On the calculation of dynamic and heat loads on a three-dimensional body in a hypersonic flow

A N Bocharov, V A Bityurin, N M Evstigneev, V E Fortov, N N Golovin, V P Petrovskiy, O I Ryabkov, I O Teplyakov, A A Shustov and Yu S Solomonov

1 Joint Institute for High Temperatures of the Russian Academy of Sciences, Izhorskaya 13 Bldg 2, Moscow 125412, Russia
2 Open Joint Stock Company “Corporation Moscow Institute of Heat Technology”, Beryozovaya Avenue 10, Moscow 127273, Russia
3 Institute for Systems Analysis of the Russian Academy of Sciences, 60-letiya Oktyabrya 9, Moscow 117312, Russia
E-mail: bocharov@ihed.ras.ru

Abstract. We consider a three-dimensional body in a hypersonic flow at zero angle of attack. Our aim is to estimate heat and aerodynamic loads on specific body elements. We are considering a previously developed code to solve coupled heat- and mass-transfer problem. The change of the surface shape is taken into account by formation of the iterative process for the wall material ablation. The solution is conducted on the multi-graphics-processing-unit (multi-GPU) cluster. Five Mach number points are considered, namely for M = 20–28. For each point we estimate body shape after surface ablation, heat loads on the surface and aerodynamic loads on the whole body and its elements. The latter is done using Gauss-type quadrature on the surface of the body. The comparison of the results for different Mach numbers is performed. We also estimate the efficiency of the Navier–Stokes code on multi-GPU and central processing unit architecture for the coupled heat and mass transfer problem.

1. Introduction
The estimation of heat and aerodynamic loads on specific body elements for a hypersonic vehicle is an essential problem. It arises during all stages of vehicle development starting from initial optimization and selection of parameters up to the testing of local units and constructions. The goal of the numerical simulation is the determination of heat fluxes and loads on specific parts of a vehicle. We are using previously developed code [1–3] to solve the coupled heat transfer problem with ablation. This problem is to be used as a benchmark and debug problem for further software development.

2. Governing equations
As in [1–3], let the gas flow be governed by continuous mechanics. Thus, the governing equations are

\[
\begin{align*}
(\rho)_t + \nabla \cdot (\rho \mathbf{u}) &= 0, \\
(\rho \mathbf{u})_t + \nabla \cdot (\mathbf{u} \otimes (\rho \mathbf{u})) + \nabla p &= \nabla \cdot \Pi, \\
(E)_t + \nabla \cdot (\mathbf{u}(E + p)) &= \nabla \cdot \mathbf{G} + \nabla (\Pi \cdot \mathbf{u}).
\end{align*}
\] (1)
The heat equations are solved for the interior of a vehicle for thermal protection system (TPS) as

$$\rho_m C_p (T_m)_t = \nabla \cdot (\lambda \nabla T_m).$$

(2)

Here, \(t\) is time; \(\cdot\) is a time derivative; \(\nabla\) is a dot product; \(E\) is a full gas energy; \(T\) is gas temperature; \(\Pi\) is a viscous stress tensor; \(p\) is a pressure; \(G\) is a heat stress; \(u\) is a velocity vector; \(\rho\) is a gas density; \(\otimes\) designates a tensor product; \(\rho_m\) is the density of body material at TPS layer, \(C_p\) is a heat capacity of the material, \(\lambda\) is a heat conduction in the material, \(T_m\) is a material temperature. The viscosity is found, using Sutherland’s law [1].

We designate the calculation domain \(\Omega\) with the initial-boundary value problem posed. The domain is decomposed in \(\Omega = \bigcup_{i=1}^{N} \Omega_i\) with \(\Omega_i\) being a convex element. Geometric flexibility of discretization is achieved by applying unstructured grid formulation. Equations (1) and (2) are used in conservative form on each \(\Omega_i\).

The code solves two problems with iterative coupling: the solution of (1) for external flow problem on \(\Omega_1\) and (2) for internal thermal protection system (TPS) layer on \(\Omega_2\). The flow-wall interface \(\partial \Omega_{12} = \Omega_1 \cap \Omega_2\) is the exchange boundary between two problems. Calculation of heat- and mass flux, estimation of the surface recession rate and re-shaping of surface is performed on \(\partial \Omega_{12}\). The latter provides information for reshape of \(W_i\). The ablating surface \(\partial \Omega_{12}\) is considered as moving, the mass flux (\(m_i\)) is defined on this surface, thus a normal velocity of surface element \(v_n = (m_i)/\rho_m\) is defined. The surface under TPS layer is assumed to be nondestructive and steady. The link between TPS characteristics and flow parameters is realized through a solution of the set of equations

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

(3)

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 computational domains \(\Omega_1\) and \(\Omega_2\): in the air and in the TPS layer. More details are found in [1].

3. Problem formulation

We are considering the problem of a body placed under the hypersonic flow conditions. At the moment we are not interested on the history of the problem as well as of the body trajectory. The main goal is to find thermal, ablating and dynamic responses in the body in case of a sudden movement in hypersonic flow. A sphere-cone with a rear-end half-sphere is considered (figures 1 and 2). We define external boundaries in \(\Omega\), namely \(\partial \Omega_{in}\) is the whole external boundary of \(\Omega\) (designated “\(l\)” in figure 1) except for the boundary with \(x = x_{\max} = \text{const}\) which is designated as \(\partial \Omega_{out}\) (designated “\(\beta\)” in figure 1). Supersonic inflow boundary conditions are posed on \(\partial \Omega_{in}\) boundary with outflow boundary posed on \(\partial \Omega_{out}\). The characteristic boundary condition for \(\partial \Omega_{out}\) is used, since the problem includes possible regions with subsonic outflow. The “sponge zone” was used in order to avoid any influence of the subsonic boundary regions on the internal flow filed. Such method was used in [6] for the fundamental research of the laminar-turbulent transition problem where perfect outflow matching was demonstrated with infinitesimal influence on the internal flow regions. Zero angle of attack is considered and gas far field parameters correspond to the 30 km altitude. Robin boundary conditions are set for the internal boundary of the TPS layer.

Calculation of computational fluid dynamics (CFD) part was conducted on 5 graphics-processing units (GPUs) on a miniclaster. We considered meshes with about \(10.2 \times 10^6\) elements (most of them are hexahedra, see figure 2), explicit second order time and space scheme, see [1] for more details on numerical algorithm. Minimum element length in normal direction to the wall of the body surface is about \(1.3 \times 10^{-6}\) m, thus a boundary layer is well represented. Inner volume for heat problem calculations consisted of about 450000 tetrahedra.
Figure 1. Surface mesh for sphere-cone geometry with external boundaries surfaces: 1—inflow boundary; 2—outflow boundary.

Figure 2. Zoom view of the boundary layer mesh (CFD part) near rear-end half-sphere.

The problem is to find dynamic loads on sphere-cone itself and a rear-end half-sphere separately under the heat loading of the vehicle with ablation.

The forces vector $\mathbf{F}$ on the body part $\partial\Omega_B$ is calculated as follows:

$$
\mathbf{F} = \int_{\partial\Omega_B} p\mathbf{n}dS + \int_{\partial\Omega_B} (\mathbf{Π}, \mathbf{n})dS,
$$

where first part corresponds to the inviscid forces and the second part corresponds to viscous
forces. Here, $\mathbf{n}$ is an outward normal vector and tensor-vector dot product is designated as $\langle \Pi, \mathbf{n} \rangle$. In order to perform numerical calculation of (4) we use Gauss quadrature:

$$
\mathbf{F} \simeq \sum_{j: \partial \Omega \subset \partial \Omega_B} \sum_{k=1}^G [p_k \mathbf{n}_j + \langle \Pi_k, \mathbf{n}_j \rangle] \omega_k \partial \Omega_j,
$$

where $\omega_k$ are Gauss quadrature wights and $G$ is defined depending on Gauss quadrature rule. Values of $p_k$ and $\Pi_k$ are calculated at Gauss quadrature points inside $\partial \Omega_j$. For second order accuracy Gauss–Lobatto points and unit wights are used that for non-canonical 2D elements (arbitrary triangles and quads) can be found in [5]. This procedure removes all errors related to numerical integration since Gauss quadrature is exact for given degree of polynomials.

4. Results

Dimensionless pressure counters, pressure gradients on the body surface, shear stress projection surface vectors and surface inflow heat fluxes are shown in figures 3–6 for Mach number 20. Mach number 28 distributions are similar and are omitted for brevity. All distributions are normalized to the free flow gas parameters. It can be seen that local pressure maxima are located on the rear sphere and on the tip. The boundary layer is separated on the cone generatrix along the normal direction to the rear half-sphere. Single main separation zone is visible with multiple separation zones located near the main vortex. It is confirmed by the analysis of pressure gradients, see figure 5. 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. These instabilities are pure three dimensional and cannot be correctly represented in 2D simulations.

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 are shown in figures 4 and 6. Heat flux maxima can be found on rear half-sphere and on spherical tip of the cone. With the increase of Mach number the maxima are increasing since far field parameters of the flow are increasing. The absolute maximum of heat flux is located on the rear half-sphere. Shear stress reaches maximum on the rear half-sphere as well. Heat flux data is used to calculate Stanton number which together with pressure distribution on the body surface are transfered to the module of the program that numerically solves (2) and (3).

Drag and Lift coefficients are defined as

$$
C_l = \frac{2F_l}{\rho_0(u_0)^2 A_l},
$$

where $F_l$ is the force in the $l$-th direction of the flow, $A_l$ is the reference area in $l$-th direction as the projection of the surface area on the plane $l = 0$, $l = x$ for $C_x$ and $l = y$ for $C_y$ with $\rho_0$ and $u_0$ being free stream parameters of density and velocity for prescribed Mach number.
Figure 3. Surface pressure distribution (relative to $\rho_0 u_0^2/2$), $M = 20$.

Figure 4. Absolute values of the surface shear stress tensor projection (relative to $\rho_0 u_0^2/2$), $M = 20$.

Results are presented in table 1. One can see the slight fall of global $C_x$ value with the increase Mach number. The value of local $C_x$ value increases with the increase of Mach number. Both $C_y$ for the whole vehicle and the rear half-sphere almost constant. Slight deviation from
Figure 5. Surface pressure gradient distribution (relative to $\rho_0 u_0^2 / D$, where $D$ is the cone maximum diameter), $M = 20$.

Figure 6. Surface heat flux (only inflow flux is shown), log scale, (relative to $\rho_0 u_0^3 / 2$), $M = 20$.

linear dependence can be due to the slight mesh deficiency for large Mach number. Another hypothesis is the shape of boundary layer separation which is more complex for $M = 28$. It is to be investigated in the future. Heat conduction and ablation of the surface were estimated
Figure 7. The temperature (in K) of the rear half-sphere, $M = 20$, $t = 0.1$ s.

during the calculation with the data, that had being obtained from the gas dynamics part of the code. Temperature distribution in the ablator layer in the rear half-sphere is shown in figures 7 and 8 for time periods 0.1 and 0.7 s. The steady state in is reached in about 0.64 s.

5. Computational efficiency

We must stress that the most time consuming operation is the preparation of the sufficiently detailed mesh which is essential in hypersonic flow calculations. Salome platform was used with custom Python scripts. Wall time consumption on the development of scripts and preparation of satisfactory mesh is about 1.2 man month. Automation of the mesh generation and its smoothness is a crucial part and will be the pinnacle of attention in our closest research.

Computational efficiency is measured in execution time. The comparison is essential for the balance between gas dynamics part of the code and heat- and mass-transfer central-processing-unit (CPU) part. The mini-cluster is used as a hardware that contained five GTX TITAN BLACK GPUs with 6GB memory for each GPU assembled on a single chasis. The CPU is a single 8 core Intel XEON E5-2640V2 Ivy Bridge-EP. The heat and mass transfer part is executed using Ansys Fluent on CPU. The mesh of 70 000 elements was used [3] and the application of CPU part of the code took only 21.5% of time during the whole timestep. With the increase of mesh up to 450 000 elements, the balance became drastically different. It takes about 7 hours of wall time to calculate gas dynamics step from initial conditions to the stationary distribution of
6. Conclusion

In this paper we are presenting the results of a coupled problem solution for gas dynamics around the hypersonic vehicle with ablation of its surface. We are using two numerical methods that where previously described in [1–3]. The results of the calculations are aimed to estimate forces acting on the whole body and its separate parts while the problem is considered in the framework of the coupled solution. A hypersonic problem for sphere-cone body with a rear half-sphere is tested as a target for our research. Spatial distributions of pressure, heat fluxes and shear stress tensor components on the surface are cross-correlated for Mach numbers varying from 20 to 28 with step 2. We calculated values of $C_x$, $C_y$ for the whole vehicle and for its part. The values of $C_x$ and $C_y$ are varying slightly. Extensive research allowed us to outline main priorities for the future work, namely, development of stand alone automated mesh generator and substitution of Ansys FLUENT part.

Figure 8. The temperature (in K) of the rear half-sphere, $M = 20$, $t = 0.7$ s.

Flow parameters. But it takes 460.63 min (7.6 h) to perform single step for FLUENT calculations on CPU resulting in 109% ratio of FLUENT part wall time to gas dynamics wall time. This situation can be improved on multi-CPU architecture but it requires installation of multi-licensed Ansys FLUENT. So it is our next priority (along with the mesh generator) to substitute Ansys to multi-GPU-oriented self-developed code.
Acknowledgments
Development and testing of physical and numerical models, as well as their implementation in commercial code were performed in the Joint Institute for High Temperatures RAS under support by the Russian Science Foundation, grant No. 14-50-00124. CFD part of computations was performed on the high performance computing mini-cluster owned by one of the authors.

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 Yu 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, Bityurin V A, Golovin N N, Evstigneev N M, Fortov V E, Petrovskiy V P, Ryabkov O I, Solomonov Yu S, Shustov A A and Teplyakov I O 2016 J. Phys.: Conf. Ser. 774 012157
[4] Scala S M and Gilbert L M 1965 AIAA Pap. 65 12
[5] Berens H, Schmid H and Xu Y 1995 SIAM J. Math. Anal. 26 468–87
[6] Evstigneev N M and Magnitskii N A 2014 Proc. ISA RAS 63 41–52