CFD Analysis of Wall Injection with Large Sized Cavity Based Scramjet Combustion at Mach 2

K. M. Pandey, P Kalita , K Barman, A. Rajkhowa and S.N.Saikia

***Abstract*—A CFD analysis of the combustion process of a scramjet engine having wall injector with cavity with a L/D ratio of 5 is carried out in this present study using FLUENT software. Both air intake and H2 injection are at Mach 2 speed and hydrogen is being injected upstream of the cavity. It is observed that a maximum temperature of 2100K can be achieved with the injection of H2 at Mach 2 speed with high thrust production and low shock formation.**

***Index Terms*—Wall injector, Mach number, Scramjet, static temperature, static pressure.**

I. Introduction

The scramjet engine is one of the most promising air-breathing propulsive systems for future hypersonic vehicles, and it has drawn the attention of an ever increasing number of researchers. The mixing and diffusive combustion of fuel and air in conventional scramjet engines take place simultaneously in the scramjet combustor. However, the incoming supersonic flow can remain in the combustor only for a very short time, i.e. for the order of milliseconds, and this restricts the further design of the scramjet engine. The presence of a cavity on an aerodynamic surface could have a large impact on the air flow surrounding A cavity wall injector is an integrated fuel injection approach, and it is a new concept for flame holding and stabilization in supersonic combustors and this makes a large difference to the performance of the engine, namely it may improve the combustion efficiency and increase the drag force.

A. Cavity Flame holders

Another fuel injection system uses a backward-facing step to induce recirculation, with fuel injected upstream of this cavity. This cavity would also provide a continuous ignition point or flame holder with little pressure drop, and hence sustained combustion. The advantage is that the drag associated with flow separation is less over a cavity than over a bluff body. The two main disadvantages are the losses in stagnation pressure due to this step, as well a reduction in total temperature. Also, the wall injection method limits the penetration of the fuel into the airflow. This means that a broad application of this method is not possible, since the ignition heavily depends on the Mach number. An injection with a cavity set up is shown in Figure 1.

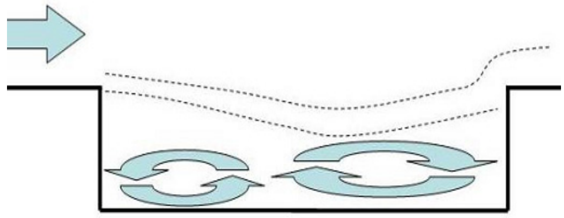

Figure 1: Rectangular cavity flame holder

With a cavity installed downstream of the fuel injection point, it was observed that the mixing efficiency as well as the combustion was greatly improved, since the mass and heat movement along the shear layer and inside the cavity are greatly increased. The depth of the cavity determines the ignition time based on the free stream conditions, while the length of the cavity has to be chosen to sustain a suitable vortex to provide sufficient mixing inside the cavity. There needs to be sufficient time for the injected fuel and free stream air to mix and ignite. An increase in the wall angle of the cavity produces greater combustion efficiency, but also a greater total pressure loss. It is also to be noted that if the injector is comparatively far from the leading edge of the cavity, the cavity forms small vortices because the mixture entering the cavity is insufficient. However, if the injector is relatively close to the cavity, the injected fuel does not penetrate into the free stream due to the flow turning into the cavity.

B. Cavity-Pylon Flame holder

Intrusive devices can enhance the interaction between a cavity-based flame holder and a fuel-air mixture in the core flow [13]. A pylon placed at the leading edge of the cavity provides such a mechanism by increasing the mass exchange between the cavity and free stream [12] and improving mixing due to pylon vortex/shock interactions [13]. Low pressure behind the pylon draws fluid out of the higher pressure cavity and into the main flow which leads to increased mass exchange between the cavity and main flow compared to a cavity-only case [14,15] (see Figure 2. Supersonic expansion at the pylon edges, results in low pressure behind the pylon .The pressure differential between the cavity and pylon base should result in a flow of cavity fluid upward behind the pylon. This Upward flow will lie between a pair of stream wise counter-rotating vortices that form as the flow over the top of the pylon spills over each side. The vortices generated by a ramp fuel injector produce a similar effect. This additional stream wise vorticity should enhance mixing of the fluid behind the pylon and the main flow.

Manuscript received July 25, 2010

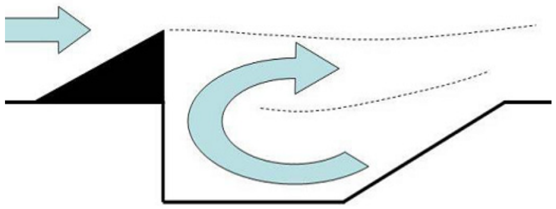

Figure 2. Cavity flame holder with inclined downstream ramp and leading edge pylon (on centerline)

In this analysis, the two-dimensional coupled implicit Reynolds Averaged Navier-Stokes (RANS) equations, the standard *k-ε* turbulence model and the eddy-dissipation reaction model have been employed to investigate the flow field in a hydrogen-fuelled scramjet .combustor with a cavity with L/D=5 and to analyze the combustion processes. The injector design also must produce rapid mixing and combustion of the fuel and air. Rapid mixing and combustion allow the combustor length and weight to be minimized, and they provide the heat release for conversion to thrust by the engine nozzle. The fuel injector distribution in the engine also should result in as uniform a combustor profile as possible entering the nozzle so as to produce an efficient nozzle expansion process.

Wall injection system uses a backward-facing step to induce recirculation, with fuel injected upstream of this cavity. This cavity would also provide a continuous ignition point or flame holder with little pressure drop, and hence sustained combustion. The advantage is that the drag associated with flow separation is less over a cavity than over a bluff body.

With a cavity installed downstream of the fuel injection point, it was observed that the mixing efficiency as well as the combustion was greatly improved, since the mass and heat movement along the shear layer and inside the cavity are greatly increased. The depth of the cavity determines the ignition time based on the free stream conditions, while the length of the cavity has to be chosen to sustain a suitable vortex to provide sufficient mixing inside the cavity. There needs to be sufficient time for the injected fuel and free stream air to mix and ignite. An increase in the aft wall angle of the cavity produces a greater combustion efficiency, but also a greater total pressure loss. It is also to be noted that if the injector is comparatively far from the leading edge of the cavity, the cavity forms small vortices because the mixture entering the cavity is insufficient. However, if the injector is relatively close to the cavity, the injected fuel does not penetrate into the free stream due to the flow turning into the cavity.

II. Literature Review

Scramjets have long been recognized as the most well-suited for hypersonic propulsion. Although a traditional ramjet is most appropriate for supersonic speeds (Mach 3 to 5), hypersonic speeds (Mach 6 to 15) can be reached only with the use of a scramjet, where combustion takes place at supersonic speeds. Because the internal flow in a scramjet is supersonic, the flow has a very short residence time during which air and fuel must mix on a molecular level, and chemical reactions have to be completed before leaving the engine. Moreover, the inlet flow is often accompanied by oblique shocks so that mixing, sustained combustion and flame anchoring become critical. Although some ground and flight experiments have successfully demonstrated the feasibility of supersonic combustion, experimental testing requires a large investment and presents numerous difficulties. Computational tools are thus a key element toward the development of an efficient, high-performance scramjet engine. In a scramjet engine, to enhance flame holding and promote its performance, One of the simplest approaches is to use the backward facing step [1]. In recent years, a cavity flame holder, which is an integrated fuel injection/flame-holding approach, has been proposed as a new concept for flame holding and stabilization in supersonic combustor. It, designed by CIAM (Central Institution of Aviation Motors) in Moscow, was used for the first time in a joint Russian/French dual-mode scramjet flight-test [2]. Experimentally, the use of a cavity after the wall injector was found to significantly improve the hydrocarbon combustion efficiency in a supersonic flow. Similar flame stabilization method, employed by Ben Yakar et al. [3] in a solid-fuel supersonic combustor, demonstrated a self-ignition as well as sustained combustion of polymethyl-methacrylate (PMMA) for supersonic flow conditions. The engine inlet is of prime importance for all air-breathing propulsion systems. Its major function is to collect the atmospheric air at free stream Mach number, slow it down (probably involving a change of direction) and so compress it efficiently. In this role the inlet is performing an essential part of the engine cycle and its efficiency is directly reflected in the engine performance. In addition, the inlet must present the air to the downstream component at the suitable velocities and with an acceptable degree of uniformity of velocity and pressure under any flight condition.

The single hole injector causes a higher penetration and a greater thickness of the mixing jet compared with the twin hole injector. The wall injector effects the lowest penetration and the smallest thickness. When using a single hole injector, the mixing jet is not in contact with the side walls of the combustion chamber. This causes a three-dimensional turbulent air flow structure around the mixing jet, generating a pair of large-scale vortices in the wake of the jet. These vortices deform the initially circular cross-section of the mixing jet into a kidney-shaped cross-section, and thereby enhances the mixing rate due to an increase of turbulence-induced convective mass transfer .There are basically two types of cavity flows viz. open and closed [4,5]. The open flow normally occurs for length to depth ratio, L/D <10 and the closed flow for L/D > 10.

The growth of a mixing layer produces a displacement effect on the surrounding flow field. This displacement in concerned flow produces pressure gradients that can affect the later development of the mixing layer, typically retarding growth. When chemical reaction occurs in a mixing layer, resulting in heat release, the growth of the mixing layer is retarded in both subsonic and supersonic flow [6,7] .The mixing and combustion experiments described earlier were numerically simulated before data was collected to assist in the experimental design. Additional simulations were also performed during and following the experimental study to compare with the measured data. Initial simulations were made with the SPARK combustion code. Additional studies with other combustion codes are being conducted [8,9,10]. There are some other works done in this area of research, which is cited here

K.M.Pandey and Amit Kumar [15] worked on Studies on Base Pressure in Suddenly Expanded Circular Ducts: a Fuzzy Logic Approach and their findings are given below. An optimum L/D ratio is evaluated in the present study using fuzzy-set theory. The fuzzy set based methodology could easily consider many attributes concurrently, while deciding the specifications of the suddenly expanded supersonic fluid flow through a straight circular duct. The methodology can be easily extended to a situation involving diverse conflicting objectives. This study can be extended to different nozzles having different geometries with variations in Mach numbers, primary pressure ratio and area ratio. It is observed that L/D ratio is 6 for base pressure for Mach numbers of 1.58, 1.74, 2.06 and 2.23, which is in very close agreement with the experimental results cited in the literature. This has been discussed with fuzzy logic as a tool for three area ratios 2.89, 6.00 and 10.00. The primary pressure ratio has been varied from 2.10 to 3.48 and L/D ratio has been varied from 1 to 6. From this analysis it is observed that L/D ratio 6 is the optimum needed keeping in view all the parameters like wall static pressure and pressure loss including base pressure.

III. Methodology

There are number of methods to compare different operating designs of wall injector (with cavity) based scramjet combustion. In the present study Computational Fluid Dynamics (CFD) was used to measure the same. For CFD analysis, the profile was made in GAMBIT and suitable boundary conditions were inserted. Two dimensional meshing was done in GAMBIT with suitable spacing. Flow analysis was carried out in FLUENT software. Suitable boundary conditions were defined and some suitable values of input parameter were taken. Iteration is done by taking 500 numbers of iteration and it is plotted. We precede our analysis when the plot got converged. Contours of static pressure, total temperature, mass fractions, kinetic turbulent energy, and x-velocity are seen for the wall of wall injector with the length along the direction of flow. Plots are being drawn between pressure variation and length of wall injector as well as between density variation and length of wall injector.

A. Detailed Design of Model

CFD analysis is done by making a profile in GAMBIT (Fig 2). Dimensions of profile that is made for analysis is given in following table (Fig.3)

Model 1: Wall injector with cavity with L/D=5

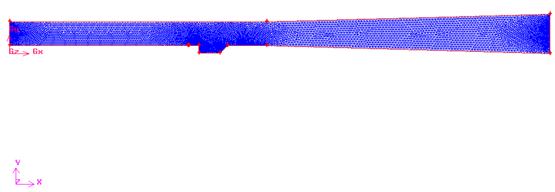

Figure 3. GAMBIT profile of wall injector (with cavity) with L/D ratio=5

TABLE 1 Dimensions of Wall injector (with Cavity) Combustor

Upstream Dia (D_u)	.0096m
Downstream Dia	.0096m
Length of cavity	.048m
L/D ratio	5
Total length of combustor	.667m
Injector Dia.	.001m
Divergence angle	2 degrees
Air inlet dia	.032m
Aft angle	45 degrees
Dist. Of fuel injector from air inlet	.22m

B. Meshing

Meshing of the the model was done with triangular meshes that had been made in GAMBIT .The boundaries were also defined at this stage as air inlet, fuel inlet, wall and outlet. The computational mesh at this level of refinement is said to have reached the limit of grid independence. The resolution of the mesh at all important areas was varied in an attempt to reach grid independent limit mesh.

Different parameters of meshing like number of cells, number of faces, number of nodes, number of partition for the profile is tabulated as follows.

TABLE 2 Parameters of Grid

	Model
Cells	20698
Faces	31407
Nodes	10710
Partition	1
Cell zones	1
Face zones	6

C. Boundary Conditions

During analysis we have taken same Mach no, same pressure and same temperature for both fuel and air for all the four models.

Pressure far field and pressure outlet conditions were taken on the left and right boundaries respectively. Pressure far field condition was taken for fuel injector. The top and bottom boundaries, which signify the sidewalls of the isolator, had symmetry conditions on them. The walls, obstacles and other materials were set to standard wall conditions. The computations were initially carried out with various levels of refinement of mesh. There exists a definite level of refinement beyond which there is no significant quantitative change in the result. The limit of that refinement is called the Grid Independent Limit (GIL).

The input parameters were for the model is shown in tabulated form .

TABLE 3

Input Parameters	AIR	FUEL
Mach No.	2	2
Temperature	1000 K	300 K
Pressure	101325 Pa	501325 Pa
Mass Fraction of O_2	.22	0
Mass fraction of H_2	0	1

IV. Analysis and Formulation

By the above methodology, the static as well as total temperature and static pressure at different point along the length of combustor can be measured. Moreover, plots can also be drawn between various parameters against different positions of the combustor length.

A. Governing Equations

In this study, ideal gas compressible flow was considered. The standard k– ε turbulence model is used.

The k– ε equations are represented as:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \quad [1]$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \quad [2]$$

The values of the five constants of the standard k- ε turbulence model are taken as:

$$C_\mu = 0.09 \quad C_{1\varepsilon} = 1.44 \quad C_{2\varepsilon} = 1.44$$

$$\sigma_k = 1.0 \quad \sigma_\varepsilon = 1.3$$

The equation for conservation of mass, or continuity, can be written in vector form as:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{v}_r) = \vec{S}_m \quad [3]$$

where, value of $\vec{S}_m$ is zero for steady-state flow.

The vectored momentum equation in terms of relative velocity, v_r can be written as

$$\frac{\partial}{\partial t}(\rho\vec{v}_r)+\nabla.(\rho\vec{v}_r\vec{v}_r)+\rho(2\vec{\omega}\times\vec{v}_r+\vec{\omega}\times\vec{\omega}\times\vec{r})=-\nabla p+\rho\vec{g}+\nabla.(\bar{\bar{\tau}}) \quad [4]$$

According to the eddy viscosity concept of the Stokes' hypothesis for Newtonian fluids, the Reynolds stress tensor, $\bar{\bar{\tau}}$ can be expressed as

$$\bar{\bar{\tau}} = \mu[(\nabla\vec{v} + \nabla\vec{v}^T) - \frac{2}{3}\nabla.\vec{v}I] \quad [5]$$

B. Modelling Details

In the CFD model, the standard k-ε turbulent model is selected. This is because of its robustness and its ability to fit the initial iteration, design lectotype and parametric investigation. Further, because of the intense turbulent combustion, the eddy-dissipation reaction model is adopted. The eddy-dissipation is based on the hypothesis of infinitely fast reactions and the reaction rate is controlled by turbulent mixing. Both the Arrhenius rate and the mixing rate are calculated and the smaller of the two rates is used for the turbulent combustion. While no-slip conditions are applied along the wall, but due to the flow being supersonic, at the outflow all the physical variables are extrapolated from the internal cells. Energy equations was considered and the solution was initialized from the air inlet for simplicity. For hydrogen-air mixing, ideal gas mixing law was followed for determination of thermal conductivity and viscosity, while density was assumed to be for ideal gas. Mass diffusivity was assumed to be following kinetic theory. The operating pressure was considered to be zero Pascal.

The Under-Relaxation factors were as follows:

1. Turbulent kinetic energy : 0.8
2. Turbulent dissipation rate : 0.8
3. Turbulent viscosity : 1

V. Results and Discussions

The various plots of properties such as static temperature, static pressure etc along the length of the combustor for the different models are given below. The red colored regions are the regions where the properties attain their maximum values. The blue colored regions indicate the regions where the properties are at their minimum.

The properties that were analyzed for the various models are-

1. Static pressure
2. Static temperature
3. Mass fraction of H_2
4. Mass fraction of O_2
5. Mass fraction of H_2O
6. Density
7. Turbulence kinetic energy
8. Turbulent intensity
9. X velocity

The **static temperature** was taken as an indication of combustion efficiency of the fuel (hydrogen). A higher combustion efficiency means a greater percentage of the injected fuel undergoes combustion resulting in a higher static temperature at the combustor exit. Study of the **mass fraction contours of H_2, O_2 and H_2O** showed evidence of fuel injection, air fuel mixing and combustion respectively. **The presence of H_2O indicated the occurrence of combustion.** Turbulent kinetic energy was an indication of vortex formation in the cavity which enhances air-fuel mixing. The X-velocity was the velocity at which the combustion products exit the combustor. It represented the **thrust available for propulsion of the scramjet.**

The static pressure and density contours and static pressure and density graphs help in visualizing the shock waves produced by the velocity of hydrogen injection. Moreover, interaction of the reflected shock waves with the air-fuel mixing boundary (visible in the density and static pressure contours) further enhanced the mixing and promoted combustion downstream of the cavity.

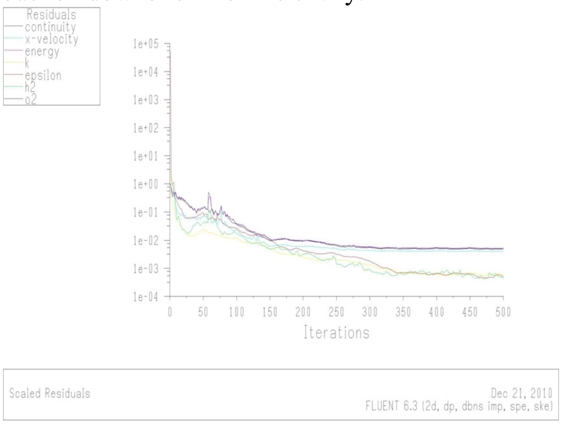

Figure 4 : Iterations curve

In FLUENT software, in order to achieve convergence of the input values, 500 numbers of iterations were performed.

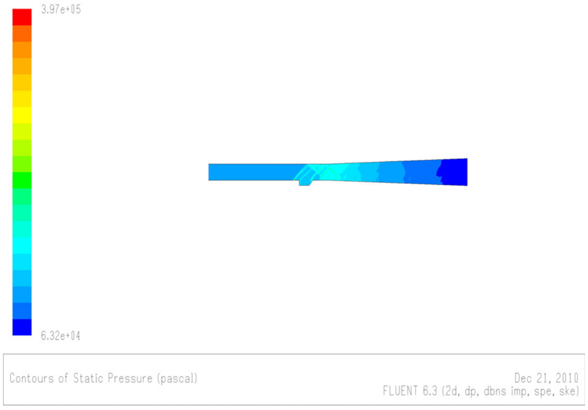

Figure5: Contours of static pressure

Static pressure variation in the combustor was visualized. It remained constant up to the fuel injection.

Pressure rise caused by shock formation is clearly visible There is a pressure rise of ~60kPa across the shock. At the outlet, the pressure decreases to a minimum of $5.56\text{x}10^{04}$ Pascal.

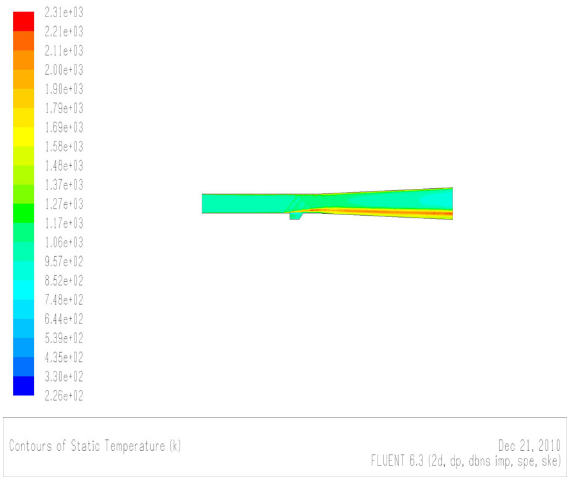

Figure 6: Contours of static temperature

Static temperature increases from 1000K at the inlet to a maximum of 2100K at the outlet. This is due to combustion of the air and injected H_2 fuel. The heat released due to combustion heats up the combustion products (water) and hence, an increase in the static temperature is observed.

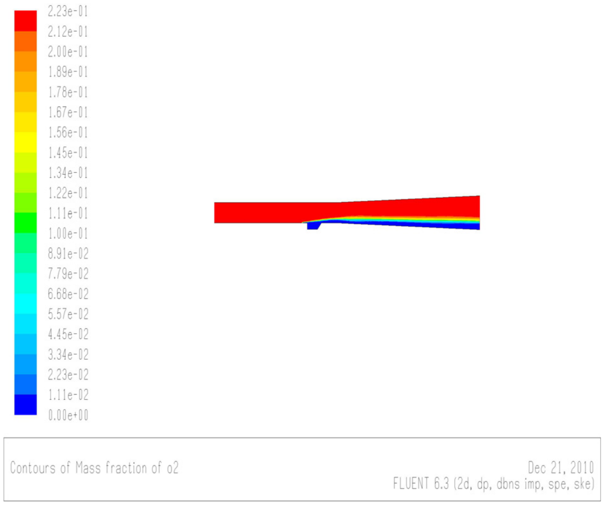

Figure 7: Contour of mass fraction of O_2

Mass fraction of oxygen was maximum at the air inlet of the combustor at .22. After fuel injection, mixing of oxygen and H2 causes a variation in the mass fraction of O_2 at any cross section of the combustor towards the outlet, as can be observed from the contour. The mass fraction varies from .22 at the interior of the combustor to almost 0 at the lower wall.

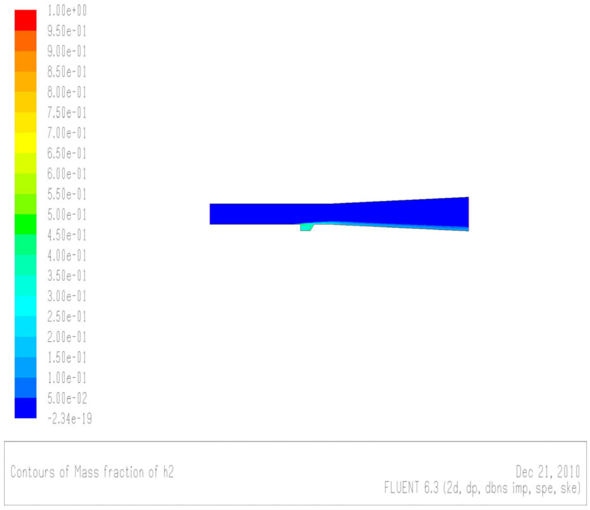

Figure 8: Contour of mass fraction of H_2

Mass fraction of H_2 is zero upto the fuel injection port. An increase in the mass fraction of H_2 beyond the fuel injector indicates the occurrence of fuel injection. The mass fraction continues to decrease downstream of the injector due to occurrence of combustion which consumes the hydrogen.

From fig. 8, it is clear that mass fraction of H_2O attains a maximum value of 0.981 at the outlet of the combustor indicating an intimate mixing of incoming air and injected H_2 and the mixture's subsequent proper combustion.

Density increases to ~0.2 kg/m^3 after fuel injection, but at the outlet, it decreased to a minimum value of 0.0529 kg/m^3 with efficient mixing and combustion. Shock formation can be visualized by the abrupt increase in density just after fuel injection.The turbulent intensity crosses the 32000% mark at the cavity due to intense turbulence caused by vortex formation. This shows efficient air-fuel mixing.

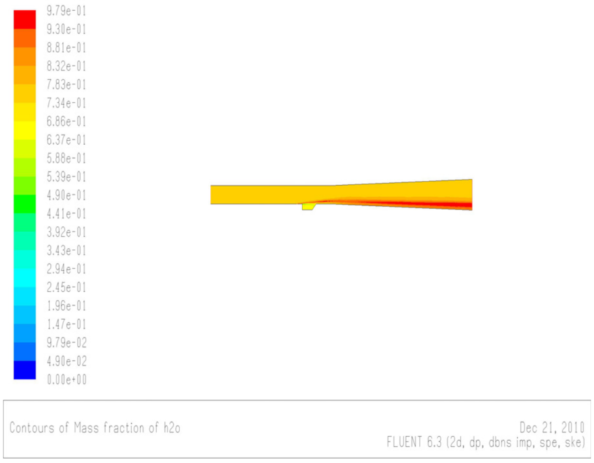

Figure 9: Contour of mass fraction of H_2O

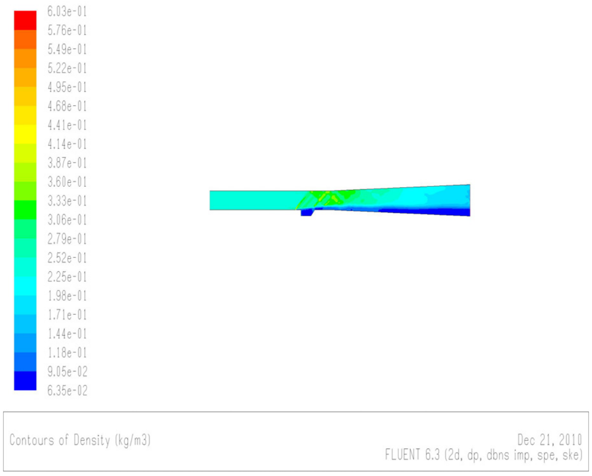

Figure 10: Contour of density

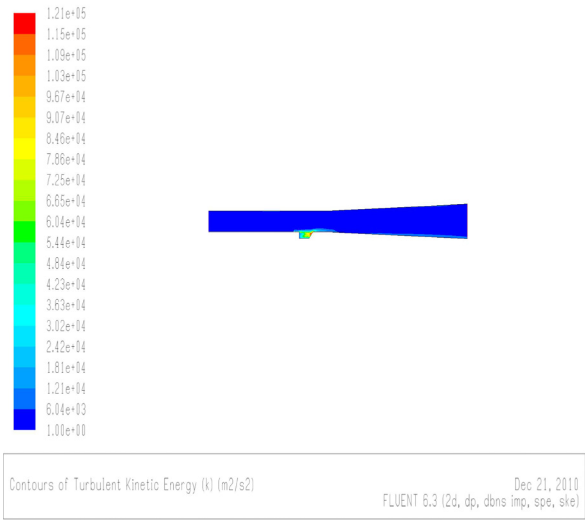

Figure 11: Contour of turbulence kinetic energy

Turbulence kinetic energy was observed to be almost constant throughout the length of the combustor, although there was an increase in the cavity to $1.2x10^5$ m^2/s^2. This indicates vortex formation in the cavity.

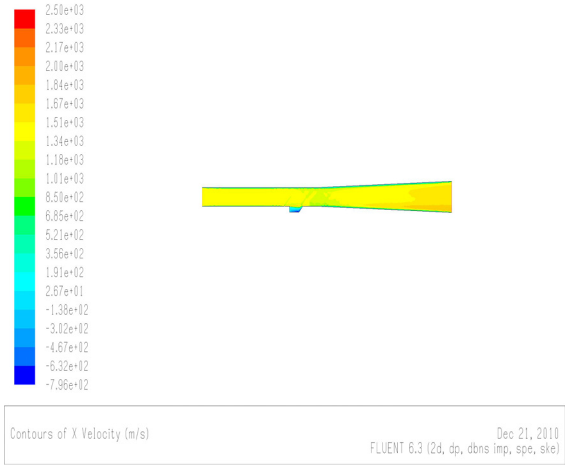

Figure 12: Contour of X velocity

X-velocity increases along the length of the combustor and attains a maximum value of 2000 m/s at the outlet, indicative of high thrust production.

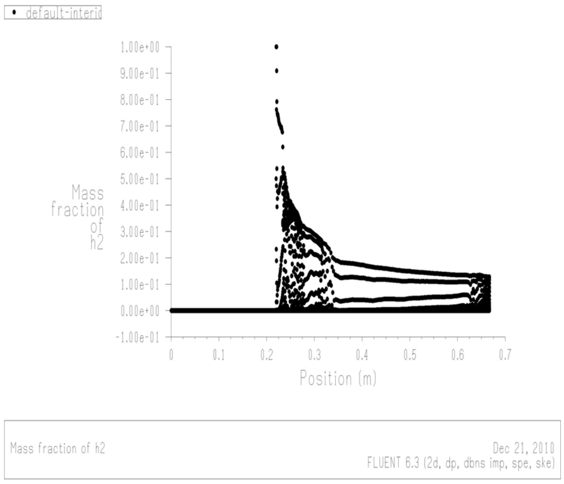

Figure13: Mass fraction of H_2 at interior

The above graph shows the distribution of H_2 in the interior of the combustor. As can be seen, the mass fraction of hydrogen is maximum at the fuel injection port and continues to decrease along the length of the combustor due to combustion. Thus, the graph provides evidence of combustion.

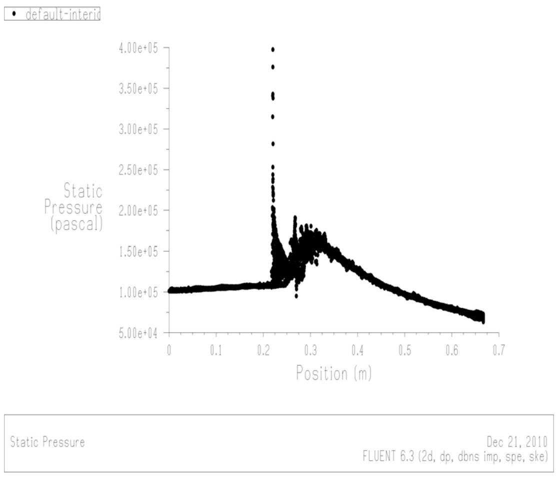

Figure 14: Static pressure distribution at the interior

By plotting the distribution of static pressure at the interior against the length of the combustor, it is observed that static pressure remains same up to the fuel injection. Then, it increases to a maximum of 3.75×10^{05} Pascal due to shock formation. With mixing and combustion of air-hydrogen mixture, it decreases gradually and at the outlet, static pressure is minimum with a magnitude of 5.2×10^{04} Pascal.

Plot of density distribution at interior shows that density increases with H_2 injection and then, it decreases gradually with mixing and combustion of air and hydrogen fuel mixture and the subsequent expansion of the combustion products.

Turbulence kinetic energy attains a maximum value of 1.5×10^5 m^2/s^2 due to injection of H_2 fuel. At the outlet, turbulence KE is 1×10^4 m^2/s^2.

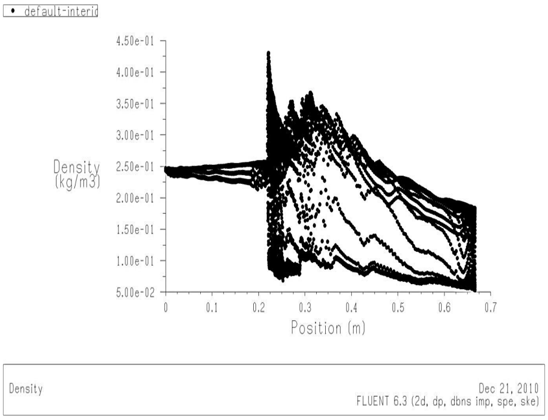

Figure15: Density distribution at the interior

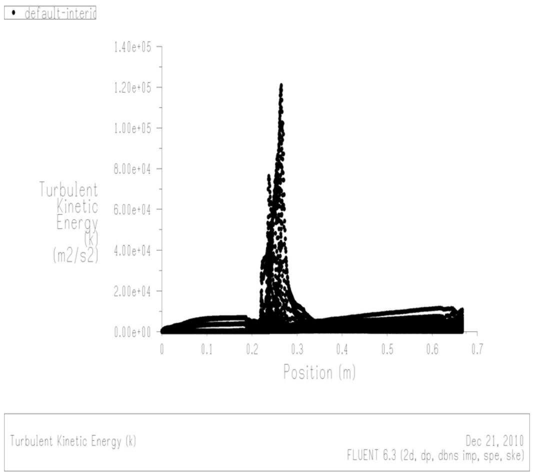

Figure 16: Turbulence KE distribution at the interior

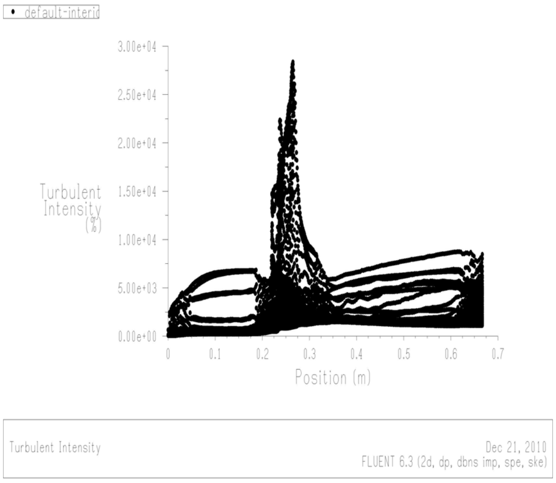

Figure 17: Turbulent intensity at the interior

In the combustor, turbulent intensity increases with the supersonic air flow. After H_2 fuel injection, the flow becomes highly turbulent and turbulent intensity crosses the mark of 32000%.

Analysis

From figures above, it can be drawn that combustion is present in this type of flow through wall injection (with cavity) based combustion. Formation of water as is evident in Fig. 8 is a clear indication of combustion. Different maximum values are reached at different positions which are tabulated below.

TABLE 4 MAXIMUM VALUES OF PARAMETERS

Parameters	Maximum value attained in the combustor
Static temperature	2000-2100 K
Turbulent Kinetic Energy	1.5×10^5 m^2/s^2
X velocity	1800 m/s
Static pressure rise across shock	50 kPa

VI. CONCLUSION

From the above analysis, it is observed that for a scramjet engine having a wall injector with a cavity of L/D=5, if hydrogen is injected at a speed of Mach 2 to an incoming air stream at Mach 2 speed, a rich air-fuel mixture can be achieved and efficient combustion of this mixture gives a maximum temperature of **2100K** at the outlet of the combustor. Moreover, a high axial velocity of **~1800 m/s** is obtained which is indicative of high thrust production. Also, there is a weak shock formation .Hence, better flame holding can be achieved if the wall injector is coupled with a cavity having a L/D ratio of 5. Due to ever increasing human need for greater speed and reduced travel time, hypersonic combustion systems will become more and more important in the future. As the mixing time for fuel in the combustor system is very less (~1ms), newer and better injection systems have to be developed that enhance fuel-air mixing and reduce ignition delay period, thus increasing both combustion efficiency and thrust.

ACKNOWLEDGEMENT

The authors acknowledge the financial help provided by AICTE from the project AICTE: 8023/RID/BOIII/NCP (21) 2007-2008. The Project id at IIT Guwahati is ME/P/USD/4.

REFERENCES

[1] Berman H.A, Anderson Jr. J.D, Drummond J.P., "Supersonic flow over a rearward facing step with transverse non-reacting hydrogen injection", AIAA.J 21(1983) 1707-1713.
[2] Roudakov A.S et al. "Flight testing an axisymmetric scramjet" Russian recent advances in 44th IFA Congress, IFA Paper 93-S.4.485,1993.
[3] Adela Ben-Yakar, Benveniste Natan, Alon Gary- "Investigation of a solid fuel scramjet combustion", J.Propul Power 14 (1).
[4] Adela Ben-Yakar, Hansen R.K- "Cavity flameholders for ignition & flame stabilization in scramjet review and experimental study", AIAA-98-3122,1998.

[5] Zhang X.-"Compressible cavity flow of oscillation due to shear layer instabilities and pressure feedback", AIAA.J.33(8)(1995)1404-1411.
[6] Dimotakis, P. E., Turbulent Free Shear Layer Mixing and Combustion. High Speed Flight Propulsion Systems, Chapter 7, Progress in Astronautics and Aeronautics, Vol. 137, 1991.
[7] Drummond, J. P., Carpenter, M. H., and Riggins, D. W., Mixing and Mixing Enhancement in High Speed Reacting Flows. High Speed Flight Propulsion Systems, Chapter 7, Progress in Astronautics and Aeronautics, Vol. 137, 1991.
[8] Cutler, A. D., and White, J. A., An Experimental and CFD Study of a Supersonic Coaxial Jet. AIAA Paper 2001-0143, January 2001.
[9] Cutler, A., Danehy, P. M., Springer, R. R., and DeLoach, D. P., CARS Thermometry in a Supersonic Combustor for CFD Code Validation. AIAA Paper 2002-0743, January 2002.
[10] Drummond, J. P., Numerical Simulation of a Supersonic Chemically Reacting Mixing Layer. NASA TM 4055, 1988.
[11] K.M.Pandey and Amit Kumar, "Studies on Base Pressure in Suddenly Expanded Circular Ducts: a Fuzzy Logic Approach", IJET 2010 Vol.2 (4): 379-386 ISSN: 1793-8244.
[12] Gardner, A Paull, A & McIntyre, T 2001, 'Upstream porthole injection in a 2-D scramjet', Shock Waves, vol. 11, pp.369-375
[13] Bogdanoff, D.W., "Advanced Injection and Mixing Techniques for Scramjet Combustors", Journal of Propulsion and Power, Vol. 10, No. 2, Mar-Apr 1994, pp. 183-190.
[14] Hsu, K., Carter, C.D., Gruber, M.R., and Barhosrt, T., "Experimental Study of Cavity-Strut Combustion in Supersonic Flow", AIAA Paper 2007-5394, July 2007.
[15] Freeborn, A.B., King, P.I., and Gruber, M.R., "Characterization of Pylon Effects on a Scramjet Cavity Flameholder Flowfield", AIAA Paper 2008-0086, Jan 2008.
[16] K.M.Pandey and T.Sivasakthivel, "Recent Advances in Scramjet FuelInjection - A Review" IJCEA 2010 Vol.1 (4): 294-301 ISSN: 2010-0221

Dr. K. M. Pandey did his PhD in Mechanical Engineering in 1994 from IIT Kanpur. He has published and presented 170 papers in International & National Conferences and Journals. Currently he is working as Professor of the Mechanical .Engineering Department, National Institute of Technology, Silchar,Assam, India..He also served the department in the capacity of head from July 2007 to July 2010. He has also worked as faculty consultant in Colombo Plan Staff College, Manila, Philippines as seconded faculty from Government of India. His research interest areas are the following; Combustion, High Speed Flows, Technical Education, Fuzzy Logic and Neural Nnetworks , Heat Transfer, Internal Combustion Engines, Human Resource Management, Gas Dynamics and Numerical Simulations in CFD area from Commercial Softwares.(Email:kmpandey2001@yahoo.com)

P Kalita and **K Barman**, S.J.Saikia and A. Rajkhowa are final year B. Tech students of Mechanical Engineering Department of National Institute of Technology, Silchar, Assam.