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

NUMERICAL SIMULATION OF HYDROGEN JET INJECTION AND IGNITION IN SUPERSONIC FLOW

Yulia V. Zakharova[1,2], Natalya N. Fedorova[1,2], Svetlana A. Valger[1]

Marat A. Goldfeld[2], Olga Vankova[2]

[1] Novosibirsk State University of Architecture and Civil Engineering (Sibstrin)
Leningradskaya str., 113, 630008, Novosibirsk (Russia)
e-mail: swetla-ya@mail.ru

[2] Khristianovich Institute of Theoretical and Applied Mechanics SB RAS
Institutskaya st., 4/1 Novosibirsk (Russia)
{nfed, vankova, julsemen, gold}@itam.nsc.ru

Keywords: supersonic flows, mixing, combustion, numerical simulations, propulsion system.

Abstract. *In the present paper, the numerical and experimental study of the flows in combustion chamber has been performed. The experiments were conducted in the hot-shot wind tunnel. 3D numerical simulations were conducted by means of ANSYS FLUENT 16.0 on the RANS-based approach. The comparison and joint analysis have been performed for the calculated and experimental data. The gas-dynamic analysis has been carried out for channel flows with multiple hydrogen jet injection from walls. In the calculation, the influence of jet injection angle on the mixing process and structure of the supersonic flow in the channel is carried out. Preliminary 2D numerical results with hydrogen jet injection has shown that zone of combustion extend from the area of injection and cover the whole cavity. Further 3D studies were carried out with multiple hydrogen jet injection in a channel with backward-facing step. In this case the combustion layer extends over the bottom and top walls of the channel.*

1 INTRODUCTION

Scramjet (supersonic combustion ramjet) engine is one of the candidates for hypersonic flight propulsion system which will be used in wide range of flight Mach numbers from 4 to 12. Scramjet should be well operated in dual and scram modes depending on the flight Mach number [1]. In particular, scramjets 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 [2], followed by successful flights of the Hyper-X vehicles in 2004 at Mach number 6.8 and Mach number 9.6 [3]. Originally, many studies have been done for hydrogen-fueled concepts of supersonic combustion chamber, primarily for space transportation systems. In this case, the upper speed bound lies in the Mach number 12-16 range [4-5 and flow velocity at the entrance of combustor can achieve Mach number of 4-5. A lot of key issues have to be addressed for developing the dual mode ramjet technology. One of them consists in designing a combustion chamber able to operate in a very large Mach number range with enough high effectiveness.

The computer design tools are widely used in developing the optimum characteristics of the combustion chamber of different types [6], which must guarantee ignition and effective operation within the wide range of Mach numbers [7-8]. The main parameters to evaluate efficiency of the combustion chamber are the mixture degree, ignition condition, combustion efficiency and pressure losses. Results of experimental researches at different Mach numbers [9-10], however, have shown essential discrepancy between real characteristics of combustion chamber and anticipated values, in particular, in realization of self-ignition and achievement of high level of combustion efficiency.

The simulation of supersonic mixing and combustion, with a focus on the supersonic combustion ramjet (scramjet), poses many challenges. Even for non-reacting flows, the way to properly treat compressibility effects on turbulence at high Mach numbers is far from being resolved. Complex geometries posed additional difficulties. When combustion dynamics are added to the problem, uncertainty exists in the turbulence-chemistry interaction, shock wave-chemistry interaction, as well as the general lack of adequate knowledge of the effects of supersonic conditions on turbulence, reaction rates, and flame regimes. Review of these problems can be found in [11-12].

Backward facing step and cavity are widely used as effective flame holders at supersonic flow velocity in the channel. The results of numerical modelling and experimental investigations of high-enthalpy turbulent flows in the neighborhood of 90-degree BFS at the Mach numbers $M_\infty = 2-4$ are presented by authors in [13-14] under conditions of cold (T_0=300 K) and high-enthalpy (T_0=2500 K) external flows at M_∞ = 2, 3, and 4. It was found that temperature conditions can substantially affect the separation zone length and the flow structure behind the step. Different vortex structures of the separation zone behind the step were obtained, depending on the temperature factor. Nevertheless, it should be noted that these investigations were performed only for the external flow and for one step configuration.

The influence of step configuration on the structure of supersonic reacting flows in the channel under adiabatic and cold wall conditions was investigated in [15]. It is shown that ignition of mixtures essentially depends on the channel geometry and temperature conditions at the walls.

Results of modeling the interaction of a plane supersonic jet with a supersonic turbulent high-enthalpy flow in a channel are presented in [16]. Parametric studies show that an increase in the angle of inclination and the mass flow rate of the jet leads to an increase in the depth of jet penetration into the flow, but more intense separated flows and shock waves are observed in this case.

The results of the numerical study on the cross-flow helium jet injection into a channel with BFS are presented in [17]. Significant 3D effects are revealed in the computations for the case of the round jet hole. It was shown that the main difference with the 2D case is the lower intensity of the interaction of the primary (air) and secondary flows due to the primary flow spreading around the jet.

The three common approaches for modeling turbulent flows usually use: and Reynolds-averaged Navier-Stokes equations (RANS) [18-19], direct numerical simulation (DNS) [20], large-eddy simulation (LES) [21-22]. It is well known that the RANS approach is the most computationally efficient and has a chance of completely modeling realistic aerospace systems. This is followed by LES, whereas DNS is still too costly for realistic engineering problems. However, the success of the RANS approach is dependent on type and complexity of flow in combustion chamber, and the procedure needs to be verified for every class of problem, making it non-universal [23]. Moreover, the approach is inherently steady and cannot deal with unsteady large-scale structures that determine the dynamics of many important flow problems. The main issue with LES, in comparison with RANS, is the computational cost.

There are a number of studies on hydrogen-fueled of scramjet, in which upper speed bound on flight Mach number was established in Mach 6-10 range. Mach number at the combustor entrance for this flight range should be 1.5-2.5. In spite of large amount of such investigations, the investigation of hydrogen combustion at high flow velocity is finding ever-increasing interest. Nevertheless, such researches were limited data for Mach numbers less than 3. The flow simulation in high-speed combustion chambers is one of important directions in these researches. The main goals of the present investigations were as follows:

- to get a new data on the flows in combustion channel;
- to investigate the influence of the injection angle, total temperature and Mach number on the self-ignition, flame propagation and combustion efficiency;
- to verify the numerical algorithm and code, including chemical schemes, on the channel flow experimental data.

2 COMBUSTOR MODEL AND EXPERIMENTAL SETUP

The combustor channel consists of rectangular channel with sudden expansion in the form of a BFS. The scheme of the model, position of injectors, and the basic dimensions of the combustor channel are presented in Figure 1. Model has two parts: multi-injector section and expanding section. In the first of part, fuel injection was carried out by means of sonic jets.

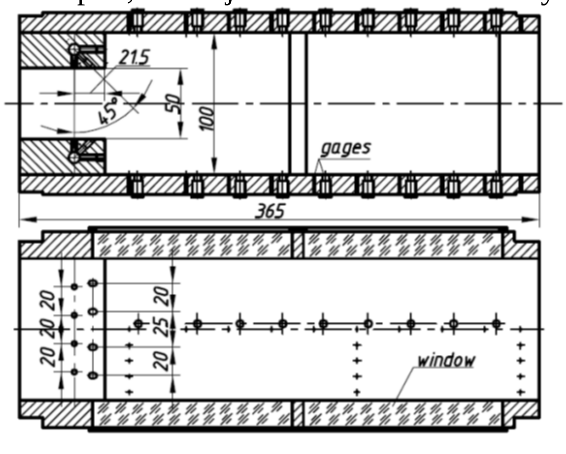

Figure 1: Model scheme of combustor and injector section.

The injecting holes have been developed as changeable elements and allowed to obtained different angles of fuel injection. Such a design enhances considerably the capabilities of the model. Variation of the fuel injection speed is possible by means of use different changeable fuel nozzles. Glass windows are placed along the whole inlet duct of the model to visualize the flow and to determine its structure (separation and attachment of the boundary layer, location and form of the flame). The system of fuel supply is intended for different types of fuel: hydrogen, kerosene and propane. Two separated systems can be used to supply the fuel into the main injectors and into the recirculation zone.

Test of combustor were performed at the hotshot wind tunnel IT-302M [24] with arc heating in the attached pipeline mode. Such mode of investigation allows an effective use of the advantages of the hotshot wind tunnel as a source of a high-enthalpy test gas (air). Photo of model and some element of the wind tunnel are shown in Fig. 2.

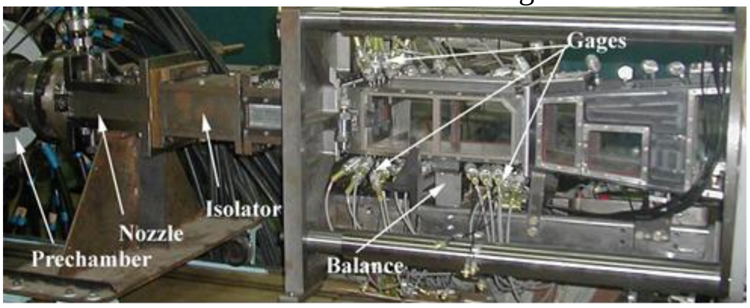

Figure 2: Elements of wind tunnel and model installation.

The choice of the initial values of air pressure in the discharge chamber and the voltage of capacitors allowed obtaining the required flow temperature. This approach ensures not only the necessary Mach number but also the required pressure and temperature at the combustor entrance. High parameters may be reached due to the absence of technological problems related to the temperature strength of the sharp edges of the model. In addition, an increase in the combustor dimensions allows one to extend significantly the capabilities of flow/flame diagnostics and to develop a modular principle of construction of the combustor for extending the range of parametric studies.

Peculiarity of this wind tunnel is the flow parameters which are falling during the operation time (100-150 ms). Therefore large numbers of runs were carried out with the pressure multiplicator for maintenance of constant equivalence ratio (ER) value. Models were tested at the following conditions at the duct entrance: Mach numbers M_{en}=2-4, total temperature T_t=2000-3000K, static pressure P_{en}=0.08-0.5 MPa, and fuel-air ER varied from 0.25 to 1.4.

During the tests, the parameters were measured as follows: the stagnation flow parameters in first and second prechambers; air and fuel flow rates; distributions of static pressure and heat flux in the model channel; Pitot pressure and temperature at model exit; base pressure distribution on backward facing step of injector device, forces acting on injector section. Large amount of measured stations allowed us to obtain detail distributions of static pressure and heat flux including transversal directions and base pressure. The video camera with frequency of about 800 frames per second was used for registration of flame images. The combustion efficiency was determined using an optical scheme of radiation registration in the ultraviolet range and one-dimensional calculation.

3 TESTING OF KINETIC SCHEMES

For numerical simulation of ignition of the hydrogen-air mixture it is necessary to use kinetic scheme that correctly predicts the ignition in supersonic external flow. Therefore, at the first stage, several kinetic schemes were tested [25-27]. Calculations were provided in terms of experimental data [28] where hydrogen was injected into a supersonic flow through a tube of d_1 = 9.5 mm diameter. The tube was installed into a supersonic nozzle with diameter of the exit section d = 65.3 mm (Fig 3). The problem was solved as 2D axisymmetric. Computations were performed under the conditions presented in Table 1.

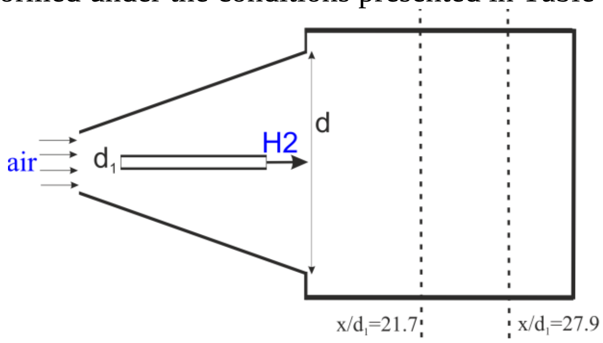

Figure 3: Calculation area for test problem.

	Jet (H_2)	Free stream
Mach number, M	2.00	1.90
Static temperature, T_{st}, K	251	1495
Velocity, u, m/s	2432	1510
Static Pressure, P_{st}, MPa	0.1	0.1
Mass Fraction:		
H_2	1.000	0
O_2	0	0.241
N_2	0	0.478
H_2O	0	0.281

Table 1: Flow parameters for test problem.

Comparison of the computed and experimental data on H_2O mass fraction in several cross-sections is provided in Fig. 4. The best agreement between the calculated and experimental data was obtained using the kinetic scheme with 19 direct and 19 reverse reactions [27]. Therefore this kinetic scheme was used for further calculations.

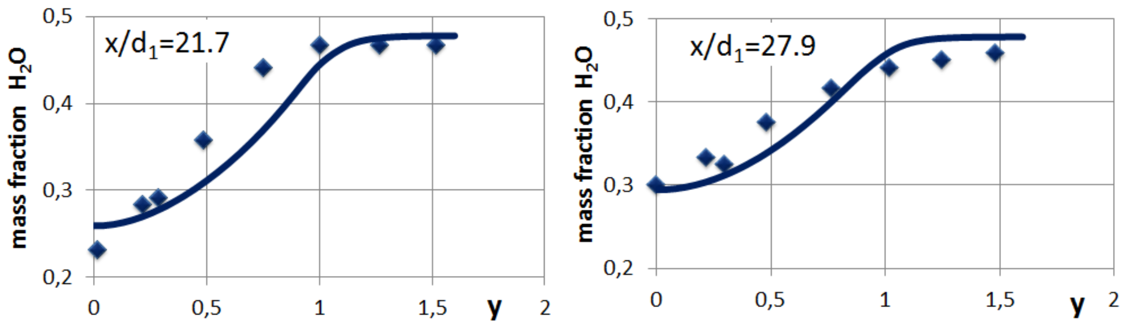

Figure 4: Comparison of experimental (symbols) and numerical (line) data on H_2O mass fraction for test problem.

4 HYDROGEN JET INJECTION IN CHANNEL WITH CAVITY

Parametric computational studies of 2D supersonic air flow in a channel with a cavity were conducted. Geometry and flow parameters are conditioned by experiments provided in the hotshot wind tunnel IT-302M. Computational domain is a channel with a cavity located at the bottom wall. The computational domain (Fig. 5) was bounded by the inlet cross section on the left, by the walls on the top and bottom, and by the outlet boundary on the right. At the inlet section, the turbulent flow parameters at Mach 2.8 were set accounting for boundary layer presence on the top and bottom walls. On the solid walls, the no-slip conditions and cold wall conditions T_w=300 K were assigned.

Hydrogen jet was injected from a slot of 2mm width located 50 mm prior the cavity front face (Fig.5). The investigation of injection angle influence on a flow in a channel was carried out in 2D approach. Computational results were obtained on the RANS-based approach closed by the k-ω/SST model. The AUSM flux vector splitting scheme of second order was used for convective term approximation and an implicit temporal approximation is utilized for time integration. A structured grid with quadrilateral cells was used; the grid was refined toward the solid surfaces. The refinement factor was chosen in such a way that the dimensionless distance to the wall in the first computational node was y_1^+~1, and the laminar sublayer contained approximately ten computational nodes, which provided a fairly accurate resolution of the turbulent boundary layer in the near-wall region. Hydrogen was injected normally to the main flow. The detailed kinetic scheme [27] was used for modeling of combustion.

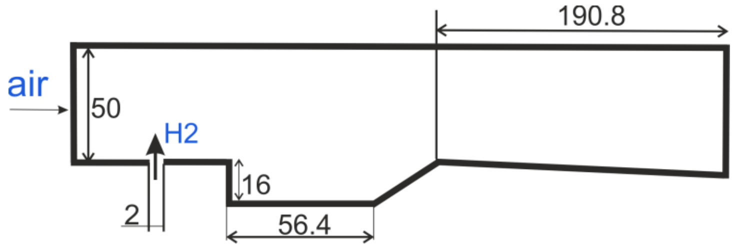

Figure 5: Channel with cavity.

The parameters of the examined flow were following: Mach number M_∞ = 2.8, the total pressure $P_0 = 8.14\times10^5$ Pa and stagnation temperature T_0 = 1798 K. Hydrogen was injected at an angles α = 30°, 60° and 90° at sonic velocity with the following parameters: the total pressure $P_0 = 9\times10^5$ Pa and static temperature T = 300 K. The fields of main parameters allow estimating the influence of hydrogen jet injection angle on wave structure and combustion process in a channel. Hydrogen jet injection leads to reconstruction of the shock wave in a channel and forming an additional separation zones. The Mach number contours for the three injection angles of α=30°, 60° and 90° are presented in Fig. 6.

The shock wave (1) that formed around the jet injection falls on the upper channel wall and then is reflected from it. Reflected shock wave (2) falls onto the mixing layer forming a subsonic region (3) in a vicinity of the rear cavity wall (Fig.6, a). The repeatedly reflected shock (3) comes to the upper wall and induces a local separation zone (4). Inside a cavity, a flow remains subsonic.

In case of 60° angle, the shock (1) is formed in front of the hydrogen jet. Furthermore, a local separation zone (3) is observed on the bottom wall before the hydrogen jet (Fig.6, b). Shock (1) falls on the upper wall and interacts with the shock (2) thereby a λ - configuration is formed.

When hydrogen is injected at an angle of 90° the flow is decelerated to a subsonic speed, as evidenced by the absence of shocks in the channel after the cavity. The presence of local subsonic regions contributes to a better mixing.

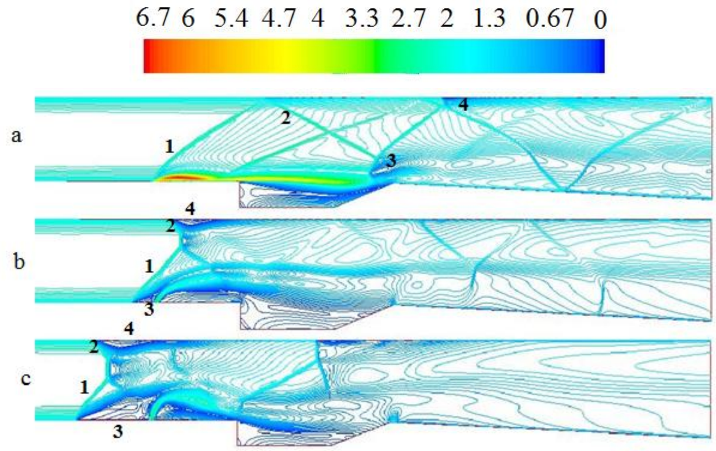

Figure 6: Contours of Mach number for different angles of hydrogen injection
a) 30°, b) 60°, c) 90°.

Further, the jet penetration depth and mixing rate of the primary and secondary flows can be estimated from the hydrogen mass fraction fields (Fig. 7). The mixture is ignited immediately behind the zone of hydrogen injection. Mixing and combustion layer extends over the cavity and along the bottom wall of the channel. When hydrogen was injected at an angle 90° the combustion process captures the whole cavity. Analysis of the H_2 and O_2 concentrations has shown that over the area of the combustion the mixture is lean that mean a lot of oxidant and lack of fuel. Inside a cavity the mixture is rich with a lack of oxidant.

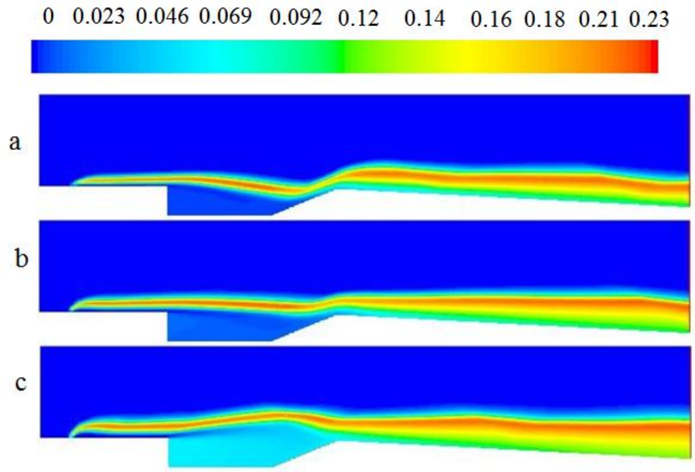

Figure 7: Mass fractions of H_2O for different angles of hydrogen injection
a) 30°, b) 60°, c) 90°.

Modeling of multiple hydrogen jet injection in a channel with BFS was carried out in 3D approach. Channel geometry is matched with the experimental model (Fig.1). The calculation were conducted for the inlet parameters as follows: free stream Mach number $M_\infty = 4$, static pressure $P = 1.06\times10^5$ Pa and static temperature $T = 600$ K. Hydrogen was injected normally to the main flow at: static pressure $P = 25.8\times10^5$ Pa and static temperature $T = 300$ K. For modeling of combustion, the detailed kinetic scheme [27] was used.

The computed mass fraction of H_2O in plane of symmetry is presented in Fig.8. It can be seen that the ignition of the hydrogen-air mixture takes place directly behind the step. Further, the flame propagates along the channel walls capturing the mixing layer. Near the exit boundary there is intense burning throughout the volume.

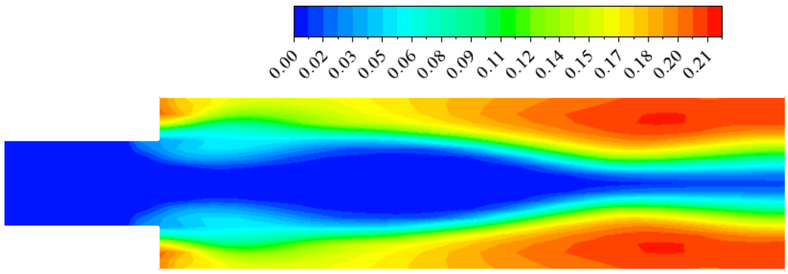

Figure 8: Mass fractions of H_2O in 3D problem.

5 CONCLUSIONS

- 2D and 3D computational analysis has been performed for the problem of hydrogen jet injection into the supersonic flow in a channel with abrupt expansion.
- Several kinetic schemes were tested in terms of experimental data.
- The jet injection angle growth results in reorganization of the shock wave structure in a channel.
- In case of multiple injections the combustion layer extends over the bottom and top walls of the channel.

6 ACKNOWLEDGMENTS

The study was supported by Russian Science Foundation, Grant № 16-19-00010.

REFERENCES

[1] Curran, E.T., Murthy, S.N.B. (Editors). *Scramjet Propulsion. Progress in Astronautics and Aeronautics*, **189**, 2001.

[2] R. R. Boyce, S. Gerard, and A. Paull, The HyShot Scramjet Flight Experiment—Flight Data and CFD Calculations Compared. *AIAA Paper 2003-7029*, 10, 2003.

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

[4] Billig, F. Research on Supersonic Combustion. *J. of Propulsion and Power*. **9**(4): 499-514. 1993.

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

[6] R. A. Baurle, Fuller,R. P., White, J. A., Chen,T. H., Gruber,M. R., and Nejad, A. S., An Investigation of Advanced Fuel Injection Schemes for Scramjet Combustion. *AIAA Paper 1998-0937*, 10, 1998.

[7] T. Cain and C. Walton, Review of Experiments on Ignition and Flameholding In Supersonic Flow, 38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, 7-10 July, Indianapolis, Indiana *AIAA 2002-3877*, 10, 2002.

[8] C. P. Goyne, C. G. Rodriguez†R. H. Krauss and J. C. McDaniel, C. R. McClinton, Experimental and Numerical Study of a Dual-Mode Scramjet Combustor , *Journal Of Propulsion and Power*, **22**, No. 3, May-June 2006.

[9] M.R. Gruber, J.M. Donbar, C.D. Carter, K.-Y. Hsu, Mixing and combustion studies using cavity-based flame holders in a supersonic flow, *Journal Propulsion and Power*, **20**, No. 5, 769–778, 2004.

[10] P.J. Ortwerth, A.B. Mathur, C. Segal, M.G. Owens, Combustion stability limits of hydrogen in a non-premixed, supersonic Flow, Proceedings of the 14th *International Symposium on Airbreathing Engines (ISABE)*, No. 1999-7137, 9, 1999.

[11] R. A. Baurle, Modeling of High Speed Reacting Flows: Established Practices and Future Challenges, *AIAA Paper 2004-0267*, 14, 2004.

[12] J. P. Drummond, Methods for Prediction of High-Speed Reacting Flows in Aerospace Propulsion, *AIAA Journal*, **52**, No. 3, 465-484, 2014.

[13] I.A. Bedarev, M.A. Goldfeld, Yu.V. Zakharova and N.N. Fedorova Investigation of temperature fields in supersonic flow behind a backward-facing step. *Thermophysics and Aeromechanics*, **16**, No. 3, 2009.

[14] Goldfeld M.A., Zakharova Yu.V., Fedorova N.N. A Numerical and Experimental Study of the High-Enthalpy High-Speed Cavity Flow. *Thermophysics Aeromechanics*, **19**, No. 6, 541 - 554. 2012.

[15] N. N. Fedorova, I. A. Bedarev, Yu. V. Zhakharova and M. A. Goldfeld Step configuration influence on combustion in premixed hydrogen-air supersonic flow. *ECCOMAS 2012*, September 10-14, Vienna, Austria, 2012.

[16] N. N. Fedorova, I. A. Fedorchenko, A. V. Fedorov Mathematical modeling of jet interaction with a high-enthalpy in an expanding channel, *Journal of Applied Mechanics and Technical Physics*, **54**, No. 2, 195–206, 2013.

[17] Irina A. Fedorchenko, Svetlana A. Valger, Dennis Daub Numerical study of helium jet injection into channel with sudden expansion, 2nd ECCOMAS Young Investigators Conference (YIC 2013), Bordeaux: France, 2 – 6 September 2013.

[18] Baurle, R. A., and Edwards, J. R., "Hybrid Reynolds-Averaged/Large-Eddy Simulations of a Coaxial Supersonic Free jet Experiment, *AIAA Journal*, **48**, No. 3, 551–571, 2010.

[19] 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, 17-28, 2015.

[20] Georgiadis, N., Yoder, D., Vyas, M., and Engblom, W., "Status of Turbulence Modeling for Hypersonic Propulsion Flowpaths, *AIAA Paper No*. 2011-5917, 2011.

[21] Peterson, D. M., and Candler, G. V., "Hybrid Reynolds-Averaged and Large-Eddy Simulation of Normal Injection into a Supersonic Crossflow," *AIAA Journal*, **26**, No. 3, 533–544, 2010.

[22] N. Georgiadis, D. Rizzetta, and C. Fureby, Large-Eddy Simulation: Current Capabilities, Recommended Practices, and Future Research, *AIAA Journal*, **48**, No. 8, 1772–1784, 2010.

[23] F. Ladeinde, X. Cai, and A. Alabi, A Critical Review of Scramjet Combustion Simulation, *47th AIAA Aerospace Sciences Meeting*, *AIAA Paper* 2009-127, 14, 2009.

[24] L.N. Pusyrev, M.I. Yaroslavtsev, Stabilization of Gas Parameters in Prechamber of Hypersonic Hot-Shot Wind Tunnel, *Izvestia Academii Nauk*, No.5, 86-93, (In Russian). 1990.

[25] Chul Park, Review of Chemical-Kinetic Problems of Future NASA Missions, I: Earth Entries// *Journal of Thermophysics and Heat Transfer*, **7**, No. 3, July-Sept. 1993.

[26] Foluso Ladeinde, A Critical Review of Scramjet Combustion Simulation (Invited). *AIAA Paper* 209-127, 13.

[27] Tien J. H., Stalker R. J., Release of Chemical Energy by Combustion in a Supersonic Mixing Layer of Hydrogen and Air// *Combustion and Flame*. No. 130, 329–348. 2002.

[28] John S. Evans, Charls J. Schexnayder Jr. Influence of Chemical Kinetics and Unmixedness on Burning in Supersonic Hydrogen Flames // *AIAA Jour*, **18**, No. 2, 188-193.