Numerical Study of the Supersonic Base Flow around Aircraft Model

Ya V Khankhasaeva, V E Borisov and A E Lutsky
Keldysh Institute of Applied Mathematics, Moscow, Russia

Abstract. This paper deals with a numerical study of the base part of the supersonic flow past an aircraft model. The simulation was carried out using the software developed by the authors for calculating 3D turbulent flow of viscous compressible gas in the framework of the unsteady Reynolds averaged Navier–Stokes equations (URANS) with the Spalart–Allmaras (SA) and Menter’s SST turbulence models. Flight modes with M = 2.5 and angle of attack \( \alpha = 0^\circ, 3^\circ \) were considered. A comparison is made of the effect of the choice of the turbulence model on the base flow.

1. Introduction

Numerical simulations of gas-dynamic processes around an aircraft is an integral stage of its design, allowing to obtain aerodynamic characteristics with small (relative to field experiments) costs. At the same time, due to the complex design of modern aircrafts, as well as high (super- and hypersonic) flight speeds, the structure of the emerging currents is quite complicated due to the presence of narrow zones of large gradients, boundary layers, shock waves, etc. Correct calculation of all these features makes high demands on the quality of the computational model.

One of the major problems in the supersonic flow around the aircraft is the simulation of currents in extended separation zones in the base part of the flow in order to determine the base pressure and wake parameters downstream. The base flow features have a significant effect on the aerodynamic characteristics of the aircraft. In particular, base drag makes a significant contribution to the total drag of the aircraft. Schematically, separated flow in the base region of the supersonic flow around a blunt cone is shown in figure 1. In front of the axisymmetric body (1), a detached shock wave (2) is formed. Boundary layer (3) adjacent to the body surface separates from the rear edge of the cone forming a viscous mixing layer (7). At the trailing edge of the cone in the fan of rarefaction waves (4) the flow turns toward the axis. As the free shear layer approaches the axis of symmetry, a recompression process (5) occurs which eventually realigns the flowfield with the axis. Current line (6) separates the recirculation zone (9) from the rest of the stream which forms a turbulent trace (8) after the point of reattachment. Because of the complexity of making reliable measurements mainly in the separated region, few experimental works were devoted to supersonic base flows. Works [1,2] contain experimental data of base pressure coefficients for bodies with different aft forms, with and without wings and fins, etc., for different Mach numbers and angles of attack. In [3] experiments were performed in order to study all features of axisymmetric base flow past a cylinder. The paper [4] is devoted to the same topic but with a non-zero angle of attack. In [5, 6] hypersonic flow around aircrafts of different configurations for a wide range of Mach numbers and angles of attack, including...
the accompanying base flow, was experimentally investigated. In [7] a generalization of a large number of experimental data obtained for a wide spectrum flight speeds is made. The main types of base ledges encountered in practice are considered, the flow pattern behind the flat, axially symmetric, three-dimensional ledges, including flow asymmetry, caused by the presence of an angle of attack, is investigated.

![Figure 1](image.png)

**Figure 1.** Scheme of the base flow.

Also in recent years a large number of numerical studies of base flow have been carried out [8-25]. Correct modeling of such flows is a difficult task, especially when calculating the flow around an aircraft in a complete layout. The following works are devoted to the numerical study of base flow in various formulations and with different numerical approaches, such as: (a) Reynolds averaged Navier–Stokes equations (RANS) with various turbulent models [8-16, 23], (b) Large Eddy Simulations (LES) [11,13,17, 21], Direct Numerical Simulation (DNS) [18], RANS/LES hybrid methods [11-13,19,20], Detached Eddy Simulations (DES) and its modifications DDES, ZDES [11,15,16,19, 21-23]. All methods have their own advantages and limitations.

The RANS methods had been developed to predict such important flow characteristics as the force, moment and velocity, etc., but were not intended to model complex unsteady coherent structures, pressure fluctuations or turbulent stresses inherent in base flow. However, such methods are widely used for analyzing the afterbody design options. LES method works well for resolving large scale unsteady typically time- and geometry-dependent flows but requires a significantly better grid resolution. Also, sub-grid models for the boundary layer and compressible flows are still not sufficiently well-developed to accurately predict supersonic base flows. Hybrid methods RANS/LES and DES use combinations of the methods described above depending on the flow characteristics. The turbulence model used in any of these methods should simulate near-wall turbulence with good accuracy.

This work presents the results of numerical study of the supersonic flow around the prototype aircraft TRK, particularly its base flow. Flight modes with $M = 2.5$ and angles of attack $\alpha = 0^\circ, 3^\circ$ were considered. The simulation was carried out using the software developed by the authors for calculating 3D turbulent flow of viscous compressible gas in the framework of the unsteady Reynolds averaged Navier–Stokes equations (URANS) with the Spalart–Allmaras (SA) and Menter’s SST turbulence models. A comparison is made of the effect of the choice of the turbulence model on the base flow.

2. Problem statement

The problem considered is the supersonic flow around aircraft TRK (figure 2). Aircraft consists of a pointed cylinder, two wings and four fins. The radius of the cylinder is taken as the unit of length. The radius of curvature of the nose is $r = 0.03$.

Calculations were performed for the Mach number $M = 2.5$ and angles of attack $\alpha = 0^\circ$ and $3^\circ$. The Reynolds number was $\text{Re} = 1.0 \times 10^7$ [m$^{-1}$].

Calculation area was a truncated cone, in the center of which the device was located. A hexagonal block-structured grid with 1 596 672 elements was used. The lateral conical boundaries were located
at a sufficient distance from the body, so that they did not intersect with the shock waves. On the wall, an adiabatic boundary condition was used.

**Figure 2.** Aircraft model and calculation area.

### 2.1. Basic equations

For calculating flows of viscous compressible gas the unsteady Reynolds averaged Navier–Stokes equations (URANS) are used which in Cartesian coordinate system \( (x_1 = x, x_2 = y, x_3 = z) \) have the following form:

\[
\frac{\partial \mathbf{q}}{\partial t} + \frac{\partial \mathbf{f}_j}{\partial x_j} = \frac{\partial \mathbf{g}_j}{\partial x_j}
\]

(1)

where the summation is over \( j = 1, 2, 3 \); \( \mathbf{q} \) – vector of conservative variables; \( \mathbf{f}_j, \mathbf{g}_j \) – vectors of inviscid and viscous fluxes, respectively:

\[
\mathbf{q} = \begin{pmatrix}
\rho \\
\rho u_1 \\
\rho u_2 \\
\rho u_3 \\
\rho E^*
\end{pmatrix}, \quad \mathbf{f}_j = \begin{pmatrix}
\rho u_j \\
\rho u_1 u_j + \delta_{1j} p \\
\rho u_2 u_j + \delta_{2j} p \\
\rho u_3 u_j + \delta_{3j} p \\
u_j \left( \rho E^* + p \right)
\end{pmatrix}, \quad \mathbf{g}_j = \begin{pmatrix}
0 \\
\tau_{1j} \\
\tau_{2j} \\
\tau_{3j} \\
\tau_{ij} u_i + h_j
\end{pmatrix}.
\]

Here \( \rho \) – density; \( u_j \) – velocity vector \( u \) components; \( \tau_{ij} \) – components of the viscous stress tensor; \( \delta_{ij} \) – Kronecker tensor; \( E^* \) – total energy of turbulent flow:

\[
E^* = E + k,
\]

where \( E \) – total energy of the mean flow, \( k \) – kinetic energy of turbulent pulsations. The thermodynamic pressure \( p \) is calculated from the equation of state of the perfect gas:
\[ p = (\gamma - 1) \rho \left( E - \frac{1}{2} \sum_{j=1}^{3} u_j^2 \right). \]

Here \( \gamma \) – adiabatic index, for air \( \gamma = 1.4 \). Components of the viscous stress tensor and heat flux vector

\[
\tau_{ij} = \mu_{\text{eff}} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu_{\text{eff}} \frac{\partial u_i}{\partial x_j} \delta_{ij} - \frac{2}{3} \rho k \delta_{ij},
\]

\[
h_j = \lambda_{\text{eff}} \frac{\partial T}{\partial x_j}.
\]

In the above equations, \( T \) is the gas temperature and is determined from formula

\[ T = \frac{p}{\rho R}, \]

where \( R \) – gas constant, for air \( R = 287 \ J/(kg \cdot K) \).

The "effective" value of the coefficients of viscosity and thermal conductivity is defined as

\[ \mu_{\text{eff}} = \mu + \mu_t, \]

\[ \lambda_{\text{eff}} = C_p \left( \frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right). \]

Here \( C_p \) – coefficient of specific heat of gas at constant pressure; \( Pr \) \( \text{and} \) \( Pr_t \) – laminar and turbulent Prandtl number, for air \( Pr = 0.7 \) ; \( Pr_t = 0.9 \). Molecular viscosity is determined by following power-law dependence:

\[ \frac{\mu}{\mu_0} = \left( \frac{T}{T_0} \right)^s, \]

where \( s = 0.76 \), \( \mu_0 \) – molecular viscosity at a reference temperature \( T_0 \).

The value of \( \mu_t \) denotes the additional turbulent viscosity, which is determined by the turbulence model.

The initial and boundary conditions for the system of equations (1) are established in the standard way [26].

2.2. Spalart-Allmaras turbulence model

The version of the one-parameter Spalart-Allmaras turbulence model (SA) for compressible flows [27] with Edwards modification [28] is used in this paper. Within the framework of this model, the averaged value of the kinetic energy of turbulent pulsations cannot be found directly, by virtue of which it is assumed \( k = 0 \).

The turbulent viscosity is given by

\[ \mu_t = \rho \tilde{v} f_{vi}, \]

\[ f_{vi} = \frac{\chi}{\chi^3 + C_{vi}}, \]

\[ \chi = \rho \frac{\tilde{v}}{\mu}, \]

where \( \tilde{v} \) – model variable, which is determined from the basic equation of the model

\[
\frac{\partial \rho \tilde{v}}{\partial t} + \nabla \cdot (\rho \tilde{v} \mathbf{u}) = \rho (P_v - D_v + T_v) + \frac{1}{\sigma_v} \nabla \cdot \left[ (\mu + \mu_t) \nabla \tilde{v} \right] + C_{v2} \frac{1}{\sigma_v} \rho \left( \nabla \tilde{v} \right)^2 - \frac{1}{\sigma_v} \rho (\mu + \rho \tilde{v}) \nabla \rho \cdot \nabla \tilde{v}.
\]
Values $P_v$ and $D_v$ responsible, respectively, for the production and dissipation of turbulence, and $T_v$ for the determination of the laminar-turbulent transition in the boundary layer, are written as

$$P_v = C_{bl}(1-f_{i2})\hat{S}_v, \quad D_v = \left[C_{wl}f_w - C_{b1}\frac{C_{b2}}{k^2}f_{i2}\right]\left[\frac{\bar{v}}{s}\right], \quad T_v = f_{i1}(\Delta U)^2,$$

$$f_w = g\left[\frac{1+C_{b3}^2}{g^2+C_{b3}^2}\right]^{1/6}, \quad g = r + C_v\left(r^6 - r\right).$$

Here $\Delta U$ – module of the difference between the velocities in the flow and the nearest point of the laminar-turbulent transition; $d$ is the distance from the solid wall.

In the Edwards modification of the Spalart-Allmaras model [28], the quantities $\tilde{S}$ and $r$ have the form

$$\tilde{S} = \sqrt{S\left[1 + f_{i1}\right]}, \quad S = 2\sum_{i,j}\left[\frac{\partial u_i}{\partial x_j}\frac{\partial u_i}{\partial x_j} - \frac{2}{3}\left(\frac{\partial u_i}{\partial x_i}\right)^2\right],$$

$$r = \tanh\left[\frac{\tilde{v}}{Sk_d^2d^2}\right]/\tanh(1.0).$$

The remaining values are constants of the SA turbulence model and are presented in table 1.

**Table 1.** Constants of SA turbulence model.

| $\sigma$ | $k_r$ | $C_{bl}$ | $C_{b2}$ | $C_{v1}$ | $C_{w1}$ | $C_{w2}$ | $C_{w3}$ |
|---------|-------|--------|--------|--------|--------|--------|--------|
| 2/3     | 0.41  | 0.1335 | 0.622  | 7.1    | $C_{wl} = \frac{C_{b1}}{k^2} + \frac{1+C_{b2}}{\sigma}$ | 0.3     | 2.0     |

When modeling a completely turbulent boundary layer, taking into account $f_{i1}$ and $f_{i2}$ does not make any significant changes to the solution, so they are usually neglected [27].

2.3. Menter’s SST turbulence model

This work uses a modified version of the Menter's two-parameter model SST (Shear Stress Transport) [29], so called SST–2003.

The turbulent viscosity is given by

$$\mu_t = \frac{\rho k}{\max(\omega, SF_2/\alpha)}$$

where, as before, $k$ is the kinetic energy of turbulent pulsations, and $\omega$ is specific dissipation rate. These quantities are determined from the equations of the model:

$$\frac{\partial \rho k}{\partial t} + \nabla \cdot (\rho ku) = \nabla \cdot \left((\mu + \sigma_t\mu_t)\nabla k\right) + P_k - \beta' \rho \omega k,$$  \hspace{1cm} (3)

$$\frac{\partial \rho \omega}{\partial t} + \nabla \cdot (\rho \omega u) = \nabla \cdot \left((\mu + \sigma_\omega\mu_t)\nabla \omega\right) + \gamma_1\frac{P_k}{\mu_t} - \beta' \rho \omega^2 + 2(1-F_t)\frac{\rho \sigma_{\omega}^2}{\omega} \nabla k \nabla \omega.$$ \hspace{1cm} (4)

The generator term $P_k$ is calculated as

$$P_k = \min\left(P, 10\beta' \rho \omega k\right)$$
Here

\[ P = \mu S^2, \quad S = \sqrt{2S_y S_z}, \quad S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right). \]

The interface functions of the SST model are determined according to formulas:

\[ F_i = \tanh \left( \arg_i \right), \quad \arg_i = \min \left[ \max \left( \frac{\sqrt{k}}{\beta \omega d}, \frac{500 \mu}{\rho \omega d^2} \right), \frac{4 \rho \sigma_{w_k} k}{CD_{w_k} d^2} \right], \]

\[ CD_{w_k} = \max \left( \frac{2 \rho \sigma_{w_k} \nabla k \nabla \omega, 1.0 \times 10^{-10}}{\omega} \right). \]

\[ F_2 = \tanh \left( \arg_2 \right), \quad \arg_2 = \max \left( \frac{2 \sqrt{k}}{\beta \omega d}, \frac{500 \mu}{\rho \omega d^2} \right), \]

where \( d \) is the distance from the solid wall.

The coefficients of the SST model are defined as:

\[ \phi = F_1 \phi_1 + (1 - F_1) \phi_2, \quad \phi = \sigma_k, \sigma_w, \gamma, \beta \]

The remaining values are model constants and are presented in table 2.

**Table 2.** Constants of SST turbulence model.

| \( \sigma_k \) | \( \sigma_{k_1} \) | \( \sigma_{w_1} \) | \( \sigma_{w_2} \) | \( \beta_1 \) | \( \beta_2 \) | \( \gamma_1 \) | \( \gamma_2 \) | \( \beta^{*} \) | \( \alpha \) | \( \alpha_1 \) | \( \alpha_2 \) | \( \alpha_3 \) |
|---------------|---------------|---------------|---------------|----------|----------|----------|----------|----------|----------|----------|----------|----------|
| 0.85          | 1.0           | 0.5           | 0.856        | 0.075    | 0.0828   | 5/9      | 0.44     | 0.09     | 0.31     | 1.0      | 0.4      | 0.2      |

3. Numerical simulation results

Numerical simulations were carried out using the software developed by the authors for calculating 3D turbulent flow of viscous compressible gas on high-performance computing systems. The basis of the computational core of the software complex was the mathematical model described above based on the URANS equations with the Spalart–Allmaras and Menter’s SST turbulence models. Approximation of the equations over space was carried out by the finite volume method with the reconstruction schemes of the 2nd (TVD) or the 3rd (WENO3) order of accuracy, both the explicit and the implicit scheme based on the LU-SGS method were used for the time approximation. Simulations were carried at the Keldysh Institute of Applied Mathematics on supercomputers K–100 and K–60 using 24 to 84 cores.

3.1. Flow at angle of attack case \( \alpha = 0^\circ \)

As shown in figure 3, there is a bow shock and the shock waves from wings and fins. Base flow (figure 4) corresponds to the described in introduction. Between the shock wave and the boundary layer there is a region of weakly viscous flow. The boundary layer separated at the trailing edge of the body forming a viscous mixing layer which constrains the recirculation region. At the trailing edge the flow turns toward the axis through the fan of rarefaction waves. At some distance from the base reattachment of the free shear layer takes place. A reattachment shock is formed. After the attachment point, a turbulent wake develops.

In general, the results obtained for SA and SST are comparable. The main differences between them are observed in the wake of base flow (figure 5). The maximum turbulent viscosity for SA is
0.020947, for SST it is 0.018235. In region after the body SA has slightly higher turbulent viscosity values.

Figure 3. Pressure distribution at $\alpha = 0^\circ$, slice $z = 0$ and model surface, SA.

Figure 4. Density distribution and streamlines in the base region, $\alpha = 0^\circ$, SA.

Figure 5. Density distribution (left) and turbulent viscosity (right) in the base region, $\alpha = 0^\circ$, SA ($y>0$) and SST ($y<0$).

Due to the lack of experimental data on the aircraft model considered in this paper, comparison of the obtained numerical information will be performed with experimental data [3] and data [11].
obtained with RANS and SA. Despite the fact that in this paper the base cut differs somewhat from that considered in [3] and [11] due to the presence of fins, such a comparison is meaningful. Since the simulation was carried out in the framework of URANS with SA and SST turbulence models, as in [11], the obtained base pressure, streamwise velocity component along the axis of symmetry from the base and the location of the reattachment point (figure 6) do not match the experimental data [3], where \( C_{p_b} = -0.102 \). However, this can also be the effect of fins, since, according to [4], the presence of fins at zero angle of attack reduces base pressure. Also for a body similar to that considered in this paper, with the exception of the absence of wings, \( C_{p_b} = -0.116 \) [4]. The difference between \( C_{p_b} \) from this work and [2] is not as great as it is for [3]. The coincidence with the numerical data [11] is good, both in the quantitative and qualitative sense. The observed change in the value of the base pressure as the flow approaches the base in the recirculation zone is due to the overestimated values of the longitudinal velocity (figures 6-8).

**Figure 6.** Base pressure coefficient (left) and streamwise velocity along the axis of symmetry from the base (right), comparison with experimental data [3] and numerical data [11] with SA.

**Figure 7.** Streamwise velocity, slices \( l/R = 0.1575 \) (a); 0.9449 (b), comparison with experimental data [3] and numerical data [11] with SA.
Figure 8. Streamwise velocity, slices $l/R = 1.8898$ (c); $2.5197$ (d), comparison with experimental data [3] and numerical data [11] with SA.

It can also be seen that the SA data is in better agreement with the data from [11] than SST because [11] also uses SA model. The shift of the reattachment point from the data [11], perhaps, is the effect of fins.

Comparison of the streamwise velocity with [3] and [11] in the slice $l/R = 0.1575$ ($l$ is the distance from the base) (figure 7a), where the boundary layer has just separated from the body, shows that RANS simulations have a somewhat overestimated streamwise velocity at the base cut as compared with experiment. In the mixing layer, the differences are not as big. An overstated streamwise velocity is also observed for the remaining slices $l/R = 0.9449$ (figure 7b), $1.8898$ (approximate start of the recompression zone) (figure 8c) and $2.5197$ (near reattachment point) (figure 8d).

Fins also affect the base flow. One of the obvious expressions of this influence is the change in the wake shape behind the body (figure 9), which now has a diamond-shaped cross section.

Figure 9. Slices of density distribution in the base region, $\alpha = 0^\circ$, SA.
3.2. Flow at angle of attack case $\alpha = 3^\circ$

In general, the structure of the flow varies little with increasing angle of attack. The differences lie in the fact that rarefaction occurs above the upper surface and pressure increases on the lower surface (figure 10). The intensity of the head shock wave on the lower part of the nose increases, on the upper part decreases. Lift force appears.

![Figure 10. Pressure distribution, $\alpha = 3^\circ$, slice $z = 0$, SA.](image)

Base flow becomes asymmetric in slice $z = 0$ (figures 11, 12). The observed base flow structures are qualitatively similar to those found in zero-angle-of-attack but are rotated to an angle roughly corresponding to the angle of attack $\alpha = 3^\circ$. Length of the recirculation region becomes slightly shorter, reattachment point shifts closer to the body.

![Figure 11. Density distribution, $\alpha = 0^\circ$ (left) and $\alpha = 3^\circ$ (right), slice $z = 0$, SA.](image)

![Figure 12. Pressure distribution and streamlines, $\alpha = 0^\circ$ (left) and $\alpha = 3^\circ$ (right), slice $z = 0$, SA.](image)
As can be seen from figure 13, base pressure coefficient for $\alpha = 3^\circ$ is lower than for $\alpha = 0^\circ$, which agrees with [2,4], where it is stated that with increasing angle of attack base pressure coefficient decreases. For SST turbulence model, the situation is exactly the same and differs a bit from results by SA, that is why it is not given here.

![Figure 13. Base pressure coefficient, $\alpha = 0^\circ$ и $\alpha = 3^\circ$, slice $z = 0$, SA.](image)

4. Conclusions
In this work numerical study of supersonic flow around aircraft model TRK, particularly its base flow, in full layout for several flight modes was performed. The simulation was carried out in the framework of the unsteady Reynolds averaged Navier–Stokes equations (URANS) with the Spalart–Allmaras (SA) and Menter’s SST turbulence models. The results of the calculations were compared with the available experimental data and the work of other authors. The qualitative and quantitative agreement of the results was demonstrated. The effect of the turbulence model used on the final result is studied separately; it is shown that for the given problem both models are comparable.

Acknowledgments
This work was supported by the Russian Foundation for Basic Research, projects No. 16-31-00399, 17-08-00909.

References
[1] Love E S 1957 Base Pressure at Supersonic Speeds on Two-Dimensional Airfoils and on Bodies of Revolution with and without Turbulent Boundary Layers, *NACA TN 3819*
[2] Moore F G, Wilcox F and Hymer T 1992 Improved Empirical Model for Base Drag Prediction on Missile Configurations Based on New Wind Tunnel Data *NSWCDD/TR–92/509*
[3] Herrin J L and Dutton J C 1994 Supersonic Base Flow Experiments in the Near Wake of a Cylindrical Afterbody *AIAA Journal* **32** 1 pp 77–83
[4] Boswell B and Dutton J 2001 Flow Visualizations and Measurements of a Three-Dimensional Supersonic Separated Flow *AIAA Journal* **39** 1 pp 113–21
[5] Surzhikov S, Borovoi V and Skuratov A 2004 Study of Convective Heating of Segmental-Conical Martian Descent Vehicle in Shock Wind Tunnel 34th *AIAA Fluid Dynamics Conference and Exhibit, Fluid Dynamics and Co-located Conferences AIAA* 2004-2634
[6] Borovoy V Y, Egorov I V, Skuratov A S and Struminskaya I V 2010 Peculiarities of flow and heat transfer in the base area of interplanetary probes *TsAGI Science Journal* **41** 3 pp 227-57
[7] Lavrukhin G N and Popovich K F 2017 Aerodynamics of jet nozzles Vol. 2 Gas flow around base cutoffs (Moscow: Fizmatlit) (in Russian)
[8] Suzen Y B and Hoffmann K A 1998 Investigation of supersonic jet exhaust flow by one- and two-equation turbulence models *AIAA Paper* 98–0322
[9] Tucker P K and Shyy W 1993 A Numerical Analysis of Supersonic Flow over an Axisymmetric
Afterbody *AIAA Paper* 93-2347

[10] Papp J L and Ghia K N 2001 Application of the RNG Turbulence Model to the Simulation of Axisymmetric Supersonic Separated Base Flow *AIAA Paper* 2001-0727

[11] Simon F, Deck S, Guillen P and Sagaut P 2006 Reynolds-Averaged Navier-Stokes/Large-Eddy Simulations of Supersonic Base Flow *AIAA Journal* **44** 11 pp 2578-90

[12] Baurle R A, Tam C J, Edwards J R and Hassan H A 2003 Hybrid Simulation Approach for Cavity Flows: Blending, Algorithm and Boundary Treatment Issues *AIAA Journal* **41** 8 pp 1463–80

[13] Kawai S and Fujii K 2005 Computational Study of Supersonic Base Flow Using Hybrid Turbulence Methodology *AIAA Journal* **43** 6 pp 1265–75

[14] Simon F, Deck S, Guillen P and Cazacu R 2006 Numerical Simulations of Projectile Base Flow *AIAA Paper* 2006-1116

[15] Forsythe J R, Hoffmann K A, Cummings R M and Squires K D 2002 Detached-Eddy Simulation with Compressibility Corrections Applied to a Supersonic Axisymmetric Base Flow *Journal of Fluids Engineering* **124** pp 911–23

[16] Xiao Z and Fu S 2009 *Acta Mech Sin* **25**: 471

[17] Fureby C, Nilsson Y and Andersson K 1999 Large-Eddy Simulation of Supersonic Base Flow *AIAA Paper* 99-0426

[18] Sandberg R D and Fasel H F 2005 Direct Numerical Simulations of Transitional Supersonic Base Flows *AIAA Paper* 2005-98

[19] Strelets M 2001 Detached eddy simulation of massively separated flows *AIAA paper* 2001–0879

[20] Fureby C and Kupiainen M 2003 Large Eddy Simulation for Supersonic Axisymmetric BaseFlow *Turbulent Shear Flow Phenomena, TSFP3. Sendai, Japan*

[21] Deck S, Garnier E 2005 Detached and Large Eddy Simulation of Unsteady Side-Loads Over An Axisymmetric Afterbody *Fifth European Symposium on Aerothermodynamics for Space Vehicles* **563**

[22] Simon F, Deck S, Guillen P, Sagaut P, Merlen A 2007 Numerical simulation of the compressible mixing layer past an axisymmetric trailing edge *Journal of Fluid Mechanics* **591** pp 215-53

[23] Garbaruk A, Shur M, Strelets M and Travina A 2009 Supersonic base flow *DESIDER — A European Effort on Hybrid RANS-LES Modelling. Notes on Numerical Fluid Mechanics and Multidisciplinary Design* **103** pp197-206

[24] Surzhikov S T 2016 Aerodynamics of the re-entry spacecraft Stardust within the hypersonic flight *Herald of the Bauman Moscow State Tech. Univ., Mech. Eng.* **3** pp 4-22

[25] Zheleznyakova A, Kuzenov V, Petrusov A and Surzhikov S 2010 The modelling of hydrodynamics for two different models of space vehicle *Physical-Chemical Kinetics in Gas Dynamics* **9**

http://chemphys.edu.ru/issues/2010-9/articles/144/

[26] Bykov L V, Molchanov A M, Scherbakov M A and Yanychev D S 2015 Computational mechanics of continuous media in the problems of aviation and space technology (Moscow: Lenand) (in Russian)

[27] Allmaras S R, Johnson F T and Spalart P R 2012 Modifications and Clarifications for the Implementation of the Spalart-Allmaras Turbulence Model *Seventh International Conf. on CFD (ICCFD7), Big Island, Hawaii*

[28] Edwards J R and Chandra S 1996 Comparison of Eddy Viscosity-Transport Turbulence Models for Three-Dimensional, Shock-Separated Flowfields *AIAA Journal* **34** 4 pp 756–63

[29] Menter F R, Kuntz M and Langtry R 2003 Ten Years of Industrial Experience with the SST Turbulence Model *Turbulence, Heat and Mass Transfer* **4** pp 625–32