Numerical simulation of transient mixing and ignition processes in a supersonic combustor chamber

Yu V Zakharova, N N Fedorova, M A Goldfeld and O S Vankova

Khristianovich Institute of Theoretical and Applied Mechanics, Siberian Division of Russian Academy of Sciences, Institutskaya, 4/1, Novosibirsk, 630090, Russia

E-mail:julsemen@itam.nsc.ru

Abstract. The paper presents the results of a numerical and experimental study of the processes of mixing and combustion in a model combustion chamber under conditions of multi-jet injection of hydrogen into supersonic (M = 4) external flow. Numerical simulation is performed for the conditions of experiments conducted in a high-enthalpy short-duration facility IT-302M. Joint analysis of experimental and calculated data allows us to obtain new data on the structure of the flow and describe the process of ignition and flame propagation through the channel of the combustion chamber under the conditions of unsteady (falling) input conditions.

1. Introduction

Nowadays, propulsion systems designed to support flights at hypersonic speeds are being actively investigated. An important aspect of the problem is to ensure the efficient operation of the supersonic combustion chamber, which determines the trust characteristics of the aircraft [1, 2]. The design of the combustion chamber should provide a quick and efficient mixing of air and fuel, ignition and intense burning, which is challenging due to high flow velocities, and, as a result, small residence times of the mixture in the combustion chamber.

To achieve fast and efficient mixing of fuel and air, various schemes for fuel supply to the combustion chamber are being actively investigated [3]. A considerable amount of research has been done for the interaction of jets with supersonic transverse flow. In [4-7], detailed data on the structure of the flow in the vicinity of a single jet were obtained, the influence of both the main flow and jet parameters on the mixing processes and flow structure in the channel of the combustion chamber is considered. Recently, the technology of multi-jet fuel injection into a supersonic flow has become widespread. In [8, 9], the cases of interaction of single and multi-jet transversal injection with an external supersonic flow are compared. The mutual influence of the jets is shown to contribute to the formation of large-scale vortex structures, as well as shock waves, and as a result, the mixing efficiency of the multi-jet injection is higher than that of the single-jet injection. Numerical simulation of multi-jet hydrogen injection from the bottom of the cavity was carried out in [10], where it is shown that increasing the injection pressure leads to the expansion of the zone of ignition inside the cavity, and in the case of multi-jet injection, the mixing efficiency is significantly increased inside and above the cavity.

Unsteady behavior of the flow structure inside a supersonic combustion chamber can disrupt the stable operation of the propulsion system. In some cases, a sudden increase in static pressure is observed, followed by a formation of a highly turbulent region of sonic flow that seem to propagate...
upstream along the duct. The nature of the dominant processes causing this unsteady pressure behavior is still not certain. Possible mechanisms that may contribute to this phenomenon are the boundary layer separation and formation of a near-normal shock wave by the region of thermally choked flow. The problem was studied in [12–15] by experimental and computational methods.

The authors of this work investigated various aspects of the operation of supersonic combustion chambers. The flow structure in the channel with a backward facing step (BFS) and cavity for different Mach numbers at the entrance to the combustion chamber was studied in [16, 17]. The processes of ignition and flame stabilization for a pre-mixed hydrogen-air mixture in a model combustion chamber at supersonic flow speeds were numerically and experimentally investigated in [18]. The influence of the BFS configuration on the length of the separation zone, the wave structure of the flow, as well as the process of ignition in the channel at Mach number of 2.8 was considered.

Recent studies [19, 20] present the results of experimental and numerical studies of the processes of hydrogen mixing and ignition in a model supersonic combustion chamber with BFS located on the walls of the channel with multi-jet hydrogen injection. In [19], it is shown that the process of stabilization of combustion is carried out in three stages. The first stage corresponds to local ignition in the separation zones formed on the channel walls due to the action of shock waves induced by jets. In the second stage, intense combustion and an increase in pressure are observed throughout the combustion chamber due to the flame propagating upstream and a significant expansion of the near-wall combustion layer. An increase in heat generation leads to an intensification of mixing and further spread of the flame upstream. The third stage of the process is characterized by steady combustion and high pressure in the entire combustion chamber, while the flux of the core remains supersonic. In [20], the transient process of the hydrogen-air mixture ignition was simulated as a consequence of several steady-state problems with the entry conditions correspondent to the various time moments of the experiments. The purpose of this work is to study the transient development of the process of mixing and combustion under the conditions of experiments conducted in a high-enthalpy short-duration facility IT-302M. Using the special computational technics developed in [21] to reproduce the unsteady input conditions, two experimental runs are simulated with different hydrogen injection angles. Joint analysis of experimental and calculated data allows us to obtain new data on the structure of the flow and describe the process of ignition and flame propagation through the channel of the combustion chamber under the unsteady (falling) input conditions typical of the facility.

2. Experimental setup and numerical algorithm

The model of a supersonic combustion chamber consists of a flat shaped nozzle, an insulator and a 100 mm wide channel that includes the injection and expanding sections. Backward facing steps of h=25 mm height located symmetrically on the upper and lower walls of the channel are used as the flame stabilizer. A series of experiments are performed in the following range of flow parameters: total temperature T0∞ = 1500 ÷ 2400 K, static pressure P∞ = 0.06 ÷ 0.25 MPa, Mach number at the entrance to the combustion chamber M∞=3.83. Hydrogen is injected under angles α=45° or 90° through 8 circular holes of 3.8 mm diameter located in front of BFS on the top and bottom model walls. During the experiments, static pressure and heat fluxes were measured along the model, and the flow was visualized with a high-speed camera. The details of the experimental model, the flow parameters and the measurement techniques can be found in [19].

3D numerical simulation is carried out using ANSYS Fluent 2019R1 on the basis of the unsteady Reynolds-averaged Navier-Stokes equations complemented with the k-ω SST turbulence model and the equations of chemical kinetics. One-stage Arrhenius type scheme [22] is used at the first stage. To reduce the number of computational cells, the computational domain is constructed taking into account the symmetry of the channel in the vertical and transverse direction (Fig. 1). It does not include nozzle and insulator and is limited by the inlet and outlet sections, side and bottom walls and by the frontal and top symmetry surfaces. Addition inlet sections corresponding to the injection orifices are created on the bottom wall.
In the computational domain, a structured hexa-grid is built with the grid points clustered near the walls and the fuel injectors. In the course of the calculation, the grid is adapted according to density and temperature gradients. The final computational grid contains about 3 million cells.

At the main inlet, the Mach number profile, corresponding to a developed 11 mm thick turbulent boundary layer, is set and $M_\infty=3.83$ at the core of the flow, which is consistent with the experimental measurements. Sonic hydrogen jets are supplied from the orifices at 45° (Case 1) and 90° (Case 2) angles having the static temperature of 240 K. In the computations, the fuel-air equivalence ratios calculated by the air and hydrogen massflow rates are 1.06 for the first case and 1.24 for the second case. Falling conditions are prescribed at the main and jet inlets with a help of User Defined Functions and custom expressions obtained by interpolation of the experimental static pressure and temperature data. At the exit boundary, the “pressure outlet” conditions are set. On the channel walls, the no-slip conditions for the velocity and cold wall conditions for the temperature $T_w=300$ K are specified. The computations are carried out in a transient mode with the initial conditions obtained from the preliminary computations without taking injections into account. These data agree well with the experimental "cold" flow data by the flow structure and static pressure distributions on the channel walls [20, 21]. At the initial moment corresponding to some fixed time moment of the experimental run, the injection starts working, what provokes mixing and ignition processes. For the case 1 the starting moment is 15 ms, and for the case 2 it is 10 ms.

Typical time step used in the computation is $10^{-7} \div 10^{-6}$ depending on the case and the stage of the flow development. It should also be noted that carrying out non-stationary calculations takes significantly more computer time as compared to the stationary case.

3. Results and discussions

Figure 2 shows the experimental static pressure distributions along the channel wall for two injection angles at various time moments after turning on the hydrogen supply. Point $x = 0$ corresponds to the BFS edge. It can be seen from the plots that the static pressure measured at the first gauge located prior the injection orifices at $x = -0.06$ m decreases with time, but it increases in the gauges located behind the step. In the first-time moments, for both cases, a sharp increase in pressure is observed at $x \approx 0.1$ m, after which the pressure decreases. The second pressure peak occurs at $x = 0.3$ m for the case 1 and $x = 0.25$ m for the case 2. At subsequent time moment, the local minimum of pressure between the two peaks disappears, and high static pressure values are observed along the entire length of the channel of constant cross-section, indicating the intense combustion. For both cases, a monotonic decrease in pressure is observed in the expanding part of the channel ($x > 0.3$ m). At an injection angle of 90° (case 2), the pressure level throughout the channel is about 30% higher than that in the case 1 (injection angle of 45°), which can be explained by the stronger shock wave generated by the jet and more intense combustion.
Figure 2. Experimental static pressure along the wall at various time moments for case 1 (a) and 2 (b).

Figure 3 represents the dynamics of the development of the ignition process for three different times $t = 1$ (a), 2 (b) and 3 ms (c, d) after starting the injection, obtained in the calculations for the case 1. Water mass concentration fields are presented in the channel longitudinal section passing through the center of the injection orifices (a-c) and in the central symmetry plane (d). Figure shows that as the jet propagates along the channel, hydrogen and air are mixing and the mixture ignites. The unevenness of the reaction product H2O distribution is due to the high hydrogen concentration and the low temperature in the jet core. As the distance from the injection site increases, the degree of mixing and temperature increase, which intensifies the combustion process. Nevertheless, analysis of the fields of mass concentration of reactants has shown that due to the high flow velocity, the most part of hydrogen remains in the near-wall layer. Oxygen is absent in the separation zone behind the BFS and in the near-wall layer in the expanding part of the channel, while about 10% of hydrogen are unburned at the exit section of the channel.

Figure 3. Computed water mass concentration fields for the case 1 at $t=1$ (a), 2 (b) and 3 ms (c, d).
Comparing Fig. 3c and d, it can be noted that there are significant differences in the intensity of combustion and the flame structure along the channel width. In the plane passing through the jet axis, an area with a lower content of water vapor is visible in the center of the jet (fig. 3c), while in the central plane (fig 3d) it is in the center of the jet where the highest values of H2O are observed.

A comparison of the calculated and experimental data on the static pressure distribution along the channel wall is shown in Fig. 4 for cases 1 and 2. Figure shows that the calculations qualitatively correctly predict the flow structure of the first stage of the combustion process described above. Quantitative differences between local minimum and maximum values, not exceeding 30%, can be due to both the difference in the fuel-air equivalence ratio, which is somewhat overestimated in the calculation comparing to the experiments, and imperfection of the computational model.

Figure 4. Experimental and computed static pressure distribution along the wall at \( t = 3 \) ms for the cases 1 (a) and 2 (b).

In order to explain the features of the distribution of static pressure on the channel walls, we analyze the static pressure field obtained in the computations of the case 1 (Fig. 5). In front of the BFS, the high pressure zone (1) induced by the injection can be seen. The compression wave, reflected from the upper symmetry plane, after passing through an expansion fan (3) formed on the BFS edge, falls on the bottom wall and causes an increase in pressure (2) corresponding to the first peak in Fig. 4a. Then the expansion fan reflected from the top symmetry plane comes to the wall, which leads to the formation of a local pressure minimum (4). The second pressure peak is caused by the arrival on the wall of a compression wave (5), re-reflected from the top symmetry plane.

Figure 5. Computed static pressure field for the case 1 at the symmetry plane.

Analysis of temperature and Mach number fields shows that in the region of the second pressure maximum, the temperature rises sharply due to intense burning, and thick subsonic zone arises. The transition to a subsonic combustion mode can trigger the formation of a hot zone, which further spreads upstream and causes disappearance of the local minimum pressure. Under certain conditions,
this can also lead to thermal choking of the channel and formation of a straight shock propagating upstream.

Conclusions
The numerical and experimental study of hydrogen mixing and combustion is performed under conditions of multi-jet injection into supersonic (M = 3.83) flow in the channel with abrupt expansion. Using the special computational techniques to reproduce the unsteady input conditions, two experimental runs are simulated with different hydrogen injection angles. Good qualitative agreement of the experimental and numerical data on the static pressure distribution along the channel wall is obtained. One of the reasons for quantitative difference between experimental and numerical data may be an imperfect kinetic scheme. For further calculations, the detailed scheme containing 38 forward and backward reactions [23] will be implemented. Based on the joint analysis of experimental and simulation data, the first stage of the hydrogen ignition process is described and the scenario of the further development is proposed. Future work will include the simulation of the next stages of the ignition process with the special attention to the prevention of the thermal choking of the channel.

Acknowledgments
The research was carried out within the framework of the Program of Fundamental Scientific Research of the state academies of sciences in 2013-2020 (project No. AAAA-A17-117030610139-4) under the partial support of the Grant No. 17-08-01158 A by Russian Foundation for Basic Research.

References
[1] Bogdanoff D W 1994 J. Propul. Power 10 183–90
[2] Urzay J 2018 Annual Rev. Fluid Mechanics 50 593–627
[3] Huang W 2018 Acta Astronaut 145 492–500
[4] Fedorova N N, Fedorchenko I A and Fedorov A V 2013 J Appl. Mech. Tech. Phys. 54 195
[5] Liua C, Zhaob Y, Wanga Z, Wanga H and Suna M 2017 Acta Astronautica 136 90–100
[6] Li L, Huang W, Fang M, Shi Y, Li Z and Peng Z 2017 Int. J. Heat Mass Transfer 132 484–97
[7] Galeazzo F C C G, Donnert G, Cárdenas C, Sedlmaier J, Habisreuther P, Zarzalis N, Beck C and Krebs W 2013 Int. J. Heat Fluid Flow 41 55–65
[8] Zhao M, Bian Y, Li Q and Ye T 2019 Aerospace Sci. Technol. 89 31–45
[9] Gerdroodbary M B, Takami M R, Heidari H R, Fallah K and Ganji D D 2016 Acta Astronautica 123 283–91
[10] Edalatpour A, Hassanvand A, Gerdroodbary M B, Moradi R and Amini Y 2019 Int. J. Hydrogen Energy 44 13923–31
[11] Song X, Wang H, Sun M, Cai Z, Liu C, Yu J, 2019 Acta Astronautica 159 584–92
[12] O’Byrne S, Doolan M, Olsen S R and Houwing A F P 2000 J. Propulsion Power 16 808–14
[13] Laurence S J, Karl S, Schramm J M and Hannemann K 2013 J. Fluid Mechanics 722 85–120
[14] Han Y, He Y, Tian Y, Zhong F and Le J 2018. Aerospace Sci. Technology 72 114–22
[15] Xue R, Xu M, Bai C, Weng C, Xu C. 2018 Int. J. Energy Res. 42 2481–93
[16] Bedarev I A, Goldfeld M A, Zakharova Y V and Fedorova N N 2009 Thermophys. Aeromech. 16 355
[17] Goldfeld M A, Zakharova Y V and Fedorova N N 2012 Thermophys. Aeromech 19 541
[18] Goldfeld M A, Zakharova Y V, Fedorov A V and Fedorova N N 2018 Comb. Explos. Shock Waves 54 629
[19] Goldfeld M A and Timofeev K Y 2017 AIP Conf. Proc 1893 030147
[20] Fedorova N N, Goldfeld M A and Zakharova Y V 2018 AIP Conf. Proc. 2027 030129
[21] Fedorova N N, Vankova O S and Zakharova Y V 2019 AIP Conf Proc. 2125 030080
[22] Bedarev I A, Rylova K V and Fedorov A V 2015 Comb. Explos. Shock Waves 51 528–39
[23] Tien J H and Stalker R J 2002 Combust. Flame 131 329–48