Simulations of a Cavity Based Two-Dimensional Scramjet Model

F. Xing[1], M.M. Zhao[2] and S. Zhang[3]

[1]Department of Aeronautics
Xiamen University Xiamen 361005, China

[2]School of Mechanical and Mining Engineering
The University of Queensland, Brisbane 4072, Australia

[2]School of Aeronautics and Astronautics
Zhejiang University, Hangzhou 310027, China

Abstract

This paper presents a computational fluid dynamics (CFD) study of non-reacting and reacting flows within a scramjet model, and for the latter the flow domain is fueled with liquid fuel and operated at shock tunnel flow conditions. This scramjet model includes an isolator, a combustion chamber and a diffuser duct. The liquid fuel is injected through lower surfaces before the cavity flame-holder. The primary goal of the study is to evaluate the detailed cavity based flame stabilization. Those CFD-predicted wall pressure distributions with available experimental data firstly to evaluate the simulation model, thus to understand underlying combusting flow physics. For the "combustion off" cases, as the distance of the corner plate increase, the boundary layer thickness of the downstream from the rear wall of the cavity increase; the entrainment air flow into the cavity and the recirculation zones in the cavity also vary with this distance changes. For the "combustion on" cases, the distance affects the position of the fuel rich regions and where the air and fuel mixture reactions occur. Despite short penetration depth of fuel injection from the simulation, the desired cavity flame stabilization has been partially achieved in the cavity flow field. The cavity flame stabilization provides a mechanism by which combustion can be achieved with mild intake compressions, which leads to greater intake efficiency (with less total pressure loss) and overall greater scramjet performance. Future work will continue to focus on the combustion instability optimized by exploring various types of cavity and corner plate configuration and cavity-based fuel injection system.

Introduction

High-speed flight in the Earth's atmosphere has many applications for transport, defence, and space access. In order to avoid carrying a large amount of on-board oxidizer, one recent trend of hypersonic flight vehicle design was the development and application of supersonic combustion ramjet (Scramjet), a variant of ramjet air breathing jet engine, in which combustion takes place at supersonic airflow speed [1]. There are several advantages of applying this engine type; e.g. flow remaining in supersonic speed and having lower static temperature and pressure even after the diffuser, reduction of dissociation problems as the gases being expanded in the engine exhaust, and reduced diffuser losses, etc. However, high velocity flow inside the scramjet combustion chamber often poses great challenges for the air/fuel mixing and the combustion progress in desirable length scales, due to complex shock-shock, shock/boundary-layer and shock-flame interactions.

Fuel injection, ignition, and flame holding are challenging issues in designing a scramjet engine. A stable flame-holding system for a wide range of operating conditions is critical to the engine performance. Various flame-holding techniques have been developed for supersonic combustors and their features were reviewed in Ref. [2]. Cavity-based flame holder, an integrated fuel injection/flame-holding approach, has lately attracted considerable attention due to its characteristics of low total-pressure loss and fuel/air mixing enhancement.

The presence of a cavity on an aerodynamic surface could have a large impact on the flow surrounding it. The flow field inside a cavity is characterized by recirculating flow that increases the residence time of the fluid entering the cavity. Because that the drag associated with flow separation is much less over a cavity than for a bluff-body, a cavity inside a combustor makes a stable flame holder with relatively little pressure drop. Researchers also suggested that cavity flow oscillations can actually be used to provide enhanced mixing in supersonic shear layers. The mixing was enhanced by the acoustic disturbance and the rate of the enhancement was controlled by cavity shape while the total pressure loss was negligibly small. However before implementing such techniques, one should carefully consider and evaluate any potential thrust loss and noise generation associated with the technique because of this unsteady nature of wave propagation, the flow may become unstable, and unstable combustion in the combustor can be induced. Several control methods have been proposed to suppress the oscillations in cavity. Among others, a cavity with an angled rear wall was devised to suppress the unsteady nature of the free shear layer by eliminating the generation of traveling shocks inside the cavity [3-4]. The most studies of cavity-based flame-holder have also been done by numerical tools without chemical reaction and experiments [5-8]. However, for a practical application to a supersonic combustion and saving the cost of research, a numerical analysis on the cavity flow for flame holding with chemical reaction is in high demand.

In the present study, two-dimensional scramjet model with a corner plate will be visited by a computational fluid dynamics (CFD) solution based on solving Reynolds-averaged Navier-Stokes equation with turbulence and combustion models. Both grid sensitivity and turbulent model assessment will be carried out and comparisons of surface pressure distribution with the experimental data will be made. In particular, three different plate height cases will be analysed in a systematic manner and CFD predicted static pressure, temperature, fuel concentration and heat release will be compared. The primary goal of the present research is to investigate how the height of the corner plate would affect the progress of the combustion, thus to assess the configuration for high-speed scramjet applications.

Model and Simulation Description

Fig.1 gives a sketch of a full scale Scramjet model. The model has two vertical sidewalls and upper and lower walls. It has a longitudinal length of 1975 mm and a span wise width of 75 mm at the inlet and the outlet planes of a rectangular duct, respectively. The isolator section is 625 mm long and the height of the inlet is 54.5 mm. The deflection angle of the diffuse section is 2°. There are two basically models with and without the corner plate. The distance from the corner plate to the lower wall is varies from 2mm, 4mm to 6mm, and all the cases are named Case A (without a corner plate), Case B-1, Case B-2 and Case B-3 respectively. After the isolator section, there is 206 mm long combustion chamber. The liquid fuel is injected from six injection ports that are located on the upper of the cavity. The ports have same cross-section diameter of 1.2 mm and an inclined angle of 90° against the wall surface. Such an injection design will result in a good air/fuel mixing to be completed prior to entrance of the cavity, thus allowing for a shorter combustion chamber.

In the present study, results from steady two-dimensional CFD simulations are going to compare with experimental data from Gruber et al. [5] for "Combustion off" (non-combustion considered cases) by using a smaller geometry cavity model. Cavity of the model with depth of 8.9 mm were used for experiment for the conditions of *L/D*=3, and with the after angle 30°.
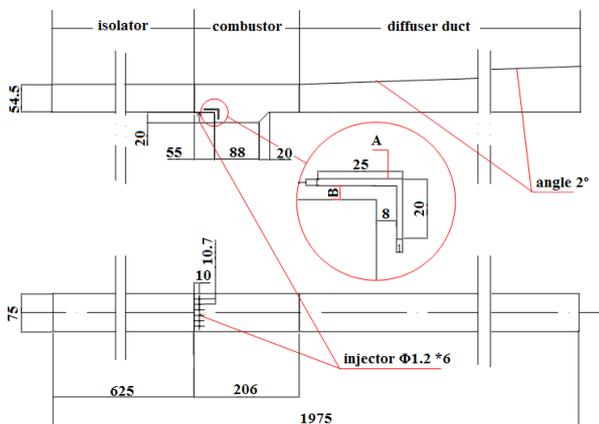
Fig.1. Schematic description of the scramjet model
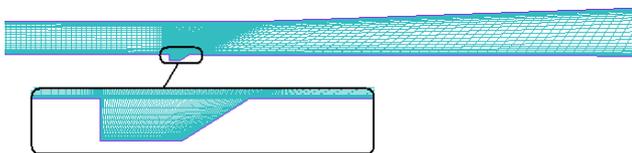
Fig.2. Sample multi-block structured mesh with close views at cavity area

A commercial CFD software ANSYS-CFX [9] is applied, that contains various sub-models to simulate turbulent combustion phenomena. Several key elements will be explored about the capabilities of the software in simulating shock/boundary-layer and shock-shock interactions, mass, momentum and heat transfer transport characteristics between injected fuel and mainstream supersonic air flow, and turbulent combustion.

For the "Combustion off" case, the absence of fuel jets means that the 3-D effects presented only in corner regions, due the existence of the side wall boundary layers. Therefore, following common practice, 2-D simulation of a mid-plane was chosen that permit computational efficient computations for wide range of parametric studies, such as grid refinement, turbulence models, and results could be comparable with those from 3-D model measurements on the centreline plane. After finishing the work of

grid sensitivity, a real size model would be used for both "Combustion off" and "Combustion on" cases. Fig.2 shows the computational mesh used in the calculations. The mesh was generated by using commercial software ICEM.

Grid Sensitivity and Validation

The flow conditions for 2-D simulation of grid sensitivity and turbulence models are incoming Mach number $(M\infty)$ of 3, static pressure $(P\infty)$ of 690 KPa, and static temperature $(T\infty)$ of 300K. These parameters are taken from the experiments measurement [5]. The boundary conditions are a uniform supersonic inflow at the inlet plane, and supersonic outflow conditions at the outlet plane. The top and bottom surfaces use no-slip condition and an adiabatic wall condition. For steady state simulation, convection term is discredited with the second order scheme, and the simulation initialize by using inlet condition [10]. Menter's shear stress transport (SST) turbulence model [11] is used in the grid sensitivity studies, which is better suitable for flow separation modelling.

A total of 3 grids were generated with different number of grid points along the stream-wise and the wall-normal as shown in Fig.3 below. Therefore, based on incoming flow conditions and grid size of the first grid next to the wall surface, different y+ value can be estimated to ensure the validity of applying SST model. In addition, the development of wall boundary layer in terms of its thickness is carefully estimated and used to resolve the wall boundary layers and the shock reflections at the wall.

In Fig.3, the effective distance comprises the cavity upstream forward wall from a separation corner, the cavity bottom and the cavity rear wall. Fig.3 shows the wall pressure distributions for *L/D*=3, and with the after angle 30°. Because of less grid points used in the wall-normal and the stream-wise, the results of Test 150*45 (y+: 80) did not agree very well with the experimental data. The simulations did not accurately capture the reflection of the shock after interacting with the boundary layer. A good agreement is observed for the computed and test results for Test 300*90 (y+: 8) and Test 600*180 (y+: 0.5). It can be seen that CFD predicts quite promising peak locations, but a little under-predicts the peak values.
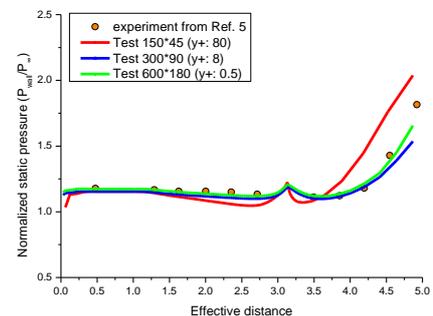
Fig.3. Comparisons of the wall pressure distributions for different meshes

Results and Discussion

Both non-reacting "Combustion off" cases and reacting "Combustion on" cases will be investigated here. The initial and inflow boundary conditions for the simulations are determined using exactly the parameters seen in Tab.1.

	Air		Liquid fuel
M_∞	2	Φ (mm)	1.2
P_∞ (Pa)	77300	ρ (kg/m³)	0.78×10^3
T_∞ (K)	502	T (K)	300
m_∞ (kg/s)	1.9	m (kg/s)	0.102
u_∞ (m/s)	2275	cone angle (°)	20

Tab.1 Scramjet simulation conditions

Combustion off
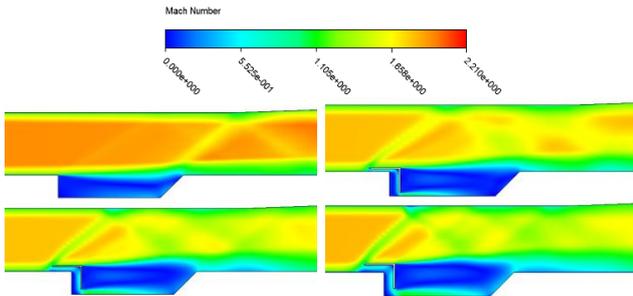
Fig.4.Mid-plane Ma number contours for different cases without injection
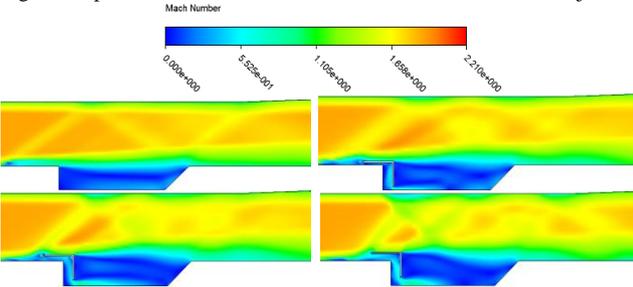
Fig.5. Mid-plane Ma number contours for different cases with injection
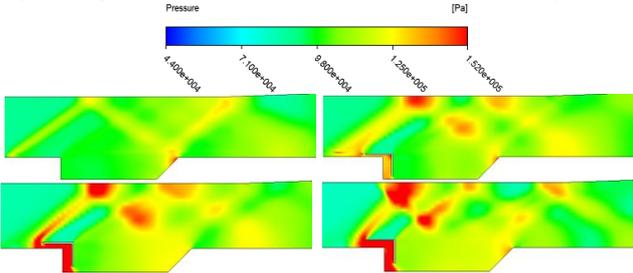
Fig.6. Mid-plane pressure contours for different cases with injection
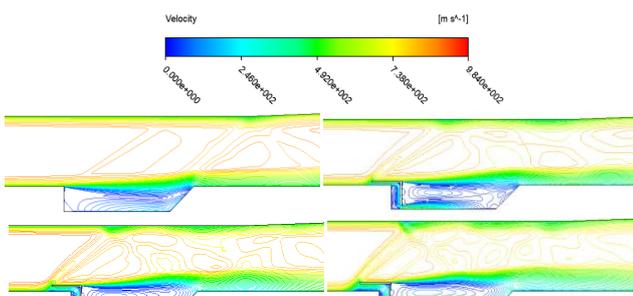
Fig.7. Mid-plane velocity contours for different cases without injection

Fig.4 and Fig.5 are pictures about mid-plane Ma number contours for with and without fuel injection. All the results are lined as Case A, Case B-1, Case B-2 and Case B-3 in order.

It can be seen from Fig.4 that in Case A, shock waves are forming from the forward wall and the rear wall of the cavity. The same things happen to the Case Bs, but the shock waves are first forming from the leading edge of the corner plate instead of the forward wall. The corner plate enhances the oscillation, so as the distance of the corner plate increase, the boundary layer thickness of the downstream from the rear wall of the cavity also increase. In Fig.5, because of high Mach number of inlet airflow, the liquid fuel injected into the transverse flow lacks sufficient momentum to penetrating in-depth into central region. But the liquid fuel adds some momentum to the boundary layer of the downstream from the rear wall of the cavity, and the air flow added to the wall again. Except of the cavity zone, the entire flow field in the scramjet kept supersonic velocity.

In Fig.6, the shocks formed and refracted can be seen more clearly. The shocks contact at the boundary layer of the upper wall and refract. These refractions continue and produce a train of gradually weakening alternate shocks and expansion waves all the way down to the diffuser duct. Fig.6. Mid-plane pressure contours for different cases with injectionFig.7 is mid-plane velocity contours for different cases. For Case A, there is a small amount of the entrainment air flow into the cavity, and there is no obvious recirculation zone in it. For Case Bs, because of the function of the corner plate, the air flow into the cavity through the "tunnel" formed by the corner plate, and produce recirculation zones in the cavities.

Moreover, the velocity profiles are better organized. In the centre regions of the cavities, the velocity of the air is small, and these regions are of critical importance because they apply sufficient time and room for the air and the fuel mixing and reacting.

Combustion on
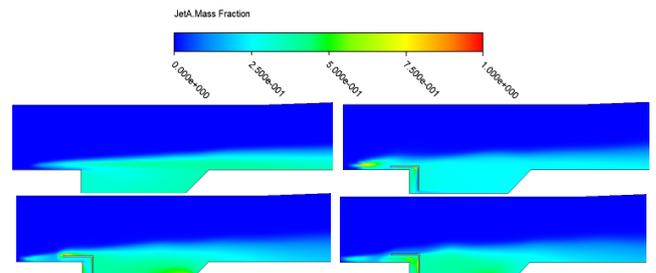
Fig.8. Mid-plane fuel mass fraction contours without combustion
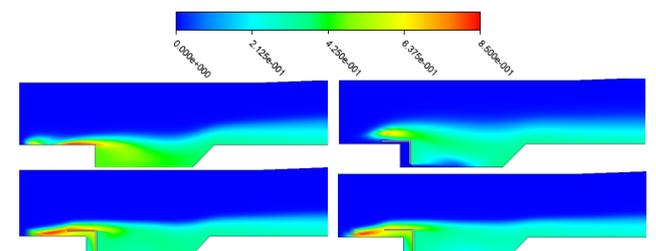
Fig.9. Mid-plane fuel mass fraction contours with combustion
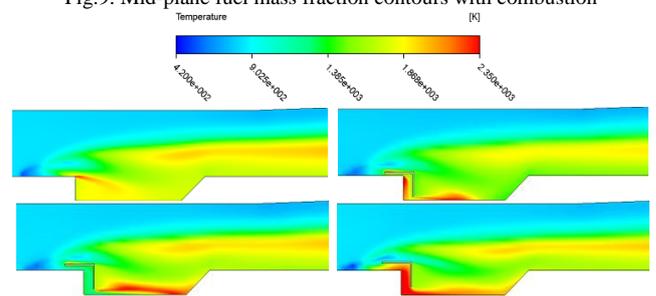
Fig.10. Mid-plane temperature contours for different cases

Fig.8 and Fig.9 are mid-plane fuel fraction contours for different cases without and with combustion. Because of high Mach number of inlet airflow, the liquid fuel injected into the transverse flow lacks sufficient momentum to penetrating in-depth into central region.

The CFD predictions showed that the fuel are almost immediately turned toward the wall surfaces and flow along it, spreading and merging with each other in the span-wise direction later. The air/fuel mixing seems to happen in the cavity and downstream for Case Bs such that the fuel fractions remain high only within a short distance from the cavity rear walls for the entire length of the scramjet. From the pictures, we can also see that the high fuel concentration regions in the combustor. For Case A, the fuel is rich in the cavity, but most of the fuel is

spread with the air stream along the lower wall direction. For Case B-1, because of the distance between the corner plate and the lower wall not big enough, the fuel could not flow through the "tunnel", so some of them concentrate at the leading edge of the corner plate. For Case B-2 and Case B-3, the fuel richest zones distribute at the bottom and the corner of the cavities. For Case B-3, because there is enough room between the lower wall and the corner plate, the fuel also gathers just in the "tunnel".

Fig.10 gives the mid-plane temperature contours. As the flow reaches the cavity regions where air and fuel mix and pressure increase above critical values, the combustion process would be reactivated. But it should be noticed that the high temperature zones are not only found inside the cavity region for Case B-2, but also in the boundary-layer near the lower walls for Case A and Case B-1. The temperature of these zones will exceed 2000 K. It can be seen in Fig.10 that the increase of distance between lower wall and the corner plate will result in combustion zones with high temperature in the cavity and the "tunnel". There is high pressure in these regions and well air and fuel mixture; hence reaction will occur in these regions as well.

Conclusions

Two-dimensional calculations of reactive flow fields within four different scramjet models incorporating liquid fuel injection into the inlet air stream have been performed. The results show that the corner plate plays an important role for these models operating. As the distance of the corner plate increase, the boundary layer thickness of the downstream from the rear wall of the cavity increase; the entrainment air flow into the cavity and the recirculation zones in the cavity vary with this distance. For the combustion simulations, the distance also affects the position of the fuel rich regions and where the air and fuel mixture reactions occur.

Based on the results and the information, the future work will continue focus on investigating using the cavity leading to the pressure loss and a counter-balancing effect of the pressure loss; the combustion instability optimized by exploring various types of cavity and corner plate configuration and cavity-based fuel injection system.

Acknowledgments

References

[1] E. T. Curran, Scramjet Engines: The First Forty Years, Journal of Propulsion and Power, (2001) Vol. 17, No. 6, pp. 1138-1148.

[2] A. B. Yakar, and R. K. Hanson, Cavity Flame-Holders for Ignition and Flame Stabilization in Scramjets: An Overview Journal of Propulsion and Power, (2001) Vol. 17, No. 4, pp. 869-877.

[3] A. B. Yakar, and R. K. Hanson, Cavity flameholders for ignition and flame stabilization in scramjets: review and experimental study, 39th AIAA Aerospace Sciences Meeting & Exhibit, (1998) January 8-11; Reno, USA.

[4] X. Zhang, Compressible cavity flow of oscillation due to shear layer instabilities and pressure feedback, AIAA Journal, (1995) Vol. 33, No. 8, pp. 1404–1411.

[5] 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, (2001) Vol. 17, No. 1, pp. 146–153.

[6] C. C. Rasmussen, J. F. Driscoll, K. Y. Hsu and etc. Stability Limits of Cavity-Stabilized Flames in Supersonic Flow, Proceedings of Combustion Institute, (2005) Vol. 30, pp. 2825-2833.

[7] J. Liu, C. Tam, T. Lu and C. K. Law, Simulations of Cavity-stabilized Flames in Supersonic Flows Using Reduced Chemical Kinetic Mechanisms. 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, (2006) July9-12, Sacramento, California.

[8] T .A. Barber, B. A. Maicke and J. Majdalani, Current State of High Speed Propulsion: Gaps, Obstacles, and Technological Challenges in Hypersonic Applications. 45th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, (2009) August 2 - 5, Denver, NV.

[9] http://www.ansys.com/products/fluid-dynamics/cfx/

[10] F. Thive, D. D. Knight, A. A. Zheltovodov, and A. I. Maksimov, Analysis of Observed and Computed Crossing Shock-Wave/Turbulent-Boundary-Layer Interactions. Aerospace Science and Technology, (2002) Vol. 6 No. 1, pp. 3-17.

[11] F. R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications, AIAA Journal, (1994) Vol. 32, No. 8, pp. 1598 – 1605.