Development and testing of non-viscous solver based on UST3D programming code

A S Kryuchkova

1 A. Ishlinsky Institute for Problems in Mechanics of the Russian academy of sciences, Vernadsky prospekt 101(1), Moscow, 119526, Russia
2 Moscow Institute of Thermal Technology, Berezovaya avenue 10(14), Moscow, 127273, Russia

E-mail: kryuchkova.arina.s@gmail.com

Abstract. Non-viscous versions of UST3D code elaborated in Laboratory of Radiative Gas Dynamics of Ishlinsky Institute for Problems in Mechanics were developed to testify the numerical method validity. Two numerical cases were examined to determine solution discrepancy between non-viscous and original algorithms. The first case consisted in computational modeling of a flow over blunted cylinder-flare configuration under zero and non-zero angles of attack. The second case represented hypersonic flow over standard HB-2 ballistic model under angle of attack. Agreement of numerical results obtained with non-viscous solver and the original code was observed. These results approve the possibility of UST3D application for non-viscous problems.

1. Introduction
During the last decades the most actual tendencies of modern aerospace industry are directed at hypersonic technologies mastering. The first attempts of hypersonic programs realization have shown the necessity of applying combined methods of hypersonic models design involving both experimental and numerical tests. While high-quality ground and flight tests require extensive investments, the accessibility of computer based simulation accounts for impetuous interest in the development of reliable digital methods. One of the major demands to be complied with these methods consists in their industrial application convenience.

UST3D (Un-Structured Tetrahedral 3-Dimentional) code [1, 2] elaborated in Laboratory of Radiative Gas Dynamics of Ishlinsky Institute for Problems in Mechanics is a numerical instrument oriented on a wide range of gas dynamics problems and currently is under active testing and development. The code performs Navier-Stokes equations numerical solution using modified method of splitting on physical processes [3, 4, 5]. For this moment the code is adapted to computations using unstructured tetrahedral grids. The choice of this adaption is related to industrial orientation of the code implying the desirability of providing comprehensible engineering application and aiming at mesh construction simplification for complex geometries. From the other hand using tetrahedral grids means that the boundary layer is not accurately resolved. However, it was still desirable to retain viscous formulation because the solution in that way is generally more stable in comparison with non-viscous formulation. Besides the current model is supposed to be applied also in future versions of the code adapted to hexaedral meshes. In this context, attention must be paid on the correct way of
employing viscous boundary conditions on surfaces for the current version of the code. It is well known that numerical solution of viscous equations performed on meshes not resolving the boundary layer is identical to the Euler, i.e. non-viscous solution. It was decided to develop a non-viscous version of UST3D code and to estimate the final solution discrepancy between the two algorithms, which could serve to verify existence of parasitic disturbances due to viscous terms contribution and present an additional verification of UST3D functionality.

2. Mathematical model

As it was already declared, UST3D code realizes numerical solution of spatial Navier-Stokes equations. In Cartesian coordinate system the equations have the form of:

$$
\frac{\partial U}{\partial t} + \frac{\partial Ec}{\partial x} + \frac{\partial Fc}{\partial y} + \frac{\partial Gc}{\partial z} = \frac{\partial Ev}{\partial x} + \frac{\partial Fv}{\partial y} + \frac{\partial Gv}{\partial z},
$$

where $U = (\rho, \rho u, \rho v, \rho w, \rho E)^T$ is conservative variables vector; $Ec = (pu, puv, pvw, pE + pu)^T$, $Fc = (\rho v, \rho vv, \rho vw, \rho pv + pvv)^T$, $Gc = (\rho w, \rho vw, \rho pw + pwv)^T$ are convective terms projections; $Ev = (0, \tau_{xx}, \tau_{yy}, \tau_{zz}, \nu_{xx} + w\tau_{xx} + q_x)^T$, $Fv = (0, \tau_{xy}, \tau_{yx}, \tau_{zy}, \nu_{xy} + w\tau_{xy} + q_y)^T$, $Gv = (0, \tau_{xz}, \tau_{zx}, \nu_{xz} + w\tau_{xz} + q_z)^T$ are viscous terms projections; $\rho$ is density; $u, v, w$ – velocity projections; $E$ – specific energy, $\tau_{ij}$ – viscous tensor components; $q_i$ – heat flux projections.

For the equations closure the ideal gas law is used:

$$
p = (\gamma - 1) \rho \left[ E - \frac{1}{2}(u^2 + v^2 + w^2) \right],
$$

where $\gamma = 1.4$ – heat capacity ratio of air.

The solver algorithm applies fictitious cells principle for boundary condition implementation. Therefore, the non-slip boundary condition realization has the form of:

$$
\vec{u}^{LB} = -\vec{u}^N, \quad T^{LB} = T^N, \quad p^{LB} = p^N,
$$

where $\vec{u}^N, T^N$ – velocity vector and temperature in the centre of the computational cell near the wall; $\vec{u}^{LB}, T^{LB}$ – velocity vector and temperature in the centre of the symmetric fictitious cell. Figure 1 graphically demonstrates the principle of boundary conditions realization using fictitious cells.

The non-viscous formulation is achieved by setting to zero viscous tangential stress components $\tau_{ij}$ and by choosing slip boundary condition on solid wall surfaces. The latter means having zero thermal fluxes and tangential orientation of velocity vector. It implies the following conditions in cells adjacent to the surface and in corresponding fictitious cells:

$$
\vec{u}^{LB} = \vec{u}^N - 2(\vec{u}^N \cdot \vec{n}), \quad T^{LB} = T^N, \quad p^{LB} = p^N
$$

Figure 1. Boundary conditions implementation: non-slip wall (a) slip wall (b), tangential velocity orientation in the cell adjacent to wall (c).
It was decided to explore additionally second type of non-viscous boundary condition formulation. The idea was to set explicitly velocity direction to tangential in the cell adjacent to the surface. This formulation corresponds to physical effect of moving the surface away from the border at the cell centre (figure 1, c) and allows estimating the order of error coming from mesh roughness. In that case static pressure needs to be corrected accordingly to momentum conservation law. The numerical implementation form is given below:

\[
\hat{u}^{LB} = \hat{u}^N - (\hat{u}^N \cdot \hat{n}), \quad \hat{T}^{LB} = T^N, \quad \hat{p}^{LB} = \frac{p^N \left(2RT^N + (\hat{u}^N)^2\right)}{\left(2RT^N + (\hat{u}^{LB})^2\right)}, \quad p^N = p^{LB},
\]

where \( \hat{u}^N, p^N, T^N \) – velocity vector, pressure and temperature in the centre of the computational cell near the wall after solver algorithm computational iteration; \( \hat{u}^{LB}, \hat{p}^{LB} \) – velocity vector and pressure after applying the boundary conditions, \( R = 287 \text{ J/kg/K} \) – gas constant of air. Here and after the formulation (5) of non-viscous boundary condition will be denoted as “\( Vn=0 \)” boundary condition while the formulation (4) will be referred as “slip” condition.

3. Simulation tests

The non-viscous solver was tested on two simulation cases. The first one represents numerical simulation of the experiment carried out in the Small Ballistic Tunnel (SBT) of N E Zhukovsky Central Aerohydrodynamic Institute (TsAGI) [6]. In this experiment a flow over blunted cylinder-flare configuration under zero angle of attack in range of Mach numbers from 1.5 to 14 was investigated. Model geometry general view is shown in figure 2(a). In numerical simulation the geometry middle diameter was assumed to be 0.2 m, nominal flow conditions were set to: \( p_\infty = 4000 \text{Pa}, T_\infty = 219 \text{K} \). Complete information on computational settings, previous computational results and their comparison with experimental data can be found in [7]. In this work the spectrum of test conditions was limited to Mach number of incoming flow \( M_\infty = 3 \). Non-zero angle of attack effect while using different boundary conditions was examined at \( \alpha=5^\circ \) and \( \alpha=20^\circ \). The goal of flow under angle of attack simulation was to verify the adequacy of velocity normal components representation in viscous and non-viscous solvers.

The second test consisted in simulation of a hypersonic flow with \( M_\infty = 9.6 \) under angle of attack \( \alpha=30^\circ \) over standard hypersonic ballistic model HB-2. The model represents one of two hypersonic ballistic configurations among the set of Advisory Group for Aerospace Research and Development (AGARD) reference models. HB-2 has analytical shape patterned in figure 2(b); the shape includes a blunted cone, cylinder, and flare. In numerical modeling the cylinder diameter \( D \) was set to 1 m. Nominal flow conditions were chosen corresponding to experiment conducted in the 1.27 m Hypersonic Wind Tunnel (HWT) of Japan Aerospace Exploration Agency (JAXA) [8]: \( p_\infty = 75 \text{Pa}, T_\infty = 52 \text{K} \).

The model flow is supposed to be laminar for the both cases. The calculi were performed on tetrahedral unstructured meshes counting 417384 cells for the first case and 1.4 million cells for the
second. The mesh density was higher round the model walls. Computational domains with indicated boundary conditions are illustrated in figure 3. Symmetry boundary condition which coincides with the pattern plane was used to reduce computational costs. Time step was chosen automatically according to condition on final discrepancy limitation.

![Figure 3](image)

**Figure 3.** General view of the computational mesh and boundary conditions settings for TsAGI (a) and HB-2 (b) test cases (zoom is shown on the right).

### 4. Results and discussions

The flow structures obtained in first test example modeling have transient behavior most strongly pronounced in the bottom area. The structure acquired in slip BC simulation occurred to be identical to the one captured with the original code. As for Vn=0 BC formulation, the solution on the boundary appears to be apparently smoother (figure 4).

Aerodynamic drag and lift force coefficients were chosen as quantitative criteria of solutions identity. The coefficients are related to the product of incident flow dynamic pressure and body midsection area \((A = \pi D^2/4)\). The summary of calculated aerodynamic drag and lift force coefficients comparison is reported in table 1. The discrepancy between the non-viscous slip BC and Navier-Stokes formulation appears to be less than 1%. Force loads on body surface while using Vn=0 boundary condition formulation are visibly lower, the difference in coefficients values calculated with original code amounts to 9.7%.
Figure 4. Mach number fields corresponding to TsAGI test case with $\alpha = 5^\circ$ obtained while applying different algorithms: original UST3D code (a), slip BC formulation (b), $V_n = 0$ BC formulation.

Table 1. Lift and drag coefficients comparison for TsAGI test case simulations.

|       | Original UST3D code | Slip BC formulation | $V_n = 0$ BC formulation |
|-------|---------------------|---------------------|--------------------------|
| $\alpha = 0^\circ$ | $C_d = 0.369; C_l = 0.00257$ | $C_d = 0.369; C_l = 0.00239$ | $C_d = 0.359; C_l = 0.00259$ |
| $\alpha = 5^\circ$ | $C_d = 0.381; C_l = 0.119$ | $C_d = 0.384; C_l = 0.118$ | $C_d = 0.371; C_l = 0.103$ |
| $\alpha = 20^\circ$ | $C_d = 0.637; C_l = 0.506$ | $C_d = 0.642; C_l = 0.51$ | $C_d = 0.603; C_l = 0.457$ |

The HB-2 case simulation was carried out only for slip BC and Navier-Stokes formulations. The same transient structure behaviour as for the flow over blunted cylinder-flare geometry was observed (see figure 5). The identity of field structures obtained with two algorithms retained actual, as well as low discrepancy in aerodynamic coefficients estimations (see table 2). Here the reference area is also model midsection area ($A = \pi (1.6D)^2 / 4$).

Table 2. Lift and drag coefficients comparison for HB-2 test case simulations.

|       | Original UST3D code | Euler solver |
|-------|---------------------|--------------|
| $\alpha = 30^\circ$ | $C_d = 0.967; C_l = 0.917$ | $C_d = 0.967; C_l = 0.918$ |
One more index of algorithm effectivity that was considered was solution convergence rate. Figure 6 presents solution convergence plots while using non-viscous solver with slip conditions and original UST3D code. It can be noticed that solution with original code converges faster in its first phase, although both solutions are stable and reach convergence after approximately 20000 iterations.

**Conclusion**
Non-viscous versions of UST3D code were implemented. The first version applied slip boundary condition on surface while the second one utilized condition of tangential velocity direction in cell adjacent to the surface. The new implementations were tested on the example of a flow over blunted cylinder-flare configuration under zero and non-zero angles of attack. Excellent agreement of the numerical results obtained with slip BC formulation and the original code was observed, while the second non-viscous formulation employment resulted in discrepancy of 9.7%. The slip BC formulation was additionally compared to the original code functioning during simulation of a hypersonic flow over HB-2 configuration. The results identity for two algorithms was confirmed which indicates adequate UST3D code functioning conformably to application on tetrahedral meshes. It was shown that the time step is more flexible while using original UST3D code. These results approve the possibility of UST3D application for non-viscous problems.

![Figure 6](image_url)

**Figure 6.** Solution convergence comparison: convergence history while using original UST3D code (a), convergence history while using non-viscous slip BC formulation (b).

**Acknowledgments**
Author is grateful to prof. S T Surzhikov and to the staff of Laboratory of Radiative Gas Dynamics (A. Ishlinsky Institute for Problems in Mechanics of the Russian academy of sciences) for useful discussions on the problems considered the study.

**References**
[1] Surzhikov S T 2017 Validation of computational code UST3D by the example of experimental aerodynamic data *J. Phys.: Conf. Ser.* **815** 012023
[2] Zheleznyakova A L and Surzhikov S T 2013 Application of the Method of Splitting by Physical Processes for the Computation of a Hypersonic Flow over an Aircraft Model of Complex Configuration *High Temperature* **51** (6) pp 816–829
[3] Ewans M V and Harlow F H 1957 The particle-in-cell method for hydrodynamic calculations *Los Alamos Scientific Lab. Rept.* N LA–2139 Los Alamos
[4] Belotserkovsky O M and Davydov Yu M 1982 *Method of the Large Particles for Gas Dynamics* (Moscow: Nauka) p 391 (in Russian)
[5] Marchuk G I 1988 *Splitting methods* (Moscow: Nauka) p 263 (in Russian)
[6] Krasil'shchikov A P and Guryashkin L P 2007 Experimental investigation of bodies of revolution in hypersonic flows (Moscow: PhysMathLit) p 208 (in Russian)

[7] Kryuchkova A S 2018 Numerical simulation of a flow over blunted cylinder-flare configuration J.Phys.: Conf. Ser. 1009 012006 DOI:10.1088/1742-6596/1009/1/012006

[8] Shigeru Kuchi-Ishi, Shigeya Watanabe, Shinji Nagai, Shoichi Tsuda, Tadao Koyama, Noriaki Hirabayashi, Hideo Sekine, and Koichi Hozumi 2005 Comparative Force/Heat Flux Measurements between JAXA Hypersonic Test Facilities Using Standard Model HB-2 (Part 1: 1.27 m Hypersonic Wind Tunnel Results). JAXA Research and Development Report