WEST-EAST HIGH SPEED FLOW FIELD CONFERENCE
19-22, November 2007
Moscow, Russia

NUMERICAL SIMULATION OF SUB-, TRANS- AND SUPERSONIC FLOW AROUND BODIES WITH VORTEX CELLS IN THE FRAMEWORK OF MULTIBANK COMPUTATIONAL TECHNOLOGIES

Sergey A. Isaev*Alexander G. Sudakov Alexander Eu. Usachov*****
and Paul A. Baranov*

** Saint-Petersburg State University of Civil Aviation E-mail: isaev@si3612.spb.edu, paul-baranov@yandex.ru*
*** Battery Company "Rigel", Saint-Petersburg E-mail: sud@mail.gtk.su*
**** Central Aerohydrodynamic Institute n.a. prof. N.E. Zhykovsky E-mail: usachov_a@mtu-net.ru*

Key words: separated viscous flow, vortex cell, suction, numerical simulation, RANS, MSST, multibank computation technology, VP2/3.

Abstract. A comparative analysis of the predicted and measured data shows that VC RANS/URANS code based on the multi-block computational technology with the use of MSST enables obtaining VC flow characteristics with a reasonable accuracy over a broad range of Mach and Reynolds numbers.
An interesting scheme of slot suction realized by a fan as a controlling tool is analysed by the example of the drag reduction of a circular cylinder in unsteady flow.

1. PRELIMINARY STUDIES

The presented Review summarizes thirty-year experience on **numerical modelling of vortex cell flows**. The generalized vortex cell (GVC) concept introduced in the Review combines flows with vortexes trapped in a number of ways. It has been demonstrated that GVC flows including those in cavities, trenches, dimples, slots and gaps between the bodies both in channels and in open space, with and without central bodies (CB) have common features regardless of vortex cell geometry. Due to that, typical flow structures developing in separating GVC flows have been identified.

The purpose of vortex cells is to improve aerodynamic characteristics of the body by means of flow control preventing its separation. The above-mentioned improvements include reduction of the drag coefficient, increase of the lift and aerodynamic quality coefficients, and creation of the restoring moment (head stabilization effect). The impact provided by *small-scale* elements, such as active vortex cells with suction, has been found to be capable in drastic *large-scale* transformation of the incident flow. As a result, flow separation behind high-drag body could be damped which has been shown in numerical simulations of steady and transient flows around thick EKIP airfoil (designed by L.Schukin) [1].

An important achievement of the long-term numerical studies of VC flows was the development of multi-block gridding methodology that has been implemented in the in-house CFD code VP2/3. The multi-block methodology, originally developed for numerical simulations of separating flows, have been successfully applied in a number of research and engineering problems. Intensive verification and validation studies (simulations of recirculating flows in cavities of different shapes including cavity flows in channels) as well assessment of RANS turbulence models (including those corrected to allow for the mean flow streamline curvature), have been preformed, and comparisons of predictions with measurements are also presented in the Review.

Shear stress transport *k*-ω model by Menter provides good agreement of the predicted velocity profiles and turbulent characteristics with the experimental data. The newer version of the model requires the turbulent viscosity to be corrected to allow for the mean flow streamline curvature, similar to other two-equation models. In the latter case, for the adjustable parameter, C_C, the value of 0.02 is recommended.

It has been found that a *passive* VC is not efficient since the flow intensity inside it is low, as well as its influence on the external flow.

Alternatively, *active* VCs can provide positive effect: they cause the reduced drag, and the drag reduction depends on suction rate or speed of rotation. Quantitative assessment of the VC efficiency should take into account energy requirements to activate the VC. Here, such an assessment was made by introducing equivalent extra drag, C_x^{add}, that depends on the power needed to maintain given rate of suction or speed of rotation. The drag coefficient, C_x, can therefore be corrected (by adding C_x^{add}) to yield the effective drag coefficient allowing for energy losses to activate VC. It has been demonstrated, that optimum rate of suction or speed of rotation exist which provide minimum value for the corrected drag coefficient, and its minimum value is observably less than the drag of a cylinder with no VCs. The dimensionless optimum value for CB suction rate was estimated as $U_{n,opt} \approx 0.034$, and corresponding effective drag is $C_x^{cor}\left(U_{n,opt}\right) = 0.35$. This value is by 53% less than the drag of a cylinder with no VCs.

Further possibility to reduce drag is concerned with proper choice of the VC edge shapes. For example, rounding the downstream VC edge (dimensionless radius is 0.05) led to reduction of drag by 10% compared to sharp edges and by a factor of nearly 2 compared to no-VC cylinder.

The key feature of the active VCs is their capability to damp flow separation, thereby reducing drag coefficient, increasing lift and aerodynamic quality coefficients at a reasonable level of extra energy losses. Detailed numerical simulations demonstrated that the VCs can only positively affect the external flow if they are placed beyond the separating zone behind the body. The other conclusions can be summarised as follows. 1. At a similar suction rate, the method of suction arrangements (either through the surface of central body or of the vortex cell) has a little effect on the flow characteristics. 2. The lift coefficient of the TA with VCs was shown to be quite high, $C_y > 1$, in a wide range of the angles of attack, $-30° < \alpha < +25°$. 3. EKIP and Gettingen airfoils exhibited similar performance for the same suction rate in the VCs. 4. A possible application of thick airfoils with vortex cells in wind blades has been presented.

2. RECENT DEVELOPMENT

Old version of VP2/3 cannot handle unstructured grids. Structured grids are beneficial in terms of efficient memory use and simpler implementation of higher order spatial approximations. In case of complicated geometries, possibility of use of unstructured grids is clearly advantageous. The development of unstructured (as well as structured and hybrid) grid option is therefore the first avenue of the software tool advancement. Also the multi-block grid concept is exploited.

Old version of VP2/3 combines pre-processor, solver, and visualisation means. It is developed to operate under Windows^{TM} platform. The second avenue for code development is to create platform-independent solver that can also operate under Linux or Windows in a multi-processor environment. Thus the pre-existing software (the RANS code VP2/3) is being adapted for modelling VC flows using parallel Linux and Windows clusters.

Extension of available incompressible flow solver to compressible flows is made.

Not only steady, but also unsteady 3D flows have been investigated, and new experimental data on VC flows have been used for further model and code validation. New experimental data on VC flows has recently been obtained at the Institute of Mechanics, Moscow State University. The novelty of the data is due to the suction through the central body inside the vortex cell. For the purpose of model validation, and for better understanding flow structure and dynamics we have performed 2D RANS simulations of the experimental flows. The conclusion from this study is streamline-curvature-modified MSST model yields reasonable agreement with the measurements.

The second group of methodical results is concerned with the RANS simulations of 2-3D and unsteady VC flows. Some simulated incompressible flows are considered: a. 3D flow around HALE setup at *CIRA* (airfoil in the channel, no VC); b. Transient vortex shedding in 3D flow around plane-mounted bluff-bodies; c. 3D unsteady flow around a circular cavity in the channel at IM MSU; d. Transient vortex shedding in 2D flow around NACA0015 airfoil and semi-cylinder (at α=0). The unsteady RANS approach is shown to be robust and informative numerical tool for predictions of flows with trapped vortexes.

A series of studies has been undertaken to investigate compressibility effects including those in vortex cell flows. Sub-, trans-, and supersonic flows around sphere, cylinder, tear-shaped body are considered [2]. The objectives of this study are: a. To develop and incorporate the all-speed pressure correction algorithm into the existing VC RANS code, originally developed as incompressible; b. To verify generalised pressure-correction procedure; c. To validate the MSST model for trans- and super-sonic flows. Choice of flows to study was determined by the experimental data available. The latter come from two types of experiments: a. Wind tunnels; b. Aero-ballistic setup. A very good agreement has been obtained for a number of separated flows thereby validating the developments into VC RANS code.

The turbulent air flow in the plane-parallel channel (Figure 1) with a circular cavity is considered as a test problem for approbation of a parallel code. The calculating multi-block grid (figure 2) shares on 8 fragments and contains 500 thousand calculation cells. Claster loading is distributed on four processors in regular intervals. The test problem is solved in two stages. First RANS is used to obtain preliminary calculation fields (4000 iterative steps are done). Then URANS is started and the formation of a cyclic vortical flow is simulated with the time step 0.02.

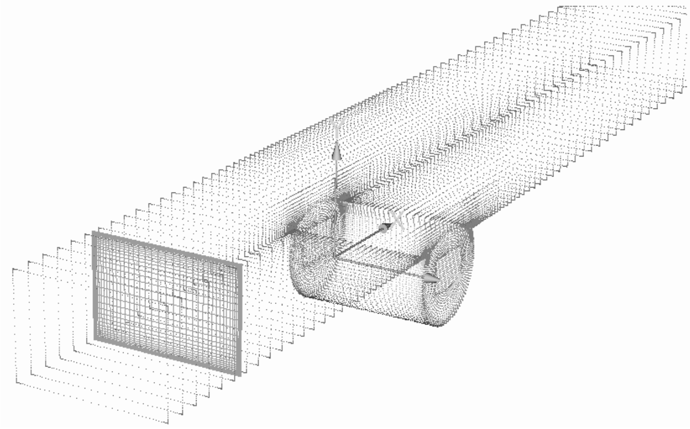

Figure 1. The plane-parallel channel with a vortex cell schematic.

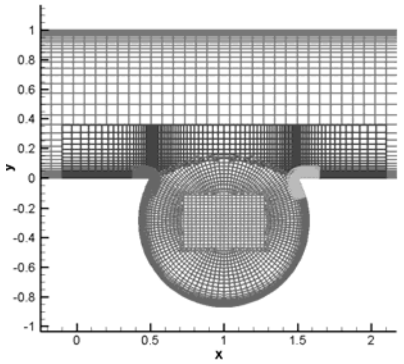

Figure 2. Multi-block grid in vicinity of a circular cavity.

It is shown, that obtained calculation results on new (parallel) and old (uniprocessor) versions of a code are coordinated among themselves. It is found, that acceleration of URANS solution at use of a parallel code makes 24 in comparison with an old uniprocessor variant. One time step on 4 processors cluster borrows for 19 seconds.

3. LAST RESULTS

From the considered set of problems it is expedient to allocate the following ones.

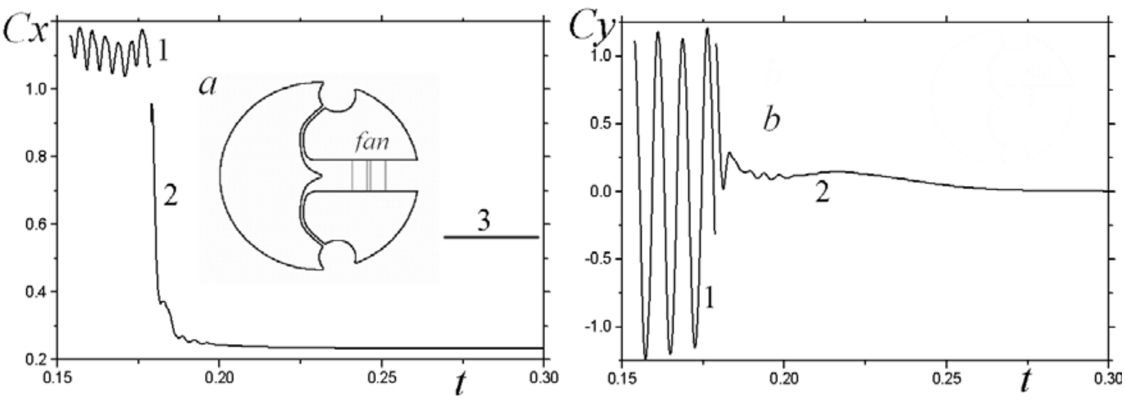

Figure 3. Evolution of drag (*a*) and lift (*b*) coefficients during the time for the cylinder without (1) and with (2,3) the fan. 3 – C*x* certain in view of power losses. Re=4×10^4

3.1. A vortical cell in a contour of the circular cylinder with fan

As known, a distributed and concentrated suction in a vortical cell in a contour of the circular cylinder at presence of a dividing plate in a near wake results in essential drag reduction. The system of slot-hole suction creation is considered due to the use of the fan in the internal channel of the cylinder (Figure 3,*a*). Pressure difference on the fan (0.0125 atm) is selected from a condition of the achievement of small value of discharge factor Cq (about 0.06) when power expenses are insignificant. Apparently from the received results, installation of the fan stabilizes a flow around the cylinder and twice reduces *Cx* (Figure 3,*b*).

3.2. A turbulent air flow in the channel with the circular cavity

The solution of test problem about a turbulent air flow in the channel with the circular cavity, obtained with the use of parallel code VP2/3, has allowed better to understand the flow formation in a vortical cell (Re=2×10^5).

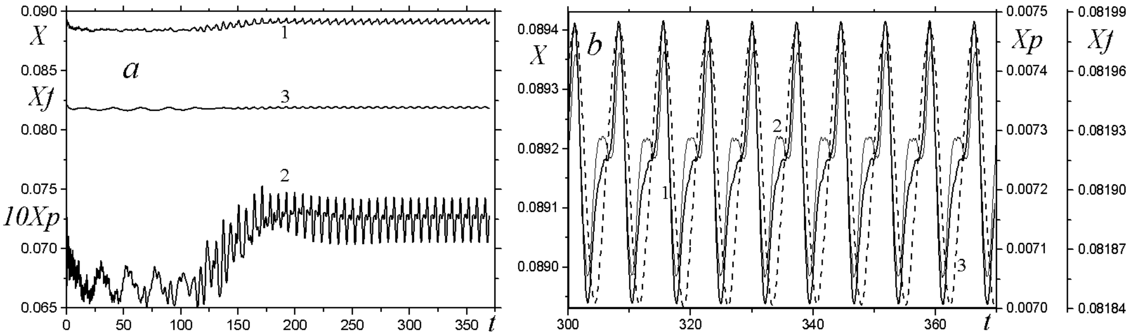

Figure 4. Dependence on time of longitudinal force (1) and its components (2,3). *a* - all range; *b* - periodic process

The self-oscillatory flow regime in the channel is established (Figure 4). The period of fluctuations of cross and longitudinal loading T = 14.6 which corresponds St=0.0685. Pressure field averaged on the period of fluctuations is symmetric. Air flow picture on lateral walls has special points such as focus from which tornado-like jets expire. The twirled jets cooperate in a median plane of the channel (figure 5,*a*). The field of pressure pulsations on walls of a cavity breaks up to four homogeneous zones (figure 5,*b*).

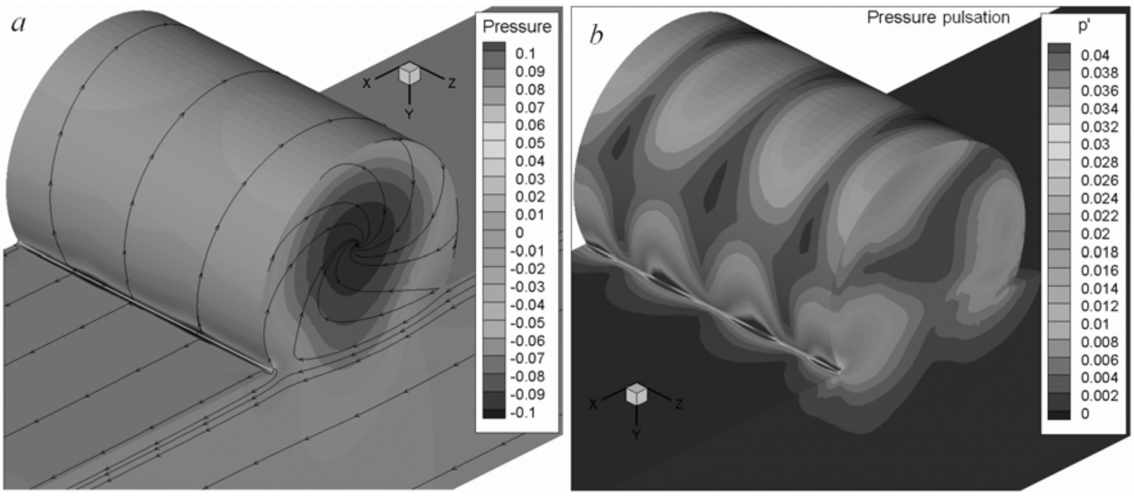

Figure 5. Flow pattern on walls of the channel (*a*) and field of pressure pulsation (*b*)

3.3. Compressible flow around thick EKIP airfoil

After test simulations of flows around the cylinder, compressible flows around thick EKIP airfoil is considered. Some results obtained are presented on Figure 6. Reynolds number is given as 10^5 and volume suction coefficient – 0.022.

The predictions obtained by the compressible RANS code can be summarized as follows. 1. The critical Mach number and VC failure mechanism is similar to that of cylinder flows. 2. Not only drag but also lift suffers if M > Mcr. 3. At higher M, relocating VCs upstream is effective. It implies that different geometry might be optimum for different velocity ranges. 4. Oscillating flow is predicted for M > 0.45, when VC periodically enters or exits the separated zone.

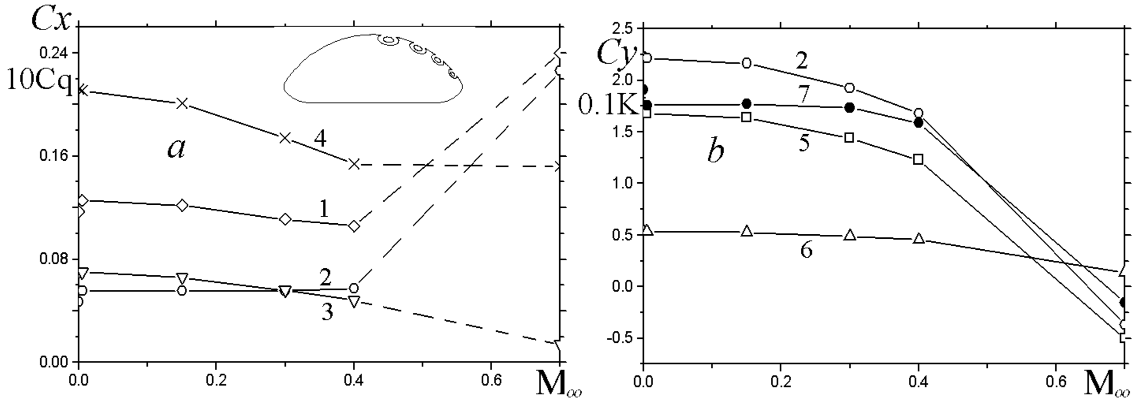

Figure 7. Dependences of drag (a) and lift (b) coefficient and also its components for EKIP airfoil, aerodynamic quality (b) and discharge factor (a) from M_∞. 1- C_{xint}; 2 - C_x; C_y; 3 - C_{xadd}; 4 - $10c_q$; 5 - C_{ypr}; 6 - C_{ycell}; 7 - 0.1K

4. CONCLUSION

A fair agreement of the oscillation period-averaged integral and local characteristics of flow around bodies, to which the 2D statement can be applied, shows that URANS is acceptable for description of periodic vortex processes.

The SST turbulence model by Menter earlier validated for incompressible flows has now been shown capable in reproducing compressible flows. The energy equation needed for compressible flow simulations is most convenient to be formulated for enthalpy.

A comparative analysis of the predicted and experimental results testifies that the VC URANS code based on the multi-block computational technology with the use of MSST allows obtaining vortex cell flow characteristics with a reasonable accuracy.

Critical Mach number (about 0.4-0.5) exists beyond which the aerodynamic characteristics rapidly deteriorate, VC is no longer effective, and the flow structure changes. The mechanism of such a VC failure is suggested. Higher suction rate and VC relocation helps to increase the critical Mach number. The effects of suction arrangements (distributed, localized) are significant and must be considered.

The work has been supported by the Russian Foundation of Basic Research on the Projects 06-08-81002, 05-01-00162 and European Project (FrameWork 6) VortexCell2050.

5. REFERENCES

[1] P.A. Baranov, S.A. Isaev, Yu.S. Prigorodov and A.G. Sudakov, “Controlling the turbulent flow past a thick airfoil by means of flow enhancement in a vortex cells using suction from central body surfaces”, *Fluid Dynamics*, 38, No. 3, 387-396 (2003).

[2] S. Isaev, A. Mikhalev, A. Sudakov and A. Usachov, “Simulation of turbulent flow around a tear-shaped body with a tapered flare”, *Technical Physics*, 52, No.8, 991-997 (2007).