Interaction of a system of supersonic jets from a body with an incoming flow

Shevchenko A V¹, Yuriev A S¹, Poniaev S A¹,², Pirogov S Yu¹, Zhitnikov T A¹, and Rotermel A R¹

¹Mozhaysky Military Space Academy
²Ioffe Institute

e-mail: artnetu@yandex.ru

Abstract. There is a need to improve the methods of control of supersonic aircraft in order to increase the efficiency and reliability of their control elements. In this work, theoretical calculations of the flow around a body with a system of supersonic jets are carried out. Using numerical methods, the possibility of control of the flow around a body using a system of jets has been investigated. The numerical study was based on a model of an inviscid perfect gas described by the Navier-Stokes equations.

1. Introduction
A promising direction for improving the aerodynamic characteristics of various bodies in a wide range of speeds, especially with significant restrictions on the shape of structural elements, is to control the flow and its movement — a targeted effect on the air flow in accordance with the required change in the aerodynamic characteristics and other parameters of the flow [1]. According to the nature of the impact on the flow, one of the most effective methods is the gas-dynamic method, implemented in jet control elements (a gas jet is blown from the body into a supersonic flow).

The interaction of the jets that blows from the body surface with the incoming flow leads to the formation of a shock wave in front of them, which does not lead to an increase in drag, allow us to create control moments during movement, and also changes the static stability of the body [2].

2. Results of numerical simulation
This article presents some results of a study of the interaction of supersonic jets with an incident non-uniform supersonic flow (\(M_s, T_s, p_s\)). The calculations were performed using the ANSYS Mechanical 15.0 software package (license of the Mozhaysky Military Space Academy No. 1020993). The studies used a body in the form of a plate with holes of diameter (\(d\)) for jets (\(M_j, T_j, p_j\)) installed at a certain distance (\(l\)) from each other (Figure 1). The outflow from the hole occurs in a mode of under-expansion.
The numerical study was based on the model of a viscous perfect gas \((k = 1.4)\), described by the Navier-Stokes equations (1-4) [3,5]:

\[
\begin{align*}
\frac{\partial \rho}{\partial t} + \nabla \cdot \mathbf{\rho} \mathbf{V} &= 0, \\
\frac{\partial \mathbf{V}}{\partial t} + \nabla \cdot (\mathbf{V} \otimes \mathbf{V} + p \mathbf{E} - T_V) &= 0, \\
\frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot ((\mathbf{E} + p) \mathbf{V} - T_V \cdot \mathbf{V} + \mathbf{q}) &= 0,
\end{align*}
\]

\[
\begin{cases}
p = R T, \\
e = C_T T, \\
E = e + |\mathbf{V}|^2 / 2,
\end{cases}
\]

\[
T_V = \mu_t (\nabla \cdot \mathbf{V}) \mathbf{E} + \mu_t \left( \nabla \otimes \mathbf{V} + \nabla \otimes \mathbf{V}^T \right),
\]

\[
q = -\lambda \nabla T,
\]

where \(\rho\) is the density, \(p\) is the pressure, \(E\) is the total energy of the gas, \(C_T\) is the heat capacity at a constant volume, \(T\) is the gas temperature, \(R\) is the universal gas constant, \(E\) is the metric tensor, \(T_V\) is the viscous stress tensor in the gas, \(\mathbf{q}\) is the heat flux vector, \(\mu\) is the coefficient of viscosity of the gas.

On a solid surface of a streamlined body, the boundary condition (5) is applied:

\[
\mathbf{V} = \mathbf{0}.
\]

At the boundary of the flow inlet, the following boundary conditions (6) are set:

\[
\begin{align*}
\rho &= \rho_b, & \mathbf{V} &= \mathbf{V}_b, & T &= T_b,
\end{align*}
\]

where \(\rho_b, \mathbf{V}_b, T_b\) are the parameters of the undisturbed flow.

At the gas jet inlet boundary, the following boundary conditions (7) are specified:

\[
\begin{align*}
\rho &= \rho_j, & \mathbf{V} &= \mathbf{V}_j, & T &= T_j,
\end{align*}
\]

where \(\rho_b, \mathbf{V}_b, T_b\) are the parameters of the gas jet.

At the boundary of the flow exit from the computational domain, one condition (8) is specified:

\[
\mathbf{V} = \mathbf{V}_\infty.
\]

The Menter shear stress transport model was used as a model of turbulence that closes the Navier-Stokes equations averaged by Reynolds. To solve the system of equations, an implicit scheme was used on a triangular unstructured grid having a concentration in areas with large gradients of gas-dynamic parameters (Figure 2). The computational grid had a dimension of \(8.6 \times 10^4\) elements. The distance from the first calculated point to the wall provided a range of normalized distance \(y+\) from 15 to 30 that
is suitable for the model used and was small enough to determine the local gas-dynamic characteristics at the surface of the body.

Due to the significant non-uniformity of the mesh cell sizes the calculations were carried out using an individual time step for each cell. Variable parameters were \( l \) and \( p_j \). The supply of gas jets into the oncoming flow was carried out in cells adjacent to the place of blowing.

The calculations of the flow around the body were performed for cases of different intensities of the jets and for different "spacing" of nozzles.

The calculation of the flow around the body for different intensities of the blown jets was performed with the following values of the varied parameters: the number of blown jets \( N = 1–3 \), the jet diameter \( d = 2 \cdot 10^{-3} \) m, \( d/l = 0.33 \), \( p_j = 25–200 \) kPa, \( T_j = 293 \) K and \( M_j = 1 \).

The flow with various nozzle spacings was studied with the following change in parameters: \( N = 2–3 \), \( d/l = 0.05–0.5 \), \( p_j = 100 \) kPa, \( T_j = 293 \) K, and \( M_j = 1 \).

The parameters of incoming flow was: \( M_\infty = 10 \), \( p_\infty = 0.287 \) kPa, and \( T_\infty = 250 \) K.

![Figure 2. Mesh in computational domain](image)

![Figure 3. Pressure distribution over the body surface at different N and p_j for jets. (а) N=1, p_j=25–200 kPa, (б) N=2, p_j=25–200 kPa, (в) N=3, p_j=25–200 kPa](image)
Investigation of a system of jets of various intensities

Figure 3 shows graphs of the pressure distribution along the axis of symmetry of the body surface for various numbers of nozzles (N = 1–3) and jet intensities (p_j = 25–200 kPa, M_j = 1, and T_j = 293 K). The static pressure measured on the lateral surface of the body was attributed to the pressure of the unperturbed flow p_\infty. A sharp increase in pressure is explained by shock waves in front of the jet obstacle [4].

Figure 4 shows the pressure distributions in the vicinity of the body (N = 1-3 and p_j = 100 kPa).

![Figure 4](image-url)  
**Figure 4.** Pressure distribution near the body for different number N and pressure p_j for jets.  
a) N = 1, (b) N = 2, (c) N = 3.

In Figure 4, it can be seen that the appearance of shock waves localized in front of the jets leads to the formation of an increased pressure zone extending to the peripheral part of the body. A small circulation zone is formed in front of the jet downstream, limited by the boundary layer detached from the body. A region of low pressure is formed behind the jet, which contributes to the formation of a stable circulation zone.

Investigation of the jet system with various nozzle spacing

The results of studying the flow around a body with a system of supersonic jets at different nozzle spacing (l/d = 0.05–0.5) with jet intensity (p_j = 100 kPa, M_j = 1, T_j = 293 K) are presented in Figures 5 and 6. The maximum increase pressure on the body surface (p/p_\infty) occurs at N = 2, p_j = 200 kPa and in this case p higher than p_\infty at 9 times (Figure 5).
Figure 6 shows the pressure distribution for the different numbers N of jets.

From Figure 6 it is seen that the shock waves localized in front of the jets lead to an increase in pressure on the surface of the body. Small circulation zones are formed in front of the jets downstream. Behind the jets, regions of reduced pressure are formed, contributing to the formation of stable circulation zones. The flow pattern is similar to that which occurs when using a system of jets of various intensities (Figure 4).

3. Conclusion

The considered gas-dynamic method of changes of the flow is promising way for solving the problem of controlling the motion and flow around bodies with good aerodynamic shapes. With increasing $p_j$, the $p/p_\infty$ ratio on the surface of the body increases, and therefore, from the point of view of increasing the efficiency of gas-dynamic control elements, it is most advantageous to use a system of jets with $p_j \geq 100$ kPa.

References

[1] A S Yuriev et al 2018 J. Phys.: Conf. Ser. 1135 012102.
[2] A Yuriev, S Pirogov, N Savischenko, E Ryzhov 2001 4-th Weakly Ionised Gases Workshop (Anaheim, CA).
[3] Surzhikov S T, 2018 Computer aerophysics of re-entry space vehicles. 3D models, (Moscow: Fizmatlit) p 760
[4] 2015 Wind Tunnels: Models, Aerodynamics and Applications ed M Russel (Clanrye International) p 228
[5] M S Liou, C J Steffen, 1993 Journal of Computational Physics 107 pp 23-39