Approach to solution of coupled heat transfer problem on the surface of hypersonic vehicle of arbitrary shape

A N Bocharov\textsuperscript{1}, V A Bityurin\textsuperscript{1}, N N Golovin\textsuperscript{2}, N M Evstigneev\textsuperscript{1}, V P Petrovskiy\textsuperscript{1}, O I Ryabkov\textsuperscript{1}, I O Teplyakov\textsuperscript{1}, A A Shustov\textsuperscript{2}, Yu S Solomonov\textsuperscript{2} and V E Fortov\textsuperscript{1}

\textsuperscript{1} Joint Institute for High Temperatures of the Russian Academy of Sciences, Izhorskaya 13 Bldg 2, Moscow 125412, Russia
\textsuperscript{2} Open Joint Stock Company “Corporation Moscow Institute of Heat Technology”, Beryozovaya Avenue 10, Moscow 127273, Russia

E-mail: bocharov@ihed.ras.ru

Abstract. In this paper, an approach to solve conjugate heat- and mass-transfer problems is considered to be applied to hypersonic vehicle surface of arbitrary shape. The approach under developing should satisfy the following demands. (i) The surface of the body of interest may have arbitrary geometrical shape. (ii) The shape of the body can change during calculation. (iii) The flight characteristics may vary in a wide range, specifically flight altitude, free-stream Mach number, angle-of-attack, etc. (iv) The approach should be realized with using the high-performance-computing (HPC) technologies. The approach is based on coupled solution of 3D unsteady hypersonic flow equations and 3D unsteady heat conductance problem for the thick wall. Iterative process is applied to account for ablation of wall material and, consequently, mass injection from the surface and changes in the surface shape. While iterations, unstructured computational grids both in the flow region and within the wall interior are adapted to the current geometry and flow conditions. The flow computations are done on HPC platform and are most time-consuming part of the whole problem, while heat conductance problem can be solved on many kinds of computers.

1. Introduction
We are considering a heat transfer problem for a hypersonic vehicle. We also assume an arbitrary shape and wide range of angle of attack and velocities. Thus a mass injection may occur and the vehicle shape may change. It is necessary to take the mass- and heat-transfer into account. So we are combining two codes that were developed for numerical simulation of the hypersonic flow over a vehicle \cite{1,2} and mass and heat transfer in the hypersonic flow for a given Stanton number \cite{3}.

2. Algorithm for the coupled problem
It was measured that the most time consuming simulation is fluid dynamics (CFD) part. The characteristic time step for the CFD part is about $1 \times 10^{-6}$ s for implicit method and $1 \times 10^{-9}$ s for explicit method; heat conduction and mass injection part (HMS) characteristic time step is...
about $1 \times 10^{-2}$ s. The CFD code is designed for the HPC platform—multi-GPU cluster under LINUX/UNIX operational system (OS). The HMS code is based on FLUENT and runs on an ordinary PC under LINUX or Windows OS. This complicates things since vehicle geometry mesh formats are different and their layout is different. The following algorithm was designed that takes all that into consideration.

Let all gas dynamics variables be stored in vector $\mathbf{U}$. We assume that all gas dynamics variables are set from the initial condition or from the previous time step. Let $\Omega \subset \mathbb{R}^3$ be the computational domain. Also, let $\Omega_1 \subset \Omega$ be the vehicle under consideration with outward boundary $\partial \Omega_1$ and $\Omega_2 \subset \Omega$ be the gas flow domain. It is obvious that $\Omega = \Omega_1 \cup \Omega_2 : \Omega_1 \cap \Omega_2 = \partial \Omega_1$. Gas variables on $\partial \Omega_1$ are indicated as $\mathbf{U}_W$. We also set number of CFD time steps for a single HMS time step as $M$ and make a bijective mapping between different methods for mesh storage, thus we know unique number of element and its vertexes on CFD part and HMS part. The algorithm is as follows.

(i) Perform $M$ CFD steps in $\Omega_2$, using $\Omega$ and $\mathbf{U}$ from previous time step or initial conditions.
(ii) Obtain the $\mathbf{U}_W$ and use it to calculate $St_W$ number.
(iii) Transfer data for all $St_W$ and $P_W$ in $\partial \Omega_1$ to HMS part.
(iv) Perform calculation of heat transfer problem $\Omega_1$ and calculate mass injection.
(v) Perform morphing of the $\Omega$ due to mass injection. Transfer morphed mesh data and $T_W \in \mathbf{U}_W$ to CFD part.
(vi) Run preprocessor part for the new mesh of $\Omega$ in CFD part and goto step 1.

Here $St_W$ stands for Stanton number on the wall and $P_W$, $T_W$ for pressure and temperature on the wall. There are some technical difficulties that are related to bijection mapping of meshes for commercial Ansys products and open source mesh generators. There is only one format type is shared (NASTRAN bulk mesh format) and it is not flexible enough. Hence this paper presents only results for one sided data transfer, namely, from CFD to HMS.

3. Code modifications
3.1. CFD code modification

As in [1, 2], let the gas flow be governed by continuous mechanics. Thus, the governing equations are as follows:

$$
(\rho)_t + \nabla \cdot (\rho \mathbf{u}) = 0; \quad (1)
$$

$$
(\rho \mathbf{u})_t + \nabla \cdot (\mathbf{u} \otimes (\rho \mathbf{u})) + \nabla p = \nabla \cdot \Pi; \quad (2)
$$

$$
(E)_t + \nabla \cdot (\mathbf{u} (E + p)) = \nabla \cdot \mathbf{G} + \nabla (\Pi \cdot \mathbf{u}). \quad (3)
$$

Here $t$—time; $()_t$—time derivative; $E$—full gas energy; $e$—internal gas energy; $T$—gas temperature; $\Pi$—stress tensor; $p$—pressure; $\mathbf{G}$—heat flux; $\mathbf{u}$—velocity; $\rho$—density; $\otimes$—tensor product. The viscosity is found, using Sutherland’s law [1]. Consider a closed domain $\Omega_2$ with the initial-boundary problem posed. The decomposition of the domain is $\Omega_2 = \bigcup_{i=1}^{N} W_i$, with $W_i$ being a convex simplex. Thus geometric flexibility of the boundary problem description is achieved. By applying the discretization, the equations (1)–(3) are transformed to conservative form.

In order to conduct a coupled simulation several modifications were made into the code, described in [1, 2] in accordance with the presented algorithm.
3.2. Mass transfer code modification

The simulation method is based on numerical solution of three-dimensional heat equation (4) within the thermo protection system (TPS), calculation of heat- and mass flux on the flow-wall interface \( \partial \Omega_1 \), estimation of the surface recession rate, and re-shaping of both surface and interior:

\[
\rho C_p \frac{\partial T}{\partial t} = \text{div}(\lambda \text{grad}(T)),
\]

where \( \rho \)—material density of body and TPS layer, \( C_p \)—heat capacity, \( \lambda \)—heat conduction, \( T \)—temperature, \( t \)—time. The ablating surface is considered as moving, the normal velocity of surface element \( v_n \) is defined with mass flux \( \dot{m} \), i.e. \( v_n = \dot{m}/\rho \). The surface under TPS layer is considered as nondestructive and steady.

We use Ansys FLUENT for solving the heat problem, which is capable of working on unstructured grids using finite volume method. The link between TPS characteristics and flow parameters (surface model) is realized through a solution of the set of equations for mass- and heat-rate at the surface:

\[
q_w = f(T_W, P_W, q_0, \ldots), \quad \dot{m}_w = f(T_W, P_W, q_0, \ldots).
\]

Here, \( q_0 \) is the heat flux from the flow (it can include the radiation flux as well). In the paper, well known model [4] was applied as an example.

Shapard’s method [5] was used to deform grid in both calculational areas \( \Omega_1 \) and \( \Omega_2 \): in the air and in the TPS layer. This method connects moving of grid nodes on the border with moving nodes inside the computational domain \( \Omega_1 \) using the following equation:

\[
\delta(x) = \frac{\sum_{i=1}^{n} w_i(x) \delta_{G_i}}{\sum_{i=1}^{n} w_i(x)}, \quad w_i(x) = \frac{1}{\|x - \mathbf{x}_i\|^p}.
\]

Here \( x \)—internal node’s coordinates, \( \delta \)—movement of the internal node, \( \delta_{G_i} \)—movement of the boundary node, \( n \)—number of boundary nodes, \( i \)—index of boundary nodes, \( w_i \)—weight function, \( p \)—positive real number. Usually \( p \in (1, 2] \), we used \( p = 2 \).

4. Some benchmarks and results

4.1. Gas dynamics

Two configurations were considered. Prototypes of configurations discussed in the paper were considered, for example, in [6, 7]. In the first case a sphere-cone was meshed with geometry, presented on figure 1. A sphere-cone with a rear-end half-sphere is considered in the second configuration (figures 2, 3, 4). These configurations are called case 1 and case 2, respectively. Case 2 is the main emphasis of the paper, since case 1 was considered in [2, 3]. The free-stream parameters correspond to the altitude 10 km, flow Mach number 16.7 and zero angle of attack.

CFD part was conducted on 5 GPUs on a miniclaster using unstructured hybrid meshes. We considered meshes with about \( 5.5 \times 10^6 \) elements, second order time and space scheme, see [2] for more details on numerical algorithm. Maximum element length in normal direction to the wall of the body surface is about \( 1 \times 10^{-6} \) m, thus a boundary layer is well represented. Only flow near body is considered in this paper, so the far field discretization was rough. Inner volume for HMS calculations consisted of 228400 prisms.

Density, pressure counters, pressure gradient on the body surface and velocities in the boundary layer near the surface for case 2 are shown on figures 5, 6, 7 and 8. It can be seen that local pressure maxima are located on the rear sphere and on the tip. It is also confirmed by the density contours. It can be also noticed that the boundary layer is separated on the cone
generatrix along the normal direction to the rear sphere. It can be confirmed by the density contours, velocities and pressure gradients in the boundary layer near the sphere base as well, see figures 5, 7, 8. Boundary layer instability can be observed. Turbulence models must be included
into simulation model for the future, especially if one is interested in the far field of the flow. The most instable region is near rear surface in the normal direction along the cone generatrix and near rear sphere, see pressure gradient (figure 8). It should be pointed out that these instabilities are pure three dimensional and cannot be correctly represented in 2D simulations.

However, as it was stated in [1], there are more important parameters to be considered, namely heat flux and shear stress on the body surface. Heat flux and shear stress on the surface of the body were also considered. Skin distributions are shown on figure 9 and 10. Heat flux maxima can be found on rear sphere and on spherical tip of the cone. The absolute maximum of heat flux on the cone tip, whereas local maximum on rear sphere is a bit less than on the cone, and it is symmetrical. Shear stress reaches maximum on tip as well, with maximum modulus of 0.02 relative to inflow flux. Rear sphere suffer maximum surface stress of 0.005 relative to inflow flux, with their maxima located symmetrically, figure 12. Heat flux data is used to calculate Stanton number which together with pressure distribution on the body surface are transfered to HMS module of the program.

Case 1 is also calculated and analyzed. Surface stress and heat flux on the body for case 1 is less than for case 2, obviously. However maximum is the same on the tip of the cone, see figures 13, 14, 15, 16. For the shear stress one can observe maximum of on the tip and local extrema at the rear sphere, The absolute maximum of heat flux on the cone tip is just like in
Figure 11. Surface stress near the cone tip sphere (relative to $\rho_0 u_0^2/2$), log scale, case 2.

Figure 12. Surface stress absolute values (relative to $\rho_0 u_0^2/2$), normal scale, case 2.

Figure 13. Density body on surface and contours around it (relative to $\rho_0$), log scale, case 1.

Figure 14. Velocity vectors in the boundary layer near surface (relative to free stream velocity $u_0$), log scale, case 1.

Figure 15. Surface heat flux (relative to inflow velocity $\rho_0 u_0^3/2$), log scale, case 1.

Figure 16. Surface stress tensor projection (relative to $\rho_0 u_0^2/2$), case 1.

case 2, whereas local maximum on rear sphere way smaller and it is also symmetrical. The boundary layer only separates near the rear sphere and this local phenomenon is minor.
We also estimated Drag coefficient for both cases. The drag coefficient is defined as:

\[ C_x = \frac{2F_d}{\rho_0(u_0)^2A}, \]

(7)

where \( F_d \) is a drag force in the direction of the flow (in \( X \) direction in our case); \( \rho_0 \) and \( u_0 \) are free stream parameters of density and velocity; \( A = W^{2/3} \)—reference area and \( W \) is a volume of the considered body.

It was found, that \( C_x = 0.05325 \) for case 1 and \( C_x = 0.4093 \) for case 2. Rear sphere greatly (7.7 times) increase drag in the flight direction.

4.2. Heat conduction and mass transfer

With the data, obtained from the gas dynamics part of the platform, heat conduction and ablation of the surface were estimated. Deformation of the ablator layer in critical point and at hemispherical elements due to the ablation are shown in the figure 17 and 18. The dependence of the temperature at the critical point on time is shown in figure 19. The steady state in frontal part is reached in about 0.5 s. Changing the thickness of the layer on time is shown in figure 20. We can see this dependence is practically linear and we can expect destruction of the ablator layer in 5 s under these condition.
Figure 18. Deformation of the ablator layer and the temperature at the hemispherical element: 1—0.1 s, 2—0.2 s, 3—0.5 s, 4—1 s, 5—1.5 s, 6—2 s, 7—3 s, 8—4 s, 9—4.6 s, case 2.

Figure 19. Temperature at the leading stagnation point (1) and at the sphere (2) versus time, case 2.

Figure 20. Thickness of the ablator layer at the leading stagnation point versus time, case 1.
5. Conclusion
In this paper, we are presenting the first results of a coupled problem solution for gas dynamics around the hypersonic body and ablation of its surface. We are using two numerical methods that where previously described in [1–3]. We show one way coupling from gas dynamics part of the platform to the heat conduction and mass injection part. A hypersonic overflow problem for sphere-cone body with a rear sphere is presented. Spatial distributions of pressure, heat fluxes and shear stress tensor components on the surface were cross-correlated. Stanton number and pressure on the wall interface were used to calculate heat conduction and surface recession rate. The distribution of the temperature along the surface was obtained. The dependence of the surface recession rate on time was close to linear.

References
[1] Bocharov A N, Balakirev B A, Bityurin V A, Gryaznov V K, Golovin N N, Iosilevskiy I L, Evstigneev N M, Medin S A, Naumov N D, Petrovskiy V P, Ryabkov O I, Solomonov Y S, Tatarinov A V, Teplyakov I O, Tikhonov A A and Fortov V E 2015 J. Phys.: Conf. Ser. 653 012070
[2] Bocharov A N, Evstigneev N M and Ryabkov O I 2015 J. Phys.: Conf. Ser. 653 012119
[3] Bocharov A N, Golovin N N, Petrovskiy V P and Teplyakov I O 2015 J. Phys.: Conf. Ser. 653 012118
[4] Scala S M and Gilbert L M 1965 AIAA Pap. 65 12
[5] Kopysov S P, Kuzmin I M and Tonkov L E 2013 Numerical Methods and Programming 14 269–278
[6] Ionov S S, Mishina E A and Kalugin W T 1992 Informtehknika 191 183–190
[7] Kalugin W T 2004 Aero- and Gas Dynamics for Control Systems of Aerospace Vehicles (Moscow: Bauman Moscow State Technical University)