ISSN 2278-3091

International Journal of Advanced Trends in Computer Science and Engineering, Vol.2 , No.1, Pages : 205 – 214 (2013)
Special Issue of ICACSE 2013 - Held on 7-8 January, 2013 in Lords Institute of Engineering and Technology, Hyderabad

CFD Analysis of Supersonic Combustor using wedge shaped strut having circular inlet with standard k-є Non premixed turbulence model

Dr K.M.Pandey[1], Binita Nath[2],
[1] Professor, Department of Mechanical Engineering,N.I.T Silchar, Assam, India,
Email: kmpandey2001@yahoo.com
[2] M.Tech Student , Department ofMechanical Engineering,N.I.T Silchar, Assam, India
Email: nath.bini@gmail.com

Abstract: In this paper the CFD analysis of supersonic combustion of hydrogen using wedge shaped strut having circular inlet with standard k-є non premixed turbulence model is discussed. In doing this a PDF(Probability Density Function) approach is created and this method needs solution to a high definition The present work is based on designing the model using ANSYS 14 software and then the FLUENT analysis is also done for analysis of combustion process with air inlet at Mach number 3 and hydrogen inlet at Mach number 1.5. The obtained results show that the numerical method used in this paper is suitable to simulate the flow field of the scramjet combustor. The eddy generated in the strut acts as a flame holder in the combustor, and it can prolong the residence time of the mixture in the supersonic flow.

Key words : Flame holder, k-ε model, Scramjet and Supersonic combustion.

1. INTRODUCTION

1.1Scramjet Engines

The name scramjet is an acronym for *Supersonic Combustion Ramjet* which is a type of jet engine intended to operate in the high velocity regime usually associated with rockets. The Scramjet engine design is an extension of the Ramjet. The difference between the two lies in flow state inside the engine. Both of them are designed to be used for supersonic flight; however a Scramjet allows the flow through the engine to remain supersonic, whereas in a Ramjet the flow is slowed to subsonic levels before it enters the combustor. Fig.1.1 shows a basic generic Scramjet design. It works by injecting fuel (typically hydrogen) into a flow of supersonic air. The air is at sufficiently high temperature and pressure for the fuel to combust, and the resulting mixture is expelled from the engine at a higher pressure.

The Scramjet is composed of four main sections: the inlet, isolator, combustor and exhaust nozzle. These sections can be seen in Fig.1.1.

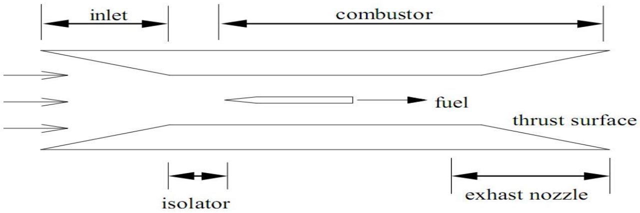

Fig.1.1: Generic Scramjet engine

The inlet heats and slows the flow through a series of oblique shockwaves. This “ram” portion of the cycle means the engine cannot be operated statically. The isolator serves to separate the combustor from the inlet of the engine, allowing further slowing of the flow. Combustion is achieved through the continuous injection of fuel (usually hydrogen) into the supersonic flow. The fuel mixes and combusts, increasing the pressure and temperature of the flow. Finally the flow is expanded via the nozzle. This serves two purposes: to allow the flow to accelerate to the external speed, and to provide a mechanism by which the increase in pressure can be converted into forward thrust.

1.2 Fuel Injectors for Scramjet Engines

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.

At moderate flight Mach numbers, up to Mach 10, fuel injection may have a normal component into the flow from the inlet, but at higher Mach numbers, the injection must be nearly axial since the fuel momentum provides a significant portion of the engine thrust. The injector design and the flow disturbances produced by injection also should provide a region for flame holding, resulting in a stable piloting source for downstream ignition of the fuel. The injector cannot result in too several local flow disturbance, that could result in locally high wall static pressures and temperatures, leading to increased frictional losses an severe wall cooling requirements. A number of options are available for injecting fuel and enhancing the mixing of the fuel and air in high speed flows typical of those found in a scramjet combustor. The Supersonic Combustion Ramjet (SCRAMJET) engine has been recognized as the most promising and sustaining air breathing propulsion system for the hypersonic flight (Mach number above 5).

1.2.1 Parallel, Normal and Transverse Injection

Parallel fuel injection is shown in Fig1.2, it consists of fuel flowing parallel to the air in the engine but separated by a splitter plate. When the splitter plate ends, a shear layer is created due to the different velocities of the fuel and air. The shear Layer is the primary source of mixing the fuel with the air so that proper combustion can be achieved.

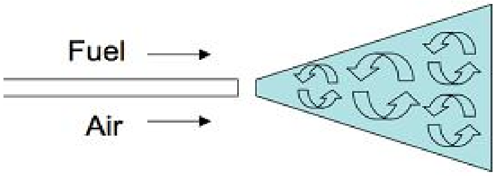

Fig.1.2: Parallel fuel injection

Normal fuel injection consists of an injection port on the wall of a scramjet. The port injects the fuel normal to the flow of air in the scramjet. Normal fuel injection creates a detached normal shock upstream of the injector which causes separation zones upstream and downstream of the injector as shown in Fig1.3. The separation zones cause increased total pressure losses which affect the efficiency of the engine. However, the downstream separation regions can be used as a flame holder.

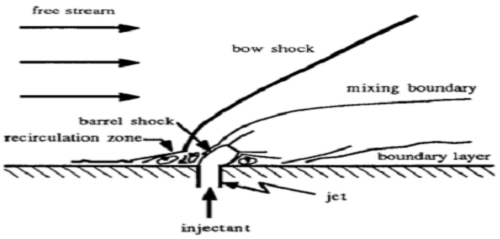

Fig.1.3: Normal fuel injection

Transverse fuel injection is a combination of parallel and normal fuel injection. In a transverse injector, the fuel is injected at an angle between normal and parallel to the flow. Transverse injection reduces some of the negatives to normal injection, but requires a larger injection pressure to achieve the same penetration height into the air flow. The increase in the injection pressure increases the total pressure loss of the scramjet which decreases the efficiency of the engine.

1.2.2 Ramp Injectors

To add axial velocity to the flow near fuel injection, ramps were added with fuel injectors on the trailing edge of the ramp injecting fuel parallel to the flow. The flow over the ramps created counter-rotating vortices that increased the mixing. Due to the supersonic flow in the scramjet, the ramps also create shocks and expansion fans which cause pressure gradients that also increase mixing. Two types of ramps were used; compression ramps are elevated above the floor while expansion ramps create troughs in the floor (Fig.1.4).

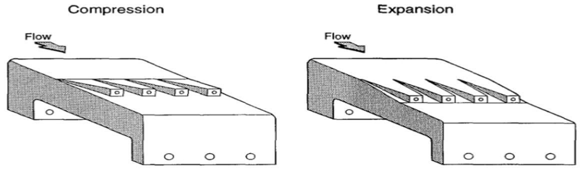

Fig.1.4: Ramp Injectors

1.2.3 Strut Injectors

Strut mixing devices covers a wide range of designs and includes both normal and parallel injection methodologies. Most struts consist of a vertical strut with a wedge leading edge. The strut is connected to both the bottom and top of the combustion section. Since it is across the whole combustion section, fuel injection occurs at several locations and allows the fuel to be added throughout the flow field. The advantages of strut injectors are the absence of strong shock waves due to a blockage caused by the fuel jet.

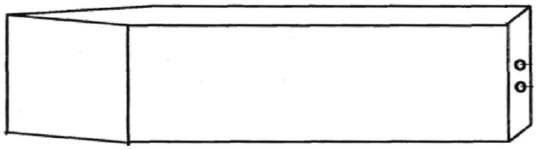

Fig.1.5: Strut with circular injector

Researchers are looked at modifying the trailing edge of the vertical strut to increase mixing. The basic strut design was similar in that the strut was connected to the top and bottom of the test section and the leading edge was a wedge. The difference came from the trailing edge designs as seen in Fig.1.6. The different trailing edges, called alternating wedge designs, create either co-rotating or counter-rotating vortices that are used to enhance the mixing. All of these designs use parallel fuel injection at the trailing edge of the strut so that the fuel is entrained into the vortices which cause the increased mixing in the combustion section.

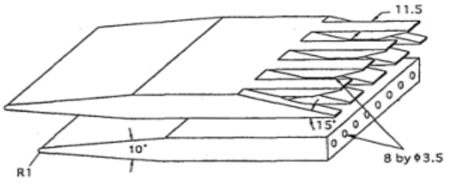

Fig.1.6: Alternating Wedge strut

1.2.4 Pylon Injection

Pylon injection is essentially injection behind a tall, narrow in-stream body, such as shown in Fig.1.7. Injection may be axial, normal, or at some other angle relative to the free stream. Many shapes and angles of injection have been investigated. The results showed much improved mixing and penetration, improved flame holding, and a lack of pressure losses and pronounced edge shocks.

ISSN 2278-3091
International Journal of Advanced Trends in Computer Science and Engineering, Vol.2 , No.1, Pages : 205 – 214 (2013)
Special Issue of ICACSE 2013 - Held on 7-8 January, 2013 in Lords Institute of Engineering and Technology, Hyderabad

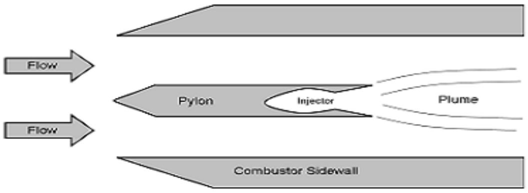

Fig.1.7: Central Pylon Fuel Injection

1.2.5 Cavity Flame holder

In this 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 prospective applications sustained combustion. An injection with a cavity set up is shown in Fig.1.8.

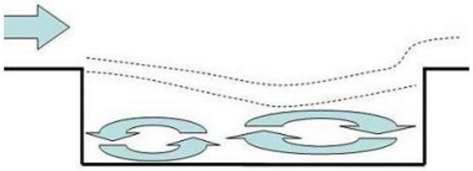

Fig.1.8: Rectangular cavity flame holder

1.2.6 Cavity-Pylon Flame holder

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 and improving mixing due to pylon vortex/shock interactions. 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.

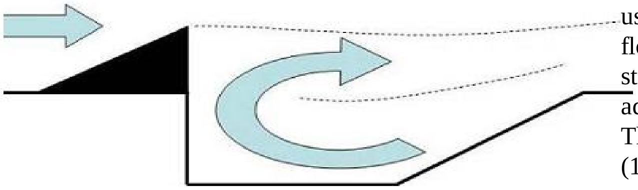

Fig.1.9: Cavity-Pylon Flame holder

1.3 Mixing, Ignition and flame holding in a scramjet combustor

Among the three critical components of the scramjet engine, the combustor presents the most formidable and challenging problems. The complex phenomenon of supersonic combustion involves turbulent mixing, shock interaction and heat release in supersonic flow. The flow field within the combustor of scramjet engine is very complex and that poses a considerable challenge in designing and development of a supersonic combustor with an optimized geometry. Such combustor shall promote sufficient mixing of the fuel and air so that the desired chemical reaction and thus heat release can occur within the residence time of the fuel-air mixture. In order to accomplish this task, it is extremely essential to have a clear understanding of fuel injection processes and have thorough knowledge of the processes governing supersonic mixing and combustion as well as the factors, which affects the losses within the combustor. The designer shall keep in mind the following goals namely,

i) Good and rapid fuel air mixing
ii) Minimization of total pressure loss
iii) High combustion efficiency.

Scramjet Fuel Injection system

Due to the extremely short residence time of the air in supersonic combustors, an efficient (rapid and with small losses in total pressure) fuel/air mixing is hard to achieve. This makes it an important issue to keep the combustor length short and to reduce the skin friction drag. In supersonic flows a rapid fuel/air mixing additionally suffers from inherently low mixing rates due to compressibility effects at high convective Mach numbers [1,2]. There are mainly two concepts for fuel injection in supersonic combustors:

Wall injectors, where hydrogen is injected through the wall [3-5] (normal or oblique to the main flow) or by ramps [6-8] mounted to the wall, **Strut injectors** [1, 9-11], which are located at the channel axis and directly inject the fuel into the core of the air stream. In some cases both types of injectors approach each other ,e.g. if a ramp injector extends over most of the channel height [8]. A good near field mixing can be achieved by wall injection, on the other hand transverse injection systems cause a significant blockage of the flow resulting in irreversibility's due to shock waves and thrust losses [12].The injected hydrogen usually acts as a coolant for the strut. Alternative to physical ramp injectors are aero ramps [13] which have a similar physical behavior but lower pressure losses [14].Aero ramps are multi-hole transverse injectors which induce pairs of counter-rotating vortices to improve mixing and fuel penetration. If strut injectors are used then usually all or most of the fuel is injected in main flow direction. This is possible without the induction of strong shock waves. Moreover, additional momentum is added by parallel fuel injection increasing the engine thrust. This may become important at high flight Mach numbers (10–15) [15]. Due to the limited mixing capabilities of parallel high speed streams, techniques for mixing enhancement are required. This can be achieved either by the use of shock waves [16,17] or by creation of stream wise vorticity. Stream wise vortices may be induced by favorable chosen strut geometry. Hydrogen should be injected in such a way that a good mixing is achieved over a short length resulting in a homogeneous temperature distribution. Local temperature peaks have to be avoided as to keep dissociation losses and nitrogen oxides low. A vital issue at low flight Mach numbers of a scramjet is auto ignition. Due to relatively low air static temperatures this may become a problem for axial strut injectors which only induce weak shock waves and small recirculation zones downstream of the strut. Thus the advantage of avoiding normal shock waves may cause problems for a stable ignition. Four different modes of combustion may be distinguished for strut injectors:

- A stable combustion where the flame is (attached) anchored directly downstream of the strut,
- A stable combustion with a lifted flame which may be stabilized by shock waves and/or a subsonic zone,

• Blow-off of the flame due to unfavorable thermodynamic conditions or bad mixing,

• Thermal choking where the heat release is too high, the flame is moving upstream, and a normal shock causes subsonic flow in the combustor. The subsequent high temperatures usually will cause damage to the strut.

The last two modes have to be avoided and the second one usually reacts sensitive to changes of the inflow conditions[18].

1.4 THEORY

A short description of the theory behind supersonic flow and Computational Fluid Dynamics is mentioned below.

1.4.1 Supersonic Flow Theory

The foremost and most important consideration with supersonic flow is that the flow is compressible. A compressible flow is one for which the density cannot be considered constant (for flow below M = 0.3 the fluid can be considered to have a constant density). Compressibility leads to two phenomena unique to supersonic flow - shockwaves and expansion waves (Fig.1.10).

Two types of shockwaves prevail: oblique and normal. Oblique shockwaves and expansion waves are generated when a supersonic flow changes direction – a shockwave when the flow converges and an expansion wave when the flow expands.

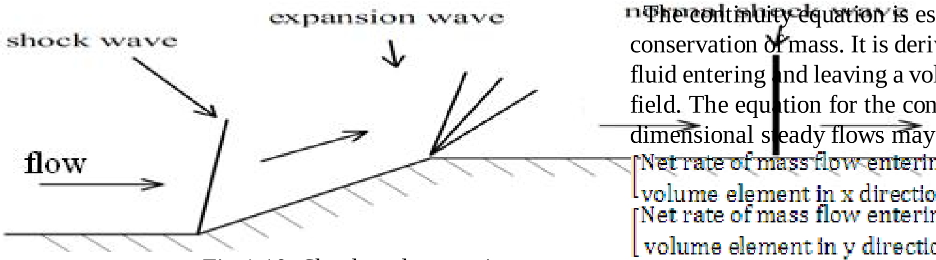

Fig.1.10: Shock and expansion waves

Oblique shock wave

An oblique shockwave is generated when the direction of a supersonic flow changes in a way that is converged. The relative conditions after an oblique shock are the same as for the normal shock – Mach number decreases and pressure, temperature, density and entropy increase. The flow can however remain supersonic. The simplest case is flow over a half wedge.

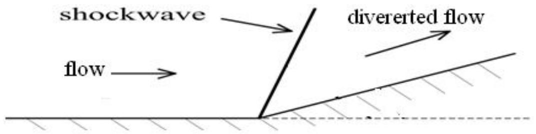

Fig.1.11: Oblique shockwave

Expansion wave

The overall effect of an expansion wave totally opposite to that of a shockwave: Mach number increases and temperature, pressure and density decrease. Entropy however remains constant. Unlike a shockwave, the flow condition across an expansion wave change gradually.

Equivalence Ratio

When referring to the Scramjet engine the term 'equivalence ratio' is often used. The equivalence ratio is a measure that speaks about the 'richness' of the air-fuel mixture. It is defined as the mass flux of fuel divided by the mass flux of air, all divided by this same ratio for a stoichiometric mixture.

1.4.2 Computational fluid dynamics

CFD is a computational software tool for analysis and calculation of fluid mechanical processes, such as mass, heat and momentum transfer. The numerical method most frequently used is the finite volume method. This method is used by both software tools utilized in the present work.

1.4.3 Governing equations of CFD

Each CFD software package has to produce a prediction of the way in which a fluid will flow for a given situation. To do this the package must calculate numerical solutions to the equations that govern the flow of fluids. For the analyst, therefore, it is important to have an understanding of both the basic flow features that can occur, and so must be modeled, and the equations that govern fluid flow. The physical aspects of any fluid flow and heat transfer are governed by three fundamental principles.

- Continuity equation
- Momentum equation and
- Energy equation.

Continuity Equation

The continuity equation is essentially the equation for the conservation of mass. It is derived by the mass balance on the fluid entering and leaving a volume element taken in the flow field. The equation for the conservation of mass for two dimensional steady flows may be stated as (1.4.1)

$$\begin{bmatrix}\text{Net rate of mass flow entering}\\ \text{volume element in x direction}\end{bmatrix}+\begin{bmatrix}\text{Net rate of mass flow entering}\\ \text{volume element in y direction}\end{bmatrix}=0$$

For an incompressible fluid, the continuity equation for a steady two dimensional flow can be written as

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (1.4.2)$$

Momentum Equation

The momentum equation are derived from Newton's second law of motion, which states that mass times the acceleration in a given direction is equal to the external force acting on the body in the same direction. The external force acting on the volume element in a flow field is considered to consist of the body forces and the surface forces.

$$[\text{Mass}]\begin{bmatrix}\text{Acceleration in}\\ \text{i direction}\end{bmatrix}=\begin{bmatrix}\text{Body forces acting in}\\ \text{i direction}\end{bmatrix}+\begin{bmatrix}\text{Surface force acting}\\ \text{in i direction}\end{bmatrix}$$

(1.4.3)

X Momentum:

$$\rho\left[u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right]=F_x-\frac{\partial p}{\partial x}+\mu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right]$$

(1.4.4)

Y Momentum:

$$\rho\left[u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}\right]=F_y-\frac{\partial p}{\partial y}+\mu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right]$$

(1.4.5)

Where F_x and F_y are the body forces per unit volume acting in the x and y direction respectively.

The physical significance of the various terms in equation (1.4.4) is as follows: The terms on the left hand side represent the inertia forces, the first term on the right hand side is the body forces, the second term is the pressure forces, and the last term in the parentheses is the viscous force on the fluid element.

Energy Equation

The temperature distribution in the flow field is governed by the energy equation, which can be derived by writing an energy balance according to first law of thermodynamics for a differential volume element in the flow field. If radiation is absent and there are no distributed energy sources in the fluid, the energy balance on a differential volume element may be stated as

$$\begin{bmatrix}\text{rate of energy}\\ \text{input due to}\\ \text{conduction}\end{bmatrix}+\begin{bmatrix}\text{rate of energy}\\ \text{input due to}\\ \text{work done by}\\ \text{conduction}\end{bmatrix}+\begin{bmatrix}\text{rate of energy}\\ \text{input due to}\\ \text{work done by}\\ \text{surface stress}\end{bmatrix}=\begin{bmatrix}\text{rate of increase}\\ \text{of energy in}\\ \text{element}\end{bmatrix}$$

(1.4.6)

The energy equation for two dimensional flow of an incompressible, constant property, Newtonian fluid is determined as

$$\rho c_p\left[u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}\right]=k\left[\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right]+\mu\phi$$

(1.4.7)

Where the viscosity-energy-dissipation function ϕ is defined as

$$\phi=2\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2\right]+\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right)^2$$

(1.4.8)

1.4.4 Methods of Discretization

There are several methods of discretizing a given differential equation. Some of the discretization methods being used are:

Finite volume method (FVM): This is the "classical" or standard approach used most often in commercial software and research codes. The governing equations are solved on discrete control volumes. FVM recasts the PDE's (Partial Differential Equations) of the N-S equation in the conservative form and then discretize this equation. This guarantees the conservation of fluxes through a particular control volume. Though the overall solution will be conservative in nature there is no guarantee that it is the actual solution. Moreover this method is sensitive to distorted elements which can prevent convergence if such elements are in critical flow regions. This integration approach yields a method that is inherently conservative (i.e. quantities such as density remain physically meaningful).

$$\frac{\partial}{\partial t}\iiint Q\,dV+\iint F\,dA\;=0 \qquad (1.4.9)$$

Where Q is the vector of conserved variables, F is the vector of fluxes, V is the cell volume, and A is the cell surface area.

Finite element method (FEM): This method is popular for structural analysis of solids, but is also applicable to fluids. The FEM formulation requires, however, special care to ensure a conservative solution. The FEM formulation has been adapted for use with the Navier-Stokes equations. Although in FEM conservation has to be taken care of, it is much more stable than the FVM approach. Subsequently it is the new direction in which CFD is moving. Generally stability/robustness of the solution is better in FEM though for some cases it might take more memory than FVM methods.

In this method, a weighted residual equation is formed:

$$R_i=\iiint W_i Q\,dV^e \qquad (1.4.10)$$

Where R_i is the equation residual at an element vertex i, Q is the conservation equation expressed on an element basis, W_i is the weight factor and V^e is the volume of the element.

Finite difference method: This method has historical importance and is simple to program. It is currently only used in few specialized codes. Modern finite difference codes make use of an embedded boundary for handling complex geometries making these codes highly efficient and accurate. Other ways to handle geometries are using overlapping-grids, where the solution is interpolated across each grid.

$$\frac{\partial Q}{\partial t}+\frac{\partial F}{\partial x}+\frac{\partial G}{\partial y}+\frac{\partial H}{\partial z}=0$$

(1.4.11)

Where $Q$the vector of is conserved variables, and F, G, H are the fluxes in the x, y, and z directions respectively.

1.4.5 Turbulence modeling

Turbulence occurs when the inertial forces in a fluid becomes considerable relative to the viscous forces, and is characterized by a high Reynolds number. The most common turbulence modeling approach and also the one used in this thesis is the Reynolds-averaged Navier-Stokes models. RANS is based on a statistical treatment of the flow. More precise, this means that some of the variables that govern the flow are divided into a time-averaged component of the flow and a fluctuating component that represents the deviation from the mean flow. A very successful and widely employed turbulence model is k-ε model. It is a two-equation model meaning that it includes two extra transport equations to represent the turbulent properties of the flow.

1.4.6 Combustion modeling

The most common combustion modeling approach and also the one used in this thesis is the eddy dissipation model. It is based on the assumption that chemical reactions are fast relative to the transport processes of the flow. When the reactants mix at a molecular level they instantaneously form products. The model assumes that the reaction rate may be directly related to the time required to mix the reactants at the molecular level.

1.4.7 Reaction mechanisms

In this thesis single step reaction model (instantaneous model) were used. The fundamental concepts behind these are outlined below. The instantaneous reaction model assumes that a single chemical reaction occurs and proceeds instantaneously to completion. The reaction used for the Scramjet was the hydrogen-water reaction:

$$2H_2 + O_2 \rightarrow 2H_2O$$

2. LITERATURE REVIEW

Peter Hyslop[19] during the course of his project used the Computational Fluid Dynamics package CFD-ACE to model the flow inside the ANU's experimental Scramjet engine. The project was split in three stages: verification of CFD-ACE with previous work; optimization of the Scramjet geometry using CFD-ACE; and experimental verification of the optimum configuration. This included development of numerous CFD models of Scramjet configurations and performing experiments in the T3 shock tunnel. Thrust generated by the Scramjet was calculated using CFD-ACE and included all pressure and shear forces generated on the walls of the Scramjet.

The field of scramjet propulsion faces metallurgical issues as the heat load in the scramjet combustion-chamber is so large that even the most advanced composite materials are not able to withstand it. At the peak value of supersonic air intake in the engine, the air entering is at extreme temperature which is thus unfit for cooling the structure. Hence a regenerative cooling cycle is being proposed so as to thermodynamically optimize the scramjet system, using the fuel (liquid hydrogen) as the primary coolant. Ansh Verma[20] obtained the properties of hydrogen from the National Institute of Standards and Technology [NIST] Chemistry Web-Book

K.M.Pandey and T.Sivasakthivel[21] applied the two-dimensional coupled implicit NS equations, the standard k-ε turbulence model and the finite-rate/eddy-dissipation reaction model to numerically simulate the flow field of the hydrogen fueled scramjet combustor with a planer strut flame holder under two different working conditions, namely, cold flow and engine ignition. They observed that the numerical method employed could be used to accurately investigate the flow field of the scramjet combustor with planer strut flame holder, and capture the shock wave system reasonably..

Malsur Dharavath, P. Manna, Debasis Chakraborty[22] presented Numerical exploration of non-reacting and reacting flow field of hydrogen fueled scramjet combustor .Mixing and combustion of hydrogen fuel injected parallely from the struts into Mach 2 vitiated air stream in a generic scramjet combustor were explored numerically and both non-reacting and reacting flows were simulated. The generic scramjet combustor with hydrogen fuel injected from the base of the strut investigated at DLR in Germany was taken as the test case for validation.

K.M.Pandey and Siva Sakthivel.T[23] worked on the topic of 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. There are number of techniques used today for fuel injection into scramjet engines. Kyung Moo Kim et.al [24] worked on the topic of "Numerical study on supersonic combustion with cavity-based fuel injection", and their findings are – When the wall angle of cavity increases, the combustion efficiency is improved, but total pressure loss increased. When the offset ratio of upper to downstream depth of the cavity increases, the combustion efficiency as well as the total pressure loss decreases. Yuan Shengxue [25] worked on the topic of "supersonic combustion", and his findings are – The calculation of deflagration in supersonic flow shows that the entropy increment and the total pressure loss of the combustion products may decrease with the increase of combustion velocity. The oblique detonation wave angle may not be controlled by the wedge angle under weak under driven solution conditions and be determined only by combustion velocity. Gruenig and F. Mayinger [26] worked on the topic of "Supersonic combustion of kerosene/h2-mixtures in a model Scramjet combustor", and their findings are – The necessary temperature level is partly achieved by the oblique shock waves in the supersonic flow with increasing combustor area ratio. K. Kumaran and V. Babu [27] worked on the topic of "Investigation of the effect of chemistry models on the numerical predictions of the supersonic combustion of hydrogen", and their findings are – Multi step chemistry predicts higher and wider spread heat release than what is predicted by single step chemistry. The single step chemistry model is capable of predicting the overall performance parameters with considerably less computational cost.

Heat sink (cooling capacity) of limited hydrocarbon fuel is not rich for the regenerative cooling of scramjet, this makes it very important to use the heat sink of fuel efficiently. Wen Bao, Xianling Li, Jiang Qin, Weixing Zhou, Daren Yu[28] focused on the effect of operating conditions of cooling system on the heat sink use of hydrocarbon fuel. Considering the coupling among flow, heat transfer and chemical reaction, a one-dimensional model was developed to evaluate the heat sink use of endothermic hydrocarbon fuel.

3. MATERIALS AND METHODS

3.1 Experimental conditions and computational details

The physical model considered here is same as the one considered by Malsur Dharavath, P. Manna, Debasis Chakraborty[22]. The analysis is done based somewhat on the experiment by Waidmann et al. [28-30] but here preheated air enters in the combustion inlet at *Ma* = 3.0. The combustor has a constant area section of 0.058 m from the combustor inlet which is followed by a divergence section (one sided divergent combustor)of 0.242 m length with 3◦ divergent angle at upper wall that is provided to compensate for the expansion of the boundary layer. The width and the length of the combustor are 0.045 m and 0.3 m respectively also the heights of the combustor at inlet and outlet are 0.05 m and 0.068 m respectively. A wedge-shaped strut is positioned at the middle in the combustion chamber at 0.035 m downstream from the combustor inlet. The length and half-wedge angle of the strut are 0.032 m and 6◦ respectively. Hydrogen fuel is injected parallel to the air stream (vitiated air) at Mach number 1.5, through 15 numbers of holes with a diameter of 0.001 m placed 0.0028 m apart (along the width) at the middle of the strut base. Fig. 1 shows the schematic diagram of the combustor geometry with the physical dimensions.

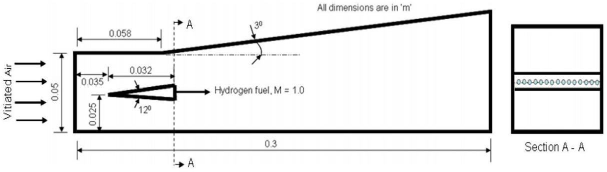

Fig1: Schematic geometry of the supersonic combustion chamber

The flow conditions of the incoming air stream and the hydrogen fuel incorporated are given in Table 1.In the original experiment the combustion was initiated by pre-burning of a small amount of O_2 in a H_2 tube by a spark. Since the injection holes are equal in size and are placed as equidistant along the width, the flow behavior of the holes and adjacent region is almost symmetrical (except the two side wall adjacent holes). That is why, one injection hole along with both side adjacent regions (up to middle of the two adjacent injection holes) has been considered for the computational domain. The computational domain and typical grid structure are shown in Fig. 2.

In the simulation undergone, X-axis is taken along the length, Y -axis along the height and Z-axis along the width with the origin being placed at middle of the bottom wall.

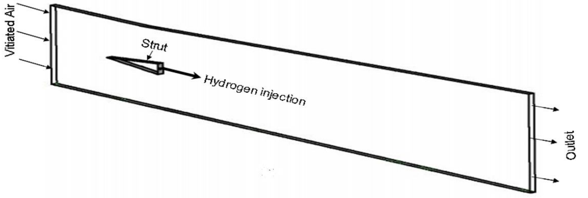

Fig 2 : Modeling of 3D Combustion Chamber and wedge shaped Strut Injector with circular inlet

3.2 Governing equations

There is an advantage of employing the complete Navier-Stokes equations as it 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 the location of shock wave as well as the physical characteristics of the shock layer, can be precisely and near accurately determined.

3.3 Boundary Conditions

In the present study three different types of boundaries are applied: inflow, outflow and fixed walls. The flow fields under consideration here are supersonic. According to the theory of characteristics all variables are prescribed at inflow boundaries, i.e. Dirichlet boundary conditions, and Neumann boundary conditions are used for all variables at outflow boundaries. At fixed walls the no slip condition are applied. All computations are initialized with the state of the incoming air. The flow conditions of the incoming air stream and the hydrogen fuel incorporated given in Table 1.

Table 1: Inflow conditions of the air stream and the hydrogen jet

Parameters	Air	Hydrogen
Mach number	3.0	1.5
Axial Velocity(m/s)	898	1984
Static temperature(K)	300	300
Static pressure(10^5 Pa)	1	1
Density(kg/m^3)	1.002	0.097
O_2 mole fraction	0.232	0
H_2O mole fraction	0.032	0
N_2 mole fraction	0.736	0
H_2 mole fraction	0	1
k(m^2/s^2)	10	2400
ϵ (m^2/s^3)	650	10^8
Mass flow rate (kg/s)	1.5	0.0015 to 0.004

Approximations and Idealizations

- The flow is considered to be in steady state
- The gas is compressible, obeying the ideal gas laws.

4. RESULTS AND DISCUSSIONS

Using definition of Global Mesh settings-**Defaults** to set Physics and Solver preferences, **Sizing** to specify sizing function (curvature, proximity, fixed), mesh sizes, growth rate, etc., **Inflation** for Prism layer growth etc the **Statistics** was viewed to mesh count and mesh quality.

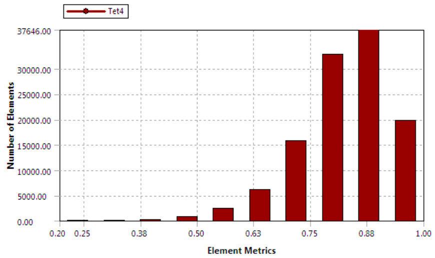

Fig 4.1: Showing the statistics of aspect ratio

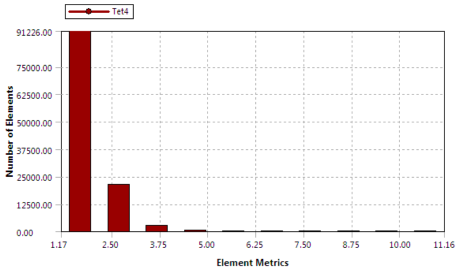

Fig 4.2 : Showing the statistics of element quality

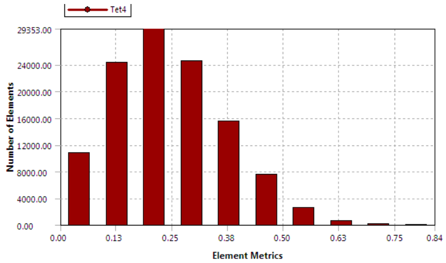

Fig 4.3: Showing the statistics of skewness

The results from the numerical simulation for supersonic combustion using wedge shaped strut injector with circular inlet with non premixed combustion model are discussed below.By defining the inputs for modeling the non-premixed combustion chemistry a Probability Density Function (PDF) table is prepared in FLUENT. To use the non-premixed combustion model, a PDF table is created that contains information on the thermo-chemistry and its interaction with turbulencemodel.

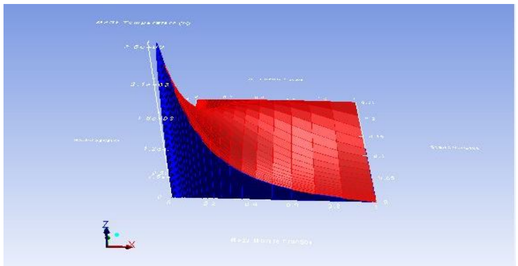

Fig 4.4: display of PDF generated

All thermodynamic data for the continuous phase, including density, specific heat, and formation enthalpies are extracted from the chemical database when the non premixed combustion model is used. These properties are transferred as the pdf mixture material, for which only transport properties, such as viscosity and thermal need to be defined.

Turbulent intensity

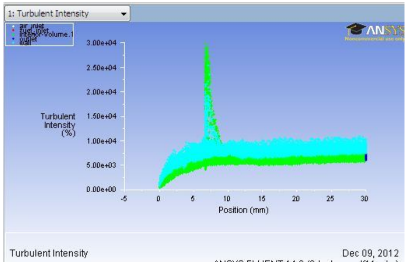

Fig 4.5: X-Y plot of turbulent intensity

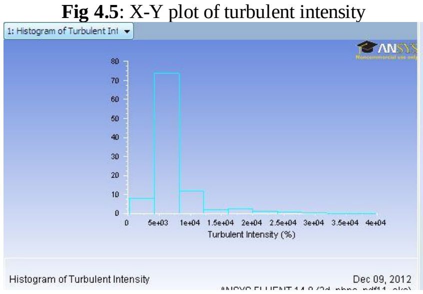

Fig 4.6: Histogram of turbulent intensity

Turbulent Kinetic Energy

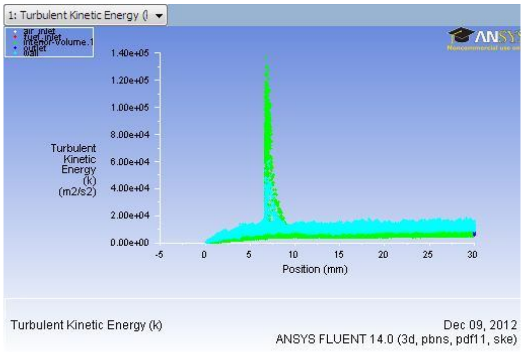

Fig 4.7: X-Y plot of turbulent kinetic energy

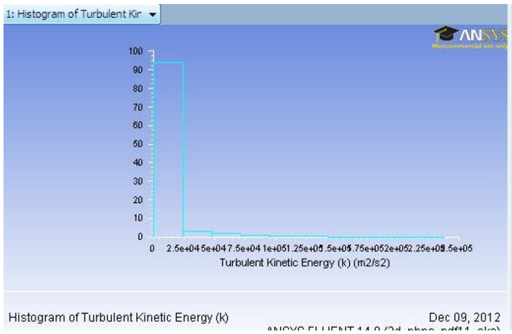

Fig 4.8: Histogram of turbulent kinetic energy

Velocity

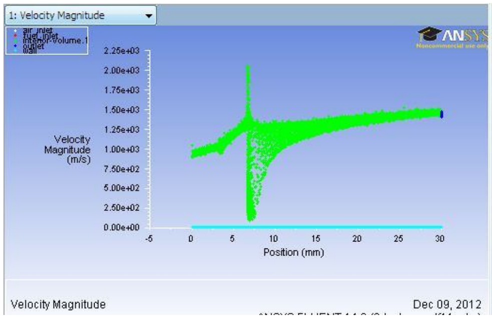

Fig 4.9: X-Y plot of velocity magnitude

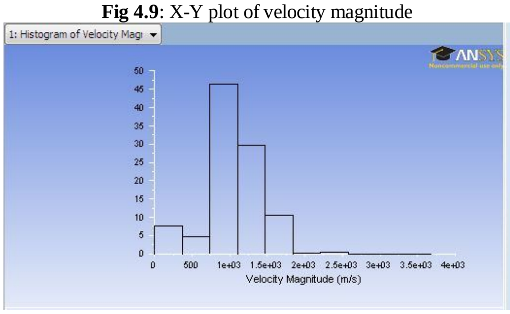

Fig 4.10: Histogram of velocity magnitude

Temperature

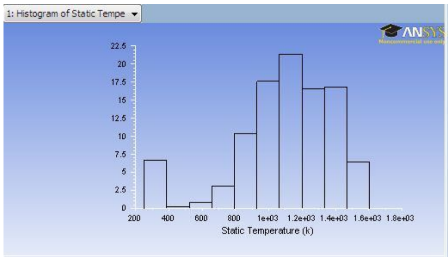

Fig 4.11: Histogram of static temperature

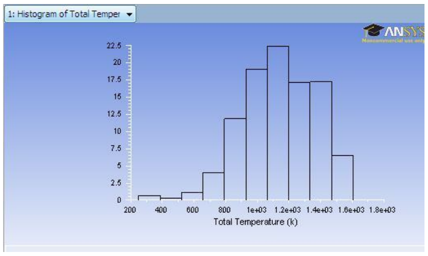

Fig 4.12: Histogram of total temperature

Pressure

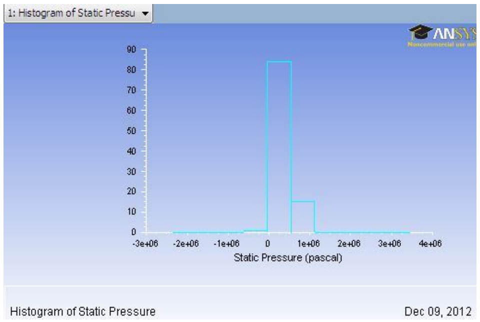

Fig 4.13: Histogram of static pressure

Species concentration

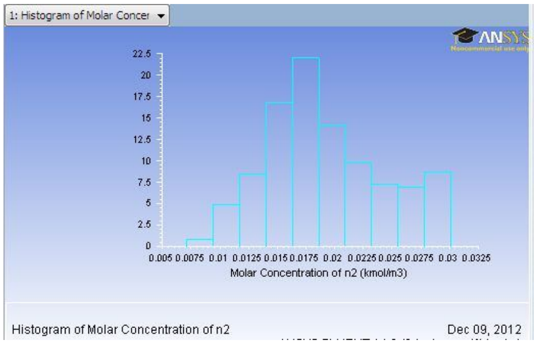

Fig 4.14: Histogram of molar concentration of N_2

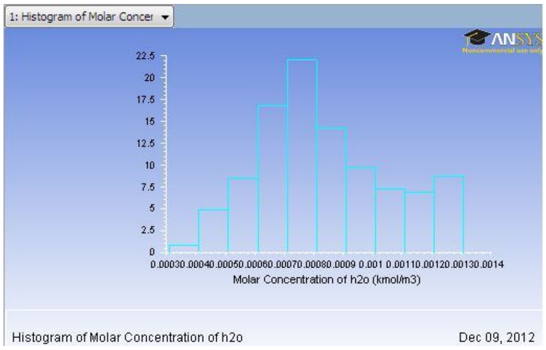

Fig 4.15: Histogram of molar concentration of H_2O

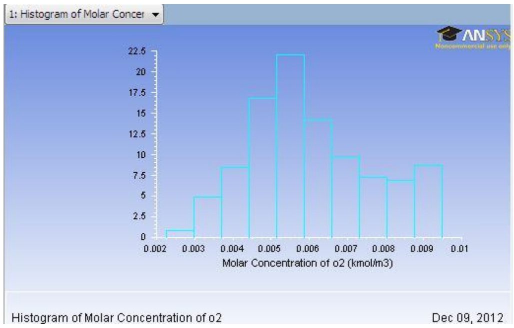

Fig 4.16: Histogram of molar concentration of O_2

From the above results it is observed that the maximum temperature attained is 1604 K. This temperature is found to occur in the recirculation areas which are produced due to shock wave interactions and fuel jet losses concentration. Due to combustion, the recirculation region behind the wedge becomes larger as compared to mixing case which acts as a flame holder for the hydrogen diffusion flame. The leading edge shock reflected off the upper and lower combustor walls hits the wake in a region where large portions of the injected fuel has been mixed up with the air which makes the setting of combustion. Also the maximum velocity is found to be 3683 m/sec.

CONCLUSIONS

The standard k- Є model has been used in the analysis which can provide considerably accurate solutions for turbulent flow only. It does not take into account the laminar flow regions for study. The maximum temperature and velocity predicted by this model are 1604 K and 3683 m/sec. The occurrence of high turbulent intensity represents a high air-fuel mixing.

Further work can be carried out to compare the results by choosing different turbulence models and selecting different Mach numbers.

REFERENCES

[1] Glawe, M. Samimiy, A. Nejad, T. Cheng, "Effects of nozzle geometry on parallel injection from base of an extended strut into supersonic flow", *AIAA paper*, pp 95-0522, 1995.

[2] D. Papamoschou, "Analysis of partially mixed supersonic ejector", Journal of Propulsion and Power, *pp 736–741*, 1996.

[3] J. Belanger, H. Hornung, "Transverse jet mixing and combustion experiments in hypervelocity flows", *Journal of Propulsion and Power*, pp. 186–192, 1996.

[4] A. Gardner, K. Hannemann, J. Steelant, A. Paull, "Ground testing of the hyshot supersonic combustion experiment in heg and comparison with flight data", *AIAA paper*, 2004-3345, 2004.

[5] T. Mitani, T. Kouchi, "Flame structures and combustion efficiency computed for a Mach 6 scramjet engine", *Combustion and Flame* , pp. 187–196, 2005.

[6] I. Waitz, F. Marble, E. Zukoski, "Investigation of a contoured wall injector for hypervelocity mixing augmentation", *AIAA Journal*, pp. 1014–1021, 1993.

[7] D. Riggins, "Thrust losses in hypersonic engines, part 2: Applications", *Journal of Propulsion and Power*, pp. 288–295, 1997.

[8] R. Baurle, R. Fuller, J. White, T. Chen, M. Gruber, A. Nejad, "An investigation of advanced fuel injection schemes for scramjet combustion", *AIAA paper*, pp. 98-0937, 1998.

[9] Strickland, T. Selerland, A.R. Karagozian, "Numerical simulation of a lobed fuel injector", *Journal of Propulsion and Power*, pp. 2950–2964, 1998.

[10] Charyulu, J. Kurian, P. Venugopalan, V. Sriamulu, "Experimental study on mixing enhancement in two dimensional supersonic flow", *Experiments in Fluids*, pp. 340–346, 1998.

[11] T. Sunami, M. Wendt, M. Nishioka, "Supersonic mixing and combustion control using stream wise vorticity", *AIAA paper*, pp. 98-3271, 1998.

[12] Riggins, C. McClinton, R. Rogers, R. Bittner, "Investigation of scramjet injection strategies for high Mach number flows", *Journal of Propulsion and Power*, pp. 409–418, 1995.

[13] Cox, R. Fuller, J. Schetz, "Vortical interactions generated by an injector array to enhance mixing in supersonic flow", *AIAA paper*, pp. 94-0708, 1994.

[14] L.Maddalena, T. Campioli, J. Schetz, "Experimental and computational investigation of light-gas injectors in Mach 4.0 crossflow", *Journal of Propulsion and Power*, pp. 1035–2006, 2006.

[15] T. Sunami, K. Itoh, T. Komuro, K. Sato, "Effects of streamwise vortices on scramjet combustion at Mach 8–15 flight enthalpies – an experimental study in heist", *ISABE paper*, pp. 2005-1028, 2005.

[16] E. Fernando, S. Menon, "Mixing enhancement in compressible mixing layers: An experimental study", *AIAA Journal*, pp. 278–285, 1993.

[17] J.-H. Kim, Y. Yoon, I.-S. Jeung, H. Huh, J.-Y. Choi, "Numerical study of mixing enhancement by shock waves in model scramjet engine", *AIAA Journal*, pp. 1074–1080, 2003.

[18] T. Sunami, A. Murakami, M. Nishioka, "Mixing and combustion control strategies for efficient scramjet operation in wind range of flight Mach numbers", *AIAA paper*, pp. 2002-5116, 2002.

[19] Peter Hyslop "CFD Modelling Of Supersonic Combustion In A Scramjet Engine", *Department Of Engineering Australian National University,1998*

[20] Ansh Verma "Ameliorative Study of a Scramjet Engine by Regenerative Cooing using Finite Element", *IACSIT International Journal of Engineering and Technology, ISSN: 1793-8236, Vol.2, No.6, December 2010.*

[21] K.M.Pandey and T.Sivasakthivel "CFD Analysis of Mixing and Combustion of a Scramjet Combustor with a Planer Strut Injector",*International Journal of Enviromental Science and Development, Vol. 2, No. 2,* April 2011

[22] Malsur Dharavath, P. Manna, Debasis Chakraborty "Thermochemical exploration of hydrogen combustion in generic scramjet Combustor",*Computational Combustion Dynamics Division, Directorate of Computational Dynamics, Defence Research & Development Laboratory, P.O. – Kanchanbagh, Hyderabad,India*

[23] K.M.Pandey and T.Sivasakthivel, "Recent Advances in Scramjet Fuel Injection - A Review," *International Journal of Chemical Engineering and Applications* vol. 1, no. 4, pp. 294-301, 2010.

[24] Kyung Moo Kim 1, Seung Wook Baek and Cho Young Han, "Numerical study on supersonic combustion with cavity-based fuel injection", *International Journal of Heat and Mass Transfer*, vol 47, pp.271–286, 2004.

[25] Yuan Shengxue, "Supersonic combustion", *Science in China (Series A),* vol. 42, no. *2,* February 1999,

[26] Gruenig and f. Mayinger, "Supersonic combustion of kerosene/h2-mixtures in a model scramjet combustor", *Institute for thermodynamics*, technical university Munich.

[27] K. Kumaran and V. Babu, "Investigation of the effect of chemistrymodels on the numerical predictions of the supersonic combustion of hydrogen", *Combustion and Flame*, vol 156, pp.826–841, 2009.

[28] W. Waidmann, F. Alff, M. Bohm, U. Brummund, W. Clauss, M. Oschwald, "Experimental investigation of hydrogen combustion process in a Supersonic Combustion Ramjet (SCRAMJET)", *DGLR Jahrbuch*, pp. 629–638, 1994.

[29] W. Waidmann, F. Alff, M. Bohm, U. Brummund, W. Clauss, M. Oschwald, "Supersonic combustion of hydrogen/air in a scramjet combustion chamber", *Space Technology*, pp. 421–429, 1995.

[30] W. Waidmann, U. Brummund, J. Nuding, "Experiments investigation of Supersonic Ramjet Combustion (Scramjet)", *8th International Symposium on Transport Phenomena in Combustion*, San Francisco, USA, 1995.