Numerical Simulation of Wall Injection with Cavity in Supersonic Flows of Scramjet Combustion

K.M. Pandey, S.K. Reddy K.K.

***Abstract*—A supersonic combustion ramjet engine (scramjet) 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. This work involves an application of computational fluid dynamics to a problem associated with the flow in the combustor region of a scramjet. A cavity wall injector is an integrated fuel injection approach, and it is a new concept for flame holding and stabilization in supersonic combustors. The presence of a cavity on an aerodynamic surface could have a large impact on the air flow surrounding it, and this makes a large difference to the performance of the engine, namely it may improve the combustion efficiency and increase the drag force. The objective of the work was to design the four wall injector model with cavity using gambit, study the combustion processes of air-fuel (h2) mixture for the wall injector models with inlet air at Mach number 2 and inlet fuel at Mach number 2 and compare the performance of the different wall injector models. There are several key issues that must be considered in the design of an efficient fuel injector. Of particular importance are the total pressure losses created by the injector and the injection processes that must be minimized since the losses reduce the thrust of the engine. In this analysis, the two-dimensional coupled implicit Reynolds averaged Navier-Stokes (RANS) equations, the standard k-ε Turbulence model, sst-kω Turbulence and the eddy-dissipation reaction model have been employed to investigate the flow field in a hydrogen-fuelled scramjet combustor with a cavity design and to analyze the combustion processes. Numerical results are obtained with the fluent solving sst-kω Turbulence model to have the best results of all models. The grid independent test was also carried out. The profiles of static pressure, static temperature, and two components of velocity and mole fraction of hydrogen at various locations of the flow field are presented. Computed values using sst-kω turbulence model are found to have good overall agreement with results obtained from literature reviews and some discrepancies were observed for static pressure and static temperature in the vicinity of the jets due to unsteadiness in the shock system.**

***Index Terms*— Scramjet engine, Mach number 2, RANS Equations, Turbulence model.**

Manuscript Accepted on February 3, 2012.

Dr. K M Pandey, is currently working as Professor, Department of Mechanical Engineering, NIT Silchar, Assam, India. Mobile no. +919435173130, Mail id: kmpandey2001@yahoo.com.

S K Reddy K.K. has done M.Tech in Thermal Engineering from Department of Mechanical Engineering, NIT silchar , Assam , India.

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. 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. A schematic of a typical cavity flame holder can be seen below

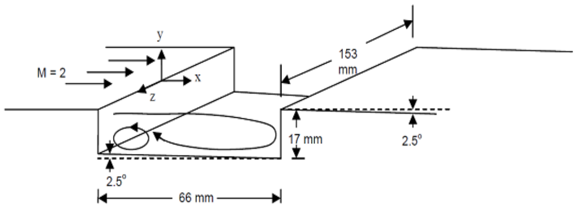

Fig.1: Mach 2 cavity pilot flow used by Lahr et al.

B. *Combustion*

Combustion technology has played an important role in the development of our civilization. Almost 80% of the worldwide energy support is provided by combustion equipment, a fact that is not expected to change in near future .Therefore, deep knowledge, understanding and control of combustion phenomena is great scientific and technological interest. Better design of combustion equipments can contribute both in the energy efficiency and in the reduction of pollution formation. Combustion is a complex phenomenon that involves several disciplines e.g. thermodynamics, heat and mass transfer, fluid dynamics and kinetics chemistry. Much advance has been achieved in the understanding of these disciplines and, furthermore, the couplings of these entire fields in a problem such as combustion have also experienced a remarkable progress. Turbulent combustion requires additional modelization such as statistical techniques to describe the flow and inherent fluctuation involved. Accurate models for molecular transport phenomenon in reactive flow are also deep interest, and much improvement has been acquired. Full chemical models for major fuels have been received great attention from the scientific community, and very completed mechanisms for hydrogen and methane are now available. Nowadays, one of the main limitations to predict and design combustion equipment (boilers, jet engines) or even predicts simple flames such as jet flames or Bunsen flames, in the resolution of the mathematical formulation. Analytical solutions of the governing equation are not feasible for most of the technological problems, and recently numerical techniques have received enormous interest. Given the ever increasing computational capacity, numerical resolution of the formulation has become a powerful tool in the last decades and, therefore, numerical simulation of combustion phenomena is becoming a very useful ingredient in the design of combustion system.

II. LITERATURE REVIEW

Wei Huang, Shi-bin Luo, Mohamed Pourkashanian, Lin Ma, Derek B.Ingham, Jun Liu and Zhen-guo Wang [1] worked on "Numerical Simulations of a Typical Hydrogen Fueled Scramjet Combustor with a Cavity Flameholder" and their findings are- As one of the most promising propulsive systems in the future, the scramjet engine has drawn the attention of many researchers. The two-dimensional coupled implicit NS equations, the standard k-ε turbulence model and the finite-rate/eddy-dissipation reaction model have been applied to numerically simulate the flow field of the hydrogen fueled scramjet combustor with a cavity flameholder under two different working conditions, namely, cold flow and engine ignition. The obtained results show that the numerical method used in this paper is suitable to simulate the flow field of the scramjet combustor. The static pressure distribution along the top and bottom walls for the case under the condition of engine ignition is much higher than that for the case under the condition of cold flow. There are three clear pressure rises on the top and bottom walls of the scramjet combustor. The eddy generated in the cavity acts as a flameholder in the combustor, and it can prolong the residence time of the mixture in the supersonic flow.

In-Seuck Jeung, Jeong-Yeol Choi [2] worked on Numerical Simulation of Supersonic Combustion for Hypersonic Propulsion and their findings are - Recently, renewed interest on the scramjet engine has been demonstrated through the many international activities along the several Asia-Pacific countries. Here, a short review of current activities on supersonic combustion in a scramjet engine will be addressed followed by the discussions on the review of numerical simulation on supersonic combustion phenomena related with scramjet engine combustors and ram accelerator. Emphasis was put on the grid refinement, scheme, unsteadiness and phenomenological differences.

Jeong-Yeol Choi, Fuhua Mab, Vigor Yang [3] worked on Combustion oscillations in a scramjet engine combustor with transverse fuel injection and their findings are - A comprehensive numerical analysis has been carried out for both non-reacting and reacting flows in a scramjet engine combustor with and without a cavity. Transverse injection of hydrogen is considered over abroad range of injection pressure. The corresponding equivalence ratio of the overall fuel/air mixture ranges from 0.167 to 0.50. The work features detailed resolution of the flow and flame dynamics in the combustor, which was not typically available in most of the previous studies. In particular, the oscillatory flow characteristics are captured at a scale sufficient to identify the underlying physical mechanisms. Much of the flow unsteadiness is related not only to the cavity, but also to the intrinsic unsteadiness in the flow-field. The interactions between the unsteady flow and flame evolution may cause a large excursion of flow oscillation. The roles of the cavity, injection pressure, and heat release in determining the flow dynamics are examined systematically.

K.M. Pandey, A.P. Singh [4] worked on Numerical analysis of combustor flow fields in Supersonic flow regime with finite rate Chemistry model and their findings are - In this numerical study, supersonic combustion of hydrogen has been presented. The combustor has a single fuel injection parallel to the main flow from the base. Coupled implicit scheme with finite rate chemistry model and K-e model have been used for modeling of supersonic combustion. The main issue in supersonic combustion is proper mixing within short burst of time. Attention is paid to the local intensity of heat release, which determines, together with the duct geometry, techniques for flame initiation and stabilization, injection techniques and quality of mixing the fuel with oxidizer, the gas-dynamic flow regime. The five main parameters were considered like Mach number stagnation temperature, mass fraction, stagnation pressure and velocity. The result shows the better mixing of fuel and the flame speed increases almost linearly. The stagnation temperature in the combustion reaches up to 2790 k. fluctuation in pressure and Mach number was due to shock train.

K.M.Pandey, T.Sivasakthivel [5] worked on Recent Advances in Scramjet Fuel Injection - A Review and their findings are - Fuel injection techniques into scramjet engines are a field that is still developing today. The fuel that is used by scramjets is usually either a liquid or a gas. The fuel and air need to be mixed to approximately stoichiometric proportions for efficient combustion to take place. The main problem of scramjet fuel injection is that the airflow is quite fast, meaning that there is minimal time for the fuel to mix with the air and ignite to produce thrust (essentially milliseconds).Hydrogen is the main fuel used for combustion. Hydrocarbons present more of a challenge compared to hydrogen due to the longer ignition delay and the requirement for more advanced mixing techniques. Enhancing the mixing, and thus reducing the combustor length, is an important aspect in designing scramjet engines. There are number of techniques used today for fuel injection into scramjet engines.

Weipeng Li, Taku Nonomura, Akira Oyama and Kozo Fujii [6] worked on LES Study of Feedback-loop Mechanism of Supersonic Open Cavity Flows and their findings are - Supersonic flow over a three-dimensional rectangular cavity with length-to-depth ratio of 2 is numerically studied by implicit large-eddy simulation to clarify the feedback-loop mechanism. A feedback-loop cycle is described and visualized with phase-averaged analysis of simulation results. Causality between the feedback acoustic wave and leading-edge shedding vortex is clearly demonstrated. Mach wave reflection at trailing edge is turned out to be the generation mechanism of feedback acoustic wave. It is convinced by investigating time-series instantaneous flow fields and auto-correlation coefficients of three simulation cases with different convective Mach number. Components of compression waves in supersonic cavity flows are summarized and their features are discussed. Proper orthogonal Decomposition (POD) in frequency domain is firstly employed to analyze wave propagations inside cavity. Results statistically show the propagation traces of notable compression waves inside cavity which are affected by high speed recirculation flows.

Y. Moriyoshi, K. Suga, M. Kubota [7] worked on Modeling of Cavitation Phenomenon inside a Nozzle under High Fuel Pressure Condition and their findings are - Direct fuel injection system is getting popular in internal combustion engines due to the superior performance in fuel economy and power. To optimize the fuel-air mixture distribution inside the cylinder, the geometry of nozzle and the mixture formation process must be well designed. To attain this, numerical simulation will be a good tool, but the prediction ability is not enough for the practical design. In this study, a two-phase flow of fuel and gas including cavitation inside an ax-symmetrical nozzle was evaluated to improve the prediction ability. Two kinds of cavitaion model (quasi-steady dynamic bubble model and discrete bubble tracking model) are proposed and implemented into a commercial code FLUENT 6.4. As a result, discrete bubble tracking model could obtain converged solutions even for a high differential pressure conditions between the nozzle entrance and the exit up to 6 MPa and predict the cavitaion phenomenon qualitatively.

Michael K. Smart [8] worked on Scramjet Inlets and his findings are - The supersonic combustion ramjet, or scramjet, is the engine cycle most suitable for sustained hypersonic flight in the atmosphere. This article describes some challenges in the design of the inlet or intake of these hypersonic air-breathing engines. Scramjet inlets are a critical component and their design has important effects on the overall performance of the engine. The role of the inlet is first described, followed by a description of inlet types and some past examples. Recommendations on the level of compression needed in scramjets are then made, followed by a design example of a three-dimensional scramjet inlet for use in an access-to-space system that must operate between Mach 6 and 12.

Md. Mahbubul Alam, Shigeru Matsuo, Toshiaki Setoguchi [9] worked on Passive Suppression of Cavity-Induced Pressure Oscillation in An Axisymmetric Supersonic Flow and their findings are - This Cavity-induced pressure oscillations have been investigated numerically for a two-dimensional supersonic flow at Mach number 1.83 at the entrance of a straight channel connected to an axi-symmetric nozzle. The control is achieved using a passive suppression method that includes a cavity partially covered by a solid surface positioned at wall of the straight channel for supersonic flow. The results showed a good reduction of amplitudes of oscillations at the region of main flow and also at the inside position of the cavity in case of flow with control.

B.V.N. Charyulu1, R. Manoj, B. Rajinikant, D.K. Tripathi, A. Rolex, Vikrant Satya, V. Ramanujachari, S. Panneerselvam [10] worked on Experimental investigations of ramp-cavity based Supersonic combustor and their findings are - Ramp-Cavity based combustor is one of the candidates of the Scramjet combustor being developed in DRDL for the Hypersonic Technology Demonstrator Vehicle project Ground testing of the ramp-cavity Scramjet combustor in connect pipe mode was carried out. Self-ignition of the kerosene fuel and sustained combustion are achieved for the complete duration (typically 5 seconds) of fuel injection. Wall pressures and temperatures near the wall boundary are measured during the static tests. The heat release during the combustion of the fuel gives rise to increase in pressure covering a large volume of combustor. The effect of fuel injection through cavities is not appreciable and hence does not affect the combustor performance. The post test inspection indicates that all the zirconia coated ramps are found to be intact without any structural damage. The thrust predicted from the experiments is found to be adequate to demonstrate positive acceleration of the Scramjet integrated Hypersonic Technology Demonstrator Vehicle.

Sean M. Torrez, James F. Driscoll, Matthias Ihme, Matthew L. Fotia [11] worked on Reduced-Order Modeling of Turbulent Reacting Flows with Application to Ramjets and Scramjets and their findings are - A new engine model has been developed for applications requiring run times shorter than a few seconds, such as design optimization or control evaluation. A reduced-order model for mixing and combustion has been developed that is based on non dimensional scaling of turbulent jets in cross flow and tabulated presumed probability distribution function flame let chemistry. The three-dimensional information from these models is then integrated across cross-sectional planes so that a one-dimensional profile of the reaction rate of each species can be established. Finally, the one-dimensional conservation equations are integrated along the downstream axial direction and the longitudinal evolution of the flow can be computed. The reduced-order model accurately simulates real-gas effects such as dissociation, recombination, and finite rate chemistry for geometries for which the main flow is nearly one- dimensional. Thus, this approach may be applied to any flow path in which this is the case; ramjets, scramjets, and rockets are good candidates. Comparisons to computational fluid dynamics solutions and experimental data were conducted to determine the validity of this approach.

Kathleen Tran [12] worked on One Dimensional Analysis Program for Scramjet and Ramjet Flow paths and his findings are - One-Dimensional modeling of dual mode scramjet and ramjet flow paths is a useful tool for scramjet conceptual design and wind tunnel testing. In this thesis, modeling tools that enable detailed analysis of the flow physics within the combustor are developed as part of a new one-dimensional MATLAB-based model named VTMODEL. VTMODEL divides a ramjet or scramjet flow path into four major components: inlet, isolator, combustor, and nozzle. The inlet module provides two options for supersonic inlet one-dimensional calculations; a correlation from MIL Spec 5007D, and a kinetic energy efficiency correlation. The kinetic energy efficiency correlation also enables the user to account for inlet heat transfer using a total temperature term in the equation for pressure recovery. The isolator model also provides two options for calculating the pressure rise and the isolator shock train. The first model is a combined Fanno flow and oblique shock system. The second model is a rectangular shock train correlation. The combustor module has two options for the user in regards to combustion calculations. The first option is an equilibrium calculation with a “growing combustion sphere” combustion efficiency model, which can be used with any fuel. The second option is a non-equilibrium reduced-order hydrogen calculation which involves a mixing correlation based on Mach number and distance from the fuel injectors. This model is only usable for analysis of combustion with hydrogen fuel. Using the combustion reaction models, the combustor flow model calculates changes in Mach number and flow properties due to the combustion process and area change, using an influence coefficient method. This method also can take into account heat transfer, change in specific heat ratio, change in enthalpy, and other thermodynamic properties. The thesis provides a description of the flow models that were assembled to create VTMODEL. In calculated examples, flow predictions from VTMODEL were compared with experimental data obtained in the University of Virginia supersonic combustion wind tunnel, and with reported results from the scramjet models SSCREAM and RJPA. Results compared well with the experiment and models, and showed the capabilities provided by VTMODEL.

III. GOVERNING EQUATIONS FOR RANS TURBULENCE

The advantage of employing the complete Navier-Stokes equations extends not only to the investigations that can be carried out on a wide range of flight conditions and geometries, but also in the process of the location of shock wave, as well as the physical characteristics of the shock layer, can be precisely determined. The three-dimensional forms of the Navier-Stokes equations below. Note that the two-dimensional forms are just simplification of the governing equations in the three dimensions by the omission of the component variables in one of the co-ordinate directions. Neglecting the presence of body forces and volumetric heating, the three-dimensional Navier-Stokes equations are derived as

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

X-Momentum Equation:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho uu)}{\partial x} + \frac{\partial(\rho vu)}{\partial y} + \frac{\partial(\rho wu)}{\partial z} = \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}$$

Y-Momentum Equation:

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho vv)}{\partial y} + \frac{\partial(\rho wv)}{\partial z} = \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z}$$

Z-Momentum Equation:

$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho ww)}{\partial z} = \frac{\partial\tau_{xz}}{\partial x}+\frac{\partial\tau_{yz}}{\partial y}+\frac{\partial\sigma_{zz}}{\partial z}$$

Momentum Equation:

$$\frac{\partial(\rho E)}{\partial t}+\frac{\partial(\rho uE)}{\partial x}+\frac{\partial(\rho vE)}{\partial y}+\frac{\partial(\rho wE)}{\partial z} = \frac{\partial(u\sigma_{xx}+v\tau_{xy}+w\tau_{xz})}{\partial x} + \frac{\partial(u\tau_{yx}+v\sigma_{xy}+w\tau_{yz})}{\partial y} + \frac{\partial(u\tau_{zx}+v\tau_{zy}+w\sigma_{zz})}{\partial z} + \frac{\partial\left(k\frac{\partial T}{\partial x}\right)}{\partial x} + \frac{\partial\left(k\frac{\partial T}{\partial y}\right)}{\partial y} + \frac{\partial\left(k\frac{\partial T}{\partial z}\right)}{\partial z}$$

IV. METHODOLOGY AND CFD CODE SET UP

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. Dimensions of profile that is made for analysis is given in following table below:

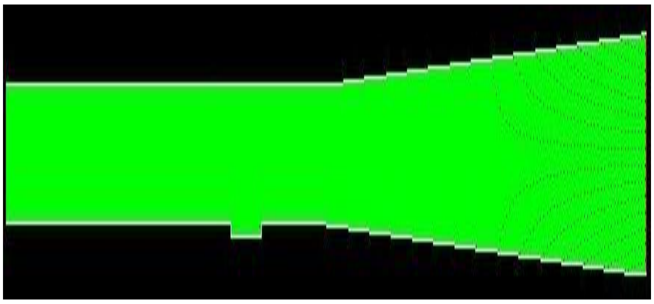

Fig 2: GAMBIT profile of wall injector (with cavity)

Table I: Dimensions Of Wall Injector (With Cavity) Combustor

Upstream diameter	0.0096m
Downstream diameter	0.0096m
Length of cavity	0.048m
L/d ratio	5
Total length of combustor	0.667m
Injector diameter	0.001m
Divergence angle	2 Degrees
Air inlet diameter	0.032m
Distance of fuel injector from air inlet	0.22m

B. Meshing:

Meshing of 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 II: Parameters of Grid

Cells	20698
Faces	31407
Nodes	10710

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

Partition	1
Cell Zones	1
Free 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 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 III: Input Parameters

Input Parameters	Air	Fuel
Mach Number	2	2
Temperature	1000K	300K
Pressure	101325 Pa	501325 Pa
Mass fraction of O2	0.22	0
Mass Fraction of H2	0	1

D. Modeling Details:

In the CFD model, the k-ω turbulent model is selected. This is because of its robustness and its ability to fit the initial iteration. 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 were 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 DISCUSSSIONS

The various plots of properties such as static temperature, static pressures 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 are-

1. Static pressure
2. Static temperature
3. Density
4. Turbulence kinetic energy
5. Turbulent intensity
6. X velocity

The static temperature was taken as an indication of combustion efficiency of the fuel (hydrogen). 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 H2, O2 and H2O showed evidence of fuel injection, air fuel mixing and combustion respectively. The presence of H2O 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.

A. Static Pressure:

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

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 across the shock. At the outlet, the pressure decreases which can be seen from the contour below.

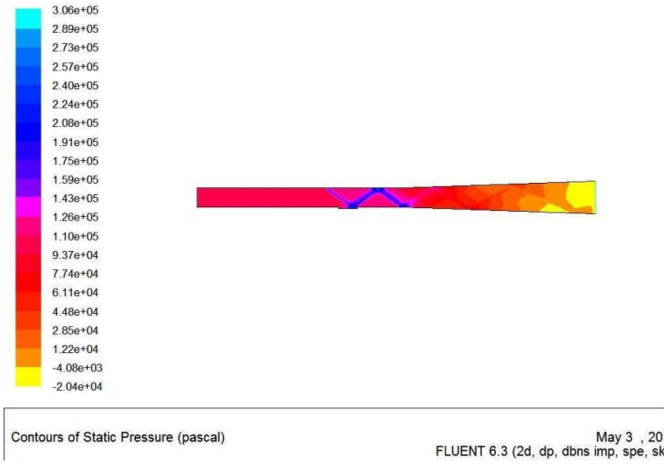

Fig. 3: Contours of static pressure

B. Static temperature:

Static temperature increases from inlet to the outlet. This is due to combustion of the air and injected H2 fuel. The heat released due to combustion heats up the combustion products (water) and hence, an increase in the static temperature is observed.

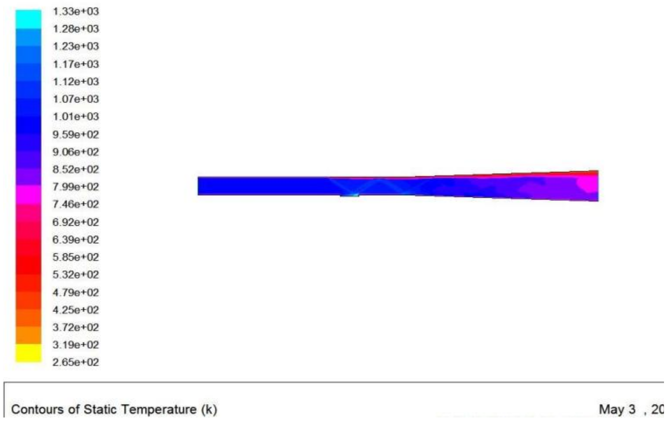

Fig. 4: Contours of static temperature

C. 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 2.3x10^4 m2/s2. This indicates vortex formation in the cavity.

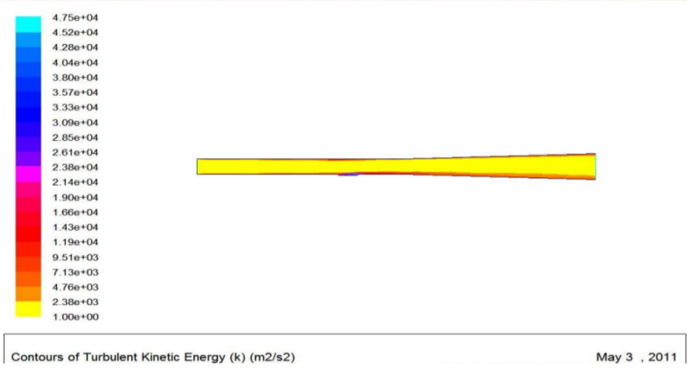

Fig. 5: Contour of Turbulence Kinetic Energy

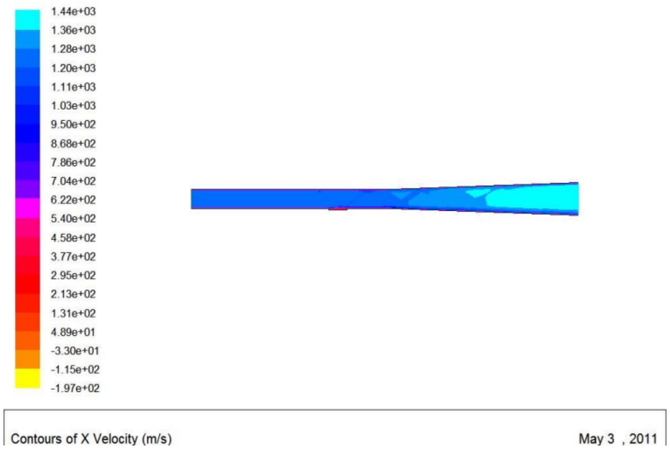

Fig. 6: Contour Of X Velocity

The below 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.

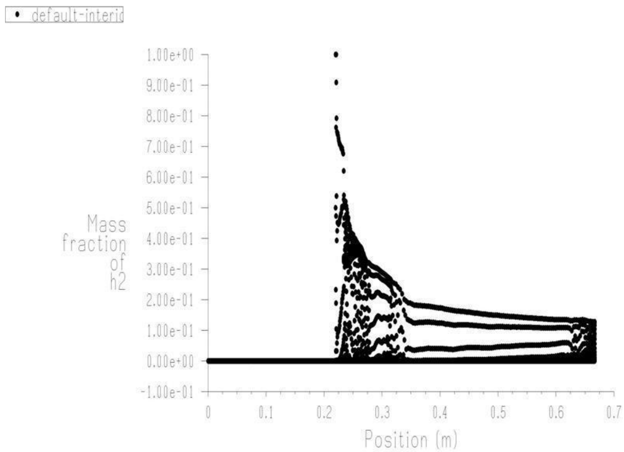

Fig. 7: Mass fraction of H_2 at interior

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 $3.75x10^{05}$ 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 $5.2x10^4$ 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 $1.5x10^5$ m^2/s^2 due to injection of H_2fuel. At the outlet, turbulence KE is $1x10^4$ m^2/s^2.

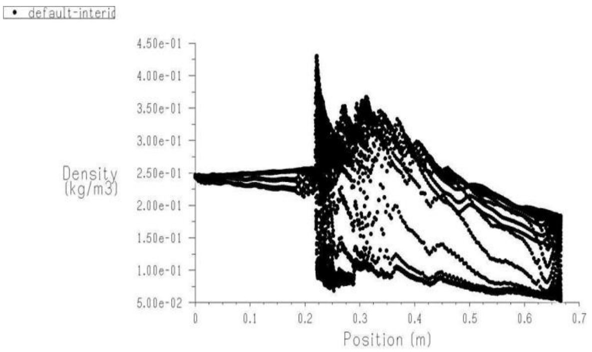

Fig. 8: Density distribution at the interior

From figures above, it can be drawn that combustion is present in this type of flow through wall injection (with cavity) based combustion.

Various analyses are performed for optimum length of the combustion chamber and the variation of the velocity magnitude with reference to the position can be seen clearly in the graph below. At the length of the chamber along with the nozzle is 65 mm the value of the velocity variation is approximately linear. If the length is increased to 120 mm the variation can be seen and the profile varies as shown in the graph below.

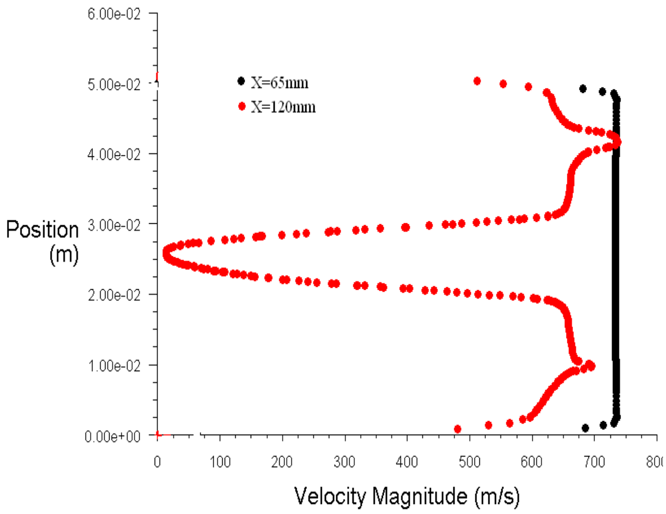

Fig. 9: Variation of velocity magnitude with length

A similar graph is provided below which also shows the variation of the velocity magnitude with respect to the length of the chamber and nozzle.

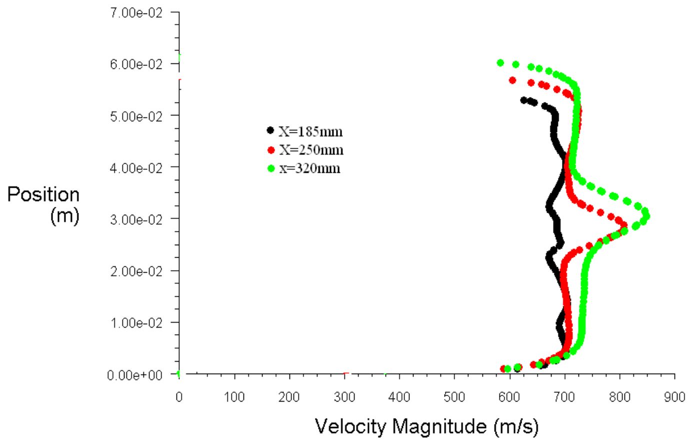

Fig. 10: Variation of velocity magnitude with varying length

From the above graphs it can be seen that the velocity magnitude increases with increase in the length of the combustion chamber.

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 1400K 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.

ACKNOWLEDGMENT

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] 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, London, UK, July 2010.

[2] In-Seuck Jeung, Jeong-Yeol Choi; "Numerical Simulation of Supersonic Combustion for Hypersonic Propulsion"; 5th Asia-Pacific Conference on Combustion, 18-20 July 2005.

[3] Jeong-Yeol Choi, Fuhua Mab, Vigor Yang; "Combustion oscillations in a scramjet engine combustor with transverse fuel injection"; Proceedings of the Combustion Institute 30, 2005, pp:2851–2858.

[4] K.M. Pandey, A.P. Singh; "Numerical analysis of combustor flow fields in Supersonic flow regime with finite rate Chemistry model"; ISST Journal of Mechanical Engineering, Vol. 1 No.2, (July - December 2010), p.p. 81-90.
[5] K.M.Pandey, T.Sivasakthivel; "Recent Advances in Scramjet Fuel Injection - A Review"; International Journal of Chemical Engineering and Applications, ISSN: 2010-0221, Vol. 1, No. 4, December 2010.
[6] Weipeng Li, Taku Nonomura, Akira Oyama and Kozo Fujii; "LES Study of Feedback-loop Mechanism of Supersonic Open Cavity Flows"; 40th Fluid Dynamics Conference and Exhibit, AIAA 2010-5112, 28 June - 1 July 2010.
[7] Y. Moriyoshi, K. Suga, M. Kubota; "Modeling of Cavitation Phenomenon inside a Nozzle under High Fuel Pressure Condition"; 11^{th} ICLASS July 2009.
[8] Michael K. Smart; "Scramjet Inlets"; Brisbane 4072 AUSTRALIA
[9] Md. Mahbubul Alam, Shigeru Matsuo, Toshiaki Setoguchi; "Passive Suppression of Cavity-Induced Pressure Oscillation in An Axisymmetric Supersonic Flow"; 29- 31 December 2007, Dhaka, Bangladesh, ICME 2007
[10] B.V.N. Charyulu1, R. Manoj, B. Rajinikant, D.K. Tripathi, A. Rolex, Vikrant Satya, V. Ramanujachari, S. Panneerselvam; "Experimental investigations of ramp-cavity based Supersonic combustor"; International Conference on Aerospace Science and Technology, Bangalore, India, 26-28 June, 2008.
[11] Sean M. Torrez, James F. Driscoll, Matthias Ihme, Matthew L. Fotia; "Reduced-Order Modeling of Turbulent Reacting Flows with Application to Ramjets and Scramjets"; Journal of propulsion and power; vol. 27, No. 2, March–April 2011.
[12] Kathleen Tran; "One Dimensional Analysis Program for Scramjet and Ramjet Flow paths"; Blacksburg, VA, December 8, 2010.

AUTHOR'S PROFILE

Dr. K.M.Pandey did his PhD in Mechanical Engineering in 1994 from IIT Kanpur. He has published and presented more than 200 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. Department, National Institute of Technology, Silchar, Assam, India. He has also served 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 Combustion, High Speed Flows, Technical Education, Fuzzy Logic and Neural networks, Heat Transfer, Internal Combustion Engines, Human Resource Management, Gas Dynamics and Numerical Simulations in CFD area from Commercial Softwares. At present, he is serving the institute in the capacity of Dean, Faculty Welfare since November 2011.

S K Reddy K.K. has done M.Tech in Thermal Engineering from Department of Mechanical Engineering, NIT silchar , Assam , India.