Numerical study of the flow structure in the supersonic inlet-isolator

R K Seleznev 1,2,3
1 Dukhov Research Institute of Automatics (VNIIA), Suschevskaya Street 22, Moscow 127055, Russia
2 Moscow Institute of Physics and Technology (State University), 9 Institutskiy per., Dolgoprudny, Moscow Region, 141701, Russia
3 Ishlinsky Institute for Problems in Mechanics RAS, Vernadsky Street 101(1), Moscow, 119526, Russia
rkseleznev@gmail.com

Abstract. In the paper three-dimensional model for hypersonic flow with turbulence model are described. Comparing the result of calculation with wall pressure experimental data, obtained in the inlet-isolator model mounted to the floor of a Mach 5 wind tunnel at the University of Texas, presented in this article. The influence of flow parameters such as the thickness of the boundary layer and turbulence models on the flow character is investigated in this paper. It was shown that a correct description of turbulence significantly affects the structure of the flow, and its consideration is critically important under comparing the calculation results with the experimental data.

1. Introduction
Development of high-speed aircraft has led to the need to create experimental installations and numerical models [1–5] for studying the process in the ramjet and scramjet. The inlet is an important part of ramjet and scramjet, and their design greatly affects the overall performance of the engine. The isolator is also an essential part of the ramjet and scramjet. The isolator is a constant cross-section passage in order to prevent choking. There are many experiments with inlets [6][7][8–10]. In this paper we consider the model of inlet-isolator mounted to the floor of a Mach 5 wind tunnel at the University of Texas [8,10,11]. In this experiment, it is important to consider the thickness of the boundary layer at the inlet. In this paper, numerical experiments are carried out that take into account the thickness of the boundary layer, as well as experiments with a uniform flow at the inlet. In this paper, we study the influence of various factors on the flow structure. The following factors are chosen as the investigated factors: the dimension of the problem, the wall temperature, the size of the computational grid, and also the influence of the turbulence model. The results of the calculation are compared with the results obtained by other authors [12–16], as well as with the results of the experiment.

2. Description of University of Texas inlet-isolator experimental setup
Wagner [8,10,11] conducted a series of experiments on the study of shock-wave structure in the supersonic inlet-isolator mounted to the floor of a Mach 5 wind tunnel (figure 1). The experimental setup consists of an input ramp with an inclination angle of 6°, and a constant-section isolator...
(figure 2). At the end of the isolator a special flap is installed, raising it can lead to nonstationary formation with subsequent choking of the flow. With the flap is completely lowered, a small part of it protrudes from the bottom wall of the chamber, as shown in figure 2. However, in this paper, as in the works of other authors[9–13], it is considered that the floor is completely smooth. The following incoming air flow parameters were used:

- Incoming flow pressure: \( P = 0.0538 \) atm;
- Incoming flow temperature: \( T = 57.4 \) K;
- Incoming flow Much Number: \( M = 4.9 \);
- Incoming flow gas mixture: Air.

**Figure 1.** Schematic of the model of the inlet/isolator model mounted on the wind tunnel floor [8].

**Figure 2.** Inlet/isolator model mounted on the wind tunnel floor [8].

### 3. Numerical model

For two-dimensional and three-dimensional calculations we used the NERAT-2D and NERAT-3D computer code [4], respectively. NERAT-3D realizes the time-relaxation method[18,19]. At each time step the following groups of governing equations were integrated successively: the Navier–Stokes and continuity equations, the equations of mass conservation of chemical species, the equation of energy conservation. These equations are formulated in the following form:

\[
\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{V}) = 0 ,
\]

(1)

\[
\frac{\partial \rho \mathbf{V}}{\partial t} + \text{div}(\rho \mathbf{V} \mathbf{V} + \mathbf{F}) = 0 ,
\]

(2)
\[
\rho c_p \frac{\partial T}{\partial t} + \rho c_p \nabla T = \text{div}(\lambda \nabla T) + \sum_{i=1}^{N_i} \rho c_{p,i} D_i \left( \nabla Y_i \cdot \nabla T \right) - \sum_{i=1}^{N_i} h_i \dot{w}_i + \frac{\partial p}{\partial t} + \nabla p + \Phi_{\mu}
\]

\[
\frac{\partial \rho}{\partial t} + \text{div} \rho \mathbf{v} = -\text{div} \mathbf{J}_i + \dot{w}_i, \quad i = 1, 2, \ldots, N_s
\]

where: \( t - \text{time}; \ \mathbf{v} = \mathbf{i}u + \mathbf{j}v + \mathbf{k}w \) – velocity vector with projections on the axis of the Cartesian coordinate system \( x, y, z; \ p, \rho \) – pressure and density; \( T \) – temperature of translational movement of particles; \( \mu, \lambda \) – dynamic coefficient of viscosity and coefficient of thermal conductivity; \( c_p \) – specific heat of the mixture at constant pressure; \( c_p = \sum_i Y_i c_{p,i} \); \( N_s \) – number of gas mixture components; \( Y_i \) – mass fraction of the \( i \)-th component of the mixture; \( c_{p,i}, h_i \) – the specific heat at constant pressure associated with translational and rotational degrees of freedom, and the enthalpy of the \( i \)-th component of the mixture; \( \dot{w}_i \) – the mass velocity of chemical transformations for the \( i \)-th component of the mixture; \( D_i \) – the effective diffusion coefficient of the \( i \)-th component of the mixture; \( \mathbf{J}_i \) – the diffusion flux density of the \( i \)-th component; \( \mathbf{J}_i = -\rho D_i \nabla Y_i \); \( N_s \) number of chemical components of the gas mixture. The components of the viscous stress tensor and the dissipative function were calculated by the formulas:

\[
\Pi_{i,j,k} = -p \delta_{i,j,k} + \mu \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] - \frac{2}{3} \delta_{i,j} \frac{\partial u_k}{\partial x_k}, \quad i, j, k = 1, 2, 3
\]

\[
\Phi_{\mu} = \mu \left[ 2 \left( \frac{\partial u}{\partial x} \right)^2 + 2 \left( \frac{\partial v}{\partial y} \right)^2 + 2 \left( \frac{\partial w}{\partial z} \right)^2 + \left( \frac{\partial v}{\partial x} + \frac{\partial w}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial x} + \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right)^2 \right] - \frac{2}{3} \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right)^2
\]

The closing relations for the system of equations to be solved include the thermal equation of state of an ideal gas:

\[
\frac{p}{\rho} = \frac{R_0}{M_\Sigma} T, \quad \frac{1}{M_\Sigma} = \sum_i \frac{Y_i}{M_i}
\]

The system of equations (1)–(5) was integrated numerically by the establishment method using computer code NERAT-3D. Equations (1)–(2) were integrated by an explicit finite-difference method according to the AUSM scheme [20]. The diffusion equations (4) and energy conservation were solved using an implicit finite-difference scheme of the second order of the Crank-Nicholson approximation. Numerical solution of finite-difference equations was carried out using multi-block technology. The number of blocks used in this technology is determined by the complexity of the geometry of the calculation area. In this case, 8 blocks structured of the grid were used. Chemical reactions are neglected.
4. Thickness of boundary layer
In the Wagner experiment observed boundary layer entering the air inlet has a thickness $\delta = 19.3$ mm, so for the correct description of the experiment is necessary to consider this fact. In this works [12–16] the simulation was run until the flow became turbulent and a target thickness for the boundary layer at the exit was reached. However, there is another approach proposed in [14] in which for forming the profile of the boundary layer of a given thickness in the cross-section at the inlet the well-known formulas for the boundary layer on a thick plate used together with the modification from the work [21]. Let $x$ be the longitudinal coordinate, and $y$ - the distance to the plate. We assume that the pressure is constant along the cross-section $x = \text{const}$, the longitudinal component of the velocity has a power-law dependence, the temperature is found according to the formula proposed by Walz [22], where $r$ – is the flux recovery constant is expressed in terms of the wall temperature $T_w$ [21]. Then the density is recalculated from the equation of state, i.e.

$$p(y) = p_\infty,$$

$$u(y,\delta) = u_\infty \left(\frac{y}{\delta}\right)^{\frac{1}{7}},$$

$$T(y,\delta) = T_\infty \left[ r \left(\frac{\gamma - 1}{2}\right) M_\infty^2 \left(1 - \left(\frac{u(y,\delta)}{u_\infty}\right)^2\right) +1 \right],$$

$$r = \frac{T_w - T_\infty}{T_\infty M_\infty^2 (\gamma - 1)},$$

$$\rho(y,\delta) = \frac{p(y)}{RT(y,\delta)}. $$

here $\delta$ – is the given thickness of the boundary layer. It is assumed that the transversal component of the velocity $w$ is zero (that is, an effectively two-dimensional problem is considered). Then, for a boundary layer on a thick plate:

$$\frac{\delta(x)}{x} = 0.382 \text{Re}_x^{\frac{1}{5}}, \quad \text{Re}_x = \frac{\rho_\infty u_\infty x}{\mu_\infty}.$$  

$$\delta(x) = 0.382 \left(\frac{\mu_\infty}{\rho_\infty u_\infty}\right)^\frac{1}{5} x^{\frac{4}{7}},$$

Substituting (12) into (7), we obtain:

$$u(y,x) = u_\infty \left(\sqrt[5]{0.382 \left(\frac{\mu_\infty}{\rho_\infty u_\infty}\right)^\frac{1}{5} x^{\frac{4}{7}}}\right),$$

$$T(y,x) = T_\infty \left[ r \left(\frac{\gamma - 1}{2}\right) M_\infty^2 \left(1 - \sqrt[5]{0.382 \left(\frac{\mu_\infty}{\rho_\infty u_\infty}\right)^\frac{1}{5} x^{\frac{4}{7}}}\right)^2 +1 \right],$$
\[ \rho(y, x) = \frac{p_{\infty}}{RT_{\infty}} \left\{ r^2 \frac{1}{2} M_x^2 \left[ 1 - \left( \frac{y}{0.382} \right)^2 + 0.4 \left( \frac{y}{\rho_{\infty} u_{\infty}} \right)^{\frac{3}{2}} \right] \right\} + 1. \] (15)

The space derivative can be presented in the following form:

\[ \frac{\partial \rho u}{\partial x} \bigg|_{(x,y)} = \frac{\rho(x + h_x, y)u(x + h_x, y) - \rho(x, y)u(x, y)}{h_x}. \] (16)

Then \( v \) can be found by numerical integration from the continuity equation:

\[ \frac{\partial \rho u}{\partial x} \bigg|_{(x,y)} = \frac{\rho(x + h_y, y)v(x, y + h_y) - \rho(x, y)v(x, y)}{h_y}, \] (17)

\[ v(x, y + h_y) = -\frac{h_y}{h_x} \frac{\rho(x + h_y, y)u(x + h_y, y) - \rho(x, y)u(x, y) + \rho(x, y)v(x, y)}{\rho(x + h_y, y)}. \] (18)

5. Turbulence model

In this paper, the algebraic turbulence model of Cebeci–Smith is used [23]. In a two-layer model, the boundary layer is considered to comprise two layers: inner (close to the surface) and outer. The kinematic eddy viscosity is calculated separately for each layer and combined using:

\[ v_i = \min(v_{in}, v_{out}). \] (19)

The inner-region eddy viscosity is given by:

\[ v_{in} = l_m^2 \left[ \left( \frac{\delta u}{\delta y} \right)^2 + \left( \frac{\delta v}{\delta y} \right)^2 \right]^\frac{1}{2}, \] (20)

\[ l_m = ky \left[ 1 - \exp \left( -y' / A^+ \right) \right], \] (21)

where: \( A^+ = 26, \kappa = 0.4, \ y^+ = \frac{y}{\nu} \sqrt{\frac{\rho}{\nu}} = \frac{y}{\nu} \left( \frac{\partial u}{\partial y} \right)_y. \)

The eddy viscosity in the outer region is given by:

\[ v_{out} = \alpha u_{\infty} \delta^* F_{klob}(y, \delta), \] (22)

where: \( \alpha = 0.0168, \ F_{klob}(y, \delta) = \left[ 1 + 5.5 \left( \frac{y}{\delta} \right)^4 \right]^{1/4}. \) \( \delta^* \) - is the displacement thickness, given by

\[ \delta^*_i = \int_y \left( 1 - \frac{u(y)}{u_{\infty}} \right) dy. \] It should be noted that in this experiment \( \delta^*_i = 8.89 \) mm [8,10,11].

6. Results

At the initial stage of calculations two-dimensional laminar code was used. At the entrance to the inlet a uniform flow is set. Figures 3–5 show the results of calculated density, transverse velocity and
pressure distribution within the experimental setup on structured mesh of 800 x 142, respectively. A distinct shock-wave structure is seen that occurs when an incoming supersonic airflow interacts with an inlet. Figure 6 shows the results of calculated pressure distribution on the bottom wall obtained in this work with the results by Lutsky [14] and Zhukov[17]. It can be seen that the flow pattern is described qualitatively. A comparison of the results of the two-dimensional and three-dimensional code was also carried out. The results of such a comparison are shown in the figure 7. For three-dimensional laminar calculation, an eight-block structured grid of dimension 120x160x8 is used in each block. It can be seen that Zhukov's laminar calculations [17] are closer to the results of two-dimensional calculations with the resolution of the first peak of pressure, and to the results of three-dimensional calculations at the resolution of the second peak of pressure. Only the results of the three-dimensional calculation will be presented below.

**Figure 3.** Density distribution within the experimental setup.

**Figure 4.** Transverse velocity distribution within the experimental setup.
However, in the experiment, the input flow was uneven with a thick boundary layer. The incoming boundary layer profiles were extracted from a method, based on well-known formulas for the boundary layer on a thick plate together with a modification from the work [14, 21]. At the first stage, a coarse calculation grid was used (30x40x8). The pressure distribution within the experimental setup is shown in figure 8. Comparison of calculated pressure distribution on the bottom wall and experimental results [8] is shown in figure 9. It should be noted that the calculated pressure amplitude is less than the experimental data. Most likely this is due to the fact that on a coarse grid numerical viscosity plays an essential role. Therefore, in the subsequent calculations we will use in the calculations a detailed grid with the dimension 120x160x8 in each block. Distribution of pressure within the experimental setup. Distribution of pressure within the experimental setup in this case is shown in figure 10. It should be noted that in the case of using this model without using the turbulence model, a vortex zone (figure 11) appears before the first shock wave, which changes the flow in the channel. The change in the structure of the flow in the channel is well evidenced by the distribution of

Figure 5. Pressure distribution within the experimental setup.

Figure 6. Comparison of calculated pressure distribution on the bottom wall obtained in this work (line) and the results obtained by Lutsky [14] (blue squares) and Zhukov [17] (green tangles).
pressure on the bottom wall, which is shown in figure 12. Therefore, in subsequent calculations, the Cebeci–Smith turbulence model will be used. In this case, the vortex before the first shock wave disappears, as demonstrated in figure 13. It may be noted that in this case, the bottom wall of the pressure increases, which is demonstrated in figure 14. This fact can be explained by the increased viscosity (the distribution of turbulent viscosity in this case is shown in figure 15). Increased viscosity slows down the main air flow, as shown in figure 16. In this paper, we also investigated the influence of the wall temperature on the flow structure. The calculation was carried out at a wall temperature equal to room temperature (Tw=300K) and equal to the flow temperature (Tw=57K). A comparison of these two cases is shown in figure 14. It can be seen that small differences appear in the second peak of pressure.

![Figure 7](image1.png)

**Figure 7.** Comparison of calculated pressure distribution on the bottom wall obtained in this work by 2D (blue dots), 3D (line) and the results obtained by Zhukov [17] (green tangles).

![Figure 8](image2.png)

**Figure 8.** Pressure distribution within the experimental setup on coarse calculation grid.
Pressure, erg/cm³

Figure 9. Comparison of calculated pressure distribution on the bottom wall obtained in this work (red line) on coarse calculation grid and experimental results [8].

Coordinate, cm

Figure 10. Pressure distribution within the experimental setup on a detailed grid without a turbulence model.
Figure 11. Vortex generation within the experimental setup on a detailed grid without a turbulence model.

Figure 12. Comparison of calculated pressure distribution on the bottom wall obtained in this work on a detailed grid without a turbulence model. (red line) and experimental results [8].

Also a comparison with the results obtained by other authors was carried out. Figure 17 shows a comparison of the pressure distribution on the chamber wall obtained in this work (black line) with the experimental results [8,10] (green squares), and the results obtained by Lutsky [14] (red squares) and Koo[13,15] (blue delta), Edwards[16] and Jang [12].
Figure 13. Pressure distribution within the experimental setup on a detailed grid using the turbulence model.

Pressure, erg/cm$^3$

Figure 14. Comparison of calculated pressure distribution on the bottom wall obtained in this work at a wall temperature equal to room temperature (Tw=300K) and equal to the flow temperature (Tw=57K) with experimental results [8]
Figure 15. Turbulent viscosity distribution within the experimental setup.

Figure 16. Longitudinal speed distribution within the experimental setup.

7. Conclusion
Results of the pressure distribution calculated by two-dimensional and three-dimensional model is presented. The present results show that two-dimensional simulations are able to predict the shock wave structures in a scramjet inlet/isolator. The influence of flow parameters such as the thickness of the boundary layer and turbulence models on the flow character is investigated in this paper. It was shown that a correct description of turbulence significantly affects the structure of the flow, and its consideration is critically important under comparing the calculation results with the experimental data.
Figure 17. Comparison of calculated pressure distribution on the bottom wall obtained in this work (red line) and the results by Lutsky [14], Koo [13,15], Edwards [16] and Jang [12] with experimental results [8].

Acknowledgments
This work was preformed within the framework of the Government program of basic research of the Russian academy of sciences (contract #AAAA-A17-117021310372-6) and partially under support of RFBR grant #016-01-00379.

References
[1] Seleznov R and Surzhikov S 2015 Quasi-One-Dimensional and Two-Dimensional Numerical Simulation of Scramjet Combustors 51st AIAA/SAE/ASEE Joint Propulsion Conference (Reston, Virginia: American Institute of Aeronautics and Astronautics) p 28
[2] Seleznov R and Surzhikov S 2016 A Quasi-One-Dimensional Analysis of Hydrogen-Fueled Scramjet Combustors 52nd AIAA/SAE/ASEE Joint Propulsion Conference (Reston, Virginia: American Institute of Aeronautics and Astronautics) pp 1–27
[3] Seleznov R K 2016 Comparison of two-dimensional and quasi-one-dimensional scramjet models by the example of VAG experiment J. Phys. Conf. Ser. 755 11001
[4] Surzhikov S, Seleznov R, Tretjakov P and Zabaykin V 2014 Unsteady Thermo-Gasdynamic Processes in Scramjet Combustion Chamber with Periodical Input of Cold Air 50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference (Reston, Virginia: American Institute of Aeronautics and Astronautics) p 25
[5] Seleznov R and Surzhikov S 2013 Generalized Newton Method For Solving Differential Equations of Chemical Kinetics 44th AIAA Thermophysics Conference (Reston, Virginia: American Institute of Aeronautics and Astronautics) pp 1–17
[6] Xing F, Zhang S and Yao Y 2012 Numerical Simulation of Shock-Induced-Combustion in Three-Dimensional HyShot Scramjet Model 50th AIAA Aerosp. Sci. Meet. Incl. New Horizons Forum Aerosp. Expo. 1–12
[7] Boon S and Hillier R 2006 Hypersonic Inlet Flow Analysis at Mach 5, 6 and 7 44th AIAA Aerospace Sciences Meeting and Exhibit (Reston, Virginia: American Institute of Aeronautics and Astronautics) pp 1–13
[8] Wagner J L 2009 Experimental Studies of Unstart Dynamics in Inlet / Isolator Configurations in a Mach 5 Flow
[9] Gamba M, Miller V, Mungal M G and Hanson R 2011 Ignition and Flame Structure in a Compact Inlet/Scramjet Combustor Model 17th AIAA Int. Sp. Planes Hypersonic Syst.
Technol. Conf.

[10] Wagner J L, Yuceil K B, Valdivia A, Clemens N T and Dolling D S 2009 Experimental Investigation of Unstart in an Inlet/Isolator Model in Mach 5 Flow AIAA J. 47 1528–42

[11] Wagner J L, Yuceil K B and Clemens N T 2010 Velocimetry Measurements of Unstart of an Inlet-Isolator Model in Mach 5 Flow AIAA J. 48 1875–88

[12] Jang I, Pečnik R and Moin D P 2010 A numerical study of the unstart event in an inlet/isolator model Cent. Turbul. Res. Annu. Res. Briefs 93–103

[13] Koo H and Raman V 2012 Large-Eddy Simulation of a Supersonic Inlet-Isolator AIAA J. 50 1596–613

[14] Borisov V E, Kudryashov I Y and Lutsky A E 2016 Numerical simulation of the pseudo-shock region formation in the channel Keldysh Inst. Prepr. 2898 1–24

[15] Koo H 2010 Large-Eddy Simulations of Scramjet Engines Doctoral dissertation

[16] Boles J, Edwards J, Choi J-I and Baurle R 2009 Simulations of High-Speed Internal Flows Using LES/RANS Models 47th AIAA Aerosp. Sci. Meet. Incl. New Horizons Forum Aerosp. Expo. 1–22

[17] Zhukov V T, Manukovskii K V., Novikova N D, Rykov Y G and Feodoritova O B 2015 Numerical simulation of the pseudo-shock region formation in the channel

[18] Shang J and Surzhikov S 2013 Numerical Prediction of Convective and Radiative Heating of Scramjet Combustion Chamber with Hydrocarbon Fuels 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition (Reston, Virginia: American Institute of Aeronautics and Astronautics) pp 1–16

[19] Shang J and Surzhikov S 2013 Radiative Heat Exchange in a Hydrogen-Fueled Scramjet Combustion Chambers 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition (Reston, Virginia: American Institute of Aeronautics and Astronautics)

[20] Edwards J, Liou M-S, Edwards J and Liou M-S 1997 Low-diffusion flux-splitting methods for flows at all speeds 13th Computational Fluid Dynamics Conference vol 36(Reston, Virginia: American Institute of Aeronautics and Astronautics)

[21] Xu S and Martin M P 2004 Assessment of inflow boundary conditions for compressible turbulent boundary layers Phys. Fluids 16 2623–39

[22] Walz A 1959 Compressible Turbulent Boundary Layers With Heat Transfer and Pressure Gradient In Flow Journal of Research of the National Bureau of Standards-B. Mathematics and Mathematical Physics 63 53–72

[23] Smith A. M. and Cebeci T. 1967 Numerical solution of the turbulent boundary layer equations (Long Beach, Calif.: Douglas Aircraft Co., Douglas Aircraft Division)