Numerical simulation of aerodynamic properties of separable elements of space-rocket systems in the LOGOS software package

S V Aksenov, D A Korchazhkin, A Yu Puzan and O A Puzan

Federal State Unitary Enterprise “Russian Federal Nuclear Center – All-Russia Research Institute of Experimental Physics” (FSUE «RFNC-VNIIEF»), Mira ave., 37, Sarov, 607188, Russia

E-mail: pppan@yandex.ru

Abstract. The work describes simulation results of the aerodynamic properties of split elements of the space-rocket systems in the LOGOS software package: the review of the advanced methods for aerodynamic properties has been carried out, the advantages of the computer simulation when solving such problems were demonstrated, verification and validation computations of the simulation problems for the split elements were performed, the produced results were compared with the reference data from available publications. The analysis of the results confirmed the applicability of the LOGOS software package for computing aerodynamic properties of the split elements.

1. Introduction

The lift-off of modern launch vehicles (LVs), including space-mission vehicles, is followed by the separation of their passive elements of structure which either exhaust their margin of energy (boost stages and propellant tanks, as well as their fragments, which are detached upon exhaustion of a propellant), or fulfill their work within the rocket-and-space system (fairings and plates protecting against a high-speed air flow and lowering the aerodynamic resistance). The large-size separable parts (SPs) weighting several hundred kilograms which did not burn up in the atmosphere have a high kinetic energy at the time of impacting the Earth’s surface and are a serious hazard to human lives and engineering structures in the impact area. Majority of SPs, which are thin-walled shells constituted by combinations of spherical, cylindrical, and conical surfaces move at random in their flight and, thereby, increase a potential impact area. Note also, that such SPs have a high lift-to-drag ratio and, as a result, the separated parts, may be drifted far away by an air flow.

To find the impact area location and size for SPs, it is necessary to be able to simulate their flightpaths and estimate the factors affecting the spread of impact points and this requires the knowledge of their aerodynamic properties within a wide range of angles of attack and skew, both at supersonic and subsonic speeds.

The experimental investigation of the aerodynamic characteristics of thin-walled elements is a complex process task in view of the specific features of mounting them in a wind tunnel. In additional, one should not rule out the influence of the test-bed parts on the results of experiments. Supporting
devices used to fasten models and their fairings significantly change the model shape and cause a significant effect on the aerodynamic characteristics of the subjects of interest. With an increasing angle of attack, the error induced by fastening parts of the test-bed also increases.

The specific feature of calculating aerodynamic characteristics using engineering analysis codes is a simplification of a streamlined model to represent its shape as a set of primitive objects (cones, cylinders, etc.), however, such a simplification does not always ensure the required accuracy of calculations. In the design practice, modern software packages for the 3D numerical simulation based on the Navier-Stokes equations [1] are used with an ever-increasing frequency to minimize errors in calculating aerodynamic characteristics using engineering analysis codes. The LOGOS software package [2] is one of them, it allows developing various structural designs of LVs [3] in the designing the stage to find an optimum structure and optimum shapes of structural elements in the course of simulating many physical processes. The LOGOS software package validation for the benefit of space and aviation industries is described in papers [4,5]. However, the numerical simulation of a flow around the thin-walled SPs of the LV structure has some typical features [6,7], including exclusive quality standards for a grid model and a description of narrow edges, the level of complexity of aerodynamic moments due to the unstable behavior of flows near the concave surfaces at small angles of an attack, and a significant effect of turbulence on the lift-to-drag ratio of a thin-walled element. In view of the foresaid, it is required to examine the applicability of the LOGOS software package to solve such classes of problems. The paper presents some calculated aerodynamic characteristics of the thin-walled SPs obtained by solving a number of typical problems.

2. The simulation of the interaction of supersonic jets with a barrier
In the CFD simulation of the stage separation process it is important to adequately describe the interaction of jets of rocket engine combustion products with the separation unit deflector in calculating the distribution of pressures over it. The LOGOS software package was validated on the problem of simulating the interaction of single-block and three-block supersonic jets with a barrier. The pressure distribution curves over a barrier from paper [8] are taken for the reference data.

The simulation was performed using block-structured grid models generated by the LOGOS preprocessor. To study the effect of typical grid model sizes on simulation results, grids varied in the number and size of cells. 1 illustrates the longitudinal cross-sections of the grid models used for the single-block and three-block CAD models. A grid of 12.5 mln cells (figure 1 (d)) for the three-block CAD model is automatically constructed using the grid reconstruction module according to the solution specifics based on the given problem solution using a grid of 3.2 mln cells. In this case, the LOGOS preprocessor automatically refines the grid in domains with typical sudden changes of the Mach number and pressure to specify the fields of distribution of their values in a given computational domain.

The steady-state problem was solved by the coupled solver of the LOGOS software package with the implicit second-order scheme using the Spalart-Allmaras turbulence model, the Low wall function $y^+$, the Courant number $C=5$, and the AUSMPW method of calculating convective flows. The following parameters were set as the boundary condition in the critical nozzle cross-section: static pressure $P_{\text{stat}} = 5 673 977$ Pa, $P_{\text{tot}} = 10 740 450$ Pa, the gas temperature $T_{\text{tot}} = 300$ K.
Figure 1. Grid models in a longitudinal cross-section of the computational domain: a) 1.4 mln cells for the single-block CAD model, b) 3.2 mln cells for the three-block CAD model, c) 5 mln cells for the single-block CAD model, d) 12.5 mln cells for the three-block CAD model.

The simulation gives us the distribution fields of the main physical quantities in the computational domain and on the barrier. Figure 2 shows the pressure profiles for the barrier calculated using the LOGOS software package in comparison with the experimental data from [8].

Figure 2. The pressure profiles for the barrier: a) the single-block CAD model, b) the three-block CAD model.

The analysis of results demonstrates that by decreasing the cell size (generation of a grid with a smaller basic size of cells, or the use of the grid reconstruction module with regard to the solution specifics) it is possible to obtain the values of pressure of on the barrier, which are closer to the experimental ones, with the time of each simulation case increasing in direct proportion.
3. The calculation of gas dynamic characteristics in the SMV stage separation process

The numerical simulation of the gas dynamics during the separation of stages of a space-vehicle (SMV) in its nominal flight was performed for a fixed position of the first and second stages of a vehicle. For the verification, the values of pressure over the deflector surface given in paper [9] were used.

The simulation was performed using one block-structured grid of 8.9 mln cells shown in figure 3. The steady-state problem with an explicit scheme using the coupled solver of the LOGOS software package was solved. The SST model of turbulence and molecular viscosity was used, the heat conductivity factor was calculated using Satherland’s formula, maximum Courant number was 0.1.

The simulations were concurrently performed on 32 processors, the problem runtime was 12 hours.

Figure 3. The grid model on the SMV surface.

Figures 4 and 5 show the velocity amplitude distribution field in a longitudinal plane of the computational domain and the distribution of pressures over the deflector surface, respectively, which were obtained using LOGOS software package. Figure 6 shows the distribution of pressures over the deflector surface obtained with the LOGOS software package in comparison with the reference data from [9].

Figure 4. The field of the velocity amplitude distribution in the longitudinal plane.
Figure 5. The pressure distribution field.

Figure 6. The distribution of pressures over the deflector surface in the plane XY.

The comparative analysis of the calculated results and reference data from [9] demonstrates that the position of the region with a relative pressure maximum on the bottom deflector in the LOGOS simulation is almost coincident with the reference one, a relative error is not higher than 5% for a given point and within the confidential interval for this class of problems.
4. The simulation of a flow around a nose fairing half

The thin-walled nose fairing halves are the most complex separable parts for which the simulation of an aerodynamic characteristic is a difficult task. The LOGOS software package was used to simulate a supersonic flow with the Mach number $M=1.2$ around the nose fairing half model at an angle of attack of $30^\circ$. The simulation results were compared with the experimental data from [9].

To solve this problem, a 3D model grid with 3.1 mln cells, predominantly hexahedral, and ten prismatic layers were generated. This model grid covering the surface of a nose fairing half is illustrated by figure 7.

The steady-state problem was solved using the Spalart-Allmaras turbulence model, the Courant number $C=10$, and the AUSMPW method of calculating convective flows. The simulations were performed in parallel on 48 processors, the problem runtime was less than 1 hour.

Figure 8 shows the fields of the pressure distribution over the upstream and downstream surfaces of the model.

![Figure 7. The model grid on the nose fairing half surface.](image)

![Figure 8. The pressure distribution fields over the model surface: a) upstream surface, b) downstream surface.](image)

Table 1 gives the values (calculated using the data obtained with the LOGOS software package) of coefficients of the aerodynamic longitudinal force $C_x$, the normal force $C_y$, and the aerodynamic pitching moment coefficient, $m_z$, in comparison with the experimental data from [9].

|        | LOGOS | Experimental data [9] | Relative error (%) |
|--------|-------|-----------------------|--------------------|
| $C_x$  | 1.24  | 1.3                   | 4.61               |
| $C_y$  | 4.28  | 4.1                   | 4.39               |
| $m_z$  | 0.52  | 0.47                  | 10.63              |
The analysis of the LOGOS simulation results for the aerodynamic flow around a model fairing half demonstrates that a relative error in the calculated aerodynamic characteristics is below 11% and admissible for a given class of problems.

5. The simulation of a supersonic flow around an LV model spent stage

Free flights of the spent first stages of launch vehicles in the Earth’s atmosphere lead to the conditions, when an object moves at a large angle of attack, with nozzles of power units being an oriented upstream. The LOGOS software package was used to simulate a supersonic flow around a spent LV stage with the Mach number $M=2$ and the angle of attack of the $174^\circ$. Pressures inside nozzles taken from paper [10] were used as reference data for the validation. The simulation was performed using in total a tetrahedral computational grid of $1\,954\,675$ cells, including $15$ near-wall prismatic layers. The model grid in figure 9 was generated by the automatic grid generator of the LOGOS software package.

The steady-state problem was solved by the LOGOS coupled solver using Spalart-Allmaras turbulence model and the equation of state of an ideal gas for the substance (air).

The simulations were performed in parallel on $32$ processors, the problem runtime was $1$ hour.

Figure 9. A grid model: a) the surface grid, b) the volume grid near nozzles.

Figure 10 shows the pressure distribution fields in the vicinity of nozzles, figure 11 shows the pressure profiles in the LV bottom part obtained using the LOGOS software package in comparison with the data from paper [10].

Figure 10. The pressure distribution fields in the vicinity of nozzles: a) in the bottom part of the model, b) in the longitudinal cross-section of the computational domain.
Figure 11. The pressure profiles in the bottom part of LV.

In the current simulation, the mean value of pressure at the nozzle bottom centers is \(1.8 \times 10^5\) Pa, the corresponding experimental value is \(1.8 \times 10^5\) Pa [10]. So, a relative error in the LOGOS mean value of pressure is 2.7%.

6. The simulation of a relative motion of a reentry vehicle and a parachute container hatch cover in the course of separation

The capability of simulating the motion of solids using grids with the “Chimera” type overlaps [11] has been implemented in the LOGOS software package. This functional capability was tested for validity on the problem of separating a parachute container hatch cover (PCHC) from a landing module. The problem is formulated, as follows: a vehicle moves in the atmosphere with a velocity of 0.6 Mach, the environment temperature and pressure are 252 K and 50507 Pa, respectively, the space angle of attack is 18°. The PCHC is separated under the action of the cover pusher whose forces are applied to the mass center of the PCHC and are normal to the body.

To simulate the separation of the PCHC from the reentry vehicle and the moving hatch dynamics, the two grid models were generated: one grid defines the whole volume, has 406 339 hexahedral cells in total, including 5 prismatic layers (figure 12 (a)); the second grid defines the volume in the vicinity of the PCHC, has 648 732 hexahedral cells, 2 near-wall prismatic layers (figure 12 (b)).

Figure 12. Grid models: a) for a landing module, b) for a parachute container hatch cover.

The two-stage simulation was performed: at the first stage, the steady-state flow around the vehicle with a motionless PCHC was simulated, at the second stage the unsteady-state problem was solved with regard to both the aerodynamic flow around the vehicle and the separation process together with
the PCHC dynamics with respect to space. The separation of the PCHC was initiated by gaining momentum, the translational movement in the atmosphere was under the action of forces, the rotational movement was under the action of momenta.

The simulations were performed in parallel on 64 processors, the problem runtime was 16 hours.

Figure 13 illustrates the Mach number distribution at the time of the PCHC separation from the landing module, figure 14 shows the coordinate X of the PCHC mass center as a function of time according to the results obtained using the LOGOS software package and the data from paper [12].

![Figure 13](image13.png)

**Figure 13.** The Mach number distribution at the time of the PCHC separation from a landing module.

![Figure 14](image14.png)

**Figure 14.** The coordinate X of the PCHC mass center as a function of time.

The comparative analysis of the results obtained, demonstrates that in the initial stage of separating the PCHC from the landing module, the mass center positions in at both curves are the same, while in the free flight of the hatch cover, they become different. This effect may be attributed to the specific features of simulations, differences in grid models and turbulence models that affect the moving body dynamics in the airflow.
7. Conclusion
This paper presents the simulation results for the aerodynamic characteristics of separable parts of space-rocket systems obtained using the LOGOS software package. Modern methods of determining the aerodynamic characteristics were analyzed, verification and validation tests on the simulation of the behavior of separable parts were performed, and the results obtained were compared with the reference data from open sources. The examination of simulation results allows making conclusions concerning the application of the LOGOS software package to calculate the aerodynamic characteristics of separable parts of space-mission vehicles and the stage separation gas-dynamics in the design activities of enterprises of the rocket and space industry.

References
[1] Landau L D and Lifshits V M 1988 Hydrodynamics (Moscow: Nauka)
[2] Safronov A V, Deryugin Yu N, Zhuchkov R N, Zelenskiy D K, Sarazov A V, Kozelkov A S, Kudimov N F, Lipnitskiy Yu M and Panasenko A V 2014 Results of the LOGOS multilibrary software package validation in solving problems of aero- and gasdynamics for the lift-off and flight of launch vehicles Mathematical Modeling 26(9) pp 83-95 (in Russian)
[3] Deryugin Yu N, Zhuchkov R N, Zelenskiy D K, Kozelkov A S, Sarazov A V, Kudimov N F, Lipnitskiy Yu M, Panasenko A V and Safronov A V 2015 Validation Result for the LOGOS Multifunction Software Package in Solving Problems of Aerodynamics and Gas Dynamics for the Lift-Off and Injection of Launch Vehicles Mathematical Models and Computer Simulations 7(2) pp 144-153
[4] Pogosyan M A, Saveljevskikh E P, Shagaliyev R M, Kozelkov A S, Strelets D Yu, Ryabov A A, Kornev A V, Deryugin D Yu, Spiridonov V F and Tsiberev K V 2013 Application of Russian supercomputer technologies to develop advanced models of aviation technology VANT, Ser. Matematicheskoie Modelirovanie Fizicheskikh Protsessov (Mathematical Simulation of Physical Processes) (2) pp 3-17 (in Russian)
[5] Betelin V B, Shagaliyev R M, Aksenov S V, Belyakov I M, Deryugin Yu N, Kozelkov A S, Korchazhkin D A, Nikitin V F, Sarazov A V and Zelenskiy D K 2014 Mathematical simulation of hydrogen–oxygen combustion in rocket engines using LOGOS code Acta Astronautica 96 pp 53–64
[6] Lutsenko A Yu, Nazarova D K and Fomin M A 2017 Aerohydrodynamic characteristics of thin-walled conical shells at supersonic velocities of mainstream Engineering journal: science and innovations 4(64) (in Russian)
[7] Dyadkin A A, Krylov A N, Lutsenko A Yu, Mikhailov M K and Nazarova D K 2016 Specific features of the aerodynamics of thin-walled structures Space engineering and technologies 3(14) pp 15–25 (in Russian)
[8] Kudimov N F, Safronov A V and Tretyakova O N 2014 Applied problems of gas dynamics and heat exchange in power units of rocket engineering (Moscow: MAI Press)
[9] Dyadkin A A, Lutsenko A Yu and Nazarova D K 2016 Numerical simulation of subsonic and transonic flow around thin shells Civil Aviation High Technologies (Nauchny Vestnik MGTU GA) (233) pp 45–50 (in Russian)
[10] Kartseva E Yu, Kolyada E O, Stroilov A V and Shmanenkov V N 2015 A study of flow pattern of first-stages of launch vehicles during the ballistic descent phase at high angles of attack Astronautics and rocket building (Kosmonavtika i raketostroenie) 4(83) pp 11–15 (in Russian)
[11] Deryugin Yu N, Sarazov A V and Zhuchkov R N 2017 Features of overset meshes methodology on unstructured grids Mathematical Modeling (Matematicheskoie modelirovanie) 29(2) pp 106-118 (in Russian)
[12] Aksyenov A A, Dyadkin A A, Moskaliev I V, Petrov N K and Simakova T V 2015 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 Space Engineering and Technologies 2(9), pp. 39–50