Numerical Simulations of Axisymmetric Aft Wall Angle Cavity in Supersonic Combustion Ramjets

Naresh Relangi1, Divyasri Garimella2, University of Rome “La Sapienza”, Rome, Italy.

Jayaraman K3, Jayaku mar Venkatesan4, Veltech University,

Jeyakumar S5, Kalasalingam University,

and

A. Ingenito6, University of Rome “La Sapienza”, Rome, Italy.

Abstract

Scramjet engines are the most favourable candidates for future air breathing propulsive system and are a promising option for high-speed flights. In scramjet engines, air-fuel mixing, flame stabilization and combustion efficiency are still a challenge. Understanding of the phenomena occurring inside the supersonic combustion chamber is therefore mandatory: that is why numerical simulations may be of help toward future SCRJ combustor design. The purpose of this paper is to investigate benefits of axisymmetric cavity with varying aft wall angle. In fact, an optimized cavity is a promising solution to stabilize the flame in supersonic flows avoiding prohibitive total pressure losses, since this enable the generation of a recirculation zone in the cavity, where the flame anchors. The objective of this paper is to numerically examine the impact of diverse toward the back-divider points on blending, pressure losses and combustion efficiency. 3D RANS simulations have been performed and compared with cold stream test data. The result of this study is calculated based on static pressure and stagnation pressure losses. The validation of results with available experimental data shows good agreement.

Results shows that cavities with different toward the back-divider points diminish the total pressure loss and move forward blending capabilities by making distribution zones.

Nomenclature

D depth of the cavity
Introduction

In the last years, hypersonic flights reached the highest attention from both the commercial and the military point of view. In fact, the consolidation of their technology is an opportunity or a threat if only few countries accomplish this knowhow. The chance of flying at hypersonic speeds lays in the possibility to reach the earth antipodal distance in about two hours when flying at \( M=8 \). Also, the high maneuverability and a not ballistic mission profile, make them impossible to be detected by the current radar systems and therefore a future threat.

In order to make this technology ready for applications, some issues must be addressed. In fact, the development of scramjet engines poses considerable challenges and it demands multidisciplinary design, analysis, modeling, simulations and system optimization. Supersonic combustion, which plays a key role in the scramjet technology, was already demonstrated in the 1960s. Subsequently NASA developed considerable expertise in airframe integration and combustion testing facility in the 1970s and 1980s. The critical component of the scramjet engine is the combustor which involves turbulent mixing, shock interaction, holding of flame and heat release in supersonic flow.

At supersonic flight speeds, the residence time of the air is on the order of 1 millisecond. The fuel must be mixed with air and burned completely within such a short time. The flow field within the combustor of scramjet engine is very complex and poses a considerable challenge in design and development of a supersonic combustor with an optimized geometry. Such combustor shall promote sufficient mixing of the fuel-air and flame stabilization avoiding excessive pressure losses and consequently penalties in thrust efficiency. The design of combustor, in fact, must account for a rapid fuel air mixing, the minimization of total pressure loss, and a high combustion efficiency.

Several studies have been carried out worldwide, and various concepts have been suggested for scramjet combustor configurations to overcome the limitations given by the short flow residence time. In fact, mixing of air and fuel at high velocity and low residence time in scramjet engine combustor is possible whereas streamline vorticity can develop. As also shown by Nishioka (2006) and Swithenbank (2001) streamwise vorticity enhances supersonic mixing down to the molecular level and leads to efficient combustion. In fact, theoretical work by Ingenito (2010) showed that mixing can be accelerated by generating and increasing streamwise vorticity. In [], it has been proven that it is possible by maximizing the baroclinic term. This can be obtained by favoring the formation of pressure and density gradients, e.g., by means of different injector designs, such as transverse, swirl, strut, wall or swept ramp injectors, and cavities.

Various flush mounted fuel injection and flame holding schemes like cavities, struts, pylons etc., have been proposed by many researchers. The use of cavities is a special perspective as flame holder and fuel injector. The design of cavities was firstly completed and applied by CIAM (Central Institute of Aviation Motors) in Moscow in a joint Russian/French dual mode scramjet flight-test. The existence of recirculation area inside the cavities enhances the combustible mixture's residence time and thus cavities are preferable candidates for flame holding. The flow field inside a cavity is characterized by recirculating flow that increases the residence time of the fluid entering the cavity. Because the drag associated with flow separation is much
less over a cavity than for a bluff-body, a cavity inside a combustor makes a stable flame holder with relatively little pressure drop. A rectangular cavity driven by a free shear layer provides a well defined configuration to study the flow separation and reattachment. Basically, there are two types of cavity flow: open or closed. The open cavity flow occurs for length-to-depth ratio, $L/D < 10$. In this case the shear layer formed at the separation corner spans the entire cavity length and reattaches somewhere along the cavity back face. For $L/D > 10$, the shear layer is unable to span the entire length of the cavity and reattaches on the cavity floor, which is the closed cavity flow. Therefore, the closed cavities are characterized by a larger drag coefficient compared with open cavities so that the latter is more desirable in a scramjet combustor.

**Literature Review**

The use of cavities for stabilization of flame in a solid fuel supersonic combustor was mainly performed by Ben Yaker et al. In his work, he revealed sustained combustion and self-ignition of poly methyl-methacrylate (PMMA) for supersonic inlet conditions. Yu and Schadow experimentally demonstrated the use of cavities to excite supersonic jets. The effect of three-dimensional cavities in supersonic flow field discloses that cavity width plays an important role in the mixing augmentation by turbulence generation. The cavities with inclined aft wall produce high drag due to stronger recompression of shear layers at the rear wall. The performance of such cavities enhances fuel–air mixing and flame holding, which is mainly dependent on the flow stability above the cavity region. The effect of backpressure and cavity length-to-depth ratio ($L/D$) on the scramjet combustor flow field for non-reacting flow was computationally studied by Huang, et al. Again Kim, et al performed computational analysis of combustion enhancement using cavity flame holder to investigate the effects of aft wall angle, cavity length, depth on pressure loss and combustion efficiency. To improve the fuel and air mixing between in supersonic flows, transverse injection from a wall orifice is one of the easiest configurations. The drawback of this configuration lays in high pressure losses due to the shock waves arising close to the fuel injection. Cavities with specific geometry may enhance mixing and flame anchoring in scramjet combustors, allowing an active recirculation and stable combustion within the cavity avoiding prohibitive pressure losses.

**Geometry Details**

The schematic view of the supersonic combustor is shown in Figure 1. The combustor may be divided into three zone, the first one upstream the cavity (REGION 1), the second one that corresponds to the cavity region (REGION 2), and the third one downstream the cavity (REGION 3). The axisymmetric combustor has an inlet diameter of 26mm and a length of 95mm. The cavities are incorporated at 20 mm from the inlet. Cavities are placed inside the combustor at 20 mm from the inlet. The cavities are axisymmetric and open type. The depth of the cavity is kept constant and length varies with the aft ramp angle. The aft wall of the cavity is inclined with two consecutive angles. Three primary and secondary angles, totally nine sets of varying aft wall angled cavities are chosen for the study and compared with the rectangular cavity. The details of the cavity configuration are tabulated in Table-1. The primary aft wall angle ($\Theta_1$) starts from the bottom of the cavity and the secondary aft wall angle ($\Theta_2$) starts from the half the depth of the cavity. The aft wall of the cavity is inclined with two angles. The primary angle ($\Theta_1$) begins from the base of the cavity and the secondary angle ($\Theta_2$) from half the depth of the cavity. The angles of 90, 60 and 30 degrees are used for primary angle and 45, 30 and 15 degrees for secondary aft wall angles.
Figure 1: Schematic view of supersonic combustor
A schematic diagram of a typical cavity that has been investigated is shown in Figure 2

![Figure 2 Schematic view of the cavity](image)

**Table 1: Geometrical detail of Cavity Configurations**

<table>
<thead>
<tr>
<th>Notation</th>
<th>Cavity, L/D</th>
<th>Effective, e/D</th>
<th>Primary angle, Θ₁ (degree)</th>
<th>Secondary angle, Θ₂ (degree)</th>
</tr>
</thead>
<tbody>
<tr>
<td>90,90</td>
<td>4</td>
<td>4.0</td>
<td>90</td>
<td>90</td>
</tr>
<tr>
<td>90,45</td>
<td>4</td>
<td>4.4</td>
<td>90</td>
<td>45</td>
</tr>
<tr>
<td>90,30</td>
<td>4</td>
<td>4.9</td>
<td>90</td>
<td>30</td>
</tr>
<tr>
<td>90,15</td>
<td>4</td>
<td>6.0</td>
<td>90</td>
<td>15</td>
</tr>
<tr>
<td>60,45</td>
<td>4.3</td>
<td>4.75</td>
<td>60</td>
<td>45</td>
</tr>
<tr>
<td>60,30</td>
<td>4.3</td>
<td>5.1</td>
<td>60</td>
<td>30</td>
</tr>
<tr>
<td>60,15</td>
<td>4.3</td>
<td>6.0</td>
<td>60</td>
<td>15</td>
</tr>
<tr>
<td>30,45</td>
<td>4.9</td>
<td>5.3</td>
<td>30</td>
<td>45</td>
</tr>
<tr>
<td>30,30</td>
<td>4.9</td>
<td>5.7</td>
<td>30</td>
<td>30</td>
</tr>
<tr>
<td>30,15</td>
<td>4.9</td>
<td>6.65</td>
<td>30</td>
<td>15</td>
</tr>
</tbody>
</table>

**Governing Equations**

The balance equations for the conserved Reynolds–Favre-averaged mass flow rate, momentum, and total enthalpy (including sensible, chemical, kinetic, and turbulent kinetic energy) in conservative form reads:

\[
\frac{\partial (\bar{\rho} \bar{u}_i)}{\partial x_i} = 0
\]

\[
\frac{\partial (\bar{\rho} \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} (\mu + \mu_t) \frac{\partial \bar{u}_i}{\partial x_j} \delta_{ij} - \frac{2}{3} \bar{p} k \delta_{ij} \right] + \bar{f}_i
\]

\[
\frac{\partial (\bar{\rho} \bar{u}_j \bar{Y}_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{\mu}{\sigma} \frac{\partial \bar{Y}_i}{\partial x_j} \right) + \bar{w}_i
\]

Assuming the Prandtl model, the turbulent viscosity reads:

\[
\mu_t = C_{\mu} \bar{\rho} \bar{u}_i \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = C_{\mu} \bar{\rho} \frac{\bar{k}^2}{\bar{E}}
\]

where \( C_{\mu} \) is a constant assumed equal to 0.09.
The turbulent viscosity $\mu_t$, the turbulent kinetic energy $k$, and the Reynolds stress $R_{ij}$ are computed with an eddy-viscosity $k-\varepsilon$ turbulent model. The turbulent kinetic energy $k$ and its rate of dissipation $\varepsilon$ are calculated by solving:

$$\frac{\partial (\bar{\rho} \bar{u}_k)}{\partial x_k} = \frac{\partial}{\partial x_k} \left[ \left( \frac{\mu}{\sigma} + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_k} \right] + \mu_t \left( \frac{\partial \bar{u}_i}{\partial x_k} + \frac{\partial \bar{u}_k}{\partial x_i} \right) \frac{\partial \bar{u}_i}{\partial x_k} - \frac{2}{3} \left( \mu_t \frac{\partial \bar{u}_i}{\partial x_k} + \rho \frac{\partial \bar{k}}{\partial x_k} \right) \frac{\partial \bar{u}_i}{\partial x_k} - \rho \frac{\partial \bar{e}}{\partial x_k}$$

$$\frac{\partial (\bar{\rho} \bar{u}_i)}{\partial x_k} = \frac{\partial}{\partial x_k} \left[ \left( \frac{\mu}{\sigma} + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \bar{e}}{\partial x_k} \right] + C_1 \frac{\bar{k}}{k} \mu_t \left( \frac{\partial \bar{u}_i}{\partial x_k} + \frac{\partial \bar{u}_k}{\partial x_i} \right) \frac{\partial \bar{u}_i}{\partial x_k} - \frac{2}{3} \left( \mu_t \frac{\partial \bar{u}_i}{\partial x_k} + \rho \frac{\partial \bar{k}}{\partial x_k} \right) \frac{\partial \bar{u}_i}{\partial x_k} - C_2 \rho \frac{\partial \bar{e}}{\partial x_k}$$

where $\sigma_k$ and $\sigma_\varepsilon$ are the turbulent Prandtl and Schmidt numbers, $C_1$ and $C_2$ are calibration constants. The total enthalpy $h_\sim$ involves both sensible and chemical enthalpy. The properties $m$, $D_k$, and $c_p$ are computed using mixing rules, thus accounting for changes temperature. The turbulent Prandtl number $Pr_t$ and the turbulent kinetic-energy Schmidt number $k$ are assumed constant.

**CFD Simulation**

In order to investigate the effect of the cavity over the combustion efficiency, 3-D RANS simulations have been simulated by means of the FLUENT code using double precision. In RANS simulations, turbulence effects have been accounted by means of the standard $k-\varepsilon$ model. A coupled implicit solver first- and second-order upwind numerical schemes have been implemented to solve the equations. The geometry is created as multi domain with interface. A monitor is placed and initialized from the air inlet. Adiabatic wall have been selected for solid domain. Ideal gas condition is assumed for air. Combustor inlet conditions are reported in Table 2.

**Inlet Boundary Conditions**

<table>
<thead>
<tr>
<th>Total Pressure</th>
<th>0.7MPa</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total Temperature</td>
<td>305K</td>
</tr>
<tr>
<td>Mach number</td>
<td>1.8</td>
</tr>
</tbody>
</table>

**CFD Results and Validation**

Comprehensive numerical simulations have been carried out to understand the influence of the flow characteristics of scramjet combustors with different geometrical shapes of its cavity. The performance of the geometries is calculated in terms of static pressure and stagnation pressure loss and compared with experimental values. The contours of static pressure and velocity are as follows. The wall static pressures, density, total pressure and Mach number for various cavity configurations under identical operating conditions are presented in Figure 3- Figure 22. In Figure 3, the density and Mach number flow fields for the configuration with $\Theta_1=90^\circ$ and $\Theta_2=90^\circ$ are shown. Due to the geometry of the combustor, a normal shock occurs upstream the cavity. Normal shocks are responsible for high total pressure losses and therefore, for the combustor efficiency decrease (see Figure 4).
Due to the normal shock, the mach number decreases to subsonic velocities. Upstream cavities, the flow is therefore subsonic: an increase in the cross area due to the cavity, increases the static pressure. Downstream the cavity, due to the rapid cross area reduction, the flow is chocked at the throat and the velocity increases to supersonic values.

Keeping the $\theta_1$ constant and reducing the aft wall angle ($\theta_2$) from 90 to 45 to 30 and 15, it can be observed that static pressure in the REGION2 is decreasing simultaneously from 2 MPa to 1.4 MPa. The higher pressure in the chamber is responsible for the position of the normal shock in the combustion chamber, upstream the cavity. In fact, as the pressure increases, the normal shock is moved upstream, this increasing the total pressure losses. Decreasing the aft wall angle $\theta_2$ to 45°, the normal shock upstream the cavity moves on the right, at the throat. Still decreasing $\theta_2$ to 30°, the shock waves is oblique, and the Mach number keeps supersonic in region 2. In fact, the Mach number in the cavity region increases by decreasing the aft wall angle from 0.7 to 1.1. This effect is critical for the total pressure losses. In fact, as the secondary aft angle decreases, the total pressure losses decrease as well. It has to be noted that, at the combustor exit, the Mach number is always supersonic.
Figure 6 Static Pressure and Total Pressure contour for the configuration with $\Theta_1=90^\circ$ and $\Theta_2=45^\circ$

Figure 7 Density and Mach number contour for the configuration with $\Theta_1=90^\circ$ and $\Theta_2=30^\circ$

Figure 8 Static Pressure and Total Pressure contour for the configuration with $\Theta_1=90^\circ$ and $\Theta_2=30^\circ$

Figure 9 Density and Mach number contour for the configuration with $\Theta_1=90^\circ$ and $\Theta_2=15^\circ$
Decreasing $\Theta_1$ to 60°, the same behavior is observed with respect to the secondary angle variation on the pressure, mach number and total pressure in the region 2. In fact, decreasing the secondary angle will decrease the pressure and will increase the velocities from subsonic to supersonic in the cavity region of the combustion chamber.

When $\Theta_2<30^\circ$, the flow is supersonic in the cavity, therefore the pressure increases and the Mach number decreases in the convergent part of the cavity and increases in the region 3.
Figure 13 Density and Mach number contour for the configuration with $\Theta_1=60^\circ$ and $\Theta_2=30^\circ$

Figure 14 Static Pressure and Total Pressure contour for the configuration with $\Theta_1=60^\circ$ and $\Theta_2=30^\circ$

Figure 15 Density and Mach number contour for the configuration with $\Theta_1=60^\circ$ and $\Theta_2=15^\circ$

Figure 16 Static Pressure and Total Pressure contour for the configuration with $\Theta_1=60^\circ$ and $\Theta_2=15^\circ$
Figure 17: Density and Mach number contour for the configuration with $\Theta_1=30^\circ$ and $\Theta_2=15^\circ$.

Figure 18: Static Pressure and Total Pressure contour for the configuration with $\Theta_1=30^\circ$ and $\Theta_2=15^\circ$.

Figure 19: Density and Mach number contour for the configuration with $\Theta_1=30^\circ$ and $\Theta_2=30^\circ$.

Figure 20: Static Pressure and Total Pressure contour for the configuration with $\Theta_1=30^\circ$ and $\Theta_2=30^\circ$. 
The effect of the cavity on the static pressure distribution along the combustor has also been calculated and compared with experimental data (see Figure 23- Figure 25). This figure shows that decreasing the secondary angle will increase the static pressure at the exit, keeping almost a constant value for the combustor exit Mach number, as shown in the previous figures. This is due to the lower total pressure losses in the last configuration where no normal shocks waves occur. Due to the convergent section in the cavity region, pressure increases establishing that the flow is supersonic.
Decreasing the primary angle to 60° and keeping the secondary angle constant, results in marginal variation in the pressure value at the exit.

![Figure 24 Static and inlet Pressure Ratio along the axial distance for the configuration with Θ1=60°](image)

Figure 24 Static and inlet Pressure Ratio along the axial distance for the configuration with Θ1=60°

Decreasing the primary angle to 30° and keeping the secondary angle constant, results again in marginal variation in the pressure value at the exit.

![Figure 25 Static and inlet Pressure Ratio along the axial distance for the configuration with Θ1=30°](image)

Figure 25 Static and inlet Pressure Ratio along the axial distance for the configuration with Θ1=30°

Finally, the effect of the cavity on the total pressure calculated as $P_t/P = (1 + \frac{\gamma-1}{2}M^2)^{\frac{\gamma}{\gamma-1}}$ has been investigated (P is static Pressure, $P_t$ is Stagnation pressure and M is Mach number). Generally, the shock waves generated due to the stronger reattachment of shear layers at the aft wall of the cavity interferes with the core flow causing an increase in the stagnation pressure loss. The stagnation pressure loss is calculated based on the difference in the stagnation pressures at the inlet of combustor to outlet of combustor along axial direction.

Figure 26 shows the stagnation pressure loss across the combustor for varying aft ramp angles of the cavity. The reduction in cavity primary aft wall angle decreases the stagnation pressure loss. However, the stagnation pressure loss is minimum for $\Theta_2=30$. For $\Theta_2=15$, stagnation pressure loss is almost equal irrespective of the cavity aft wall angle $\Theta_1$, indicating that decreasing the secondary aft wall angle below 45 provides a stable flow field in the cavity region. A secondary cavity aft wall angle higher than 45 produces a stagnation pressure higher than 18%. The increase in stagnation pressure loss is due to the shock waves strength in the combustor.
Figure 26 Stagnation Pressure loss % for the different configurations

The numerical simulation results are validated with the experimental tests case performed by the authors. The experimental values and the numerical analysis can be found in same plot for each geometry. The computed wall static pressure distributions have been compared with experimental data. It is assured that the numerical results are in good agreements with the experimental outcome in the operating conditions. Therefore, the present CFD technique is appropriate to analyze the flow fields in axisymmetric cavity combustor. Small discrepancies are at the combustor exit due to the model limitations, axisymmetric assumption and boundary conditions.

Conclusions:

An investigation of axisymmetric cavity in scramjet combustors was simulated in order to investigate cavity influences on flame and combustion stabilization. CFD analyses of cavity based cold flow are carried out at Supersonic flow. Results agree reasonably well qualitatively and quantitatively with experiments.

The wall static pressure distribution shows that a reduction in the secondary aft wall angle below 45 increases the static pressure at the exit and also decreases the total pressure losses. Rectangular cavity provides a higher stagnation pressure loss than the aft wall angle cavity. Decrease in aft wall angle reduces the stagnation pressure loss indicating that a stable flow field is accomplished in the combustor. The cavity is a good choice to stabilize the flame in the hypersonic flow, and it generates a recirculation zone in the scramjet combustor. Further, if its geometry can be designed properly, it can act as an igniter for the fuel combustion, but the material of the cavity flame holder should be considered for operating at those high temperatures. Since promising results have been obtained, further work will be performed to analyze the effect of a secondary fuel injection and combustion in a cavity based supersonic combustion ramjet.

Introduction: