International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 1, No. 2, July 2012

Research Paper

CFD ANALYSIS OF DIFFERENT GEOMETRIC COMBUSTOR AT SUBSONIC FLOW

Chandra Shekhar[1] and M Shiva Shankar[2*]

Corresponding Author: **M Shiva Shankar, ✉ shishank2000@yahoo.com*

In the design of a jet engine the problem of fuel injection as well as flame holding is very important role. In combustion due to short residence times, it is essential to specifically adopt a device or strategy to enhance the mixing between the fuel and oxidizer to achieve combustors of reasonable size and weight. At the same time the mixing device/strategy should not lead to large total pressure losses for the flow in the combustor, because it would leads to thrust losses. Compared to other mixing devices cavity has low pressure losses and high combustion efficiency. It is essential to understand the behavior of cavity. Computational Fluid Dynamics (CFD) investigations are carried out by varying geometric parameters to study the cavity influence on the combustion at subsonic speed, in a simple combustor with cavity and without cavity. The different fuels like hydrogen, kerosene and high octane are injected at Mach number 0.8 to Mach number 0.3 air stream. The combustion efficiency determined different above mentioned fuels that hydrogen is greater than two fuels. Also in hydrogen fuel there is 20.05% increase in combustion efficiency of geometric combustor with cavity compared to without cavity. The present study indicates that combustion efficiency is high in the presence of cavity at subsonic flows. This work was carried by using computational software package fluent and gambit.

Keywords: CFD, Cavity, Combustor, Combustion efficiency

INTRODUCTION

Supersonic flow over cavities has been extensively studied for many years because of their relevance to aerodynamic configurations (Anderson, 1995). A cavity, exposed to a flow, experiences self sustained oscillations, which can induce fluctuating pressures, densities, and velocities in and around the cavity, resulting in drag penalties. This problem motivated many experimental and computational studies, which have been directed toward improving the understanding

[1] M.Tech Student, Siddaganga Institute of Technology, Tumkur.
[2] Associate Professor, Siddaganga Institute of Technology, Tumkur.

of the physics of cavity flows and the means to control their nature. In the design of supersonic combustion ramjet (scramjet) engine a problem of fuel injection as well as cavities. It has been essential to understand the cavity behavior to incorporate cavities in scramjet engines.

In this paper we studies have been performed on cavity flows, cavities are typically divided into two categories, open and closed. Open cavities are those for which the separated shear layer re-attaches at the end-wall, while for closed cavities the shear layer re-attaches on the floor of the duct. Open cavities are typically preferred for combustor designs because the drag penalty is lower. The mixing and the flame- holding properties of a scramjet can be significantly improved through the addition of cavity. The length of the cavity determines the mass entrainment characteristics, whereas the depth of the cavity determines the residence time. Therefore, cavities can be tuned to achieve either mixing enhancement or flame holding by changing the length and depth of the cavity. Cavities produce a recirculation of radicals, with sufficient residence time for ignition to occur without the need for long combustion chamber length.

COMBUSTION

Combustion or burning is a complex sequence of exothermic chemical reactions between a fuel and an oxidant accompanied by the production of heat or both heat and light in the form of either a glow or flames. Combustion is the chemical reaction which happens when substances burn.

Combustion is the conversion of a substance called a fuel into chemical compounds known as products of combustion by combination with an oxidizer. The combustion process is an exothermic chemical reaction, i.e., a reaction that releases energy as it occurs.

$$\text{Fuel} + \text{Oxidizer} \rightarrow \text{Products of Combustion} + \text{Energy} \quad ...(1)$$

A simpler example can be seen in the combustion of hydrogen and oxygen, which is a commonly used reaction in rocket engines:

$$2H_2 + O_2 \rightarrow 2H_2O(g) + \text{Heat} \quad ...(2)$$

This is a measure for the anti-knocking properties of octane.

The octane rating is not related to the amount of octane contained in the gasoline.

Octane reacts with oxygen and forms CO_2 and H_2O as shown in Equation (3).

$$C_8H_{18} + 12.5O_2 = 8CO_2 + 9H_2O \quad ...(3)$$

Kerosene is a mixture of hydrocarbons of 12 to 15 or more carbons.

Octane became well-known in American popular culture in the mid- and late-sixties.

$$C_{15}H_{32} + 23O_2 \rightarrow 15CO_2 + 16H_2O \quad ...(4)$$

Kerosene is a thin, clear liquid formed from hydrocarbons, with density of 0.78-0.81 g/cm^3. Kerosene is obtained from the fractional distillation of petroleum between150 °C and 275 °C, resulting in a mixture of carbon chains that typically contain between 6 and 16 carbon atoms per molecule. The flash point of kerosene is between 37 and 65 °C (100-150 °F).

Combustor

A combustor is a component or area of a gas turbine, ramjet or pulsejet engine where combustion takes place. It is also known as a burner or flame can depend on the design.

Combustors are designed to contain and control the burning fuel-air mixture. Chamber itself which contains the flame and the fuel injection system. The combustor normally consists of three components: an outer casing that acts as a high pressure container, the combustion.

In a gas turbine engine, the main combustor or combustion chamber is fed high pressure air by the compression system and feeds the hot exhaust into the turbine components of the gas generator. Over cavities of different sizes on subsonic flow field. Compressibility effects, So cavity with self sustained oscillation can be used to enhance mixing in free shear layer.

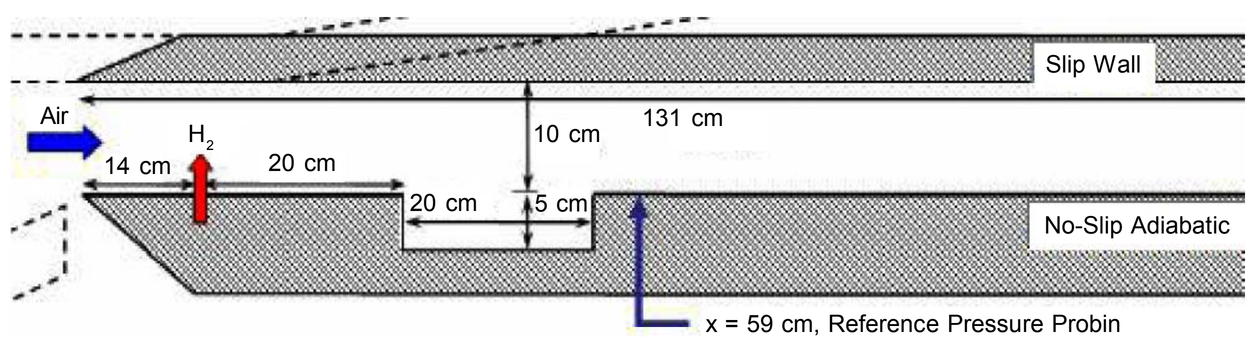
Flame holding is known to play very important role. In supersonic combustion due to short residence times, it is essential to specifically adopt a device to enhance the mixing between the fuel and oxidizer to achieve combustor of reasonable size and weight. At the same time, the mixing device should not lead to large total pressure losses for the flow in the combustor, because this would lead to thrust losses. There are several techniques for improving the mixing, based on the generation of stream wise vorticity such as ramps, tabs, lobe mixers, etc.

GEOMETRICAL MODELING AND GRID GENERATION

In the present study a two-dimensional model of a simple channel type combustor of 10 cm height and 131 cm length is composed of transverse fuel injection vertically through a slot of 0.1 cm width to the combustor through a choked nozzle. With and without cavity is used to analyze the combustion flow properties with different position of fuel injections has been modeled using GAMBIT pre-processor.

The following model has taken from the paper Numerical Simulation of Supersonic Combustion for Hypersonic Propulsion 5th Asia-Pacific Conference on Combustion (Eddington, 1921). The University of Adelaide, Adelaide, Australia July 18-20, 2005. So this work is about to analyze the combustion, using the same model and changing the cavity dimensions as (L/D = 4 and L/D = 3) and replacing the fuel injecting nozzle near to the cavity at subsonic speed with different fuels and comparing which fuel is efficient for combustion. Using this as the basic model five model types has been analyzed.

Figure 1: Simple Combustor Configuration Combustor Model Types



Combustor Model Types

Model 1: Without cavity.

Model 2: With cavity L/D ratio is 4.

Model 3: With cavity Injector placed 10cm near to the cavity L/D ratio 4.

Model 4: With cavity L/D ratio is 3.

Model 5: With cavity Injector placed 10cm near to the cavity L/D ratio 3.

NUMERICAL PROCEDURE

Presently there are several commercially available CFD software packages namely FLUENT, FLOW 3D, ANSWER, PHOENICS, STAR-CD, etc., for solving complex fluid flow problems. However, the basic steps involved in solving the flow problem are the same regardless of the package and can be grouped under three stages.

Stage 1: Pre-processing,

Which involves

- Geometric modeling
- Grid-generation
- Flow specification

Stage 2: Solution stage involving the algebraic equations.

Stage 3: Post processing stage.

This involves analyzing the results from vector plots, contour plots, surface plots and other data visualization tools.

This process of numerical discretization each term within a partial differential equation is translated in to a numerical analogue that the computer can be programmed to calculate. The distinct streams of numerical discretization techniques are finite difference, finite element and finite volume method. In the present study the finite volume method is used.

Solution Procedure

There are several numerical algorithms for solving the discrete equations among which the SIMPLE algorithm for correct linkage between pressure and velocity and the TDMA line-by-line solver of the algebraic equations are the most popular. The success or otherwise of such numerical solution algorithms is determined from the mathematical concepts of convergence and stability.

COMPUTATIONAL DETAILS OF THE PRESENT STUDY

The flow inside the domain has been simulated by solving equations for conservation of mass, momentum, and energy. Finite volume method has been used. The pressure-velocity coupling has been achieved by SIMPLE algorithm. The convective terms are discretized by First order upwind schemes for all equations while the diffusive terms are discretized by central differencing schemes. Turbulence in the flow has been modeled using the standard k-e turbulence model. This work deals with the computational details viz. governing equations that are solved using the solution procedure, geometrical modeling and the details of geometry, grid generation for the configurations under study, various boundary conditions that are enforced are discussed and presented in this chapter (Table 1). In order to render the problem tractable for analysis with limited computational sources and time, the following assumptions are made.

Assumptions

The following important assumptions pertaining to the flow in the cavity have been made in the present study:

Table 1: Boundary Conditions Zones

Name	ID	Type
Fluid	2	Fluids
Fuel Inflow	3	Velocity-Inlets
Air Inflow	4	Velocity-Inlet
Proutlet	5	Pressure-Outlet
Wall	6	Walls
Default-Interior	8	Interior

- Flow is steady, incompressible and turbulent.
- Isothermal flow throughout the domain.
- Reacting flow inside the cavity.
- Radiation effects are neglected.

Governing Equations

Continuity Equation

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \vec{v}) = S_m$$

Where ρ is density, v is the mean velocity component in i direction and the source term S_m is the mass added to the continuous phase due to vaporization of liquid droplets. For the non-reacting flow S_m takes a value of zero.

Momentum Equation

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla(\rho \vec{v}\vec{v}) = -\nabla p + \rho g + \vec{F}$$

Where ρ is static pressure, g and F are gravitational body forces and external body forces (e.g., forces that arise from interaction with the dispersed phase), respectively. F also contains other model-dependent source terms such as porous-media and user defined sources.

Energy Equation

$$\frac{\partial}{\partial t}(\rho E) + \nabla(\vec{v}(\rho E + P)) = -\nabla \sum_j h_j j_j + S_h$$

S_h includes the heat of chemical reaction, where sensible enthalpy h is defined for ideal gases as where m_j, is the mass fraction of the species j

Mesh Refinement

The refinement of mesh has done and the respective face meshed model is in Figure 2.

Mesh is exposed as mesh file, which is to be read in the solver. The grid is as shown in the Figure 3.

Figure 2: Meshed Model with Cavity L/D Ratio is 4

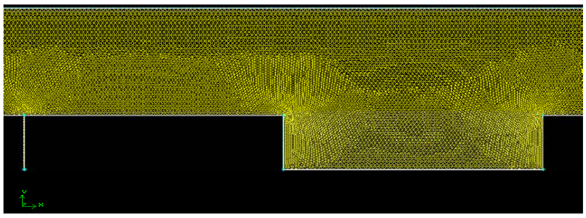
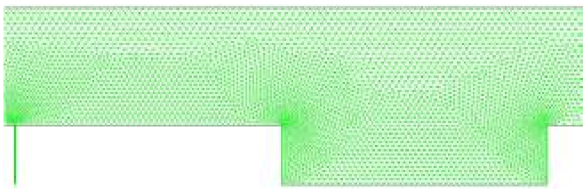


Figure 3: Grid with Cavity Model with Cavity L/D Ratio is 4



The analysis carried out is presented in the following steps. Flow behavior is studies inside a combustor with and without cavity.

- Analysis is carried out in the combustor.
- Comparison of fuels and geometric variation studies is done.

Operating Condition (Table 2)

Operating Pressure: –101325

Air Inflow Velocity: 102 m/s (Mach 0.3)

Table 2: Operating Condition

Condition (Inlet)	Value
Air Temperature	600 K
Fuel Inflow Velocity	272m/s (Mach 0.8)
Fuel Temperature	300 K
Outlet Temperature	600 K
Wall Temperature	400 K

RESULTS AND DISCUSSION

The combustion inside a combustor using different fuels, with and without cavity with transverse fuel injection vertically through a slot of 0.1 cm width has been successfully simulation using the FLUENT commercial code.

The Figures 4 and 5 shows the temperature distribution contours in the combustor.

Inside the cavity is low compare to the total velocity in the combustor, due to the recirculation the residing time of fuel and air is high in the combustor with cavity (Figure 6).

This study investigated the cavity-injector combustor behavior using Hydrogen, High-octane and Kerosene fuels and the effects of combustor over cavities of different sizes on subsonic flow field, fuels are injected at Mach number 0.8 to Mach number 0.3 airstream (Table 3). The main objective of this work is to examine the subsonic flow over the cavity and

Figure 4: Temperature Contour of Model Without Cavity

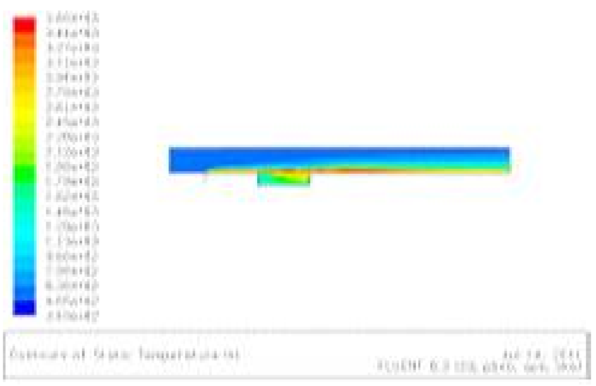


Figure 5: Temperature Contour of Model with Cavity

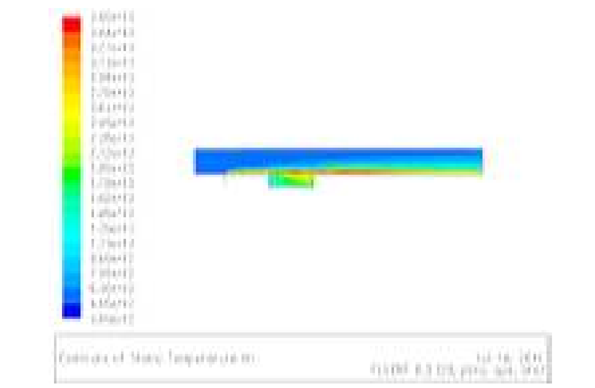


Figure 6: Velocity Contour Recirculation (Near L/D4)

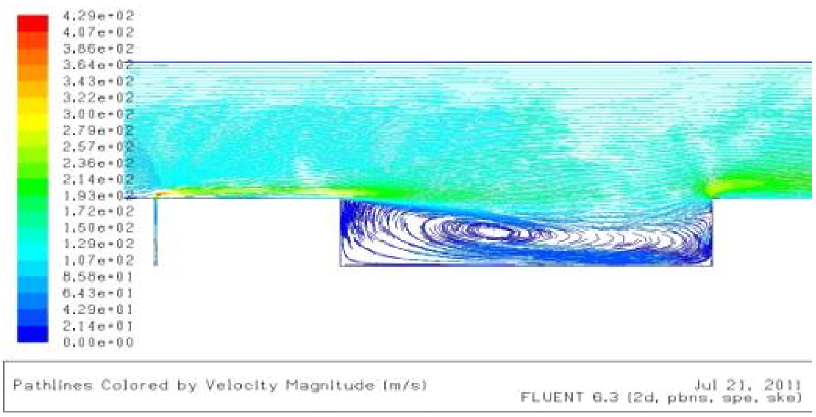


Table 3: Species

Inlet Required	Value
Oxygen	0.23
Hydrogen	1.00
Octane	1.00
Kerosene	1.00

examining which fuel is efficient for combustion, the cavity has influenced subsonic combustion and as per our results hydrogen fuel has higher efficiency with High Octane and Kerosene fuel. For a good combustor the exit temperature must be low, it has been found when combustion occurs with Hydrogen fuel. The model without cavity has the low pressure loss than the model with cavity, but the model with cavity has higher combustion efficiency than the model without cavity.

CONCLUSION

As per the result from the present work combustion efficiency is high at supersonic speeds, with the presence of cavity. This work is carried out for subsonic flows to examine if the cavity influences subsonic combustion, as per the results from the present study has shown that the combustion efficiency is high in the presence of cavity at subsonic flows.

Overall flow features like recirculation zone inside the cavity, combustion in a combustor with cavity and without cavity, mixing of air and fuel, velocity of flow and mass fraction contours are Comparing with models without and with cavity with different fuels the combustions efficiency increases in a model with cavity, as below:

- In Hydrogen Fuel there is 20.05% increase in combustion efficiency.
- In High Octane Fuel there is 19.90% increase in combustion efficiency.
- In Kerosene Fuel there is 14.55% increase in combustion efficiency.

The best combustion efficiency is observed in the model when the Injector placed near cavity L/D4 with hydrogen fuel.

REFERENCES

1. 12th AIAA International Space Planes and Hypersonic Systems and Technologies, December 19, 2003, Norfolk, Virginia.
2. Adela Ben-Yakar and Ronald K (2001), *Journal of Propulsion and Power*, Vol. 17, No. 4, July-August, Hanson Stanford University, California.
3. AIAA/CIRA 13th International Space Planes and Hypersonic Systems and Technologies, Australian National University, Canberra.
4. Anderson J D (Jr.) (1995), "Computational Fluid Dynamics", McGraw-Hill, Inc.
5. Eddington S (1921), The Role of Plasma Igniters and Fuel Injectors in Hypersonic Technology (British Astrophysicist, 1882-1933).
6. *International Journal of Heat and Mass Transfer*, Vol. 47, No. 2, January 2004.
7. Kim K M, Baek S W and Han C Y (2004), "Numerical Study on Supersonic Combustion with Cavity-Based Fuel Injection", *International Journal of Heat and Mass.*
8. Michael A Kegerise, Eric F Spina Sanjay Garg and Louis N Cattafesta III (2004), "Mode-Switching and Nonlinear Effects in

Compressible Flow Over a Cavity”, *Physics of Fluids*.

9. 5th Asia-Pacific Conference on Combustion, July 18-20, 2005, “Numerical Simulation of Supersonic Combustion for Hypersonic Propulsion”, The University of Adelaide, Adelaide City, Australia.

APPENDIX

Nomenclature			
Symbol	**Description**	**Symbol**	**Description**
η	Efficiency	h	Enthalpy (J/KG)
m_j	Mass Fraction	m	Gas Viscosity (kg/m-s)
k	Turbulent Kinetic Energy	P	Density (kg/m^3)
p	Static Pressure	S_h	Heat of Chemical Reaction