FLOW OSCILLATIONS IN SUPERSONIC CAVITY- INDUCED COMBUSTION

Rajarshi Das
Department of Mechanical Engineering
Andong National University
Andong, Republic of Korea
das75@anuis.andong.ac.kr

Aran Ye
Department of Mechanical Engineering
Andong National University
Andong, Republic of Korea
aranye@anuis.andong.ac.kr

Toshiaki Setoguchi
Institute of Ocean Energy
Saga University
Saga, Japan
setoguci@me.saga-u.ac.jp

Heuy Dong Kim
Department of Mechanical Engineering
Andong National University
Andong, Republic of Korea
kimhd@andong.ac.kr

ABSTRACT

Pressure oscillations induced by cavities in a Mach 2.08 flowfield under different thermal conditions is investigated. The flow takes place in a confined three-dimensional rectangular duct where cavities of L/D ratios 2 and 5 are mounted on one of the walls. Length of the cavity is kept constant at 50mm for all the test cases. Pressure oscillations are measured from numerically obtained unsteady flow data. A comparative study on flow oscillations under cold and hot thermal conditions is carried out during the course of the work. Unsteady shear layer structure over the cavity is also investigated and an attempt has been made to relate this unsteady motion to the flow oscillations.

INTRODUCTION

Rapid and efficient fuel air mixing is an important requirement of a supersonic combustion ramjet. Entrainment of secondary jet into the primary flow largely affects distribution of fuel inside the combustor and is the initial step in forming a uniformly distributed combustible mixture. Unsteadiness in the flow field produced by emission of acoustic oscillations from wall mounted cavities has been proposed as an effective tool to enhance entrainment and fuel air mixing. To this end, extensive work on various cavity shapes and designs have been carried out by various groups.

Most of the related flow investigations have been performed using hydrogen as the preferred fuel. Two emerging scramjet applications namely (i) hydrogen fueled engine to access space, and (ii) hydrocarbon fueled engines for air launch missiles have been identified in a review by Curran (2001). For optimal combustion, the spread of fuel into the supersonic jet and creation of a proper combustible fuel-air mixture is essential. Yu and Schadow (1994) were the first to propose cavity based mixing phenomena who observed that spreading of cavity actuated mixing layer is much higher than spreading of ordinary mixing layer for the same velocity and density ratios. Wall mounted cavities of different L/D ratios were observed to effect mixing of two supersonic streams within a short mixing tube (Mingbo et al., 2009).

Ebrahimi and Gaitonde (2007) investigated the overall combustion efficiencies of a cavity based combustor with different fuel injection locations. Their results revealed that injector located on the ramp of the cavity wall gave the maximum combustion efficiency. Transverse injection of fuel in the upstream region of the cavity produced bow shock in the supersonic mainstream thereby leading to severe pressure losses affecting net thrust produced by the engine. The allied fluid dynamic phenomenon was explained by Yakar and Hanson (1998). Angled injection of fuel at upstream of the cavity was also probed but the shock related pressure losses were found to exist. With the advent of powerful computers and robust numerical algorithm, CFD nowadays is complimenting difficult to perform experiments and is playing a very major role in development and design of scramjet combustors through analysis of various thermo-chemical parameters obtained from numerical simulations (Chitsomboon and Northam, 1991, Kim, 1995). Despite many years of research, large number of issues related to description of supersonic turbulent reactive flows still remains unanswered. One such important aspect is the shift in the flow oscillation pattern during combustion from that occurring during cold flow through the cavity based combustor. Similar flow unsteadiness during combustion was investigated by Choi et al. numerically (Choi et al., 2005). They observed that the flow unsteadiness during combustion bears close resemblance to that for non-reacting flows. Experimental investigation by Hong Bo et al. (2013) revealed that the dominant oscillation frequencies shifted to higher frequencies during combustion.

In the present study, oscillations characteristics of two cavity based combustors for both cold and hot flow conditions have been explored numerically. Hydrogen is injected as fuel through an injector located at the trailing edge wall of the cavity at a fuel-air equivalence ratio of 0.05. The structure of the shear layer over the cavity has also been investigated through numerical visualization. An attempt has been made to correlate the shear layer structure with the pressure oscillations under different thermal conditions.

TEST SETUP

One-half portion of the combustor model is shown in Fig.1, cut into half along its length which is used as the flow domain for the numerical simulations. The length of the cavity is L=50mm which serves as the non-dimensional parameter for the other flow domain dimensions. The cavity is located 2L=100mm downstream of the flow inlet region of the test section. An injector having a diameter of 0.5mm is located 3mm below the channel floor on the trailing edge wall of the cavity with its centre coinciding with the longitudinal symmetric plane of the test section.

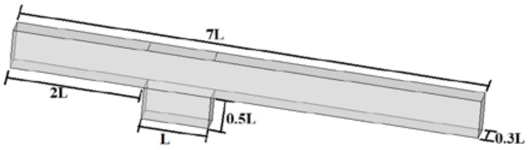


Fig.1 Test setup

Two cavities, each of length L=50mm and depth D=25mm and 10mm are used for the investigation thereby giving L/D ratios of 2 and 5. The inlet conditions are numerically simulated such that a M=2.08 flow is established in the channel.

The flow domain was created using commercially available software ICEM-CFD. The domain was meshed into finely structured hexahedral meshes with 2.195 million mesh elements. Clustering of mesh was done near to the wall to give a y+~1 for the steady converged solutions.

NUMERICAL METHODOLOGY

The flowfield inside the test section is numerically simulated using commercially available software FLUENT v 15.0. Three dimensional, steady and unsteady density based Navier Stokes formulation is applied for the solution combined with the species transport model. The coupled formulation of density based solver is used for its applicability in compressible high Mach number flows. Pressure inlet and pressure outlet are chosen as boundary conditions at inlet and exit of the flow domain from the various options available in the software. Non-slip walls and symmetry boundary conditions are imposed at the other regions. A single step finite rate chemistry following reversible reaction between fuel and oxidizer is used to model the chemical interaction between fuel and air. Connaire et al. (2004) observed that with addition of turbulence-chemistry interaction, there is dramatic increase in the turbulent diffusivity throughout the flame region and different chemical schemes do not alter the mixing and combustion process significantly. For the current study a finite rate eddy-dissipation model is adopted to define the turbulence-chemistry interaction. Through the species model, inlet air is described to constitute 0.77% nitrogen and 0.23% oxygen. This oxygen serves as the oxidizer for the fuel injected. Spatial discretization was achieved by second order upwind scheme.

The mass flow rate of the mainstream and the secondary fuel injection is controlled by specifying total and static pressure at the inlet and injection location. Previous investigations have indicated that the finite rate single step chemistry model performs reasonably well in predicting the overall mixing process in the combustor (Chandra Murthy et al., 2010). The mixing rate is determined from the eddy dissipation model (EDM) based on the density and mass fractions of the fuel, oxidizer and products. The total temperature of the entire flowfield was kept at 300K for the cold flow cases and was increased up to 1800K for the combustion cases in accordance to the general procedure adopted by earlier researchers (Chandra Murthy et al., 2010). The fuel inlet total temperature was kept constant at 300K throughout the course of this investigation.

Turbulence closure is achieved by means of Wilcox k-ω SST model for the steady flow initially. The SST model has been observed to provide good prediction for mixing layers and jet flows and is lees sensitive to initial values in numerical simulations (Chandra Murthy et al., 2010). Smagorinsky-Lily large eddy simulation (LES) model is used for the unsteady cases. This model, although consumes large computational resources, is preferred over other models for high end applications including mixing and combustion.

For the unsteady simulation, a time step of $\Delta t=10^{-7}$s is implemented for the transient formulation. A steady case was run initially for which convergence of mass flow imbalance was monitored at the inlet, injection and outlet locations. After the mass flow imbalance fell below the order of 10^{-5} the unsteady formulation was implemented. The wall y+ value at a location 10mm upstream of the cavity was observed to be ~1 for the converged steady solutions.

The pressure oscillations within the cavity are investigated using numerical tools. Time resolved static pressure is measured at the trailing edge wall of the cavity at a location 2mm below the bottom wall of the flow channel. The pressure data is sampled over a time of 0.05s for every time step spanning $\Delta t=10^{-7}$s. The periodic fluctuations in the flow field is illustrated in these unsteady pressure data. The data is then transformed into an amplitude vs frequency domain by fast fourier transform method. Commercially available software MATLAB is used for this transformation. A Hamming window with 50% overlap is used for a window length of 2048 samples thereby giving a transform length of 4096 samples. Thus the frequency resolution obtained after processing is 48.82Hz. Repeated runs were carried out and the experimental uncertainty in frequency resolution of the various modes was observed to be within ±100Hz over a span of 6runs of the solution. The shear layer over the cavity that governs the pressure oscillations for cavities with L/D>1 is numerically visualized based on the flow vorticity in the test domain. These are shown and discussed in subsequent sections.

Verification of numerical scheme

The overall approach to this investigation has been validated against a number of steady and unsteady flow problems including combustion in similar flow domain. First, a grid independence study is carried out for different grid sizes in the test domain. The static pressure variation at the centre of cavity floor along its length for the L/D=2 cavity at various grid sizes used is shown in Fig.2 for the steady cases. It is observed that there is not much difference between the pressures predicted by the three grids chosen for computation. However, depending on the time required for computation, the intermediate grid size was chosen for the analysis.

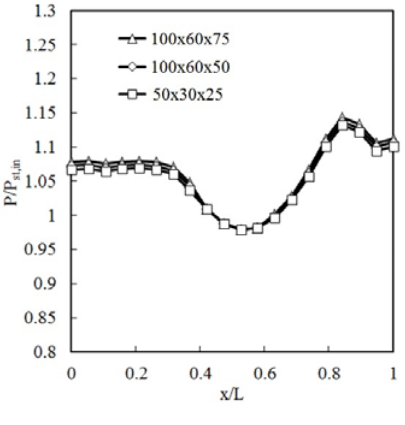


Fig.2 Static pressure variation at cavity floor for different grids

A validation study for the combustion model adopted for this study and as discussed previously was done. The model test domain used is similar to the combustor configurations of Burrow and Karkov's (1973) experimental study. Researchers have simulated the flow in this combustor previously as test cases for their numerical study (Chandra Murthy et al., 2010). The static temperature at the exit of the combustor obtained at the exit of the combustor for the present study is compared with that obtained by previous researcher and is shown in Fig.3. The numerically calculated static temperature at the combustor exit during the validation study shows good match with results obtained by other researchers. This helps to validate the various schemes used to model the combustion inside the combustion chamber for the present study.

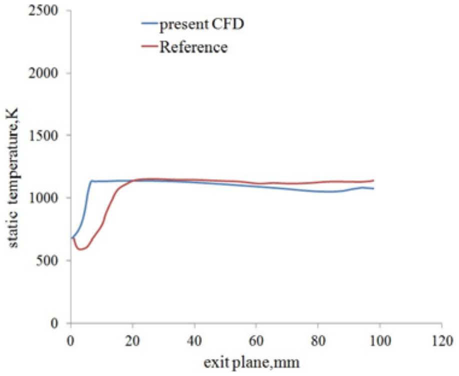


Fig.3 Static temperature variation at exit of combustor used for validation study

The unsteady scheme used for the analysis is verified from the pressure oscillation data of the L/D=2 cavity. The frequencies of the various modes of oscillations obtained from the pressure spectrum is compared with the theoretical values predicted by Rossiter's modified formula-

$$Stn = \frac{m-\xi}{\left\{\dfrac{M_\infty}{\left[1+(\gamma-1)M_\infty^2/2\right]^{1/2}}+\dfrac{1}{K}\right\}} \qquad (1)$$

where,

L – length of the cavity, V_∞ – free stream velocity, m – mode number (1,2,3,…), M_∞ – flow Mach number, K – ratio of convective velocity of vortices to the freestream velocity and ξ – a factor to account for time lag between the passage of a vortex and emission of acoustic pulses at the trailing edge.

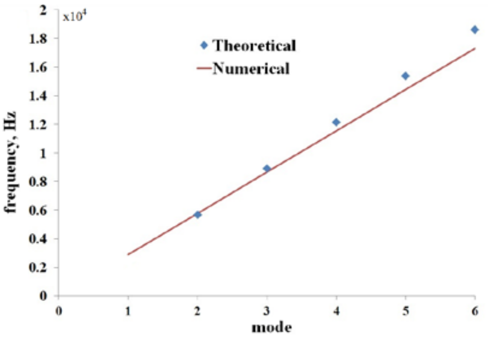


Fig.4 Mode comparison for pressure oscillations

Values of empirical constants ξ and 1/K were evaluated through best fit of measured data as $\xi = 0.25$ and $1/K = 1.75$. These values were observed to depend on the geometric aspects of the cavity (Heller et al., 1971). A comparative plot of the theoretically and numerically obtained mode frequencies is shown in Fig.4. Frequencies of the lower modes are observed to match well with each other. The pressure oscillations of the L/D=2 cavity shown in Fig.4 corresponds to cold flow analysis numerically. After these verifications the numerical scheme has been applied to the present investigations.

RESULTS AND DISCUSSION

The pressure oscillations obtained from the unsteady flow analysis is presented in a transformed PSD versus frequency domain in this section. The cold flow unsteady pressure data for the L/D=2 and L/D=5 cavities are shown in Figs. 5(a) and 5(b). From the power spectrum plot for the L/D=2 cavity it is observed that there are multiple modes of oscillations, one of them being dominant. The frequency of the dominant mode is observed to be 5.86 kHz corresponding to the second Rossiter's mode. Frequencies of the different modes were within ±100Hz of the analytical data obtained for similar flow conditions and geometrical aspects of the test channel. Other high frequency, low amplitude modes were observed which might be generated by the large scale vortical structures interacting with the trailing edge of the cavity. The flapping of the shear layer which causes periodic inflow and outflow of fluid into the cavity and from it is oscillates mostly at the dominant frequency.

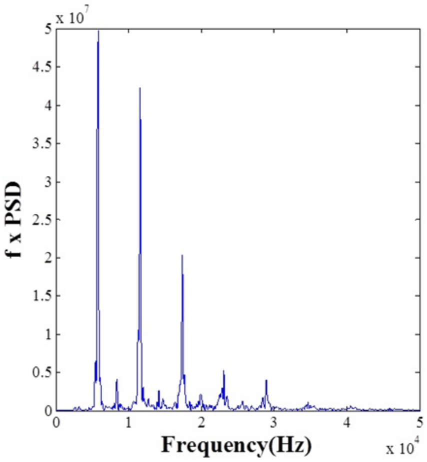


Fig.5(a) PSD of pressure oscillations of L/D=2 cavity

Pressure spectra of the L/D=5 cavity at cold flow conditions shows 2 discrete peaks as is seen in Fig.5(b). The first peak occurs at 5.37 kHz and the second peak which is the dominant one occurs at a frequency of 12.94 kHz. This corresponds approximately to the second and the fourth mode predicted by eqn(1). As the length of the cavity remains same the oscillation frequencies observed for both the models are almost same.

The pressure data recorded during combustion is shown in similar PSD versus frequency domain for the 2 cavities as is shown in Figs.6(a) and 6(b). More broadband type oscillations at high amplitudes are observed from the pressure spectra. The pressure level for the dominant mode is observed to be much lower during combustion as compared to the cold flow case. This is observed for both the cavity models. However, the other high amplitude modes are observed during the combustion process.

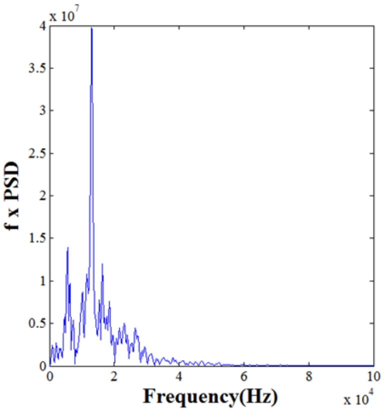


Fig.5(b) PSD of pressure oscillations of L/D=5 cavity

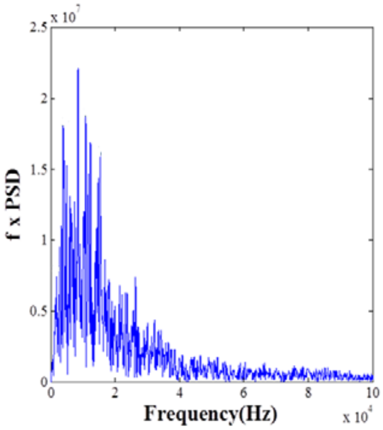


Fig.6(a) PSD of pressure oscillations of L/D=2 cavity during combustion

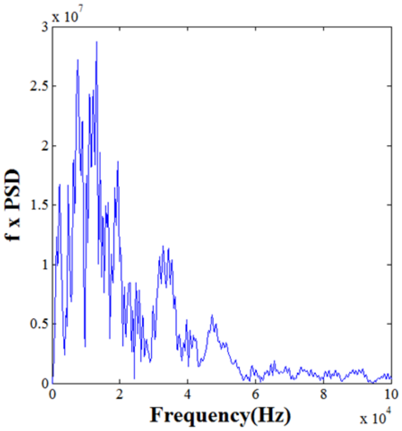


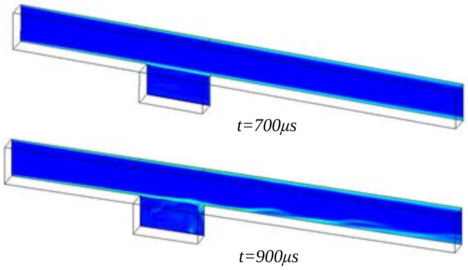
Fig.6(b) PSD of pressure oscillations of L/D=5 cavity during combustion

During cold flow, the pressure oscillations are characterized by discrete peaks, whereas during combustion broadband oscillations at higher amplitude are observed. Prominent mode frequencies observed during both cold and hot flow are summarized in Table 1. The oscillation energy is distributed and the overall energy is much higher during combustion. This is due to the increased pressure and energy level in the flow during combustion. The change in oscillation mode from discrete peaks to more of broadband type is further studied on the basis of the shear layer structure over the cavity during the two processes. As the shear layer is defined by vorticity, the contour of vorticity is numerically visualized along the axial symmetric plane of the flow domain.

Table 1. Mode frequencies.

L/D=2(Hz)		L/D=5(Hz)	
cold	hot	cold	hot
5865	3616	5379	2192
11626	8318	12949	4781
17490	10669	16335	7769
23251	15370	18526	10757

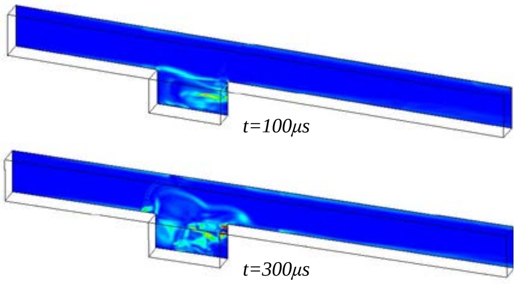
The shear layer structure over the L/D=2 cavity in terms of vorticity magnitude is shown in Figs. 7(a) and 7(b) at different time instants. The colour bar is not shown as the only goal of this visualization is to see the shear layer structure over the cavity. During cold flow the shear layer structure is observed to be almost similar at different times. An entirely different phenomena is observed during combustion process when the shear layer almost breaks down over the cavity.



$t=700\mu s$

$t=900\mu s$

Fig.7(a) Contours of vorticity for L/D=2 cavity under cold flow conditions



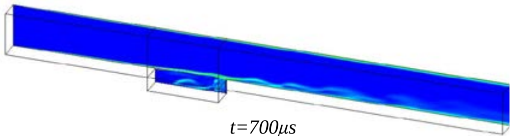
$t=100\mu s$

$t=300\mu s$

Fig.7(b) Contours of vorticity for L/D=2 cavity under hot flow conditions

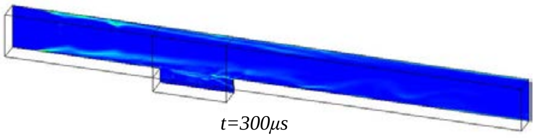
The shear layer over the L/D=5 cavity during cold flow process is observed to span over the entire cavity length as is observed in Fig.8(a). During combustion the boundary layer is observed to separate much upsteam of the cavity as is observed from Fig.8(b). However, no thermal choking was observed and the inlet Mach number remained at 2.08 over the entire time. No proper structure of the shear layer is observed for the L/D=5 cavity during combustion.

The discrete pressure oscillations observed during the cold flow process is not observed during combustion for both the cavity models. The breakdown in shear layer structure may be causing this.



$t=700\mu s$

Fig.8(a) Contours of vorticity for L/D=5 cavity under cold flow conditions



$t=300\mu s$

Fig.8(b) Contours of vorticity for L/D=5 cavity during combustion

CONCLUSIONS

Unsteady flow field in a model cavity based scramjet combustor is investigated. The results show that, under different thermal conditions, the pressure oscillation characteristics in the flow field changes. The pressure oscillations are defined by discrete modes during cold flow. Broadband, high amplitude modes are observed during combustion. The acoustic energy is observed to be higher and spread over several oscillation modes during combustion. Substantial variation in shear layer structure during these two processes may be causing these changes in oscillation characteristics.

REFERENCES

Curran, E.T., 2001, “Scramjet Engines: The First Forty Years,” *Journal of Propulsion and Power*, Vol.17, No.6, pp.1138-1148.

Yu, K.H. and Schadow, K.C., 1994, “Cavity Actuated Supersonic Mixing and Combustion Controls,” *Journal of Combustion and Flame*, No-99, pp. 295- 301.

Ming-bo, S., Geng, H., Jian, L. and Wang. Z., 2009, “Mixing Characteristics in a Supersonic Combustor with Gaseous Fuel Injection Upstream of a Cavity Flameholder,” *Journal of Flow Turbulence and Combustion*, Vol. 8, pp. 271-286.

Ebrahimi, H. and Gaitonde, D., 2007, “Parametric Study of 3D Hydrocarbon Scramjet engine with Cavity,”*AIAA-200-645*, pp.1-6.

Yakar, A.B. and Hanson, R.K., 1998, "Cavity Flameholders for Ignition and Flame Stabilization for Scramjets: An Overview," *Journal of Propulsion and Power*, Vol. 17, No.4, pp. 869-877.

Chitsomboon,T. and Northam, G.B., 1991, "Computational Fluid Dynamics Prediction of the Reacting Flowfield inside a subscale Scramjet Combustor," *Journal of Propulsion and Power*, Vol. 7, No.1, pp. 44-48.

Kim, S.W., 1995, "Numerical Investigation of Chemical Reaction-Turbulence Interaction in Compressible Shear Layer," *Journal of Combustion & Flame*, No. 101, pp. 197- 208.

Choi, J.Y., Ma, F.and Yang, V., 2005, "Combustion Oscillations in a Scramjet engine Combustor with Transverse Fuel Injection," *Proceedings of Combustion Institute* 30 (20), pp. 2851-2858.

HongBo, W., Zhen, G.W., MingBo, S.and HaiYan, W., 2013, "Nonlinear Analysis of Combustion Oscillations in an Cavity based Supersonic Combustor," *Science China Technological Sciences*, Vol-56, No-5, pp.1093-1101.

Connaire, M.O., Curran, H.J. Simmie, J.M., Pitz, W.J. and Westbrook, C.K., 2004, "A Comprehensive Modeling Study of Hydrogen Oxidation,“ *International Journal of Chemical Kinetics*, Vol. 36 , No. 11, pp. 603-622.

Chandra Murty, M.S.R., Mishal, R.D. and Chakraborty, D., 2010, "Numerical Simulation of Supersonic Combustion with Parallel Injection of Hydrogen Fuel," *Journal of Defense Science*, Vol. 60, No. 5, pp. 465-475.

Burrows, M.C. and Kurkov, A.P., 1973, "Analytical and Experimental Study of Supersonic Combustion of Hydrogen in Vitiated Airstream," *Report No. NASA-TMX-2828*.

Heller, H.H, Holmes, D.G, Covert, E.E., 1971, "Flow Induced Pressure Oscillations in Shallow Cavities," *Journal of Sound and Vibration*, Vol. 18, No. 4, pp. 545-553.