

See discussions, stats, and author profiles for this publication at: <https://www.researchgate.net/publication/324130164>

Numerical Study of Cavity-Based Flame-Holder with Slot Injection for Supersonic Combustion

Article · April 2018

DOI: 10.24247/ijmperdapr2018102

CITATIONS

0

READS

191

2 authors, including:



Bhupinder Singh

Chandigarh University

3 PUBLICATIONS 0 CITATIONS

SEE PROFILE

NUMERICAL STUDY OF CAVITY-BASED FLAME-HOLDER WITH SLOT INJECTION FOR SUPERSONIC COMBUSTION

SHUBHANKAR BHAKTA & BHUPINDER SINGH

Department of Automobile Engineering, Chandigarh University, Mohali, Punjab, India

ABSTRACT

The current paper has focused on finding the optimum cavity properties to enhance the mixing phenomenon and combustion processes in supersonic flow using Computational Fluid Dynamics (CFD) ANSYS Fluent. Various models have been modeled with different values of offset ratios (OR) and aft ramp angles (θ) while keeping the length-to-depth ratio constant. All cavity flows have taken open type, i.e. length-to-depth ratio <10 . A cavity with a width of 1mm was used for hydrogen fuel injection over and done with transverse slot nozzle into a supersonic air stream flow with Mach number 3.75. The free stream pressure, temperature and Reynolds number were 1.2MPa, 299K, and 2.07×10^7 respectively. The cavity provides a stable flame, rapid mixing of fuel and oxidizer and minimizes the total pressure losses thus making cavity a promising flame-holder technique. From a viewpoint of combustion efficiency and total pressure loss the cavity influence is discussed and compared with the without cavity model. When compared with the without cavity model the mixing properties and combustion processes were enhanced. Both the total pressure loss and combustion efficiency are increased with increasing aft ramp angle θ . Whereas, an increment in OR has the adverse effect on both because of less obstruction generated on the supersonic flow field. Higher ramp angle caused higher eddy-viscous turbulent flow and wall drag which increased the total pressure losses.

KEYWORDS: Cavity, Nozzle, Supersonic Combustion, Combustion Efficiency, Flame-Holder & Total Pressure Loss

Received: Feb 10, 2018; **Accepted:** Mar 03, 2018; **Published:** Mar 16, 2018; **Paper Id.:** IJMPERDAPR2018102

NOMENCLATURE

D_u or D	=	Upstream depth.
D_d	=	Downstream depth.
OR	=	Offset ratio D_u/D_d .
M	=	Mach number.
P_w	=	Static wall pressure.
P_t	=	Total pressure.
∞	=	Free stream condition.
L/D	=	Cavity length-to-depth ratio.
V	=	Velocity.
U	=	u-direction velocity.

m	=	Mass.
Θ	=	Aft ramp angle.
ρ	=	Density.
Φ	=	Equivalence ratio, mass flow rate.
ref	=	Reference value.

1. INTRODUCTION

Supersonic Combustion ramjet engine (SCRAMJET) is a new type of air-breathing propulsion device for the hypersonic flight ($M > 5$) which uses the vehicle's forward motion to compress incoming air an axial or rotatory compressor [1]. A scramjet basically contains three components namely converging inlet, combustor chamber and diverging nozzle. The process starts with the inlet air which is compressed by the high speed of the vehicle in the first chamber and in the combustion chamber where fuel is combusted. At diverging nozzle, the exhaust jet expands at a higher speed than the inlet air [3]-[5]. Unlike Ramjet, a scramjet has no moving parts. Figure 1 shows the components and the processes of scramjet engine based on the stages. The incoming air is responsible for operating the supersonic combustion ramjet engine at extremely high speed: theoretically, the top speed of a scramjet has been projected between Mach 12 (16,000km/h) to Mach 24 (25,000km/h) [10]. After 1991, when first Scramjet flight test was claimed the researchers and developments have been boosted the study of combustion in supersonic flows. Supersonic combustion is an important technology for improving the efficiency of Advanced Technology Vehicles (ATVs). This offers a substitute to a rocket-driven system which conveyances fuel and oxidizer [15].

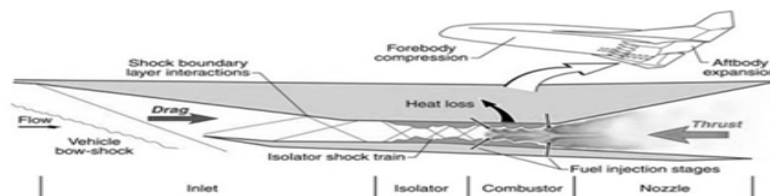


Figure 1: Schematic of a Scramjet Engine Indicating Engine Processes and Components. Courtesy: NASA Langley

Compound design and continuous thermodynamic turbulent fluid dynamics make the modeling of a supersonic combustion highly complex process. Computational outlays are directly related to the time gauges of combustion reactions and turbulent eddies [4]. Major subjects like FUEL-AIR mixing, flame holding, cavity drag, pressure losses, flame stabilization and thermal loading management, ignition, an interaction between the isolator and the combustor are essential to be solved for the successful design and application of hydrogen and/or hydrocarbon-fueled Scramjet engines [1]-[14]. In order for the scramjet to work efficiently, designers must choose a fuel with rapid burn properties generating a large amount of thrust. Hydrogen and Hydrocarbons are the most suitable solution. Above technological challenges make Scramjet a complex phenomenon to master.

A cavity can have a larger impact on the flow dynamics and surroundings aerodynamically. M. R Gruber et al has condemned drag the reason for lower flow separation over a cavity than bluff-body, with relatively little pressure drop, which makes the cavity a stable flame -holder device inside the combustor [9]. Different fuels affect the specific impulse of the exhaust gases of supersonic combustion. The SCRAMJETS are the fastest among all another type of jets.

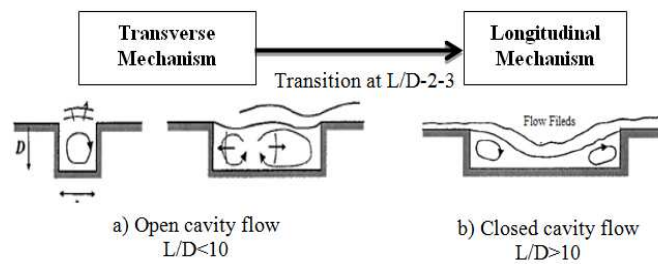


Figure 2: Flow Field Schematics of Cavities with Different L/D Ratios

Basically, there could be 2 types of cavity flow, namely: Open and/or Closed [1], [8], [12]. Flow-fields are shown in figure 2 schematically. The length-to-depth ratio for open cavity flow is $L/D < 10$. For this, the shear layer formation occurs at the separation point and attaches somewhere at the downstream depth of cavity [1], [7], [12], [13]. Unlike open cavity flow, in the closed cavity flow ($L/D > 10$) the shear layer doesn't go for entire cavity length and instantly reattach on the cavity bottom floor. The open cavities are more appropriate in scramjet combustor because of the drag coefficients are considered larger in closed cavity [1], [13], [15].

2. METHODOLOGY

2.1. Modeling Methods

In this paper the parallel simulation of a Scramjet combustor with compressive, viscous, supersonic flow over the cavity when fuel is injected upstream is simulated. The 2D steady-state Navier-Stokes equations along with the κ - ϵ two-equation turbulence model with finite-rate chemistry equations are used to model hydrogen/air combustion in Computational Fluid Dynamics CFD on ANSYS Fluent. The pressure based fluent solver with second-order scheme has been used for the study. Due to finite volume method used, the governing equations are discretized in a conventional form. The equations are integrated over control volume where the time derivatives are estimated with an implicit first-order Euler scheme. The turbulent kinetic energy has been approximated with second-order scheme. To satisfy the discrete implicit continuity equations the pressure correction and the final momentum fields are combined.

2.2. Preliminary Design

The CATIA Modeling software and ANSYS DESIGN MODELER has been used for 2D and 3D modeling of the combustor model.

2.2.1. Test Model Design

2.2.1.1. 2D Design and Meshing

figure 3 shows the schematic two-dimensional geometry of the Supersonic engine combustor. The figure is representative of a complete the combustor but without a complex geometry. The area section of combustor is 500x150mm. Selecting a 2D geometry for the model, the design of the engine starts by defining the computational boundary conditions, shown in table 1, with free stream Mach number 3.75 (taken from Kim et al. [7] and Aso et al. [13]).



Figure 6: 2D Meshing of Model

A mesh refinement has been provided on the bottom wall and the injection area of the combustor as the results at these zones of combustor is acuter. To enhance the quality of the results, the bottom wall of the cavity is also refined because it worked as the combustion and mixing phenomenon device.

2.3. Cavity Geometry Specifications

Table 1 shows the geometric structures of the cavities studied. All types of models have the same upstream depth $D_u = 15\text{mm}$ and length L is defined in figure 6, accordingly. The length-to-depth ratio has been taken 3 because of medium cavity length i.e $L/D = 3$ the combustion efficiency is used to be highest [7].

Table 1: Cavity Geometry Specifications

Name	L/D_u	OR= D_u/D_d	Ramp angle Θ , deg
LD3-0.5-60	3	0.5	60
LD3-0.5-90	3	0.5	90
LD3-01-35	3	1	35
LD3-01-60	3	1	60
LD3-01-90	3	1	90

The aft ramp angle has been taken 35, 60 & 90 and the offset ratio OR has been chosen 0.5 and 1 to see the effect of downstream depth upon the flame-stabilization and combustion process. The visualization of the flow field and output parameters have taken on the engine centerline and through the cavity wall section.

2.4. Boundary Conditions

For validation and simulation of a reactive flow with upstream fuel injection of the cavity the following free stream inlet and injection, conditions were applied, given in table 2. The free stream conditions have been considered by Kim et al. 2003 [7] and Also et al. 1991 [13]. Slot converging sonic throat nozzle having width 1mm is used while hydrogen gas is injected into the flow with a total pressure of $P_{inj} = 10.29P_\infty$ [7].

Table 2: Computational Boundary Conditions

Properties	Free stream Conditions	Secondary Injection Conditions
Pressure (MPa)	1.2	12.348
Temperature (K)	299	600
Mach Number	3.75	10
Mass flow rate Φ	-	0.32
Reynolds number	2.07×10^7	10.6×10^7

3. RESULTS AND DISCUSSIONS

3.1. Code Validation

The foremost objective of present works in to observe the compressible, viscous supersonic flow over and within the cavity in the upstream injection of secondary flow. For the validation purposes of the present computational method,

two computing cases are considered here.

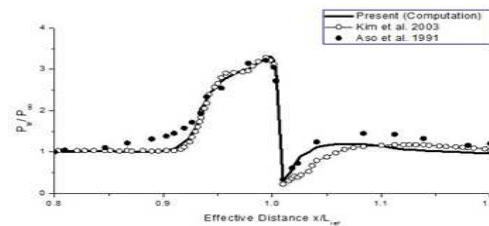


Figure 7: Wall Pressure Distribution

First Test Case

The flow conditions for this case the numerical and experimental data of Kim et al. [7] and Also et al. [13] are considered, respectively. The combustor geometry comprises a slot upstream injection of 1mm width and the distance of nozzle is $L_{ref} = 355\text{mm}$ from the leading edge of the combustor. The computation conditions are as follow in table 2. Figure 7 shows the wall surface pressure distribution with respect to the effective distance and compared with the experimental work by Also et al. and Kim et al.

Second Test Case

A study of Gruber et al. [9] is considered. They studied the various cavity geometry configurations for Mach 3 flow. To fill the research gap the current study has used slightly different cavity geometry configurations and inlet boundary parameters compared than Gruber et al., shown in table 3 and 4, respectively.

Table 3: Cavity Geometry Configuration

Parameters	Present (Computation)	Gruber et al. 2001
D_u (mm)	15	8.9
L/D	3	3 with aft angle 30° 5 without aft angle

Table 4: Free-Stream Conditions

Parameters	Present (Computation)	Gruber et al. 2001
Free stream Pressure	1.2 MPa	690kPa
Free stream Temperature (K)	299	300

Figure 8 presents the validation of current work with the study of Gruber et al. [9]. The comparison between present model LD3-01-35 and Gruber's LD3-01-30 of the surface pressure distribution along cavity wall has shown.

In the present computation study, the wall pressure has slightly decreased at the bottom right corner of the cavity just before the ramp of the cavity. The reason behind this is the use of a larger upstream depth of cavity which decreases the wall pressure as the cavity depth increases [9]. Because an increase in the aft ramp angle in the present study has generated a greater shock wave at trailing edge of cavity thus, an increment of wall pressure than the Gruber's data. The stream-line of the pressure distribution of present computation has shown in Figure 9 and compared with the Gruber's experiment one.

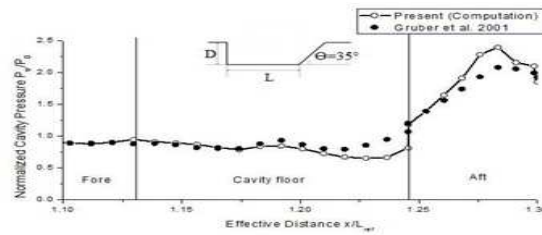


Figure 8: Comparison of Cavity Wall Static Pressure Distribution

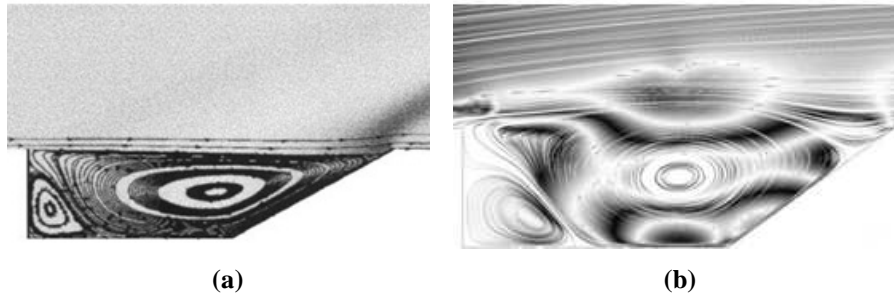


Figure 9: Computational Stream-Line Contour (a) Gruber's Computation (b) Present Computation

3.2. Present Computational Visualization and Results

3.2.1. Computational Visualization

The increase in the H_2O mass fraction as the gas expands around and after the cavity causes an improvement of injection scheme for better flame-holding [7] and the combustion efficiency is defined as the measure of the degree of decreasing the H_2 species after the injection and completeness of combustion [9]. Mathematically, Combustion efficiency defined;

$$\eta_c = \frac{\dot{m}_{fuel,in} - \dot{m}_{fuel,x}}{\dot{m}_{fuel,in}} \quad (1)$$

Where, $\dot{m}_{fuel,x}$ = local fuel mass flow rate.

The total pressure loss in supersonic flow is defined as;

$$\eta_{loss} = 1 - \frac{\int P_t \rho u dA}{\int P_{t,ref} \rho u dA} \quad (2)$$

Where, $P_{t,ref}$ = total pressure of the reference. The total pressure losses have various affecting factors like viscous forces in boundary wall, shock waves, wall stresses, flow separation etc. To see the effect of various cavity geometries on pressure loss, the pressure contours of all the models have been plotted down and compared with each other and with the pressure contours of without cavity flow field in figure 10.

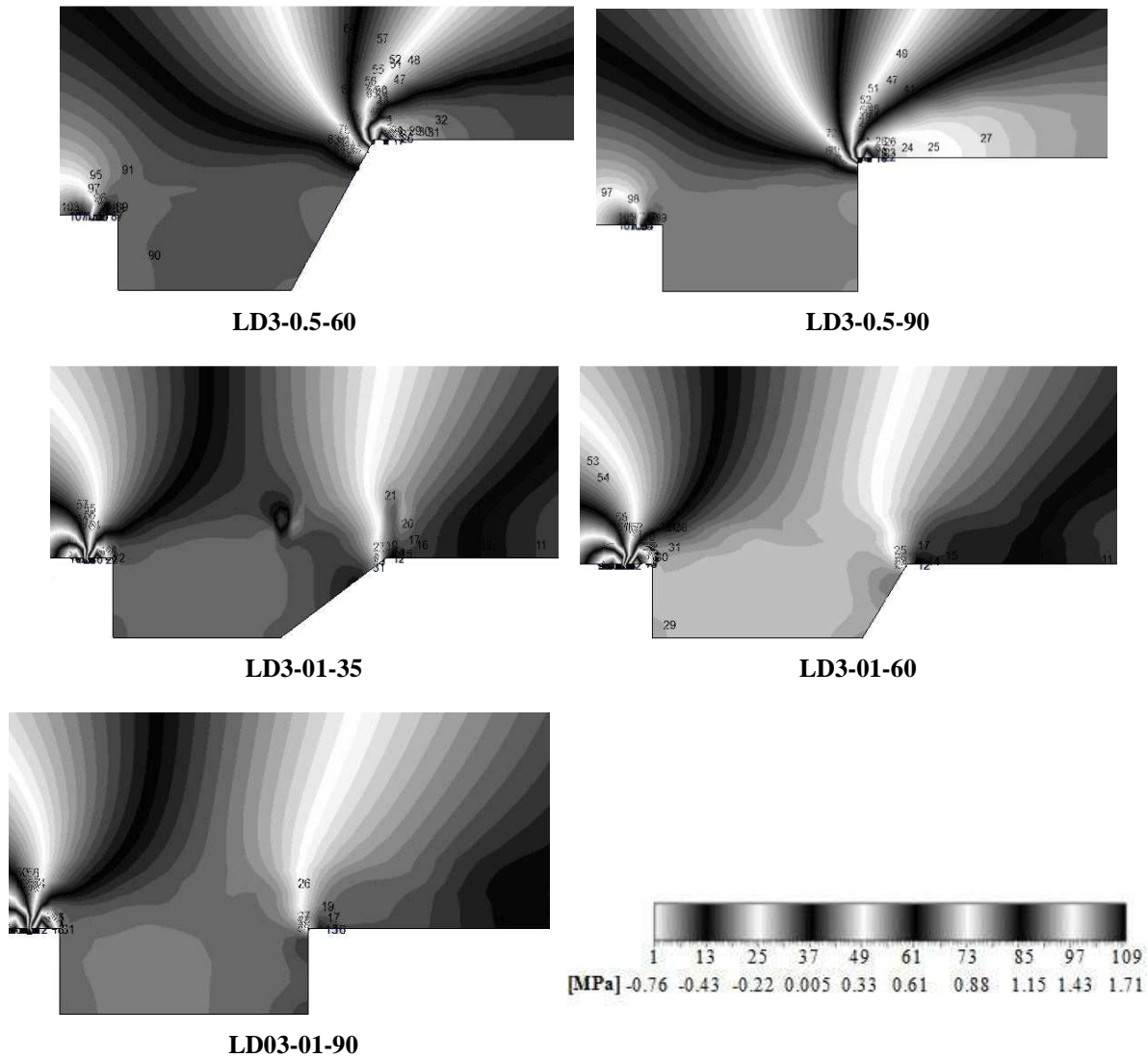


Figure 10: Computational Results of Static Pressure Contour for all Models

Generation of the shock wave in unsteadiness zones of cavities of rectangular geometries is uncommon when eddy-adhesive based turbulence flows are employed. Stronger eddy shock waves are produced at the trailing edges of the models with $OR = 0.5$ compared with $OR = 1$, thus, increasing the pressure losses due to low viscosity and cavity wall drag. The increasing in ramp angle Θ , wall of cavities showed the several pressure patterns and increases the pressure losses shown in figure 11.

3.2.2. Total Pressure Loss and Combustion Efficiency Data

Based on equations (1) and (2) the combustion efficiency and pressure losses are plotted with respect to the effective distance of the combustor for all models and compared with the base model (flow field without cavity).

a. Total Pressure Loss

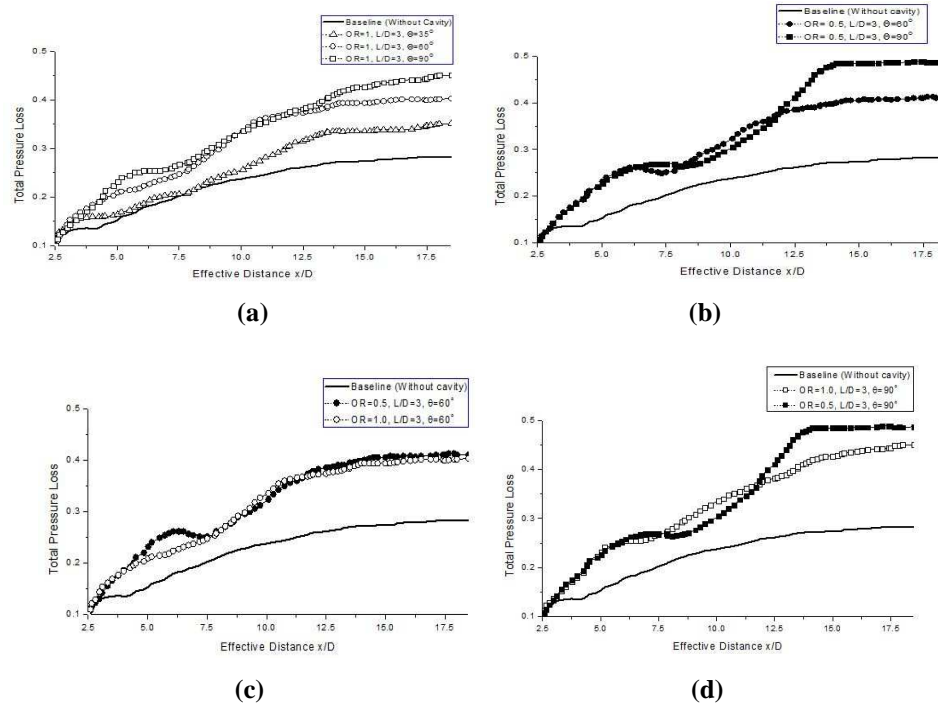


Figure 11: Total Pressure Loss for

- (a) Various aft Ramp Angle θ for $OR=1$ (b) Various aft Ramp Angle θ for $OR=0.5$
(c) Various OR for aft Ramp Angle $\theta=60^\circ$ (d) Various OR for aft Ramp Angle $\theta=90^\circ$

b. Combustion Efficiency

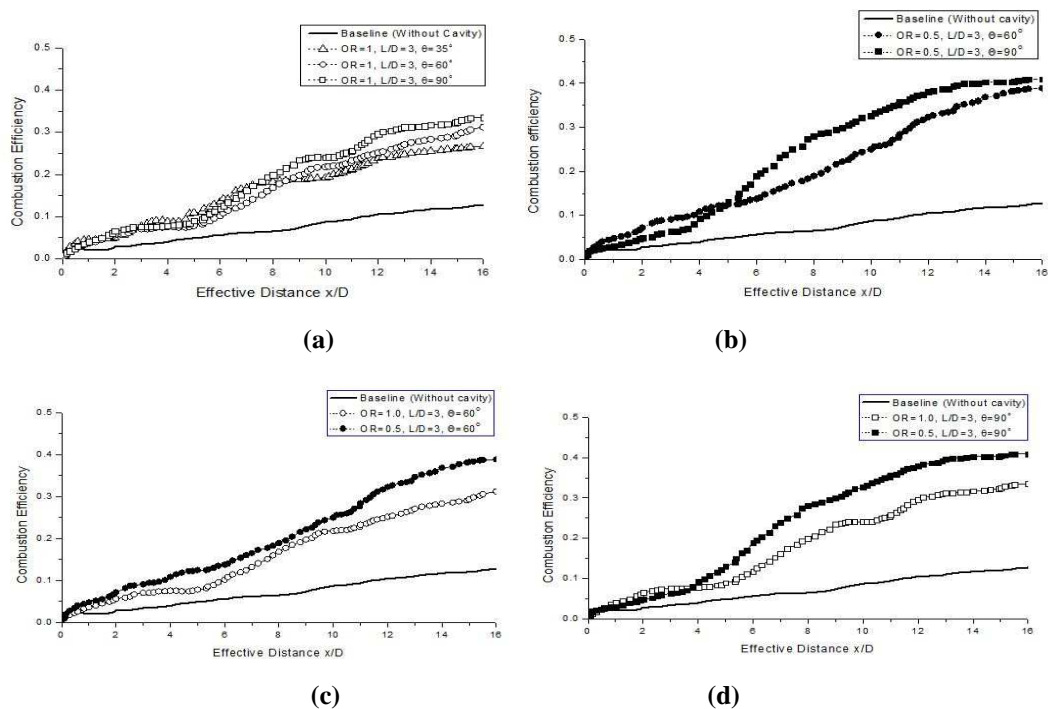


Figure 12: Combustion Efficiency for

- (a) Various aft Ramp Angle θ for $OR=1$ (b) Various aft Ramp Angle θ for $OR=0.5$
(c) Various OR for aft Ramp Angle $\theta=60^\circ$ (d) Various OR for aft Ramp Angle $\theta=90^\circ$

4. CONCLUSIONS

In this computational study, the flow field properties and effect of the various cavity geometries on the upstream hydrogen injection with a slot injector have been investigated. Their flow characteristics were visualized using CFD ANSYS FLUENT. The parameters i.e. wall static pressure, velocity, density, mass flow rate were measured span-wise at the bottom centerline to ensure the optimum effect of all the output parameters. Following results were found by varying the input parameter such that aft ramp angle Θ and offset ratios OR.

- By decreasing the offset ratio, both total pressure loss and the combustion efficiency increases, figure 11 (c) and (d) /figure 12 (c) and (d), respectively.
- As the aft ramp angle Θ increases, again both total pressure loss and the combustion efficiency increases, figure 11 (a) and (b) / figure 12 (a) and (b), respectively.
- As the objective of this study to discover the optimum cavity geometry among all the cavity geometry's specifications, the cavity with a moderate length-to-depth ratio, higher offset ratio OR and rectangular open cavity ramp angle $\Theta=90^\circ$ is the best optimum solution so far we, based on the input boundary conditions.

ACKNOWLEDGEMENT

The authors are gratefully acknowledged by the Department of Automobile Engineering, Chandigarh University.

REFERENCES

1. Adela Ben-Yakar & Ronald K. Hanson. *Cavity Flame-Holders for Ignition and Flame Stabilization in Scramjets: An Overview. Journal of propulsion and power. Vol. 17, No. 4, August 2001.*
2. Donald P. Rizzetta. *Numerical Simulation of Slot Injection into a Turbulent Supersonic Stream. AIAA JOURNAL Vol. 30, No. 10, DOI: 10.2514/3.11244, October 1992.*
3. Dr. Richard P. Hallion. *The Hypersonic Revolution Vol. II. Boiling AFB: Air Force History and Museums Program, (1998).*
4. F. Moura, V. Wheatley, T. J. McIntyre and I. Jahn. *On the development of a Mach 10 scramjet engine for investigation of supersonic combustion regimes. 20th Australasian Fluid Mechanics Conference, Perth, Australia, December 2016.*
5. K. M. Pandey, P Kalita, K Barman, A. Rajkhowa and S. N. Saikia. *CFD Analysis of Wall Injection with Large Sized Cavity Based Scramjet Combustion at Mach 2. IACSIT International Journal of Engineering and Technology, Vol.3, No.2, April 2011.*
6. K. Yu, K. J. Wilson, K. C. Shadow, *Effect of flame-holding cavities on supersonic combustion performance, AIAA-99- 2638, (1999).*
7. Kyung Moo Kim Seung, WookBaek, Cho Young Han. *Numerical study on supersonic combustion with cavity-based fuel injection, International Journal of Heat and Mass Transfer 47 (2004) 271–286. July 2003.*
8. M. R. Gruber J. M. Donbar, C. D. Carter and K.-Y. Hsu. *Mixing and Combustion Studies Using Cavity-Based Flame-holders in a Supersonic Flow, Journal of propulsion and Power, AIAA, DOI: 10.2514/1.5360. Vol. 20, No. 5, October 2004.*
9. M. R. Gruber, R. A. Baurle, T. Mathur and K.-Y. Hsu. *Fundamental Studies of Cavity-Based Flame holder Concepts for Supersonic Combustors. JOURNAL OF PROPULSION AND POWER Vol. 17, No. 1, DOI: 10.2514/2.5720. February 2001.*

10. Naidu KS and Bajaj DK. Modeling and Exhaust Nozzle Flow Simulations in a Scramjet. Naidu and Bajaj, *J Astrophys Aerospace Technol* 2015, 3:2. (2015).
11. Nithin. N. Numerical analysis of Supersonic combustion engine with Double Cavity configuration at Mach 2 fuel injection. *International Journal of Science and Research (IJSR) ISSN (Online): 2319-7064 Index Copernicus Value (2013): 6.14 / Impact Factor (2013): 4.438.* (2013).
12. R. A. Baurle, M. R. Gruber, A study of recessed cavity flowfields for supersonic combustion applications, AIAA- 98-0938, (1998).
13. Shigeru ASO, Satoshi UYAMA, Masafumi KAWAI and Yasunori ANDO. Experimental Study on Mixing Phenomena In Supersonic Flows With Slot Injection. AAIA 91-0016. January 1991.
14. TarunMathurMark Gruber, Kevin Jackson, Jeff Donbar, Wayne Donaldson Thomas Jackson and Fred Billig, Supersonic Combustion Experiments with a Cavity-Based Fuel Injector, *Journal of propulsion and Power*, AIAA, DOI: 10.2514/2.5879. Vol. 17, No. 6, December 2001.
15. ThangaduraiMurugan, Sudipta De and V. Thiagarajan. Validation of Three-Dimensional Simulation of Flow through Hypersonic Air-breathing Engine. *Defence Science Journal*, Vol. 65, No. 4, July 2015, pp. 272-278, DOI: 10.14429/dsj.65.6979. (2015).
16. Wang Lua, b, QianZhansena, b, GaoLiangjie. Numerical study of the combustion field in Dual-cavity Scramjet combustor. *Procedia Engineering* 99 (2015) 313 – 319. (2015)
17. Wu Xianyu, Li Xiaoshan, Ding Meng, Liu Weidong, Wang Zhenguo. Experimental Study on Effects of Fuel Injection on Scramjet Combustor Performance. *Chinese Journal of Aeronautics* 20(2007) 488-494, November 2007.
18. X. Zhang, A. Rona, J. A. Edwards, The effect of trailing edge geometry on cavity flow oscillation driven by a supersonic shear layer, *Aeronaut. J.* (1998) 129–136. (1998).
19. Y. S. Chena, Y. Y. Liana, T. H. Choub, Alfred Laib, J. S. Wub. Mixing Effectiveness Study in Scramjet Combustion. *Procedia Engineering* 67 (2013) 218 – 229. (2013).
20. Zheng Zhong-hua& Le Jia-ling. Parallel Modeling of Three-Dimensional Scramjet Combustor and Comparisons with Experiment's Results. China Aerodynamics Research & Development Center. (2002).

