INVESTIGATION OF SUPERSONIC COMBUSTION WITH CAVITY BASED INJECTION IN A SCRAMJET COMBUSTOR

J. Sandeep*

K. Bhardwajan**

Dr.M.Y.Ali***

Dr.PV.Raman Murti****

T.Tirupathi*****

ABSTRACT

CFD analysis of supersonic combustion of air with hydrogen fuel has been performed for a scramjet engine. As the combustion is taking place at supersonic speeds, the flow has very less residence time (milliseconds) in the combustor. The Primary objective of this analysis is to improve residence time, thereby increasing fuel-air mixing and combustion efficiency. The eddies or vortices generated in the cavity acts as a flameholder and increases the residence time of flow. The two-dimensional coupled implicit Navier Stokes equations, reailizable k-ε turbulence model and the finite-rate/eddy-dissipation reaction model have been applied to numerically simulate flow field of the hydrogen fueled scramjet combustor with a cavity flameholder under two different working conditions, namely, cold flow and engine ignition. Hydrogen and H_2O *mass fractions left at the outlet are considered as complete fuel-air mixing and combustion efficiency.*

Ke Terms: *Aft ramp angle, Vorticity, Flame holding.*

*Assistant Professor, Department of Aeronautical Engineering, Malla Reddy College of Engineering and Technology, Maissamaguda, Dhulapally, Secunderabad, AP.

**Associate Professor, Department of Aeronautical Engineering, Institute of Aeronautical Engineering, Dundigal, Hyderabad, AP.

***Principal, Geethanjali Institute of Science and Technology, Hyderabad.

****Principal and Director, Bandari Srinivas Institute of Technology, AP.

***** Head of the Department, Dept of Aeronautical Engineering, Malla Reddy College of Engineering and Technology, Maissamaguda, Dhulapally, Secunderabad AP.

1. INTRODUCTION

Over the past 40 years considerable effort has been directed to the development of an air-breathing engine which can give a speed of Mach 6 and above. A most viable engine to be studied in this regard is a SCRAMJET (supersonic combustion RAMJET) which is an extension of ramjet and the marked distinction between these two engines is in with respect to the flow states inside the engine. Scramjet allows the flow through engine to be supersonic, whereas in ramjet the flow is slowed to subsonic levels before it enters the combustor. Ramjet engines are optimal for Mach of 3-6. But after Mach 6, slowing the flow to subsonic levels dissociates the oxygen and nitrogen gases, thus causing combustor entrance temperature and pressure to become too high. Combustion in dissociated flow is extremely inefficient because the heat released by exothermic combustion reactions is negated by the heat absorbed through endothermic dissociation reactions.

The available time for fuel and air in the combustor is limited to the order of 1 millisecond, which makes fuel-air mixing and maintaining supersonic combustion a difficult task. Consequently, realization of this engine is strongly linked to effective design of fuel injection, enhanced fuel–air mixing, and combustion process in the combustor. Good fuel-air mixing can be achieved by creating a recirculation area where the fuel and air can be mixed partially at low velocities.

Wei Huang[1] proposed concluded k-ε turbulence model and finite-rate/eddy-dissipation reaction model can exactly simulate flow field of a hydrogen fueled scramjet combustor with cavity flameholder. M.R.Gruber et al [2] cited that shear layer from the leading edge reattaches at the end of cavity trailing edge, thus generating an oblique shock. The high pressure and temperature region behind this shock can also act as an ignition source. Ben Yakar and Hanson[3] observed cavities which include aft ramp angle (between θ = 45 and 16 deg) have minimum drag penalties. Further investigation is therefore required to design an effective and effective cavity for supersonic flame holding, ensuring systematic study of cavities both in nonreacting and reacting flows and their interaction with fuel jets. Hyungseok Seo et al [4] inferred grater vorticity magnitude indicates stronger rotation thereby allowing flowfield to provide better mixing for air and fuel. Increasing cavity sizes (within a limited range) increases vorticity and enhances fuel-air mixing.

All previous scramjet combustion work discussed in literature survey included low hypersonic flight Mach numbers with an combustor inlet Mach number ($1<M<3$) where scramjet operation is observed. This provided motivation for the present study at flight Mach numbers of magnitudes 10 and above, whose inlet Mach number is between 3-5. At such high speeds hydrogen will be the only fuel suitable for fast reactions (as discussed in the combustion phenomenon chapter).

Essentially, current numerical work focuses to analysis using CFD of cavity based supersonic combustor for efficient fuel air mixing and combustion and performance, evaluation is carried out for non reacting flows and reacting flows.

2. DESCRIPTION OF PHYSICAL MODEL AND CFD METHOD

The geometry is foreseeded with an isolator for 0.22 m and a staged step of 0.0032 m on the upper wall with 0.096 m as combustor length, followed a divergent section with 1.7^0 angles spread over 0.35 m. Hydrogen is injected from a slot of width 0.001 m at 0.0128 m distance from the end of isolator. Geometry modeling and meshing is obtained using a preprocessing tool (GAMBIT).

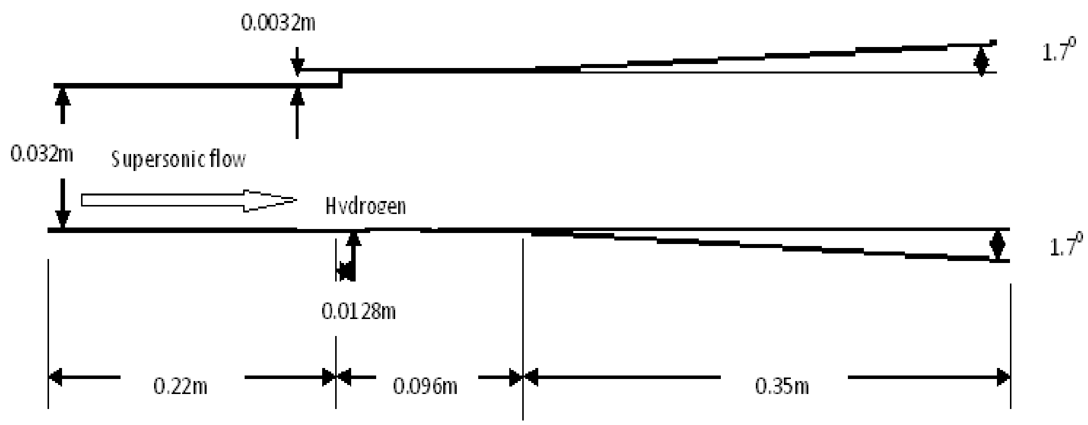


Fig 1: Geometry of Scramjet Combustor

Cavity geometries are characterized by its length to depth (L/D) ratio. A cavity is termed open if the ratio is less than 10. For L/D<10 these ratios, the free shear layer will reattach to the rear face of the cavity. For L/D >10 cavity is known as closed because the shear layer will reattach on the cavity floor. Generally cavities are designated as LD3-01-30. Here LD3 indicates length to depth ratio as 3, 01 indicates offset ratio (OR) and 30 indicates aft ramp angle (θ) of the cavity as shown in fig2 below.

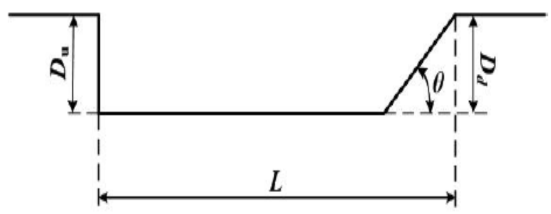


Fig 2: Cavity nomemclature

In the present model D_u= 0.0032 is mainlined constant and offset ratio (OR) = D_u / D_d = 1 is also maintained constant but L / D ratio and ramp angle are varied for a better design.

2.1 Grid Generation:

As hydrogen is injected at the middle of combustor geometry and because the flow is essentially supersonic then more variations in flow properties, formation of a series of shock and expansion waves occur. So the mesh should be highly dense after the isolator section. Structured mesh is (preferred for better captivation of the flow properties and shock waves emanating from within the cavity and therefore) made with 60880 elements.

2.2 Inlet Boundary Conditions:

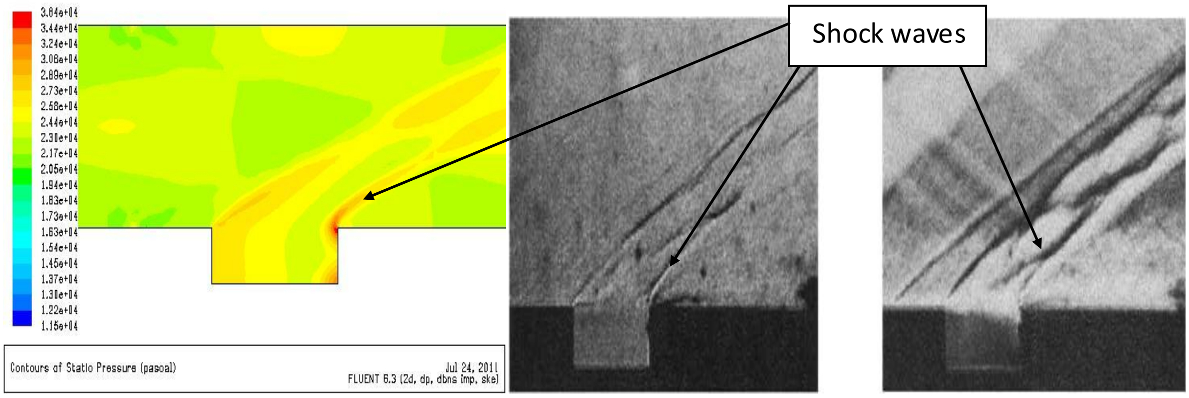
PARAMETERS	INJECTOR	FREESTREAM FLOW
Mach number	1	4.5
Temperature (K)	1000	1300
Pressure (Pa)	506625	101325
Mass fraction of Hydrogen	1	0
Mass fraction of Oxygen	0	0.23
Mass fraction of Nitrogen	0	0.77

Table 1: Input conditions

For turbulence modeling in the CFD model, the realizable k-ε model is choosen. This is because of its robustness and its ability to strongly predict turbulence, recirculation region and vortices. Further, because of intense turbulent combustion, finite-rate/eddy-dissipation reaction model is adopted for the chemistry. The finite-rate/eddy-dissipation is based on hypothesis of infinitely fast reactions and the reaction rate is controlled by turbulent mixing. Both the Arrhenius and mixing rates are calculated and smaller of two rates is used for the turbulent combustion. While no-slip conditions are applied along the wall, because the flow being supersonic, at outflow all regions physical variables are extrapolated from the internal cells.

3. MODEL VALIDATION

In order to validate accuracy of the turbulence model used, a cavity flow is applied and compared with the experimental work done by Gruber et al [2], which further shows unison with the formation of shockwaves with respect to location and intensity as well.



(a)Shadowgraph photograph **(b)Schlieren photograph**

Fig 3: Comparision of CFD Pressure contours of non reacting flow in cavity with Experimental photographs by Gruber et al[2]

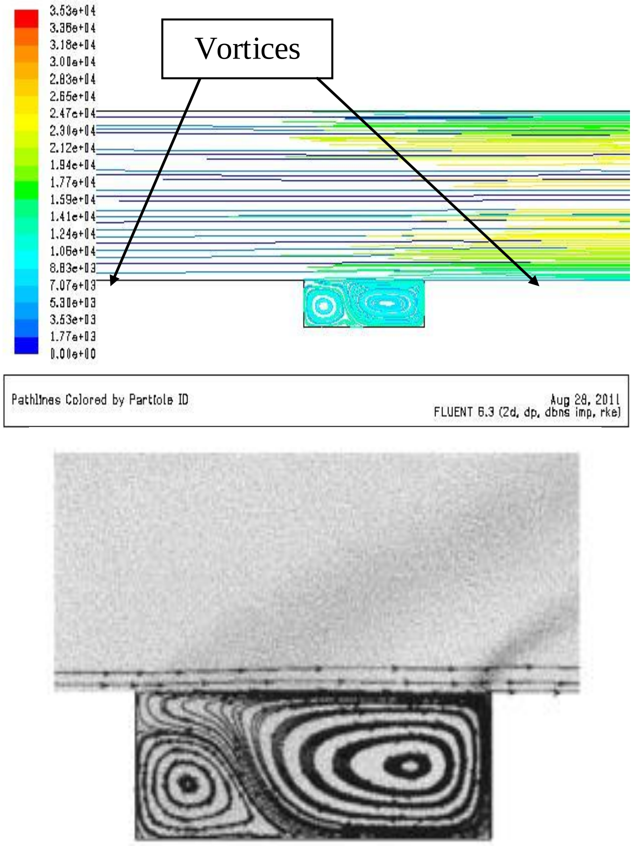


Fig 4: Vortices by CFD and experiment

Validation with experimental results obtained by Gruber et al [2] were conducted with a Mach 3 facility nozzle operated at a stagnation pressure of 690 KPa for all the cavity models. By these we can validated that realizable k-ε turbulence model predicts the shock and expansion waves as shown in Fig 3, vortices in the cavities as shown in Fig 4

4. RESULTS AND DISCUSSION

4.1 Non-Reacting flows:

Solving flow, energy and species equations with reactions disabled is known as cold flow or non-reacting flow. Here the basic flow pattern formation of shock waves and vortices due to geometric changes can be observed by pressure contours and pathline contours where in mixing intensity of fuel-air can be understood.

Vorticity is used as a measure of extent of mixing taking place inside the combustor. It represents the strength of rotation along each axial direction. Greater magnitude of vorticity indicates stronger rotation of flow field which enhances air-fuel mixing.

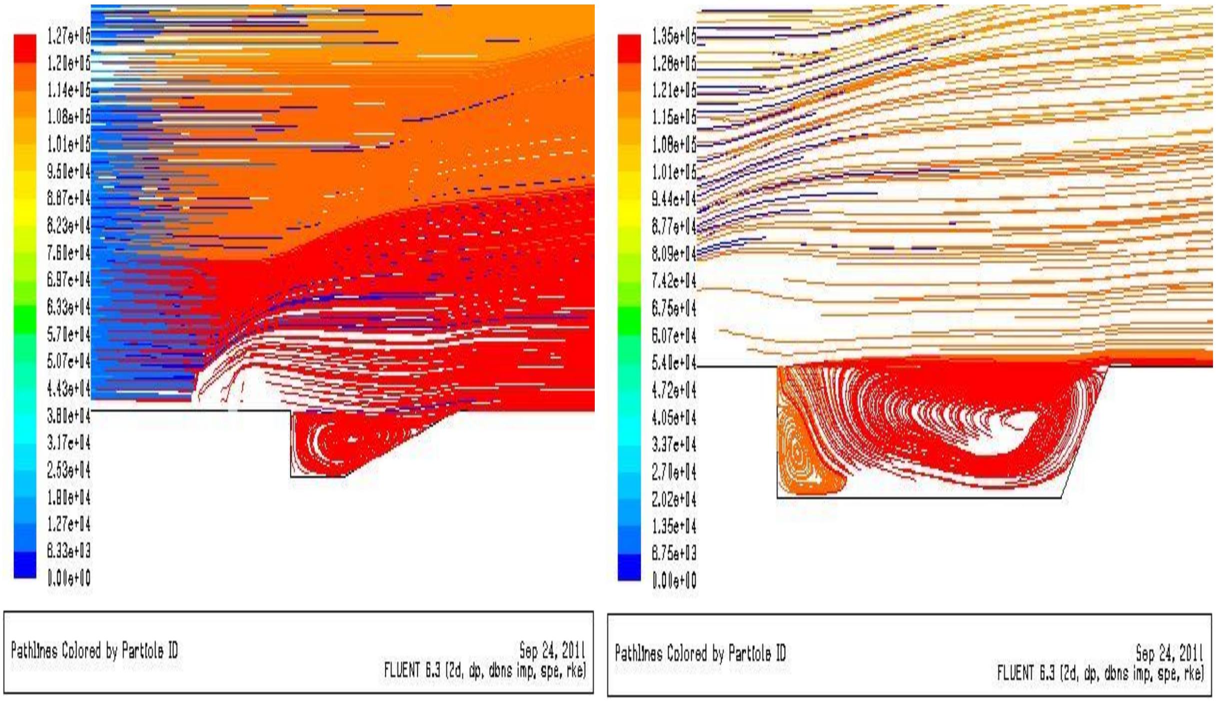
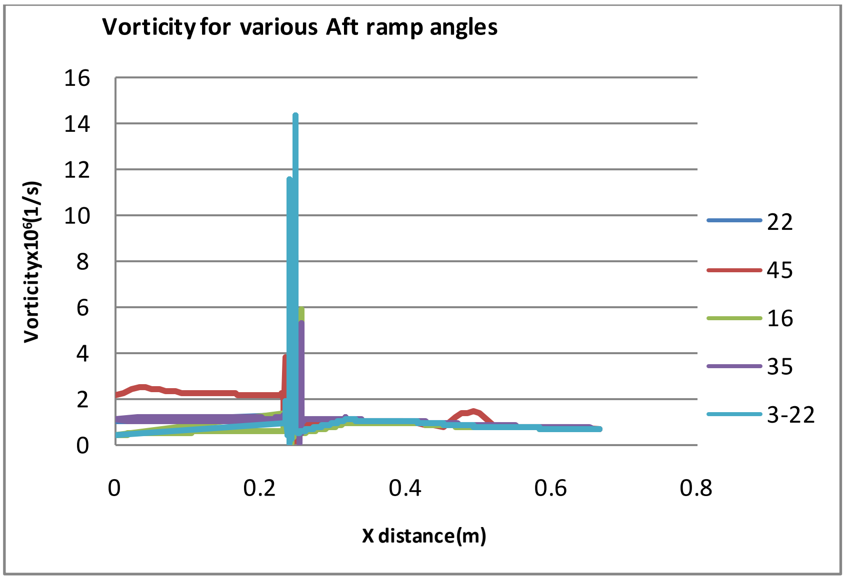


Fig 5: Pathlines of flow with cavity 16^0 ramp angle and cavity with 45^0

Cavities enhance the fuel-air mixing by formation of recirculation zones. From the analysis in the above figures two vortices (one large scale and other small scale) have formed in the cavity. As the aft wall ramp angle is increasing two vortices are formed, one at the leading edge of cavity and other at trailing edge of cavity as shown in Fig 5. A single large scale vortex induces more vorticity than two smaller vortices.



Graph 1: Graph of vorticity variation with cavity aft ramp angles

In Graph the first four curves are with cavities of L/D=5 and the maximum vorticity is attained for a cavity with 22^0 aft ramp angle because the shock occurred at the trailing edge of cavity is stronger than any other shocks. The last fifth curve is obtained for a cavity with L/D=3 and aft ramp angle 22^0. Shorter cavities create a single large vortex such that more vorticity than larger cavities.

Table 2: Residence time of flow in combustor

S.no	Cavity	Residence time (milli sec)
1	L/D=5, 16^0	0.39
2	L/D=5, 22^0	7.4
3	L/D=5, 25^0	4.3
4	L/D=5, 35^0	1.7
5	L/D=5, 45^0	0.67
6	L/D=3, 22^0	10.63

Vorticity is not the only factor for good fuel-air mixing; even available residence time for air and fuel should be high. The vortices in L/D=3 and L/D=5 causes a large scale difference in residence time between rectangular cavities. Decreasing length of the cavity decreases its volume correspondingly i.e. decreasing residence time, but the mass exchange rate is also decreased thus increasing residence time as observed from Table 2 above.

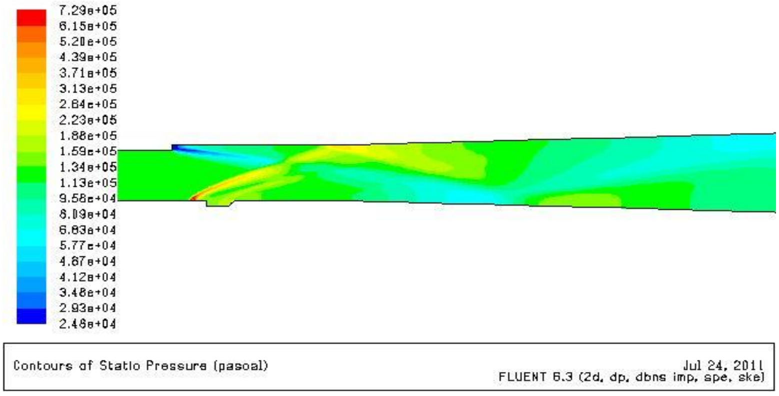


Fig 6: Pressure contour for non reacting flow in combustor with cavity 45^0

From Fig 6 presence of cavity and injection of fuel, both contribute to formation of a structure of oblique shock waves in the scramjet duct. Shocks occurring at the fuel injection point and cavity trailing edge induce a vertical velocity component, affecting the downstream region which increases the vorticity. Intersection of these shocks and reflected shocks with expansion wave from the step of top wall increases the vorticity of the flow. As vorticity increases a good fuel-air mixing can occur.

4.2 Reacting flow:

Higher combustion efficiency means a higher percentage of the injected fuel undergoes combustion resulting in higher magnitudes of static temperature was at exit. Presence of H_2O indicates occurrence of combustion. Total Pressure loss represents the losses in the combustor

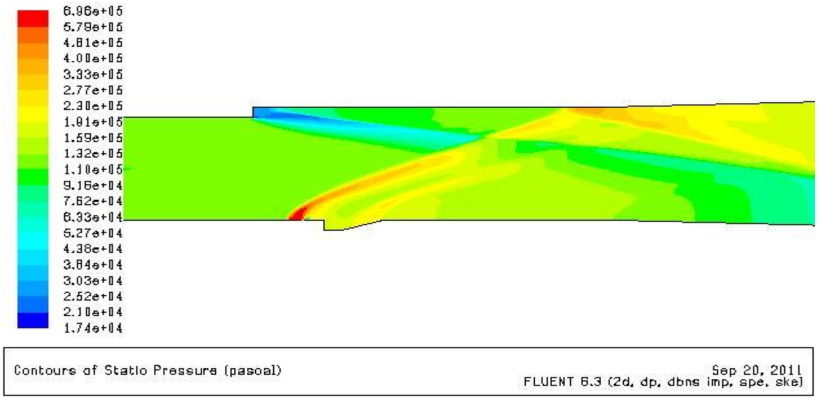
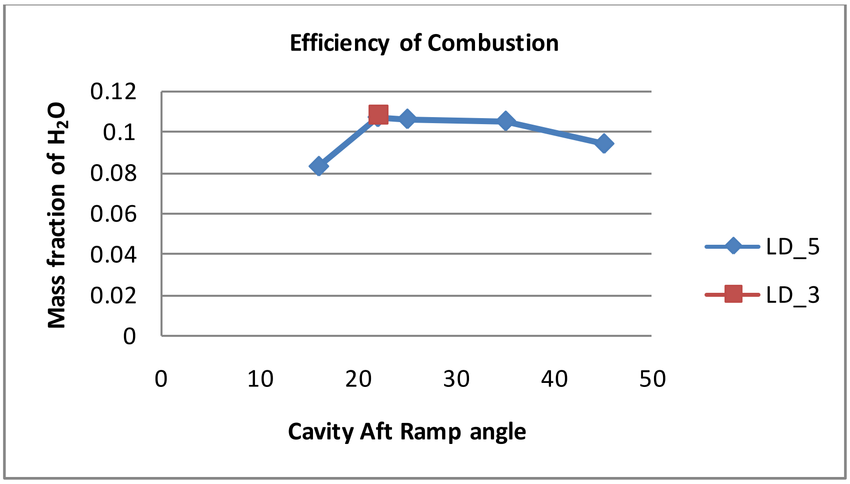


Fig 7: Pressure contour for reacting flow in combustor with cavity 16^0

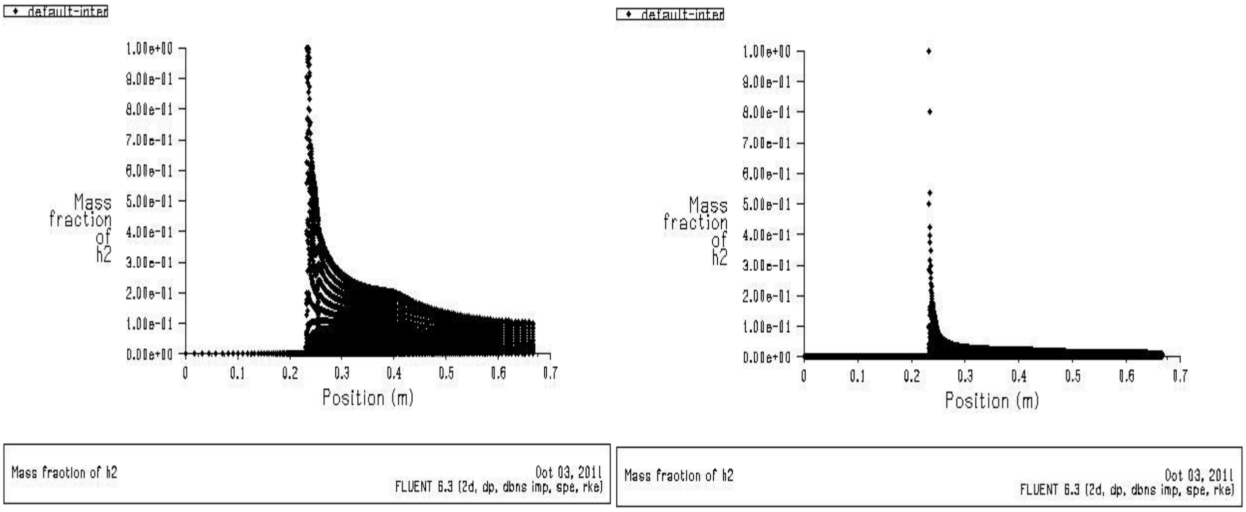
A Compression wave formed at the fuel injection slot results in increased pressure and temperature measurements in the cavity and leads to autoigniton. From the Fig 7, pressure contours indicate that strong shocks are formed at the leading edge and trailing edge which acts as main source for ignition.

The most likely location for initial combustion reaction to occur is just above the cavity at the main air/fuel interface where the temperature measurements obtained have not reduced as much as in the cavity. Cavity not alone supports in the mixing process but also allows in fast proceeding of combustion reactions to an extent of providing continual radicals to main flow. These radicals would remain chemically frozen until they reach the region of higher temperature where in combustion occurs.



Graph 2: variation of Mass fraction of H_2O for combustor with cavity varying rampangles

From Graph 2 mass fraction of H_2O is maximum for cavity with L/D=3 and 22^0 ramp angle. The trailing edge shock formed is strong which increases the pressure to 2.7 MPa leading to high combustion efficiency. And also recirculation region in the cavity causes local temperature gradients to be quite high, because the local velocities are low and kinetic energy of main stream is already fully converted into thermal energy.



Graph 3: Mass fraction of H_2 for combustor with cavity L/D=5 and 22^0 and cavity L/D=3 and 22^0

Observations from Graph 3 [mass fraction of H_2] for cavities with L/D=5, ramp angle 22^0 and L/D=3, ramp angle 22^0 indicates that only 0.001 mass fraction has not been reacted in the cavity with L/D=3 rather than 0.1 mass fraction in L/D=5. Thus citation suggest that combustion efficiency of cavity with L/D=3 is greater than cavity with L/D=5.

5. CONCLUSIONS

The following conclusions are drawn based on the simulations performed:

- As the aft ramp angle is decreased in cavity vorticity increased due to the trailing edge shocks and at 22^0 maximum vorticity is observed.
- Compared to L / D=5 cavity, vorticity is more in cavity with L/D=3. This is due to the formation of a single large scale vortex in the cavity with improvement in residence time for fuel-air.
- Because fuel is injected from within the walls of combustor, the fuel cannot mix with the core flow of combustor and leads to poor combustion efficiency.
- The total pressure losses are more due to strong shock formed at transverse injection of fuel locations and shocks emanating at the cavity trailing edges.

6. SCOPE FOR FUTURE WORK

The present work is restricted for two dimensional models of the scramjet and involoving single step reaction models. A suitable combustor design and standard equivalence ratio is to be determined with minimum pressure losses then we can extend the project for three dimensional models. To improve combustion efficiency floor injection can also be proposed thereby full scale combustion can be achieved.

BIBLIOGRAPHY

1. Wei Huang, Shi-bin Luo, Mohamed Pourkashanian, Lin Ma, Derek B.Ingham, Jun Liu and Zhen-guo Wang "Numerical simulations of a Typical Hydrogen Fueled Scramjet Combustor with a cavity Flameholder" WCE 2010, June 30-july2, 2010, London, U.K.
2. Gruber MR, Baurle RA, Mathur T and Hsu KY. Fundamental studies of cavity-based flameholder concepts for supersonic combustors. Journal of Propulsion and power, 17(1): 146-153, 2001.
3. Wei Huang, Zhen-guo Wang, Shi-bin Luo and Jun Liu "Effect of cavity flame holder configuration on combustion flow field performance of integrated hypersonic vehicle" Sci China Tech Sci, 53:pp. 2725-2733, 2010.
4. Adela Ben-Yakar and Ronald K. Hanson "Cavity Flame-Holders for Ignition and Flame Stabilization in Scramjets: An Overview" JOURNAL OF PROPULSION AND POWER Vol. 17, No. 4, Aug 2001
5. Hyungseok Seo, Hui Jeong, Jaewoo Lee and Yunghwan BYUN " A Study of the mixing Characteristics for cavity sizes in Scramjet combustor" Journal of the Korean Physical Society, Vol. 55, No. 5, pp. 2180~2186, Nov 2009.
6. Eunju Jeong, Sean O'Byrne, In-seuck Jeung and A.F.P. Houwing "Investigation of Supersonic combustion with Angled injection in a Cavity-Based Combustor" JOURNAL OF PROPULSION AND POWER Vol.24, No. 6, Nov-Dec 2008.