UDC: 519.63

A.V. Sokhatsky, Dr of Science
(Institute of transport systems and technology,
Ukrainian National Academy of Science)

On the aerodynamic calculation of high-speed ground transport vehicles

The up-date state of problem, concerned to t development of high-speed transport vehicles with the use of aerodynamic effects is analyzed. To solve the problem it is necessary to improve the continuum mechanics models and methods, to develop computational techniques, algorithms and applied program packets for aerodynamic calculations. The emphasis is laid on the use of numerical methods based of Navier-Stokes equations.

Introduction

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 [1-6]. The effect is achieved if the vehicle is equipped with an aircraft wing.

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.

Methods of numerical simulation

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 semi empirical turbulence models. Semi empirical 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.

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.

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.

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.

For uniting of initial equalizations the method of complete volume is applied. Shorting of the system of equalizations is executed with the use of oneself-reactance differential model of turbulence of Spalarta- Allmarasa [6].

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.. 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.

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

6. Spalart P.R., Allmaras S.R. A one-equation turbulence model for aerodynamic flows // La Recherche Aerospatiale. – 1994.- N1.- P.5-21.