CFD PACKAGE "FLOW VISION" FOR SIMULATIONS AND IMITATION MODELING OF HYPERSONIC VEHICLES

Son E.E.*, Degtyar V.G., Aksenov A.A.*, Zhluktov S.V.*, Hlybov V.I.**, Kalashnikov S.T. ***, Son K.E.***
***Moscow Institute of Physics and Technology (State University),**
****Academician V.P. Makeyev Space Rocket Centre**
***** Federal State Budgetary Scientific Institute "South Ural Scientific Centre"**

Keywords: *hypersonic, vehicles, cfd modeling*

Abstract

CFD package "Flow Vision" has been developed since late 1960th and now is one of the mostly used in Russia and elsewhere, In the present report we describe the applications of the "Flow Vision" for hypersonic vehicles (HV) design, Imitation modeling and optimizations. We describe and use it for external aerodynamics, simulations of inlets, fuselages, wings, combustion chambers and nozzles. We simulated HV for different models like "Hexafly", X01-MIPT and other vehicles at the laboratory of Hypersonic and Plasma Technologies of Moscow Institute of Physics and Technology.

1 CFD "Flow Vision" description

The software package Flow Vision [5,6] is designed for numerical modeling of 3D laminar and turbulent, stationary and non-stationary flows of liquid and gas. The software package is based on the finite volume method, high-precision difference schemes, effective numerical methods and reliable mathematical models of physical processes. Numerous models make it possible to calculate complex flows accompanied by swirling flows, the motion of free/contact surfaces, shock waves, conjugate heat transfer, combustion, etc. Flow Vision is a Cartesian, locally adaptive grid. Local dynamic adaptation of the initial grid is performed in accordance with user-defined criteria. The initial grid consists of rectangular cells. Near the boundary of the computational domain, a boolean subtraction of uncountable volumes from rectangular cells occurs, as a result of which polyhedral cells of arbitrary shape are formed. There is no simplification of frontier cells. Grid generation is fully automated.

The software package Flow Vision runs on computers that have a mixed architecture, combining inter-node MPI-parallelization with stream parallelization in a node, as on a computer with shared memory. Using mixed parallelization allows you to achieve high-quality scaling of the software complex when working on a large number of processors.

Flow Vision allows simulate flows near moving and deformable bodies in conjunction with the finite element software Abaqus, Fidesys.

Flow Vision allows solve the problem of optimizing the shape of objects in conjunction with the optimization software complex IOSO by visualization and data processing tools implemented in Flow Vision

The software package Flow Vision is a tool for modeling the flow of liquid and /or gas in technical devices or natural conditions with subsequent analysis of the results of calculations. Flow Vision is focused on flow simulation in conditions characterized by the following features: a complicated form of the boundaries of the flow region, which allows one to simulate the functioning of real technical devices, flow turbulence, arbitrary flow velocities (from incompressible fluid flow to supersonic flows), convective and radiant heat exchange, thermal conductivity and conjugate heat exchange, non-Newtonian rheology, FSI (fluid-structure interactions), the presence of a free liquid/gas or liquid/liquid surface, diffusion and chemical reactions between components and combustion.

1.1 Brief technical description

The software package FlowVision is equipped with a variety of means of preparing the draft calculation, which include:

- facilities of importing geometric objects that make up the boundaries of the calculation area, from programs of geometric modeling (CAD-systems),
- facilities for creating geometric objects of elementary form,
- facilities of automatic generation of a computational grid, taking into account the features of the shape of objects in the calculation area,
- facilities for specifying the boundary and initial conditions of flow and heat transfer in the computational domain.

The calculation block of the software provides a numerical solution of a set of equations describing the motion of a liquid and/or gas in the calculation area, including:

- conservation equations of mass, momentum and energy,
- equations of state
- equations of turbulence models
- equations of special models (free gas / liquid surface, combustion, radiative heat transfer, flow in a thin gap, etc.

The solution of the system of equations is carried out on a Cartesian grid, which is automatically locally is crushed. Grinding the grid can be concentrated in the region of high gradients or complex geometric shapes. The grid cells intersecting with the boundaries of the computational domain and the calculated subregions are cut off by the boundary surfaces.

In the FlowVision, the technology of flow computation with bodies moving in the computational domain relatively fixed bodies. The FlowVision design unit can function on computers of different configurations: personal computers, in a network of computers or on a cluster. Parallelization of computational procedures is performed automatically and ensures efficient use of multiprocessor technology.

FlowVision includes a variety of tools for analyzing simulation results:

- controls and express analysis of intermediate calculation results,
- facilities of accumulation of integral characteristics of various processes,
- analysis and visualization of calculation results.

2 Method for calculating the Navier-Stokes equations in the Flow Vision

2.1 Numerical Schemes

To solve 3D nonstationary Navier-Stokes equations, a large number of different methods have been developed that can be divided into two large groups: methods using pressure-velocity variables and methods using variable density-velocity. The methods using pressure-velocity variables are well suited for the modeling of incompressible and weakly compressible fluid and gas flows. The methods using variable density-velocity are intended, mainly, for solving problems of super- and hypersonic aerodynamics. Simulation of incompressible and weakly compressible flows requires a significant complication of the initial algorithm: the introduction of preconditioners, double integration over time, and so on.

A real supersonic gas flow near an arbitrary object is characterized by the presence of zones in which the gas velocity is small. These are the boundary layers, the zones of inhibition and recirculation of the flow. The flow in these zones is practically incompressible. Therefore, the development of methods using pressure-velocity variables for modeling super- and hypersonic flows is an urgent task of computational fluid mechanics.

Chorin [2], Patankar [3], Belotserkovsky, Gushchin, Shchennikov [4] proposed the method of splitting in terms of physical variables. In all these methods, the Navier-Stokes equations are initially solved with the pressure from the previous step, then the obtained velocity is corrected by the pressure increment gradient over the time step, for which the additional equation is solved. The problem with this approach is that they can not be used for calculating the Navier-Stokes equations, although they can be used to calculate other equations, for example, the law of conservation of energy. The use of non-divergent and inconsistent transfer rates for the Navier-Stokes equations and other convective-diffusion equations leads to a weak convergence of the whole splitting algorithm and does not allow the solution of supersonic problems with large steps in time.

The implicit method of splitting by the physical variables, which uses divergent velocities from the current time step to solve the Navier-Stokes equations. FlowVision, which allows modeling unsteady flows with moving bodies, with free and contact surfaces, as well as with the interaction between the liquid and the deformable structure. The method makes it possible to carry out calculations for large Mach numbers with an integration. The paper presents the results of calculations of known tests, which confirm the high efficiency of the proposed method.

2.2 Modification of the splitting method on physical variable

The scheme of splitting can be written in the form of the conservation equations for momentum and pressure.

$$\mathbf{W}_f^{n+1} = (\rho_c^n \tilde{\mathbf{V}}_c)_f + (p_c^{n+1} \frac{d\rho}{dp} \tilde{\mathbf{V}}_c)_{f1} - (p_c^n \frac{d\rho}{dp} \tilde{\mathbf{V}}_c)_{f1} - \tau$$

$$\rho_c^{n+1} \mathbf{V}_c^{n+1} = \tilde{\rho}_c \tilde{\mathbf{V}}_c - \tau \nabla p^{n+1} + \tau \nabla p^n,$$

where the term

$$\mathbf{W}_f^{n+1} = (\rho_c^n \mathbf{V}_c^n)_f + (p_c^{n+1} \frac{d\rho}{dp} \mathbf{V}_c^n)_{f1} - (p_c^n \frac{d\rho}{dp} \mathbf{V}_c^n)_{f1} - \tau(\nabla_f p^{n+1} - \nabla_f p^n)$$

Density ρ_c^{n+1} can be found from the equation

$$\rho_c^{n+1} = \rho_c^n + \frac{d\rho}{dp}(p_c^{n+1} - p_c^n)$$

Next we solve the energy equation

$$\frac{\rho^{n+1} H^{n+1} - \rho^n H^n}{\tau} + CD(\mathbf{W}_f^{n+1}, H^{n+1}) = \frac{p^{n+1} - p^n}{\tau}$$

from where the temperature could be found and corrections with temperature follows from EoS could be corrected.

3 Validation of Flow Vision Method for calculating the Navier-Stokes equations in the supersonic flows

2.1 Sphere in the flow

Flow around sphere for supersonic blunt body (Lunev, 2007)

$$p' = \frac{p_b}{p_0} = 1 - \left(1.2 - \frac{1.5}{M_\infty^2}\right)\sin^2\omega + \left(\left(0.27 - \frac{1.5}{M_\infty^2}\right)\right)\sin^4\omega$$

where p_b – pressure at the body surface, p_0 – pressure at the stagnation point. Distribution Max number is shown in fig.

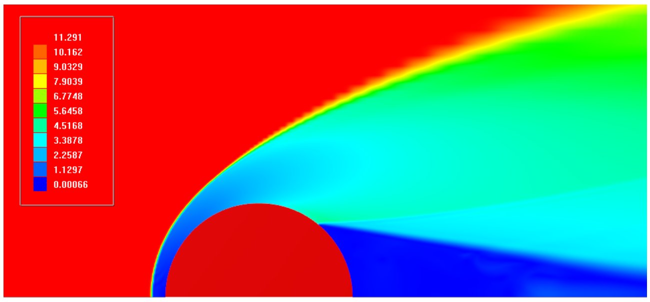
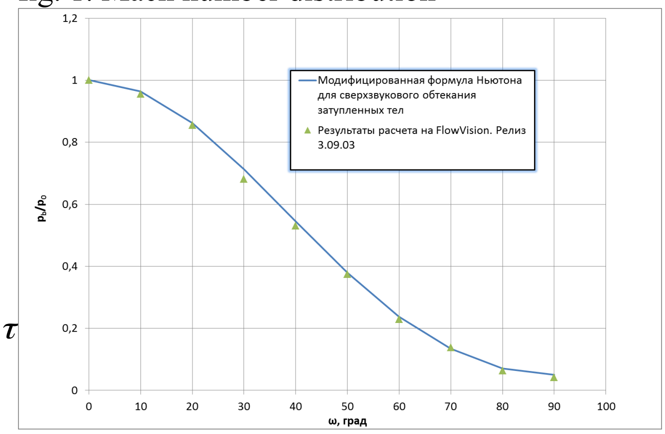


fig. 1. Mach number distribution



.

Fig. 2. Pressure distribution at the sphere surface

3 “Hexafly” simulation

The “Hexafly“ project is the joint Europe-Russia for Hypersonic Vehicle with the Hydrogen fuel at Mach number 6-8, flight at the higher atmosphere.

3.2 Numerical grid

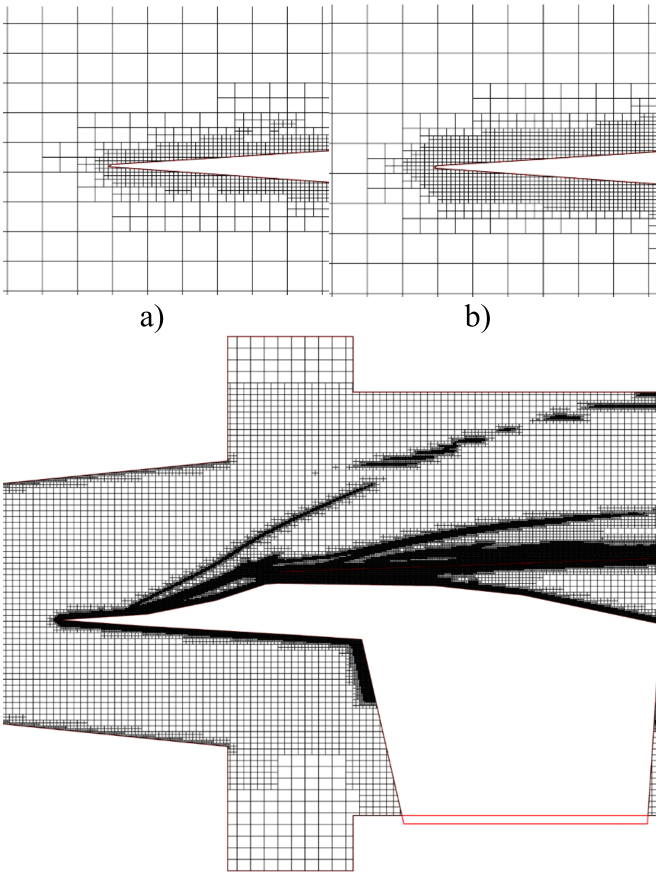


Fig.3. Calculation grid

3.3 Results of simulations

In figs 4-7 Mach number static pressure, and temperature are shown

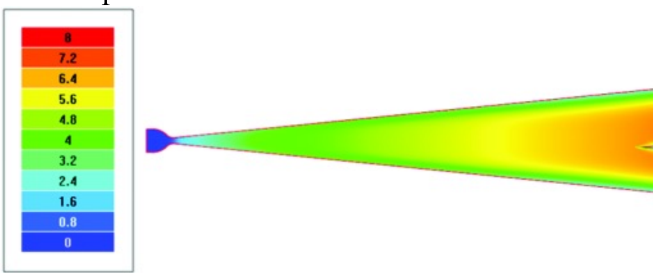


Fig. 4. Mach number distribution in the symmetry plane

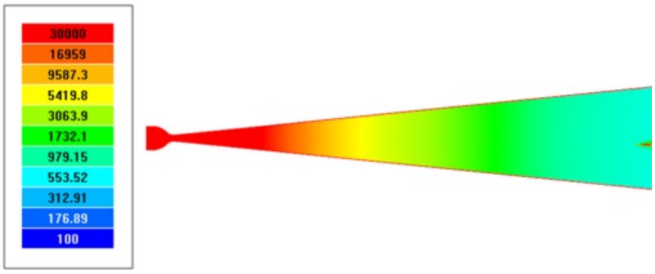


Fig. 5. Pressure distribution (Pa)

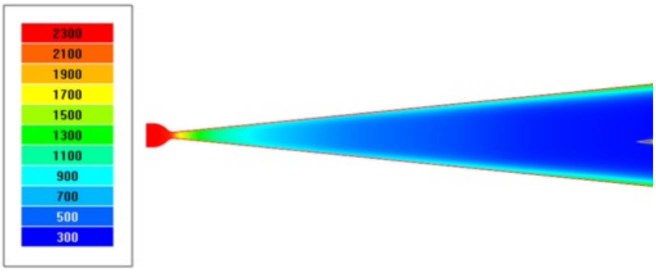


Fig.6. Static temperature distribution in the symmetry plane (K)

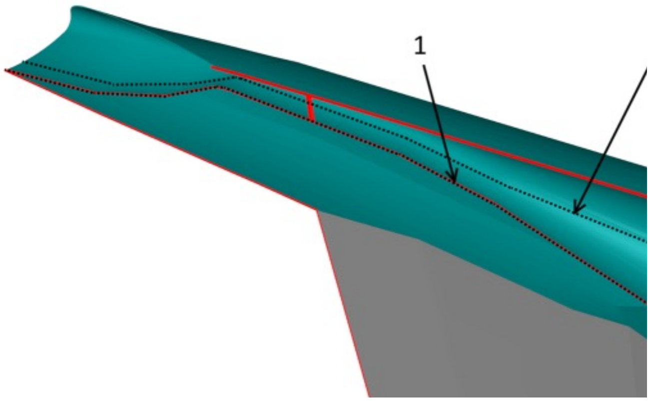


Fig 7. Positions of pressure probes at the surface of model HEXAFLY: 1 – central line probes, 2 –side line of probes.

References

[1] Degtyar V.G. and Son E.E. *Hypersonic Vehicles* 1st edition, Janus-K Publisher, 2016, 840 P., 2nd edition, Janus-K Publisher, 2018, vol.1 980 P., vol.2 1050 P.

[2] Chorin A.J. Computational Fluid Mechanics, 1st edition, Elsevier, 1989, 240 P.

[3] Patankar S., Numerical Heat Transfer and Fluid Flow, CRC Press, 1980, 214 P.

[4] Belotserkovsky O.M., Gushchin V.A., Shchennikov V.V., 1975. The method of splitting in terms of physical variables.

Copyright Statement

The authors confirm that they, and/or their company or organization, hold copyright on all of the original material included in this paper. The authors also confirm that they have obtained permission, from the copyright holder of any third party material included in this paper, to publish it as part of their paper. The authors confirm that they give permission, or have obtained permission from the copyright holder of this paper, for the publication and distribution of this paper as part of the ICAS proceedings or as individual off-prints from the proceedings.