Numerical study of the flow structure in the supersonic inlet-isolator with mechanical throttle

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 Institutsky 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 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. A study is being conducted on the phenomenon of unstart of air inlet-isolator. The results of the distribution of pressure, the Mach number and the temperature inside the inlet-isolator at successive points in time presented.

1. Introduction

Recently, of work on the creation of a scramjet is actively underway in the world [1–4]. An essential part of such an engine is an inlet-isolator. In the inlet-isolator, the kinetic energy of the flow is converted into static pressure. The inlet-isolator performs the same function in the scramjet as the compressors in gas-turbine engines. This is a carefully designed and manufactured design, the performance of which determines the reliability of the scramjet engine and the achievement of the required characteristics in all operational flight modes. High speeds greatly increase the cost of conducting experiments, which leads to the need to create computational codes that describe these phenomena. Since the structure of the flow in the scramjet is very complex [5] and sensitive to the inflow parameters, then in order to increase the accuracy of predictive flow simulation into the scramjet, it is necessary to calculate the full layout of the scramjet (hypersonic aircraft [6], inlet-isolator [7–15], combustor [16–21], nozzle [22]). In this paper, the inlet-isolator is studied in detail. Disturbances in the system, such as those caused by fuel combustion, can lead to such a phenomenon as unstart. Study of this phenomenon are devoted to many experimental and theoretical works. In this paper, as a result of numerical simulation, we study the flow structure in an experiment at the University of Texas with a supersonic air inlet with a raised throttle. The calculation results were compared with the results of the experiment and the results of other authors.

2. Description of University of Texas inlet-isolator experimental setup

Wagner et al. [23–25] had conducted a series of experiments on shock-wave structure in the supersonic inlet-isolator. Which was mounted on the floor of a Mach 5 wind tunnel. Experimental
facility shown in figure 1 (all the dimensions in millimeters). It should be taken into account for further numerical study of these experimental data that input conditions for gas flow in the input cross-section were non-uniform. Input experimental setup consisted of an inlet compression ramp with an inclination angle of 6°, and connected to a constant-section isolator. At the exit of the isolator a special flap with angle of deviation of 28º was installed.

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

![Figure 1](image)

**Figure 1.** Schematic of the model of the inlet/isolator model mounted on the wind tunnel floor [23].

### 3. Numerical model

As well as in the works modified three-dimensional NERAT-3D computer code [20] used. NERAT–3D realizes the time-relaxation method [26, 27]. 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:

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

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

\[
\rho c_p \frac{\partial T}{\partial t} + \rho c_p \mathbf{V} \text{grad} T = \text{div}(\lambda \text{grad} T) + \sum_{i=1}^{N_s} \rho c_i D_i (\text{grad} Y_i \cdot \text{grad} T) - \sum_{i=1}^{N_s} \hat{h_i} \hat{w_i} + \frac{\partial \rho_p}{\partial t} + \mathbf{V} \text{grad} p + \Phi_{\mu},
\]

\[
\frac{\partial \rho_p}{\partial t} + \text{div} \rho_p \mathbf{V} = -\text{div} \mathbf{J}_i + \hat{w_i}, \quad i = 1, 2, \ldots, N_s
\]

where: \( t \) – time; \( \mathbf{V} = iu + jv + kw \) – 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; \( c_p = \sum_{i=1}^{N_s} Y_i c_{p,i} \); \( N_s \) – number of gas mixture components; \( Y_i \) – mass fraction of the i-th component of the mixture; \( c_{p,i}, \hat{h}_i \) – 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; \( J_i \) – the diffusion flux density of the i-th component; \( J_i = -\rho D_i \text{ grad} Y_i \); \( N_i \) number of chemical components of the gas mixture. The components of the viscous stress tensor and the dissipative function were calculated by the formulas:

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

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

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

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

(5)

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 [28]. 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, eight blocks structured of the grid were used. Dimension of the computational grid in each block was 40x60x8. 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 [29–33] 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 [31] 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 [34]. 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 [35], where \( r \) – is the flux recovery constant is expressed in terms of the wall temperature \( T_w \) [34]. Then the density is recalculated from the equation of state, i.e.

\[
p(y) = p_v, \quad u(y, \delta) = u_v \left( \frac{y}{\delta} \right)^{\frac{1}{2}},
\]  

(6)  

(7)
\[
T(y, \delta) = T_\infty \left[ r \left( \frac{\gamma - 1}{2} M_\infty^2 \left( 1 - \left( \frac{u(y, \delta)}{u_\infty} \right)^2 \right) + 1 \right) \right],
\]

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

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

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:

\[
\frac{\delta(x)}{x} = 0.382 \text{Re}^{\frac{1}{5}}, \quad \text{Re}_x = \frac{\rho_x u_x x}{\mu_x}.
\]

\[
\delta(x) = 0.382 \left( \frac{\mu_x}{\rho_x u_x} \right)^{\frac{1}{5}} x^\frac{4}{5},
\]

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

\[
u(x, y) = u_\infty \left\{ y \left[ 0.382 \left( \frac{\mu_x}{\rho_x u_x} \right)^{\frac{1}{5}} x^\frac{4}{5} \right] \right\}^{\frac{1}{7}},
\]

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

\[
\rho(y, x) = \frac{p_\infty}{RT_\infty} \left[ r \left( \frac{\gamma - 1}{2} M_\infty^2 \left( 1 - \left( \frac{u(y, \delta)}{u_\infty} \right)^2 \right) + 1 \right) \right].
\]

The space derivative can be presented in the following form:

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

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

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

\[
v(x, y + h_y) = -\frac{h_y}{h_x} \frac{\rho(x + h_x, y)u(x + h_x, y) - \rho(x, y)u(x, y)}{\rho(x, y + h_y)} - \frac{\rho(x, y) v(x, y)}{\rho(x, y + h_y)}.\]
5. Turbulence model
In this paper, the algebraic turbulence model of Cebeci–Smith is used [36]. 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:

\[ \nu_i = \min (\nu_{in}, \nu_{out}) \]  

(19)

The inner-region eddy viscosity is given by:

\[ \nu_{in} = l_m^2 \left[ \left( \frac{\partial u}{\partial y} \right)^2 + \left( \frac{\partial v}{\partial x} \right)^2 \right]^{1/2}, \]

(20)

\[ l_m = ky \left[ 1 - \exp \left( -\frac{y^+}{A^+} \right) \right], \]

(21)

where: \( A^+ = 26, k = 0.4, y^+ = \frac{y}{v} \sqrt{\frac{\tau_w}{\rho}} = \frac{y}{v} \sqrt{\frac{\partial u}{\partial y}}, \)

The eddy viscosity in the outer region is given by:

\[ \nu_{out} = \alpha u_e \delta^* F_{Kleb} (y, \delta), \]

(22)

where: \( \alpha = 0.0168, F_{Kleb} (y, \delta) = \left[ 1 + 5.5 \left( \frac{y}{\delta} \right)^6 \right]^{-1}. \delta^* - \) is the displacement thickness, given by \( \delta^* = \int_0^\delta \left( 1 - \frac{u(y)}{u_\infty} \right) dy \). It should be noted that in this experiment \( \delta^* = 8.89 \text{ mm} \) [23–25].

6. Results
This paper presents the results of a time history of pressure, Mach number and temperature contours for the unstart event due to raised throttle. Figures 2–7, Figures 8–13 and figures 14–19 show the results of calculated pressure, Mach number and temperature, respectively, at successive times (\( t = 4.47 \text{ ms}, 8.36 \text{ ms}; 12.24 \text{ ms}; 16.13 \text{ ms}; 20 \text{ ms}; 23.9 \text{ ms}. \)). At the initial stage, a shock wave arises, which leads to the separation of the boundary layer from the throttle as shown in figures 2,8,14. This recirculation area is lengthened, causing the formation of a shock system that strikes the upper surface, which effectively reduces the area occupied by the supersonic core and leads to the formation of additional compression waves.

![Figure 2. Pressure distribution within the experimental setup t = 4.47 ms.](image)
As a result, the gas flow in the lower half of the isolator becomes subsonic, and the flow pattern stops changing from approximately 18 ms. Comparison of calculated pressure distribution on the bottom wall at that time and experimental results [23] is shown in figure 20 ($P_{inf}$=0.0538 atm is Incoming flow pressure). The experimental results [23], as well as the results of other authors [29–31, 33, 37], qualitatively similar the flow pattern obtained in these calculations.

Figure 3. Pressure distribution within the experimental setup $t = 8.36$ ms.

Figure 4. Pressure distribution within the experimental setup $t = 12.24$ ms.

Figure 5. Pressure distribution within the experimental setup $t = 16.13$ ms.
Figure 6. Pressure distribution within the experimental setup $t = 20$ ms.

Figure 7. Pressure distribution within the experimental setup $t = 23.9$ ms.

Figure 8. Mach distribution within the experimental setup $t = 4.47$ ms.

Figure 9. Mach distribution within the experimental setup $t = 8.36$ ms.
Figure 10. Mach distribution within the experimental setup $t = 12.24$ ms.

Figure 11. Mach distribution within the experimental setup $t = 16.13$ ms.

Figure 12. Mach distribution within the experimental setup $t = 20$ ms.

Figure 13. Mach distribution within the experimental setup $t = 23.9$ ms.
Figure 14. Temperature distribution within the experimental setup $t = 4.47$ ms.

Figure 15. Temperature distribution within the experimental setup $t = 8.36$ ms.

Figure 16. Temperature distribution within the experimental setup $t = 12.24$ ms.

Figure 17. Temperature distribution within the experimental setup $t = 16.13$ ms.
Figure 18. Temperature distribution within the experimental setup $t = 20$ ms.

Figure 19. Temperature distribution within the experimental setup $t = 23.9$ ms.

Figure 20. Comparison of calculated pressure ($t = 23.9$ ms) distribution on the bottom wall obtained in this work (red line) and experimental results [23].
7. Conclusion
The paper presents the results of 3D simulations of an air inlet-isolator with a raised throttle. The unstart process of the inlet-isolator demonstrated in the calculations. The resulting flow structure is qualitatively similar to the one observed in the experiment. A comparison is made of the pressure distribution on the bottom wall at the time of the unstart of inlet-isolator in comparison with the experimental results. The use of more detailed computational grids and more accurate models of turbulence will allow obtain a better match with the experimental results. This may constitute the object of future studies.

References
[1] Heiser W, Pratt D, Daley D and Mehta U 1994 Hypersonic Airbreathing Propulsion vol 26 (Washington, DC: American Institute of Aeronautics and Astronautics, Inc.)
[2] Curran E T, Heiser W H and Pratt D T 1996 Fluid Phenomena in Scramjet Combustion Systems Annual Review of Fluid Mechanics 28 323 – 60
[3] Curran E T 2001 Scramjet Engines: The First Forty Years J. of Propulsion and Power 17 1138 – 48
[4] Bose T 2012 Airbreathing Propulsion (New York, NY: Springer New York)
[5] Seleznev R K 2018 History of scramjet propulsion development J. of Phys.: Conf. Series 1009 012028
[6] Im S and Do H 2018 Unstart phenomena induced by flow choking in scramjet inlet-isolators Progress in Aerospace Sciences 97 1 – 21
[7] Veeran S, Pestyridis A and Ganippa L 2018 Ramjet Compression System for a Hypersonic Air Transportation Vehicle Combined Cycle Engine Energies 11 2558
[8] Kotov M A 2017 Experimental and numerical study of supersonic flow over two blunted wedges J. of Phys.: Conf. Series 815 012025
[9] Xu K, Chang J, Li N, Zhou W and Yu D 2018 Preliminary investigation of limits of shock train jumps in a hypersonic inlet-isolator European J. of Mechanics - B/Fluids 72 664 –75
[10] Varshney M, Ansari M D, Baig M F and Sharma I 2018 Control of Unstart Phenomenon in Scramjet Engines 2018 Joint Propulsion Conf: 1–11
[11] Urzay J 2018 Supersonic Combustion in Air-Breathing Propulsion Systems for Hypersonic Flight Annu. Rev. Fluid Mech 50 593 – 627
[12] Im S kyun and Do H 2018 Unstart phenomena induced by flow choking in scramjet inlet-isolators Progress in Aerospace Sciences 97 35 – 60
[13] Seleznev R K 2018 Numerical study of the flow structure in the supersonic inlet-isolator J. of Phys.: Conf. Series 1009 012034
[14] Seleznev R K 2018 Validation of two-dimensional model by the example of a supersonic inlet-isolator J. of Phys.: Conf. Series 1009 012030
[15] Seleznev R K 2018 Validation of 3D model by the example of a supersonic inlet-isolator J. of Phys.: Conf. Series 1009 012031
[16] Cheng F, Tang S, Zhang D and Li Y 2017 Quasi-One-Dimensional Modeling and Analysis of RBCC Dual-Mode Scramjet Engine Int. J. of Turbo & Jet-Engines
[17] Seleznev R K 2016 Comparison of two-dimensional and quasi-one-dimensional scramjet models by the example of VAG experiment J. of Phys.: Conf. Series 755 011001
[18] Seleznev R and Surzhikov S 2016 A Quasi-One-Dimensional Analysis of Hydrogen-Fueled Scramjet Combustors 52nd AIAA/SAE/ASEE Joint Propulsion Conf. (Reston, Virginia: American Institute of Aeronautics and Astronautics) pp 1–27
[19] Seleznev R and Surzhikov S 2015 Quasi-One-Dimensional and Two-Dimensional Numerical Simulation of Scramjet Combustors 51st AIAA/SAE/ASEE Joint Propulsion Conf. (Reston, Virginia: American Institute of Aeronautics and Astronautics) p 28
[20] Surzhikov S, Seleznev R, Tretjakov P and Zabaykin V 2014 Unsteady Thermo-Gasdynamic Processes in Scramjet Combustion Chamber with Periodical Input of Cold Air 50th
[21] Seleznev 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 25

[22] Storozhev D 2018 Numerical simulation of the underexpanded plume spectral radiance using Monte-Carlo method J. of Phys.: Conf. Series 1009

[23] Wagner J L 2009 Experimental Studies of Unstart Dynamics in Inlet / Isolator Configurations in a Mach 5 Flow Dissertation The University of Texas at Austin

[24] 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

[25] 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

[26] 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

[27] 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)

[28] 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 Conf. vol 36 (Reston, Virginia: American Institute of Aeronautics and Astronautics)

[29] Jang I, Pečnik R and Moin D P 2010 A numerical study of the unstart event in an inlet/isolator model Center for Turbulence Research Annual Research Briefs 93 –103

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

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

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

[33] Boles J, Edwards J, Choi J-J and Baurle R 2009 Simulations of High-Speed Internal Flows Using LES/RANS Models 47th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition (Reston, Virginia: American Institute of Aeronautics and Astronautics) pp 1–22

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

[35] Walz A 1959 Compressible Turbulent Boundary Layers With Heat Transfer and Pressure Gradient In Flow 63 53 –72

[36] Smith A. M. and Cebeci T 1967 Numerical solution of the turbulent boundary layer equations Douglas Aircraft Co Long Beach Ca Aircraft Div №. DAC--33735

[37] 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 Keldysh Institute Preprints 5 1 – 23