Three-dimensional calculation of the aerothermodynamics of a double cone 25°/55° on an unstructured grid

I A Koryukov\textsuperscript{1,2} and I A Kryukov\textsuperscript{1}

\textsuperscript{1}Ishlinsky Institute for Problems in Mechanics RAS, Moscow, 119526, Russia
\textsuperscript{2}Moscow Institute of Physics and Technology (State University), Dolgoprudny, Moscow region, 141701, Russia

E-mail: iakoryukov@mail.ru

Abstract. A three-dimensional numerical simulation of 25°/55° double-cone high-speed aerothermodynamics on an unstructured grid was carried out. For numerical simulation of the flow, a geometric model of a double cone 25°/55° was built. Numerical simulation using air and N\textsubscript{2} (nitrogen) was carried out. For numerical simulation, the CFD computer code UG3D developed in the Institute for Problems in Mechanics of the Russian academy of Sciences was used. This solver is based on the model of Navier-Stokes equations for a perfect gas solved on an unstructured grid. Verification and validation of the results were carried out.

1. Introduction
The high-speed flow around a double cone of 25°/55° presents considerable practical and theoretical interest. An object of interest for this flow lies at the junction of two cones, where the shock wave from the cone 25° interacts with the boundary layer of the 55° cone. Despite the simple construction of the body, in the case of numerical simulation of the flow there are some significant difficulties with exact calculation of the region of interaction of the shock wave, or the boundary layer. It makes the problem of numerical simulation of such flow a good benchmark for computer codes for high-speed flows. And as a good benchmark the problem has a lot of approved experimental data obtained by various experimental groups. One of the well-known groups is Calspan-University at Buffalo Research Center (CUBRC) \cite{9}. CUBRC conducted many rigorous experiments, which form a significant experimental database for high-speed flows. The database is widely used for verification and validation of numerical computer codes \cite{1-5}.

In the Institute for Problems in Mechanics of the Russian academy of Sciences, there is an experimental program for the formation of database for verification and validation of computer codes for high-speed flows \cite{6-8}. The database also contains some experimental results for a hypersonic flow near a double cone.

Experimental data for the flow near a 25°/55° double-cone obtained in CUBRC were estimated and then chosen as a good data set for validating the UG3D code. Most published results of numerical simulation of the flow \cite{1-5} were obtained with 2D numerical codes. So, one of the goals of this work was 3D numerical simulation of the flow. We chose the laminar case with the Mach number M = 9.59 for code validation. Obtained results are discussed and compared with CURRC experimental data and calculated results of other authors. Some features of the obtained results are considered in more detail.
2. The computational flow model

For the numerical simulation of double cone flow 25°/55° aerodynamics, it is assumed that the flow is determined by compressible Navier-Stokes equations, which describe the conservation of mass, momentum, and energy in a viscous fluid:

$$\frac{\partial w}{\partial t} + \frac{\partial F^x(w)}{\partial x} + \frac{\partial F^y(w)}{\partial y} + \frac{\partial F^z(w)}{\partial z} = \frac{\partial G^x(w)}{\partial x} + \frac{\partial G^y(w)}{\partial y} + \frac{\partial G^z(w)}{\partial z},$$  
(1)

where $F^x$, $F^y$, $F^z$ – are the projections of the convective flow vector (2); $G^x, G^y, G^z$ – projections of the viscous flow vector (3); $\rho$ - density; $P$ - pressure; $u, v, w$ - components of the velocity vector; $E$ - is specific total energy of the gas; $\tau_{\alpha\beta}$ - components of the viscous stress tensor ($\alpha = x, y, z$; $\beta = x, y, z$); $q_x, q_y, q_z$ - components of the heat flux vector [13].

$$w = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E \end{pmatrix}, \quad F^x = \begin{pmatrix} \rho u \\ \rho u^2 + P \\ \rho u \nu \\ \rho u w \\ \rho u E + Pu \end{pmatrix}, \quad F^y = \begin{pmatrix} \rho v \\ \rho v^2 + P \\ \rho v \nu \\ \rho v w \\ \rho v E + Pv \end{pmatrix}, \quad F^z = \begin{pmatrix} \rho w \\ \rho w^2 + P \\ \rho w \nu \\ \rho w v \\ \rho w E + Pw \end{pmatrix},$$  
(2)

$$G^x = \begin{pmatrix} \tau \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xz} \\ \mu \tau_{xx} + \nu \tau_{xy} + \omega \tau_{xz} - q_x \end{pmatrix}, \quad G^y = \begin{pmatrix} \tau \\ \tau_{yy} \\ \tau_{xy} \\ \tau_{yz} \\ \mu \tau_{yy} + \nu \tau_{xy} + \omega \tau_{yz} - q_y \end{pmatrix}, \quad G^z = \begin{pmatrix} \tau \\ \tau_{zz} \\ \tau_{yz} \\ \tau_{zz} \\ \mu \tau_{zz} + \nu \tau_{yz} + \omega \tau_{zz} - q_z \end{pmatrix},$$  
(3)

A complete system of Navier-Stokes equations is used together with the perfect gas equation of state (4):

$$P = (\gamma - 1) \rho U = (\gamma - 1) \rho \left[ E - \frac{1}{2} (u^2 + v^2 + w^2) \right],$$  
(4)

where $\gamma = c_p / c_v$ - heat capacity ratio; $c_p, c_v$ - specific heat at constant pressure and volume, respectively; $U = c_v T, E$ - specific internal and total energy of gas; $T$ – temperature [13]. The working fluid is assumed to be a calorically perfect gas, and the ratio of specific heats is assumed to be 1.4.

The flow solver ug3D is a density-based cell-center finite-volume Navier-Stokes solver and can be considered to be a Godunov-type solver. In order to achieve second-order spatial accuracy, a piecewise linear reconstruction MUSCL approach is used [12].

3. Statement of the problem of the high-speed flow past a model

For three-dimensional numerical modeling of aerothermodynamics of the object, computer geometry of the model was created on the basis of data from open sources [10, 11] in the format *.iges in CAD (figures 1, 2).

The initial data of the oncoming stream from the Run 28, which is used in the calculations, is presented in the table 1 [11], where parameters with the index "\(\infty\)" refer to the incoming stream.

4. Grid models

After the CAD geometric model of the double cone 25°/55° was built, a volume mesh was generated. For this purpose, the model was exported to the mesh generator. The CFD module of aerothermodynamics UG3D is able to perform calculations both on structured and unstructured grids.
However, in this case an unstructured tetrahedral volume mesh of 2.1 million cells was generated (figure 2).

![Figure 1. Schematic view of the model [11].](image1)

![Figure 2. General view of the double cone model 25°/55°.](image2)

Table 1. The initial parameters.

| №  | $V_\infty$, m/s | $\rho_\infty$, kg/m$^3$ | $T_\infty$, K | $T_w$, K | $M_\infty$ | Gas     | $\gamma$ |
|----|----------------|------------------------|-------------|---------|-----------|---------|---------|
| Run 28 | 2664.0      | 0.0006546             | 185.6       | 293.3    | 9.59      | N$_2$ (Nitrogen) | 1.39   |
| Run 28 (Air) | 2664.0 | 6.546                | 185.6       | 293.3    | 9.59      | Air     | 1.4     |

The unstructured type of grid was chosen because the structured grid requires a lot of computer resources to generate. In the future work, it is planned to perform a three-dimensional calculation on a structured grid. The obtained tetrahedral unstructured volume grid has regions of concentration along the entire surface of the geometric model (figures 3–6).

![Figure 3. General view of the grid. The number of tetrahedron = 2.1 million, the number of nodes = 391135.](image3)

![Figure 4. Grid thickening over the entire surface of the geometric model. The number of tetrahedron = 2.1 million, the number of nodes = 391135.](image4)
The quality of the grid models created was evaluated by a number of generally accepted criteria [14, 15]. The constructed grid models completely satisfy the requirements for efficient operation of the aerodynamic calculation code UG3D.

5. Results of numerical simulation

Calculations were made using the air and N₂ (nitrogen) as test gases. As an illustration of computation results, three-dimensional distributions of gas-dynamic parameters with zero angle of attack are presented (figures 7, 8, 11-13). For example, the distribution of density, Mach number and pressure are presented below. Comparison of obtained results of the pressure coefficients with the experimental data, analytical solution in the area of 25° cone (figures 9, 10) and calculations by James N. Moss was carried out [9]. Moss uses the Monte Carlo method (DSMC) in his calculations [11].

Analytical solution of pressure coefficient on the 25° cone was described by the Taylor-McCall equations [14]:

\[
\frac{\gamma - 1}{2} V_{\text{max}}^2 - V_r^2 - \left(\frac{dV_r}{d\theta}\right)^2 = 2V_r \frac{dV_r}{d\theta} \cot \theta + \frac{d^2V_r}{d\theta^2} - \frac{dV_r}{d\theta} \left[ V_r^2 \frac{dV_r}{d\theta} + dV_r \left( \frac{d^2V_r}{d\theta^2} \right) \right] = 0; \quad \frac{dV_r}{d\theta} = V_\theta
\]

(5)

where \(V_r\) – tangential component of the velocity vector; \(V_\theta\) – normal component of the velocity vector; \(\theta\) – intermediate angle of inclination of the conical generator; \(\gamma\) – adiabatic index; \(V_{\text{max}}\) – maximum isentropic flow rate.

All obtained results show density and pressure growth in the area of shock wave-boundary layer interaction. Using the air as a test gas the viscous and inviscid flow was computed. Firstly, an inviscid calculation was performed (figure 9). Obtained pressure coefficient has a bad agreement with experimental and Moss data in a 25–55 cone junction (experimental and Moss data show \(C_p\) growth beginning prior to junction). Calculated maximum rate of \(C_p\) is much higher than the experimental one, and also shifted by X coordinate. The second considered case was viscous calculations (figure 10). A viscous calculation shows a good agreement of pressure coefficient with Moss data in the region of cones junction as well as \(C_p\) peak values. However, the \(C_p\) maximum value is still shifted along the X coordinate.

Comparing calculated results of the pressure coefficient distribution using the N₂ as a test gas with the results of Moss [11] and experiment [9], there is a great agreement of \(C_p\) peak value and the placing of \(C_p\) growth beginning with Moss’ data [11] (figure 14). Comparing the results of the \(C_p\) distribution with the experimental data, one can see the difference in the peak values in the region of interaction between the shock wave and the boundary layer. Perhaps this is due to the choice of the calculation method used. Because Moss [11] uses the Monte-Carlo method and UG3D code uses the Godunov method.

Figure 5. Grid thickening on the surface of the first cone 25°.

Figure 6. Grid thickening on the surface of the second cone 55°.
**Figure 7.** Distribution of density [kg/m$^3$] in air. $M_\infty = 9.59$.

**Figure 8.** Distribution of the Mach number in the air. $M_\infty = 9.59$.

**Figure 9.** Graph of comparison of the pressure coefficient $C_p$ with the experiment Moss [11] and analytical calculation. Inviscid calculation in the air.

**Figure 10.** Graph of comparison of the pressure coefficient $C_p$ with experiment and Moss [11] and analytical calculation. Viscous calculation in the air.

**Figure 11.** Distribution of density [kg/m$^3$] in N$_2$. $M_\infty = 9.59$.

**Figure 12.** Pressure [Pa] distribution in N$_2$. $M_\infty = 9.59$. 
Figure 13. Temperature [K] distribution in N₂. \( M_\infty = 9.59 \).

Figure 14. Graph of the comparison of the pressure coefficient \( C_p \) with the experiment and Moss (DSMC). Viscous calculation in nitrogen.

Comparison of calculated data with the results of Gnofo calculation [10] and experimental shadowgraph are also considered. Figure 15 presents a pressure-contour comparison of Gnofo and UG3D numerical simulations. It can be seen from the comparison that the area of maximum pressure in the shock wave – boundary layer region is shifted below, locally closer to the 25°-55° cone conjunction. It is also seen that the region of interaction of the shock wave with the boundary layer is smaller than in the results of Gnofo. Gnofo uses the LAURA code based on the Navier-Stokes equations [16]. An experimental shadowgraph [9] is compared to numerically calculated density gradient in figure 16.

Figure 15. The result of comparing the pressure distribution on the model surface (bottom) with the results of Gnofo [10] (upper part).

Figure 16. The result of comparing the numerical simulation of the density gradient (bottom) with the experimental data [9] (upper part).

Differences in calculated and experimental maximum rates of gas-dynamic parameters in shock wave – boundary layer interaction region may be explained by a mesh detailed insufficiently for
adequate modeling of all the effects of the boundary layer in the calculation. The problem of more detailed grid with a separately constructed boundary layer generation to obtain better results is a part of the future works.

6. Conclusion

Three-dimensional numerical simulation of 25°/55° double-cone high-speed aerothermodynamics on an unstructured grid was carried out. Three-dimensional numerical simulation was carried out using the air and N₂ (nitrogen) as test gases. The CFD UG3D solver, developed in the Institute for Problems in Mechanics of the Russian academy of Sciences, was used for numerical simulation. This computer code is based on the model of Navier-Stokes equations for a perfect gas solved on an unstructured grid.

For numerical simulation of the flow, a geometric model and unstructured tetrahedral mesh of a double cone 25°/55° was generated. Verification and validation of the results were carried out. Obtained numerical results have a good agreement of pressure coefficient with the analytical solution along the 25° cone surface in all cases. A viscous calculation shows a good agreement of pressure coefficient with Moss' data in the region of cones junction as well as Cp peak values. However, the Cp maximum value is still shifted along the X coordinate. There is a great agreement of Cp peak value and the placing of Cp growth beginning with Moss' data using the N₂ as a test gas. In all cases, there is an agreement in the character of calculated pressure coefficient distribution with the experimental data but the peaks in the shock wave - boundary layer interaction region are shifted.

Further simulations of 25°/55° double-cone high-speed aerothermodynamics with a more detailed computational grid in the boundary layer area are supposed to improve the agreement of calculations with the experiment results.

The presented study was partially supported by Russian State Assignment under contract No.AAAA-A17-117021310372-6.

References

[1] Jiaoo H, Jingying W and Chunhian L 2017 Numerical Simulation of High-Enthalpy Double-Cone Flows AIAA J. 55 7 pp 2471–2475
[2] Xiao H, Shang Y, Wu D 2014 DSMC simulation and experimental validation of shock interaction in hypersonic low density flow, Scientific World J.
[3] Druguet M C, Candler G V and Nompelis I 2003 Simulations of viscous hypersonic double-cone flows: Influence of numerics. In 16th AIAA Computational Fluid Dynamics Conf.
[4] Candler G V, Nompelis I, Druguet M C, Holden M S, Wadhams T P, Boyd I D et al 2002 CFD validation for hypersonic flight: Hypersonic double-cone flow simulations. In 40th AIAA Aerospace Sciences Meeting and Exhibit
[5] Reichel S and Groll R 2013 Numerical Simulation And Experimental Validation Of A Hypersonic Flow, For International Journal of Computational Methods and Experimental Measurements
[6] Kotov M A, Kryukov I A, Ruleva L B, Solodovnikov S I and Surzhikov S T 2013 June 24-27 Experimental Investigations of an Aerodynamic Flow of Geometrical Models in Hypersonic Aerodynamic Shock Tube 31st AIAA Applied Aerodynamics Conference San Diego, CA, AIAA Paper 2013–2931 15 p
[7] Kotov M A, Kryukov I A, Ruleva L B, Solodovnikov S I and Surzhikov S T 2015 Supersonic Air Flows Around Some Geometrical Primitives, 33rd AIAA Applied Aerodynamics Conference, AIAA AVIATION Forum, AIAA Paper 2015-3012
[8] Kotov M A, Kryukov I A, Ruleva L B and Solodovnikov S I 2017 The Investigation of Shock-Wave Interaction with Aerodynamic Models, 55th AIAA Aerospace Sciences Meeting, AIAA SciTech Forum, AIAA Paper 2017-0262
[9] Holden M S and Wadhams T P Jan. 2001 Code Validation Study of Laminar Shock/Boundary
Layer and Shock/Shock Interactions in Hypersonic Flow Part A: Experimental Measurements, *AIAA Paper* 2001-1031

[10] Gnoffo P A 2003 Computational Fluid Dynamics Technology For Hypersonic Applications, *NASA Langley Research Center Hampton, VA* 23681-0001

[11] James N Moss 2001 Hypersonic Flows About a 25° Sharp Cone *NASA Langley Research Center, MS 408A, Hampton, VA* 23681-2199

[12] Barth T J and Jespersen D C 1989 The design and application of upwind schemes on unstructured meshes, *AIAA Paper* 1989-0366

[13] Surzhikov S T 2017 Validation of computational code UST3D by the example of experimental aerodynamic data *J. of Physics: Conference Since* 815 012023

[14] Liseikin V D 1999 *Grid Generation Methods* (Berlin: Springer)

[15] Thompson J F, Soni B K, Weatherill N P 1998 *Handbook of Grid Generation* (CRC Press)

[16] Cheatwood F M and Gnoffo P A 1996 User’s Manual for the Langley Aerothermodynamic Upwind Relaxation Algorithm (LAURA), *NASA TM 4674*