The Numerical Simulation of the Flow of Vehicles by Turbulent Flow

Anatolii Sokhatskyi [0000-0002-3593-6517]

University of Customs and Finance 2/4 Vernadskoho str.Dnipro, Ukraine
Institute of Transport Systems and Technologies of the National Academy of Sciences of Ukraine .5 Pysarzhevskoho St. Dnipro. Ukraine

Abstract. The creation of advanced transportation vehicles moving on new physical principles is an important task for designers. The turbulent state of the atmosphere and the instability of the kinematic parameters of the air environment complicate the problem. To date, mathematical modeling of turbulent flows remains one of the most challenging problems in fluid and gas mechanics. Reliable prediction of the characteristics of turbulent flows is an extremely important scientific problem. Further search for ways and development of new methods is necessary. A methodology, algorithms, algorithms, and a set of programs for solving the problem of vehicle aerodynamics have been developed.
The software package is written in the Fortran-95 programming language. The Reynolds-averaged Navier-Stokes equations were used with the Spalart-Allmaras turbulence model in the DES implementation. A number of problems were solved numerically. The calculations and their comparison with the results of experimental studies have confirmed the efficiency of the methodology and the developed set of programs.

Keywords: Aerodynamics of transport vehicles; Numerical modeling; Navier-Stokes equation; Trbulence models.

1 Introduction

Creating transportation vehicles based on new physical principles and improving the developed ones is an important task for designers. Today, there is a problem of creating surface vehicles. The influence of a close interface on aerodynamics and motion dynamics is an insufficiently studied issue. The presence of a turbulent environment and the instability of the kinematic parameters of the air environment complicate the problem of ensuring the specified movement of the vehicle. In addition, mathematical modeling of turbulent flows remains one of the most difficult problems in fluid and gas mechanics. Reliable prediction of the characteristics of turbulent flows is an extremely important scientific problem. This is due to the complexity and insufficient study of turbulence as a physical phenomenon.

The paper deals with the problems of constructing a mathematical model, numerical method, algorithm for solving the problem, and software development for studying the aerodynamic characteristics of vehicles moving near the interface. The possibilities of using the Reynolds-averaged Navier-Stokes equations with the use of empirical turbulence models are evaluated. A methodology, algorithms, and a set of programs for solving the problem of aerodynamics of vehicles moving near the interface are developed. Examples of numerical solutions using personal computers are given. Preliminary studies show that the presence of a closely spaced interface has a significant impact on the flow characteristics around the vehicle. The development of mathematical models is necessary for further research.

2 The state of modeling of viscous turbulent flows

One of the most difficult problems in creating high-speed ground and surface vehicles is the task of finding a rational aerodynamic layout. The movement of such a vehicle at high speed occurs near the interface in atmospheric conditions close to the parameters of the standard atmosphere at sea level. The proximity of the water surface contributes to the formation of the screen effect, a phenomenon that increases lift, reduces the inductive component of drag, and changes the aerodynamic momentum dependencies. Due to the screen effect, a high value of aerodynamic quality of winged surface and ground vehicles can be achieved.

Aerodynamic processes have a decisive influence on the technical characteristics of vehicles. To ensure a given mode of movement of a vehicle, it is necessary that its aerodynamic, geometric, mass, strength and dynamic parameters are in a certain range, and their time derivatives have the required values. The solution to this problem is to conduct a whole range of aerodynamic studies to provide the vehicle with a rational aerodynamic layout.

2.1. Methods of modeling turbulent flows

Methods of modeling turbulent flows can be divided into three groups with a certain degree of conventionality: approaches based on the use of the Reynolds Averaged Navier-Stokes equations (Reynolds Averaged Navier-Stokes - RANS); two classical approaches - Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES); hybrid approaches based on the joint use of RANS and LES approaches for different flow regions (see Fig.1) [1-3].

Today, the most common approaches are those based on the use of Reynolds Averaged Navier-Stokes equations (RANS). They are closed with the help of one or another semi-empirical turbulence model [4-6].

The classical vortex separation approaches are the most advanced. These are direct numerical simulation of turbulence (Direct Numerical Simulation - DNS) and the method of large eddy simulation (Large Eddy Simulation - LES). The DNS method is based on the direct numerical solution of three-dimensional unsteady Navier-Stokes equations with the distinction of all spatial and temporal scales of turbulence. It is based on the physical principles of aerodynamics and is completely free from empirical assumptions

In the LES method, the same equations are solved immediately after their preliminary spatial filtering. This makes it possible to exclude some of the spatial and temporal scales from consideration. This operation significantly reduces the requirements for spatial and temporal resolution. This reduces the requirements for the necessary computing resources.

To take into account the influence of the filtered ("subgrid") turbulence scales, one or another semi-empirical model is used. In the scientific literature, to emphasize the fundamental differences between the LES method and the approaches used to close the RANS, they are called "subgrid".

The third group includes hybrid approaches based on the joint use of RANS and LES approaches in different areas of the flow. They are the most common for practical use, based on the capabilities of computing technology. In accordance with this approach, the calculation of turbulent compressible fluid flows is performed by directly solving the Navier-Stokes equations averaged over Reynolds for density and pressure.

Regardless of the nature of the averaged flow, its dimensionality, and stationarity or nonstationarity, it is necessary to solve the three-dimensional unsteady Navier-Stokes equations. This is due to the fact that turbulence is a fundamentally three-dimensional and unsteady phenomenon. It should be noted that for a number of models, such as DNS, it is necessary to ensure sufficient accuracy in distinguishing all spatial and temporal scales of turbulence.

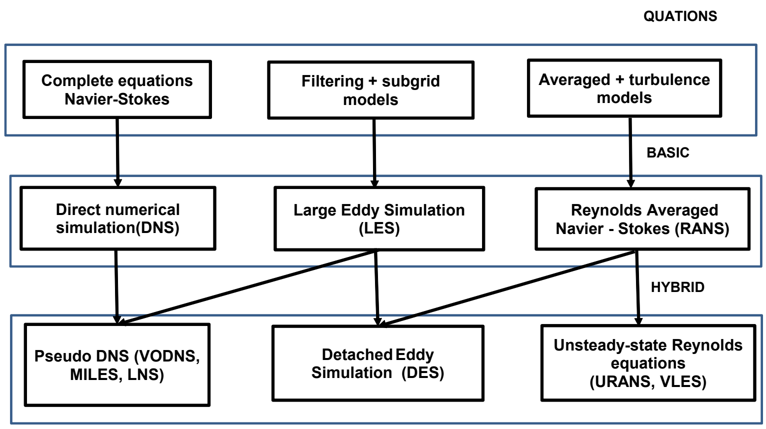


Fig. 1. Levels of turbulent flow models

The advantage of the RANS equations over the original Navier-Stokes equations is that they are formulated directly with respect to the time-averaged flow characteristics that are of primary interest in aerodynamic problems. Thus, it is possible to eliminate the need to calculate local unsteady characteristics of turbulent flows by integrating three-dimensional unsteady Navier-Stokes equations (DNS). In real flow conditions, at the present time of development of computing technology, the calculation of local unsteady characteristics of turbulent flows is considered absolutely impossible. It should be noted that the RANS equations are open-ended, since they contain an unknown Reynolds stress tensor and a turbulent heat flux vector. Therefore, for their practical use, additional relations are required to relate these quantities to the characteristics of the averaged motion. They can be obtained only with the use of some empirical information. These relations are commonly called turbulence models for Reynolds stresses or second moments.

An alternative approach to solving the problem of closure of the Reynolds equations is to use the Reynolds stress transfer equations, which can be formally derived from the Navier-Stokes equations by using a time averaging procedure similar to that used in the derivation of the Reynolds equations. However, these equations contain the so-called third-order moments ($\overline{u_i' u_j' u_k'}$). Their relationship with the parameters of the averaged motion and the components of the Reynolds stress tensor (second-order moments) is unknown, and their determination requires the use of transfer equations for the third-order moments. These equations, in turn, contain fourth-order correlations, etc., so that it is impossible to obtain a strict closed system of equations regarding the statistical characteristics of turbulence in principle. In this situation, it seems a reasonable compromise to limit the modeling to the second-momentum transfer equations.

In [1], the required computational resources were estimated and the prospects for the practical application of various approaches to modeling turbulent flows were outlined.

Readiness means the ability to calculate one option within a day on the most powerful computers available. This means LES with wall-side RANS modeling; in the case of LES up to solid walls, the costs are comparable to those of DNS. Readiness means the use of a computer with a performance of one teraflop, the calculation time is 5000 years!

Significant progress in the construction of various semi-empirical turbulence models was achieved in the 60s and 70s of the last century. They gave false hope for the creation of a universal RANS model that could be suitable for calculating any, at least most, turbulent flows.

During the second half of the last century, scientific institutions conducted numerous experimental and computational studies of turbulent flows. They have convincingly shown that local averaged characteristics of turbulent flows are subject to a significant global influence of stable, large-scale, with dimensions of the order of the macro-scale of the flow, fundamentally three-dimensional and unsteady structures.

The characteristics of these structures depend on the specific geometry of a given flow and boundary conditions. Thus, the hypothesis of localization and averaged characteristics of turbulent flows, on which RANS turbulence models were supposed to be based implicitly, is not fulfilled.

This, in principle, makes it impossible to construct an ideal model of this type and makes the above-mentioned hopes for the possibility of constructing a universal RANS turbulence model essentially unrealizable. This statement applies equally to simple models based on the Boussinesq hypothesis of a linear relationship between the Reynolds stress tensors and strain rates, as well as to models of Reynolds stress transfer.

The most striking examples of flows characterized by the formation of coherent turbulent structures with dimensions of the order of the streamlined body dimensions are flows with large separation zones. It is for this reason that the results of calculations of such flows using RANS models are usually unsatisfactory

Table 1. Computational resources and prospects for the practical application of various approaches to modeling turbulent flows [5].

Method	The number of grid nodes required	The required number of steps in terms of time	Readiness *
3D Steady RANS	10^7	10^3	1985
3D Unsteady RANS	10^7	$10^{3,5}$	1995
DES	10^8	10^4	2000
LES**	$10^{11,5}$	$10^{6,7}$	2045
DNS	10^{16}	$10^{7,7}$	2080***

3 Statement of the research problem

The averaged Navier-Stokes equations (RANS equations) for a perfect compressible gas in a curved coordinate system are written as follows:

$$\frac{\partial \hat{Q}}{\partial t} + \frac{\partial\left(\hat{E} - \hat{E}_v\right)}{\partial \xi} + \frac{\partial\left(\hat{F} - \hat{F}_v\right)}{\partial \eta} + \frac{\partial\left(\hat{G} - \hat{G}_v\right)}{\partial \zeta} = \hat{H} \,, \tag{1}$$

where $\hat{Q}$ – is a vector of unknown variables; $\hat{E}, \hat{F}, \hat{G}$ – vectors of inviscid flows; $\hat{E}_\text{v} = \xi_x E_v + \xi_y F_v + \xi_z G_v$, $\hat{F}_\text{v} = \eta_x E_v + \eta_y F_v + \eta_z G_v$, $\hat{G}_\text{v} = \zeta_x E_v + \zeta_y F_v + \zeta_z G_v$ – viscous flow vectors; $\hat{H} = 1/j\, H$ – vector of source terms.

In the system of equations (1), the n - component vectors $\hat{Q}, \hat{E}_i, \hat{F}_i, \hat{G}_i, \hat{E}_\text{v}, \hat{F}_\text{v}, \hat{G}_\text{v}$ have the corresponding form depending on the turbulence model.

The vectors $\hat{Q}, \hat{E}, \hat{F}, \hat{G}, E_v, F_v, G_v$ are defined by the following relations

$$\hat{Q} = \frac{1}{J}\begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ E_t \end{bmatrix}, \quad \hat{E} = \frac{1}{J}\begin{bmatrix} \rho U \\ \rho U u + \xi_x p \\ \rho U v + \xi_y p \\ \rho U w + \xi_z p \\ (E_t + p)U - \xi_t p \end{bmatrix}, \quad \hat{F} = \frac{1}{J}\begin{bmatrix} \rho V \\ \rho u V + \eta_x p \\ \rho v V + \eta_y p \\ \rho w V + \eta_z p \\ (E_t + p)V - \eta_t p \end{bmatrix}, \hat{G} = \frac{1}{J}\begin{bmatrix} \rho W \\ \rho u W + \zeta_x p \\ \rho v W + \zeta_y p \\ \rho w W + \zeta_z p \\ (E_t + p)W - \zeta_t p \end{bmatrix}, \tag{2}$$

$$E_v = \frac{1}{J}\begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xz} \\ u\tau_{xx} + v\tau_{xy} + w\tau_{xz} - q_x \end{bmatrix}, \quad F_v = \frac{1}{J}\begin{bmatrix} 0 \\ \tau_{xy} \\ \tau_{yy} \\ \tau_{yz} \\ u\tau_{xy} + v\tau_{yy} + w\tau_{xz} - q_y \end{bmatrix}, \quad G_v = \frac{1}{J}\begin{bmatrix} 0 \\ \tau_{xz} \\ \tau_{yz} \\ \tau_{zz} \\ u\tau_{xz} + v\tau_{yz} + w\tau_{zz} - q_z \end{bmatrix}, \quad (3)$$

where $\xi_x, \xi_y, \xi_z, \eta_x, \eta_y, \eta_z, \zeta_x, \zeta_y, \zeta_z$ - metric coefficients, $J = \partial(\xi,\eta,\zeta)/\partial(x,y,z,)$ - Jacobian transformation of coordinates, $\tau_{xx}, \tau_{yy}, \tau_{zz}, \tau_{xy}, \tau_{xz}, \tau_{yz}$ - components of the stress tensor, q_x, q_y, q_z components of the heat flow vector. $E_t = \rho\left[e + \frac{1}{2}\left(u^2 + v^2 + w^2\right)\right]$.

The pressure value in compressible fluid flows is found from the equation of state

$$p = (\gamma - 1)\left[E_t - \frac{1}{2}\rho\left(u^2 + v^2 + w^2\right)\right].$$

where $\gamma = C_p / C_v$ – is the ratio of specific heat capacities.

Differential turbulence models

Recently, there has been an increasing search for suitable one-parameter differential turbulence models for calculating spatial flows. One of them is the Spalart-Allmaras model [1].

The turbulent effects are described within the framework of the Boussinesq hypothesis of tangential stress representation using a semi-empirical model for turbulent viscosity. Equation (1) is closed by the differential transport equation of the vortex kinematic pseudo-viscosity

$$\frac{\partial(\rho\tilde{\nu})}{\partial t} + \frac{\partial(\rho\tilde{\nu}u_j)}{\partial x_j} = E_t + F_t - G_t + T_t, \quad (4)$$

where $E_t = \frac{1}{\sigma}\left[\frac{\partial}{\partial x_j}\left(\rho(\nu + \tilde{\nu})\frac{\partial\tilde{\nu}}{\partial x_j}\right) + C_{b2}\rho\frac{\partial\tilde{\nu}}{\partial x_j}\frac{\partial\tilde{\nu}}{\partial x_j}\right]_t$ - diffusion term satisfying the boundary conditions on the wall $\tilde{\nu} = 0$; $F_t = C_{b1}(1 - f_{t2})\rho\tilde{S}\tilde{\nu}$ - is an expression that describes the production of turbulence in the region and supports the description of the flow in the laminar sublayer; $G_t = C_{w1}f_w\rho\left(\frac{\tilde{\nu}}{d}\right)^2$ - is an expression describing the turbulence decay in the laminar sublayer; $T_t = f_{t1}\rho\Delta U^2 + f_{t2}\rho\frac{C_{b1}}{\kappa^2}\left(\frac{\tilde{\nu}}{d}\right)^2$ - expression of an approximate description of the transient mode with smoothing functions f_{t1}, f_{t2}, that ensure the transition from laminar to turbulent regime in the wall region.

The vortex viscosity is calculated by the following relationship:

$$\mu_{tur} = \rho \tilde{\nu} f_{v1}, \quad (5)$$

where $f_{v1} = 1 - \chi^3 / \left(\chi^3 - C_{v1}^3\right)$ - damping function for the ratio of kinematic viscosities $\chi = \tilde{\nu} / \nu_{lam}$, that corresponds to the Van Dries damper.

Auxiliary ratios are determined from the expressions

$$\tilde{S} = f_{v3}\omega + \frac{\tilde{\nu}}{(\kappa d)^2} f_{v2},$$

where d - nearest distance to the wall,

$f_{v2} = 1 - \chi / \left(1 + \chi f_{v1}\right)$, $\omega = \left|\nabla \times \tilde{\nu}\right|$ - vortex module,

$$f_{v2} = \left[1 + \frac{\chi}{c_{v2}}\right]^{-3}, \quad f_{v3} = \frac{\left(1 + \chi f_{v1}\right)\left(1 - f_{v2}\right)}{\chi}, \quad f_w = g\left[\left(1 + C_{w3}^6\right) / \left(g^6 + C_{w3}^6\right)\right]^{1/6},$$

$$g = r + C_{w2}\left(r^6 - r\right), \quad r \equiv \tilde{\nu} / \left(\tilde{S}\kappa d^2\right),$$

$$C_{w1} = C_{b1} / \kappa^2 + \left(1 + C_{b2}\right) / \sigma, \qquad C_{w2} = 0{,}3, \; g = r + C_{w2}\left(r^6 - r\right),$$

$$C_{w3} = 2, \; f = g\left(\frac{1 + C_{w3}^6}{g^6 + C_{w3}^6}\right),$$

$$f_{t1} = c_{t1} g_t \exp\left(-c_{t2} \frac{\omega_t^2}{\Delta U^2}\left[d^2 + g_t^2 d_t^2\right]\right), \; g_t = \min\left(0.1, \Delta U / \omega_t \Delta x\right),$$

$$f_{t2} = c_{t3} \exp\left(-c_{t4}\chi^2\right),$$

$$c_{v1} = 7.1, \; c_{v2} = 5.0, \; c_{t1} = 1, \; c_{t2} = 2, \; c_{t3} = 1.1, \; c_{t4} = 2,$$

$$C_{b1} = 0{,}1355, \; C_{b2} = 0{,}622, \; C_{b3} = 2/3.$$

The Detached Eddy Simulation (DES) method is formed by replacing the variable d with $\tilde{d}$, which is determined by the formula [6]

$$\tilde{d} \equiv \min\left(d, C_{DES}\Delta\right), \quad (6)$$

де $\Delta \equiv \max\left(\Delta x, \Delta y, \Delta z\right)$, $C_{DES} = 0{,}65$ - model steel DES.

The paper uses the Spalart-Allmaras turbulence model in the DES implementation.

4 Solving the problem

The advantage of the RANS equations over the original Navier-Stokes equations is that they are formulated directly with respect to the time-averaged flow characteristics that are of primary interest in aerodynamic problems. Thus, it is possible to eliminate the need to calculate local unsteady characteristics of turbulent flows by integrating three-dimensional unsteady Navier-Stokes equations (DNS). In real flow conditions, at the present time of development of computing technology, the calculation of local unsteady characteristics of turbulent flows is considered absolutely impossible.

It should be noted that the RANS equations are open-ended, since they contain an unknown Reynolds stress tensor and a turbulent heat flux vector. Therefore, for their practical use, additional relations are required to relate these quantities to the characteristics of the averaged motion. They can be obtained only with the use of some empirical information. These relations are commonly called turbulence models for Reynolds stresses or second moments.

The Reynolds-averaged Navier-Stokes equations closed by the one-parameter Spalart-Allmaras turbulence model in the realization of separated vortices were used to calculate the flow [6]. Figures 2-3 show the results of numerical calculation of the flow parameters around a circular cylinder. The calculation results are compared with the experimental data of [7] To close the Reynolds-averaged Navier-Stokes equations, the Spalart-Allmaras turbulence model in the DES implementation was used. The initial system of equations was written and solved in a curved three-dimensional coordinate system. The set of programs was written in the FORTRAN-95 programming language.

5 Numerical implementation of the method

The control volume method was used to numerically solve the system of equations (3). The basic principles of the control volume method (CVM) are that we consider the classical equations of balance of a certain quantity Q in a control volume bounded by a surface $S=\sum S_k$ with an external normal Integrating equation (1) over the control volume $\Delta\Omega$, we obtain

$$\iiint\limits_{\Delta V}\left[\frac{\partial\hat{Q}}{\partial t}+\frac{\partial\left(\hat{E}-\hat{E}_v\right)}{\partial\xi}+\frac{\partial\left(\hat{F}-F_v\right)}{\partial\eta}+\frac{\partial\left(\hat{G}-\hat{G}_v\right)}{\partial\zeta}-H\right]d\Omega=0\ . \tag{7}$$

Applying the mean and Ostrogradsky-Gauss theorems to equation (5), we obtain:

$$\frac{\partial\widetilde{Q}}{\partial t}=-\frac{1}{\Delta\Omega}\oint\!\!\oint\limits_{S}\left[\left(\hat{E}-\hat{E}_v\right)n_x+\left(\hat{F}-\hat{F}_v\right)n_y+\left(\hat{G}-\hat{G}_v\right)n_z\right]dS+\widetilde{H}\ , \tag{8}$$

where S – is the surface around the control volume $\Delta\Omega$; $\vec{n}$ - is the vector of the external normal to the surface S .

The upper sign [~]means the average value of the desired function over the volume:

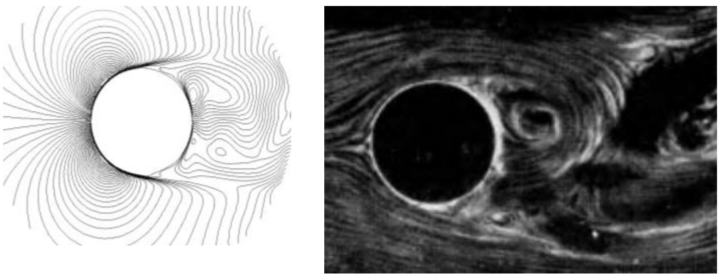
$$\widetilde{f}=\frac{1}{\Delta\Omega}\iiint\limits_{\Delta V}f\,d\Omega$$

By solving the Reynolds Navier-Stokes averaged problems closed by the Spalart-Allmaras turbulence model in the realization of separated vortices (4,5,6), the flow around a circular cylinder is calculated. The calculation results (Fig. 4) are compared with experimental data [7,8].

The paper deals with the calculation of the flow of a flat-bottomed transport vehicle, the hull of which is a semicylinder with a wedge-shaped bow and stern.

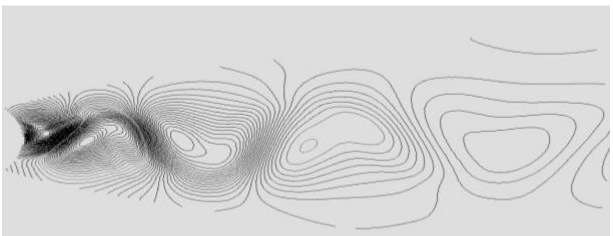
An important component of numerical modeling is the correct construction of a computational mesh around the body under study. We use a zonal approach. The total calculation area is divided into several blocks. A separate mesh is built for each block. Algebraic methods and methods based on solving differential equations are used to build the mesh in separate blocks

The blocks are constructed in such a way that the calculated hydro-gas-dynamic parameters can be correctly transferred from one block to another. For this purpose, an appropriate methodology for setting boundary conditions at the boundaries of individual blocks was developed. The computational domain consists of 6 blocks. Figure 5 shows the structure of the calculation grid blocks. Fig. 6 shows the design grid of block No. 1

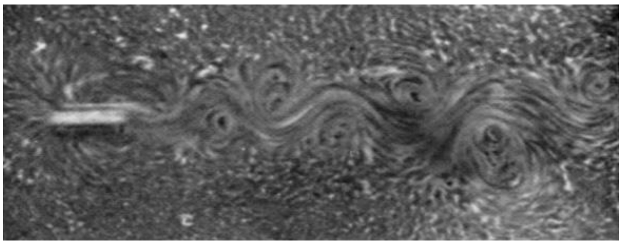


a) *b)*

Fig. 2: Turbulent flow around a cylinder with the descent of a vortex track:
a - calculation (isoline V), Re=10000; *b* - experiment [7], Re=10000.

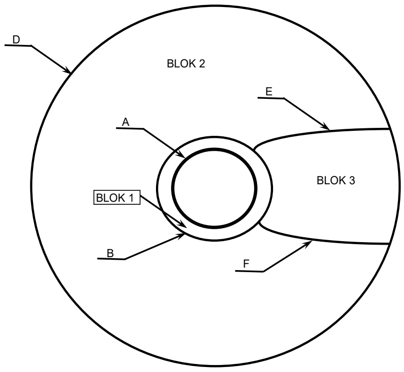


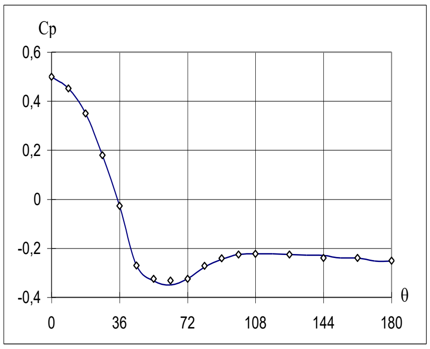
a)



b)

Fig. 3. Vortex track in the wake of a circular cylinder:
a - calculation of Re=19300; *b* - experiment Re=19300 [7].





a) *b)*

Fig. 4. Numerical modeling of flow around a transversely arranged circular cylinder
a – Block approach to building a computational domain around a circular cylinder,
b – Distribution of the pressure coefficient on the cylinder surface for Re=14000:
___- calculation of Re=14000; O - experiment Re=14500 [8].

The total number of nodes is 1816096. The distance to the plane-shaped track structure is h=0.0125 of the maximum transverse size of the vehicle's midsection. Calculations were performed for the Mach number M=0.3. The distribution of isomachies over the upper surface of the vehicle and the pressure distribution around the surface of the vehicle body are shown in Figs. 7 and Figs. 8.

Numerical calculations have confirmed the efficiency of the developed methodology, algorithms, and a set of programs.

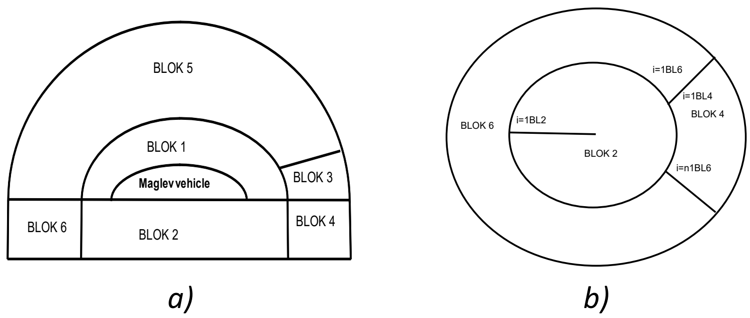


Fig. 5. Multi-block structure of the computational domain around the hull of a high-speed vehicle:

a - vertical section; *b* - horizontal section

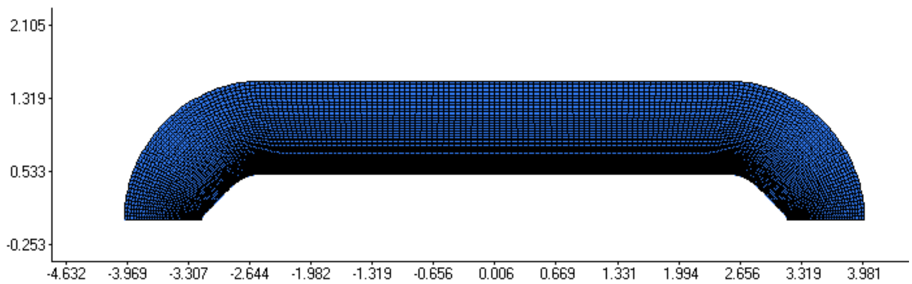


Fig. 6. Design grid of block No. 1

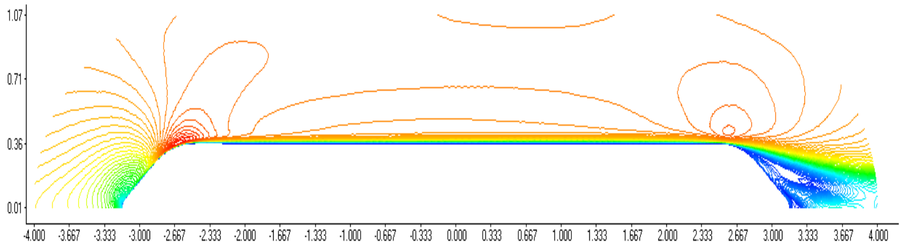


Fig. 7. Distribution of isomachs over the upper surface of the vehicle

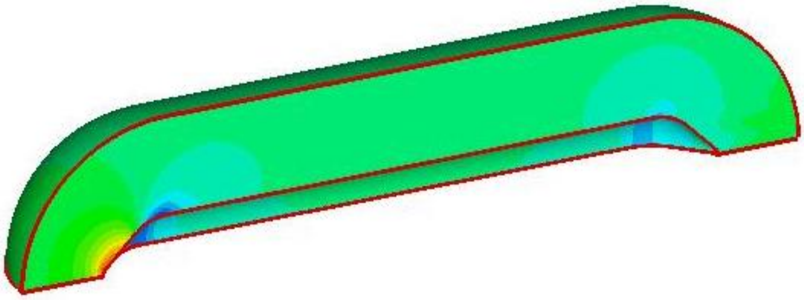


Fig. 8. Pressure distribution around the surface of the vehicle body

6. Acknowledgements

The development of methods, algorithms, software and a set of studies was carried out in accordance with the research topic of the Institute of Transport Systems and Technologies of the National Academy of Sciences of Ukraine "Development of theoretical foundations for the development and research of magnetically levitating transport and innovative energy systems". I would like to express my gratitude to the Director of the Institute of Transport Systems and Technologies of the National Academy of Sciences of Ukraine, Doctor of Technical Sciences, Professor Viktor Dzenerskyi for providing support in the research.

References

1. P. R. Spalart, “Strategies for turbulence modeling and simulations”, Int. J. Heat Fluid Flow, 2000, v. 21, pp. 252–263.
2. J. H. Ferziger, “Recent Advances in Large Eddy ”, Engineering Turbulence Modelling and Experiments 3. W. Rodi and G. Bergeles (Editors), 1996, Elsevier, pp. 163- 176.
3. P. R Spalart. "Philosophies and fallacies in turbulence modeling", Progress in Aerospace Sciences. 2015, 74(1), pp. 1–15.
4. A.V.Sokhatskyi, To the problem of mathematical modeling of turbulent flows around transport vehicles. Systems and technologies. 2021. № 2 (62). C.5-36.
5. F.R. Menter, Two-equation eddy-viscocity turbulence models for engineering applications. AIAA Journal, 1994. v. 32, N 8. P. 1598-1605.
6. P.R., Spalart, S.R. Allmaras, A one-equations turbulence model for aerodynamic flows. AIAA paper, 1992, Vol. 0439, 21p.
7. G. K. Batchelor, An Introduction to Fluid Dynamics. Cambridge University Press, 1970.
8. A. Roshko, On the drag and shedding frequency of two-dimensional bluff bodies. NACA Tech. Note, 1954, №3169. 29p.