Computational Investigation of Combustion in a Cavity

Vijaya Kumar R[1], Syed Mohsin[2]
[1,2]Assistant Professor, Department of Aeronautical Engineering
MVJ College of Engineering, Bangalore, Near ITPB, Channasandra-560067

ABSTRACT: CFD investigations are carried out by varying geometric parameters to study the cavity influences on the combustion at subsonic speed, in a simple combustor with and without cavity. The kerosene, hydrogen and high octane fuels are injected at Mach number of 0.8 to Mach number 0.3 airstream. The Combustion efficiency for the different fuels are found and compared that to be hydrogen is greater than high octane and kerosene fuels. The pressure losses in the combustor with hydrogen fuel are less compared to that of kerosene and high octane fuels. In hydrogen fuel there is 20.05% increase in combustion efficiency in a combustor with cavity compared to without cavity, in high octane fuel there is 19.90% increase in combustion efficiency, in kerosene fuel there is 14.55% increase in combustion efficiency. As per the results from the present study that the combustion efficiency is high in the presence of cavity at subsonic flows. Hydrogen Fuel is efficient for combustion compare to that of kerosene and high octane fuels. This work was carried by using computational software package Fluent and Gambit.

Key words: Cavity, Combustion efficiency, hydrogen fuel, Computational Fluent and Gambit.

1 Introduction

In the design of turbojet engine a problem of fuel injection as well as flame holding is known to play a very important role. In combustion due to the 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 [1]. At the same time, the mixing device/strategy should not lead to large total pressure losses for the flow in the combustor, because this would lead to thrust losses [2]. There are several techniques for improving the mixing, some of them are based on the generation of streamwise vorticity, such as ramps, tabs, lobe mixers, chevrons, etc. Some others are based on the self-excitation such as with cavities [1]. It is then essential to understand the cavity behavior to incorporate cavities in engines. The cavity is of interest because recirculation flow in cavity would provide a stable flame holding while enhancing the rate of mixing or combustion. The total pressure losses incurred by these devices are less compared with other mixing-enhancement devices.

While many studies have been performed on cavity flows, the current understanding of cavity-flow physics remains Incomplete. 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 [4]. Open cavities are typically preferred for combustor designs because the drag penalty is lower. The mixing and flame-holding properties of an engine can be significantly improved through the addition of a cavity. Cavities produce a recirculation of radicals, with sufficient residence time for ignition to occur without the need for long combustion chamber lengths. In this work we study the cavity-injector combustion behaviour using hydrogen, high-octane and kerosene fuels and the effects of combustion over cavities of different sizes on subsonic flow field. This work describes the study of the effect of combustor with and without cavity in a viewpoint of total pressure loss and combustion efficiency.

2. 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.
The hydrogen, kerosene and high octane fuel reacts with oxygen as given in below reaction equations.

$2H_2 + O_2 \rightarrow 2H_2O$ **(g) + heat** (hydrogen reaction)

$C_{15}H_{32} + 23O_2 \dashrightarrow 15CO_2 + 16H_2O$ (kerosene reaction)

$C_8H_{18} + 12.5O_2 = 8CO_2 + 9H_2O$ (high octane reaction)

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. Pre-processing was done by Gambit, Solving & Post processing was done by fluent software package. The numerical discretization technique finite volume method is used.

4 GEOMETRICAL MODELLING

The basic combustion model has taken from the paper **Numerical Simulation of Supersonic Combustion for Hypersonic Propulsion** 5th Asia-Pacific Conference on Combustion, The University of Adelaide, Australia 18-20 July 2005. The transverse fuel injection vertically through a slot of 0.1 cm width is as shown in figure. So this work was analyzed the combustion, using the same model and changing the cavity dimensions as (L/D=4 & L/D=3) and replacing the fuel injecting nozzle near to the cavity 10cm at subsonic speed with different fuels and comparing which fuel is efficient for combustion. Using this as the basic model four other model types has been analyzed.

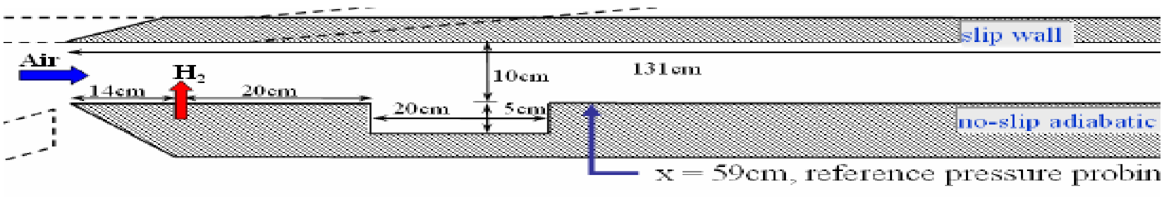

Fig 1: - Simple Combustor configuration

Model types

- Model without cavity
- Model with cavity L/D ratio is 4
- Model with cavity Injector placed 10cm near to the cavity L/D ratio 4
- Model with cavity L/D ratio is 3
- Model with cavity Injector placed 10cm near to the cavity L/D ratio 3

5. 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*-ε turbulence model.

5.1 Assumptions

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

(i) Flow is steady, incompressible and turbulent
(ii) Isothermal flow throughout the domain
(iii) Reacting flow inside the cavity
(iv) Radiation effects are neglected

5.2 Governing equations

Continuity Equation:

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

Momentum Equation:

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

Energy Equation:

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

5.3 Meshing

The refinement of mesh has done and the respective face meshed model is as shown in below figure.

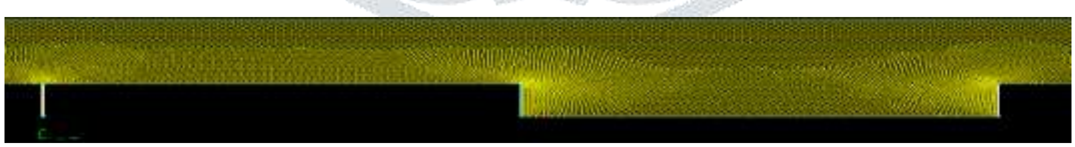

Fig 2:- Meshed Model with cavity L/D ratio is 4

Mesh is exported as .msh file, which is to be read in the solver. The grid is as shown in the below figure.

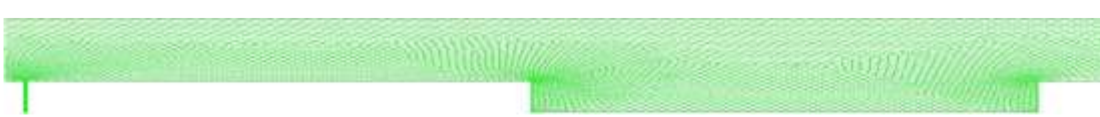

Fig 3:- Grid with cavity L/D ratio is 4

5.4 The analysis carried out is presented in the following steps

- Grid independency is studies for selecting proper interval mesh size.
- 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.

Boundary Conditions

Zones

NAME	ID	Type
Fluid	1	fluids
Fuel inflow	2	velocity-inlet
Air inflow	3	velocity-inlet
Proutlet	4	pressure-outlet
Wall	5	walls

Inlet:

Condition	Value
Air inflow Velocity	102 m/s (Mach 0.3)
Air temperature	600 K
Fuel inflow Velocity	272m/s (Mach 0.8)
Fuel temperature	300 K
Outlet temperature	600 K
Wall temperature	400 K

Convergence criterion:

Variable	Check Monitor Convergence	Convergence Criterion
Continuity	yes	0.0001
X-Velocity	yes	0.0001
Y-Velocity	yes	0.0001
Energy	yes	1e-06
K	yes	0.0001
Epsilon	yes	0.0001
CH4	yes	0.0001
O2	yes	0.0001
CO2	yes	0.0001
H2O	yes	0.0001

6. 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 simulated using the FLUENT commercial code.

The below contours shows the pressure distribution and temperature (combustion) in the combustor.

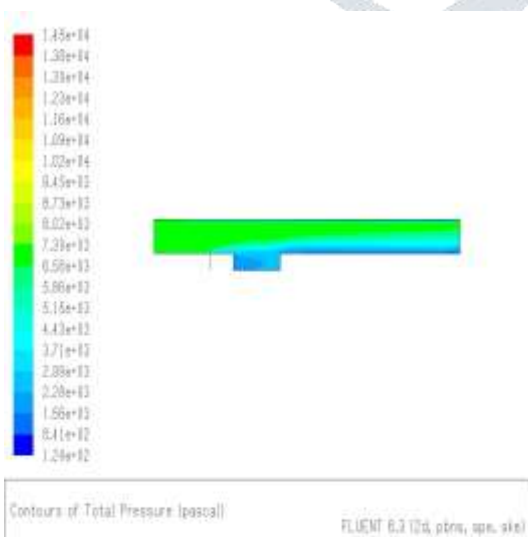

Fig 4: - Pressure contour

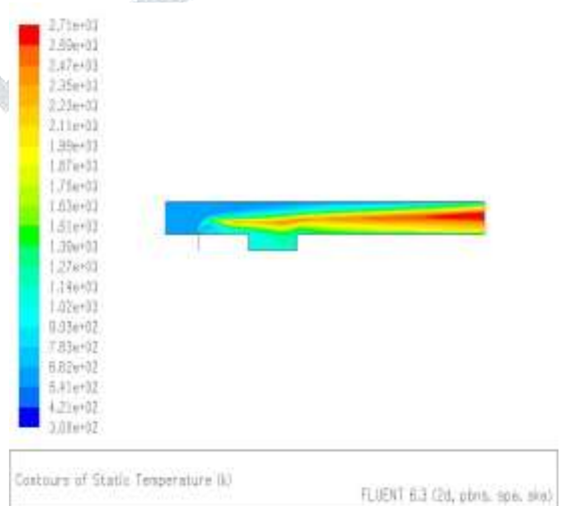

Fig 5: - Temperature contour

The pressure inside the cavity is low as shown in the figure 4. The combustion of fuel and air takes place as shown in figure 5.

The recirculation inside the cavity is as shown in below figure 6, the velocity 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.

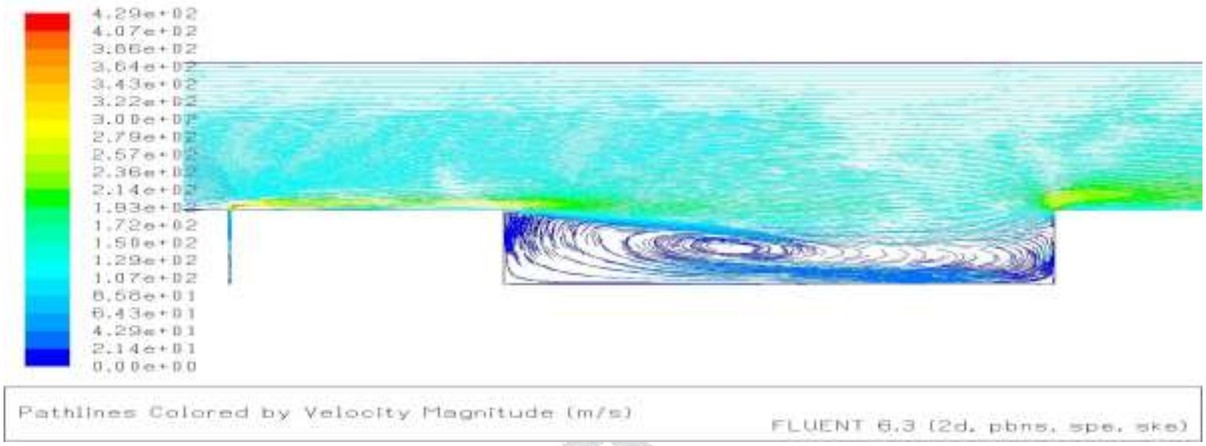

Fig 6:- Recirculation inside the cavity

This study investigated the cavity-injector combustion behaviour using hydrogen, high-octane and kerosene fuels and the effects of combustion over cavities of different sizes on subsonic flow field, fuels are injected at Mach number 0.8 to Mach number 0.3 airstream. The main objective of this work is to examine the subsonic flow over the cavity & examining which fuel is efficient for combustion, the cavity has influenced subsonic combustion and as per our results hydrogen fuel has higher efficiency comparing with high Octane & 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.
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.**

7. REFERENCES

1) Adela Ben-Yakar and Ronald K.Hanson. “JOURNAL OF PROPULSION AND POWER”, Stanford University, California 94305 Vol. 17, No. 4, July–August 2001
2) Kyung Moo Kim, Seung Wook Baek, Cho Young Han.“Numerical study on supersonic combustion with cavity-based fuel injection”, International Journal of Heat and Mass Transfer 47 (2004) 271–286.
3) S.Eddington.“The Role of Plasma Igniters and Fuel Injectors in Hypersonic Technology”. (British Astrophysicist, 1882-1933).
4) TarunMathur, MarkGruber, KevinJackson. “Supersonic Combustion Experiments with a Cavity-Based Fuel Injector”, Journal article, NOV 2011.
5) Wai-Sun Don, David Gottlieb, Jae-Hun Jung, “A multidomain spectral method for supersonic reactive flows”, Journal of Computational Physics 192 (2003) 325–354.
6) Tutorial 11, Modelling Species Transport and gaseous combustion. Fluent Inc. November 27, 2014.
7) O. L. Gülder, AER 1304 Fundamentals of Combustion, Fall 2008 – UTIAS University of Toronto.
8) Fluent 6.3 User’s Guide, Copyright @ 2006 by Fluent Inc