ECCOMAS Congress 2016
VII European Congress on Computational Methods in Applied Sciences and Engineering
M. Papadrakakis, V. Papadopoulos, G. Stefanou, V. Plevris (eds.)
Crete Island, Greece, 5–10 June 2016

RESEARCH OF UNSTEADY FLOW REGIMES IN CHANNEL OF HYPERSONIC INLET

Natalia N Fedorova[1, 2], **and Marat A. Goldfeld**[1]

[1] hristianovich Institute of Theoretical and Applied Mechanics SB RAS
4/1 Institutskaya, Novosibirsk, Russia,

[2]Novosibirsk State University of Architecture and Civil Engineering (Sibstrin)
113 Leningradskaya st., Novosibirsk, Russia
{nfed, gold}@itam.nsc.ru.

Keywords: Supersonic Flow, Channel, Numerical Simulation, Experimental Investigation, Wind Tunnel, Sock Wave. Boundary Layer Separation

Abstract. *In the paper, the numerical study of the flows in channels of variable cross section has been performed under the conditions of experiments carried out in blow down wind tunnel. The main goals of the present investigations were as follows: to get a new data on the flows in inlet channel at M=2-8 under conditions which close to start-unstart of inlet; to investigate the influence of the channel geometry, Mach and Reynolds number on the flow structure and inlet characteristics; to study the influence boundary layers on the channel flow structure and the inlet performances. The internal part of the channel has complicated configuration can be regulated over the length. The experimental investigations of the model have been carried out in the blow-down wind tunnel T-313 at Mach numbers from 2 to 6 and in the hot-shot wind tunnel IT-302M at Mach numbers from 5 to 8 in Reynolds number range from 8 to* $56 \cdot 10^6$ *1/m. The comparison and joint analysis has been performed for the calculated and experimental data. The gas-dynamic patterns have been constructed for the complicated flows with multiplex interactions of the shock waves with the boundary layers developing on the compression surfaces. The parametric computations performed within the wide range of flow and geometric parameters allowed to carry out the experiments and provide a basis for explanation of flow features and the choice of optimum configurations.*

1 INTRODUCTION

Hypersonic airbreathing propulsion offers great potential for reliable and economical access to space as well as atmospheric flight. In particular, scramjets (supersonic combustion ramjets) are a promising technology that can enable efficient and flexible transport systems by removing the need to carry oxidizers and other propulsion limitations of conventional rocket engines. Supersonic combustion via the use of scramjet technology was successfully demonstrated for the first time worldwide by the HyShot II Program 2002 [1], followed by successful flights of the Hyper-X vehicles in 2004 at Mach number 6.8 and Mach number 9.6 [2].

One of basic elements of the scramjet is air inlet. Its influence on effectiveness of propulsion is dominant, especially at application of integral construction configurations. Feature of air inlet of the scramjet consists in fact that it should ensure operation of the engine in is unprecedented a wide range of flight conditions at high efficiency of all elements of the engine for reception of the maximum thrust. From this point of view, high level of mass flow rate and minimum total pressure losses must be ensured in a wide range of a flight velocities and altitudes [3, 4]. A lot of key issues have to be addressed for developing the dual mode ramjet technology. One of them consists in designing an inlet able to operate in a very large Mach number range, with enough pressure recovery while providing the necessary air mass-flow to the combustion chamber. The use of a sophisticated variable geometry simplifies the aerodynamic design of the inlet. But, in the same time, the mechanical feasibility of the inlet is more and more difficult task for implementation because of the very severe environment the inlet will encounter during the flight. At the contrary, a fixed geometry inlet will be much more feasible from the mechanical point of view but it will lead to the very difficult compromises at the choice of the design Mach number. At the too large contraction ratio, a very large spillage will reduce the installed thrust at low Mach number. If the contraction ratio will be too restricted, the general performances will be dramatically reduced at high Mach number [5].

Several studies have been carried out on optimizing inlet performance using internal inlet components. Amongst other parameters, inlet performance also depends on upstream flow. Different research groups have investigated the effect of major component properties on inlet characteristics, e.g., the modification of the inlet internal geometry [6] and configurations of side walls [7]. Variations of entrance configuration and inlet position reveal different effects on performances [8] and studies at the position of the inlet diffuser were carried out to avoid flow separation [9, 10]. Other studies included unstart criteria and inlet buzzing [11, 12], which must be prevented [13] to ensure the functionality of the ramjet and prevent destruction of the structure of inflowing and the inlet unstart [14, 15]. One technique is to control the inlet normal shock by adjusting internal parts of the inlet [16, 17]. All examples deal with inlet design and a given unchangeable main engine design [18, 19]. On the basis of still open issues from these studies, the influence of the upstream flow on inlet performance was identified as one of the key issues.

Reliable in-flight starting of the inlet is of critical importance for the successful operation of scramjet engines, particularly integral configurations with high-contraction inlets [20]. The present research is undertaken to examine the capabilities of various inlet starting methods based on two principles: boundary layer effects and variable geometries. If the overall Mach number range is limited (Mach 4-Mach 10), a completely fixed geometry can be defined for the scramjet inlet with relatively good possible performance in the whole flight envelope. At the contrary, an extension of the Mach number range needs to provide some variation of the geometry of the inlet channel in order to obtain acceptable performances. Indeed for higher Mach numbers, it will be necessary to use a high contraction ratio for the inlet in order to limit the supersonic Mach number at the entrance of the combustion chamber and to reduce diverging combustion chamber, as Mach number will increase. At the contrary, in order to extend the flight range to lower Mach numbers, the inlet contraction ratio must be reduced (with or without reduction of the capture area) to avoid a too large cowl spillage and the corresponding additive drag section of the combustion chamber must be open.

Therefore it is important to ensure the necessary mass flow rate and pressure recovery by means of adjustment of geometry and control of boundary layer bleed. Investigations of performances of adjustment inlet with or without boundary layer bleed were presented in works [21, 22].

Nevertheless these researches were limited to a narrow range of speeds of flight (Mach numbers) and did not assume an adjustment of cross-section of inlet channel depending on flight conditions. The computer design tools are widely used in developing the optimum geometry of the engines and their elements (inlet, combustion chamber, and nozzle) [23]. At the same time numerical simulation widely employs for determination of aerodynamic scramjet characteristics (thrust and drag). The flow simulation of high-speed inlets is one of important directions in these researches [24-26].

In the paper, the numerical study of the flows in channels of variable cross section has been performed under the conditions of experiments in wind tunnels. The comparison and joint analysis has been performed for the calculated and experimental data. The gas-dynamic analysis has been carried out for the complicated flows with multiple interactions of the shock waves with the boundary layers developing on the inlet surfaces.

The main goals of the present investigations were as follows:

- to get a new data on the flows in channel at M=2-8;
- to investigate the influence of the channel geometry, Mach and Reynolds number on the flow structure and inlet characteristics
- to verify the algorithm and code on the channel flow experimental data;
- to study the influence of the boundary layer before channel entrance on the channel flow structure and the inlet characteristics.

2 MODEL AND FACILITIES

The experimental model presents a channel with flat walls and 3D central body [4, 7]. The quasi-two-dimension configuration of the central body (nose part) allows one to simulate a boundary layer which is developed on the aircraft nose part and guarantees preliminary flow deceleration in two shock waves. The internal part of the channel has a complicated configuration with the adjusted area over the channel length. The variation of inlet geometry was performed by means of motion of the cowl. Several model configurations with different cowl position at different relative throat area have been considered (Fig. 1) for the corresponding Mach numbers to provide the inlet “start” at different test conditions.

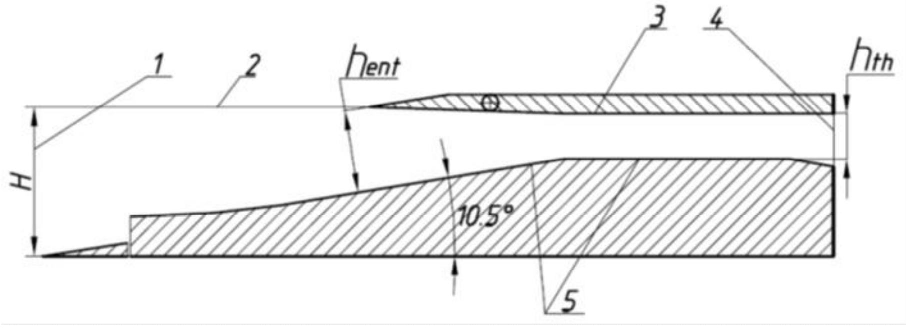

Figure 1: Scheme of model and calculation domain

Change of the channel configuration was accompanied by change of value of internal compression, which decreased at reduction of Mach number to provide inlet start in all range of Mach numbers. Change of internal contraction (relative height of a throat) is presented in Table 1 together with total contraction of captured air stream.

M	2	3	4	5	5.5	6	7	7.5	8
H/h_{th}	3.0	3.36	4.0	4.5	4.8	5.3	5.66	6.21	7.06
h_{ent}/h_{th}	0.96	1.14	1.44	1.68	1.82	2.06	2.23	2.51	2.91

Table 1: Channel configurations

The bottom and top walls of the model were equipped with static pressure taps to measure the distribution of static pressure along the longitudinal axis. Additionally, in three cross-sections the installation of Pitot pressure probes has been provided to determine the local Mach numbers and total pressure recovery in the channel. The side walls of model and wind tunnels facility was equipped with transparent windows to provide Schlieren visualization of the internal flow structure. Glass windows are placed along the inlet throat of the model to visualize the flow to determine its structure (separation and attachment of the boundary layer, location and form of shock waves). During test, shadow, oil and sparkle visualizations were performed to analyze the flow structure at the entrance and in the duct of the inlet.

The experimental investigations of the model have been carried out in the blow-down wind tunnel T-313 at Mach numbers from 2 to 6 and in the hot-shot wind tunnel IT-302 at Mach numbers from 5 to 8. The blow-down wind tunnel T313 has been equipped profiled nozzles for the Mach number range from 2 to 6 and has test duration up to 600 s. Working section of wind-tunnel has a rectangular cross-section and located in a pressure chamber. The hot-shot wind tunnel IT 302M has been equipped profiled nozzles with exit diameter of 400 mm for the Mach number range from 5 to 8. Test duration can change from 100 to 200 ms depending on Mach number. Typical flow characteristics for both wind tunnels are presented in Table 2.

	Blow-down wind-tunnel T-313					Hot-shot wind-tunnel IT-302M (time range 10 - 70 ms)		
M	2	3	4	5	6	6	7	8
P_t, bar	2.1	4.2	10.5	8.0	8.8	52÷31	330÷56	340÷124
T_t, K	288	280	2760	374	540	1780÷1500	2200÷1340	2000÷1600
$Re*10^6/m$	24.8	36.2	54.4	16.9	9.4	16.8 ÷ 10.4	22.9÷6.8	14.9÷7.6

Table 2: Condition of the experiments in T-313 and in IT-302.

Peculiarity of IT-302M wind tunnel is decrease of flow parameter during operation time. That is why flow parameters for this tunnel are shown for typical duration of wind tunnel operation (from 10th ms to 70th ms) when regime of wind tunnel remains quasi-stationary.

3 MATHEMATICAL MODEL AND CALCULATION METHOD

The calculations have been performed on the basis of the full transient Favre-averaged Navier-Stokes equations and Wilcox two-equation turbulence model [27]. The original numerical algorithm [28] used earlier for the 2-D turbulent supersonic flow computations and showed a good potential to predict the properties of these flows. The same methodical approach was extended to a new class of internal supersonic flows. The temporal approximation presents a four-step finite-difference scheme of splitting according spatial variables. At each fractional step the finite-difference scheme was realized by scalar sweeps. The TVD-scheme of Flux Vector Splitting by van Leer of the third order of accuracy has been used for the approximation of convective terms. The viscous terms have been approximated with the central finite-difference relations of second order of accuracy.

The computation region was bounded by the bottom wall (central body) 5 from below and by upper wall 3 or free surface 2 from above. On the left and on the right the computational domain was bounded with input and output sections 1 and 4, respectively. In the input section 1, the free flow conditions or the profiles of all gas-dynamic and turbulent parameters obtained from the computation of the turbulent boundary layer on the plate have been assigned. The coincidence with experimental boundary layer integral parameters and skin friction coefficient has been provided in computations if such data were obtained in the experiments. The no-slip conditions for velocity have been specified on boundaries 4 and 5. Adiabatic condition for temperature was used in the computation for M=2 ÷6 corresponding blow-down wind tunnel and constant temperature ("cold wall") was specified for M=6 ÷8 computations in conditions of hot-shot wind tunnel. On the top boundary 2, so-called "simple wave" conditions were assigned in order to provide all the disturbances to way out from the computation region. The "soft" boundary conditions have been set on boundary 5. Special study was performed for determination of the temperature factor influence on the flow structure.

4 INLET FLOW SIMULATION

The computations were performed for the inlet configurations shown in Figure 1 under the conditions of experiments described in Table 1.

Fig. 2 presents the static pressure distributions along the central body and the cowl obtained in computations for Mach 2 to 6 carried out under the conditions in blow down wind tunnel. Data was made dimensionless by the value of pressure in the entrance section of the calculation region. The data obtained indicate a big lengthwise pressure non-uniformity and high level of pressure, which increase together with Mach number. At Mach numbers $M \geq 4$, a saw-toothed structure appears in the pressure distribution related to the alternation of shock waves and rarefaction waves inside the channel

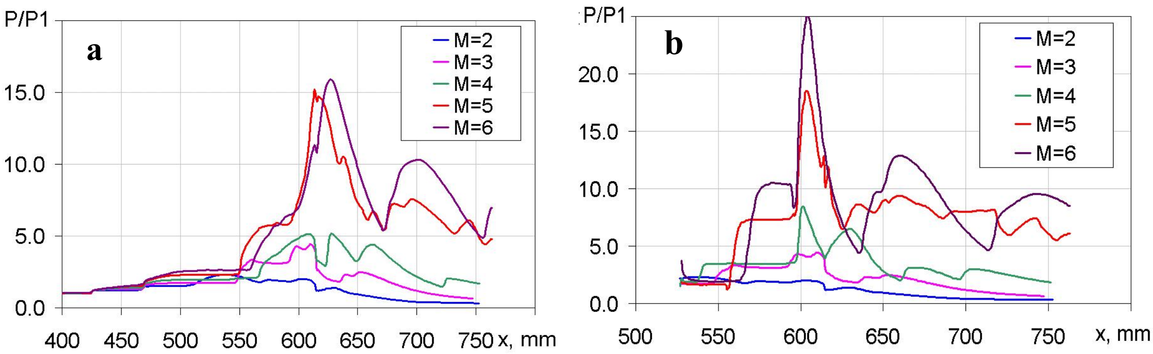

Figure 2: Computed pressure along central body (a) and cowl (b)

The computed distributions of relative total pressure $P0/P0_\infty$ are presented in Figure 3. Here pressure was averaged over the channel width along the computation grid lines. These data allow one to determine the total pressure losses in any cross section which is a good complement to the direct measurements in some assigned cross-sections. The joint analysis of the calculated and experimental results has permitted one to conclude that investigated model configuration are satisfactory with respect to the total pressure recovery coefficient within the whole given range of Mach numbers.

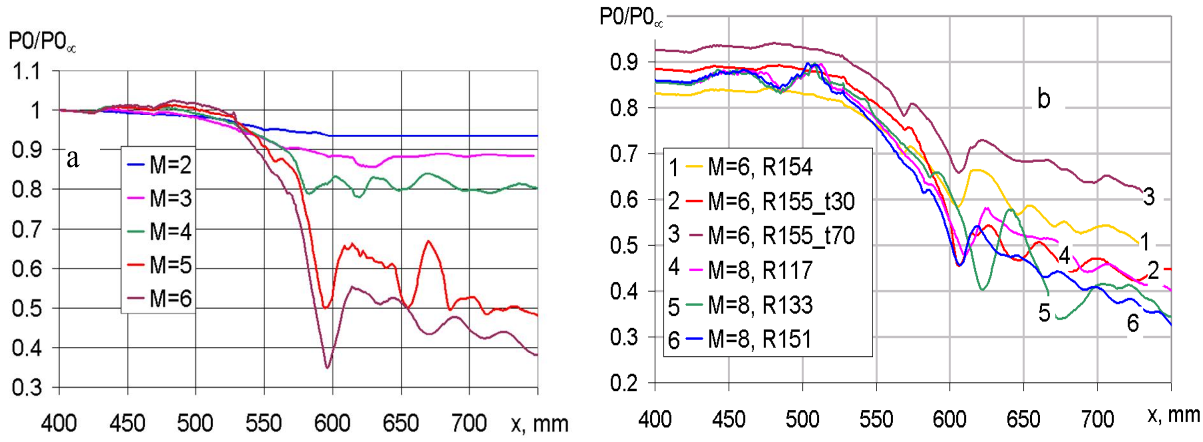

Figure 3: Averaged total pressure under blow down (a) and hot-shot (b) wind tunnel conditions

Comparison of results of calculation of static pressure distribution under test conditions in blow down wind tunnel is presented in Figure 4. This data shows good agreement of computed and experimental data on major part of the channel length of model. Appreciable quantitative difference is observed in the end of computational domain where experimental values can exceed results of calculation. Probably, this is effect of influence of subsonic diffuser, in which the throttling device should be installed. This effect becomes more significant at the rise of Mach number.

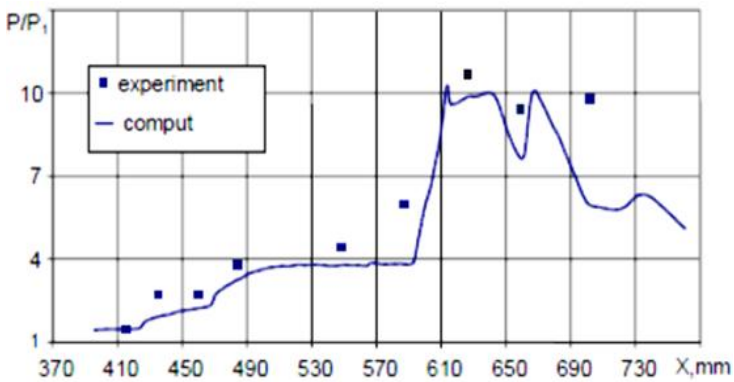

Figure 4: Central body pressure distribution at Mach number 6in blow down wind tunnel

The increment of shock wave intensity with Mach number rise results in the formation of an extensive separation region on the channel walls behind channel entrance. This region covers the channel section and causes the significant pressure rise. Determination of skin friction distribution on the walls allows detecting these regions and, correspondingly, area location of maximum pressure and heat flux. Results of calculation of skin friction distribution on the walls of a central body at Mach numbers 2-6 is presented in Figure 5. The presence of the boundary layer separation region arise at Mach number M ≥ 4 when the skin friction coefficient become less than zero. One can see that separation zone increases at Mach number rise. Size of separation zone can reach 5 height of throat as it occurred at Mach number M=6. These results were confirmed by means of Schlieren visualization of flow in the model throat.

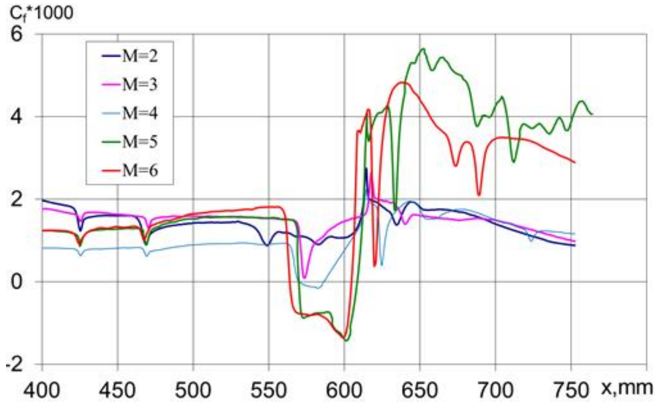

Figure 5: Computed skin friction along the central body for various Mach numbers

Three geometrical configurations were investigated at Mach number M=8 (Table 1) for three different relative internal contraction h_{ent}/h_{th}=3.11, 2.91 and 2.32 that corresponded experimental conditions of the runs 133, 151 and 117. Here h_{ent} and h_{th} is height of entrance in the channel and throat height, correspondingly. The static pressure distributions along the central body (a) and the cowl (b) are presented in Fig. 6. Comparison data obtained indicate qualitative agreement of the experimental and computational data.

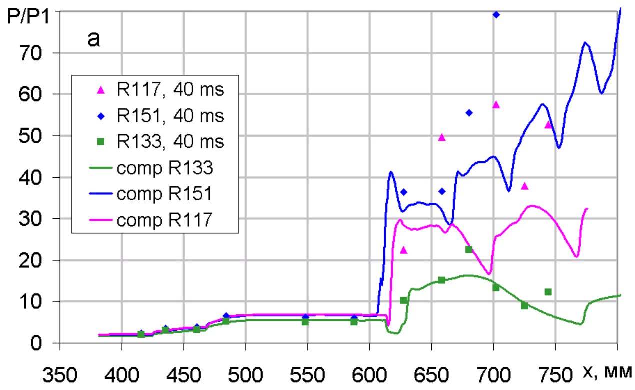

Figure 6, a: Static pressure distribution along central body at M=8

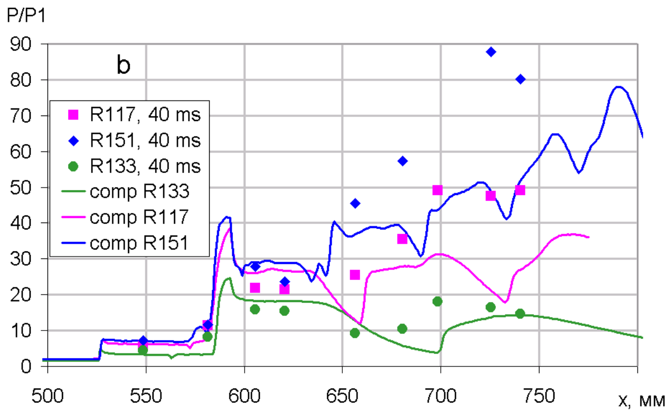

Figure 6, b: Static pressure distribution along cowl at M=8

The possible reason of disagreement between computed and experimental values may cause the fact that the cowl of the real model has the cavity and subsonic diffuser downstream behind the end of computation domain. Such geometrical configuration is source of formation of the massive flow separation. The separation zone moving upstream, leads to a significant static pressure rise inside the channel and, as a result, to increasing of the losses of total pressure. For computations under condition of hot-shot wind tunnel there is also a problem to define properly the incoming flow parameters, including turbulence level, and wall temperature. Computations were performed for constant conditions for fixed time, whereas they were changing in real flows.

The flow in inlet has very complex structure because of multiple shock waves and expansion fans and their interactions with boundary layer on channel walls, which is difficult-to-analyzed on the basis of only measurements and even visualization. Here mathematical modeling gives the instrumentation that helps one to construct the flow field picture. According to Schlieren visualization data, the M=4 flow structure is consistent with the calculated density contours flows, as it shown in Figure 7. The density contours are presented in the whole calculation region (a) and in the part of channel (c), which corresponds to the experimental flow visualization picture in model throat (b). Comparing Figure 7b to Figure 7c it is quite easy to identify the wave structures and corresponding flow regions on the experimental and calculation schemes. The gas-dynamic patterns of the flows constructed on the basis on a joint analysis of the computation and the visualization and measurements gives more deep understanding of the processes in the duct and helps one to design the efficient devices

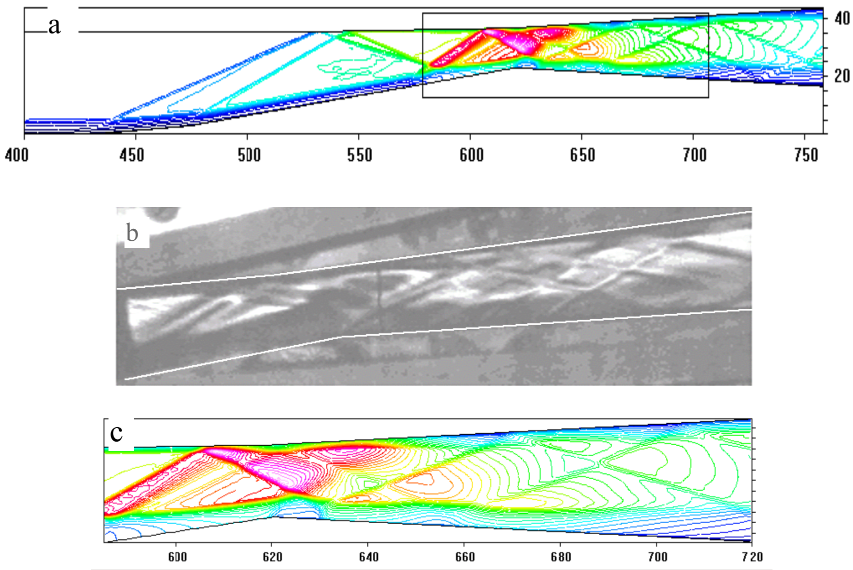

Figure 7: The computed density contours (a), (c) and experimental flow visualization (b) at M=4

In computations, the “start -unstart” of the channel was modeled depend on Mach number and corresponding internal contraction. When compression grows for example, in consequence of increase of angle of attack is growing, the separation zone on the central body and cowl becomes larger that covers the channel and brings about a significant pressure rise inside the channel. If cross section is deficient for realization of supersonic inflowing, separation zone is moving upstream along channel and its size is increasing. One can see the separation zone growth and movement of the separation zones and the shock waves system upstream in Figure 8. This process is accompanied by subsequent pressure rise up to moment of break of inflowing and appearance of normal shock wave before channel entrance. These results conform to experimental data of pressure measurement and flow visualization in blow down wind tunnel.

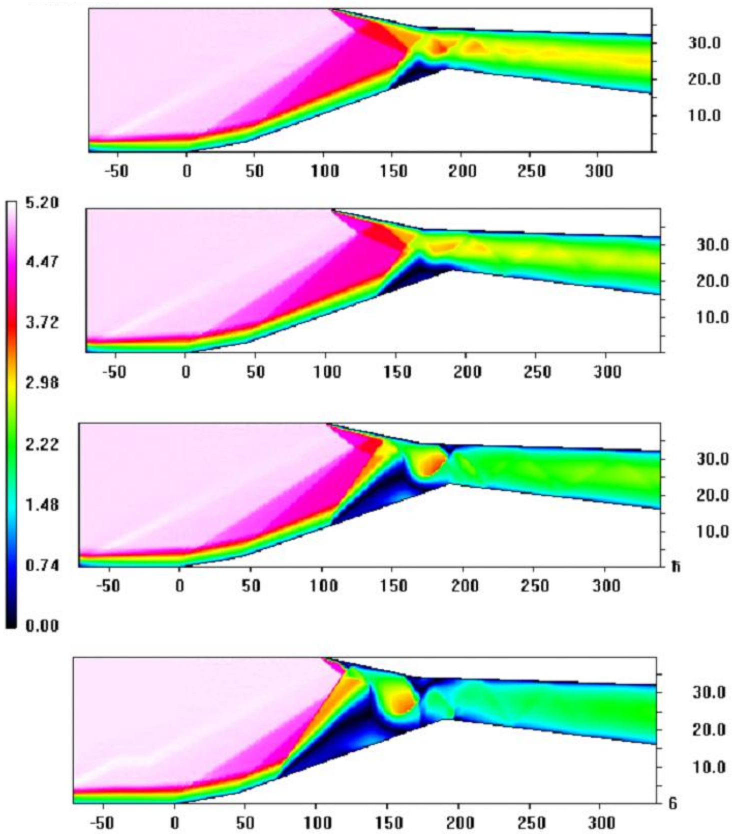

Figure8: Modelling the inlet "unstart" process at M=6

Thus, there is a transition from "start" to "unstart" state of the channel. Presented data show the presence of extensive subsonic areas, considerable decrease of flow velocity in a channel and increase of a boundary layer thickness.

The Figure 9 shows change of skin friction coefficient at process realization "unstart" the channel. It can be seen that the area of negative values of a skin friction increases in process of growth of the sizes of separation area and shifts upstream to a channel entrance. The data obtained allows defining extent of a separation region, which can reach till ten heights of the throat that corresponds approximately to half of length of the channel.

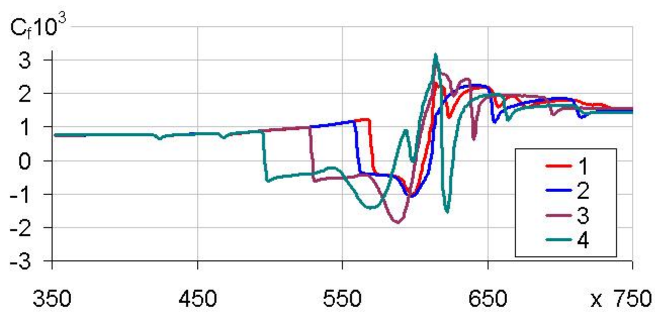

Figure 9: Skin friction coefficient distribution in the channel with separation zone.
Lines 1-4 corresponds pictures in Figure 8 from the upper to lower

5 CONCLUSIONS

Thus, the methods of physical experiment and computational modeling have been used in the work to study the properties of complex flows in the inlets with changeable geometrical configuration in a wide range of Mach number. The experimental findings present a basis for the mathematical model and calculation algorithm verification. At the same time, the parametric computations performed within the wide range of flow and geometric parameters help one to carry out the experiment and provide a basis for the choice of optimum configurations and explanation of flow features.

ACKNOWLEDGEMENTS

This research was partially supported by the Russian Fund for Basic Research, project No. 16-08-00917.

6 REFERENCES

[1] R. R. Boyce, S. Gerard, and A. Paull, The HyShot Scramjet Flight Experiment—Flight Data and CFD Calculations Compared, *AIAA Paper* No. 2003-7029, Dec. 2003, 10p.

[2] C. R. McClinton, X-43—Scramjet Power Breaks the Hypersonic Barrier: Dryden Lectureship in Research for 2006, *AIAA Paper* 2006-1317, Jan. 2006, 12p.

[3] J. Seddon, and E. L. Goldsmith, Practical Intake Aerodynamic Design, *AIAA Education Series, AIAA, New York,* 1993, 398p.

[4] F. Falempin, and M. Goldfeld, Design and Experimental Evaluation of a Mach 2 – Mach 8 Inlet, *AIAA Paper No.* 2001-1890, Kyoto, 2001, 8p.

[5] E. T. Curran, W. H. Heiser, and D. T. Pratt, Fluid Phenomena in Scramjet Combustion Systems, *Annual Review of Fluid Mechanics,* **28**, Jan. 1996, pp. 323–360.

[6] D. M. Van Wie, Scramjet Inlets, *Scramjet Propulsion, Progress in Astronautics and Aeronautics, AIAA, Reston*, VA, Vol. **189**, Chap. 7, 2001, pp. 447-512.

[7] F. Falempin, M. Goldfeld, R. Nestoulia, A. Starov, Evaluation of a variable geometry inlet for dual mode ramjet, *AIAA Paper No.* 2002-5232, 2002, 8p.

[8] C.P. Wang, K.Y. Zhang, K.M. Cheng, Pressure Distribution Measurements in Scramjet Isolators under Asymmetric Supersonic Flow, *AIAA Paper No*. 2006-818, 2006. 10p.

[9] N. Bachchan, R. Hillier, Effects of Hypersonic Inlet Flow Non-Uniformities on Stabilising Isolator Shock Systems, *AIAA Paper No.* 2004-4716, Aug. 2004, 10p.

[10] A., Hamed, and J. S. Shang, Survey of Validation Data Base for Shockwave Boundary-Layer Interactions in Supersonic Inlets, *Journal of Propulsion and Power*, **7**, No. 4, 1991, pp. 617–625.

[11] K. Matsuo, Y. Miyazato, H.-D. Kim, Shock Train and Pseudo-Shock Phenomena in Internal Gas Flows, *Progress in Aerospace Sciences,* **35**, No. 1, 1999, pp. 33–100.

[12] C. Cox, C. Lewis, and R. Pap, Prediction of Unstart Phenomena in Hypersonic Aircraft, *AIAA Paper No.* 1995-6018, April 1995. 9p.

[13] P. J., Waltrup, and F. S. Billig, Structure of Shock Waves in Cylindrical Ducts, *AIAA Journal,* **11**, No. 10, 1973, pp. 1404–1408.

[14] B. U. Reinartz, C. D. Herrmann, and J. B. Ballmann, Aerodynamic Performance Analysis of a Hypersonic Inlet Isolator Using Computation and Experiment, *Journal of Propulsion and Power,* **19**, No. 5, 2003, pp. 868–875.

[15] J. L. Wagner, A. Valdivia, K. B. Yuceil, N. T. Clemens, D. S. Dolling, An Experimental Investigation of Supersonic Inlet Unstart, *37th AIAA Fluid Dynamics Conference and Exhibit, 2007 Miami, FL, AIAA Paper No.* 2007-4352, 2007, 11p.

[16] D. M. Van Wie, F. T. Kwok, R. F. Walsh, Starting Characteristics of Supersonic Inlets, *AIAA Paper No.* 96-2914, July, 1996, 9p.

[17] X. Veillard, R. B. Tahir, E. V. Timofeev, S. Mölder, Limiting Contractions for Starting Simple Ramp-Type Scramjet Intakes with Overboard Spillage. Journal of Propulsion and Power, **24**, No. 5, 2008, pp. 1042–1049.

[18] J. Haberle, A. Gulhan, Investigation of Two-Dimensional Scramjet Inlet Flowfield at Mach 7, Journal of Propulsion and Power, **24**, No. 3, 2008, pp. 446-459.

[19] S. Srikant, J.L. Wagner, M.R. Akella, N. Clemens, Unstart Detection in a Simplified-Geometry Hypersonic Inlet–Isolator Flow, Journal of Propulsion and Power, **26**, No. 5, 2010, pp. 1059-1071.

[20] M. T. Nair, N. Kumar, S. K. Saxena, Computational Analysis of Inlet Aerodynamics for a Hypersonic Research Vehicle, *Journal of Propulsion and Power,* **21**, No. 2, 2005, pp.286-291.

[21] D. Schulte, A. Henckels, R. Neubacher, Manipulation of Shock/Boundary-Layer Interactions in Hypersonic Inlets. *Journal of Propulsion and Power*, Vol. 17, No. 3, 2001, pp.411-417.

[22] D. Rozario, and Z. Zouaoui, Computational Fluid Dynamic Analysis of Scramjet Inlet, *5th AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada, AIAA Paper* No. 2007-30, 2007, 20p.

[23] G.V. Candler, P. K. Subbareddy, J.M. Brock, Advances in Computational Fluid Dynamics Methods for Hypersonic Flows. *Journal of Spacecraft and Rockets*, **52**, No. 1, 2015, pp.17-28.

[24] D. Knight, Automated optimal design of supersonic and subsonic diffusers using CFD, *European Congress on Computational Methods in Applied Sciences and Engineering ECCOMAS, Barcelona, September* 2000, 21 pp.

[25] C. Bourdeau, G. Carrier, D. Knight, and K. Rasheed, Three Dimensional Optimization of Supersonic Inlets, *AIAA Paper No.* 99-2108, 1999, 10p.

[26] S. Mölder, E. V. Timofeev, and R. B. Tahir, Flow Starting in High Compression Hypersonic Air Inlets by Mass Spillage, *AIAA Paper No*. 2004-4130, July 2004, 11p.

[27] D.C. Wilcox, *Turbulence Modeling for CFD. La Canada, California: DCW Industries Inc.* 1993. 460 p.

[28] A.V. Borisov, N.N. Fedorova, Numerical simulation of turbulent flows near the forward-facing steps, *Thermophisics and Aeromechanics*. **4**, No.1, 1996, pp. 69-83.