Numerical simulation of gas flow in nozzle channels with a central body

Yu V Kaun, N A Brykov and M V Chernyshov

Baltic State Technical University “VOENMEH”, Saint Petersburg, Russia

E-mail: y.kaun13@gmail.com

Abstract. The development of a reusable launch system is one of the most promising directions in the development of cosmonautics. World rocket companies are striving to create a space vehicle with improved tactical and technical characteristics. The main task is to ensure the most efficient flight mode of the reusable spacecraft from the point of view of the full load of the injection system. Since the main costs are spent on transporting the fuel of the rocket and its own design, the developers strive to fully load such a system not only when sending into orbit, but also when returning back. The implementation of the idea of a reusable spacecraft entails large energy losses, which suggests the idea of creating a single-stage reusable launch system.

1. Introduction

An important criterion for evaluating the efficiency of a power plant is the property of autoregulation, the ability to maintain thrust efficiency over the entire range of flight altitudes, from the Earth's surface to outer space.

Wedge air nozzles (annular nozzles with a central body) are the closest to a perfectly adjustable nozzle. Such nozzles significantly benefit from an increase in altitude characteristics: thrust and specific impulse. This is especially noticeable at low flight altitudes, where the property of autoregulation (establishment of a flow close to the design regime in a supersonic flow with a free boundary) is so important. Wedge-air nozzles of jet engines can become a competitive alternative for reusable launch vehicles [1-2].

The main advantages of nozzles with a central body in front of the classic rocket engine nozzle are:

1) flow stabilization in the required direction;
2) maximizing thrust by transforming the movement of the jet behind the engine in the desired direction.

2. Mathematical model of the solver

When setting up a numerical experiment for the flow of a compressible gas in a nozzle channel with a central body, at the initial stage, a high Reynolds model of turbulence of the type $k - \varepsilon$. Turbulence model with two differential equations, because models with one equation for the kinetic energy of turbulence are low in quality due to additional non-universal algebraic relations for dissipation. The choice of the second variable greatly affects the properties of the model, since it should not change much, since otherwise, these equations are difficult to solve numerically, as well as boundary conditions for the second variable.
In the case of a compressible gas, averaging occurs according to Favre, in which the density $\rho$ and pressure are averaged according to Reynolds, and for other variables, weighted average values are introduced:

$$ f = \frac{\rho f}{\rho} $$

(1)

The averaged Navier-Stokes equations for a compressible perfect gas (the averaging signs are omitted) have the following form (for an incompressible one, the energy equation is excluded, $\rho = \text{const}$):

$$ \begin{align*}
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) &= 0 \\
\frac{\partial (\rho \vec{u})}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) &= -\nabla p + \nabla \cdot (\tau_m + \tau_t) \\
\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho \vec{u} H) &= \nabla \cdot [\vec{u} \cdot (\tau_m + \tau_t) + (\vec{q}_m + \vec{q}_t)] \\
\rho &= \frac{p m}{R T}
\end{align*} $$

(2)

Here is the velocity vector of the averaged flow with components - the molecular and turbulent components of the viscous stress tensor, is the total gas energy, is its total enthalpy, are the molecular and turbulent components of the heat flux density vector, is the temperature, is the specific heat capacity of the gas at constant volume, is the specific heat capacity of gas at constant pressure, $m$ - molar mass of gas, $J / (\text{mol} \cdot \text{K})$ - universal gas constant.

$$ \begin{align*}
\rho &= \frac{C_p T}{C_p - R/m} C_p m R = 8,31434 \\
\frac{D k}{D t} &= \nabla \cdot \left( \left( \frac{\nu + \nu_t}{\sigma_k} \right) \nabla k \right) + P_k - \varepsilon; \\
\frac{D \varepsilon}{D t} &= \nabla \cdot \left( \left( \frac{\nu + \nu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right) + C_1 \frac{\varepsilon}{k} P_k - C_2 \frac{\varepsilon^2}{k}; \\
P_k &= -\tau_{ij} \frac{\partial U_i}{\partial x_j} = \nu_t S^2; \\
\nu_t &= C_\mu \frac{k^2}{\varepsilon}.
\end{align*} $$

(3-6)

The constants are found on the basis of dependence and jet flows: $C_2 = C_1 - \frac{k^2}{\sigma_\varepsilon \sqrt{\nu_t}} \sigma_k = 1.0, \sigma_\varepsilon = 1.3, C_1 = 1.44, C_2 = 1.92, C_\mu = 0.09.$

- $\varepsilon$ - specific dissipation;
- $k$ - kinetic energy of turbulence;
- $P_k$ is the generation term in the equation for kinetic energy;
- $\nu_t$ - Kolmogorov’s formula;

At the stage of the two-dimensional formulation of the numerical experiment, the ideal gas model was used.
3. Construction of the central body in a two-dimensional setting.

The model of the central body, providing the maximum thrust characteristic, is sensitive to any deviation of the geometry of the central body from the ideal shape (at which the work of the nozzle is close to that of the design mode).

The ideal contour or shape of the outer generatrix of the aerodynamic nozzle is determined using the adiabatic theory of supersonic flows. Then, depending on whether the flow is unexpanded or overexpanded, either Prandtl-Meier expansion flow analysis or shock system analysis is performed to determine the angle of intersection of the flow boundary with the edge of the primary nozzle. According to the results of the analysis of oblique shock waves, the angle of inclination of the shock waves for the overexpanded flow is determined and displayed as a contour of the outer boundary forming the primary nozzle.

This technique analyzes rarefaction waves from the throat of a rocket engine nozzle, where the Mach number is one, to the exit section as a series of simple expansion waves. On the outer generatrix of the wedge, a Prandtl-Meier rarefaction flow is observed from the edge of the primary nozzle to the edge of the outer generatrix of the wedge, which acts as the central body of the nozzle (figure 1).

To construct the contour of the central body OC, we define on it the position of an arbitrary point B. The intermediate rarefaction wave AB falls into point B. The flow velocity w (or M) along the rarefaction wave (characteristics) AB is constant. The flow section F, in which the velocity w, is defined as the projection of the lateral surface of the truncated cone, formed by the rotation of the segment AB around the nozzle axis, onto the plane normal to the velocity [5-6].

![Figure 1](image_url)

**Figure 1.** To the calculation of the contour of the nozzle with the central body:
1 – axis of the nozzle; 2 – free surface.

Flow area F and angle \( \omega \) between the flow direction and the tangent line AE are determined by the relations

\[
\frac{F}{F_{sp}} = \frac{1}{M} k_{cm} \left( 1 + \frac{k_{cm} - 1}{2} M^2 \right)^{\frac{k_{cm} + 1}{2(k_{cm} - 1)}}
\]

\[
\omega = \sqrt{\frac{k_{cm} + 1}{k_{cm} - 1}} \arctan \left( \frac{k_{cm} - 1}{k_{cm} + 1} \left( M^2 - 1 \right) \right) - \arctan \sqrt{M^2 - 1}
\]

The angle \( \mu \) between the directions of the characteristics AB and the velocity w (Mach angle) is determined by the ratios:

\[
\sin \mu = \frac{1}{M}
\]

If R is the distance from point B to the nozzle axis, and the length AB is equal to L, then
\[ F = \pi L (R + R_a) \cos(90^\circ - \mu) = \pi L (R + R_a) \sin \mu \]  

(9)

Denoting by \( \psi \) the angle between the directions of the tangent AE and the normal to the axis of the nozzle, from the geometric relations one can obtain

\[ \omega_a = 90^\circ - \psi \]  

(10)

\[ \angle BAC' = (\mu - \omega) + \omega_a = \mu - (\omega + \psi) + 90^\circ \]

(11)

Denoting the angle \( BAC' = \chi \), we get:

\[ \chi = 90^\circ + \mu - (\omega + \psi) \]

\[ \angle DBA = 90^\circ - \chi = -\mu + (\omega + \psi) \]

Since \( BD = Ra - R \), then

\[ L = \frac{R_a - R}{M \cos(\mu - (\omega + \psi))} \]

(12)

\[ F = \frac{\pi (R_a^2 - R^2)}{M \cos((\mu - (\omega + \psi))} \]

Taking into account the relation \( F_a = \pi R^2 a \), we have

\[ R = R_a = \sqrt{1 - \frac{F}{F_a} M \cos(\mu - (\omega + \psi))} \]

(13)

As a result, we get:

\[ L = \frac{R_a}{\cos(\mu - (\omega + \psi))} \left[ 1 - \sqrt{1 - \frac{F}{F_a} M \cos(\mu - (\omega + \psi))} \right] \]

(14)

Relations (12) and (14) are calculated for constructing the contour of the central body. A nozzle path with a central body was modeled in a two-dimensional setting. The outflow of the compressible gas occurred from two variants of high-altitude nozzles with a central body, shown in figure 2 at different flight altitudes.
The primary nozzle includes an annular supply channel with a combustion chamber and a supersonic part with a minimum cross-section, which may be absent. The secondary nozzle consists of a central body, tapering to the axis of symmetry. The nozzle with the central body is conventionally divided into two parts: the primary nozzle and the secondary nozzle [7-8]. The central body nozzle can be divided into two parts: a primary nozzle and a secondary nozzle. The primary nozzle includes an annular supply channel with a combustion chamber and a supersonic part with a minimum cross-section, which may be missing. The secondary nozzle consists of a central body that tapers towards the axis of symmetry.

4. Results of a numerical experiment.

As a result of numerical simulation, it was possible to obtain patterns of the velocity distribution at low flight altitudes (figure 3). Initially, a model of a high-altitude nozzle with a truncated cone-shaped central body was calculated (figure 4).

Figure 2. Geometric model of an “Aerospike” nozzle with a truncated (a) and solid (b) central body.

Figure 3. Distribution of Mach numbers at flight altitude: a) 0 km; b) 10 km; c) 20 km; d) 30 km

Figure 4. Distribution of Mach numbers at flight altitude: a) 0 km; b) 10 km; c) 20 km; d) 30 km

The configuration with a wedge-shaped central body noticeably brings the nozzle operation closer to the design mode at low flight altitudes (Fig. 3 a-c). Starting from an altitude of 30 km (Fig. 3 - d), the
flow turns in the direction of the outer walls of the nozzle. This effect is also traced during the outflow of a supersonic jet into vacuum.

On the surface of the wedge, the maximum Mach number is about 3, which is less than the theoretical value of 3.41. A decrease in the Mach number indicates the presence of a shock wave at the exit from the generatrix of the primary nozzle. The shock wave on the surface of the wedge is a deviation from the ideal flow and causes excessive heating of the surface. The structure of the jet-separated flow at the exit from the nozzle has been studied. At large degrees of off-design, it contains three rarefaction waves: at the outer edge of the nozzle cut, at the point of intersection of the hanging shock generated by the compression wave from the profile of the central body, with the boundary of the jet, and at the angular edge of the bottom cut of the central body. Behind the bottom cut of the central body, a large separation region of an irregular conical shape with subsonic return flows is formed, behind which a tail shock occurs.

The described design of the aerodynamic wedge is superior to that of conventional conical nozzles. However, a significant disadvantage of the truncated cone is that at high altitudes behind the base, a turbulent wake is formed, leading to high resistance and a decrease in efficiency. The problem can be solved by injecting a secondary flow into the area behind the base.

The circulation of this secondary stream and its interaction with the engine exhaust gases create an "aerodynamic wedge". In this case, the nature of the flow remains similar to the case of using a full-fledged wedge. In addition, the secondary flow re-circulates upward, creating additional thrust.

Simulation data indicates areas that are overstressed during normal operation, which can lead to engine and launch vehicle failure. Subsequent modeling will determine the effectiveness of any proposed design modifications.

5. Conclusion
A promising scheme for organizing a jet engine is considered to be a wedge-air jet engine capable of providing a design mode for the outflow of combustion products in a wide range of altitudes. The performed calculations of the gas flow through a nozzle with a central body were carried out with the aim of organizing a central body, which makes it possible to bring the operation of the nozzle closer to the automatic control mode at different flight altitudes.

Acknowledgments
The study was supported by a grant from the Russian Science Foundation No. 21-79-00100, https://rscf.ru/en/project/21-79-00100/.

References
[1] Lukashevich V P, Afanasyev I B. 2009 Space wings (M: LenTa Wanderings) p 498
[2] Dorofeev A 2010 Foundations of the theory of thermal rocket engines. Theory, calculation and design: a textbook for aviation and rocket-building specialties of universities vol 1 (MSTU im. N. E. Bauman) p 463
[3] Garbaruk A V, Strelets M Kh, Shur M L 2012 Modeling of turbulence in the calculations of complex flows (Polytechnic University, St. Petersburg) p 456
[4] Silnikov M V, Chernyshov M V 2018 Shock Waves. Vol. 28. No. 4 pp. 765-784
[5] Klyushnikov V Yu 2019 Ultra-light launch vehicles: a niche in the launch services market and promising projects (Aerospace sphere No. 4) Part 2 pp 64-75
[6] Yaskin A V 2013 The theory of the device of rocket engines (Biysk) p 262.
[7] Kaun Yu V, Teterina I V, Chernyshov M V 2020 Gas injection into the supercritical region of the nozzle as a flight control method for ultra-light launch vehicles Actual problems of protection and security: Proceedings of the XXIII All-Russian scientific-practical conference RARAN vol 1 (Armament and military equipment. - St. Petersburg) pp 161-166.
[8] Gordeyev S and Jumper E 2010 Progress in Aerospace Sciences pp 388-400