COMPUTATIONAL ANALYSIS OF CAVITY COMBUSTOR WITH SUBSONIC SPEED

A J SRIGANAPATHY[1], R. LOGANATHAN [2], S. MONISHRAJ [3], R. RANJITHKUMAR [4],

R. SATHISHKUMAR[5]

[1]Assistant Professor, Department of Aeronautical Engineering, Mahendra Institute of Engineering and Technology, Namakkal, India.
[2,3,4,5] UG Scholar, Department of Aeronautical Engineering, Mahendra Institute of Engineering and Technology, Namakkal, India.
sri.aero87@gmail.com[1], loganathanr6115@gmail.com[2],monishraj586@gmail.com[3], ranjith242000@gmail.com[4], sathishkumaraero1234@gamil.com[5]

***

Abstract - *In the design of a supersonic combustion ramjet (scramjet) engine a problem of fuel injection as well as flame holding is known to play a very important role. Because of the short residence periods in supersonic combustion, a system or technique to improve the mixing between the fuel and oxidizer is needed to achieve combustors of acceptable size and weight. There are many techniques for improving mixing; some of them, such as ramps, tabs, lobe mixers, chevrons, and others, are based on the generation of streamwise vorticity. Others, such as cavities, operate on the theory of self-excitation. Cavity has been proved to be a modern reliable flame keeper for supersonic combustion, and it has been used in a variety of scramjet models. Cavities are commonly categorized into two types: open and closed. Because of the lower drag penalty, open cavities are commonly favored for combustor designs. The use of a cavity is a trade-off between flame retention for effective combustion and the drag penalty that results. The addition of a cavity to a scramjet will greatly boost its mixing and flame-holding capabilities. The mass entrainment characteristics are determined by the cavity length, while the residence time is determined by the cavity depth. As a result, adjusting the length and depth of the cavity may be used to achieve either mixing enhancement or flame retention. The proposed work's aim is to investigate the effect of cavity on total pressure loss and combustion efficiency. The effects of combustion over cavities of various sizes on the subsonic flow field are investigated in this study using methane fuel and cavity-injector combustion action. The primary goal of this research is to investigate subsonic flow over the cavity when fuel is pumped into the combustion chamber.*

Key Words: Cavity, Combustor, Flame holder, Fuel injector, Scramjet.

1. Introduction

Computational Fluid Dynamics (CFD) is the method of replacing the differential equation that governs fluid flow with a series of algebraic equations (a process known as Discritization), which can then be solved using a digital computer to obtain an approximate solution. CFD has been able to solve a wide range of flow problems, including those that are compressible or incompressible, laminar or turbulent, chemically reacting or non-reacting, thanks to the advent of high-speed and large-memory computers. Finite Difference Method (FDM), Finite Volume Method (FVM), Finite Element Method (FEM), and Boundary Element Method are some of the well-known Discritization methods used in CFD (BEM).

1.1 Supersonic air vehicle

Since we know that supersonic air vehicles will play an important role in the future where combustion would occur at supersonic speeds, Supersonic Combustion RAMJET (SCRAMJET) is needed for this type of air vehicle. It's extremely difficult to get a good mixture of fuel and air, as well as a flame, in a SCRAMJET. Because of the short duration of residence time and the high rate of flow, holding properties exist (around Mach 2). Different types of cavities (open cavity / closed cavity) have been used by the researchers to develop flame keeping and mixing properties. Open cavities have a segregated shear layer that reattaches at the end-wall. The use of a cavity is a trade-off between flame retention for effective combustion and the drag penalty that results. This work examines the cavity-injector combustion behavior using high octane and kerosene fuel, as well as the effects of combustion over cavities of various sizes on the subsonic flow area. This research analyses the total pressure loss and combustion efficiency of various fuels to determine which fuel is the most effective and which model is the most efficient for subsonic combustion. The aim of this research is to compare the effects of combustor with and without cavity on total pressure loss and combustion efficiency when comparing different fuels and determining which fuel is the most effective at subsonic speeds.

2. Literature Survey:

The aim of supersonic combustor designers is to create a combustor with a high efficiency while minimising total pressure losses caused by the fuel injection and flame holding devices. Since thrust is a feature of both the amount of fuel burned and the pressure losses caused inside the combustor, high performance does not always lead to higher engine thrust. To achieve high efficiencies, the injector must produce rapid fuel and air mixing and combustion, allowing for a shorter combustor and therefore weight and drag savings. Struts (Masuya et al., 1995), ramps (Sands et al., 1997), and cavities (Baev et al., 1983, Mathur et al., 1999 and 2000) have all been used to improve mixing and provide a subsonic setting. a region from which to anchor a flame, but at the expense of pressure losses caused by these systems [2]. Ramps, tabs, lobe mixers, chevrons, and other techniques for enhancing mixing are focused on the generation of stream wise vorticity. Others, such as cavities [3], are dependent on self-excitation. To integrate cavities in scramjet engines, it is important to first understand cavity behaviour. Cavity has been shown to be a new legitimate flame holder for supersonic combustion and has been used in a variety of scramjet models [4]. The Central Institution of Aviation Motors (CIAM) in Moscow designed the cavity flame holders. Since recirculation flow in the cavity will provide a steady flame while increasing the rate of mixing or combustion, it is of interest. When compared to other mixing-enhancement systems, these devices experience less total pressure losses [3]. [In-Seuck Jeung and colleagues, 2005] This paper provides a brief overview of current activities in supersonic combustion in scramjet engines, which will be accompanied by a description of numerical simulations of supersonic combustion. Scramjet engine combustors and ram accelerator-related phenomena Grid refinement, scheme, unsteadiness, and phenomenological variations were all highlighted. [2004, Kyung Moo Kim 1 et al.] This paper explains numerical investigations into the combustion enhancement when hydrogen fuel is pumped through a transverse slot nozzle into a supersonic hot air stream using a cavity. Since recirculation flow in the cavity will provide a steady flame while increasing the rate of mixing or combustion, it is of interest. The reactive flow characteristics of many inclined cavities with different aft wall angles, offset ratios, and lengths are investigated. From the standpoint of total pressure loss and combustion efficiency, the cavity effect is addressed. As compared to the case without a cavity, the combustor with a cavity improves mixing and combustion while increasing pressure loss. However, in terms of combustion efficiency and total pressure loss, there is an acceptable length of cavity. [Tianwen Fang Meng Ding et al., 2008].The effects of cavities of various sizes on the supersonic flow area were studied experimentally and numerically, and the properties of supersonic cold flows over cavities were investigated. The findings show that the length-to-depth ratio L/D in the 5–9 range has no effect on cavity flow integral structures. Within the range of 30°–60°, the bevel angle of the rear wall has no effect on the overall structure of the cavity flow, but it can have a noticeable effect on the evolution of the shear layer and vortexes in cavities. [Wai-Sun Don and friends, 2003] This paper serves two purposes: it introduces a multidomain Chebyshev method for solving the two-dimensional reactive compressible Navier–Stokes equations, and it reports on the effects of using this code to numerically simulate reactive flows with large Mach numbers in recessed cavities. To stabilise the computations, the computational method makes use of newly derived interface boundary conditions as well as an adaptive filtering technique. The simulation findings are applicable to flame-holders with recessed cavities.

3. CAVITY FLOW

In SCRAMJET, the term cavity refers to a passive system that can achieve two different properties. One is to increase the mixing of fuel and air, while the other is to keep the flame alive in a Supersonic Combustion Ram-jet (SCRAMJET). These two distinct properties can be accomplished by altering the Cavity's aspect (L/D) ratio. Some excellent review papers on aspects of high-speed flow behaviour over the cavity have been published. The boundary layer ahead of the cavity divides at the leading edge, forming a free shear layer around the cavity in general. Flow recirculation occurs within the cavity [2]. The shear layer reattaches downstream at a different location. The reattachment point is defined by the cavity's geometry as well as the external flow conditions. The cavities are graded as “open” or “closed” depending on the reattachment point [3, 6]. The reattachment takes place on the cavity's back wall in an open cavity and on the lower wall in a closed cavity. The aspect ratio of open cavities is less than 7-10, while the aspect ratio of closed cavities is higher. The separated free shear layer can be detected locally upwards or downwards, creating a shock wave or an expansion wave at the leading edge, depending on the pressure within the cavity. The pressure loss in the cavity is caused by the leading edge expansion wave, flow separation, flow recirculation, reattachment of the free shear layer, and trailing edge shock, all of which must be held low for reasonable results.

3.1 Cavity as a Flame Holder in High-Speed Flow:

The following observations are made from high-speed cavity flame holder studies [9].

1. Mass entrainment rate and residence time was dependents upon the cavity geometry. Longer the length of cavity, lesser was the cavity residence time Slanted rear wall also reduced the cavity residence time.
2. Longer the length of cavity, higher was the mass entrainment inside the cavity, whereas depth of cavity residence time.
3. Higher the aspect ratio, higher was the drag coefficients. In addition as the rear wall angle decreased from 90 degree, the cavity drag increased.
4. Cavities with o set ratio more than 1 reduced the pressure at the cavity base and thereby increased the drag.

4. COMBUSTION

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. Thus combustion may be represented symbolically by

Fuel + Oxidizer ⍰ Products of combustion+ Energy

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 the design. 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.

4.1 Combustor Requirements and Problems

- Comparatively low temperature in outflow
- Specified temperature profile in outflow
- High velocities in front of and behind
- Require wide stability range for pressure level
- Relightning at wind-milling condition
- Small volume, low weight
- Low pressure drop
- No pollutant production

5. 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.

(i) Pre-processing, which involves a. Geometric modeling
b. Grid-generation
c. Flow specification
(ii) Solution stage involving the solution of algebraic equations
(iii) Post processing stage, which involves analyzing the results from vector plots, contour plots, surface plots and other data visualization tools.

5.1 Basic steps involved in CFD analysis

I. Problem Identification and Pre-Processing
1. Define your modeling goals.
2. Identify the domain you will model.
3. Design and create the grid.
II. Solver Execution
1. Set up the numerical model.
2 Compute and monitor the solution.
III. Post-Processing
1. Examine the results.
2. Consider revisions to the model.

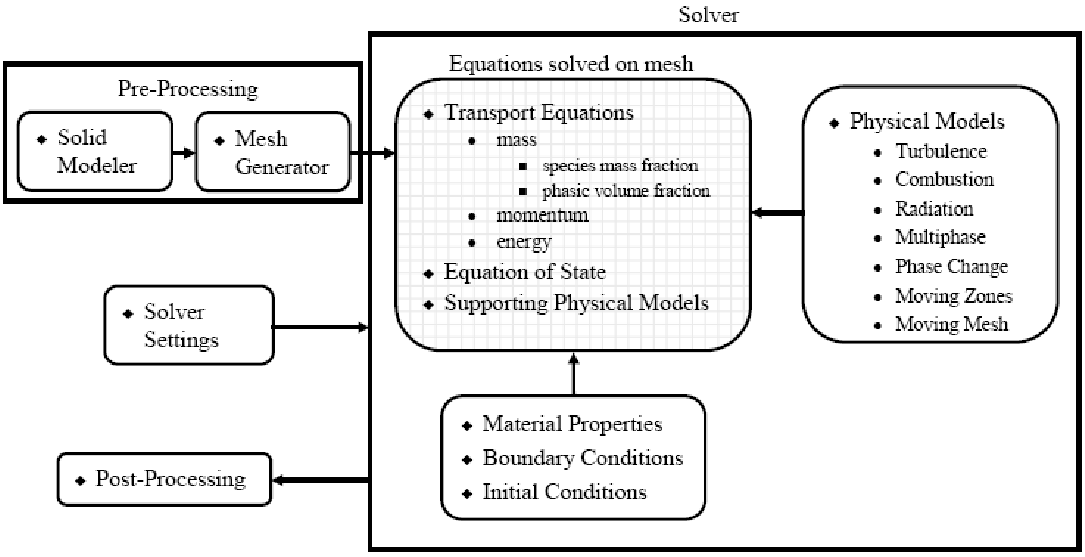

Fig .1 General overview of CFD modeling

5.2 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

5.3 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 **5.3.1Numerical Simulation of Supersonic Combustion for Hypersonic Propulsion** 5th Asia-Pacific Conference on Combustion, The University of Adelaide, Adelaide, Australia 18-20 July 2005. So this work is about to analyze 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 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.

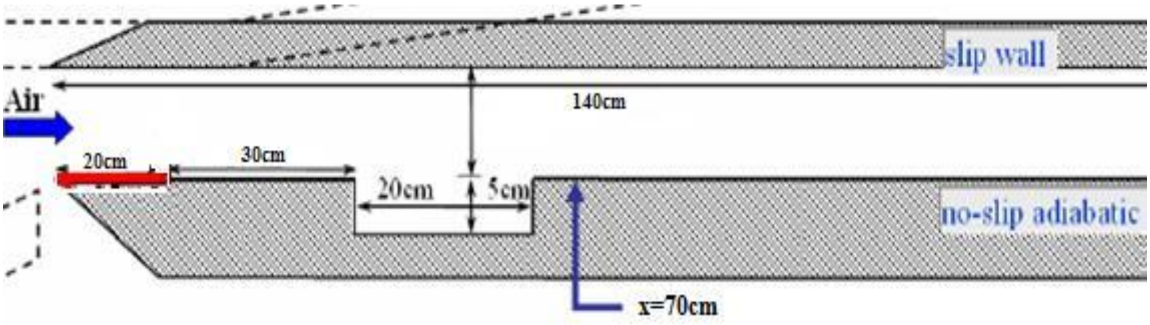

Fig.2.Simple Combustor configuration

5.4 Model types

Model without cavity Injector size is 10cm.

- Model without cavity Injector size is 20cm.
- Model with cavity L/D 4, Injector size is 10cm.
- Model with cavity L/D 4, Injector size is 20cm.
- Model with cavity L/D 3, Injector size is 10cm.
- Model with cavity L/D 3, Injector size is 20cm.

Combustion studies using different fuels are analyzed, using the above models and calculating the total pressure loss and combustion efficiency of each fuel and computing which fuel and which model is efficient for combustion.

5.5 Procedure followed for modeling geometry and Grid Generation.

1. Points are created as per the geometry.
2. The edges are created by joining the points and face is formed by using all edges.
3. The geometry is transferred into the meshing section.
4. The boundary layer is created as per boundary layer thickness
5. The edges are meshed by using the edge mesh tools
6. The full face of combustor with and without cavity is meshed by using the face mesh tool.
7. Boundary zones are specified.
8. The refinement of mesh has done and the respective face meshed models are as shown in below figures.

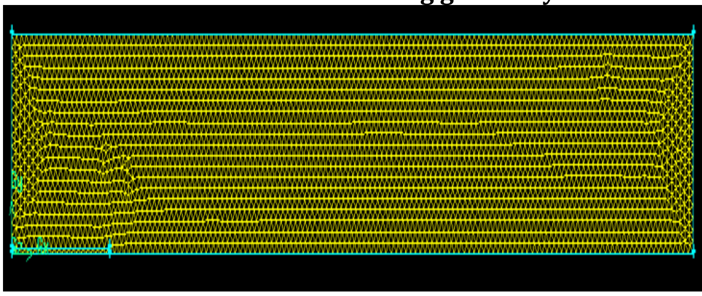

Fig.3 Mesh model without cavity, Injector 10cm

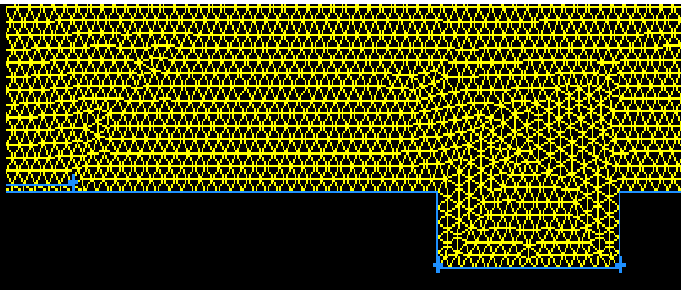

Fig .4Mesh model with cavity L/D ratio is 3, Injector 10cm

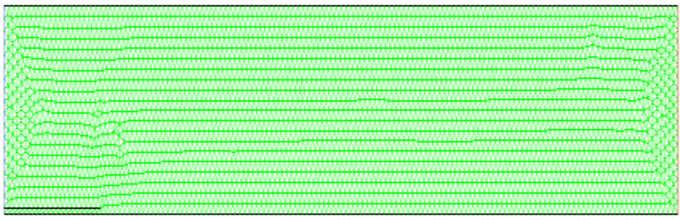

Fig.5.Grid without cavity, Injector 10cm

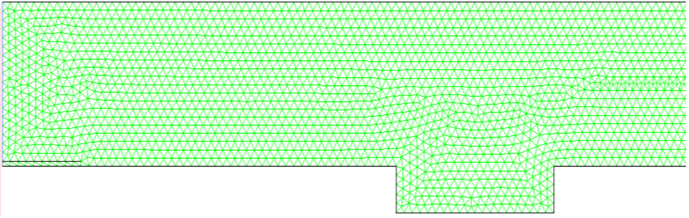

Fig.6.Grid with cavity L/D ratio is 3, Injector 10cm

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 present chapter gives a detailed account of the results obtained including Mesh Refinement, the contours of different parameters viz. Temperature, Pressure, Flow visualization, Velocity magnitude, Mass fraction, Streamline, and the comparison of different models with Methane, Kerosene and Octane fuels. From the beginning, inlet boundary condition was simulated as uniform values for all the models, but posed in convergence problem. So, by considering this problem as uniform inlet boundary condition, inlet profile boundary has been simulated which converged nicely. The converged solution is explained below.

6.1 Temperature contours of different models with different fuels.

The below contours shows the combustion or Peak temperatures of METHANE, OCTANE, KEROSENE fuels with different models respectively.

6.2 Model without cavity, fuel injector size is 10cm

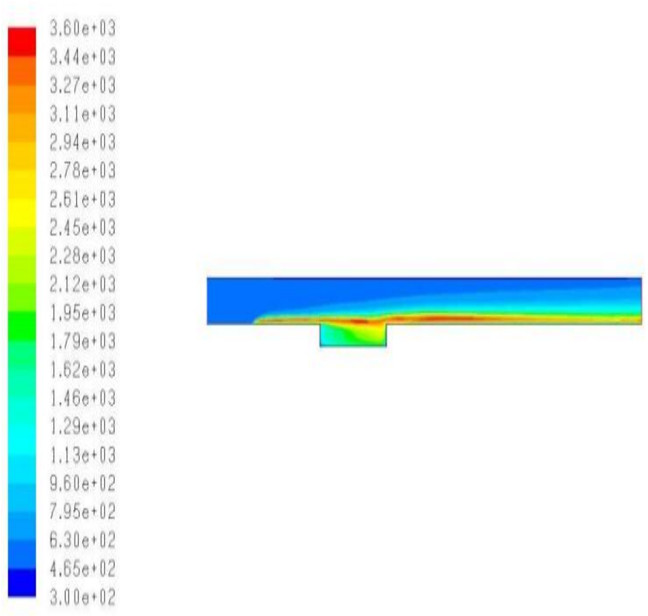

Fig.7. Temperature contour of Methane fuel with cavity L/D 3

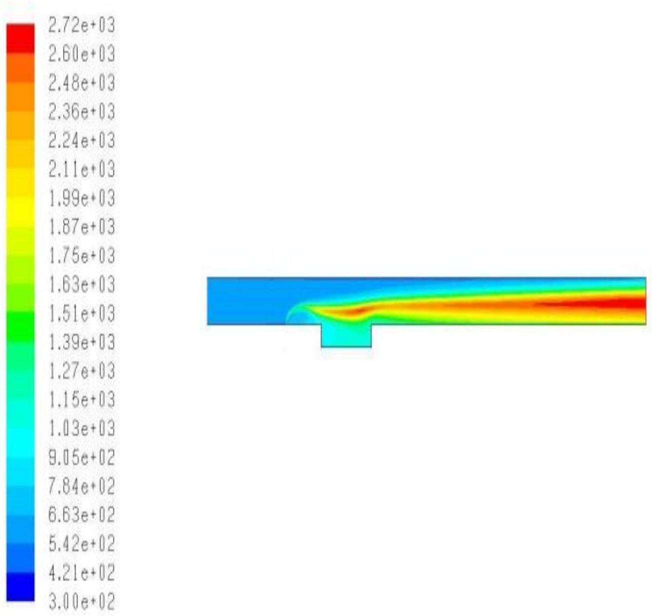

Fig.8. Octane fuel with cavity L/D 3

Table.1.Combustion Efficiency of different Fuels

Model type	Enthalpy input (J/KG)	Enthalpy output (J/KG)	Combustion efficiency %
Without cavity	295249.2	164871.7	44.15
With cavity L/D 4	299199.9	147159.2	50.81
With cavity L/D 3	300703.3	239573.8	46.92
Injector placed 10 cm far L/D 4	299177.3	137016.2	**54.20**
Injector placed 10 cm far L/D 3	298919.7	150789.9	49.55

Table.2.Combustion efficiency of Octane fuel

Model type	Enthalpy input (J/KG)	Enthalpy output (J/KG)	Combustion efficiency %
Without cavity	276573.19	192311.34	30.466
With cavity L/D 4	280445.4	154203	45.01
With cavity L/D 3	280581.8	148496.3	47.07
Injector placed 10 cm far L/D 4	280488	178625	36.31
Injector placed 10 cm far L/D 3	280559.8	139167.2	**50.396**

Table.3. Combustion efficiency of Kerosene fuel

Model type	Enthalpy input (J/KG)	Enthalpy output (J/KG)	Combustion efficiency %
Without cavity	279884.91	191322.67	31.64
With cavity L/D 4	283848.7	172765	39.13
With cavity L/D 3	283895.8	166362.9	41.40
Injector placed 10 cm far L/D 4	283793.3	185118.9	34.76
Injector placed 10 cm far L/D 3	284005.4	152812	**46.19**

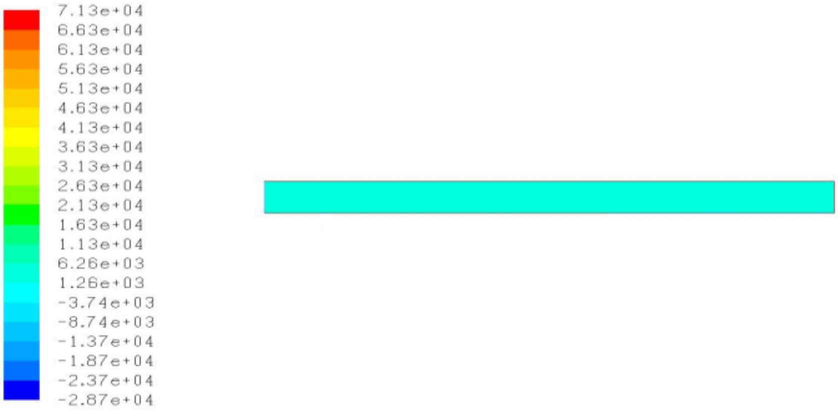

Fig.9. Pressure contour of model without cavity

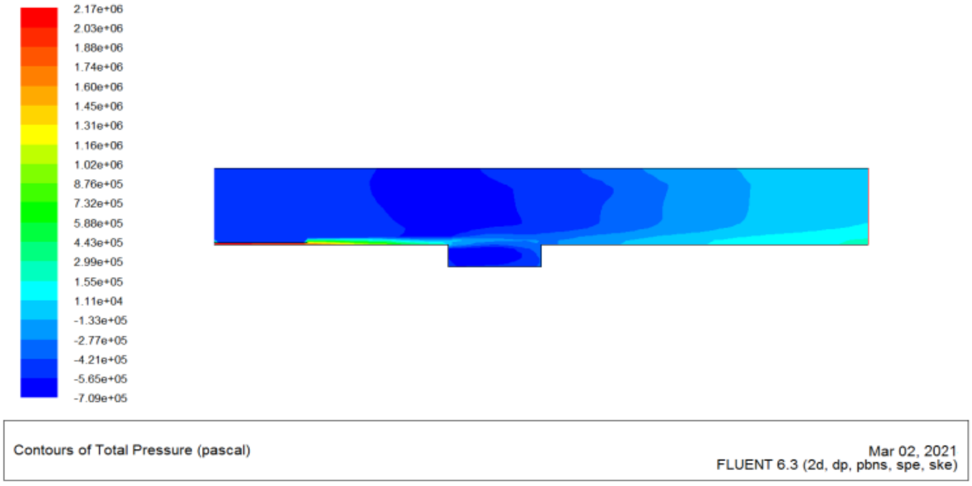

Fig.10.Pressure contour of model with cavity L/D 4

6.3 Total pressure contours

Fig 9 &10 shows the total pressure contours over the whole computational domain which indicates lower total pressure inside the cavity. As it is well known that the cost of getting flame holding and mixing properties by using cavity is total pressure loss. The total pressures loss is calculated as the ratio of difference of total pressure at the inlet and outlet of the domain and inlet total pressure. The values of total pressures are based on mass-weighted average

6.3.1 The total pressures loss calculation

The total pressures loss is calculated as the ratio of

difference of total pressure at the inlet and outlet of the domain and inlet total pressure. The values of total pressures are based on mass- weighted average.
The total pressure loss = $(P_{01} - P_{02}) / P_{01}$

- P_{01} = the inlet mass-weighted average pressure. P_{02} = the outlet mass-weighted average pressure.
- The values of total pressure input and output for methane, octane and kerosene fuel is as noted in below tables, and the total pressure loss is calculated.
- The total pressure loss for the model without cavity = (5929.85 - 4910.73) /(5929.85)
- The total pressure loss = 17.18
- As same total pressure loss is calculated for all models.
- Pressure loss of Methane fuel

Table.4.1.Pressure loss of Methane fuel

Model type	P_{01} (Pa)	P_{02} (Pa)	Pressure
Without cavity	15935.11	6648.731	58.3
With cavity L/D 4	15622.125	5785.748	62.9
With cavity L/D 3	6413.088	6244.105	22.95
Injector placed 10 cm far L/D 4	7166.50	5463.86	23.75
Injector placed 10 cm far L/D 3	6902.98	5392.84	**21.87**

Table.4.2.Pressure loss of Octane fuel

Model type	P_{01} (Pa)	P_{02} (Pa)	Pressure Loss %
Without cavity	58487.145	12265.567	**79.02**
With cavity L/D 4	74569.23	12660.48	83.02
With cavity L/D 3	67953.78	12572.17	81.49
Injector placed 10 cm far L/D 4	70855.44	12205.87	82.77
Injector placed 10 cm	74737.65	12010.28	83.93

Table.5.Pressure loss of Kerosene fuel

Model type	P_{01} (Pa)	P_{02} (Pa)	Pressure Loss %
Without cavity	94773.91	13975.78	**85.25**
With cavity L/D 4	128933.3	14502.76	88.75
With cavity L/D 3	116331.2	14842.75	87.24
Injector placed 10 cm far L/D 4	122381.8	14021.49	88.54
Injector placed 10 cm far L/D 3	141487.8	14393.48	89.82

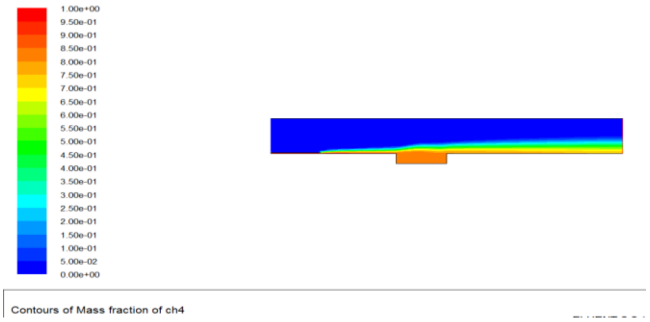

Fig.11.Mass fraction of CH_4

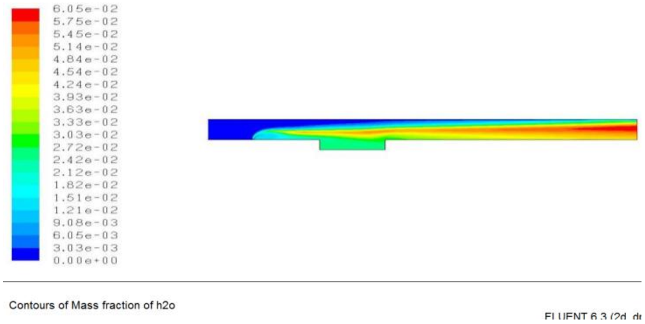

Fig.12.Mass fraction H_2O

6.4 Comparison of Pressure loss and Combustion Efficiency of different models with different fuels.

Table.6. Pressure loss comparison of different models with different Fuels

Model type	Pressure loss% methane	Pressure loss % octane	Pressure loss % kerosene
Without cavity	**17.18**	**79.02**	**85.25**
With cavity	23.02	83.02	88.75
With cavity	22.95	81.49	87.24
Injector placed 10 cm far L/D 4	23.75	82.77	88.54
Injector placed 10 cm far L/D 3	21.87	83.93	89.82

Table.7.Combustion Efficiency comparison of different models with different Fuels

Model type	Combustion efficiency % METHANE	Combustion efficiency % OCTANE	Combustion efficiency % KEROSENE
Without cavity	44.15	30.466	31.64
With cavity	50.81	45.01	39.13
With cavity	**54.20**	36.31	34.76
Injector placed	46.92	47.07	41.40
Injector placed	49.55	**50.396**	**46.19**

6.5 Graph which shows the combustion efficiency and total pressure loss of different fuels with different models

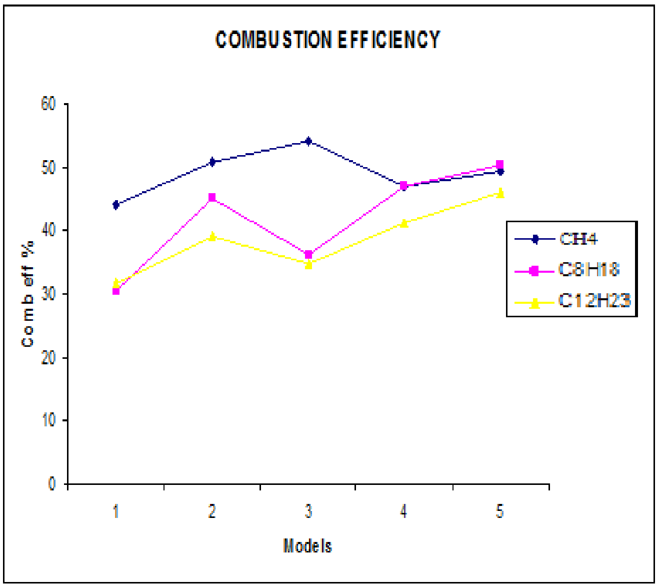

Fig.13. Combustion efficiency v/s Different models

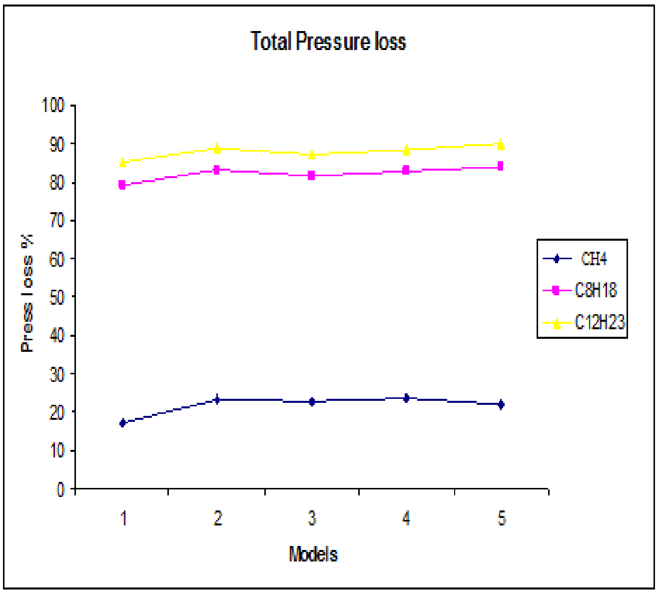

Fig.14.Total pressure loss v/s Different models

In the graph (fig.13) it has been clearly shown that the Methane fuel has higher combustion efficiency than the Octane and Kerosene fuels. In the graph (fig.14) it has been clearly shown that the Methane fuel has less pressure loss than the Octane and Kerosene fuels. So from this we can conclude that methane fuel is better for the model which we have analyzed for combustion at subsonic speed. Comparing the models with and without cavity with Methane, Octane and Kerosene fuels, the pressure losses are less in a combustor when Methane Fuel is used for the combustion process. The model without cavity has less pressure loss compared to the models with cavity. In the models with cavity the pressure loss is less when the fuel injector is placed far the cavity L/D 3. Comparing the models with and without cavity with Methane, Octane and Kerosene fuels, it is observed the combustion efficiency is better when methane fuel is used in a combustor .The combustion efficiency is high in the model when the injector is placed far cavity L/D ratio 4, it is compared in the table.

6.5 Peak temperature, Exit temperature and Exit velocity values of different models and different fuels are noted in the below tables.

Table.8.Outlet values of Methane fuel

Model type	Peak temperature $*10^3$	Exit temperature	Exit velocity
Without cavity	3.77	855.05	151.27
With cavity L/D 4	3.60	932.02	166.05
With cavity L/D 3	3.66	929.38	164.23
Injector placed 10 cm far L/D 4	3.65	926.52	166.14
Injector placed 10 cm far L/D 3	3.68	896.77	162.54

Table.9.Outlet values of Octane fuel

Model type	Peak temperature $*10^3$	Exit temperature	Exit velocity
Without cavity	2.70	2009.10	366.415
With cavity L/D 4	2.71	2041.13	375.06
With cavity L/D 3	2.68	2102.16	386.34
Injector placed 10 cm far L/D 4	2.76	1973.83	362.07
Injector placed 10 cm far L/D 3	2.75	2019.12	370.84

Table.10.Outlet values of Kerosene fuel

Model type	Peak temperature $*10^3$	Exit temperature	Exit velocity
Without cavity	2.81	1916.40	344.20
With cavity L/D 4	2.73	1941.4	351.93
With cavity L/D 3	2.67	1958.41	353.59
Injector placed 10 cm far L/D 4	2.79	1871.97	339.30
Injector placed 10 cm far L/D 3	2.72	1842.09	332.70

The values as shown in table(8-10), comparing to the Kerosene and Octane fuel, Methane fuel have less exit temperature, so a good combustor requires less exit temperature as per this Methane fuel is efficient for combustion, but the peak temperature is high and exit velocity is low. The exit velocity is high when Kerosene fuel is used for combustion.

Conclusion

As per the Computational analysis, the combustor with cavity is efficient for combustion.

- In this research we analyzed the fuel injector with different dimensions for their combustion efficiencies and pressure losses.
- Overall flow features like recirculation zone inside the cavity, combustion in a combustor with cavity, mixing of air and fuel, & velocity, Stream function of flow contours are noted and examined.
- The different models are explained and flow is analyzed, as per the computational analysis the combustor with cavity L/D ratio 4, 10cm fuel injector is more efficient for the methane fuel.
- The different models are explained and flow is analyzed, as per the computational analysis the combustor with cavity L/D ratio 3, 10cm fuel injector is more efficient for both the fuels octane and kerosene.
- By the comparison of pressure losses the combustor without cavity and 10cm fuel injector has low pressure loss for all fuels methane, octane and kerosene.
- From this research it can be concluded that the combustor with cavity L/D ratio
- 3 & 10cm fuel injector is good and we can implement experimentally to a subsonic combustion chamber.

References

[1]. Anderson, J.D. Jr. “Computational fluid dynamics”, McGraw-Hill, Inc.1995

[2]. S. Eddington. “The Role of Plasma Igniters and Fuel Injectors in Hypersonic Technology”. (British Astrophysicist, 1882-1933).

[3]. Adela Ben-Yakar and Ronald K. Hanson. “JOURNAL OF PROPULSION AND POWER”, Stanford University, California 94305 Vol. 17, No. 4, July– August 2001.

[4]. In-Seuck Jeung1and Jeong-Yeol Choi. “Numerical Simulation of Supersonic Combustion for Hypersonic Propulsion”. 5th Asia-Pacific Conference on Combustion. The University of Adelaide, Adelaide, Australia 18-20 July 2005.

[5]. A. J. Neely, C. Riley, R. R. Boyce, N. R. Mudford. “12th AIAA International Space Planes and Hypersonic Systems and Technologies”. 15 – 19 December 2003, Norfolk, Virginia.

[6]. Kyung Moo Kim 1, 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.

[7]. Takahashi Shuhei, Tanaka hideyasu, Wakai Kazunori. (Univ. of Tokyo, Fac. Of Eng.), “Supersonic Combustion with Liquid Hydrocarbon Fuel”, Journal Title; Proceedings of the Space Sciences and Technology Conference, Vol.46th, no.Pt.2, PAGE.566-570(2002).

[8]. Chadwick C. Rasmussen, Sulabh K. Dhanuka, James F. Driscoll. “Visualization of flame holding mechanisms in a supersonic combustor using PLIF Proceedings of the Combustion Institute”. Volume 31, Issue 2, January 2007, Pages 2505-2512.

[9]. Frederick S. Billig. “Supersonic combustion of storable liquid fuels in Mach 3.0 to 5.0 air streams Symposium (International) on Combustion”, Volume 10, Issue 1, 1965, Pages 1167-1178.

[10].A.Paull, R.J.Stalker And D.J.Mee. “Experiments on supersonic combustion ramjet propulsion in a shock tunnel”, Department of Mechanical Engineering, The University of Queensland, Brisbane, 4012 Queensland, Australia (Received 26 July 1994 and in revised form 28 February 1995).
[11].Tianwen Fang Meng Ding and Jin Zhou. “Supersonic flows over cavities Journal Frontiers of Energy and Power Engineering in China”, Research Article, Publisher higher education press, June 11, 2008.
[12].Tarun Mathur, Mark Gruber, Kevin Jackson. “Supersonic Combustion Experiments with a Cavity-Based Fuel Injector”, APPROVED FOR PUBLIC RELEASE, Journal article, NOV 2001.
[13].William H. Allen Jr. “Fuel-Air Injection Effects on Combustion in Cavity- Based Flameholder in a Supersonic Flow”. APPROVED FOR PUBLIC RELEASE, Master's thesis, MAR 2005, report number A223434.
[14].Fernando Schneider, Peter Gerlinger and Manfred Aigner. “Enhanced Mixing in Supersonic Combustion High Performance Computing in Science and Engineering”, 04 Transactions of the High Performance Computing Center Stuttgart (HLRS) 2004

BIOGRAPHIES (Optional not mandatory)

A J SRIGANAPATHY, Assistant Professor, Department of Aeronautical Engineering, Mahendra Institute of Engineering and Technology, Namakkal, India.

R. LOGANATHAN, UG Scholar, Department of Aeronautical Engineering, Mahendra Institute of Engineering and Technology, Namakkal, India.

S. MONISHRAJ, UG Scholar, Department of Aeronautical Engineering, Mahendra Institute of Engineering and Technology, Namakkal, India.

R. RANJITHKUMAR, UG Scholar, Department of Aeronautical Engineering, Mahendra Institute of Engineering and Technology, Namakkal, India.

R. SATHISHKUMAR, UG Scholar, Department of Aeronautical Engineering, Mahendra Institute of Engineering and Technology, Namakkal, India.