JST: Engineering and Technology for Sustainable Development
Volume 32, Issue 2, April 2022, 024-031

Appraisal Burning Characteristic and Analysis Effect of Cavity in Scramjet Combustion Chamber

Van-Minh LE, Cong-Truong DINH*, Quoc-Khanh PHAM, Duc-Huy TA, Quang-Sang VO, Hong-Quan LUU, The-Mich NGUYEN, Anh-Tuan NGUYEN

School of Mechanical Engineering, Hanoi University of Science and Technology, Hanoi, Vietnam
**Email: truong.dinhcong@hust.edu.vn*

Abstract

The combustion chamber clearly plays an critical role in generating thrust force so the aircraft can move forward. A scramjet (supersonic combustion ramjet) is a variant of a ramjet airbreathing jet engine in which combustion takes place in supersonic airflow. Researchers are constantly working to improve the efficiency of ultrasonic combustion furnaces by various methods such as: optimize fuel injectors, optimize combustion chamber geometry design, create hole cavity. In this research, the characteristic of supersonic airflow were investigated, and a comparison between the standard chamber and advanced chamber was made to determine the effects of a circular hole (cavity) on pressure and velocity of the fuel mixture through the scramjet. Two dimensional Reynolds-Averaged Navier-Stokes governing(RANS) equations with $k-\varepsilon$ turbulence model and finite rate/eddy dissipation chemistry model have been considered for modelling chemical reacting flows. From the comparison of numerical results, it is found that the development of recirculation regions and additional shock waves from the edge of cavity flame holder is increased and achieved stabilized combustion. From this research analysis, the performance of the scramjet engine with cavity is significantly improved as compared to the design without cavity.

Keywords: Combustion chamber, scamjet, cavity, RANS analysis, shock wave, thrust.

1. Introduction

The scramjet engine is superior to today's aviation vehicles. The scramjet engine is designed to avoid high drag and low combustion efficiency at high Mach numbers by keeping the supersonic flow negative throughout the engine especially in the combustion chamber. Avoiding strong impact waves like the Ramjet has significantly reduced engine drag. The reaction time of only a few milliseconds and the limitation of the combustion's length are the two main issues that hinder the engine's efficiency. The way that the stored fuel is injected into the compressed air also plays a pivotal role. Therefore, researchers are trying to find the optimal locations for fuel injection to achieve higher performance at supersonic speeds.

Combustor geometry has a huge effect on combustion process. Araújo et al. [1] has researched the characteristics of a two-dimensional combustor at transonic velocity (Mach 5-10) with the fuel is the mixture of air and products of burning hydrocarbon C_xH_y. The two-dimensional model in the current work is inspired by this paper, but with a simplified condition – burning process is just between H_2 and O_2, Mach numbers inlet is 2.0. Kummitha and Pandey [2] experimented with the same two-dimensional model but with wavy wall strut. They found that the changes in strut wall changed the mixing process behavior and increased the combustion efficiency significantly. Choubey and Pandey [3], and Kummitha et al. [4] used different strut designs, struts numbers and angles of attack, in the combustor and found that the modified model increased combustion efficiency and decreased the ignition delay. Another research on the cavity inside combustion chamber was conducted by Ben-Yakar and Hanson [5] but with different dimensions – 3 mm depth, as a result, the cavity has a certain effect in keeping and mixing the fuel in the combustion chamber.

Huang and Zhang [6] studied two different combustion models i.e., ultrasonic and subsonic using numerical simulation. A scramjet test engine of the German Aerospace Center (DLR) was selected with a parallel fuel injection system. The kinetic chemistry in the scramjet combustion chamber under dual-mode was explained using mode transition by Shen et al. [7] and Abu-Farah et al. [8], Who explained hydrogen's combustion behaviors with struts are improved at fuel injectors. Data investigation was performed by selecting the LES model to find out the effect of the chemokinetic mechanism between hydrogen and air by Liu *et al.* [9]. The non-burn behavior in a two-

---

ISSN 2734-9381
https://doi.org/10.51316/jst.157.etsd.2022.32.2.4
Received: February 14, 2022; accepted: February 27, 2022

dimensional model was investigated by Gruber *et al.* [10]. Non-combustion mode is a process where the fuel mixture is kept cold to evaluate the behavior of the fuel flow in the combustion zone caused by changes in the geometry and mixing process.

As shown in the literature review above, only this system or the transverse injection system has been studied by other researchers. Parallel fuel injection or bulkhead fuel transfer systems have not yet been extensively studied for combustion chamber efficiency. On the basis of previous research results as well as experimental results for scramjet engine combustion chambers, the content of this paper is to propose a reliable numerical simulation model and method. CFD simulation with k-ε model and simple H_2-O_2 combustion is applied to achieve faster and more accurate results. In addition, an assessment of the influence of the cavity design on the efficiency of the furnace is carried out, thereby drawing conclusions about the more optimal design.

2. Numerical Analysis

2.1. Design Description

In this work, the research object is a scramjet engine. It is a kind of engine designed to operate at hypersonic speeds (over Mach 5), but the combustion process in combustion chamber happens in supersonic mode. A typical scramjet engine has four main components: inlet, isolator, combustor, and ultrasonic exhaust (nozzle) as shown in Fig. 1 [7].

The designed model used in this paper is presented in Fig. 2. The total length of the combustion chamber is 340 mm and height at the inlet is 50 mm and at the outlet 62 mm. The method of fuel injection is parallel with the help of a symmetrical wedge-shaped strut. The symmetrical wedge strut is 6 mm high and 32 mm long containing the injectors that guide the combustion fuel. The wedge tip is positioned 77 mm from the inlet. The diameter of each fuel injector is 1 mm with 15 consecutive holes (with a constant distance between the holes of 2.4 mm).

In the work, we consider the injector at eighth hole with the cross section between the nozzles. The effect of circular cavity on combustion performance is also included as shown in Fig. 3. The hole has the radius of 15mm, and is put after the wedge strut, the distance between the hole and inlet is x ($80\ mm < x < 260\ mm$).

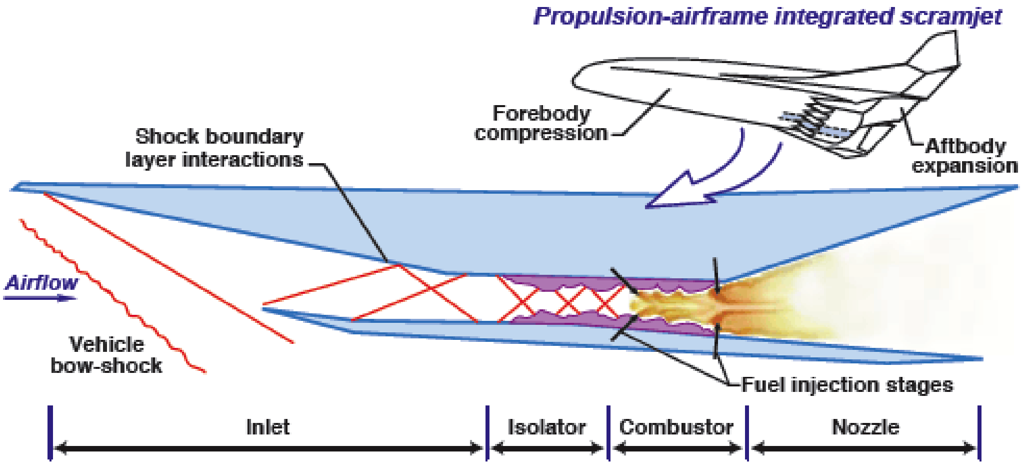


Fig. 1. Scramjet's components

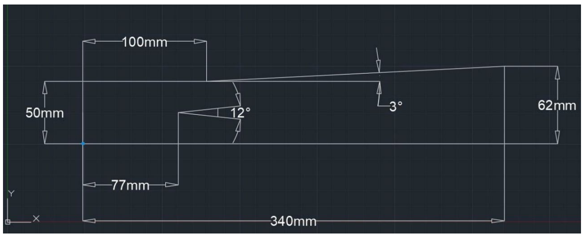


Fig. 2. Combustor's parameters of scramjet engine

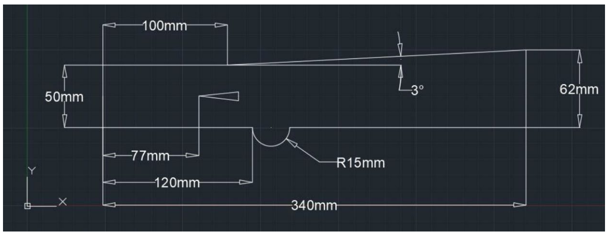


Fig. 3. Model with cavity

2.2. Numerical Method

The computational model was built using ANSYS Fluent 19.1 [11]. Details of the selection models are presented in the following sections. The Reynolds-averaged Navier Stokes (RANS) equations were solved with k-ε turbulence model. The Reynolds Average Navier Stokes Equation (RANS) model is useful for capturing behavioral variables of the flow. The RANS average method is suitable for 2D simulations such as considering the gas flow through the fuel injectors and the combustion in the combustion chamber under consideration.

Combustion modelling is used to capture the potential interactions between fuel and air. The feature transport model with Turbulence-Chemistry Interaction used as Eddy-Dissipation was chosen to explore the properties of the current field. Fuel mixing efficiency and combustion efficiency are the two most important characteristics to evaluate the efficiency of the scramjet combustion chamber. Mixing efficiency is the ratio of the density of fuel injected in the combustible area A to the total fuel injected. The efficiency of a combustion chamber with unreactive (non-combustible) flow with the addition of fuel mass is characterized by the mixing efficiency (η_m), defined by the equation:

$$\eta_m = \frac{\int_A \alpha.\rho_{gas}.Y_{H_2} u dA}{\dot{m_{H_2}}}, \; with \; \alpha = \begin{cases} 1, \; \phi < 1 \\ \frac{1}{\phi}, \phi \geq 1 \end{cases} \tag{1}$$

in which ρ_{gas} is the density of the air; Y_{H_2} is mass ratio of hydrogen with air ($Y_{H_2} = \rho_{H_2}/\rho_{gas}$) ; u is the normal velocity; A is the cross-sectional area and ϕ is the equivalence ratio, defined as $\phi = M_{O_2} Y_{H_2} / 2 M_{H_2} Y_{O_2}$ with M is the molar mass and Y is the mass of the gas; $\dot{m_{H_2}}$ is the flow rate of the injected hydrogen. A good fuel mixing efficiency will give good combustion efficiency. Combustion efficiency can be defined as the ratio of the amount of hydrogen burned to the total gas intake of hydrogen. Combustion is a fast chemical reaction with a high degree of turbulence. The mathematical equation of fire can be written as follows:

$$\eta_{comb}(x) = 1 - \frac{\int (A(x)) \rho_{gas}.Y_{H_2} u dA}{\dot{m}_{H_{2,inj}}} = 1 - \frac{\dot{m}_{H_{2,x}}}{\dot{m}_{H_{2,inj}}} \tag{2}$$

in which $\dot{m}_{H_{2,x}}$ is the mass flow of hydrogen at a given cross-sectional position x; $\dot{m}_{H_{2,inj}}$ is the total mass flow of injected hydrogen and η_{comb} is combustion efficiency.

The geometry and the computational grid were generated using Design Modeler and ICEM-CFD, respectively. A structural grid was used with quadrangular elements, refined at the edges, cutting points and fire zones to clearly capture the flow phenomenon. The configuration of the wedge is isosceles triangle, so there are not many asymmetrical detail areas, so using a structural grid provides easier grid control, shorten the computing time while still providing accuracy as shown in Fig. 4. The detail of boundary conditions are presented in Tables 1 and 2 below.

Table 1. Boundary conditions

Indicator	Air inlet	Fuel
Mach number	2.0	1.0
Static pressure (Pa)	100 000	100 000
Static temperature (K)	340	250
Density (kg/m^3)	0.9734944	0.09698617
Velocity (m/s)	756.1424	1203.324

Table 2. Proportion between air and fuel inlet

Mass ratio	O_2	H_2O	N_2	H_2
Air	0,232	0,032	0,736	0
Fuel	0	0	0	1

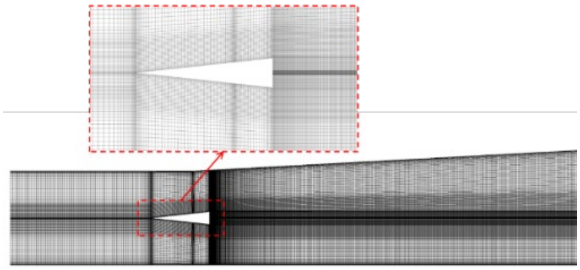


Fig. 4. Mesh of Model

3. Result and Discussion

3.1 Grid Independence Test and Validation Results

Performing meshing and calculating the pressure on the lower wall and the middle wall as shown in Fig. 5, it was found that from about 110,000 mesh, the pressure value starts to stabilize and at more than 162,000 mesh, the optimal value is reached. In fact, the number of nodes in the article is 162,486 grids, checking the effect of mesh number on pressure's stability is acceptable.

First, the investigation was made with cold and not-burnt fuel (hydrogen) to observe the shock waves and evaluate the density, pressure, Mach number and so on before burning fuel in the combustion chamber. The shock waves generated at the tip of the fuel injector are complex. Extended fan-shaped regions behind the symmetrical wedge are followed by oblique collision waves and reflected waves with corresponding deflection amplitudes. The results from simulations are close with experiments about the shock waves's position, structure, and profile (Fig. 6). In Fig. 6, at *x* = 142 mm and *x* = 227 mm, there is a difference between the reflected wave at subsonic frequencies and the visible supersonic fuel flow. Between 137 mm < x < 142 mm the complex interaction of the two shock waves generated at the tip of the wedge and reflected at the channels top and bottom walls with the hydrogen jet is resolved in detail. Because of the one-sided divergent channel, the upper shock wave hits the subsonic hydrogen jet a bit further downstream than the lower shock wave yielding a non- symmetric flow field. The lower shock wave generates a strong pressure gradient in the hydrogen jet leading to a slight expansion of the jet. On the lower side of the hydrogen jet there is only a compression wave but not a shock wave. In the supersonic flow region at *x* = 227 mm the shock waves are not reflected at the jet, rather they are deflected by crossing the jet. There are two shock waves in the shadow picture originating from the base of the wedge. These shock waves are generated by some irregularities on the wedge surface. Clearly, these shock waves and the resulting flow structures are not caught by the simulation.

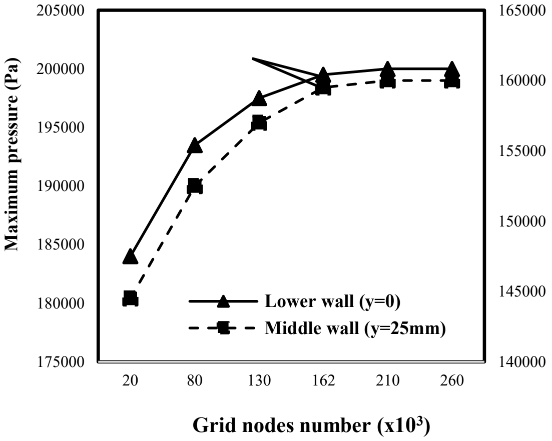


Fig. 5. Grid dependency test.

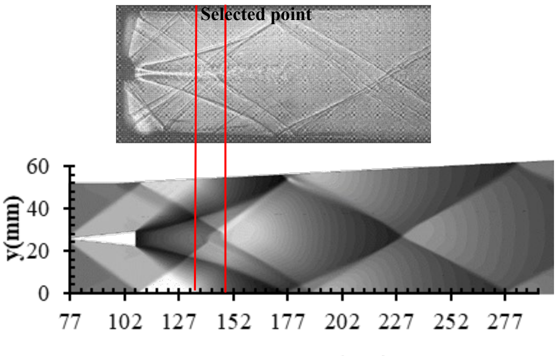
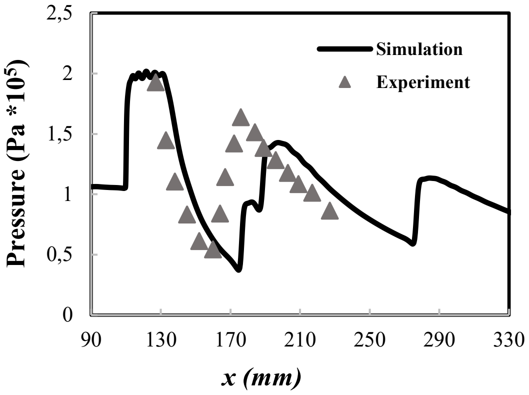


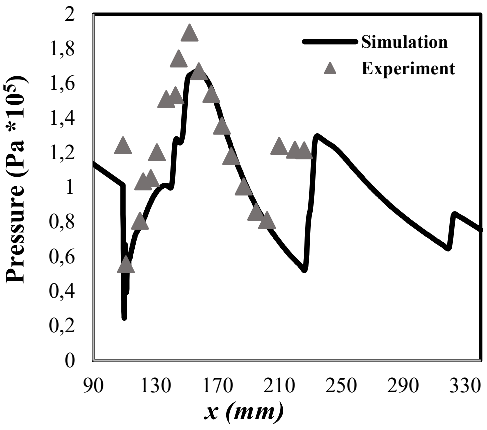
Fig. 6. Validation results: Experiment result – shadow picture (top); Numerical pressure (bottom)

Although the overall agreement between the experiment and the calculation (Fig. 7) is good, the pressure peaks at *x* = 130 mm at the lower channel wall and at *x* = 110 mm in the middle of the channel are missing in the computation. The reasons for the discrepancy could be the influence of three-dimensional effects. In the experiment the hydrogen is injected through 15 holes resulting in a truly three-dimensional flow field. Furthermore, the pressure measurements were taken near the side walls of the channel where corner and boundary layer effects are present. Another possible explanation are interactions of wall boundary layers with the shock waves. These effects are not taken into account here since slip boundary conditions have been applied.

In the experiment, researchers used Laser Doppler Velocimetry (LDV) for velocity measurement [12]. Fig. 8 compares simulation results and experimental results of axial velocity at 4 sections: *x* = 120 mm, *x* = 167 mm, *x* = 199 mm and *x* = 275 mm. Peak velocity at four different x values in simulation are 805 m/s, 831 m/s, 771 m/s and 751 m/s respectively. These two results above are quite likeness and trustable.

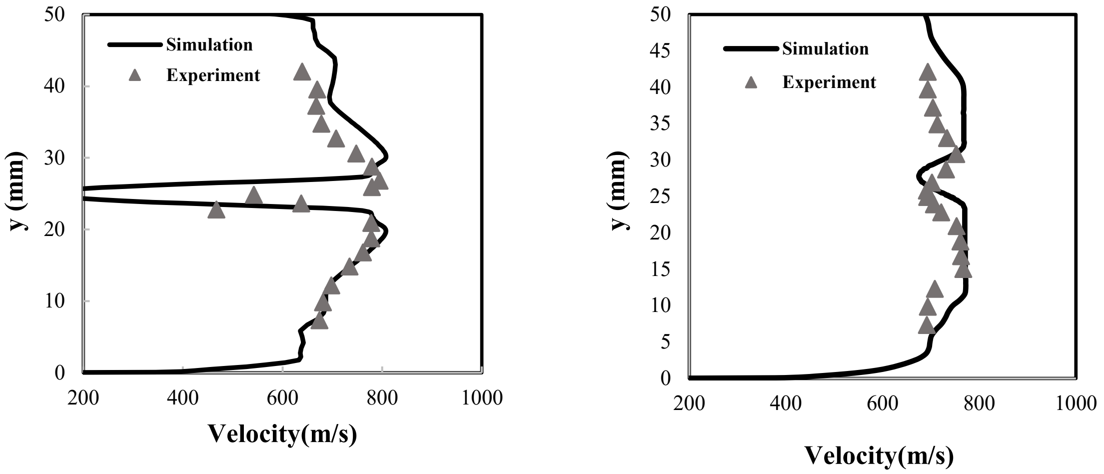


a) Bottom wall (y=0mm)



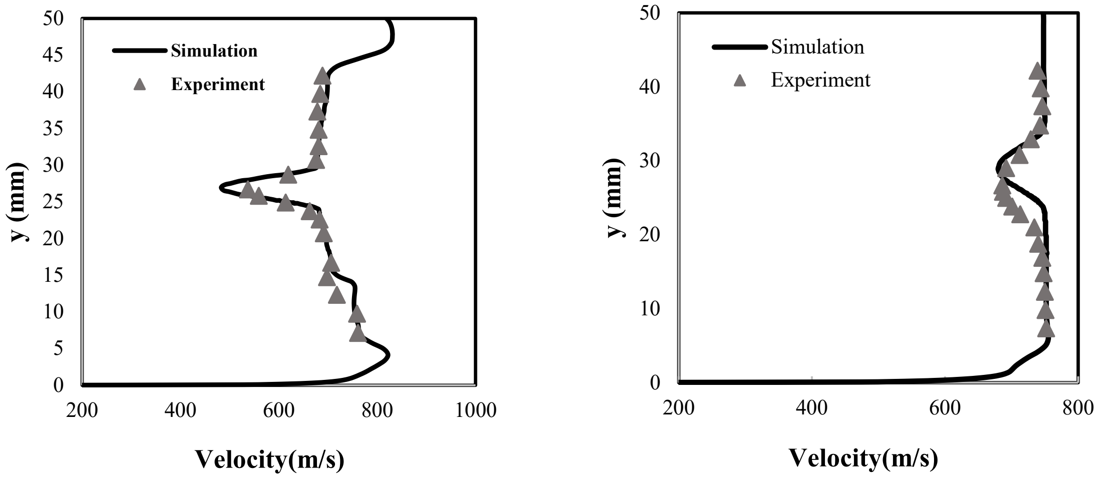
b) Middle (y=25mm)

Fig. 7. Pressure distribution



a) Axial velocity (x=120 mm)

c) Axial velocity (x=199 mm)



b) Axial velocity (x=167 mm)

d) Axial velocity (x=275 mm)

Fig. 8. Axial velocity

In the following section, the numerical results' validity will be examined when fuel mixture was set to burn. Hydrogen fuel is injected and burned at M = 1 and ambient temperature. Both velocity and temperature of streamline are included below. The overall structure and the difference between simulation calculation and experimental results can be seen in Fig. 9 with flow density and pressure calculation. Compared to the shadow picture the calculated combustion zone is too broad at x = 145 mm with a stronger narrowing further downstream. Besides the acceleration by thermal expansion in the flame, there is an acceleration by the constriction of the subsonic jet for x >145 mm. The stronger contraction in the computation results in a larger speed than experimentally observed.

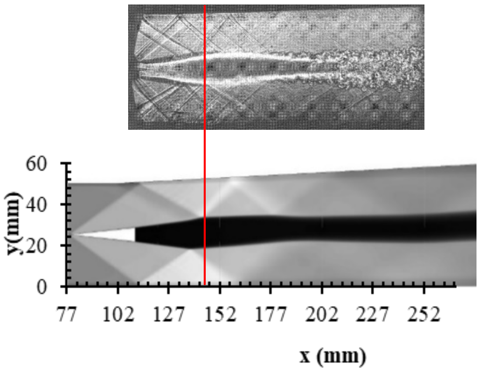
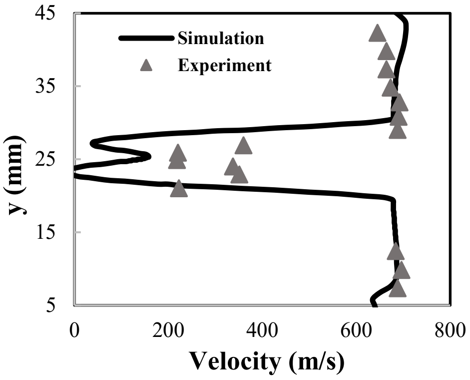
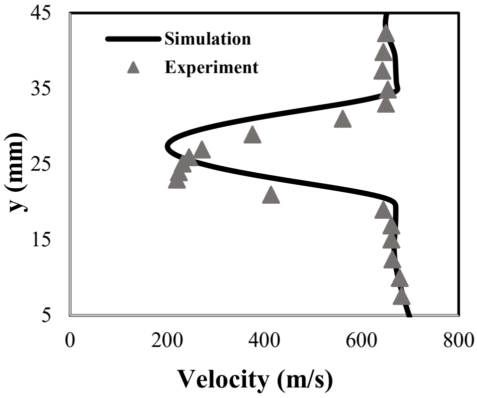


Fig. 9. Comparison: Experiment result – shadow picture (top); Numerical pressure (bottom)



a) x = 120 mm



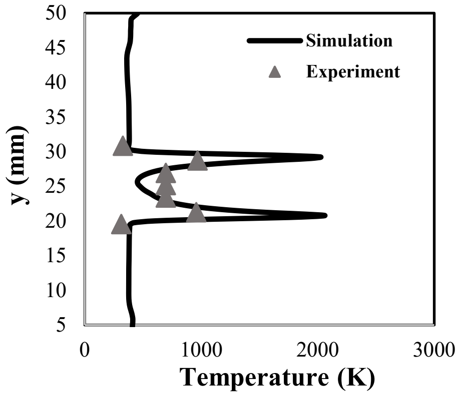
b) x=167 mm

Fig. 10. Axial velocity at cross section: (a) x=120mm; (b) x=167mm

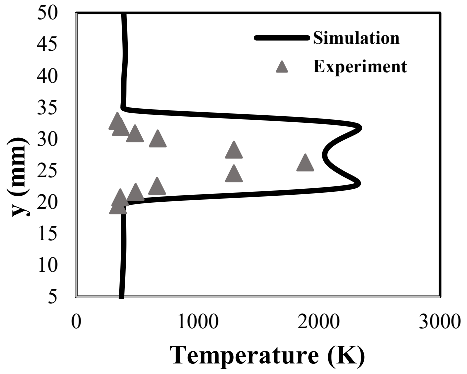
Fig. 10 shows axial velocity at cross section x = 120 mm and x = 167 mm. Peak velocity at both cross sections is match with experiment results, which are 700 m/s at x = 120mm and 674 m/s at x = 167 mm. Fig. 11 plots temperature at cross section x = 120 mm, x = 167 mm and x = 275 mm in simulation and experiment. Maximum temperature approach 2139K at x = 120 mm, 2335K at x = 167 mm and 2139K at x = 275 mm around middle y value.

There are some overcoming at peak value of temperature in experiment results compared with experiment results. These deviations occur because of the simple combustion model (one-order chemical equation for one-order reaction).

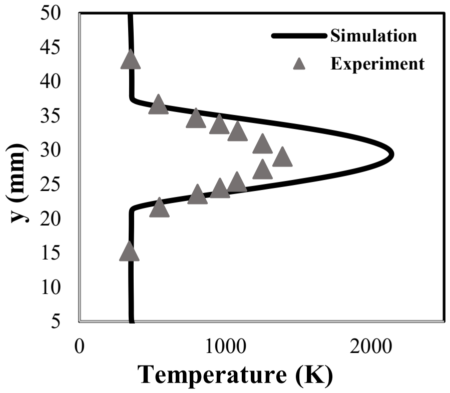
The model used in this case is simple but provides quite precise results and well-known in simulation community. It adapts well in different conditions in simulation models, too. The simulation results confirm this. Though there are some missing at some value, in general, the trend and the coincident between simulation and experiment results is superb and reliable, suitable in investigating further research.



a) Temperature at cross section x=120 mm



b) Temperature at cross section x=167 mm

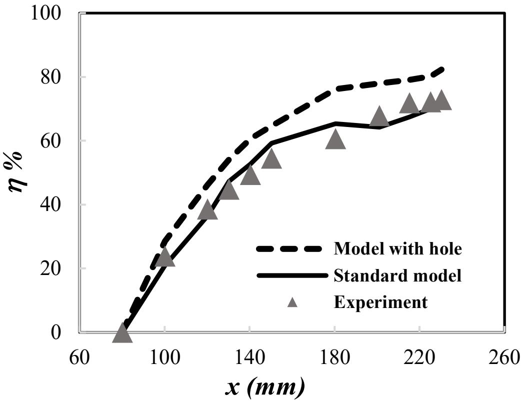


c) Temperature at cross section x=275 mm

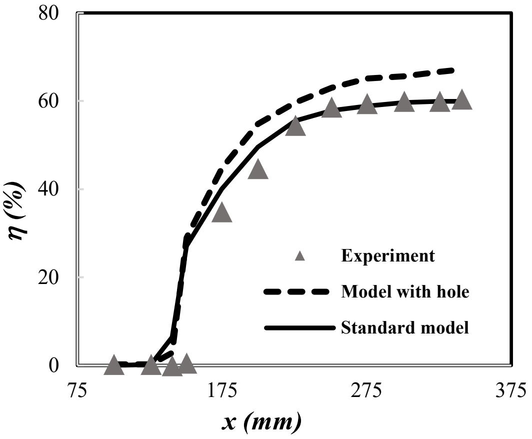
Fig. 11. Temperature comparison at cross section: (a) x=120mm; (b) x=167mm; (c) x=275mm

3.2. Effect of Circular Cavity on Combustion Performance

In the following discussion, a round hole cavity was drilled behind the fuel injector, for the purpose of changing the impact wave interaction, improving the fuel mixing efficiency. Results archived in simulation are compared with data of experiment with standard model.



a) Mixing efficiency



b) Combustion efficiency

Fig. 12. Comparison mixing and combustion efficiency in different models: a) Mixing efficiency; (b) Combustion efficiency

Fig. 12 shows the mixing and combustion efficiency of jet burners with and without a circular hole. Data of simulation with standard model and experiment are very coincident, which proves the precision of simulation and experiment model. All peak values are significantly increased after adding the hole in mixing and combustion efficiencies, about 8.5% and 7,2% respectively than experiment combustion.

Fig. 13 and 14 compare pressure and density distribution in simulation in 2 combustion chamber models (with or without cavity hole). If combustion chamber has cavity hole, several shockwaves are added under the mainstream, after the hole, while streamline above mainstream seem unchanged. The model with the round hole has pushed the mixing area closer to the injector, the wave density and distribution of pressure are thicker.

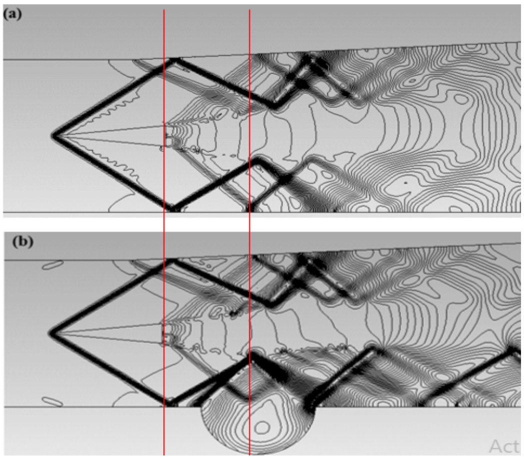


Fig. 13. Pressure distribution: (a) without hole; (b) with hole

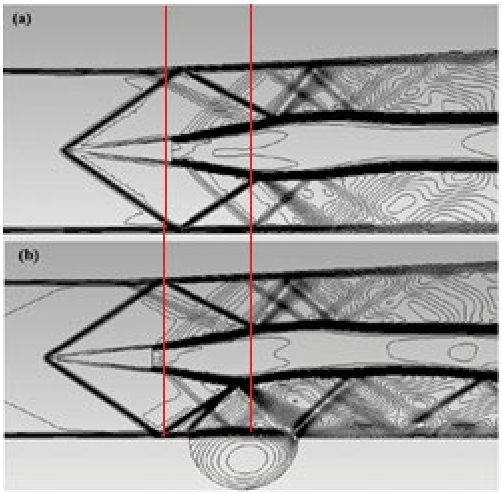


Fig. 14. Density distribution: (a) without hole; (b) with hole

4. Conclusion

In this work, numerical simulations of the injection of hydrogen fuel into a supersonic airstream are performed with and without the combustion progress. The simulation results are similar incredibly to the experimental measurements. The numerical approach is capable of computing chaotic diffuse flames in complex geometries, has good advantages, and gives reliable results. The results once again prove the validation and the variety in choosing boundary conditions of simulation model in this work. On the other hand, a rounded cavity in Scramjet combustion chamber has more mixing zones, shock waves, and vortex zones and all of this has improved the timing and mixing of the fuel with the air. The stability and efficiency are enhanced as well. From all figures and graphs, it is observed that pressure and temperature in the vicinity of the circular cavity and the boundary layer separation are increased.

From all the results and evaluations, it can be concluded that the performance of the scramjet combustion chamber with the circular cavity has been increased compared to the standard scramjet model without the cavity.

Further work should be focused to improve the physical model: three-dimensional effects and different fire models. The effect of cavity addition such as aerodynamic efficiency, additional mass effect or material structure analysis will be investigated in the future as well.

Acknowledgment

This study is funded by Hanoi University of Science and Technology (HUST) under grant numbers T2021-TT-009 and T2021-PC-039.

References

[1] P. P. B. Araújo, M. V. S. Pereira, G. S. Marinho, J. F. A. Martos, and P. G. P. Toro, Optimization of scramjet inlet based on temperature and Mach number of supersonic combustion, Aerospace Science and Technology, vol. 116, 2021 https://doi.org/10.1016/j.ast.2021.106864.

[2] O. R. Kummitha and K. M. Pandey, Hydrogen fueled scramjet combustor with a wavy-wall double strut fuel injector, Fuel, 304, 2021 https://doi.org/10.1016/j.fuel.2021.121425

[3] G. Choubey and K. M. Pandey, Effect of variation of angle of attack on the performance of two-strut scramjet combustor, International Journal of Hydrogen Energy, vol. 41, no. 26, pp. 11455–11470, 2016, https://doi.org/10.1016/j.ijhydene.2016.04.048.

[4] O. R. Kummitha, K. M. Pandey, and R. Gupta, Numerical analysis of hydrogen fueled scramjet combustor with innovative designs of strut injector, International Journal of Hydrogen Energy, vol. 45, no. 25, pp. 13659–13671, 2020 https://doi.org/10.1016/j.ijhydene.2018.04.067.

[5] A. Ben-Yakar and R. K. Hanson, Cavity flameholders for ignition and flame stabilization in scramjets: Review and experimental study, In 34th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, American Institute of Aeronautics and Astronautics Inc, AIAA, 1998, https://doi.org/10.2514/6.1998-3122.

[6] Z. Huang H. Zhang, Numerical investigations of mixed supersonic and subsonic combustion modes in a model combustor, International Journal of Hydrogen Energy, vol. 45, no. 1, pp. 1045–1060, 2020 https://doi.org/10.1016/j.ijhydene.2019.10.193.

[7] W. Shen, Y. Huang, Y. You, and L. Yi, Characteristics of reaction zone in a dual-mode scramjet combustor during mode transitions, Aerospace Science and Technology, vol. 99, 2020 https://doi.org/10.1016/j.ast.2020.105779.

[8] L. Abu-Farah, O. J. Haidn, and H. P. Kau, Numerical simulations of single and multi-staged injection of H2 in a supersonic scramjet combustor, Propulsion and Power Research, vol. 3, no. 4, pp. 175–186, 2014, https://doi.org/10.1016/j.jppr.2014.12.001.

[9] B. Liu, G. Q. He, F. Qin, J. An, S. Wang, and L. Shi, Investigation of influence of detailed chemical kinetics mechanisms for hydrogen on supersonic combustion using large eddy simulation, International Journal of Hydrogen Energy, vol. 44, no. 10, pp. 5007–5019, 2019 https://doi.org/10.1016/j.ijhydene.2019.01.005.

[10] M. R. Gruber, R. A. Baurle, T. Mathur, and K. Y. Hsu, Fundamental studies of cavity-based flameholder concepts for supersonic combustors, Journal of Propulsion and Power, vol. 17, no. 1, pp. 146–153, 2001 https://doi.org/10.2514/2.5720.

[11] ANSYS Fluent-19.1, 2018, ANSYS Inc.

[12] W. Waidmann, F. Alff, U. Brummund, M. Böhm, W. Clauss, and M. Oschwald, Experimental investigation of the combustion process in a supersonic combustion ramjet (SCRAMJET) Combustion Chamber, in: DGLR-Jahrestagung 1994; 04. - 07.10.1994; Erlangen.