Numerical simulation as an important tool in developing novel hypersonic technologies

A N Bocharov¹, B A Balakirev², V A Bityurin¹, V K Gryaznov³, N N Golovin², I L Iosilevskiy⁴, N M Evtigneev³, S A Medin¹, N D Naumov¹, V P Petrovskiy¹, O I Ryabkov¹, Yu S Solomonov², A V Tatarinov¹, I O Teplyakov¹, A A Tikhonov² and V E Fortov¹

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 of Problems of Chemical Physics of the Russian Academy of Sciences, Academician Semenov Avenue 1, Chernogolovka, Moscow Region 142432, Russia
⁴ Moscow Institute of Physics and Technology, Institutskiy Pereulok 9, Dolgoprudny, Moscow Region 141700, Russia
⁵ Institute for Systems Analysis of the Russian Academy of Sciences, 60-letiya Oktyabrya 9, Moscow 117312, Russia
E-mail: bocharov@ihed.ras.ru

Abstract. Development of novel hypersonic technologies necessarily requires the development of methods for analyzing a motion of hypervelocity vehicles. This paper could be considered as the initial stage in developing of complex computational model for studying flows around hypervelocity vehicles of arbitrary shape. Essential part of the model is a solution to three-dimensional transport equations for mass, momentum and energy for the medium in the state of both LTE (local thermodynamic equilibrium) and non-LTE. One of the primary requirements to the developed model is the realization on the modern heterogeneous computer systems including both CPU and GPU. The paper presents the first results on numerical simulation of hypersonic flow. The first problem considered is three-dimensional flow around curved body under angle of attack. The performance of heterogeneous 4-GPU computer system is tested. The second problem highlights the capabilities of the developed model to study heat and mass transfer problems. Namely, interior heat problem is considered which takes into account ablation of thermal protection system and variation of the surface shape of the vehicle.

Development of novel hypersonic technologies requires inevitably the development of methods for analyzing a motion of hypervelocity vehicles. Complexity of such analysis is first of all owing to a large variety of physical processes related with hypervelocity flight and their mutual influence.

Obviously important are the processes determining aerothermodynamics of hypervelocity vehicle including plasma formation and thermo-mechanical destruction of the vehicle surface due to its interaction with high speed flow. In addition, a structure of ionized flow in a vehicle wake is also of great interest. Comprehensive analysis of present-day vehicles as well as the development of perspective ones is hardly possible without new analysis technologies, in particular, computational technologies.
Numerical simulation is a powerful tool to study many of critically important processes connected with the motion of hypervelocity body in planetary atmosphere. Demands to computational tools are due to not only complex physics of hypervelocity motion, but also due to complex geometry of vehicle (variation of surface shape, rotation, precession, and other). Obviously, computational technologies under developing should provide a substantial analysis for reasonable time.

This paper could be considered as the next stage in developing of complex computational model for studying flows around hypervelocity vehicles of arbitrary shape. Essential part of the model is a solution to three-dimensional transport equations for mass, momentum and energy for the medium in the state of both LTE (local thermodynamic equilibrium) and non-LTE. One of the primary requirements to the developed model is the realization on the modern heterogeneous computer systems including both CPU and GPU. One more important requirement relates to a computational grid topology. The developed model is capable of working with any known topologies of elements of unstructured grid (prism, pyramid and tetrahedron).

Calculation of thermodynamic properties of flow can be done in several ways depending on flow conditions and analysis purposes. Thermo-chemical non-equilibrium airflow model as well as LTE airflow model (as development of existing code PlasmAero) was implemented. More sophisticated model is planned to be implemented which allows one to calculate thermodynamic properties of medium with arbitrary chemical composition including multi-component plasma under LTE approach. The model is being developed as an extension to the existing code SAHA developed for calculations of thermodynamic properties of non-ideal plasmas with complex composition. Thermal state of vehicle interior can be analyzed from solution to the conjugate heat and mass transfer problem. Heat problem is also solved on unstructured grid, and variation of the vehicle’s surface due to interaction with high temperature flow is taken into account.

The paper presents some results on developing a HPC (high performance computing) tool for numerical simulation of hypersonic flows. The first problem considered is three-dimensional flow around curved body under angle of attack. The performance of heterogeneous 4-GPU computer system is tested. The second problem highlights the capabilities of the developed model to study heat and mass transfer problems. Namely, interior heat problem is considered which takes into account ablation of thermal protection system (TPS) and variation of the surface shape of the vehicle.

The flow solver developed realizes solution to 3D Navier–Stokes equations on both CPU clusters and multiple GPU clusters. Arbitrary cell-topology grid can be used constructed with any unstructured grid generator. Cell-shape independent technologies are used to construct both inviscid and viscous numerical fluxes. About twenty inviscid-flux methods were implemented and tested for hypersonic flow calculations including Godunov-type methods, flux-splitting methods and other. Both explicit and implicit time integration schemes can be applied for flow computations.

To demonstrate the current solver features the Mach 17 flow around a sphere-edged cone at non-zero angle of attack has been calculated on 4-GPU cluster (GeFORCE GTX TITAN BLACK). Figure 1 presents the computational grid on the body surface. Distribution of pressure over the surface is shown in figures 2 and 3 for two angles of attack.

Efficiency of the solver was tested for the grids from 1M to 4.25M number of grid cells. In the latter case, Efficiency of the solver was measured to be 76% of peak efficiency for implicit time integration and is 94% for explicit that. (It should be noticed that in the case of 4.25M grid, computation time for implicit scheme is about order of magnitude less than for explicit one) For 4.25M grid, performance of the solver is nearly 4.1 TFlops, which is close to performance of 32-CPU cluster with Intel Xeon E5 4-core unit.

Another class of hypersonic flow problems relates to accurate assessment of thermal state of the vehicle’s interior. This is important from many points of view: ability of materials to
support heat loads, structural mechanics problems, packaging of equipment and other. The first problem of the list closely relates with conjugate heat transfer problem aggravated with radiation from the shock layer, material ablation, and, as consequence, changes in the vehicle’s shape. To meet design requirements, aerospace designers should have computational tools capable of accounting for complicated heat transfer on the vehicle surface, re-shaping vehicle surface, rebuilding the whole computational geometry/grid. This paper shortly reviews the numerical technique developed to simulate thermal state within a vehicle’s interior taking into account arbitrary and varying shape of outer surface, complicated heat and mass transfer on it, changes in the bulk material sizes and properties.

The technique is based on approach in which separate solvers are used for hypersonic flow (like that reported above) and for the interior of the vehicle (wall solver). The latter is realized as an extension to widely used program Ansys Fluent. The developed technique provides appropriate data exchange between flow and wall solvers, solves the non-linear heat transfer problem at the surface using data obtained from the flow simulations. Next, the unstructured computational grid is re-built taking into account the surface recession, and the heat problem is solved for a whole bulk of the interior. All these steps are coupled through iterations at each time step.

As an example 3D unsteady numerical simulation of thermal state within a sphere-cone shaped body like that mentioned above was carried out. The hypersonic flow around the body was preliminarily calculated with the JIHT RAS PlasmAero code. Unstructured tetrahedral mesh with about of 100,000 grid cells was used for this test. Some results are presented in figure 4 and 5.

The developed technique seems to be a promising tool in solving many problems related with external and internal hypersonic flows and real craft design. The future work assumes implementation to the HPC code of several physics models concerning the plasma formation,
both LTE and non-LTE; implementation of turbulence models; implementation of real material ablation models; development of appropriate flow-solver/wall-solver interface and data exchange.

Figure 4. Temperature over the body surface for several time instances.

Figure 5. TPS recession rate over the body surface for several time instances.