CHAPTER 2
Numerical Study on the Mixing Characteristics of a Combustor Using Pulsed Fuel Injector for a Scramjet

Dr. P. Maniarasan
Nehru Institute of Engineering and Technology, India
Dr. M. Santhosh
Nehru Institute of Engineering and Technology, India
Mr. S. Balaji
Nehru Institute of Engineering and Technology, India
Dr. D. Satish Kumar
Nehru Institute of Technology, India

ABSTRACT

Today's fuel injection methods for scramjet engines are still an emerging topic. Usually, liquids or gases are used as the fuel for scramjets. To ensure effective and efficient combustion, the fuel and air need to be mixed exactly in stoichiometric proportions. Effective and efficient fuel–air mixing plays a critical role in the successful operation of scramjet engines. In this work, the pulsed fuel injection technique in a realistic scramjet combustor flow condition with five different geometries is numerically investigated to improve the fuel-air mixing in supersonic combustion systems with a short flow residence time. The time dependent sinusoidal pulse is achieved by using an mathematical expression given at the fuel inlet. For the investigation, the two equation k-ω SST turbulence model is implemented, and Reynolds-Averaged Navier-Stokes equations are solved using ANSYS FLUENT. The findings demonstrate that extremely high amounts of turbulent kinetic energy (TKE) are seen particularly in the downstream side of the fuel injectors and low percentage of unburnt hydrogen is observed at the outlet. Increased TKE and less percentage of unburnt hydrogen at the outlet is proven to improve mixing efficiency for all types of pulsed fuel injection .

Keywords:*efficient combustion,ANSYS FLUENT, SST turbulence model etc*

INTRODUCTION

The basic principle of the Computational Fluid Dynamics (CFD) is to reduce the cost of the experimental studies. Analysis and design of the Scramjet engines are not an exception to this trend. In order to reduce the number of experiments and so, reduce the costs of design and speed up this process, Computational Fluid Dynamics tools are being used. Different algorithms and methods are being improved and

ISBN: 978-93-5980-498-9, www.aetsjournal.com — Handbook of research in Modeling of Transportation and Aviation Processes

implemented to these CFD tools to have a better understanding of the problems prior to their real life applications.

In this thesis, a CFD tool is developed to analyze the flow through the Scramjet engine combustor. Since the experimental studies of the high speed flows and especially hypersonic speeds are cumbersome and expensive, the use of Computational Fluid Dynamics tools become more crucial. Testing of Scramjet engines require highly developed laboratories with high-tech instruments which are only available in some of the countries today. In order to improve the vehicles flying at hypersonic speeds faster and with a lower cost, most of the design and analysis work should be done with modeling in computer areas, i.e. using CFD tools. Scramjet combustion chamber is basically the most important part of the Scramjets. The design and analysis of the combustor is difficult compared to other parts of the engine since peak temperatures occurs at this part due to combustion process. In this thesis, the analysis of Combustor is done by developing a CFD tool which takes into account most of the processes happening in the combustion chambers such as mixing of the fuel-air, reactions, flame holding, etc.

LIMITATION OF THE STUDY

In this thesis, the analysis are done in very high mach number in both steady and transient state for comparison of results with simple geometry. The analysis are limited to the combustion chamber of the Scramjet engine. Other parts of the engine namely inlet, isolator and nozzle is not part of analysis.

LAYOUT OF THE STUDY

Chapter 2 encloses the literature study about scramjet engines and hypersonic flow. This includes historical background of the supersonic air- breathing propulsion systems with a focus on scramjet engines. Different parts of the scramjet engine with their working principles are introduced. Different methods used to enhance the performance of the supersonic combustion are presented.

Chapter 3 gives theoretical background of the scramjet engine governing equations. The equations of motions for the flow analysis of a scramjet combustor analysis are briefly explained.

Chapter 4 contains the computational algorithms and methods used to solve the problem for scramjet combustor. In this chapter, numerical discretization of the equations are shown.

In Chapter 5, the results and contours were taken for five different geometries using normal and pulsed fuel injection technique. Analysis of the model is done and variation of flow variables and species are demonstrated by 2D contours.

Chapter 6 contains the general conclusions of the study. Moreover, the recommendation for the future work is also provided in here.

LITERATURE REVIEW

This chapter is devoted to historical background of the development of the Scramjets. Moreover, brief explanation of the Scramjet working principles for different parts of the engine especially for combustion chamber is given. Most recent researches on the development of the cavity recessed scramjet combustors is presented.

INTRODUCTION TO SCRAMJET ENGINES

Scramjet engines are named as Supersonic Combustion RAMJETS (SCRAMJETS). Since Scramjets are descendants of the ramjets, the definition of ramjets should be given in advance.

Ramjet engines are kind of air-breathing propulsion systems which uses the forward motion of the engine and specific inlet design to compress the air without the use of axial compressor. Therefore, Ramjet engines have no moving parts. Ramjet compresses the oncoming flow by passing it through one or more oblique shocks. After decelerating the flow into subsonic speeds the combustion occurs. Since ramjet engines need an oncoming flow at supersonic speeds, they are unable to operate at zero speed, i.e. at take-off. As a result, assisted take-off, by using other engine types will be required during take-off.

Since at high Mach numbers, typically above 5, decelerating the flow into subsonic speeds cause excessive pressure and temperature increase in the combustion chamber, the use of ramjet engines became non-profitable at these high speeds. Therefore, the idea of the supersonic combustion is risen where the incoming flow is not needed to be decelerated to subsonic speeds. Therefore, the new type of ramjets for which combustion occurs at supersonic speeds are introduced as supersonic combustion ramjet (Scramjet).

HISTORICAL BACKGROUND

More than a century has passed since Rene Lorin discovered the idea of using ram pressure (pressure that is exerted on a body when it flows through a fluid) in propulsion systems in 1913 . There has been lots of improvements in the area of hypersonic propulsion systems since then and many studies and experiments are done. Here some of the most important contributions is highlighted.

Rene Leduc started the conceptual design of the ramjet engine in 1920s but, got the patent with an airplane that has a ramjet in 1934 . In 1928, Albert Fono from Hungary, patented by designing a propulsion system which had all of the components of today's ramjets. Unfortunately, the propulsion system designed by Fono was never built. Although Leduc got patent in 1934, the flights was delayed because of World War II. In 1946, experimental aircraft (Leduc 010) was constructed with the concept of the Leduc's design and its first powered flight took place in 1949. Interests in ramjet reached its maximum in 1950s and a lots of researches were done at that time.

However, the development of the scramjet did not start until the late 1950s or early 1960s. In 1958, Weber and McKay discovered that by the use of shock-wave interactions, efficient combustion (reduced loses) can be considered for flows at supersonic speeds. They indicate that with the proper inlet geometry, the scramjet can be more efficient than ramjet at flight speeds exceeding Mach 5. In 1960s, Antonio Ferri demonstrated a scramjet for the first time. Following the works of Weber, McKay and Ferri, lots of research projects on the development of the scramjet engines were started.

The most important of these projects was NASA's HRE (Hypersonic Research Engine) project. The primary aim of the project is to test a hypersonic scramjet engine in flight using X-15A-2 research plane which should be modified to carry hydrogen for scramjet engine. Since the repair expenses of the X-15A-2 became too high, the X- 15 project was cancelled in 1968 and the in flight test of the scramjet engine was not achieved . Most of the research projects until then and afterwards for some years were continued on the development of scramjet by ground testing.

The first successful verified scramjet combustion in flight environment was achieved by Australian HyShot program in July 2002. The engine operated effectively and demonstrated the supersonic combustion. However, it was designed for the purpose of demonstrating the technological achievement of

ISBN: 978-93-5980-498-9, www.aetsjournal.com — Handbook of research in Modeling of Transportation and Aviation Processes

the supersonic combustion and it is not used to provide thrust to propel an aircraft. Later in 2004, the first flight with scramjet propulsion system which produced thrust is achieved by X- 43A (Figure 2.2). On May 2010, X-51A wave rider broke the record of longest hypersonic flight time of 140 seconds by using Pratt & Whitney Scramjet engine.

OVERVIEW OF THE SCRAMJET COMPONENTS

At high Mach numbers, above 3, pressure increase can be achieved by changing the inside geometry of the engine. In other words, there is no need for rotating elements such as compressors to increase the pressure of the incoming flow. Scramjets are designed based on this principle. Scramjet lacks any moving part and provides pressure increase necessary for the burning cycle by changing shape and area of the engine's inner geometry . Scramjet engines basically consist of four main parts; inlet, isolator, combustor and nozzle. Theoretically scramjet design is very simple but practically, many problems arise which will be mentioned later. The components of the scramjet engine are shown schematically in Figure 2.3.

The inlets are designed to capture the air needed for the engine, decelerate the flow into speeds required by the engine with least possible loss in the total pressure and generating drag as small as possible. Inlets of the scramjet engines are not favored to be separated from the fuselage. This is due to the fact that separating the body and engine will cause an increase in drag of the vehicle. Moreover, hypersonic boundary layers do not separate commonly compared to flows at low Mach numbers. Inlet design for the scramjets can be challenging because of the machinery limitation. To clarify, cooling might be needed for the inlet material and inlets might be required to change their geometry while flying. In other words, they can be required to have variable geometries to adapt for low and very high Mach numbers .

Increased heat values because of the combustion occurring in the combustor cause back pressure and reduction in the mass flow of the air which effects the inlet flow and may cause unstart. In order to prevent unstart, a component named 'isolator' is placed between inlet and combustor. Isolator contains the shock train created by the back pressure of the combustor and prevents it from reaching to the inlet.

Combustion chamber is a component where mixing of the fuel-air and supersonic combustion occurs. Detailed information about the scramjet combustor is given in the following section.

Nozzle is desired to expand the flow beyond combustor. Nozzle of a scramjet is a divergent duct which is suggested to be open type nozzle in order to adapt for the large pressure ratio required . Open type are kind of nozzles which use vehicles aft-body as part of the nozzle.

COMBUSTION CHAMBER

In scramjet, the combustion occurs at supersonic speeds. Since decelerating the flow into subsonic regions decreases the efficiency of the engine and cause other machinery problems, supersonic combustion is necessary at high Mach numbers . However, air entering combustion chamber at supersonic speeds results in additional difficulties which needs to be overcome. This makes the design of the combustion chamber to be the most challenging part of the scramjet propulsion system design. Since flow must be maintained at supersonic speeds throughout the combustor, the following problems arise

- Poor fuel-air mixing rate
- Reduced residence time
- Difficult flame holding

ISBN: 978-93-5980-498-9, www.aetsjournal.com — Handbook of research in Modeling of Transportation and Aviation Processes

In order to overcome these problems, several studies were done and

some methods were proposed. Most of the proposed ideas, solved some of the aforementioned difficulties but they also possessed some new issues. The main idea behind most of these methods is to use physical obstacle in the combustor. Obstacles in the flow path enhances the mixing and combustion efficiency by increasing the residence time. However, physical obstacles need cooling which is a severe problem in high enthalpy flows. Moreover, pressure loss and increase in drag are additional problems. Some of the proposed and studied methods are introduced here.

RAMP INJECTORS

In order to enhance the mixing in the combustor, ramp injectors can be used. Ramp injector's principle is to increase the fuel-air mixing by adding axial velocity to the parallel injection. This type of mechanism increases the mixing by producing counter-rotating vortices and creating shock and expansion waves caused by supersonic flow passing over the ramps. There are two types of ramp injectors; compression and expansion ramps. Both types are illustrated in Figure 2.4. Ramps which are raised in the flow path are compression types while the ramps recessed in the floor are of expansion type. Compression ramps create a stronger vortex but, since they do not reach to the smaller scale, expansion ramps result in better combustion efficiencies. Moreover, expansion ramps attain their maximum efficiencies in less distance then the compression ramps .

Although Ramp injectors enhance the fuel-air mixing and combustion efficiency, they possess some crucial disadvantages. Since the fuel is injected along the wall, mixing can only be done only near to the wall until the shear layers expand enough through the core flow which will happen at the far downstream . In addition, placing obstacles in flow path will cause pressure losses and consequently increase in the drag. Physical obstacles in high enthalpy flows creates high temperatures which cause severe problems for the materials used.

STRUT INJECTORS

Struts are placed vertically in the combustion chamber from bottom to the top. Struts are designed with a wedge at the leading edge and fuel injectors at the trailing edge. Fuel injection is efficient in struts because the fuel is added to the flow throughout the whole flow field from several locations of the trailing edge. However, since struts are in-stream devices, they have significant pressure losses and remarkable contribution to the drag

To decrease the pressure loss, some researchers tried to modify the leading and trailing edge of struts. Considering this, NASA conducted experiments with different variations in the shape of the struts to determine the effect of different parameters such as thickness, length, leading edge sweep, etc. . It is concluded that thickness of the strut has the most contribution to the drag. Moreover, a study on wedge-shaped and diamond-shaped strut injection and its results show that the wedge-shaped strut injectors are more efficient . Although strut injectors provide proper fuel injection to the flow but, high pressure losses and increase in drag because of pressure loss has not been solved yet.

PULSED INJECTOR:

Another type of fuel injection is pulsed injection conventionally;

Fuel is injected as a continuous stream from injection ports into the combustion chamber where it ignites. This type of injection injects the fuel in a series of pulses, which allows for greater mixing between the fuel and air. Combustion occurs more rapidly as well as more efficiently, thus producing a greater thrust output. The time between pulses is dependent on the free stream conditions, and is coordinated to achieve near stoichiometric combustion. An advantage of this method is that combustion always remains in a transient state, and never reaches a steady state condition. Transient combustion further enhances fuel-air mixing, as well as allowing for a greater dispersal of the heat load on the combustor

CAVITY FLAME HOLDERS

In late 90s, cavity flame holders were proposed as a new concept for flame stabilization in supersonic combustion chambers . In this concept, fuel injection methods are combined with flame holding techniques. Cavities were first employed by Central Institution of Aviation Motors (CIAM) in Russia. Cavity technique were first used for flight tests of a Russian/French dual-mode Scramjet . In the following experiments it is observed that implementations of cavities increased the hydrocarbon combustion efficiency remarkably .

The principle idea behind cavity technique is to create a recirculation region where the mixing of the fuel and air occurs at relatively low speeds . Since cavities are recessed in the combustor, pressure losses are decreased compared to other techniques where the devices are placed in-stream. By creating low speed recirculation regions, cavities increase the residence time and so, mixing and combustion becomes more efficient and stable. Since there are many factors affecting the performance of the cavity flame holders such as cavity geometry, fuel injection patter and fuel type and so on, cavity stabilization method has not been fully understood yet. Many researches are going on studying cavities with different geometries and varying characteristics. Generally, cavities are separated into two categories regarding their geometry: open and closed cavities . The parameter used to characterize cavities is Length- to-Depth ratio (L/D) of the cavity. Cavities are called “open” when their length to depth ratio is lower than ten (L/D < 10). In open cavity flows, the shear layer separated from the upstream corner reattaches to the aft-wall of the cavity downstream. In open cavity flow regimes with smaller aspect ratio (L/D < 2-3) where single large vortex is formed in the cavity, transverse oscillations are dominant. Whereas, in open cavities of higher length to depth ratio where cavity is filled with several vortices (Figure 2.6), longitudinal oscillations are more dominant . Shear layer impingement at the aft- wall of the open cavities results in high pressure and so increases the drag.

The second category of the cavities which are called “closed”, have length to depth ratio higher than ten, L/D > 10. In closed cavity flow regimes, shear layer formed at the upstream corner cannot pass the entire cavity and reattaches to the cavity floor. Closed cavities generate great drag coefficients because of pressure increase at the aft- wall and pressure decreases at front wall . Drag values are higher in closed cavities compared to open ones. Hence, open cavities are preferred for the use in scramjet combustion chambers.

A study from Hsu et al. has shown that closed cavities results in unstable flames while open cavities with low aspect ratio does not provide the volume necessary for flame holding. Stable combustion was achieved for a limited range of L/D which corresponds to minimum drag and entrainment.

The most efficient cavities are open cavities with higher aspect ratios. However, in this flow regimes oscillations are controlled by longitudinal mechanisms. The longitudinal oscillations are explained with two basic models . As shear layer forms at the upstream corner of the cavity and then reattaches to the aft-wall causes an increase in the cavity pressure. Therefore a compression wave is being generated which travels upstream to the front wall. The first model suggests that this wave produces vortices in the front wall which amplify while traveling downstream as can be seen in the Figure 2.8.

a) First Model   b) Second Model

OSCILLATIONS

Unlikely, the second model proposes that shear layer deflection is a result of compression wave reflection of itself from the front wall, not the induction of vortices (Figure 2.8). Since shear layer interaction with the cavity aft-wall is the basic factor for fluctuations in the cavity, controlling the formation of the shear layer can prevent the cavity oscillations. For this purpose, passive and active methods to control the shear layer are introduced. In passive methods, control of the shear layer is done by installing vortex generators or spoilers upstream of the cavity or by inclining the cavity aft-wall. In this way, formation of shear layers and their reattachments to the aft-wall will be controlled and consequently compression waves

will not be reflected into the cavity (Figure 2.9.). On the other hand, in active control methods, formation of shear layer is controlled by mechanics, acoustic or fluid injection methods. Upstream mass injection as can be seen. Since active control methods can be adapted to various conditions, they are more efficient than the passive methods.

Ben-Yakar demonstrated the effect of inclined cavity aft-wall on the reattachment of the shear layer and its stabilizing effect. From the experiment results, it is concluded that for cavity with 90 degree aft-wall, compression waves were propagated into the cavity. This occurs after the generation of shock waves in the shear layer reattachment location, at the cavity trailing edge. However, in the inclined cavities, shear layer reattaches to the aft-wall in a steady manner so that no acoustic waves are reflected inside of the cavity.

Figure 2.12 Steady Shear Layer Reattachment at Inclined Cavity Aft- wall. Pressure drag produced in the cavity is a result of following reasons. First, pressure difference between the aft-wall back face and free-stream pressure. This pressure difference will cause net force in the x-direction, i.e. drag force will be generated. Second, high pressure region will be generated at the location of shear layer reattachment. This creates a net force in x-direction on the front wall face inside the cavity .

Experimental studies of Zhang and Edwards shown that in longitudinal mode ($L/D > 3$), cavity drag increases significantly as L/D increases. This is due to the fact that by increasing L/D, oscillations are damped at the reattachment location and so pressure increases at the aft-wall of the cavity. Moreover, because of momentum diffusion, pressure drops at the front wall. These pressure fluctuations cause an increases in the cavity drag as shown in Figure 2.13.

As can be concluded from the aforementioned studies, cavity drag is mostly affected by the aft-wall of the cavity. Therefore, several studies are done regarding the design of back wall. Researches of Gruber et al. demonstrated that drag increases for small angles ($\theta = 16$) of the aft-wall. Small angles lead to creation of expansion waves at the cavity upstream which increases drag since pressure decreases at the aft- wall face. Therefore, large pressure difference contributed to high drag values. However, the studies of Zhang et al. shows that decreasing back wall angle from $\theta = 90$ to $\theta = 45$, decreased the drag coefficient. In another study from Samimy et al. a cavity with aft-wall angle of 20 degree is chosen to minimize the pressure difference upstream and downstream of the cavity. This is the key for decreasing the cavity drag. From all these studies it can be concluded that minimum drag coefficient may occur at the wall angle between $\theta = 16$ and $\theta = 45$ degrees.

NUMERICAL METHODS

ISBN: 978-93-5980-498-9, www.aetsjournal.com ― Handbook of research in Modeling of Transportation and Aviation Processes

In order to analyze the flow in the scramjet combustor different computational models can be implemented. Reynolds averaged Navier-Stokes equations (RANS), Large Eddy Simulations (LES) and Direct Numerical Simulations (DNS) are used for simulation purposes. DNS is very expensive to solve the complex reacting flows. Therefore, RANS and LES methods are the mostly used methods in flow analysis of the scramjets. Baurle and Eklund used the VULCAN Navier-Stokes code in their studies. In this code, Reynolds averaged equations were solved with cell-centered finite volume method. For chemical reaction modeling, finite rate kinetics model was implemented. In their study, Menter Baseline (BSL) and Menter Shear Stress Transport (SST) turbulence models were used. Edwards low- diffusion flux split scheme was employed with Van Leer flux limiter.

Lin et al. used the CFD++ code to perform analysis and simulations on scramjet model. The simulations were done based on finite volume method, multi- dimension TVD (Total Variation Diminishing) schemes and Harten-Lax- van Leer- Contact (HLLC) Riemann solver with minmod flux limiter. Two- equation $\kappa-\varepsilon$ turbulence model was employed. Reduced finite rate kinetics model was used for modeling reacting flows. In a study of Ghodke et al. [27], second order accurate block structured finite volume method was used to solve the LES equations. Linear Eddy Mixing (LEM) and Artificial Neural Network (ANN) models were used for chemical reaction calculations. In order to evaluate the fluxes a hybrid methodology characterized by shock/interaction interactions was implemented.

PHYSICAL MODELING

In the scramjet combustor, the flow is characterized by the fluid motion along with chemical reactions. In other words, reacting flow is the basic characteristic of the flows which experience combustion. Governing equations of these flows are principle conservation laws of fluid mechanics and chemical reaction mechanisms. Theoretical background are given as follows.

Governing Equations of Fluid Motion

Differential form of the conservation of mass, momentum and energy are applied for a distinctive volume of the fluid which can be shown mathematically in the following manner for Cartesian coordinate systems.

Conservation of mass in its differential form which is known as

continuity equation is given as:

Momentum equations which are referred to as Navier-Stokes equations are shown in the following.

Momentum in the x-direction:

Momentum in y-direction:

Momentum in the z-direction:

Conservation of Energy:

In addition to fundamental conservation laws of fluid mechanics, conservation of mass should be applied for every species in reacting flows.

FUEL TYPES

Fuel selection is one of the important parameters in designing the propulsion systems. Besides to its effect in engine performance, it has great effect in the design of the vehicle. For instance, choosing a fuel with higher density will occupy less volume in the vehicle and so will increase the room available for other parts such as payload areas and etc. Therefore, fuel should be selected not only to provide good performance and efficiency for the engine but also, to increase the engine's application potential .

Throughout years, several fuels have been tested to be used in scramjets. Among those fuels, hydrogen and hydrocarbon fuels are used extensively. Fry shows that for flight Mach numbers above 8, hydrogen fuels are beneficial. On the other hand, for scramjet powered flight Mach numbers below 8, hydrocarbon fuels should be preferred which can be seen from Figure 3.1. However, in a study of Waltrup it is concluded that hydrocarbon fuels can have better performance than hydrogen fuels up to Mach numbers of 10. Certainty of the efficiency of hydrogen in high Mach numbers above 10 is due to its fast reaction rate and cooling capacity compared to hydrocarbons.

Overall, hydrogen fuels have the advantages of fast burning, short ignition delay time and high cooling capacity. However, its low density and requirement for large fuel tank area are disadvantageous. Unlikely, hydrocarbon fuels are advantageous because of high density (require less volume), easy handling (not highly reactive) and more energy per volume. Hydrocarbon fuel have disadvantages of slow reaction and long ignition delay.

COMPUTATIONAL MODELING

Basic principle of Computational Fluid Dynamics (CFD) is to model flows as realistic as possible. However, there are many obstacles to accomplish this goal. In order to model the fluid motion precisely, solution of the complex equations are needed for complex problems. These models require high computation time and very large computers. Therefore, solving these complex equations becomes inefficient both computationally and economically. For this purpose, lots of research are done to get better solutions with rather simplified equations and reduced CPU times. Numerical algorithms and methods are developed to solve the flow problems accurately with less computation time i.e. to reach the goal of accurate modelling with efficient computation process.

In this study, the coupled equations of Navier-Stokes and finite rate chemical reactions are solved. In order to have efficient computation, computational space is transformed from Cartesian coordinates to generalized coordinates. Moreover, different flux splitting methods are introduced to the solution algorithm. First and second order schemes are presented with the addition of flux limiters for second order scheme.

NAVIER-STOKES EQUATIONS IN CARTESIAN COORDINATES

Three dimensional steady coupled equations of Navier-Stokes and finite rate chemistry model equations can be written in vector form in the generalized coordinate as Equation (4.1). Flux vectors are divided into convective (inviscid) and viscous fluxes for better representations of the flow physics and simplifying the equations for the numerical approach.

$(Fc - Fv) + (Gc - Gv) + (Hc - Hv) - S$

$= 0$

$\partial x \qquad \partial y \qquad \partial z$

ISBN: 978-93-5980-498-9, www.aetsjournal.com ― Handbook of research in Modeling of Transportation and Aviation Processes

where Fc, Gc and Hc are inviscid (convective) flux vectors and Fv, Gv and Hv are viscous flux vectors in x, y and z-direction, respectively. Moreover, S is the source.

In order to reduce the computation time, the conservation of mass equation for one of the species is removed. This is possible because of the fact that summation of all of the densities should be equal to the total density of the flow and since total density is also introduced to the equation, one of the species' density can be removed from the equations to reduce the size of the vector. In other words, independent species one less than the total number of species.

NUMERICAL DISCRETIZATION

Governing equations of fluid motion can be written in both differential and integral form and solved accordingly. The methods used to solve these equations have different characteristics. Therefore, methods which are advantageous for the problem and domain in question should be applied. In other words, there is no globally efficient method for different type of problems. Basic principle of numerical approximations is to divide the solution domain into discrete points, areas or volumes according to domain dimensions. In order to solve the differential form of the governing equations finite difference method (FDM) is generally utilized. Finite difference methods are accurate in computational domains which are divided by equally spaced points. Hence, implementation of finite difference equations for complex geometries with discontinuities will cause numerical problems. Whereas, finite element methods (FEM) and finite volume methods (FVM) attempt to solve the governing equations of fluid motion in their integral form. In order to use finite element method, the domain of solution should be divided to finite number of elements where approximations are done by using interpolation function at each element. Unlike finite difference method, FEM can handle complex geometries with good accuracy. However, finite element methods require high CPU time for fluid flow problems . In order to solve flow problems in complex geometries with less computational time, finite volume method is introduced.

FINITE VOLUME METHOD

In finite volume method, discretization of the domain is done by dividing the solution domain into small (finite) volumes or cells. Then, integral form of the governing equations are applied to each volume. The goal is to approximate the flow variables defined at each cell in a way to approach the exact solution for which conservation laws are satisfied. Since conservation laws are stronger in integral form, it is physically intuitive to apply equations of motion in their integral form for complex flows. Therefore, finite volume method is promising method for flows with discontinuities such as shocks and complex geometries.

ORDER OF ACCURACY

In order to provide the information for the flux vectors, flow variables from the centers of the neighboring cells should be implemented into the flux calculation at the cell interface. To do this, first or second order interpolation is applied.

FIRST ORDER SCHEMES

In first order schemes, flow variables at cell interfaces are taken to be equal totheir values at cell centers to the left and right of that interface. First order TVD (Total Variation Diminishing) schemes, are monotone schemes. In other words, first order schemes have dissipative characteristics and tend to decrease the effect of discontinuities in the flow (such as shocks) or in the geometry. To capture the effect of discontinuities precisely, second or higher order schemes can be employed.

SECOND ORDER SCHEMES

In second order schemes, the information of the flow variables at cell interfaces are computed by interpolating the flow variable values at the neighboring cells. The variation of flow variables between the cells are taken into account in second or higher order schemes. By increasing the order of interpolation, higher order schemes can be constructed. Second order schemes can be established for the left and right values of the flow variables using the MUSCL (The Monotonic Upstream-Centered scheme for Conservation Laws) scheme

Constructing higher order schemes improves the accuracy of the solutions. Unlike first order schemes, higher order schemes do not have dissipative characteristics. In the regions of discontinuities, the second order schemes tends to capture abrupt changes in the flow variables precisely. Second order schemes provide more information about the flow variables by taking the small variations into account. Depending on the complexity of the flow or geometry and the stability of the numerical methods applied, oscillation may be generated and the solutions might not be converge. In order to avoid these problems at local discontinuous regions, flux limiters can be employed.

BOUNDARY CONDITIONS

Implementation of boundary conditions can be difficult depending on the physical domain of the problem. However, transforming the solution domain from physical to computational space makes the application of boundary conditions simpler . Since in this study, the computational space is used to solve the equations of fluid motion, enforcement of boundary conditions are rather simple.

In order to employ the boundary conditions properly, ghost cells are generated at the outside of the computational boundaries. The ideal is to assign values of the flow variables in these computational cells such that the proper boundary conditions are imposed. Inflow, outflow, wall and symmetry boundary conditions should be specified to solve the proposed problem. These boundary conditions may vary in different flow patterns of inviscid and viscous flow.

Inflow boundary condition is set to be as the free stream conditions and outflow condition is set to be free without any limitations.

Wall boundary conditions are different considering inviscid or viscous flows. For inviscid flows, the values of normal velocity at ghost cells are taken as opposite of the normal velocity in the interior cell. The tangential velocity at the wall boundary condition is not zero for inviscid flows. However, for viscous flows, no- slip boundary condition is applied at the walls. In no-slip boundary condition, the values of velocities in all directions are set to be zero. This is done by imposing the velocity at the ghost cells in the opposite direction of the velocity values of the interior cell.

SOLUTION METHOD

The current study is done on the density based solver with transient approach and the results were taken for 5 milliseconds. The k- ω SST turbulence model were used in this study. Because of the higher supersonic speed the flow could not be captured for longer time so the time step is reduced 10^-5 . And the solution runs for 500 time steps to visualize the flow for 5 milliseconds. Since the time step size is very much small the explicit formulations were used to compute the results. And the pulsed mode is implemented at fuel inlet using the expression given.

RESULTS AND DISCUSSIONS

Numerical Study on the Mixing Characteristics of a Combustor Using Pulsed FIS

The system of equations described in the previous chapters are solved and the results are shown here. In order to identify the effect of different parameters on the flow analysis, different parametric studies are done. Since solving three dimensional Navier-Stokes equations with chemical reactions require great computational effort, the parametric studies are done on simplified Euler equations. The validity of the results obtained by the transport equations are investigated and approved. Therefore, evaluation of results for different geometry with and without using sinusoidal pulse are shown in the following. Finally, results obtained using Navier-Stokes equations are given.

The different types of geometry of scramjet combustion chamber used in this study is taken from the literature review. The mentioned scramjet is taken as reference since it is one of the most recent experimental scramjets used. Also, data from the experimental studies of this module are available in the literature which is crucial for code validation purposes. The different geometries were shown in Figure 5.1.

The location of the fuel injectors is highly important in increasing the mixing efficiency of the fuel-air. Fuel injectors located far upstream will cause flame holding problems because of high velocities at the combustor entrance. Mixing efficiency reduces at high speeds because of the low residence time of the flows at these speeds. Therefore, flames will be unstable and continuous combustion will not occur. On the other hand, if the fuel injectors were placed on the edge of the cavity front wall, the fuel will not be able to penetrate into the core flow by taking advantage of the reduced velocity and recirculation region in the cavity. The shear layer will interrupt the fuel- air mixing and so efficiency of the mixing will decrease in a great amount. Therefore, fuel injection should be done from a small distance upstream of the cavity front wall. In this way, higher mixing efficiency of fuel-air will be achieved in the cavity region. This is one of the important characteristics of the cavities which make them more efficient compared to other flame holding methods in scramjet combustors.

Before proceeding with the solutions, the validity of the solutions should be confirmed. For this purpose, experimental and numerical data from different studies are employed. In order to verify the results of the present study, the experimental data are taken from the literature are used for the validation and grid refinement study. The combustor inlet conditions applied in the solution of the present study are taken from the experimental data and can be summarized as in Table 5.2.

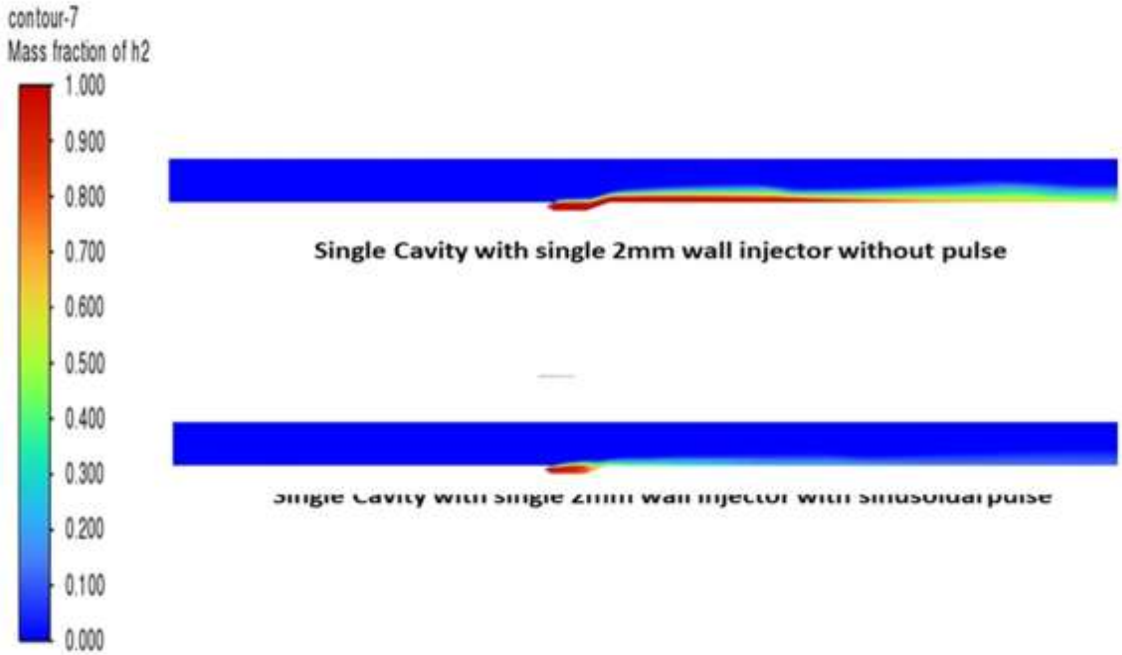
The pulsed mode of injection is achieved by giving an expression at the fuel inlet. The expression is obtained from the simple harmonic wave motion equation by altering it with a frequency of 15KHz. The positive sinusoidal expression is obtained by squaring the value of sine . The time dependent sinusoidal pulse is achieved by using an mathematical expression given at the fuel inlet. And the sinusoidal pulsating plot for 5milliseconds is also given below.

The above sinusoidal pulse is implemented at the analysis of the scramjet engine by giving it in the fuel inlet boundary conditions. The contours were taken for the properties such as static temperature, static pressure, density, turbulent kinetic energy, velocity and mass fraction of hydrogen. The main parameter which is used to compare the results were the turbulent kinetic energy at the downstream of the fuel injectors and the percentage of unburnt hydrogen at the outlet of the scramjet engine.

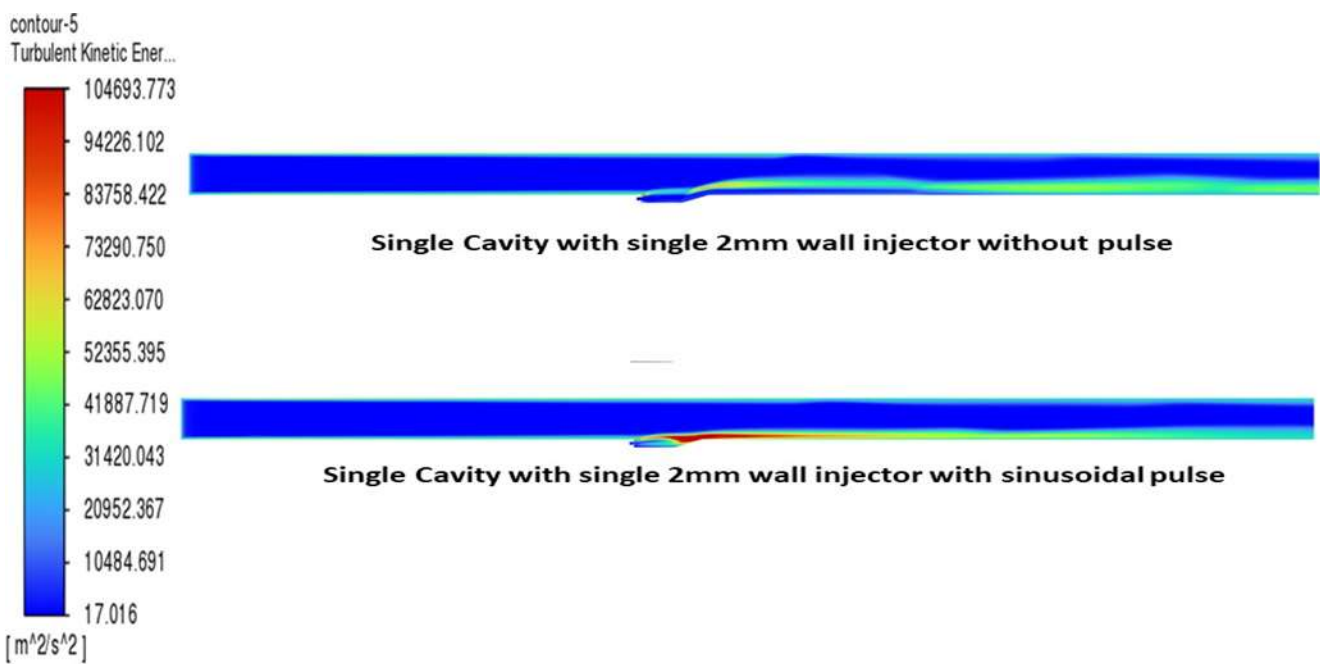
The percentage of unburnt hydrogen at the outlet will greatly help to determine the mixing characteristic of the combustor and so it is the main parameter to compare the mixing characteristic of normal and pulsed mode fuel injection.

CONTOUR COMPARISON FOR VARIOUS GEOMETRIES

ISBN: 978-93-5980-498-9, www.aetsjournal.com ― Handbook of research in Modeling of Transportation and Aviation Processes



TEMPERATURE CONTOURS



DENSITY CONTOURS

VARIOUS GEOMETRY	OUTLET TEMPERATURE	UNBURNT HYDROGEN AT THE OUTLET IN PERCENTAGE
Single Cavity with 2mm injector without pulse	1713.144	22.17
Single Cavity with 2mm injector with sinusoidal pulse	**1667.614**	**10.78**
Single Cavity with double 1mm injector without pulse	1402.294	26.33
Single Cavity with double 1mm injector with sinusoidal pulse	**1603.334**	**9.75**
Double Adjacent Cavity with double 1mm injector without pulse	844.829	35.39
Double Adjacent Cavity with double 1mm injector with sinusoidal pulse	**1410.964**	**11.73**
Double Opposite Cavity with single 2mm injector without pulse	1490.643	21.55
Double Opposite Cavity with single 2mm injector with sinusoidal pulse	**1583.911**	**15.53**
Single Cavity with 2mm wall injector without pulse	1595.185	12.09
Single Cavity with 2mm wall injector with sinusoidal pulse	**1750.318**	**3.16**

CONCLUSIONS AND FUTURE WORK

In this thesis, a CFD analysis of five different geometries of scramjet engine is done using normal and pulsed mode of fuel injection and the results and contours were taken and studied. The findings demonstrate that extremely high amounts of turbulent kinetic energy (TKE) are seen particularly in the downstream side of the fuel injectors and low percentage of unburnt hydrogen is observed at the outlet while using pulsed mode injection. Increased TKE and less percentage of unburnt hydrogen at the outlet is proven to improve mixing efficiency for all types of pulsed fuel injection.

The table below shows the outlet temperature and the percentage of unburnt hydrogen at the outlet for the five different types of geometries and it is evident that the percentage of unburnt hydrogen is greatly reduced while using pulsed mode of fuel injection and the flame temperature at the combustion chamber is higher and the temperature at the outlet is also tabulated below.

From the table it is evident that the pulsed mode of fuel injection is very efficient in all types of geometry and the percentage of unburnt hydrogen is greatly reduced in all types of geometry while using pulsed mode of fuel injection.

The current work is done with sinusoidal pulse in simple geometries to prove the improved mixing characteristics of fuel and air and our future work is to implement the various other modes of pulsed injection like cosine pulse, rectangular pulse, square pulse etc,, with complex geometries. The flow calculation in our current study is 5 milliseconds and in our future work it will be increased more and the analysis will be done accordingly.

REFERENCES

[1]. Dora, E. Musielak (2015) 'Numerical Simulation of Cavity Fuel Injection and Combustion for Mach 10-12 Scramjet', HSABP TC Report

[2]. Menter, F. R. (1994) 'Two-equation eddy-viscosity turbulence models for engineering application', AIAA J. 32(8) 1598–1605

[3]. Pecnik, R., Terrapon, V. E., Ham F, Iaccarino, G. and Pitsch, H. (2012) 'Reynolds-averaged Navier–Stokes simulations of the HyShot II Scramjet', AIAA J. 50(8) 1717–1732.

[4]. Roncioni, P., Natale, P., Marini, M., Langener, T. and Steelant, J. (2015) 'Numerical simulations and performance assessment of a scramjet powered cruise vehicle at Mach 8', Aerospace Science Technology 42(1) 218–228.

[5]. Schramm, J., Karl, S., Hannemann, K. and Steelant, J. (2008) 'Ground testing of the HyShot II scramjet configuration in HEG', AIAA Paper 2008- 2547.

[6]. Segal, C. (2009) 'The Scramjet Engine: Process and Characteristics' Cambridge University Press.

[7]. Shi, H., Wang, G., Luo, X., Yang, J. and Lu, X. (2016) ', Large-eddy simulation of a pulsed jet into a supersonic crossflow', Computational Fluids 140 320–333.

[8]. Song Chen and Dan Zhao, (2019) 'RANS Investigation of the effect Of pulsed fuel injection on scramjet HyShot II engine', Aerospace Science Technology 84 182-192

[9]. Vanyai, T., Bricalli, M., Brieschenk, S. and Boyce, R. R. (2018) ‘Scramjet performance for ideal combustion process’, Aerospace Science Technology 75(1) 215–226.

[10]. Williams, N. and Moeller, T. M. (2016) ‘Numerical investigations of high frequency pulsed fuel injection into supersonic crossflow AIAA Paper 70(6) 3483.

ISBN: 978-93-5980-498-9, www.aetsjournal.com — Handbook of research in Modeling of Transportation and Aviation Processes