Numerical simulation of a flow over blunted cylinder-flare configuration

A S Kryuchkova$^{1,2}$

$^1$Ishlinsky Institute for Problems in Mechanics RAS, Vernadsky prospekt 101(1), Moscow, 119526, Russia
$^2$Moscow Institute of Thermal Technology, Berezovaya avenue 10(14), Moscow, 127273, Russia

E-mail: kryuchkova.arina.s@gmail.com

Abstract. Numerical modeling of a flow over blunted cylinder-flare configuration using UST3D code was performed. Flow variables fields and aerodynamic drag coefficient distribution over a wide range of Mach numbers were obtained. Good agreement of the numerical results with experimental data was observed. The numerical simulation can serve as a supplementary validation of the UST3D computer code which proved to be a reliable instrument for aerodynamic calculations of widespread configurations.

1. Introduction

The paper presents results of numerical modeling of a flow over blunted cylinder-flare configuration. Thanks to the availability of an extensive experimental foundation of aerodynamics for simple bodies it has become possible to conduct corresponding calculations and hence to validate numerical approaches by using well-known cases. The experiment considered in this paper was carried out in the Small Ballistic Tunnel (SBT) of the N E Zhukovsky Central Aerohydrodynamic Institute (TsAGI) [1]. Model aerodynamic drag coefficients as well as flow structure (figure 1) were investigated.

Figure 1. Shadowgraph of a typical flow structure experimentally obtained in SBT.

Models of blunted cylinder-flare shape were shot to the test section at speeds from 500 up to 2000 m/s. The Mach numbers of the test section had the values of 2.5, 3.0, and 3.5. In this way the resulting
Mach numbers of the incoming flow made up values from 6 to 14. The method has allowed investigation of flow characteristics in a wide range of Mach numbers. For experimental modeling of total Mach numbers from 1.5 to 6 the models were launched to a pressure chamber with undisturbed air. Reynolds numbers in the experiments were changing with the flow rate from $10^6$ up to $20 \cdot 10^6$. The measurements were taken for zero angle of attack. The model geometry is shown in figure 2. Dimensions were determined with aid of the following relationships: $d/2R = 1.0$ (semi-spherical leading part); aspect ratio $L/d = 1.5$; cone half-angle $\theta = 15^\circ$; the model fullness was characterized by the ratio of cylinder to middle diameters $d/D = 0.333$.

![Figure 2. Model geometry.](image)

### 2. Numerical method

UST3D (Un-Structured Tetrahedral 3-Dimentional) code was used for computational modeling of the problem [2,3]. The software solves spatial Navier-Stokes equations using modified method of physical processes splitting [4,5,6]. In Cartesian coordinate system the equations have the form of:

$$
\frac{\partial U}{\partial t} + \frac{\partial E_c}{\partial x} + \frac{\partial F_c}{\partial y} + \frac{\partial G_c}{\partial z} = \frac{\partial E_v}{\partial x} + \frac{\partial F_v}{\partial y} + \frac{\partial G_v}{\partial z},
$$

where $U = (\rho, \rho u, \rho v, \rho w, \rho E)^T$ is conservative variables vector; $E_c = (\rho u, \rho u^2 + p, \rho uv, \rho vw, \rho wE + pu)^T$, $F_c = (\rho v, \rho uv, \rho v^2 + p, \rho vw, \rho vE + pv)^T$, $G_c = (\rho w, \rho uw, \rho vw, \rho w^2 + p, \rho wE + pw)^T$ are convective terms projections; $E_v = (0, \tau_{xx}, \tau_{xy}, \tau_{xz}, \tau_{yx}, \tau_{yy}, \tau_{yz}, \tau_{zx}, \tau_{zy}, \tau_{zz})^T$, $F_v = (0, \tau_{xy}, \tau_{yx}, \tau_{yy}, \tau_{yz}, \tau_{zy}, \tau_{zz})^T$, $G_v = (0, \tau_{xz}, \tau_{zx}, \tau_{yz}, \tau_{zy}, \tau_{zz})^T$ are viscous terms projections; $p$ is density; $u, v, w$ – velocity projections; $E$ – specific energy, $\tau_{ij}$ – viscous tensor components; $q_i$ – heat flux projections.

The flow is supposed to be laminar. For the equations closure the ideal gas law is used:

$$
p = (\gamma - 1) \rho [E - \frac{1}{2}(u^2 + v^2 + w^2)],
$$

where $\gamma = 1.4 –$ heat capacity ratio of air.

The UST3D solver is adapted for unstructured tetrahedral grids. The computational code consists of a preprocessor, a solver and a postprocessor. The preprocessor stocks all necessary data concerning computational cells topology. The solver actually realizes numerical computations and the postprocessor records the results output in a convenient format for graphical visualization. Time step is chosen automatically according to condition on final discrepancy limitation.

The calculi were performed on two tetrahedral unstructured meshes counting 417 384 and 668 675 cells. The mesh density was higher round the model walls, so that the layer between the body and the bow shock wave numbered at least 6 cells in the direction normal to the boundary. Computational domain is patterned in figure 3. Boundary conditions are also indicated in the figure, where: 1 – inflow boundary condition; 2 – flow at infinity; 3 – supersonic exit; 4 – viscous wall. Symmetry boundary condition which coincides with the pattern plane was used to reduce computational costs.
3. Results and discussion

In the numerical modeling the middle diameter was assumed to be 200 cm (approaching to real flight vehicle dimensions). Nominal flow conditions were set to: \( p_\infty = 40000 \text{erg/cm}^3 \), \( T_\infty = 219 \text{K} \), \( T_{\text{surf}} = 300 \text{K} \).

The resulting flow structure includes typical elements such as bow shock, oblique shock near the generating line break, and transient flow in the bottom area. It was found that for this configuration there is no separation zone in the whole range of Mach numbers, which agrees with experimental observations [1].

Calculated drag coefficient comparison with experimental data from [1] is shown in figure 4. Drag coefficient is related to the product of incident flow dynamic pressure and body midsection area (note that it coincides with axial force coefficient at zero angle of attack). According to the study drag coefficient decreases with Mach number growth and stabilizes when Mach number mounts to 7. Good agreement with the experimental curve is observed. The computational accuracy for both meshes is approximately the same and does not exceed the experimental data spread. Flow structure and its transient behavior are illustrated in figure 5.

![Computational mesh and boundary conditions](image)

**Figure 3.** Computational mesh and boundary conditions (zooming is shown on the right).

![Drag coefficient comparison](image)

**Figure 4.** Drag coefficient comparison.
One more factor testifying the accuracy of the numerical modeling of a flow over a body of revolution is bow shock deviation. The shock shape and its distance from the body are mostly determined by the shape of the body leading part. Since the leading part of the considered configuration is semi-spherical, the shock deviation distance on the axis of symmetry $\varepsilon$ can be assessed by shock deviation tables and curves obtained for a sphere. There is a set of widely known papers dealing with sphere bow shock deviation [7, 8].

Comparison of the computed deviation distance on the axis of symmetry with experimental and numerical results given in [7] is shown in figure 6. The numerical simulation is noted to give a satisfactory estimation of bow shock deviation.

After validating numerical method with help of the tests mentioned above a set of computations of a flow under various angles of attack ($5^\circ$, $10^\circ$, $15^\circ$, and $20^\circ$) for $M_\infty = 3$ was performed. The purpose of the computations was to demonstrate angle of attack effect on drag force as well as to verify UST3D stability in more complex problems. Computed drag coefficient dependence on the angle of attack is presented in figure 7. The drag coefficient is shown to increase while angle of attack rises and it can be seen that there is still no stalling in the considered angle of attack range. Flow structure for $M_\infty = 3$ and angle of attack $\alpha = 20^\circ$ is shown in figure 8.
Figure 6. Normalized bow shock deviation distance $\varepsilon_0 = \varepsilon / R$ obtained in UST3D (red diamonds) compared to numerical data from [7] (black curve) and [9] (crosses) as well as to experimental data from [10] (white circles) and [11] (black points).

Figure 7. Angle of attack effect on drag coefficient for $M_\infty = 3$.

Figure 8. Mach number field for the case of $M_\infty = 3$ and angle of attack $\alpha = 20^\circ$. 
4. Conclusion

Computer modeling of a flow over blunted cylinder-flare configuration using UST3D code was carried out. Comparison of resulting computational and experimental drag coefficient distributions over a wide range of Mach numbers has shown good accuracy of numerical method predictions at zero angle of attack. Furthermore, it was demonstrated that UST3D program estimates the bow shock deviation distance reasonably well. An additional computation with non-zero incident flow angle of attack was performed to illustrate angle of attack effect on drag force and on the flow structure. The numerical simulation has provided an extra UST3D code validation and has testified the program to be a robust instrument for aerodynamic calculations of widespread configurations.

Acknowledgments

Author is grateful to professor S T Surzhikov and to the staff of Laboratory of Radiation Gas Dynamics (Ishlinsky Institute for Problems in Mechanics RAS) for useful discussions on the problems considered in the study.

References

[1] Krasil'shchikov A P and Guryashkin L P 2007 Experimental investigation of bodies of revolution in hypersonic flows (Moscow: PhysMathLit) p 208 (in Russian)
[2] Surzhikov S T 2017 Validation of computational code UST3D by the example of experimental aerodynamic data J. Phys.: Conf. Ser. 815 012023
[3] Zheleznyakova A L and Surzhikov S T 2013 Application of the Method of Splitting by Physical Processes for the Computation of a Hypersonic Flow over an Aircraft Model of Complex Configuration High Temperature 51 (6) pp 816–829
[4] Ewans M V and Harlow F H 1957 The particle-in-cell method for hydrodynamic calculations Los Alamos Scientific Lab. Rept. N LA–2139 Los Alamos
[5] Belotserkovsky O M and Davydov Yu M 1982 Method of the Large Particles for Gas Dynamics (Moscow: Nauka) p 391 (in Russian)
[6] Marchuk G I 1988 Splitting methods (Moscow: Nauka) p 263 (in Russian)
[7] Bazzhin A P, Blagosklonov V I, Minailos A N and Pirogova S V 1971 Supersonic flow of a perfect gas over a sphere Uch. Zap. TsAGI 2 (3).
[8] Krasil'shchikov A P and Podobin V P 1968 Experimental study of sphere aerodynamic characteristics in free flight up to M ~ 15 Fluid Dynamics 3 (4) pp 132–134
[9] Lyubimov A N and Rusanov V V 1970 Gas Flows Over Blunt Bodies (Moscow: Nauka) p 668 (in Russian)
[10] Lipman G V and Roshko A 1960 Elements of Gas Dynamics (Moscow: IL) p 352 (in Russian)
[11] Maslennikov V G 1967 Study of detached shock wave location for supersonic flow past ellipsoids of revolution in gases with different intramolecular structure Aerophysical Studies of Supersonic Flows (Moscow, Leningrad: Nauka) p 241 (in Russian)