Computational analysis of novel cavity-based flameholder designs for supersonic combustion engines

Sanchayata Naya1, Sampada Lamb2, Devabrata Sahoo3, N. V. Raghavendra4, S.A.I. Bellary5

1-5Annasaheb Dange College of Engineering and Technology, Ashta, Sangli, Maharashtra, India

E-mail: nayal.sanchayata@gmail.com

Abstract: Computational analysis for novel cavity-based flameholder designs with different fore-wall and aft-wall inclinations has been presented. The flameholder performance has been evaluated based on the following key parameters: pressure distribution, temperature distribution, recirculation zones and drag force. On comparing the different cavity designs, it has been found that both the fore-wall and aft-wall angles affect the flameholder performance. It was observed that an obtuse fore-wall angle gave favourable results numerically. Further studies can analyze cavities with different obtuse fore-wall ramp angles for optimum flameholder performance.

1. Introduction

The ever-increasing demand for faster flight vehicles has led to an active research in the area of propulsion mechanisms. Today, the fastest air-breathing engine is the SCRAMJET engine, which operates in supersonic conditions of Mach number greater than 5. The flow in the combustion chamber is supersonic, which results in extremely short residence time for injection, mixing and combustion of fuel with the incoming air. The high speed of air also affects the stability of the flame in the combustion chamber. Elevated temperatures of the combustor walls pose another technical challenge, which must be addressed to devise a plan for optimum use of heat-resistant materials [1].

Extensive research has been done on different types of flameholders, especially the cavity-based type, where multiple cavity configurations have been analyzed numerically as well as experimentally [2, 3]. A detailed analysis of different cavity configurations along with pylon-aided fuel injection and combustion studies has been done by Gruber et al [4-6], both experimentally and computationally. He concluded that smaller aft-wall ramp angles lead to lower fore-wall pressure, higher drag coefficient and smaller residence time. K H Yu et al [7] performed experimental study of rectangular and inclined cavities with varying aspect ratios (L/D), and found that short, inclined cavities and two tandem cavities give good flameholder performance and volumetric heating. Cavity-based flameholders are often combined with other types of flameholders like the strut-based flameholders [8, 9] and the pylon-based flameholders [5, 10] to develop larger recirculation regions and better fuel-air mixing characteristics for better combustor performance. A S Vishnu et al [11] conducted a detailed computational investigation on the effect of different heat fluxes on the cavity flow physics which was validated experimentally.
It was noted that general research based on cavity-type flameholder involved variation in the aft-wall ramp angle only. We specifically note that research has not been done to evaluate the effect of modification of fore-wall ramp angle on the flameholder performance.

The present work aims to provide parametric analysis of the effect of fore-wall ramp angle on the performance of a cavity-based flameholder. Computational simulations for supersonic flow over cavity configurations with fore-wall angle modifications have been performed. The analysis of results from these simulations can lead to better flameholder designs, and also help in material selection for the combustor walls.

1.1 Cavity-based Flame-holders
A common device to facilitate supersonic combustion is a flameholder, which introduces low-speed eddy-like recirculation zones in the incoming high-speed flow, as shown in Figure 1. These recirculation zones enable efficient mixing of fuel and air, resulting in continuous combustion (and prevent flame-out).

Extensive research has been done to understand the complex flow physics of cavity-based flameholders. This has led to progress in supersonic combustion and development of successful SCRAMJET prototypes.

The geometry of a typical cavity-based flameholder is shown in Figure 2.

The performance of a cavity-based flameholder design is evaluated based on the following broad parameters:

- Higher static pressure in cavity improves flameholder performance
- Higher temperatures in the cavity encourage better mixing of fuel and air, and also flame stability
- Large eddies or recirculation zones result in better mixing
- Low drag force on the cavity is desirable
In the present work, four novel cavity designs with fore-wall ramp angle modifications are proposed, and computational analysis of the above-mentioned parameters are performed.

1.2. Research objectives

Figure 3 depicts the definition of the fore-wall ramp angle $\theta_f$ and aft-wall ramp angle $\theta_a$ of a cavity. To understand the effect of fore-wall and aft-wall ramp angles on cavity performance, different combinations of $\theta_f$ and $\theta_a$ are considered. Computational analysis of supersonic flow over these cavity designs is carried out and results are obtained for the various performance parameters.

![Figure 3. Notations for fore-wall and aft-wall ramp angles of a cavity-based flameholder](image)

2. Methodology

A total of six cavity designs are analyzed computationally. It may be noted that the first two designs with 90° fore-wall ramp angle correspond to the test cases studied in Gruber paper [4], and serve to validate our computational methodology. Remaining cavities are our novel designs with variation in fore-wall ramp angle.

For choosing the fore-wall ramp angle, there were two options: Acute angle and Obtuse angle. For the sake of uniformity, one acute angle and one obtuse angle were chosen. The mean difference from right angle of about 20° was kept constant for both the acute and obtuse fore-wall ramp angles. This resulted in fore-wall ramp angles of 70° and 110° calculated as per figure 3.

These two fore-wall ramp angles were combined with two different aft-wall ramp angles: 90° and 16°, which were taken from the reference of the Gruber paper [4]. This led to the creation of 4 novel cavities design with both acute and obtuse fore-wall ramp angles along with two standard aft-wall ramp angles. A list depicting the cavity geometries is provided in Table 1.

2.1. Computational Setup

Computational Fluid Dynamics (CFD) simulations are performed using ANSYS-Fluent® software, for steady supersonic viscous flow approaching the cavity. It is a two-dimensional, non-reacting, cold-flow investigation wherein only air is introduced in the cavity so we don’t consider the actual combustion of fuel and air and the resulting thermodynamic effects. This is in accordance with our baseline reference paper [4].

For the Initial Flow conditions, the freestream conditions from the baseline paper [4] are used as the test conditions for our present work:

- Freestream Mach number: 3
- Stagnation pressure: 690 KPa
- Stagnation temperature: 300 K
- Working fluid: Air (considered as ideal gas)
Table 1. Cavity designs studied in present work

| Sr. No. | Cavity Nomenclature | Cavity Design | Remarks |
|---------|---------------------|---------------|---------|
| 1       | 90_90               |               | Validation Case |
|         |                     | θf = 90°      |         |
|         |                     | θa = 90°      |         |
| 2       | 90_16               |               | Validation Case |
|         |                     | θf = 90°      |         |
|         |                     | θa = 16°      |         |
| 3       | 70_90               |               | Novel design  |
|         |                     | θf = 70°      |         |
|         |                     | θa = 90°      |         |
| 4       | 70_16               |               | Novel design  |
|         |                     | θf = 70°      |         |
|         |                     | θa = 16°      |         |
| 5       | 110_90              |               | Novel design  |
|         |                     | θf = 110°     |         |
|         |                     | θa = 90°      |         |
| 6       | 110_16              |               | Novel design  |
|         |                     | θf = 110°     |         |
|         |                     | θa = 16°      |         |

2.1.1. Computational mesh
A structured two-dimensional mesh is used as the computational domain, as shown in Figure 4. The height of the computational domain is about 20 times the cavity depth, to simulate far-field conditions at the top boundary. There are 250 x 200 cells in the cavity region and about 150 x 100 cells in the regions above. The smaller cell-size inside the cavity is used to capture the viscous effects of the flow like boundary layer, shear layer, shock formation and eddy formation. To get satisfactory results, y⁺ is less than unity for all cavity grids. Further away from the cavity surface, the viscous effects become less important and hence the cell size is larger for quicker computations.

2.1.2. Boundary conditions
The boundary conditions for the numerical simulation are depicted in Figure 4. The lower part of the computational domain comprises of no-slip stationary wall condition which represents the combustor wall of the SCRAMJET engine. Inlet corresponds to the freestream boundary conditions mentioned in section 2.1. Both Outlet and Farfield boundary conditions correspond to Pressure-Farfield boundary type where the freestream Mach number is set as 3.
2.1.3. Flow solver setup
The flow solver is configured to use an explicit density-based solver, with Green-Gauss cell-based scheme for spatial discretization, and Roe-FDS scheme for the inviscid fluxes. Turbulence model is set to k-ω, as adopted in the reference paper [4]. The desired computational accuracy is configured to second order in both time and space.

![Computational domain](image)

**Figure 4.** Computational domain

3. Results And Discussion

3.1. Validation of the computational methodology
Our computational methodology is first used to obtain results for baseline cavities presented in reference [4]. The results obtained are compared with experimental results presented in the reference [4]. As discussed below, there is a close agreement between the two sets of results. This helps in validating our computational methodology. The same methodology is then used for computational analysis of the novel cavity designs presented in this work.

3.1.1. Normalized Pressure Plots
The baseline reference paper [4] provides the experimental data for the cavity designs 90_90 and 90_16. These designs are part of our parametric study as shown in Table 1. Figures 5 and 6 show the comparison of our present computational results (represented using solid lines) with the experimental results from Gruber et al [4] (represented using circles). The comparison shows a close agreement between the two results, thereby validating our computational method.

3.2 Performance parameters of novel cavity-based flameholder designs
CFD simulations are performed for Mach 3 supersonic flow approaching each of the four novel cavity designs: 70_90, 70_16, 110_90, 110_16; and results are obtained pertaining to the performance parameters – pressure distribution, temperature distribution, recirculation zones and drag force. Details of these results are presented below and discussed to evaluate the performance of each of these designs as a flame holder.

3.2.1 Pressure distribution
The results for normalized pressure distribution for the novel cavity designs are presented below. The pressure is normalized with respect to the inlet static pressure.

Figure 7 (a) shows the normalized pressure distribution for the cavity 70_90. It is seen that there is a slight increase in the pressure after the separation point, indicating a weak shock formation. The aft-
wall corner point shows higher pressure than the rest of the cavity. When compared to the test cavity 90_90, we see more pressure variations with higher magnitude.

Figure 5. Normalized Pressure plot for cavity 90_90

Figure 6. Normalized Pressure plot for cavity 90_16

Figure 7 (b) shows the normalized pressure distribution for the cavity 70_16. It is seen that the pressure decreases after the separation point, indicating a weak expansion wave formation. The aft-wall corner point shows higher pressure than the rest of the cavity. When compared to the test cavity 90_16, not much variation is seen.

Figure 7 (c) shows the normalized pressure distribution for the cavity 110_90. The pressure increase after the separation point is the highest when compared to the other cavities. The aft-wall corner point shows much higher pressure comparatively. When we compare our results for cavity 110_90 with those obtained for all the other cavities in Table 1, a higher-pressure distribution is observed for the novel cavity 110_90. Consequently, this design ranks best amongst the considered cavity designs for pressure distribution. Figure 7 (d) shows the normalized pressure distribution for the cavity 110_16. It is seen that the distribution is similar to that of cavity 70_16 and to the test cavity 90_16 also.

3.2.2 Temperature distribution

The results for normalized temperature distribution for the novel cavity designs are presented below. The temperature is normalized with respect to the inlet temperature.

Figure 8 (a) shows the normalized temperature distribution for the cavity 70_90. The shear layer is completely visible. It is seen that the temperature increases as we travel from free stream towards the cavity vertically downwards. The aft-wall corner point shows increase in temperature along the cavity line. When we compare the results from our computation for the test cavity 90_90, we see a lower temperature distribution for cavity 70_90.

Figures 8 (b) shows the normalized temperature distribution for cavity 70_16. On comparing with the results we obtained for the baseline test cavity 90_16, not much difference is seen, though temperature distribution along the aft-wall is relatively lower.

Figure 8 (c) shows the normalized temperature distribution for the cavity 110_90. The aft-wall corner point shows increase in temperature along the cavity line. The bottom corner of the aft-wall shows higher temperature than the interior of the cavity. The temperature variation also indicates the recirculation zones. When compared to all the remaining cavities, we see lower temperature distribution for 110_90 cavity, making it a better design choice.

Figure 8 (d) shows the normalized temperature distribution for the cavity 110_16. It is seen that the distribution is similar to that of cavity 70_16 and to the test cavity 90_16 also.
3.2.3 Recirculation zones in the cavity

Figure 9 depicts the results for the recirculation zones in the novel cavity designs. We see that the novel cavity design 110_90 shows stronger recirculation zones with contours of higher velocities, as compared to other cavities. This makes the cavity 110_90 a suitable candidate for flameholder design.

![Normalized pressure contours](image)

Figure 7: Normalized pressure contours for the novel cavity designs

3.2.4 Drag force on the cavity

The drag force obtained from ANSYS-Fluent® computations is normalized with respect to the drag force we had obtained for the baseline 90_90 cavity. Results from the computations show that cavity design 70_90 experiences the lowest drag amongst other cavities, making it a better cavity design with respect to the cavity drag. Results for the normalized drag force for the novel cavity designs are discussed in the next section 3.2.5.

3.2.5 Discussion and comparison of results

Based on each of the performance parameters, the four novel cavity designs are ranked as in Table 2. Cavity design which performs best for a parameter get a score of 4, while the one with relatively poorest performance get a score of 1.

The desirable performance parameters are:
- High pressure distribution
- High temperature distribution
- Large eddies or recirculation zones
- Low drag force on the cavity
Based on the relative performance scores in Table 2 and the discussions above, we see that cavity design 110.90 offers the best overall performance as a flameholder. In an incoming supersonic flow, this cavity develops good pressure distribution which promotes efficient combustion. This cavity lags in terms of high temperature distribution, which may slightly reduce fuel-air mixing. However, the strong recirculation zones in the cavity should bring about efficient mixing of fuel and air. The
relatively lower temperatures at the cavity wall can be helpful in reducing damage to the material of the wall. Our finding of best overall performance with design 110_90 also corroborates with that of Gruber et al [4], who concluded that 90° for the aft-wall ramp angle works best for flameholder performance.

Table 2: Relative performance scores of the novel cavity designs

| Cavity  | Performance Parameters as a Flameholder |
|---------|----------------------------------------|
|         | Pressure Distribution | Temperature Distribution | Recirculation Zones | Normalized Drag Force | Total Score |
| 70_90   | 3                        | 1                        | 2                   | 4                     | 10          |
| 70_16   | 1                        | 4                        | 3                   | 2                     | 10          |
| 110_90  | 4                        | 2                        | 4                   | 1                     | 11          |
| 110_16  | 2                        | 3                        | 1                   | 3                     | 9           |

4. Conclusion
In this paper, computational analysis is done to qualitatively and quantitatively assess the performance of novel cavity designs based on key performance parameters of pressure distribution, temperature distribution, recirculation zones and drag force. The following conclusions are drawn:

- The fore-wall ramp angle of a cavity can affect its performance as a flameholder. Specifically, an obtuse fore-wall angle can result in stronger recirculation zones and efficient mixing of fuel and air.
- With respect to the aft-wall ramp angle, a setting of 90° works best for flameholder performance.
- Cavities with different obtuse fore-wall ramp angles can be analyzed for optimum flameholder performance.

References
[1] Yang W, Fu J, Ma X and Xing R 2018 IOP Conference Series: Materials Science and Engineering 408 150.
[2] Ben-Yakar A and Hanson R K 2001 Journal of Propulsion and Power 17 869–877.
[3] Santhosh K Gugulothu 2019 Heat Transfer: Asian Research 48 3657-3681.
[4] Gruber M R, Baurle R A, Mathur T and Hsu K Y 2001 Jr. Propulsion and Power 17 146–153.
[5] Gruber M R, Carter C D, Montes D R, Haubelt L C, King P I and Hsu K Y 2008 Journal of Propulsion and Power 24 460–470.
[6] Gruber M R, Donbar J M, Carter C D and Hsu K Y 2004 Journal of Propulsion and Power 20 769–777.
[7] Yu K H, Wilson K J and Schadow K C 2001 Journal of Propulsion and Power 17 1287–1295.
[8] Hsu K Y, Carter C D, Gruber M R, Barhorst T, Smith S 2010 Jr. Propulsion and Power 26 6.
[9] Choubey G, Devarajan Y, Huang W, Kulmani Mehar, Tiwari M and Pandey K M 2019 International Journal of Hydrogen Energy 44 13895-13909.
[10] Freeborn A B, King P I and Gruber M R 2009 Journal of Propulsion and Power 25(3).
[11] Vishnu A S, Aravind G P, Deepu M and Sadanandan 2019 International Journal of Heat and Mass Transfer 141 1140-1151.