UDC 629.78:519.8 DOI: 10.18698/1812-3368-2021-2-37-51

PARAMETRIC CALCULATIONS OF THE AERODYNAMICS OF A DESCENT VEHICLE

V.N. Dubovik[1] vdubovik@laspace.ru
V.T. Zhukov[2] zhukov@kiam.ru
K.V. Manukovskii[3] manu@itep.ru
N.D. Novikova[2] nn@kiam.ru
D.V. Tulin[1] tulin@laspace.ru
O.B. Feodoritova[2] feodor@kiam.ru

[1] Lavochkin Association, Khimki, Moscow Region, Russian Federation
[2] Keldysh Institute of Applied Mathematics, Russian Academy of Sciences, Moscow, Russian Federation
[3] National Research Center Kurchatov Institute — ITEP, Moscow, Russian Federation

Abstract

We present the methodology and results of parametric aerodynamic studies of vehicles descending into the planet's atmosphere. The proposed computational approach might serve as the basis for solving a number of problems such as predicting and optimizing the descent trajectory of the vehicle, the search for a rational aerodynamic layout of the vehicle, i.e., tasks requiring massive parametric calculations. The systematization of such calculations is the first step towards the creation of a specialized database that includes sets of input and output data (flight speed, angles of attack, drag and lift coefficients, aerodynamic pitching moment, etc.) and the corresponding three-dimensional fields of gas-dynamic quantities together with computational meshes of various granularity and parameters of the computational model. Additional information to each element of the database might be a set of variables, parameterizing the geometry of the vehicle, experimental data, etc. The probability of forming the information content of such a database using modern supercomputer systems is shown. The capabilities of the domestic supercomputer aerodynamic code *NOISEtte* are demonstrated in the field of multiparametric three-dimensional calculations of descent vehicles based on the numerical solution of the Navier — Stokes equations on three-dimensional unstructured meshes

Keywords

Descent vehicle, aerodynamic characteristics, unstructured meshes, Navier — Stokes equations

Received 27.05.2020
Accepted 01.07.2020

Introduction. The current state of computational research of aerospace crafts is characterized by the widespread use of mathematical models, which require significant computer resources for numerical implementation. One of the high-cost aerospace problems is the three-dimensional numerical simulation of the aerodynamics of space descent vehicles (DV) based on the Navier — Stokes system of equations to solve important problems, for example, predicting and optimizing the flight path of the vehicle in the atmosphere of the Earth or other planets. The solution to this problem requires massive parametric calculations to create multidimensional tables of aerodynamic characteristics (ADC) depending on flight parameters (speed, angle of attack, etc.) in a wide range of values of the listed quantities. Typical requirements for the volume of a table — dozens of values for each variable parameter — lead to the need to perform thousands of calculations in a complex flow pattern (with the presence of shock-wave and turbulent phenomena [1–3], etc.). The general situation for calculating the ADC for one point of the table is to solve a three-dimensional system of equations for the dynamics of a compressible viscous heat-conducting gas based on unsteady Reynolds-averaged Navier — Stokes equations with a suitable turbulence model [3]. To calculate the trajectory of the DV in the planet's atmosphere, this aerodynamic problem must be solved together with the ballistic problem — a system of ordinary differential equations describing the motion of the center of mass of the DV in the planet's gravitational field. Under the assumption that the DV motion occurs under the action of aerodynamic forces and gravity, the system of ballistic equations includes a subsystem of equations for three variables (range, flight altitude, lateral displacement) [4].

This subsystem determines the movement of the center of mass by its velocity V, the angle θ of inclination of the velocity vector to the local horizon and the directional angle ψ. Another subsystem is written concerning the vector $b=(V,\ \theta, \psi)$ of unknowns in the form $db/dt = F(b,\ C_x,\ C_y, C_z)$, where $C_x,\ C_y, C_z$ are the coefficients of the components of the aerodynamic force along the axes of the trajectory coordinate system [4]. In turn, these coefficients depend on the shape and size of the vehicle, its orientation in space relative to the velocity vector, as well as on the properties and current state of the atmosphere. The chosen mathematical model takes into account viscous friction in the boundary layer, wave resistance, and vortex formation. At hypersonic speeds, viscous friction for a vehicle with good aerodynamics can account for more than half of the total resistance, and upon transition to a turbulent state, viscous friction and heat fluxes on the surface of the vehicle increases many times [5–7].

With the joint integration over time of the ballistic and aerodynamic problems, the values of the aerodynamic coefficients C_x, C_y, C_z, which are necessary to determine the parameters of the ballistic problem are taken to be constant at each time step. The joint ballistic-aerodynamic calculation is laborious, usually, trajectory and aerodynamic studies are carried out separately, see [5–12], where these two calculations are considered using different degrees of approximation.

Another important task is to find a rational aerodynamic configuration of the DV. In a simple formulation, this problem consists of parametrizing the geometry of the spacecraft with a small set of parameters and search by enumerating the best option, for example, from the position of determining a suitable trajectory of atmospheric descent. This means that for each geometry of the vehicle it is required to solve a joint ballistic-aerodynamic problem. In the case of optimization of the trajectory (for example, according to the criterion of the minimum final speed), such a problem must be solved many times. Of course, there are optimization methods based on variational principles, but the use of such methods is limited to simple formulations. Consequently, the computational complexity of the discussed problem will be large.

The solution of two problems can be built based on creating a specialized database, which includes sets of input and output data and corresponding three-dimensional distributions of velocity, density, pressure, together with computational meshes of various granularity. Additional information for each element of the database can be a set of data that parameterize the geometry of the vehicle, the values of ADC obtained from field experiments and other calculations, etc.

The practical result of the work is the formation of the information content of such a database using modern supercomputer capabilities. The problem of creating a supercomputer methodology for parametric aerodynamics studies of the spacecraft is solved, which can be transferred to supercomputers of a new generation as they appear within the paradigm of high-performance computing using parallel programming mechanisms *OpenMP, MPI, CUDA*, etc.

The developed supercomputer methodology is based on the numerical solution of the Reynolds-averaged Navier — Stokes equations (RANS) using a semi-empirical turbulence model [13, 14] on three-dimensional unstructured meshes using the *NOISEtte* code [15]. This code was developed in the sector of computational aeroacoustics of the Keldysh Institute of Applied Mathematics, Russian Academy of Sciences, and includes numerical techniques for low- and high-speed flows in a wide range of Mach numbers. There are several turbulence models in the code, adapted to the tasks of the class of interest to us.

It is assumed that the code will be developed to take into account chemical reactions, that is, where it is essential to take into account the reactions, to reduce the complexity of the calculations, one can use the constructed database, filling it with new data. This corresponds to the proven approach to the calculation of multicomponent mixtures of reacting gases: at high flight speeds, the complete calculation is most effectively implemented in stages — a multicomponent calculation is performed sequentially without taking into account chemical reactions and only then the calculation taking into account chemical reactions and heat transfer in boundary layers.

Problem statement. The proposed methodology was used in practice for calculating the DV with varying their geometric shape. Here we have limited ourselves to considering a vehicle with a fixed geometry, and the variable parameters are the Mach number and the angle of attack. Let us recall the basic definitions, see, for example, [16]. When the DV moves in the air, it is acted upon by the pressure forces and viscous friction, which depend on the speed, air density, body shape, and body position in the flow. Their resulting force is the integral of the pressure forces over the surface of the body and passes through a point called the pressure center. The integral of the moments of pressure forces concerning a point called the center of mass gives an aerodynamic moment, which is equal to the product of the resulting force and its arm relative to the center of mass.

Aerodynamic forces are applied to this center of mass and can be expressed in terms of dimensionless aerodynamic coefficients C_x, C_y, C_z. Usually, computational work on the analysis of ADC is carried out using related coordinate axes and their corresponding coefficients — the coefficients of the longitudinal (C_A) and normal (C_N) forces. In this work, the information content of the ADC database is formed under the assumption that the lateral component of the force is absent, that is, it is possible to limit ourselves to the equations of motion describing a flat trajectory of a point mass (the motion of the center of mass). Then, to determine the descent trajectory, it is enough to use tables with coefficients C_A and C_N.

Mathematical model. Practical calculations of the DV are based on the model of a continuous gas medium described by the system of Navier — Stokes equations taking into account the effects of turbulence. A fairly standard problem statement is considered, supplemented by the implementation of high-precision methods on parallel computing systems of the petaflops class. The system of averaged Navier — Stokes equations (the so-called RANS model) has the form (see, for example, [1–3]):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0;$$
$$\frac{\partial (\rho \vec{u})}{\partial t} + \nabla \cdot (\rho u \otimes u) = -\nabla p + \nabla \cdot (\tau_m + \tau_t) + \rho f;$$
$$\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho u H) = \nabla \cdot [u \cdot (\tau_m + \tau_t) + (q_m + q_t)].$$

Here t is the time; u is the velocity vector of the averaged flow with components (u_1, u_2, u_3); p is the pressure; τ_m, τ_t are molecular and turbulent components of the viscous stress tensor; f is the vector of external forces, for example, gravity; $E = e + |u|^2 / 2$ is the specific total energy of the gas, e is the specific internal energy of gas; $H = E + p / \rho$ is the total enthalpy; q_m, q_t are molecular and turbulent components of the vectors of heat flux density. Turbulence plays an essential role in the pattern of physical phenomena. The work was limited to the use of the Spalart — Allmaras turbulence model [13, 14]. Its choice is due to the simplicity of the model (one equation, labor-saving formulas) and physically reasonable modeling of various flows (boundary layers, jets, separated flows, etc.).

Computational model. The process of mathematical modeling involves several stages: the selection of a computational model, determination of computational domains and subdomains, boundary and initial conditions, as well as carrying out a cycle of methodological and parametric calculations. In modern calculations, it is also necessary to take into account the possibility of good parallelization of the algorithm. The presented calculations were performed using the *NOISEtte* software package [15]. Aerodynamic calculation of a real three-dimensional object can be successful only when using a geometric computational mesh of acceptable quality; the preparation of such a mesh is one of the laborious parts of the computational process. To model and calculate a complex flow, it is required to construct an initial mesh and provide, if necessary, its subsequent modification to obtain a more accurate solution. In the course of a computational experiment, the mesh quality significantly affects the accuracy, convergence, and computation time. In this case, the properties of the computational mesh are determined by the features of the solution to the problem. Let us list the main stages of mesh building. First, a triangulation of the surface bounding the considered three-dimensional body is created. As a rule, the surface is imported from the CAD-package and can be edited if necessary. At the same time, the set of proposed description formats and geometry editing tools is quite wide. Thus, the basis for constructing a computational volumetric mesh outside the body is one or several (in the case of a multi-domain problem) triangulated surfaces. The model of the studied DV is shown in Fig. 1. The object consists of a spherically rounded circular cone with a smooth conjugation and two circular cylinders of different radii located in the mid-section domain of the DV. The streamlined vehicle has several rounded angle points, near which expansion or compression flows can be locally realized. A triangular mesh with the required quality (the degree of smoothness in domains with high curvature, the degree of resolution of thin places, the growth rate of the characteristic size of surface cells when moving away from the areas with high granularity, preservation of topological features, local mesh refinement, etc.) is created on the surface of the DV.

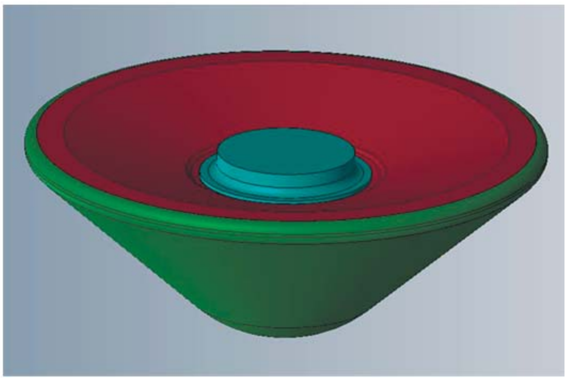


Fig. 1. DV model

The requirements for the absence of self-intersections, closeness, etc. are imposed on surface meshes. In working with real objects, the resulting CAD-surfaces meet these requirements in rare cases. As a rule, additional surface preparation is required to obtain a volumetric mesh of the required quality. For this, several automatic instruments with a wide range of quality control tools are used. First, defects of CAD-geometry (holes, self-intersections, etc.) are corrected, which allows one to get a closed surface with the required granularity. The resulting surface mesh can have a triangulation of low quality (there can be a significant inhomogeneity of the distribution of the vertices, a large anisotropy of triangles, etc.). The use of such surface triangulation greatly complicates the process of creating a three-dimensional computational mesh and can degrade its quality. Therefore, an important stage is the construction of a surface triangulation of the required quality level. Rough surface triangulation acts as an initial approximation. Then, using an iterative method based on the Delaunay criterion, the mesh is locally refined and rebuilt until the specified quality criteria are met. Moreover, in the process of rebuilding the surface mesh, the required granularity (characteristic size of mesh elements) is achieved, either specified by the user or automatically calculated based on the geometric characteristics of the surface (local curvature, the proximity of various model elements, etc.). The resulting surface mesh is used as input for the procedure for creating a volumetric computational mesh. The general view of the surface at half of the DV cut off along the plane of symmetry is shown in Fig. 2. In domains with increased curvature of the surface, the mesh is adaptively refined.

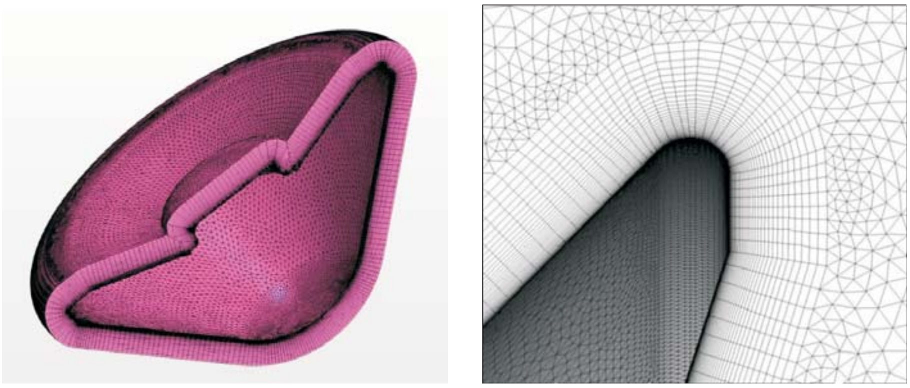
Fig. 2. General view of a triangulated surface

As already noted, the resulting surface mesh is the basis for constructing a volumetric computational mesh. At this stage, it is necessary to determine the size and shape of the computational domain.

At a zero angle of attack, the flow is axisymmetric, but the presence of even a small angle of attack makes the problem three-dimensional. In the case of a change in the angle of attack only in the vertical plane, the problem retains symmetry about the vertical plane, which makes the calculations more economical, since it allows calculations in a half-space. In the presence of horizontal and vertical components of the forward flow velocity, the problem is solved on a full spatial mesh. Peculiarities of the aerodynamics of flows dictate some rules for constructing volumetric meshes. Such a mesh consists of two main parts: 1) prismatic near the streamlined surfaces; 2) arbitrary polyhedral at a sufficient distance from surfaces. Two approaches are used to create a wall prismatic layer with a given number of cells in the direction from the wall, with a given law of cell size growth and the total layer thickness.

In the first approach, a new surface mesh is built, spaced from the original surface by the full thickness of the wall layer. The resulting wall volume is cut into a given number of prismatic layers. This approach usually does not guarantee the creation of a prismatic layer with the given parameters and the required total thickness in domains with complex geometry. In such cases, it is better to use the method of sequential creation of prismatic layers, which allows modification of the surface mesh of the current layer (collapse of edges, cutting of cells, optimization of the position of vertices, etc.), which allows creating a layer of prismatic cells that penetrates much further from the walls into the computational domain.

It is the second approach that is used in the process of creating volumetric meshes. The general view of the computational domain with a wall layer consisting of 40 layers of prismatic cells is shown in Fig. 3 *a*, the smoothness of the mesh in the rounding domain is shown in Fig. 3 *b*.



a *b*

Fig. 3. General view of the computational domain with a wall layer (*a*) and its fragment (*b*)

The structure of the volumetric computational mesh is shown in Fig. 4. To fill the internal volume outside the wall layers, various designs are used. The most acceptable meshes are hybrid meshes, which belong to the class of conformal polyhedral meshes since the simplest and most efficient discretizations are designed specifically for such meshes. The *NOISEtte* code allows the use of meshes consisting of triangular prism cells in the wall layers and tetrahedrons outside such layers.

Computational studies. The following is a description of the calculations of a uniform forward flow around a typical DV with specified aerodynamic characteristics. The type and diagram of the DV are shown in Fig. 1 and 5. The initial part of the vehicle is a frontal spherically blunt circular cone. The rear cowling of the vehicle introduces certain difficulties in constructing a mesh adapted to this geometry and also complicates the calculations. The investigated range of speeds is very wide and includes sections of hyper- and supersonic flights,

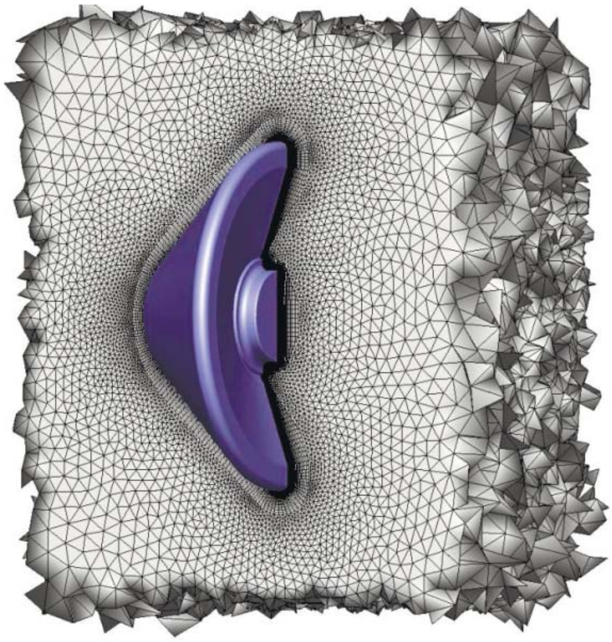


Fig. 4. Structure of the volumetric calculation mesh

as well as sections of sonic and subsonic flights up to the section of controlled deceleration in the immediate vicinity of the planetary surface. Among the ADCs, the aerodynamic drag coefficients C_x, of the lift force C_y (longitudinal and normal components of the force in a coupled system) and the aerodynamic pitching moment m_z.

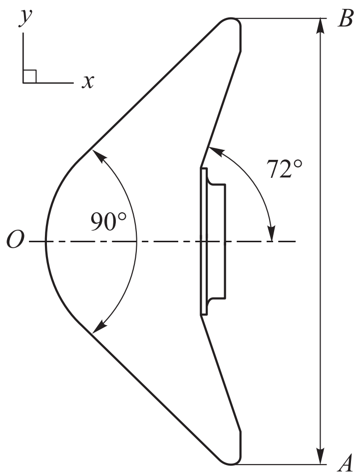


Fig. 5. DV diagram

The pitching moment is calculated relative to the *Oz* axis of the right *Oxyz* coordinate system with the origin at point *O* (see Fig. 5). In this case, the calculated flight modes of the DV lie in a wide range of Mach numbers (0.2, 0.4, 0.6, 0.8, 1, 1.2, 1.6, 1.9, 2.3, 3, 5, 7, 10, 15, 22, 28), angles of attack of the incident flow α (0, 1, 2, 5, 10, 15, 20, 25, 30, 35, 40, 45°), i.e., the calculation was made 16 × 12 = 192 options.

When determining the dimensionless aerodynamic coefficients, the diameter *AB* of the mid-section of the DV is taken as the characteristic size (see Fig. 5), and its area is taken as the characteristic area. All calculations are carried out under the assumption of a perfect gas model with constant values of the adiabatic index $\gamma = 1.4$ and Prandtl number $\mathrm{Pr} = 0.71$. The descent vehicle is placed in an undisturbed oncoming flow, the basic characteristics of which (density, pressure, and Reynolds number) differ for different flight modes and are given by a sufficiently detailed table as functions of the Mach number in the range M = 0.2–28. The DV understudy has an axisymmetric shape, which makes it possible to use a hemisphere as the outer boundary of the computational domain. The diameter of this hemisphere is equal to 20 diameters of the mid-section of the vehicle located inside the hemisphere. The topology, shape, and size of the computational domain correspond to the goals of the computer simulation and problem statement.

Three boundary surfaces are selected for calculations. On the surface of the vehicle, the adhesion condition is specified, for pressure and temperature — the equality of derivatives along the normal to the surface to zero. At each point of the 'input' boundary, the direction of the gas-dynamic flow is determined, and either the specified values of the forward flow are set, or the values of the quantities are transferred from the computational domain according to the characteristics. On the plane of symmetry, the zero normal component of the velocity and the conditions for the equality of the normal derivatives of pressure and temperature to zero are specified. The characteristic scales of the problem are the values of the variables in the undisturbed flow ρ_∞, u_∞ and the size L, equal to the mid-section diameter. The initial data is a homogeneous flow with parameters $\rho = 1, p = (1/\gamma)\mathrm{M}^2$ and a velocity vector which is collinear with the forward flow velocity vector. Using the constructed geometric model we generate a three-dimensional hybrid tetrahedral mesh with prismatic layers in the boundary layer domains (near solid surfaces) with a total volume of approximately 1,3 million computational nodes. Additionally, for verification, a mesh with a doubled number (~ 2,8 million) of nodes was constructed to check the practical convergence of the solution. The number of cells in the prismatic layer is about 30 % of their total number. The thickness of the prismatic layer is approximately $6.5 \cdot 10^{-2}$; the size of the boundary cell $h \approx 10^{-4}$ (relative units). An essential element of a reliable computational experiment is the construction of a histogram of the mesh quality. When forming this histogram, the basic mesh indicators are calculated and compared with some accepted consensual values. For example, these include the minimum internal angle of the element (values should be within 18–160°), the aspect ratio, which shows how the control volume is stretched (< 10 000), etc. To control the quality of the constructed meshes, we used as the own tools of the *NOISEtte* software complex, and the *checkMesh* utility belonging to the open package *OpenFoam*. Quality requirements include preservation of the sign of the volume of cells (no overlap), non-admission of almost degenerate cells, absence of large values of dihedral angles, etc. Attention is paid to such characteristics of the mesh as non-orthogonality, skewness, and aspect ratio. Since aerodynamic coefficients are of interest in a wide range of Mach numbers, including sub-, trans-, supersonic, and hypersonic modes, the simulation strategy changes when moving from one mode to another. For subsonic flight modes ($M < 0.8$) Roe's scheme was used together with linear-one-dimensional reconstructions of the fifth order, for transonic modes — *HLLE* scheme based on the fifth-order *WENO*-type reconstructions, for $M \geq 3$ modes — the *HLLE* scheme with linear reconstructions [15]. The indicated switches must be included in the database content as control parameters. For time integration, an implicit first-order method based on linearization by Newton's method was used. Systems of linear equations are solved by the method of stabilized bi-conjugate gradients, and the symmetric Gauss — Seidel method is used as a preconditioner with a block size equal to the number of unknown functions, but without taking into account the equations describing turbulence. The relative accuracy of solving linear systems was set equal to 0.01.

To automate the calculations, a script has been written, it allows group calculations to be performed without user intervention. Computational clusters of the Keldysh Institute of Applied Mathematics, Russian Academy of Sciences were used for the calculations. The computation time for one variant varied from 2 to 12 hours using 64–256 cores.

Let us illustrate the calculation results with graphs. Recall that we are considering flow in the absence of physical and chemical processes. The dependence f the aerodynamic coefficients on the angle of attack for different flight modes (pre-, trans-, supersonic, and hypersonic) is shown in Fig. 6. In the entire range of angles of attack and Mach numbers, the coefficient C_x smoothly decreases from the values $\approx 0.8–1.7$ to the minimum values $\approx 0.3–1.0$, and the coefficient C_y smoothly increases from 0 to the maximum values $\approx 0.3–0.4$. The dependence of the pitching moment on the angle of attack is shown in Fig. 7: at a zero angle of attack, there is a stable equilibrium (the moment is equal to zero, and its derivative with respect to the angle is negative) for all values of the Mach numbers. To check the accuracy of the obtained numerical results, comparative studies of several DV flow modes on various meshes were carried out. Usually, a twofold increase in the number of nodes of the computational mesh, that built based on the above-described qualified technology, clarifies the gas-dynamic pattern of the flow, but the difference in the integral coefficients, which are of main interest in the work, is several percent.

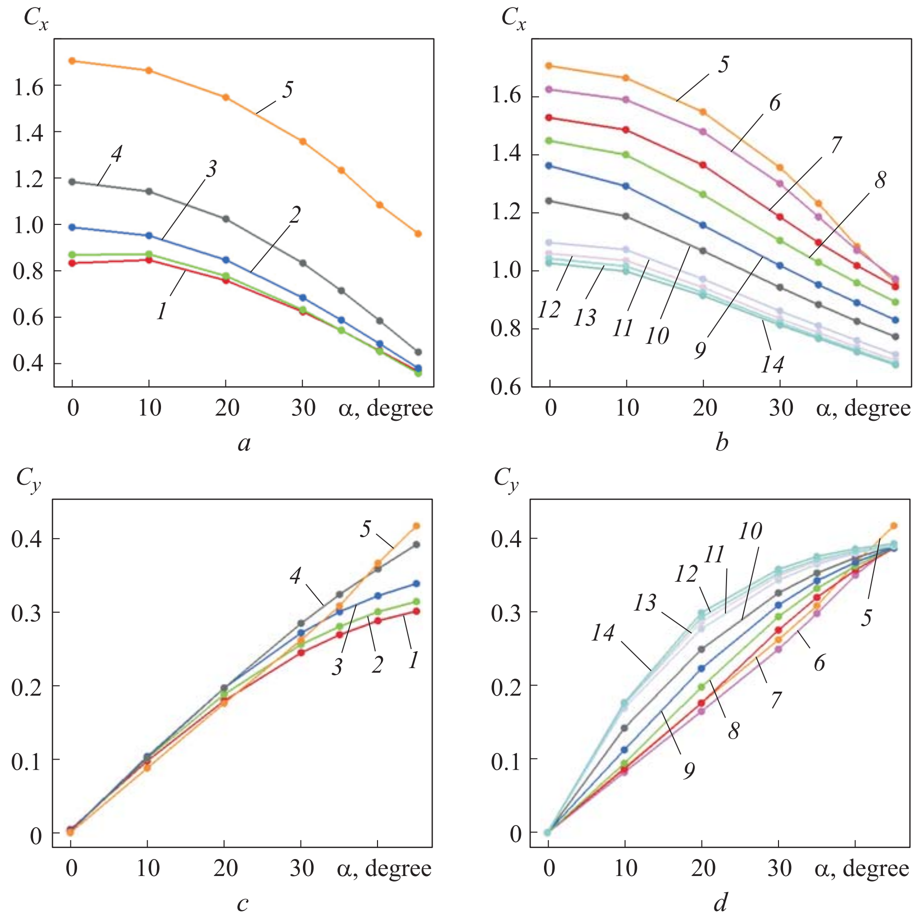


Fig. 6. Dependencies of aerodynamic coefficients C_x (*a*, *b*) and C_y (*c*, *d*) on the angle of attack α at Mach numbers M = 0.2 (*1*), 0.4 (*2*), 0.6 (*3*), 0.8 (*4*), 1.0 (*5*), 1.2 (*6*), 1.6 (*7*), 1.9 (*8*), 2.3 (*9*), 3.0 (*10*), 5.0 (*11*), 7.0 (*12*), 10 (*13*), 28 (*14*)

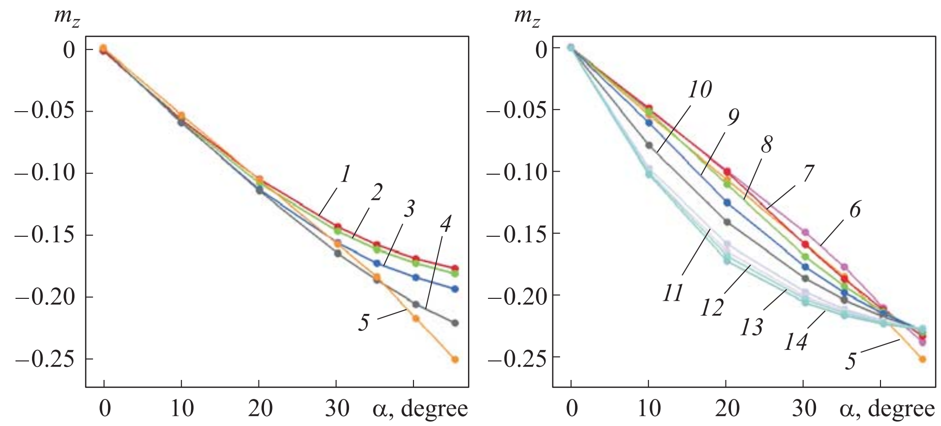


Fig. 7. Dependence of the pitching moment coefficient on the angle of attack at different Mach numbers M = 0.2 (*1*), 0.4 (*2*), 0.6 (*3*), 0.8 (*4*), 1.0 (*5*), 1.2 (*6*), 1.6 (*7*), 1.9 (*8*), 2.3 (*9*), 3.0 (*10*), 5.0 (*11*), 7.0 (*12*), 10 (*13*), 28 (*14*)

Conclusion. The presented results show that the use of supercomputer technologies allows solving the problems of determining the ADC in a large range of changes in flight and geometric parameters at a fundamentally new level. The supercomputer technology of aerodynamic calculations based on the domestic code [15] makes it possible to obtain the necessary set of aerodynamic force coefficients and the moments of the DV and thereby close the system of equations of the vehicle motion without using empirical relations. The given results illustrate the good accuracy of the calculated values of the aerodynamic coefficients of the DV. Mass parametric calculations using parallel supercomputer technology are promising as a tool for building a database that includes sets of input and output data and corresponding three-dimensional fields of gas-dynamic quantities together with computational meshes and parameters of the computational model. Such a database can be used to refine the trajectory calculations of the DV, as well as in the design of new structures.

Translated by K. Zykova

REFERENCES

[1] Loytsyanskiy L.G. Mekhanika zhidkosti i gaza [Fluid mechanics]. Moscow, Drofa Publ., 2003.

[2] Landau L.D., Lifshits E.M. Teoreticheskaya fizika. T. 6. Gidrodinamika [Theoretical physics. Vol. 6. Thermodynamics]. Moscow, FIZMATLIT Publ., 2001.

[3] Tirskiy G.A., ed. Giperzvukovaya aerodinamika i teplomassoobmen spuskaemykh kosmicheskikh apparatov i planetnykh zondov [Hypersonic aerodynamics and heat exchange of descent spacecraft and planetary probes]. Moscow, FIZMATLIT Publ., 2011.

[4] Dmitrievskiy A.A., Lysenko L.N. Vneshnyaya ballistika [External ballistics]. Moscow, Mashinostroenie Publ., 2005.

[5] Golomazov M.M., Ivankov A.A. Examination of the aerodynamics of a prospective descent module descending in the atmosphere of Mars. *Vestnik NPO im. S.A. Lavochkina*, 2018, no. 2, pp. 56–61 (in Russ.).

[6] Golomazov M.M., Finchenko V.S., Shmatov S.I. Teoretiko-eksperimental'nye metody resheniya zadach aerodinamiki kosmicheskikh apparatov, spuskaemykh v atmosferakh planet. V kn.: Proektirovanie avtomaticheskikh spuskaemykh apparatov dlya fundamental'nykh nauchnykh issledovaniy [Theoretical experimental methods for solving aerodynamic problems of descent spacecraft in planetary atmospheres. In: Design of Automated Descent Spacecraft for Fundamental Scientific Research]. Moscow, MAI-Print Publ., 2014, pp. 962–1034 (in Russ.).

[7] Zheleznyakova A.L., Kuzenov V.V., Petrusev A.S., et al. The modelling of hydrodynamics for two different models of space vehicle. *Fiziko-khimicheskaya kinetika*

v gazovoy dinamike [Physical-Chemical Kinetics in Gas Dynamics], 2010, vol. 9, no. 1 (in Russ.). Available at: http://chemphys.edu.ru/issues/2010-9/articles/144

[8] Andreev V.N., Borovkov A.I., Voynov I.B., et al. Aerogasdynamics behavior of the escape system separable nose assembly with operating propulsion system. *Kosmicheskaya tekhnika i tekhnologii* [Space Technique and Technologies], 2014, no. 4, pp. 10–20 (in Russ.).

[9] Matyushin M.M., Sokolov N.L., Ovechko V.M. Optimal control of a spacecraft during descent in Mars atmosphere. *Vestnik NPO im. S.A. Lavochkina*, 2016, no. 2, pp. 54–60 (in Russ.).

[10] Alabova N.P., Bryukhanov N.A., Dyad'kin A.A., et al. Role of computer simulation and physical experiment in investigations of space rocket system aerogasdynamics throughout the designing. *Kosmicheskaya tekhnika i tekhnologii* [Space Technique and Technologies], 2014, no. 3, pp. 14–21 (in Russ.).

[11] Aksenov A.A., Dyad'kin A.A., Moskalev I.V., et al. Computer simulation of the flow and the relative motion of the reentry vehicle and the parachute compartment hatch cover in the course of their separation during descent. *Kosmicheskaya tekhnika i tekhnologii* [Space Technique and Technologies], 2015, no. 2, pp. 39–50 (in Russ.).

[12] Lutsenko A.Yu., Nazarova D.K., Slobodyanyuk D.M. Computation of aerodynamic characteristics and parameters of flow around the launch vehicle nose fairing half in the ANSYS CFX package. *Inzhenernyy zhurnal: nauka i innovatsii* [Engineering Journal: Science and Innovation], 2018, no. 5 (in Russ.).
DOI: http://dx.doi.org/10.18698/2308-6033-2018-5-1766

[13] Spalart P.R., Allmaras S.R. A one-equation turbulence model for aerodynamic flows. *30th Aerospace Sciences Meeting and Exhibit*, 1992, no. 1992-0439.
DOI: https://doi.org/10.2514/6.1992-439

[14] Garbaruk A.V. Sovremennye podkhody k modelirovaniyu turbulentnosti [Modern approaches to the modeling of turbulence]. St. Petersburg, St. Petersburg Univ. Publ., 2016.

[15] Abalakin I.V., Bakhvalov P.A., Gorobets A.V., et al. Parallel research code NOISEtte for large-scale CFD and CAA simulations. *Vychislitel'nye metody i programmirovanie* [Numerical Methods and Programming], 2012, vol. 13, no. 2, pp. 110–125 (in Russ.).

[16] Feoktistov K.P., ed. Kosmicheskie apparaty [Aircraft]. Moscow, Voenizdat Publ., 1983.

Dubovik V.N. — Deputy Head of Department, Lavochkin Association (Leningradskaya ul. 24, Khimki, Moscow Region, 141402 Russian Federation).

Zhukov V.T. — Dr. Sc. (Eng.), Chief Researcher, Keldysh Institute of Applied Mathematics, Russian Academy of Sciences (Miusskaya ploshchad 4, Moscow, 125047 Russian Federation).

Manukovskii K.V. — Cand. Sc. (Phys.-Math.), Senior Researcher, National Research Center Kurchatov Institute — ITEP (Bolshaya Cheremushkinskaya ul. 25, Moscow, 117218 Russian Federation).

Novikova N.D. — Researcher, Keldysh Institute of Applied Mathematics, Russian Academy of Sciences (Miusskaya ploshchad 4, Moscow, 125047 Russian Federation).

Tulin D.V. — Deputy Head of Complex, Lavochkin Association (Leningradskaya ul. 24, Khimki, Moscow Region, 141402 Russian Federation).

Feodoritova O.B. — Senior Researcher, Keldysh Institute of Applied Mathematics, Russian Academy of Sciences (Miusskaya ploshchad 4, Moscow, 125047 Russian Federation).

Please cite this article as:

Dubovik V.N., Zhukov V.T., Manukovskii K.V., et al. Parametric calculations of the aerodynamics of a descent vehicle. *Herald of the Bauman Moscow State Technical University, Series Natural Sciences*, 2021, no. 2 (95), pp. 37–51.
DOI: https://doi.org/10.18698/1812-3368-2021-2-37-51