ON THE AERODYNAMIC CALCULATION OF HIGH-SPEED GROUND TRANSPORT VEHICLES

O. Prykhodko, A. Sokhatsky

Institute of transport systems and technology,

Ukrainian National Academy of Science

Further development of the ground transport calls for solution variety of problems among which aerodynamic problems are very important. The up-date state of high-speed ground transport problem shows that the use of aerodynamic effects will make it possible to optimize the technical and economic performances of vehicles.

A research conducted has shown that operation of the advanced high-speed transport vehicle with superconducting magnets must be based on a dynamic air cushion principle [1]. The effect is achieved if the vehicle is equipped with an aircraft wing.

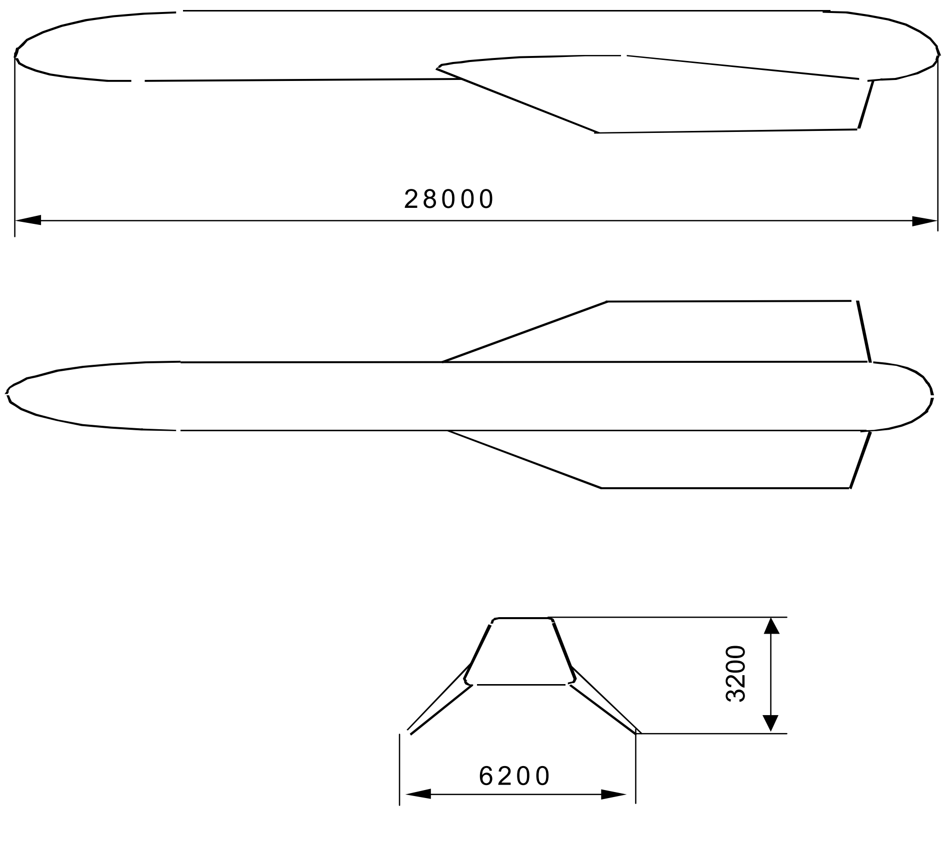

Fig.1. Aircraft-shaped high-speed transport vehicle

When such a vehicle is moving in ground proximity, the air under it's wing is dragged and pressure is increased. As a result, an air cushion and addition lift are formed affecting the wing and the whole vehicle. It gives a scope for applying the magnets with less power and for reduction the electricity expenditure, for improving the traffic stability and safety.

Early aerodynamic ground transport investigations were in the main empirical. Theoretical approaches weren't in the wide use because they allowed to investigate the flow processes for the simplest in design geometries only. That is why mathematical and experimental investigations of flows around the transport vehicle became especially actual. Mathematical simulation gives opportunity to investigate mechanism of the phenomena which are sometimes disappeared from an explorer. The importance of mathematical simulation rises with the development of personal computers and with the improvement of numerical methods and models which are in use. Complementing each other and competing at the same time calculation and experiment give to explorers new opportunities for investigating the complex interdependent processes.

The turbulence model, adequate to physical process, is an important factor for successful numerical calculation of turbulent flows. A lot of works are devoted to the problem. Some of them consider the models based on the averaged Navier-Stokes equations involving the Reynolds stress tensors.

The known turbulence models are subdivided in two types; mean velocity field models and averaged turbulence characteristic field models. Closed models, based on averaged turbulence characteristics are subdivided into the Reynolds stress field models and the kinetic turbulence energy field models.

The experience has shown that above mentioned turbulence models can be applied to the simple flows. In the case of complex flows parameter distribution in the turbulent flow considerably differs from the results received with the use of simple semiempirical turbulence models. Semiempirical turbulence models, based on the concept of mixing path length and turbulent viscosity, suppose the equilibrium of boundary layer structure, while in each point the averaged flow generation of turbulence energy should be equilibrated by dissipation.

It is assumed in the Reynolds stress turbulence models that the turbulence stresses are proportional to mean velocity of deformation. At the time exact expression for defining the relation between Reynolds stress tensor and distribution of the averaged flow parameters is not found.

Aerodynamic characteristics are received for the relative distances h=0,016; 0,024; 0,032; 0,04; 0,052 (where h=H/b,and H is a distance to the ground; b is the airfoil chord. The angles of attack $\alpha=-0.5^\circ, 0^\circ, 0.5^\circ, 1.0^\circ$. Relationships between lift coefficients, drag, longitudinal moment, aerodynamic quality and the angle of attack and also distance to the ground are established (fig.2).

The experimental results have shown that lift is increased when angle of attack is increased. When the distance to the ground decreases the lift coefficient increases. The drag rises when the airfoil approaches to the ground. When angles of attack are negative the force, affecting the airfoil, is negative and it is holding the airfoil down to the ground. Derivative $\partial C_m / \partial\alpha$ has a negative value in the investigated range of distances to the ground and angles of attack. It indicates the longitudinal stability. When the airfoil approaches to the ground, the area of increased pressure on its subsurface is extended. As a result a longitudinal moment is increased.

Flow visualization in the airfoil leading edge area has showed that in the tested range of distances to the ground and angles of attack flow separation does not occur.

Among the factors governing a quality and efficiency of viscous flow numerical simulation, the statement of boundary conditions for the flow parameters, especially for turbulent parameters, is very important.

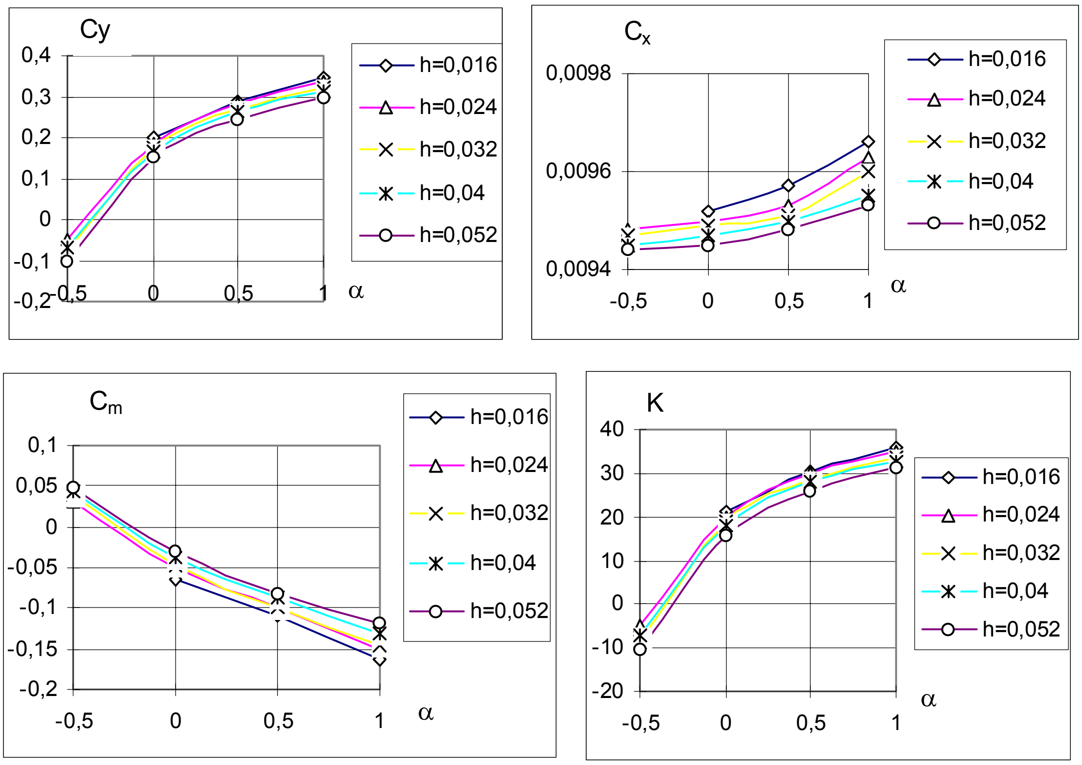

Fig. 2. Lift coefficients, drag, pitching moment and aerodynamic quality for the CLARK-Y-4% airfoil against the angle of attack

In terms of numerical method practical applications the grid construction is a key problem in computational hydrodynamic. To solve the problem it needs to take two important decisions: to choose a type of the grid and to choose a method for a grid construction. There are two types of methods for grid construction: algebraic methods and differential ones. Both of them have their own advantages and limitations. Differential methods don't allow to get such a grid orthogonity as in the case when algebraic methods are used. The main advantages of algebraic methods are good checking of entire grid point distribution, especially while constructing the grids local-orthogonal to the boundaries. It can be explained that algebraic methods require less computer memory and computer time. Nevertheless the type of a grid must be chosen according to the particular problem.

The corresponding grid design techniques based on algebraic relations and on the solution of differential equations are developed by the authors.

2. It is thought that experimental investigations give the most true solutions for many hydroaerodynamic problems. But it isn't always simple to simulate the real flow conditions. At present time the following methods are used for simulation of ground effect in a wind tunnel: fixed plate, inverted image, boundary layer control, moving belt. But no one of these approaches gives opportunity to realize a real flow pattern next to the transport vehicle [1]. Thus, during modeling the flow in the ground proximity the number of difficulties appear, which are related to the realization of actual flow conditions. Ideally there must be a relative movement between the ground and vehicle model. It is difficult to simulate this physical process in a wind tunnel adequately.

To estimate an effect of the vehicle elements on its aerodynamic performances the experimental investigations of flow around an airfoil in ground proximity were carried out. Flow singularities and aerodynamics of CLARK-Y-4% airfoil were investigated in a closed - layout wind tunnel T-3 of Kharkov Aviation Institute. The tunnel had an open working section. The experiments were carried out with the use of inverted imagine method at small distances to the ground. The main object was to study the flow in ground proximity and to get the date for a testing and for the computational techniques developed.

3. The main factors, giving rise to the progress in computational hydroaerodynamic and heat transfer are: level of fluid-gas mechanics models, efficiency of numerical algorithms, quality of computational grid construction methods, computer power. Optimum mathematical model can considerably reduce requirements to the computes resources and that is very important in terms of practical application. The complete loop of mathematical simulation consist of the stages: physical phenomenon; mathematical model program for computer simulation; computational experiment. In so doing, original mathematical model and numerical technique are of crucial importance. That is why the second and the third stages of simulation process are stressed by the authors.

There are four main stages in the development of computational aerodynamics. At each following stage more complete approximation the Navier-Stokes equations is used:

I – analytical approximation and linearised equations;

II – gas-hydroaerodynamic equations without accounting of dissipative effects;

III – averaged complete and simplified Navier-Stokes equations;

IV – nonaveraged Navier-Stokes equations.

The simplest methods for the perfect fluid computation are based on the assumption of potential flow. They are subdivided into two classes: singularity methods and field methods. Singularity methods, which are also known as boundary equation methods, are based on the use of different analogues of the Green identities: solution in an arbitrary chosen point of the flow area can be expressed through the integrals of dependent variable distribution and are also through the integrals of its derivatives. Such approach is used for designing the deflected flap airfoil, moving in ground proximity. Oppositely, in the field theory methods for the solution in an arbitrary point of the flow area if is necessary to know solution not only on the boundary, but also in the middle of the calculated area. Finite difference methods, finite volume-and finite element methods refer to this class of methods.

It is thought that the models based on Navier-Stokes equations, map onto the physical process of real fluid flow in the best way. Regular development of the flow calculation technique, based on the compressible viscous gas equations throughout the computational domain began at the end of the last century. However in many practical cases the flow can be considered as incompressible. Variety of objects and problems, arising ahead of investigators, requires him to be oriented into the whole model range beginning with the simplest approximations and ending with complete Navier-Stokes equations

$$\frac{\partial q}{\partial t}+\frac{\partial E}{\partial x}+\frac{\partial F}{\partial y}+\frac{\partial G}{\partial z}=H_q ,$$

where $q=\{u,v,w,p,k,\varepsilon\}^T$; E,F,G - is a combined convection – cum - diffusion flows, Hg –is a source term.

Construction the optimum mathematical model can considerably reduce a level of requirement to the computer resource and that is very important in terms of its practical application.

Mathematically, the flows in ground proximity are of wide field for the design of models and methods, describing variety of phenomena and physical effects. Presently during investigating a wide range of physical phenomena the problem how to set up the applicability limits and to verify the mathematical models and numerical techniques in use is posed more and more actively [2-5]. The actual problem is to develop for application such simple mathematical models, which would allow to study complex physical phenomena with satisfactory accuracy.

Using the perfect fluid model the present work takes the discrete vortex approach (extended in [2,3]) as a base for the algorithms developed. Flows around an airfoil in ground proximity are calculated which are characterized by large Reynolds numbers. Viscous effect in the flows is substantial only in the thin layers, in the main domain it is negligible. Mathematical simulation of flow was boiled down to the definition of discrete vortex circulation at both the main and imaged airfoil. Using the values received, velocity and pressure at any point of the plane outwards the airfoil and on it's surfaces were calculated. The aerodynamic coefficients were defined by integration over the airfoil surface.

Based on the program, algorithm and technique developed, numerical calculations of flow around the CLARK-Y and CLARK-YH airfoils were made for the case of infinite flow around the body in ground proximity.

Ground distance and wing tilt angle effects on the aerodynamic performances were investigated (fig.3). The analysis and comparison with experimental date are given [6].

4. Numerical simulation of near-ground flows on the base of the Navier-Stokes equations. In the last few years the models and numerical techniques, based on the Navier-Stokes equations, receive wide acceptance in theoretical investigations of different physical phenomena. The main problems in numerical solution of the Navier-Stokes equations are related to the way of representing them in the adaptive

curvilinear nonorthogonal coordinate system, to the construction of the computational grid, to the selection of differential approximation of the assumed equations, to the setting up of boundary conditions , to the field pressure calculation, to the testing of the algorithms developed and also to the estimation of scheme effects.

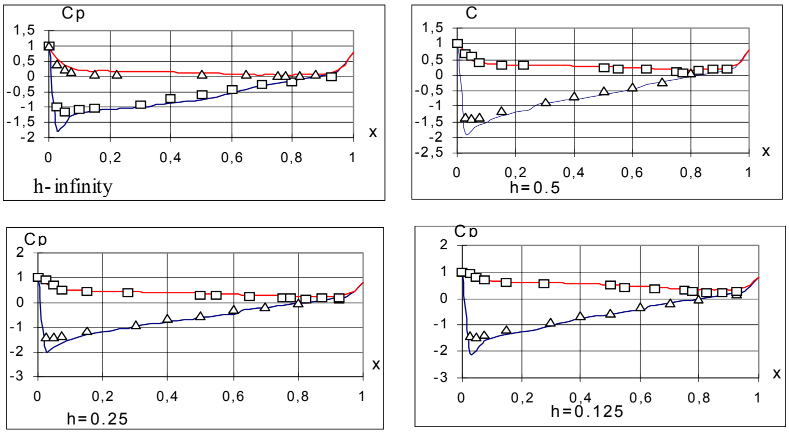

Fig. 3. Pressure coefficient distribution over the airfoil in ground proximity:

___ - calculation, □,Δ - experiment [6]

At statement the viscous incompressible flow problems on the base of the Navier-Stokes equations for laminar and turbulent regimes physical variables and variables "vorticiti-flow function" are used. Two parameter (k-ε) turbulence model is applied for closing the averaged Navier-Stokes equations. At numerical simulation on the basis of solution the differential equation systems in partial derivatives with the use of finite-difference methods the choice of the algorithms for transformation the assumed equations in partial derivatives into the system of algebraic equations is the most important stage. The techniques, algorithms and programs for the solution of Navier-Stokes equations, written in curvilinear nonorthogonal coordinate system were developed with the applications of the following schemes: contra-flow scheme; hybrid-scheme; Leonardo scheme of the second and the third orders of approximation; upper relaxation scheme; predictor-corrector scheme with the time splitting by the variable direction - matrix factorization method.

One of the problem, arising during solution the incompressible viscous flow Navier-Stokes equations in the "vorticity-flow function" variables, is related to the calculation of pressure field by the known velocity field or flow function. Possible approaches for the pressure field calculation were examined and four techniques for the estimation of a pressure field in the curvilinear nonorthogonal coordinate system with the use of Poisson equations were developed and realised: for pressure and Bernulli variable, for the direct integrating of momentum equations, for SIMPLER technique [1].

Boundary conditions on the streamlined surfaces and a ground are realized with the use of several techniques [1].

Testing of the techniques developed is carried on the problem related to the development of the laminar flow in a cave and to the flow around the isolated cylinder. For the turbulent flow regime the flows around a plate and an isolated cylinder are calculated.

Every technique developed was realized in the integrated packet of applied program. Some results for the flow around the lifting system components (airfoil and circular cylinder) in ground proximity are presented below. In the fig. 4-5 comparison of the result with experimental data and with the calculations of other authors is given for a circular cylinder and an airfoil [7-8].

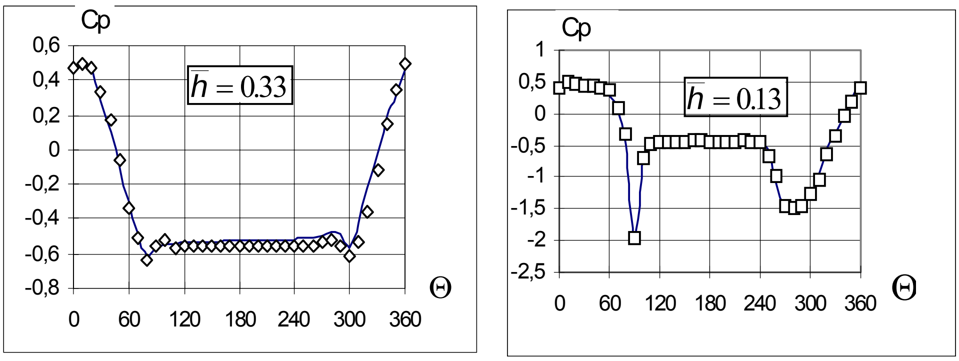

Fig. 4. Pressure distribution over the cylinder in ground proximity:

- ◇calculation, , □ - experiment [7-8]

Under condition of flow around the cylinder in ground proximity two asymmetrical vortexes are formed in the wing wake during which the upper one has more proper form and larger sizes. It is due to the slit effect in the place of the smallest clearance between the cylinder and the ground. Velocity flow past the cylinder at the ground surface in much higher than in the undisturbed mainstream.

Because of that some amount of fluid, being introduced into that part of flow, is carried out of the base of the cylinder lower part . Forward and backward stagnation points are displaced to the ground. Negligible displacements of the separation points can be explained by the stabilizing effect of the ground on aerodynamic wake. The ground decelerates the eddy blobs movement at once behind the cylinder and thus stabilizes the location of the separation points.

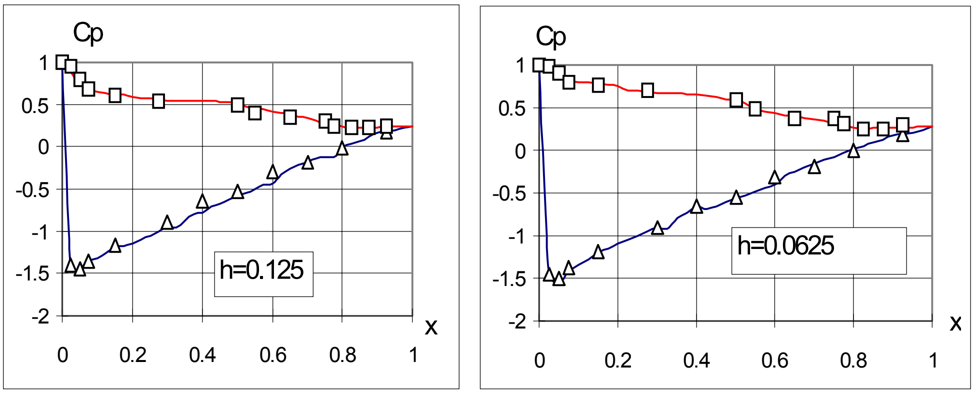

Fig.5. Pressure coefficient distribution over the airfoil in ground proximity.

□,Δ- experiment [2], ___ – calculation.

5. Conclusions

Flow properties for transport system elements (components) in ground proximity are experimentally and numerically investigated in the study. Discrete vortex method and incompressible Navier-Stokes equations are used for the calculating of laminar and turbulent flow regimes. Results for the flow around a cylinder and an airfoil in ground proximity are presented. Angle of attack- distance to the ground dependence of aerodynamic coefficients are received.

Calculations when used in balanced combination with wind tunnel experiments can considerably reduce time and funds expense under developing the new transport vehicles.

Numerical models constructed, algorithms and program realized can be used during transport vehicle designing and also during perfecting the fluid-gas mechanics models and methods.

As computer speed becomes higher and computer storage becomes lager and also as continuum mechanics models and methods become more perfect it is needs to develop methods, algorithms and applied program packets for aerodynamic calculations with the use of Navier-Stokes equations.

The further investigations will be related to the choice of optimum configuration for the transport vehicle with superconducting magnets.

References

1. Prykhodko O.A., Sokhatsky A.V. Mathematical and experimental simulation of transport vehicle aerodynamics in ground proximity. - Dniepropetrovsk: Science and Education , 1998. – 154 p.

2. Belotserkovsky S.M. Nisht M.I. . Separated and nonseparated flow of the ideal fluid around thin airfoils . - M.: Science, 1978.-276 p.

3. Belotserkovsky S.M., Ginevsky A.S. Simulation of turbulent jets and trails with the use of discrete vortex method . - M.: Publishing corporation of physical and mathematical literature, 1995. - 368 p.

4. Belov I.A., Isaev S.A., Korobkov V.A.Calculation of the separated incompressible flows. Problems and Methods. - M.:Sudostroyeniye , 1989.- 256p. with.

5.Phletcher K. Computational techniques in fluid dynamics. In II volumes. - M.: Mir, 1991.- V. 1. – 501p.- V. 2. – 552p.

6. Serebriysky J.M., Biyachuyev Sh. Tube investigation of the horizontal steady flow around the wing in ground proximity. // Transactions of CAGI .-Issue 437.- 1939.- 32p.

7. Kovalenko V.M., Bichkova N. M., Kisel G.A., Dikovskaya N.D. Flow around the rotational and fixed cylinder in a plane ground proximity. Information 1 // Izvestiya SD AS USSR. Techn . Sci. Series.- 1983.- №13, issue 3.- p.50-59.

8. Kovalenko V.M., Bichkova N.M., Kisel G.A., Dikovskaya N.D. Flow around the rotational and fixed cylinder in a plane ground proximity. Information 2 // Izvestiya SD AS USSR. Techn . Sci. Series.- 1984.- №4, issue 1.- p.78-88.