Analysis of the high speed gas flow over a sphere in the range of Mach numbers 2–12

N A Kharchenko\textsuperscript{1,2,3}, M A Kotov\textsuperscript{1}

\textsuperscript{1} Ishlinsky Institute for Problems in Mechanics RAS, Vernadsky prospect 101(1), Moscow, 119526, Russia
\textsuperscript{2} Tactical Missiles Corporation JSC, Ilica street 7, Korolev, Moscow region, 141080, Russia
\textsuperscript{3} Moscow Physical–Technical Institute, Institutskiy per. 9, Dolgoprudny, Moscow region, 141701, Russia

E-mail: kharchenko.na@phystech.edu

Abstract. In this paper, the results of analysis of the high speed gas flow over a sphere are presented. Numerical investigations were carried out using the UST\textsuperscript{3D} code developed at the Ishlinsky Institute for Problems in Mechanics of RAS (IPMech RAS). The code implements numerical simulation of aerothermodynamics of aircrafts in the context of integration of the full system of Navier–Stokes equations on unstructured mesh using the perfect gas model.

1. Introduction

During many years the investigations of the drag of the sphere moving in the gas flow remains relevant. The papers [1–7] are devoted to investigation of this problems.

The main goal of this work is the validation of the UST\textsuperscript{3D} code in a context of analysis of a high speed flow over a sphere [8]. The code-calculated data were validated against experimental data obtained during study of aerodynamic characteristics of a sphere in a free flight [9]. Tests were performed in the aeroballistic research facility of the Central Aerohydrodynamic Institute (TsAGI) [10] in the range of Mach numbers 1.5 – 15.2 and in the range of Reynolds numbers $10^6$–$10^7$.

During the study, a large amount of experimental data concerning the drag coefficient $C_x$ was obtained. Experiments were performed as follows: the sphere was shot from the launch tube with a caliber of 14.5 mm in the direction opposite to the supersonic flow in the wind tunnel [11]. The diameter of spherical model used was about 10 mm.

2. Computational model

The UST\textsuperscript{3D} code is designed for numerical investigation of aerothermodynamics of high-speed aircrafts and can be applied in a wide range of Mach numbers, altitudes, and angles of attack [8, 12]. The code is based on solving the three-dimensional unsteady system of Navier–Stokes equations (1)–(4) [13, 14].

\[
\frac{\partial w}{\partial t} + \frac{\partial F^x(w)}{\partial x} + \frac{\partial F^y(w)}{\partial y} + \frac{\partial F^z(w)}{\partial z} = \frac{\partial G^x(w)}{\partial x} + \frac{\partial G^y(w)}{\partial y} + \frac{\partial G^z(w)}{\partial z},
\]
\[
\mathbf{w} = \begin{pmatrix}
\rho \\
\rho u \\
\rho v \\
\rho w \\
\rho E
\end{pmatrix},
\]

\[
F^x = \begin{pmatrix}
\rho u \\
\rho u^2 + p \\
\rho uv \\
\rho uw \\
\rho uE + pu
\end{pmatrix}, \quad F^y = \begin{pmatrix}
\rho v \\
\rho uv \\
\rho v^2 + p \\
\rho vw \\
\rho vE + pv
\end{pmatrix}, \quad F^z = \begin{pmatrix}
\rho w \\
\rho uw \\
\rho vw \\
\rho w^2 + p \\
\rho wE + pw
\end{pmatrix},
\]

\[
G^x = \begin{pmatrix}
0 \\
\tau_{xx} \\
\tau_{xy} \\
\tau_{xz} \\
u \tau_{xx} + v \tau_{xy} + w \tau_{xz} - q_x
\end{pmatrix}, \quad G^y = \begin{pmatrix}
0 \\
\tau_{xy} \\
\tau_{yy} \\
\tau_{yz} \\
u \tau_{xy} + v \tau_{yy} + w \tau_{yz} - q_y
\end{pmatrix}, \quad G^z = \begin{pmatrix}
0 \\
\tau_{xz} \\
\tau_{yz} \\
\tau_{zz} \\
u \tau_{xz} + v \tau_{yz} + w \tau_{zz} - q_z
\end{pmatrix},
\]

The complete set of Navier–Stokes equations is used in conjunction with the equation of state for ideal gas (5).

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

The UST3D code performs numerical integration of this system by using the method of splitting on physical processes. The method is explicit and has the first order approximation with respect to time and the second order approximation with respect to space. Solution of the problem is obtained as the result of relaxation [8, 12].

As one of approaches to calculation of flows through interface boundaries of mesh cells used in the UST3D, the HLLE solver was implemented [15]. The HLLE solver was used in order to prevent emergence of the carbuncle-type effect during computations. Emergence of this effect is a characteristic for bodies with large-radius blunt noses.

In all Godunov-type methods, the flow through the cell interfaces is calculated from the Riemann problem solution [16]. Using the exact Riemann solver proposed by S. K. Godunov requires a costly iterative procedure. However, a large number of approximate Riemann solvers are available now, which makes the S. K. Godunov’s approach quite efficient for solving the problems of unstructured meshes. In this work, the HLLE method is used as an approximate Riemann solver [15].

The procedure of approximate Riemann solver for flux computation (well-known as HLL) was proposed by Harten, Lax, and van Leer [17]. The method is based on consideration of a simplified two-wave configuration without taking into account the contact discontinuity. The key difference of the HLLE (Harten – Lax – van Leer – Einfeldt) method is the Roe-averaged density of the sound speed used to find the maximum and minimum wave speeds.

The flux of the HLLE method is determined from the following relation [18]:

\[
\]
\[ F_{HLLR} = \begin{cases} F_L & \text{при } 0 \leq S_L \\ S_R F_L - S_L S_R (w_R - w_L) & \text{при } S_L \leq 0 \leq S_R \\ S_R - S_L & \text{при } 0 \geq S_R \\ F_R & \end{cases} \]

Speeds of the left and right shock waves \( S_L \) and \( S_R \) are determined from the minimum and maximum disturbance speeds [18]:

\[ S_L = V_L - c_L, \quad S_R = V_R + c_R, \quad (7) \]

\[ S_L = \min \{ V - c, V_L - c_L \}, \quad S_R = \max \{ V + c, V_R + c_R \}, \quad (8) \]

\[ \bar{V} = \frac{\sqrt{\rho_L V_L} + \sqrt{\rho_R V_R}}{\sqrt{\rho_L} + \sqrt{\rho_R}}, \quad \bar{c} = \left[ (\gamma - 1) \left( H - \frac{1}{2} \bar{V}^2 \right) \right]^\frac{1}{3}, \quad \bar{H} = \frac{\sqrt{\rho_L H_L} + \sqrt{\rho_R H_R}}{\sqrt{\rho_L} + \sqrt{\rho_R}}, \quad (9) \]

The pressure distribution over the interface can be calculated from the following relations:

\[ \bar{p} = 0.5 (p_L + p_R) - 0.5 (V_R - V_L) \bar{c}, \]

\[ \bar{\rho} = 0.5 (\rho_L + \rho_R), \quad \bar{c} = 0.5 (c_L + c_R), \quad (10) \]

Hight viscosity of this method allows eliminating the carbuncle-type effects during the computation.

3. Problem statement for computing flows around the sphere

For numerical simulation of the flow, a three-dimensional surface of the sphere was created with the diameter of 10 mm.

In this work, simulation of three-dimensional flow field over a sphere was performed using unstructured tetrahedral meshes. The computational mesh was refined towards the surface of the sphere. The number of triangles on the surface was 36 354 elements, and the overall number of tetrahedrons in the volume was 2 920 576 cells (figure 1).

**Figure 1.** Volume computational mesh.
4. Results obtained
A series of computations of a flow over a sphere was carried out for Mach numbers from 2 to 12, in increments of one Mach. The pressure and density parameters in incident flow were $p_\infty = 0.405 \times 10^4 \text{ Pa}$ and $\rho_\infty = 0.646 \times 10^1 \text{ kg/m}^3$ respectively at $Re = 10^4–10^5$. The results of computations for $M = 6$ are shown in figures 2–4. Results obtained with the UST3D code where the HLLE solver is implemented are presented in the right-hand side of figures 2–4. Evidently, carbuncle-type effects are absent in the leading shock wave, when fluxes through computational cells are calculated using the HLLE method.

![Figure 2: Pressure distribution ($p/10^5$) over the surface and in the vicinity of the sphere.](image1)

![Figure 3: Density distribution ($\rho/\rho_\infty$) over the surface and in the vicinity of the sphere.](image2)
Figure 4. Mach number distribution over the surface and in the vicinity of the sphere.

Figure 5. Change of the aerodynamic drag coefficient of the sphere as a function of the Mach number.

From figure 5 one can see that obtained calculated data are in a close agreement with the experimental data [9, 11]. Difference in obtained values of the drag coefficient $C_x$ can occur due to the presence of carbuncle-type effect in computations.
Conclusion
Within the framework of the computer code validation a good agreement between the calculated and the experimental data for the air flow over the sphere was obtained. This is important for the future UST3D code applications to aerothermodynamics analysis of aircraft with large-radius blunt noses.

The presented study was partially supported by Russian State Assignment under contract No. AAAA-A17-117021310372-6 and partially supported by Russian Foundation for Basic Research grant No. 16-01-00379.

References
[1] Hodjes A J 1957 The Drag Coefficient of Very High Velocity Spheres *J. of the Aeronautical Sciences* pp 755–758
[2] Charters A C and Thomas R N 1945 The Aerodynamic Performance of Small Spheres from Subsonic to High Supersonic Velocities *Journal of the Aeronautical Sciences* pp 468–476
[3] May A and Witt W R 1953 Free-Flight Determinations of the Drag Coefficients of Spheres *J. of the Aeronautical Sciences* pp 635–638
[4] Bailey A B and Hiatt J 1972 Sphere Drag Coefficients for a Broad Range of Mach and Reynolds Numbers *AIAA J.* 10 pp 1436–1440
[5] Lyons W C, Brady J J and Levensteins J 1964 Hypersonic Drag, Stability and Wake Data for Cones and Spheres *AIAA J.* 2 pp 1448–1456
[6] Ross F W and Willmarth W W 1971 Some Experimental Results on Sphere and Disk Drag *AIAA J.* 9 pp 285–291
[7] Kotov M A, Kryukov I A, Ruleva L B, Solodovnikov S I, and Surzhikov S T 2015 Supersonic Air Flows Around Some Geometrical Primitives 33rd *AIAA Applied Aerodynamics Conf.* AIAA 2015–3012
[8] Surzhikov S T 2017 Validation of computational code UST3D by the example of experimental aerodynamic data *J. of Physics: Conference Series* 815 012023
[9] Krasilshikov A P and Podobin V P 1968 Experimental research of free flight ball aerodynamic characteristics up to the Mach numbers M ~ 15. *Fluid Dynamics* 4 pp 190–193
[10] http://www.tsagi.ru
[11] Krasilshikov A P and Guryashkin L P 2007 Bodies of revolution experimental research in hypersonic flows (Moscow: Fizmatlit) p 208 (in Russian)
[12] 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 pp 816–829
[13] Surzhikov S T 2013 Convective heating of small-radius spherical blunting for relatively low hypersonic velocities *High Temperature* 51 pp 231–245
[14] Surzhikov S T 2017 Spatial Multiphysics Models of the Radiation Gas Dynamics of Super Orbital Re–Entry Space Vehicles. *J. of Physics: Conference Series* 837 012021
[15] Einfeldt B, Munz C D, Roe P L and Sjögreen B 1991 On Godunov–Type Methods near Low Densities *J. of Computational Physics* 92 pp 273–295
[16] Godunov S K 1959 A finite-difference method for the numerical computation and discontinuous solutions of the equations of fluid dynamics *Mat. Sb.* 47 pp 271–306
[17] Harten A, Lax P D and van Leer B 1983 On Upstream Differentiating and Godunov–Type Schemes for Hyperbolic Conservation Laws *SIAM Review* 25 pp 35–61
[18] Toro E F 2009 Riemann Solvers and Numerical Methods for Fluid Dynamics *Springer* p 724