweig. Simulations were performed on computational grids up to 100 million cells on the high-performance computational cluster "Tornado" of the Supercomputer Center "Polytechnichesky" with the use of the hybrid (MPI/Open MP) parallelization technique. The paper presents an outline of the computational problem setup, numerics used and computational resources needed, along with the major obtained physical results.

Key words: Turbulence, Scale-resolving simulation, Wakes, Adverse Pressure Gradient, Massively parallel computations.

1. Introduction

Wings of modern airliners have a complex 3-element configuration (a so-called high lift wing which consists of a main wing, slats, and flaps) aimed at ensuring sufficiently high lift at the landing and takeoff stages, when the flight speed is relatively low. Design and optimization of such wings is a complicated process requiring multi-variant calculations, which are now routinely performed with the use of the Reynolds Averaged Navier-Stokes (RANS) equations combined with semi-empirical turbulence models. RANS approach is computationally efficient and is proven to provide required accuracy for the cruise flight regime. However, for the take-off and landing stages even the best existing RANS models are known to be not accurate enough [1].

One of the reasons of this failure is inability of RANS to predict evolution of the turbulent wakes of the slat and the main wing when it is subjected to a strong adverse pressure gradient (APG) generated by the deflected flap. In particular, RANS models tend to under-predict velocity deficit caused by APG [2]. This motivated a joint German-Russian Project "Wake flows in Adverse Pressure Gradient" aimed at deeper understanding of the underlying physics of the evolution of turbulent wakes exposed to adverse pressure gradient and at development of improved RANS models capable of accurate prediction of the class of flows in question.

For developing such RANS models and for their calibration and verification high fidelity data are needed not only on the mean flow but also on its statistical parameters some of which (first of all, the dissipation rate of turbulence kinetic energy), cannot be measured in physical experiment with high enough accuracy. Thus, along with experiments, numerical simulations must be conducted with the use of scale-resolving approaches, which require high computational resources but provide reliable prediction of all the essential flow characteristics thanks to the direct resolution of a major part of turbulence spectrum. This paper presents the first results of a series of scale-resolving unsteady 3D simulations (zonal RANS-IDDES) of the wake of the flat plate subjected to adverse pressure gradient created by two pairs of thin liner foils at conditions corresponding to the concurrent experimental studies being conducted in the framework of the project at the Technical University of Braunschweig (TU BS). Simulations were performed with the use of the in-house multi-purpose CFD code Numerical Turbulence Simulation (NTS code) [3] on a high-performance computational cluster "Tornado" of the Supercomputer Center "Polytechnichesky") of the Saint-Petersburg Polytechnic University.

---

[*] The study was funded by DFG and RBRF (Grants No. RA 595/26-1 and No. 17-58-12002). Computations were performed with the use of resources of the Supercomputer Center "Polytechnichesky".

The paper starts from a description of the NTS code used in the simulations (Section 2). Then, Section 3 provides a brief outline of the computational setup used for the simulation of the considered flow and a description of the computational resources used. In Section 4 major results of the computations are presented, and in Section 5 some conclusions are formulated based on the study performed.

2. NTS code

NTS code is a cell-vertex finite-volume in-house code developed in SPBPU for simulations of turbulent flows. It allows computing steady and unsteady flows in a wide range of Mach numbers including incompressible flows with the use of structured multi-block overset grids of the Chimera type. Most of the currently available approaches to turbulence modelling are implemented in the code, including RANS-based approaches with different semi-empirical models, Large Eddy Simulation (LES), hybrid RANS-LES approaches, and Direct Numerical Simulation (DNS). The code has been verified by numerous comparisons with other CFD codes (CFL3D code of NASA, GGNS and BCFD of The Boeing Company, TAU code of DLR, ANSYS-CFX and ANSYS-FLUENT) and is currently considered as one of the most reliable codes for computations of turbulent flows.

For massively parallel computations, the code employs a so-called hybrid Message Passing Interface (MPI) / Open Multi Processing (Open MP) parallelization scheme, which ensures a high efficiency of simulations on very large computational grids required for the scale-resolving simulations. Within this approach both MPI libraries (with distributed memory technologies) and Open MP libraries (with shared memory technologies) are used for parallelization of computations in different grid blocks (or set of grid blocks). This strategy is very flexible and is easy to adapt to computers with different architecture by manually varying a set of managing parameters. The code is written on FORTRAN-90 and does not use any specific libraries other than the standard MPI and Open MP.

As mentioned above, the code accepts structured multi-block overset grids. Computational domain and grid are divided into a set of grid blocks taking into consideration both the flow geometry and effective usage of computers with large amount of nodes/cores. The blocks are overlapping (see fig.1) which allows retaining a high order of spatial approximation used in the inner nodes of the blocks at the block boundaries. The size of a single grid block should not be less than 50000-100000 cells since otherwise the efficiency of parallelization decreases [4].

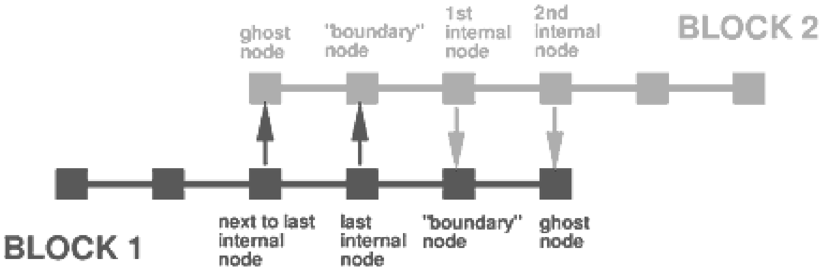

Figure 1. Schematic of overlapping grid blocks in NTS code

3. Experimental flow model, computational setup and computational resources used

Figure 2 shows a sketch of the experimental flow model which detailed description along with an outline of the experimental setup is presented in [5]. The model includes a flat plate (FP) generating the wake and two pairs of symmetrically installed liner foils (LF1 and LF2) creating APG. The Reynolds number based on the plate length L = 1.058m and free stream velocity U_0 varied from 1.6 to 3.2 million, and in the present work computations were performed at Re = 3.2 million. Corresponding Mach number was around 0.14, which justifies using the incompressible flow assumption in the simulations.

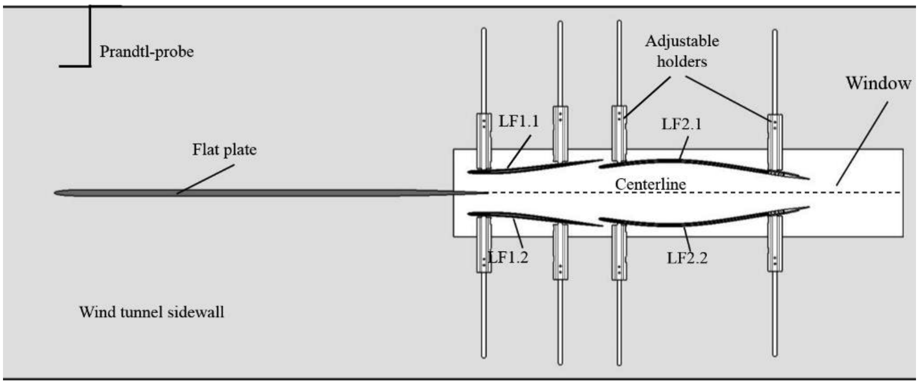

Figure 2. Sketch of experimental flow model

Within the zonal RANS-IDDES approach [6] computational domain is subdivided into two sub-domains (zones): RANS and IDDES. This allows reducing computational resources by using fine grids required for IDDES only in a limited part of the domain ("focus region"), while in the rest (major) part of the domain much coarser grids sufficient for RANS may be used. RANS and IDDES sub-domains manually prescribed in the present simulations and the computational grid in *XY*-plane are shown in the Figure 3.

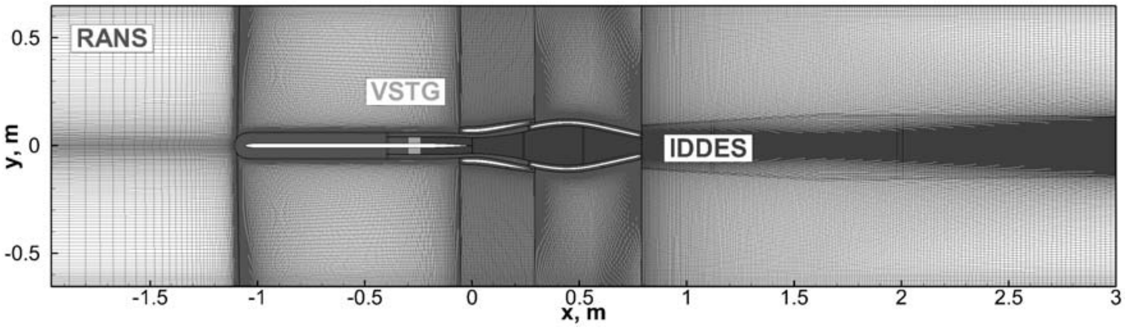

Figure 3. Computational domain, grid, and RANS & IDDES sub-domains in *XY*-plane

The RANS zone extends from the inlet boundary of the domain to the section x = -0.3m and includes the outer part of computational domain farther downstream as well as the boundary layers forming on the liner foils. The IDDES zone covers the rest (downstream) part of the FP boundary layer and the wake, which is the focus region in this study. To introduce resolved turbulent structures into the IDDES subdomain, which is essential for ensuring a rapid transition from the fully modeled turbulence (RANS sub-domain) to mostly resolved turbulence (IDDES sub-domain) the Volume Synthetic Turbulence Generator (VSTG) [6, 7] was used at the RANS-IDDES interface. In the RANS zone the k-ω RANS model of Menter [8] is applied, whereas in the IDDES zone we use the IDDES approach [9] with the same underlying RANS model.

The incompressible branch of the code used in present simulations employs the flux-difference splitting method of Rogers and Kwak [10]. In the RANS sub-domain the inviscid fluxes in the governing equations are approximated with the use of a 3rd-order upwind-biased scheme and in the IDDES sub-domain a 4th-order central scheme is used. The viscous fluxes are approximated with the 2nd-order central scheme. For the time integration, an implicit 2nd-order backward Euler scheme with sub-iterations is applied.

Simulations are performed with the use of the periodic boundary conditions in the spanwise direction. Hence, in order to check whether the span size of the domain is sufficient to ensure span-size independent mean and statistical flow characteristics, two simulations were carried out in the computational domains with the span size L_z = 0.1m and 0.2m. The grids for both domains had 24 blocks and were identical in *XY*-plane. They were clustered near the FP and liner foils walls so that the size of the first near-wall step in the wall-normal direction would be less than 1.0 in wall units. In the IDDES sub-domain the grid steps were sufficiently small for obtaining nearly grid-independent solution [11].

As a result, the total grid counts for the simulations in the narrow and wide domains around 30 and 60 million cells, respectively.

The time step used in simulation was chosen to ensure less than 1.0 Courant number (0.25 in major part of the domain). The physical time of the simulations needed to achieve statistically steady-state flow and to accumulate the unsteady flow data for getting reliable turbulence statistics was about $15(L/U_0)$. This corresponds to running the simulation for $3{\cdot}10^4$ time steps or $6{\cdot}10^5$ iterations (20 internal sub-iterations in pseudo time per a time step were conducted, which ensured not less than two-order drop of residuals).

Computations were performed on the 11 nodes of the cluster “Tornado” (each node has two processors with 14 Intel Xeon E5-2697 v3 cores and 64 GB RAM). The wall-clock time per iteration was 2.0 and 4.2 seconds for the narrow and wide computational domains, respectively.

4. Major Results of the Simulations

As mentioned in the Introduction, the major purpose of this study is obtaining reliable mean and statistical data for wake flows exposed to APG needed for building and validation of improved RANS model(s) capable of predicting this class of flows. A capability of the RANS-IDDES approach used in the present work to provide such data is illustrated by Figures 4- 10 below.

In particular, Figures 4, 5 visibly demonstrate that this approach ensures a resolution of fine-grained (consistent with the grid used) turbulence in both the FP boundary layers downstream of the VSTG and in the wake. Other than that, the figures clearly suggest that the approach does captures a specific feature of the wakes subjected to APG, namely, a presence of stagnation region which is known to be poorly predicted (underestimated) by RANS models.

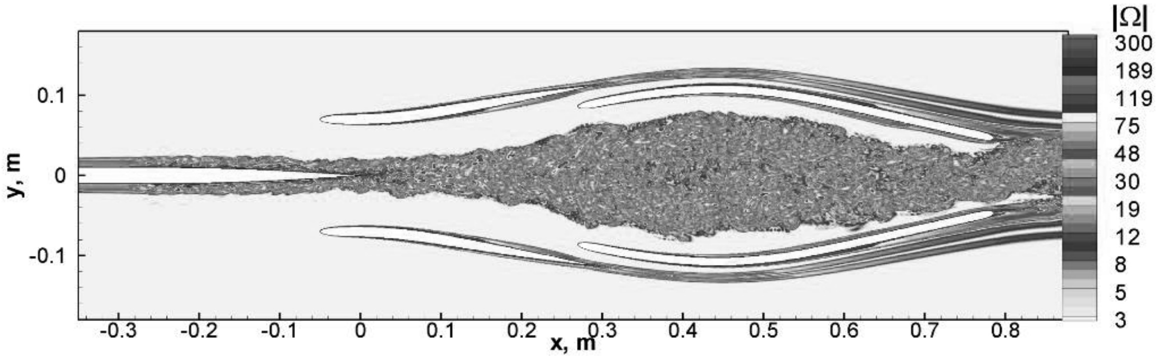

Figure 4. Instantaneous field of vorticity magnitude in XY-plane obtained on narrow domain

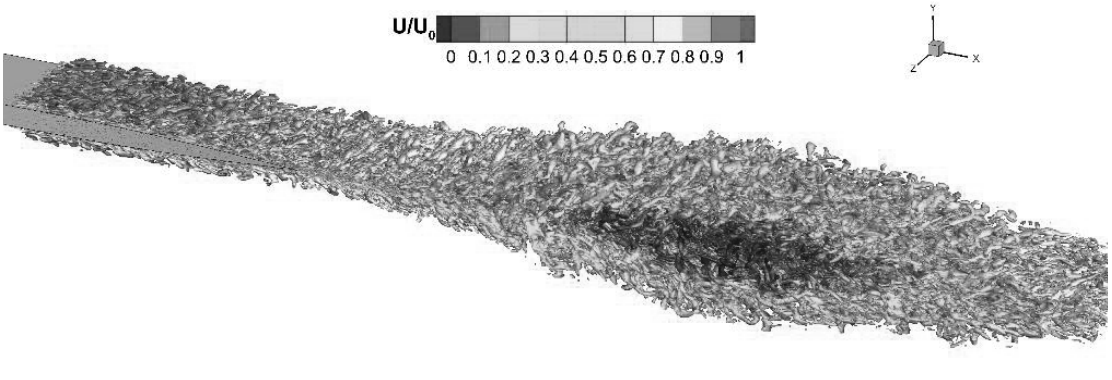

Figure 5. Isosurface of Q-criterion colored by streamwise velocity obtained on narrow domain (liner foils are not shown)

Figures 6, 7 compares flow visualizations (snapshots of the vorticity magnitude) in *XZ* (Fig.6) and *ZY* (Fig. 7) planes from the simulations in the narrow and wide domains. Analysis of these snapshots suggests that the simulation in the wide domain does not reveal any 3D large-scale turbulence structures which could not be represented in the simulation in the narrow domain.

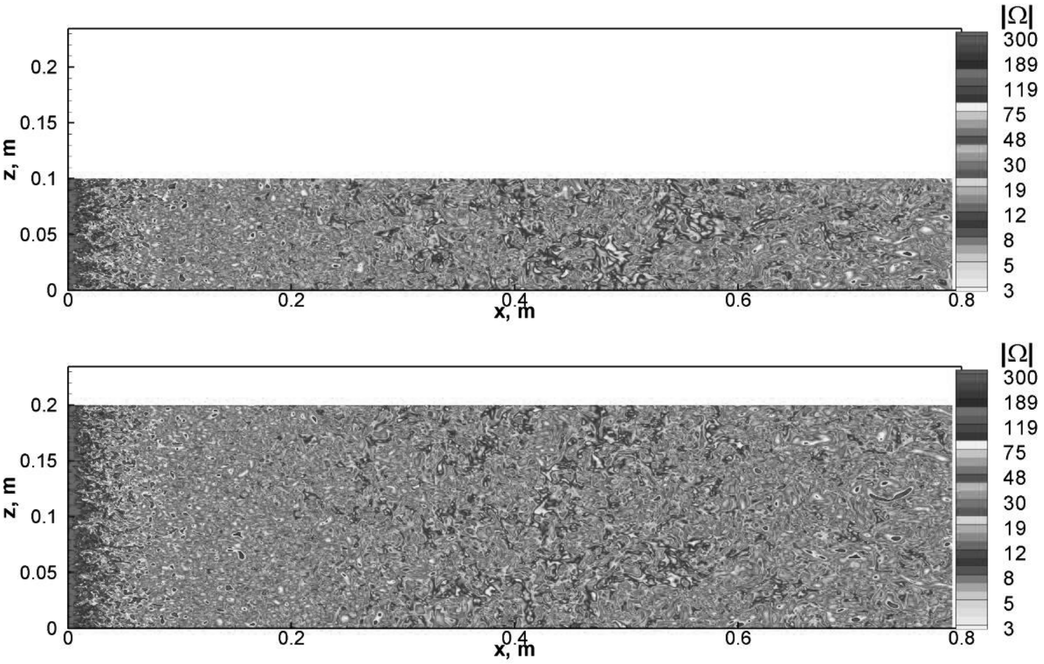

Figure 6. Instantaneous fields of vorticity magnitude in symmetry *XZ*-plane ($y = 0$) from the simulation in narrow (upper frame) and wide (lower frame) domains

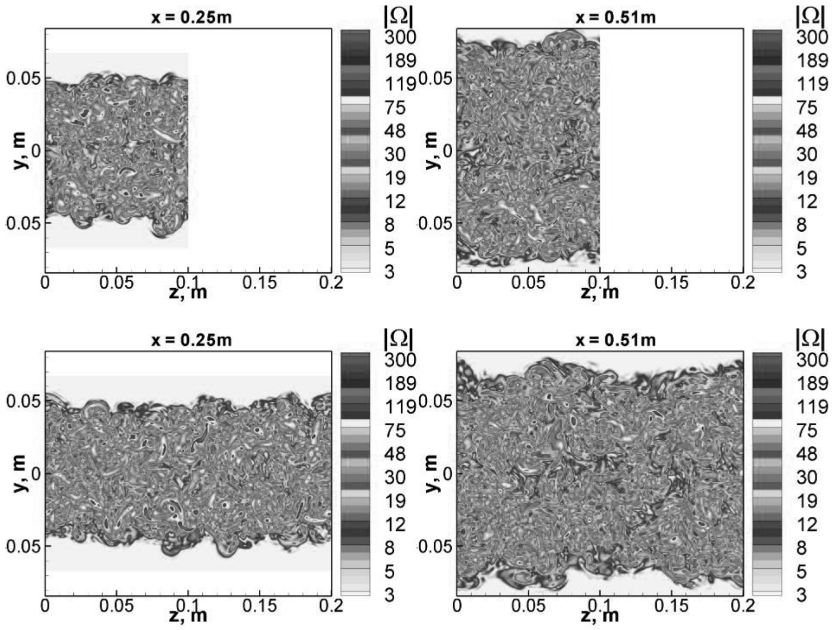

Figure 7. Instantaneous fields of vorticity magnitude in two *ZY*-planes from the simulation in narrow (upper frame) and wide (lower frame) domains

This observation is confirmed quantitatively by the comparison of the results obtained in the wide and narrow domains presented in figs 8-10. As seen in these figures, the mean flow velocity, resolved kinetic turbulent energy, and its dissipation rate (the latter is computed based on the balance of the *k*-transport equation [12]) predicted by the simulations in the narrow and wide domains are virtually identical, which justifies the use of the narrow domain in the further simulations of the experimental flow model (cases with other Reynolds numbers and higher APG).

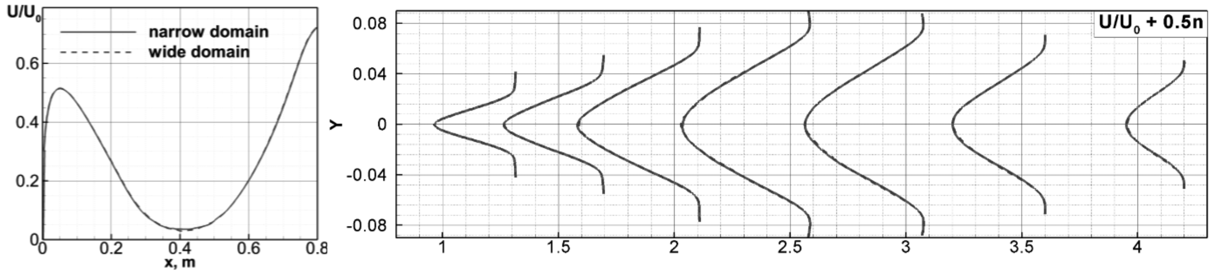

Figure 8. Comparison of mean velocity distribution along the wake symmetry plane $y = 0$ and of velocity profiles at sections $x_n = (0.1n)$ m, $n = 1, 2,\ldots,7$ computed in narrow and wide domains

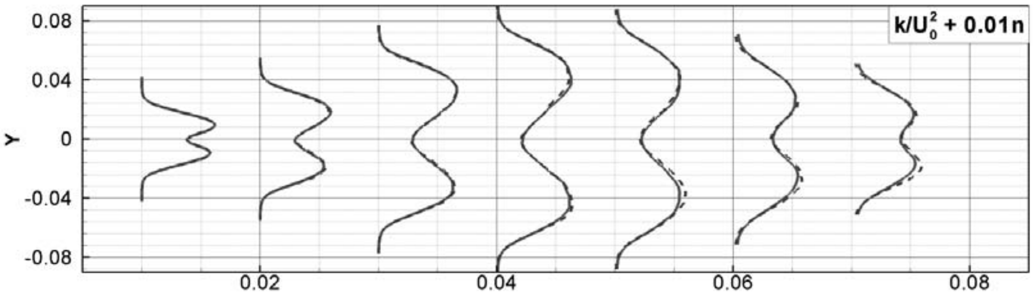

Figure 9. Comparison of profiles turbulent kinetic energy at sections $x_n = (0.1n)$ m, n = 1, 2,…7 computed in narrow and wide domains (see Fig. 8 for the legend)

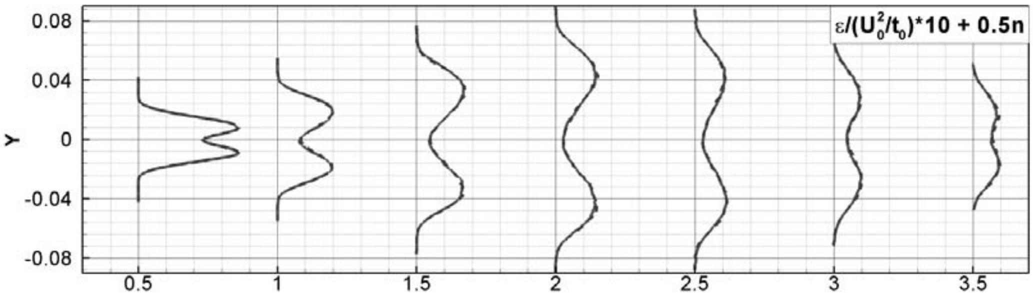

Figure 10. Comparison of profiles of viscous dissipation rate at sections $x_n = (0.1n)$ m, $n = 1, 2,\ldots,7$ computed in narrow and wide domains (see Fig. 8 for the legend)

5. Conclusions and outlook

The paper presents the current (initial) stage of the computational part of the German-Russian project “Wake flows in Adverse Pressure Gradient”, launched in 2017. Results are presented of the first scale-resolving RANS-IDDES simulations of the wake of the flat plate subjected to adverse pressure gradient created by two pairs of thin liner foils at conditions corresponding to the concurrent experimental studies being conducted at the Technical University of Braunschweig [5]. The simulations were conducted with the use of computationally efficient massively parallel version of the in-house SPbPU code NTS [4] with the use of 11 nodes of the high performance cluster “Tornado”. Results have shown that the simulation in the wide domain does not reveal any large-scale turbulence structures, which are not represented in the simulation in the narrow domain. Hence the a further work on the project will be focused on such simulations on the narrow domain at different experimental conditions with an ultimate objective of creating a reliable combined (experimental/numerical) database for calibration and validation of enhanced RANS turbulence model for the considered class of flows.

References

1. Rumsey C.L., Ying S.X. Prediction of high lift: review of present CFD capability // Progress in Aerospace Sciences. 2002. Vol. 38. P. 145-180.
2. Driver D.M., Mateer G.G. Evolution of a Planar Wake in Adverse Pressure Gradient. 2016. *NASA/TM–2016-219068*
3. Shur, M., Strelets, M., and Travin, A., High-order implicit multi-block Navier-Stokes code: Ten-years experience of application to RANS/DES/LES/DNS of turbulent flows. https://cfd.spbstu.ru/agarbaruk/doc/NTS_code.pdf (date of access 15.04.2019)
4. Belyaev K.V., Garbaruk A.V., Shur M.L., Strelets M.K., Spalart P.R. Experience of Direct Numerical Simulation of Turbulence on Supercomputers // Supercomputing. RuSCDays 2016. Communications in Computer and Information Science. 2016. Vol 687. P. 67-77.
5. Breitenstein W., Scholz P., Radespiel R., Burnazzi M, Knopp T., Guseva E., Shur M., and Strelets M. A Wind Tunnel Experiment for Symmetric Wakes in Adverse Pressure Gradients. 2019. AIAA 2019-1875
6. Shur M.L., Spalart P.R., Strelets M.Kh, Travin A.K. Synthetic Turbulence Generators for RANS-LES Interfaces in Zonal Simulations of Aerodynamic and Aeroacoustic Problems // Flow, Turbulence and Combustion. 2014. Vol. 93. P. 63-92.
7. Shur M., Strelets M., Travin A. Acoustically adapted versions of STG // Notes Num. Fluid Mech. and Multidisciplinary Design. 2017. Vol. 134. P. 62-69.
8. Menter F.R. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications // AIAA Journal. 1994. Vol. 32. P. 1598–1605.
9. Shur M.L., Spalart P.R., Strelets M.Kh, Travin A.K. A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities // Int. J. Heat and Fluid Flow. 2008. Vol. 29. P. 1638-1649.
10. Rogers S. E., and Kwak D. An upwind differencing scheme for the incompressible Navier-Stokes equations // Appl. Numer. Math. 1991. Vol. 8. P. 43–64.
11. Guseva E.K., Strelets M.Kh., Travin A.K., Burnazzi M. and Knopp T. Zonal RANS-IDDES and RANS computations of turbulent wake exposed to adverse pressure gradient // J. Phys.: Conf. Ser. 2018. Vol. 1135. 012092
12. Dejoan, A., Leschziner, M.A. Large eddy simulation of a plane turbulent wall jet // Phys. of Fluids. 2005. Vol. 17**.** 025102