NUMERICAL INVESTIGATION OF TWO-PHASE GAS–PARTICLE FLOW IN A HYPERSONIC SHOCK TUNNEL

Anatoly A. Veryovkin and Yury M. Tsirkunov

Baltic State Technical University, Faculty of Aerospace Engineering
1, 1st Krasnoarmeiskaya str., 190005 Saint Petersburg, Russia
e-mail: tsrknv@bstu.spb.su

Key words: Two-phase gas–solid flow, unsteady flow, shock tube, Laval nozzle, hypersonic flow over a body

Abstract. The paper describes the results of computational simulation of an unsteady two-phase gas-particle flow in a hypersonic shock tunnel. The tunnel consists of a heated shock tube, a convergent-divergent nozzle linked up with the low-pressure chamber of the tube, a test section where a model is placed, and a exhaust chamber. In calculations, the configuration of the tunnel, and the initial parameters in the high-pressure and low-pressure chambers of the shock tube were taken as in experiments carried out at TsAGI (Russia). The designed Mach number at the nozzle exit was equal to \( M = 6.01 \). Particles were injected into the high-pressure chamber of the shock tube just before the diaphragm destructed. Their initial concentration was very low, so that the reverse effect of the dispersed phase on the carrier gas was negligible. The carrier gas flow was described by the Euler equations which were solved by the finite-volume method of the second order. In the calculation domain, an unstructured grid was used with a high concentration of nodes near the critical cross-section of the nozzle and in the vicinity of the body in the test section. Triangular and polygonal finite-volume cells were taken. The fluxes at cell boundaries were calculated using four different algorithms of the TVD-type. The best quality of the numerical solution for the flow field was obtained with the use of the modified second-order Godunov method for fluxes applied to the polygonal cells. The particle-phase flow was simulated using the Lagrangian method. A fine structure of two-phase flow was investigated from the instant of destruction of a diaphragm between high- and low-pressure chambers of the shock tube to the end of the quasi-steady-state flow over a body in the test section.

1 INTRODUCTION

A hypersonic shock tunnel is a powerful experimental tool in supersonic aerodynamics. The flow in such tunnels was a subject of intensive study in the midtwentieth century. The list of scientific publications on this subject compiled by the editors of the collection of papers [1] contains about seven hundred references. The first major problem faced engineers that time was to obtain a uniform steady-state gas flow with a high supersonic Mach number at the
inlet of the test section for a time interval sufficient for measuring the flow parameters near a model in this section. The main directions of efforts of researchers were to develop a theory of flow in a shock tube of variable cross-section including a theory of shock wave propagation, and to estimate the influence of numerous effects not included in the theory. At that time, a direct numerical simulation of flow in a hypersonic shock tunnel was impossible, and fine flow structure in the whole tunnel remained unknown for a long time. Quite recently, shock tunnels have begun to be used in experiments on hypersonic dusty gas flows over bodies, and the first studies here were carried out in the frame of an extensive international research programme related to a possible expedition to the Mars. The presence of dispersed particles can modify essentially the effect of flow onto a model compared with the case of a pure gas flow. For example, the heat flux at the stagnation point of a blunt body in a dusty gas flow is several times as greater as in a flow of a pure gas, even if particles are very small, so that they do not reach the body surface and do not deposit on it [2]. For understanding such and other phenomena, as well as for correct interpretation and explanation of experimentation data, it is necessary to determine the behaviour of both phases from the instant of opening a diaphragm between the high-pressure and low-pressure chambers of a shock tube up to the end of quasi-steady-state two-phase flow around a model. First steps on computational simulation of gas flow in a shock tube with an adjacent convergent-divergent nozzle were made apparently in the very late 1960s (see, for example, [3]). Since then many separate aspects of gas flow in a shock tube have been investigated numerically. One of the recent work here has been devoted to the effect of diaphragm opening time on a shock-tube flow [4]. At the same time, the behaviour of two-phase flow in a hypersonic shock tunnel remains an open question at present, and, as a result, very expensive experiments have no an adequate theoretical accompaniment.

The aim of the present study is to investigate numerically the flow of a dusty gas in a hypersonic shock tunnel including the flow over a model placed in a test section. Configuration and sizes of the setup components corresponded in calculations to the shock tunnel UT-1M used at TsAGI (Russia). The initial parameters in the high-pressure chamber of the shock tube and in the exhaust chamber were taken as in experiments [5].

2 SETTING OF THE PROBLEM FOR A TWO-PHASE FLOW IN A HYPERSONIC SHOCK TUNNEL

2.1 Schematic of a hypersonic shock tunnel and scheme of a dusty gas flow

A schematic of the TsAGI UT-1M shock tunnel is shown in Fig.1. The length of a high-pressure chamber is 6 m. The gas in this chamber is preliminary heated up to 600 K to avoid its condensation when expanding in the divergent part of the nozzle and in the exhaust chamber. The diameter of the shock tube is 76 mm. The diaphragm is positioned at the distance of 350 mm in front of the nozzle inlet. The nozzle is contoured with the throat and exit diameters of 38 mm and 300 mm, respectively. The designed Mach number at the nozzle exit is equal to 6.01 for the air. Particles are being introduced into the high-pressure chamber from a two-phase mixer through the inlet pipes just before the start up of the shock tube. More detailed description of
the mixture inlet system is given in [5]. In the high-pressure chamber, the initial pressure is 17 to 32 bar and in the exhaust chamber it is $10^{-2}$ bar or less. The particle size and the particle concentration can be varied.

Figure 1: Schematic of a hypersonic shock tunnel

Just after opening of the diaphragm, a shock wave travels towards the nozzle and a rarefaction wave propagates through the gas at rest in the high-pressure chamber. The shock wave interacts with the nozzle boundaries. This process is being accompanied by forming a very complicated shock-wave structure in the area near the nozzle throat that results in a non-uniformity gas flow in a nozzle and further in a test section. Particles are being carried along the tunnel, however their distribution at the cross-section of the tunnel can become non-uniform due to the non-uniformity of the gas flow and the particles’ reflection from the nozzle walls. Particles, due to their inertia, have a velocity lag relative to the gas flow, and this lag depends on the particle size. It is quite clear that for correct interpretation and explanation of the experimental results, it is very important to know the flow fields of both phases in the test section.

2.2 Governing equations, boundary and initial conditions for the carrier gas

The actual particle concentration in the flow is very low (the initial particle volume fraction in the high-pressure chamber has not exceeded $10^{-6}$), so that we may assume that the reverse effect of the dispersed phase on the carrier gas is negligible. In this case, the two-phase flow problem can be reduced to two sequential problems: modelling of the carrier gas flow and calculation of the particle’s motion in the predetermined gas flow field.

Theoretical formulation of the problem on the carrier gas flow includes the balance and constitutive equations, and the appropriate boundary and initial conditions. Introduce the cylindrical coordinates $(x, y, \varphi)$: the axis $x$ is directed along the axis of symmetry of the tunnel from the bottom of the high-pressure chamber towards the test section, $y$ is normal to it, $\varphi$ is the azimuthal angle. For description of unsteady axially symmetric flow (flow parameters do not depend on $\varphi$) of the carrier gas (air) in the tunnel, we use the Euler equations which can be written in the following compact form [6]

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{R}{y} = 0,$$\hspace{1cm}(1)
\[ \mathbf{U} = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho \mathbb{E} \end{pmatrix}, \quad \mathbf{F} = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ (\rho e + p)u \end{pmatrix}, \quad \mathbf{G} = \begin{pmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ (\rho e + p)v \end{pmatrix}, \quad \mathbf{R} = \begin{pmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ (\rho e + p)v \end{pmatrix}. \] (2)

We assume air to be thermodynamically perfect with constant specific heats, then we have
\[ e = c_v T + \frac{u^2 + v^2}{2}, \quad p = \rho R T. \] (3)

Here \( t \) is the time; \( \rho, p, e, T \) are the density, the pressure, the total energy, and the temperature, respectively; \( u \) and \( v \) are the \( x \)- and \( y \)-components of the velocity vector \( \mathbf{v} \); \( c_v \) is the specific heat at constant volume; \( R \) is the gas constant.

The calculation domain in the \((xy)\)-plane was bounded by the contour of the tunnel, the axis of symmetry, and the exit cross-section of the test section. At the rigid boundaries, we used the conditions of zero normal velocity. At the axis of symmetry, the velocity component \( v \) and the derivatives \( \partial u / \partial y, \partial p / \partial y, \partial T / \partial y \) were taken to be zero (the conditions of symmetry). At the test section exit, no conditions are to be imposed.

### 2.3 Governing equations and the initial conditions for the particles.

**Particle–wall collision model**

The dispersed phase is treated as a set of a large number of simulated particles which move without collision between each other, but they can collide with the walls of the tunnel (for example, in the convergent part of the nozzle). The motion of simulated particles are described in the Lagrangian formulation.

We assume the particles to be solid spheres of equal radii \( r_p \). The momentum and the angular momentum equations, and the relation for the position vector of a particle can be written in the form
\[ m_p \frac{d\mathbf{v}_p}{dt} = \mathbf{f}_D + \mathbf{f}_M, \quad I_p \frac{d\omega_p}{dt} = \mathbf{l}_p, \quad \frac{d\mathbf{r}_p}{dt} = \mathbf{v}_p, \] (4)

Here, \( m_p = (4/3)\pi r_p^3 \rho, I_p = (2/5)m_p r_p^2 \), \( \mathbf{v}_p \) and \( \omega_p \) are the mass, the moment of inertia, the velocity, and the angular velocity of a particle. The model of action of the carrier gas on a particle includes the drag force \( \mathbf{f}_D \), the lift Magnus force \( \mathbf{f}_M \), and the damping torque \( \mathbf{l}_p \) (these components are the most important in the problem under consideration)
\[ \mathbf{f}_D = \frac{1}{2} C_D \pi r_p^2 \rho |\mathbf{v} - \mathbf{v}_p| (\mathbf{v} - \mathbf{v}_p), \quad \mathbf{f}_M = \frac{4}{3} C_\omega \pi r_p^3 \rho [(\omega - \omega_p) \times (\mathbf{v} - \mathbf{v}_p)], \quad \omega = (1/2)\text{curl} \mathbf{v}, \] (5)
\[ \mathbf{l}_p = \frac{1}{2} C_l r_p^5 \rho |\omega - \omega_p| (\omega - \omega_p). \]
The coefficients $C_D$, $C_\omega$ and $C_I$ in (5) depend on different dimensionless parameters associated with the flow around a particle. These coefficients are calculated from the approximate formulae which are constructed on the basis of theoretical solutions, numerical results and experimental data. We have used the Henderson relations [7] for $C_D$

$$C_D(\text{Re}_p, M_p, T_p/T) = \begin{cases} C_{D1}^1, & 0 < M_p \leq 1, \\ C_{D1}^1 + \frac{4}{3}(M_p - 1)(C_{D2}^2 - C_{D1}^1), & 1 < M_p \leq 1.75, \\ C_{D2}^2, & M_p > 1.75, \end{cases}$$

where

$$C_{D1}^1(\text{Re}_p, M_p, T_p/T) = 24 \left\{ \text{Re}_p + \sqrt{\frac{2}{2}} M_p \left[ 4.33 + \frac{3.65 - 1.53 T_p/T}{1 + 0.353 T_p/T} \exp \left( -0.247 \sqrt{\frac{2 \text{Re}_p}{\gamma M_p}} \right) \right] \right\}^{-1} + \left[ \frac{4.5 + 0.38(0.03 \text{Re}_p + 0.48 \sqrt{\text{Re}_p})}{1 + 0.03 \text{Re}_p + 0.48 \sqrt{\text{Re}_p}} + 0.1 M_p^2 + 0.2 M_p^4 \right] \exp \left( -\frac{M_p}{2 \sqrt{\text{Re}_p}} \right) + 0.6 \sqrt{2} M_p \left[ 1 - \exp \left( -\frac{M_p}{\text{Re}_p} \right) \right],$$

$$C_{D2}^2(\text{Re}_p, M_p, T_p/T) = \left[ 0.9 + \frac{0.34}{M_p^2} + 1.86 \sqrt{\frac{M_p}{\text{Re}_p}} \left( 2 + \frac{8}{\gamma M_p^2} + \frac{2.116}{\gamma M_p} \sqrt{\frac{T_p}{T}} - \frac{4}{\gamma^2 M_p^4} \right) \right] \left( 1 + 1.86 \sqrt{\frac{M_p}{\text{Re}_p}} \right)^{-1}.$$

Here $\text{Re}_p = 2 \rho |\textbf{v} - \textbf{v}_p| r_p/\mu$ and $M_p = |\textbf{v} - \textbf{v}_p|/\sqrt{\gamma k} T$ are the relative particle Reynolds and Mach numbers, $C_{D1}^1$ is the value of $C_D^1$ at $M_p = 1$, and $C_{D2}^2$ is the value of $C_D^2$ at $M_p = 1.75$. The dependence of $C_D$ on $T_p/T$ is very weak in the flow under consideration. That is why we have ignored this dependence, and the ratio $T_p/T = 1$ has been taken as unity.

For calculation of $C_\omega$ we have used the exact solution by Rubinow and Keller [8] and the formula suggested by Oesterlé and Bui Dinh [9]

$$C_\omega = \begin{cases} 3/4, & 2 \gamma_\omega < 0.45, \\ (3/8) \gamma_\omega [0.45 + (2 \gamma_\omega - 0.45) \exp(-0.075 \gamma_\omega^{0.4} \text{Re}_p^{0.7})], & 2 \gamma_\omega \geq 0.45, \end{cases}$$

where $\gamma_\omega = \omega_p r_p/|\textbf{v} - \textbf{v}_p|$. 

5
The expression for the coefficient \( C_l \) has been taken in the form [10]

\[
C_l = \frac{C_{l1}}{\sqrt{\text{Re}_{pw}}} + \frac{C_{l2}}{\text{Re}_{pw}},
\]

where \( \text{Re}_{pw} = \rho|\omega - \omega_p| r_p^2 / \mu \), and \( C_{l1} \) and \( C_{l2} \) are given in Table 1.

The initial conditions for the system of equations (4) are specified from the following considerations. It is assumed that at the moment of start up of the shock tube (\( t = 0 \)), the particles in the high-pressure chamber are suspended in the carrier gas forming a uniform two-phase mixture, and their velocities and angular velocities are equal to zero.

Being involved into the motion by the carrier gas, particles, due to their inertia, do not follow the gas streamlines and can collide with the walls of the tunnel. We have used the semi-empirical particle–wall collision model [11] for calculation of the parameters of a particle just after its rebound. The model is based on the laws of mechanics and the experimental data for the restitution coefficients of the normal and tangential to the wall velocities of the center of gravity of a particle. This model is valid at moderate and high particle impact velocities. The final relations for the normal, tangential, and rotational velocities of a particle after its rebound take the form

\[
v_{p2} = -a_n v_{p1},
\]

\[
u_{p2} = \begin{cases} u_{p1}a_r + \omega_{p1} r_p (a_r - 1), & \beta_1 < \beta_s, \\ u_{p1}a_r - \frac{2}{7} \omega_{p1} r_p, & \beta_1 \geq \beta_s. \end{cases}
\]

\[
\omega_{p2} = \begin{cases} \frac{5}{2} u_{p1} (a_r - 1) + \frac{5}{2} \omega_{p1} \left( a_r - \frac{3}{5} \right), & \beta_1 < \beta_s, \\ -\frac{u_{p1}}{r_p} a_r + \frac{2}{7} \omega_{p1}, & \beta_1 \geq \beta_s. \end{cases}
\]

Here, \( v_{p1}, u_{p1}, \) and \( \omega_{p1} \) are the normal, the tangential and the angular velocity of a particle just before its collision; \( a_n \) and \( a_r \) are the restitution coefficients; \( \beta_1 \) is the angle of collision (the angle between the velocity vector of a particle before a collision \( v_{p1} \) and the wall); \( \beta_s \) is the critical value of \( \beta_1 \): if \( \beta_1 < \beta_s \) we have sliding collision, if \( \beta_1 \geq \beta_s \) a collision is non-sliding [11].
We have used the following formulae for calculation of the restitution coefficients [11]

\[ a_n = 1 - \left[ 1 - \exp(-0.1 \nu p_1^{0.61}) \right] \sin \beta_1, \]

\[ a_r = C_0 + C_1 \left( \frac{\pi}{2} - \beta_1 \right)^2 + C_2 \left( \frac{\pi}{2} - \beta_1 \right)^4 + C_3 \left( \frac{\pi}{2} - \beta_1 \right)^6, \]

The coefficients in the last formula depend on the wall and particle materials. In calculations, they have been taken as \( C_0 = 0.690, C_1 = -0.288, C_2 = 0.1140, \) and \( C_3 = 0.0219 \) that corresponds to hard particles like corundum or silicon dioxide and a steel wall.

At the axis of symmetry, the condition of specular particles’ reflection has been used.
At the test section outlet, no conditions for particles are needed.

3 NUMERICAL METHOD

Theoretically, the two-phase flow problem formulated above can be reduced to the sequence of two problems. At first, an unsteady carrier gas flow field is found from the equations (1)–(3) with the appropriate initial and boundary conditions, and then the motion of particles is determined in this flow field from the equations (4). However, it is much more convenient in computational simulation to calculate flow parameters of both phases simultaneously. Below, we consider numerical algorithms for the carrier gas and the particles.

3.1 Unstructured grid and configuration of control volumes

The computation domain in \((xy)\)-plane was bounded by the walls of the shock tube, the nozzle, the test section and the body, the axis of symmetry, and the outlet of the test section. A strongly nonuniform unstructured triangular grid (the ratio of the maximal cell size to the minimal one reached \( \sim 400 \)) was constructed in this domain with high concentration of nodes in those areas where the flow structure was expected to be complicated (in the nozzle throat and in the shock layer of a body in the test section). Then two types of finite-volume meshes associated with this grid were introduced. The first mesh was triangular and coincided with the grid, whereas another one has polygonal (hexagonal, as a rule) finite-volume cells. A fragment of a grid and configuration of two types of finite-volume cells are shown in Fig. 2.

![Figure 2: Fragment of a grid with triangular (left) and polygonal (right) finite volumes](image-url)
Calculations using the mesh of polygonal finite volumes gave better resolution and quality of numerical flow fields compared with those for the mesh of triangular finite volumes. All numerical results presented in this paper are obtained with the use of polygonal finite volumes. The total number of triangular grid nodes in calculations was about $3.25 \cdot 10^5$.

3.2 Finite-volume relations for the carrier gas

An explicit finite-volume method was used for numerical solving a carrier gas flow field. Consider a polygonal $i$-th cell $C_i$ (Fig. 3), and let $\Omega_i$ and $S_i$ be its volume and surface, respectively. Integrating the equations (1) over a control volume $\Omega_i$ and applying the Gauss's theorem we obtain the following integral form of the Euler equations

$$
\frac{\partial}{\partial t} \int_{\Omega_i} U \, d\Omega + \int_{S_i} (F \, n_x + G \, n_y) \, dS + \int_{\Omega_i} \frac{R}{n_i} \, d\Omega = 0,
$$

where $U$, $F$, $G$ and $R$ are defined by (2), and $n(n_x, n_y)$ is the unit vector normal to $S_i$ and pointing outward the volume $\Omega_i$, so that we have

$$
F \, n_x + G \, n_y = \begin{pmatrix}
\rho (v \cdot n) \\
\rho u (v \cdot n) + p n_x \\
\rho v (v \cdot n) + p n_y \\
(\rho e + p) u (v \cdot n)
\end{pmatrix},
$$

The same relation can be written for every cell of a finite-volume mesh.

![Figure 3: Polygonal cell $C_i$ as a control volume $\Omega_i$ for integral form of the Euler equations](image)

Applying the mean value theorem for the integrals over the volume $\Omega_i$ in (6) and using the cell-centered finite-volume discretization, one can obtain

$$
\frac{\partial U}{\partial t} \mid_{\Omega_i} + \int_{S_i} (F \, n_x + G \, n_y) \, dS + \frac{R}{n_i} \mid_{\Omega_i} = 0,
$$
where \( \mathbf{U}_i \) and \( \mathbf{R}_i \) are assigned to the cell center (more exactly, to the center of gravity of the polygon \( C_i \) which may, in the general case, not to coincide with the node of the triangular grid inside \( C_i \)), \( y_i \) is the distance of the cell center from the axis of symmetry. The control volume surface \( S_i \) consists of several faces \( S_j \) (7 faces in Fig. 3). Representing the integral over \( S_i \) as a sum of integrals over its faces and applying the mean value theorem to every summand we obtain

\[
\int_{S_i} (\mathbf{F} n_x + \mathbf{G} n_y) \, dS = \sum_j (\mathbf{F} n_x + \mathbf{G} n_y)_j S_j,
\]

where \( (\mathbf{F} n_x + \mathbf{G} n_y)_j \) is the vector of mean fluxes of the corresponding flow properties (mass, momentum, or energy) through the face \( S_j \).

In calculations, the predictor-corrector algorithm which yields the second-order of accuracy in time has been used

\[
\mathbf{U}_i^{n+1} = \mathbf{U}_i^n - \Delta t \left[ \frac{1}{V_i} \sum_j (\mathbf{F} n_x + \mathbf{G} n_y)_j S_j + \frac{\mathbf{R}_y}{y} \right]_i^n, \quad i = 1, \ldots, N_{CV}, \quad (8)
\]

\[
\mathbf{U}_i^{n+1} = \mathbf{U}_i^n - \Delta t \left[ \frac{1}{V_i} \sum_j (\mathbf{F} n_x + \mathbf{G} n_y)_j S_j + \frac{\mathbf{R}_y}{y} \right]_i^*, \quad i = 1, \ldots, N_{CV}. \quad (9)
\]

Here \( N_{CV} \) is the number of finite volumes of the mesh. The summation in (8) and (9) is over all faces of the control volume surface \( S_i \).

A time step \( \Delta t \) was chosen from the Courant–Friedrichs–Levy condition [6].

### 3.3 Calculation of fluxes through the faces of the control volume

For calculation of the flux terms in (8) and (9), we applied four methods: the modified second-order Godunov method [12], the Kotov scheme [13], the Rodionov scheme [14, 15], and the Osher–Solomon scheme [16]. In preliminary test calculations, the best results were obtained with the use of the Godunov method key features of which are described below.

The components of the flux vector \( (\mathbf{F} n_x + \mathbf{G} n_y)_j \) can be determined if we know the fluxes of the mass, the momentum, and the total energy through the face \( S_j \) between the control volume \( C_i \) and the immediate neighbour \( C_j \) (see Fig. 3). These components are calculated in the present study from the exact solution of the Riemann problem. The Riemann solver is applied to the flow parameters which are calculated at the middle of the face \( S_j \) from a linear approximation of the components of \( \mathbf{U} \) in the neighbouring cells \( C_i \) and \( C_j \).

Consider in detail the technique of calculation of \( \mathbf{U} \) for the Riemann solver. We denote the middle of the face \( S_j \) by the double subscript \( ij \). Going round the cell \( C_i \) in an anticlockwise direction we define flow properties in the cell \( C_i \) as \( left \) and the ones in every neighbouring cell as \( right \). Thus, \( (\mathbf{U}_{ij})_l \) and \( (\mathbf{U}_{ij})_r \) are the vector \( \mathbf{U} \) calculated at the middle of \( S_j \) from linear varying of flow properties across the cells \( C_i \) and \( C_j \), respectively. A linear function \( \mathbf{U}(x, y) \)
within $C_i$ can be determined if we know $U_i$ and two derivatives $\partial U_i/\partial x$ and $\partial U_i/\partial y$. Consider the cell $C_i$ and the series of two cells $C_j$ and $C_{j+1}$ (see Fig. 4). Denote $U$ at the centers of these cells by $U_i$, $U_j$ and $U_{j+1}$, respectively. Let the co-ordinates of the centers be $(x_i, y_i)$, $(x_j, y_j)$, and $(x_{j+1}, y_{j+1})$. Then we may write

\[
U_j = U_i + \left( \frac{\partial U}{\partial x} \right)_{ij} (x_j - x_i) + \left( \frac{\partial U}{\partial y} \right)_{ij} (y_j - y_i), \tag{10}
\]

\[
U_{j+1} = U_i + \left( \frac{\partial U}{\partial x} \right)_{ij} (x_{j+1} - x_i) + \left( \frac{\partial U}{\partial y} \right)_{ij} (y_{j+1} - y_i). \tag{11}
\]

The system of equations (10) and (11) is linear with respect to $(\partial U/\partial x)_{ij}$ and $(\partial U/\partial y)_{ij}$, and these derivatives can be easily found. The same procedure being applied to every set of three cells results in the series of $(\partial U/\partial x)_{ij}$ and $(\partial U/\partial y)_{ij}$ depending on $j$ at fixed $i$. Then we calculate the required derivatives $\partial U_i/\partial x$ and $\partial U_i/\partial y$ from the following relations

\[
\frac{\partial U_i}{\partial x} = \begin{cases} 
\text{sign} \left( \frac{\partial U}{\partial x} \right)_{ij} \min_j \left( \frac{\partial U}{\partial x} \right)_{ij} & , \quad \min_j \left( \frac{\partial U}{\partial x} \right)_{ij} \cdot \max_j \left( \frac{\partial U}{\partial x} \right)_{ij} > 0, \\
0, & \text{otherwise}.
\end{cases} \tag{12}
\]

\[
\frac{\partial U_i}{\partial y} = \begin{cases} 
\text{sign} \left( \frac{\partial U}{\partial y} \right)_{ij} \min_j \left( \frac{\partial U}{\partial y} \right)_{ij} & , \quad \min_j \left( \frac{\partial U}{\partial y} \right)_{ij} \cdot \max_j \left( \frac{\partial U}{\partial y} \right)_{ij} > 0, \\
0, & \text{otherwise}.
\end{cases} \tag{13}
\]

In a similar manner, the derivatives can be calculated for every cell of a mesh. Substantially, the relations (12) and (13) introduce a limiter which allows to avoid oscillations behind strong
shock waves. Using this limiter we have the second order of accuracy in space in those areas where the flow field is smooth and the first order in the vicinities of discontinuities (shock waves and contact surfaces).

Assuming variation of $U$ to be linear within every cell, one can obtain the following formulae for $(U_{ij})_l$ and $(U_{ij})_r$

\[
(U_{ij})_l = U_i + \frac{\partial U_i}{\partial x} (x_{ij} - x_i) + \frac{\partial U_i}{\partial y} (y_{ij} - y_i), \quad (14)
\]

\[
(U_{ij})_r = U_j + \frac{\partial U_j}{\partial x} (x_{ij} - x_j) + \frac{\partial U_j}{\partial y} (y_{ij} - y_j). \quad (15)
\]

The flow parameters calculated on the basis of $(U_{ij})_l$ and $(U_{ij})_r$ represent the input data for the Riemann solver which gives the fluxes of the mass, the momentum, and the total energy through the face $S_j$. These fluxes are used for calculation of components of the flux vector $(F_{nx} + G_{ny})_j$ in (8) and (9). At the faces of control volumes coincided with the boundaries of the calculation domain, the flux vector is found with the use of the boundary conditions.

### 3.4 Numerical method for calculation of the particle motion

For solving the ordinary differential equations (4) describing the motion of particles we applied the predictor-corrector method of the second order

\[
v^n_p = v^n_p + \frac{\Delta t_p}{m_p} (f^n_D + f^n_M), \quad \omega^n_p = \omega^n_p + \frac{\Delta t_p}{I_p} l^n_p, \quad r^n_p = r^n_p + \Delta t_p v^n_p; \quad (16)
\]

\[
v^{n+1}_p = v^n_p + \frac{\Delta t_p}{2 m_p} (f^n_D + f^n_M + f^n_D + f^n_M), \quad \omega^{n+1}_p = \omega^n_p + \frac{\Delta t_p}{2 I_p} (l^n_p + l^n_p), \quad (17)
\]

\[r^{n+1}_p = r^n_p + \frac{1}{2} \Delta t_p (v^n_p + v^n_p).
\]

A time step $\Delta t_p$ was chosen from the condition that a particle does not travel during $\Delta t_p$ over a distance more than a cell size.

### 4 NUMERICAL RESULTS AND DISCUSSION

In computational simulation, the carrier gas was air, the particles’ material was silicon dioxide, the material of setup walls was ductile steel (this is important in a particle–wall collision model). The setup sizes are given in Subsection 2.1. The initial pressure and temperature in the high-pressure chamber were equal to 25 bar and 600 K. In the low-pressure chamber, the nozzle, the test section, and the exhaust chamber, these parameters were equal to 0.01 bar and 290 K.
4.1 Flow structure of the carrier gas flow

At the instant of opening a diaphragm between the high-pressure and low-pressure chambers \((t = 0)\), a strong shock wave is travelling towards the nozzle, and the front of a rarefaction wave propagates towards the closed end of the shock tube. A contact surface separating gases located at the initial instant in the high-pressure and low-pressure chambers is moving behind the shock wave. The shock wave when reaches the nozzle inlet interacts with the nozzle walls resulting in appearance of rather complex shock-wave structure in the flow. It is fully demonstrated by Figs. 5 and 6 from which it is clearly seen, that this interaction is accompanied by numerous regular and Mach reflections of shock waves from the walls and the axis of symmetry. Besides, two well-defined eddies arise (they are shown in Fig. 9 and will be discussed later). As a result, two main shock waves are being formed one of which is travelling in the same direction as the initial shock, whereas another one is travelling in the opposite direction towards the high-pressure chamber. A contact surface (the boundary between red and green areas in Fig. 5 at \(t = 0.280 \text{ ms}\)) meets the latter shock wave \((t = 0.292 \text{ ms})\) and then moves through it. Initially flat shape of the contact surface is becoming distinctly curved (contact surface is identified by the left boundary of the dark area in a bottom picture of each pair in Fig. 7). In contrast, the shock wave travelling towards the closed end of the shock tube being at first complex shape (see, for example, flow field at \(t = 0.340 \text{ ms}\) in Fig. 6), is becoming flat \((t = 600 \text{ ms})\).

In the developing supersonic flow in the divergent part of the nozzle, we can distinguish the main shock wave (boundary between dark and light blue in Fig. 7) propagating through the stagnant gas of low pressure, the contact surface having a curvilinear shape, and the upwind propagating secondary shock wave (boundary between green and light blue). The high-Mach-number flow is passing through the secondary shock wave and is being decelerated that is accompanied by forming an eddy and further deformation of the contact surface shape. This configuration of two shock waves and the strongly curved contact surface is moving towards the test section.

At first, the main shock wave reaches a body placed in the test section and diffracts around it (see pictures at \(t = 1.800 \text{ ms}\) to \(t = 2.600 \text{ ms}\) in Fig. 8). Then the secondary shock wave with a high Mach number supersonic wake flow interacts with the body \((t = 2.800 \text{ ms}\) and \(t = 3.000 \text{ ms}\)). As a result, we have a high Mach number supersonic quasi-steady-state flow over a body (beginning from \(t \approx 3.640 \text{ ms}\)). Note that the flow in front of the body is not rigorously uniform. There is some cross-wise distribution of flow parameters. The duration of the quasi-steady-state carrier gas flow in the test section for the input data is approximately 14 ms.

4.2 Particle flow patterns in the shock tunnel

Flow patterns of particles of two sizes \((r_p = 0.5 \mu \text{m} \text{ and } 10 \mu \text{m})\) were investigated. The characteristic Stokes number for small and large particles was of order of 0.01 and 10, respectively. In computational simulation, the total number of particles was about \(1.2 \cdot 10^6\). This was enough for high resolution of the time-dependent particle-phase flow structure and for calculation with
high accuracy the particle concentration.

Small particles follow the gas flow with a very little lag. The behaviour of small particles depends essentially on a fine flow structure of the carrier gas. As was found in computational simulation, two ring eddies (Fig. 9) are being formed in the nozzle throat during the interaction of shock waves (compare patterns in Fig. 9 with the flow fields at the same or close instants in Fig. 5), and they influence significantly the motion of particles of radius $0.5 \mu m$ (see Figs. 10 and 12). Particles of radius $10 \mu m$ are much more inertial, and they lag considerably behind the gas flow (see Fig. 11). A wide area free of particles is observed near the walls of the nozzle (Fig. 11) and the test section (Fig. 13). Large particles after their reflection from the wall of the convergent part of the nozzle are being focussed near the axis of symmetry. In the test section, the velocity lag for small and large particles was approximately $5 m/s$ and $100 m/s$, and for particles of both sizes, the cross-wise profiles of concentration were not uniform.

5 CONCLUDING REMARKS

The fine flow structure of the carrier gas and the particles’ behaviour in a hypersonic shock tunnel was investigated, and important flow features were described in detail. In contrast to the traditional viewpoint, the gas flow in such tube is far from quasi-one-dimensional. Large-scale eddies developing in the nozzle influence significantly the motion of small particles. The behaviour of large particles differ dramatically from that of small particles. A considerable number of large particles impinge the wall of the convergent part of the nozzle, reflect from it and then the reflected particles are focussed near the axis of symmetry. The present results give the basis for understanding the particles’ behaviour in the shock tunnel, and they are to be taken into account in interpretation of experimental data.

6 ACKNOWLEDGEMENTS

This work was supported by the Russian Foundation for Basic Research (Grant 05-08-50075), and the INTAS. The authors would like to thank S.V. Panfilov for helpful discussions on various aspects of the numerical method and computational results.

REFERENCES


Figure 5: For caption see next page.
Figure 6: Time-dependent field of Mach number in the nozzle throat. Arrows show direction of motion of main discontinuities (shock waves and a contact surface).
Figure 7: Time-dependent field of Mach number and instant patterns of massless particles-markers in the nozzle. Mach number scale is given in Fig. 8.
Figure 8: Time-dependent field of Mach number in the test section.
Figure 9: Instant patterns of massless particles-markers in the nozzle throat.

Figure 10: Instant patterns of small dispersed particles in the nozzle: $r_p = 0.5 \mu m$.

Figure 11: Instant patterns of large dispersed particles in the nozzle: $r_p = 10 \mu m$. 
Anatoly A. Veryovkin and Yury M. Tsirkunov

Figure 12: Pattern of small dispersed particles in the test section. $r_p = 0.5 \mu m$, $t = 3.0$ ms.

Figure 13: Pattern of large dispersed particles in the test section. $r_p = 10.0 \mu m$, $t = 3.7$ ms.