Numerical investigation of fuel mixing with upstream crescent cavities in a scramjet combustor

Tim Roos, Adrian Pudsey, Mathew Bricalli, Hideaki Ogawa

A R T I C L E   I N F O

Keywords:
Upstream cavity
Scramjet
Mixing enhancement
Computational fluid dynamics
Crescent cavity

A B S T R A C T

Cavities are commonly employed in scramjet combustors for flameholding and mixing enhancement, but the mechanism used to enhance mixing is absent at high supersonic Mach numbers, limiting their operational scope. The present study investigates the ability of crescent-shaped cavities placed upstream of a fuel injector to enhance mixing through vorticity generation, a mixing enhancement mechanism that is also effective at high supersonic Mach numbers. The mixing performance of five crescent cavity designs, two of which incorporate hybrid fuelling, is investigated using unsteady Reynolds-Averaged Navier–Stokes (URANS) computations of a chemically frozen flow with hydrogen as the fuel. It is found that the crescent cavities enhance mixing by up to 22.6% without the hybrid fuelling arrangement and by up to 90.1% with the hybrid fuelling arrangement. While vertical jet penetration is lower for all cavity cases, lateral penetration is higher and the cavity cases incur no or negligible total pressure loss compared to the baseline at the domain outflow, within the margin of error. Wall drag is also lower than in the baseline for some cavity cases. The primary mechanism driving mixing is found to be enhanced streamwise vorticity in the vicinity of the cavity, caused by the cavity vortex leaving the cavity and wrapping around the injector. The cavity flowfields are also found to be oscillatory in nature, although the oscillations are lateral and the harmonic frequencies are much lower than those of the longitudinal oscillations characteristic of conventional cavity flow. The mechanisms driving these oscillations are discussed, as are the flowfields for the best performing cavity cases. Several flowfield features of the crescent cavities are also highlighted and discussed, demonstrating how the hybrid injection cavity cases enhance mixing.

1. Introduction

Mixing augmentation is one of the primary research topics for practical supersonic combustion ramjet (scramjet) systems [1–12]. Since scramjet combustor residence times are on the order of 1 ms it is imperative that fuel is mixed quickly and efficiently, in order to maximise heat release inside the combustor. This is especially important when fuels with longer ignition delay times are used [13–15], necessitating the use of mixing enhancement techniques.

Common approaches to mixing enhancement include vorticity generation and enhancement, inlet injection (to increase mixing length) and employing a range of injector geometries and fuelling strategies [2–4,9,16–23] as summarised recently by Huang [23]. Vorticity generation in particular is known to be promising, due to the relatively large gains in mixing it can provide [4,23].

Vorticity generation is most commonly achieved using ramps, struts or vortex generators [23–51], with optional injection of fuel from these structures. The main downside of using ramps or struts is that they protrude into the main combustor flow, incurring (significant) losses through shockwaves and base drag. A less intrusive option is the aero-ramp, where fuel is injected through a number of carefully arranged fuel injectors, resulting in a ramp-like aerodynamic displacement in the combustor flow [25–27]. In the nearfield the aero-ramp has been shown to perform better in terms of mixing than a physical ramp, however in the far-field the physical ramp features better mixing [26]. A dependence on injection pressure was also observed.

Most practical scramjet systems will likely use hydrocarbon fuels because of their high energy density [13]. The ignition delay time of hydrocarbon fuels can be significantly longer than the combustor residence time however, requiring the use of flameholding and ignition enhancement devices in the combustor. These devices, commonly wall-mounted cavities [32–35], also incur total pressure losses, further increasing combustor drag. Ideally, a single device would provide both mixing enhancement and flameholding to limit combustor losses while still achieving viable combustor performance.

* Corresponding author.
E-mail address: tim.roos@rmit.edu.au (T. Roos).

https://doi.org/10.1016/j.actaastro.2020.08.022
Received 23 June 2020; Received in revised form 24 July 2020; Accepted 15 August 2020
Available online 24 August 2020
0094-5765/© 2020 IAA. Published by Elsevier Ltd. All rights reserved.
Previous numerical and experimental work has shown that upstream cavities have the potential to be such a device; Handa et al. [36] qualitatively showed that mixing was enhanced when a three-dimensional cavity was placed upstream of a set of injectors in a $M = 1.58$ freestream, through the induction of what was termed a “secondary flow” that increased streamwise vorticity. Huang et al. [37] later performed a numerical study on this cavity geometry and an improved version of the design and confirmed that the cavity cases offer better mixing and penetration than the corresponding flat plate case.

An earlier numerical study by these authors [38] showed that mixing was enhanced by up to 9% relative to a flat plate case when an $L/D = 15$ rectangular cavity was placed 0.5 mm upstream of a circular injector in a Mach 4.5 freestream, demonstrating that upstream cavities could be enhance mixing across a range of Mach numbers.

In the latter study the primary mixing enhancement mechanism was observed to be vorticity enhancement through shielding of the fuel jet, increasing baroclinic torque production and the amount of fluid exposed to this vorticity. Reacting flow simulations of these configurations also showed that combustion was enhanced by up to 8% [39], demonstrating that the upstream cavity can improve both mixing and combustion performance.

The present work attempts to improve upon the upstream cavity design by utilising the vorticity contained inside the cavity. When a cavity is exposed to a supersonic crossflow a subsonic region of recirculating flow develops inside the cavity, Fig. 1. Based on the cavity aspect ratio $L/D$ the cavity shear layer will either span the cavity and create a single recirculation region (open cavity flow, $2−3 < L/D < 12$) or attach to the cavity floor before separating and attaching to the cavity aft wall, creating two recirculation regions (closed cavity flow, $L/D > 13$) [40]. A previous study of the upstream cavity flowfield showed that for upstream cavity arrangements open cavity flow can exist at $L/D > 13$ however [41].

If the recirculation is contained inside the cavity and away from fuel injectors it does not significantly contribute to freestream mixing. Alternatively, fuel can be injected directly into the cavity [35,38] to achieve mixing inside the cavity however direct fuelling has other significant downsides; fuel penetration into the freestream is limited and a very rich mixture can form inside the cavity, limiting its effectiveness as a flameholder [42,43].

The purpose of the current study is to attempt to improve the upstream cavity design and take advantage of the vorticity contained inside the cavity to enhance freestream mixing while limiting the negative impacts on overall combustor performance. Specifically, a number of small, three dimensional (as opposed to spanwise infinite) crescent cavities will be studied, that have been designed to introduce the vorticity contained in the cavity into the main combustor flow. These cavity designs have the potential use the cavity vorticity to enhance freestream mixing, while maintaining freestream penetration and the flameholding ability of the cavity and limiting drag penalties.

2. Numerical approach

2.1. Flow solver

The commercial code CFD++ was used for all simulations in this work [44]. The code solves the steady or unsteady compressible Reynolds-Averaged Navier–Stokes equations within a unified grid framework, facilitating unified treatment of a number of cell types in both fully structured, unstructured or hybrid grid topologies.

The code include a wide range of turbulence models than can be used both with and without wall models and incorporates multispecies finite rate chemistry. Multi-dimensional Total Variation Diminishing (TVD) polynomials are used to discretise inviscid fluxes and cell interface fluxes are calculated using a compressible Riemann solver.

CFD++ has previously been validated for various types of supersonic flow, including transverse injection and supersonic flow over a ramp, [45–48] and for cavity flow more specifically [45], including in a previous study of the rectangular upstream cavity flowfield by the present authors [41].

All calculations in this study are set up with the unsteady RANS (URANS) equations, using an unsteady implicit (backward Euler) numerical scheme that is second order accurate in both time and space.

The global time step was set to $Δt = 1.5 \times 10^{-6}$ s to capture any large-scale cavity oscillations inside the cavity, if present. In conventional open cavity flow with freestream Mach numbers below $M \approx 3.5$ there are self-sustained longitudinal oscillations inside the cavity, the frequency of which is commonly predicted using the modified Rossiter formula [49]. There is experimental evidence that at $M > 3.5$ the cavity acoustics are more accurately approximated by a closed-box acoustics model however, as discussed in [50]. While it is not clear if the these models hold for complex cavity geometries placed upstream of a fuel injector, as studied in this work, it is the best available approximation of the harmonic frequencies that could be expected inside the cavity.

For the conditions listed in Section 2.3 and a cavity with length $L = 6 \text{mm}$ the frequencies of the first three cavity oscillation modes are calculated to be 77.6, 181.1 and 284.7 kHz using the modified Rossiter equation and 69.6, 139.2 and 208.9 kHz using the closed-box acoustics model. These modes are sampled at least 2–14 times using a time step of $1.5 \times 10^{-6}$ s, which is considered sufficient for this study. The temporal stability of the flowfield and the validity of using this time step is discussed in more detail in Section 3.3.

The dual-time stepping scheme that is available within CFD++ to accelerate convergence was used for all time steps. The simulations were conducted from a set of initial conditions that correspond to a jet about to issue into the combustor freestream and the total simulation time was $3 \times 10^{-4}$ s. This is in excess of the analytically predicted establishment time for self-oscillation of $r = 200D/U_∞ = 1.95 \times 10^{-4}$ s, calculated using the relation by Zhang et al. [51] for a cavity with depth $D = 2.4 \text{mm}$ and $M = 4.33$.

The present study only considered chemically frozen flow, so reactions were not activated. All calculations were set up using the 13 species, 33 reactions H₂-air reaction mechanism by Jachimowski [52] however to enable reacting flow simulation in future investigations. Turbulence was modelled using Menter’s 2003 two-equation shear stress transport (SST) model [53,54], which blends from the $k−\omega$ model near the wall to the $k−\epsilon$ model in the freestream. This model has been used in preceding (upstream) cavity studies [11,41,55,56] and was used here because it handles adverse pressure gradients relatively well. The models is also less sensitive to specified freestream turbulence levels than the $k−\omega$ model and has been shown to have good agreement with experimental results in mixing layers.
For the hybrid injection cases a δ value of 3.1 was observed in a small region. For the hybrid injection cases a γ value of 3.1 was observed in a small region. For the hybrid injection cases a γ value of 3.1 was observed in a small region.

2.2. Geometry and mesh

Fig. 2 schematically shows the geometry that was modelled in this study. The geometry comprises a flat plate with a δ = 2 mm circular injector and a crescent-shaped cavity located directly upstream of this injector. The flow direction is left to right. The domain length, height and width were set to 130 mm, 30 mm and 60 mm, respectively, to ensure that the fuel plume did not touch the lateral or upper boundaries. The domain inflow plane was located 30 mm upstream of the injector centre and the origin of the domain was located in the injector centre.

The cavity configurations that were examined in this study all had the same basic shape that is shown in Fig. 2; on the cavity centreline the cavity has a length L and a depth D, with the cavity aft wall located 0.5 mm upstream of the injector leading edge. Away from the symmetry plane the cavity depth gradually decreases, until the cavity floor meets the combustor wall at x = 0 mm. Both the front wall and the aft wall trace a circular path when viewed from above, i.e. the distance between the cavity wall and the injector centre is the same at x = 0 mm and z = 0 mm. Three cavity length and depth combinations were examined in this study, summarised in Table 1. The cavities with L = 3d were sized to be approximately twice the length of the separation region upstream of an under-expanded jet injected into a crossflow, whereas cavity depth was based on previous work and selected to maintain L/D > 2. The naming convention used for the cavity cases consists of (1) the cavity length, normalised by injector diameter and (2) the cavity depth normalised by injector diameter. The cases in Table 1 are from here on referred to as the single injector cases.

Two hybrid fuelling arrangements were also studied for case 3LD1.2, where fuel is injected both inside and outside the cavity [3]. In the arrangements studied here a δ = 0.5 mm injector was added on the cavity centreline on either the aft (arrangement 1) or front (arrangement 2) cavity wall, as shown in Fig. 3.

Finally, all cavity cases were compared to a no-cavity flat plate injection case, hereafter referred to as the baseline. To obtain the baseline geometry the cavity was simply removed in Fig. 2.

The grids for this study were generated using the commercial grid generation software GridPro [57]. All grids were structured in three dimensions and were refined in the vicinity of the cavity. A closeup of the grid for the aft wall hybrid injection case is shown in Fig. 4. The region where the cavity floor meets the combustor floor was especially refined, due to the expected high vorticity and steep gradients in this region.

The height of the first cell on the combustor wall, cavity floor and cavity walls was set to 1 × 10⁻³ mm, 2 × 10⁻³ mm and 3 × 10⁻³ mm respectively, to ensure a γ⁺ value of at most 1.23 everywhere.¹ A growth rate of 1.3 was used to distribute cells in the wall normal direction.

Cell counts ranged from 12.9 million for the baseline case up to 14.4 million for case 4.5LD0.8, with the number of cells inside the cavity varying from case to case. The freestream (i.e. outside the cavity) cell distribution is equivalent for all cases. To obtain the grid for the baseline case the cavity was removed but clustering was applied as if the cavity were there. In other words, the cavity itself was removed but the clustering in the domain above the combustor wall was kept the same to ensure grid similarity.

2.3. Flow and boundary conditions

The combustor inflow conditions are representative of a scramjet flying at Mach M = 8.4, altitude h = 38.4 km and dynamic pressure q = 17 kPa. The combustor inflow conditions representative of this flight condition were based on shock tube studies by Kirchhartz [58] and are summarised in Table 2.

The combustor inflow conditions were applied to the domain inflow using a supersonic inflow profile. This profile contained a 5 mm boundary layer, to account for the boundary layer that would start to grow on the scramjet inlet. The inflow profile was generated on a two-dimensional, structured grid on a flat plate of length 2 m and leading

---

¹ For the hybrid injection cases a γ⁺ value of 3.1 was observed in a small number of cells where the fuel jet impacts the cavity wall, however this is not expected to significantly impact results.
edge radius \( r = 0.25 \text{ mm} \), with a vertical cell distribution identical to the distribution in the three-dimensional grids. Where the boundary layer thickness \( (\delta = 0.99u_{\text{wall}}) \) reached 5 mm the primitive flow variables were extracted from the wall to the freestream and applied to the domain inflow.

The combustor and cavity walls were modelled as isothermal walls with \( T_{\infty} = 1800 \text{ K} \) and were solved directly to the wall, without the use of a wall model. This temperature was selected to represent the steady state thermal environment in a scramjet. The injector wall was modelled as an inviscid wall and no plenum was modelled to relax grid requirements [59]. Centroidal extrapolation (no conditions prescribed) was stipulated on the combustor outflow plane, the upper wall and the lateral boundaries. This removes shock reflections from the domain and allowed for a fundamental study of the jet interaction. Inflow turbulence levels were set to 3% with a turbulence length scale of 0.01 m. Stagnation pressure and temperature were applied at the injector inflow boundary.

For the single injector cases \( H_2 \) was injected sonically at a stagnation pressure of \( p_i = 2.48 \text{ MPa} \) and a stagnation temperature of \( T_i = 313 \text{ K} \), with a fuel mass flow of 4.7 g/s. For the hybrid injection cases the same global equivalence ratio (and hence total fuel mass flow) was maintained and fuel mass flow was divided between the injectors based on area, i.e. the stagnation conditions for the two injectors were the same. With an area ratio of 4:1, a stagnation pressure of \( p_i = 1.984 \text{ MPa} \) and a stagnation temperature of \( T_i = 313 \text{ K} \) yield a total fuel mass flow of 4.7 g/s, with 80% of this mass flow injected into the freestream and the remaining 20% injected into the cavity. Note that it was verified afterwards that the injected mass flow was indeed the same for the hybrid and single injector cases.

An oft-cited parameter in injection studies is the jet-to-freestream dynamic pressure ratio, \( I \):

\[
I = \frac{(p_f M^2)}{(p_f M^2)_{\infty}} \tag{1}
\]

where \( p \) is the static pressure, \( r \) is the ratio of specific heats and \( M \) is the Mach number. The subscript denotes conditions in the jet (j) or combustor freestream (\( \infty \)). For the cases with a single injector \( I = 0.92 \) while for the cases with two injectors \( I = 0.72 \).

2.4. Treatment of numerical accuracy

All solutions achieved a reduction in the normalised residuals for the mass, momentum and energy equations of at least three orders of magnitude. A typical example of the convergence history for the cases studied in this work is presented in Fig. 5, which shows the convergence history for case 3LD1.2.

To assess grid dependence the method proposed by Roache [60] was used and implemented as outlined by Celik et al. [61]. This method quantitatively estimates the discretisation error in a grid using three grid refinements and Richardson extrapolation, estimating the error in the limit of grid spacing tending to zero. For more information the reader is referred to [61]. For the present study, two additional grids were created for the aft wall hybrid injection case. The aft wall hybrid injection case was selected for grid refinement because its flowfield is the most complex, hence sensitivity to the grid resolution was expected to be the greatest for this case. The variable that was used for the grid convergence study was mixing efficiency, defined in Section 2.5, since it is a primary variable of interest in this work.

To obtain the grid refinements the characteristic cell size \( h \) was first computed for the medium grid using Eq. (2):

\[
h = \left[ \frac{1}{N} \sum_{i=1}^{N} (\Delta V_i)^{\frac{1}{3}} \right]^{\frac{1}{3}} \tag{2}
\]

where \( N \) is the number of cells and \( \Delta V_i \) is the volume of cell \( i \). Cell density in all three grid dimensions was then either increased (fine mesh) or decreased (coarse mesh) and the characteristic cell size of the new grids was computed, until a grid refinement factor \( r = h_{\text{coarse}}/h_{\text{fine}} \) of at least 1.3 was achieved, as recommended in [61]. Details of the three grids that were obtained are provided in Table 3.

The grid convergence study was performed using RANS computations. While the crescent cavity flowfields are unsteady in nature, as will be discussed in Section 3.3, the unsteadiness is mostly concentrated inside the cavity, which is a well resolved region of the domain. The mean macro flowfield predicted by the RANS and URANS simulations was observed to be similar and no sudden steep gradients appeared in the URANS simulations, hence it is considered valid to use RANS to assess if the grid resolves the combustor flow physics. For the RANS simulations the same numerical set-up as outlined above was used, with the only difference that a steady implicit scheme was used, still with second order spatial discretisation.

Using the method outlined above the discretisation error at \( x/d = 35 \) was obtained. This location is away from the high-gradient region in the flowfield near the injector and the flow has had time to develop, allowing any discrepancies between the flow physics in the three grids to stabilise themselves. Evaluation of the results at this location yielded an approximate relative error of \( e_{\text{rel}}^{21} = 0.804\% \) and an extrapolated relative error of \( e_{\text{rel}}^{21} = 1.04\% \), leading to a grid convergence index (GCI) of 1.29% in the medium grid with an enforced apparent order of \( p = 2 \). The Richardson extrapolate at \( x/d = 35 \) is shown in Fig. 6, along with the values of mixing efficiency obtained for the three grid refinements. The trend of mixing efficiency through the domain is also shown in Fig. 7 for all three grids.

The figure and the GCI values demonstrate that the solution is in the asymptotic region, especially in the farfield, approaching the Richardson extrapolate as grid density increases and demonstrating that the results are sufficiently independent of the grid. In the nearfield

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>( p )</td>
<td>78.8</td>
<td>kPa</td>
</tr>
<tr>
<td>( T )</td>
<td>835</td>
<td>K</td>
</tr>
<tr>
<td>( u )</td>
<td>2468</td>
<td>m/s</td>
</tr>
<tr>
<td>( M )</td>
<td>4.33</td>
<td></td>
</tr>
</tbody>
</table>

Table 2

<table>
<thead>
<tr>
<th>Grid</th>
<th>( n_{\text{coarse}} )</th>
<th>( n_{\text{fine}} )</th>
<th>Refinement factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fine</td>
<td>5.9M</td>
<td>1.92 \times 10^{-1}</td>
<td>( r_f = 1.33 )</td>
</tr>
<tr>
<td>Medium</td>
<td>13.9M</td>
<td>2.56 \times 10^{-1}</td>
<td>( r_f = 1.33 )</td>
</tr>
<tr>
<td>Coarse</td>
<td>33.2M</td>
<td>3.42 \times 10^{-1}</td>
<td></td>
</tr>
</tbody>
</table>

Table 3

Grid information for grid convergence study.

Fig. 5. Convergence history for case 3LD1.2. The normalised residuals for the mass, momentum and energy equations are shown.
the discrepancy between the fine and medium grids is larger, however this is likely because vorticity (which is sensitive to grid density) is dominating mixing in this region of the domain. Overall all three grids predict similar one-dimensional performance through the domain.

To investigate the flowfield in the three grids Fig. 8 compares contours of fuel mass fraction at $x/d = 5$ downstream of the injector for the three grid refinements.

The overall shape of the jet plume is largely the same for all three grids, although the fuel is drawn slightly more towards the combustor wall as grid density increases. This is likely because the cavity vortex is predicted to be stronger as grid density increases, causing more fuel to be drawn to the wall. This is discussed in more detail in Section 3. The coarser grids are also observed to be more dissipative (lower peak fuel mass fraction) than the finer grids, as is expected. Overall the three grids are observed to predict similar jet plume behaviour.

To ensure the same macro flowfield features such as shocks and separations are captured by all three grids Fig. 9 compares contours of static pressure on the combustor wall for the three grids. The figure shows that the overall flowfield structure is similar for all three grids, with only slight discrepancies in the locations and size of some flow features. The largest discrepancy between the three grids is near the cavity front wall, where the secondary fuel jet impinges on the cavity wall and the unsteadiness in the cavity originates, but even in this area the overall flowfield structure is similar in all three grids.

Given that all three grids capture the same global flowfield phenomena and performance and that the solution is in the asymptotic range, the medium grid is selected for the present study; this grid offers improved resolution over the coarse grid, while still maintaining reasonable computational cost compared to the fine grid, especially considering that the present study uses URANS modelling.
2.5. Performance parameters

Complex, three dimensional flowfields are commonly one-dimensionalised using either area, mass flow or stream thrust (flux conserved) averaging [62]. In the present study mass flow averaging will be used to compare the one-dimensional performance of the different combustor configurations, and the one-dimensional performance parameters that will be used are introduced in this section. For each of the parameters presented below a surface integral is taken over a range of streamwise slices (yz planes) to obtain the streamwise evolution of that performance parameter through the domain.

Mixing efficiency is an important parameter in assessing the performance potential of a scramjet combustor, since combustion is mixing limited in scramjets. In the present study mixing efficiency is taken from [63] and is defined as the amount of fuel mixed to at least stoichiometric conditions, i.e. it is an indicator of the amount of fuel that would react fully if the flow were chemically reacting; if the local fuel mass fraction is equal to or less than the stoichiometric ratio it is considered fully mixed, while it is considered increasingly unmixed as the local fuel mass fraction exceeds the stoichiometric ratio and approaches 1. In the present definition of mixing efficiency, this is quantified in the ‘reacting’ fuel mass fraction \( a'_{H_2} \), defined as:

\[
\begin{align*}
 a'_{H_2} &= \left\{ \begin{array}{ll}
 a_{H_2} & a_{H_2} \leq a_{H_2}^{st} \\
 1 - \frac{1}{1 - \frac{a_{H_2}}{a_{H_2}^{st}}} & a_{H_2} > a_{H_2}^{st}
\end{array} \right.
\end{align*}
\]  

(3)

where \( a_{H_2}^{st} \) is the stoichiometric mass fraction for hydrogen and air, \( a_{H_2}^{st} = 0.0292 \), and \( a_{H_2} \) is the local fuel mass fraction. The mass flow averaged mixing efficiency is then calculated as:

\[
\eta_{\text{mixing}} = \frac{\int \rho u a'_{H_2} \, dA}{\int \rho u a_{H_2} \, dA}
\]  

(4)

where \( \rho \) is the local density in kg/s, \( u \) the axial velocity in m/s and the other variables are common with Eq. (3). Using this definition, the local mixing efficiency is 1 when the local fuel mass fraction is stoichiometric or less, and exponentially goes to 0 as the fuel mass fraction approaches 1.

As stated before the purpose of the crescent cavity is to increase streamwise vorticity in the flowfield. To assess if the crescent designs achieve this design objective streamwise circulation is calculated using Eq. (5):

\[
\Gamma = \frac{1}{w d} \int \int \int [\omega_3] \, dA
\]  

(5)

where \( \omega_3 \) is the streamwise vorticity and the other variables are common with Eq. (4). Note that only streamwise vorticity was included here, since it is this component of vorticity that contributes the most to mixing in the freestream.

For effective combustion it is not only important to achieve significant mixing, but also that this mixing is widespread through the combustor. It is therefore important to achieve good vertical penetration into the freestream, which is assessed in this study by calculating the vertical centre of mass of the fuel plume using Eq. (6):

\[
Y_k = \frac{\int \rho u a_{H_2} y \, dA}{\int \rho u a_{H_2} \, dA}
\]  

(6)

where \( y \) is the distance normal to the combustor wall and the other variables are common with Eq. (4).

Finally it is important that any increase in mixing performance does not come at the cost of excessive flow losses. In the present study flow losses are quantified by calculating the mass flow averaged total pressure loss through the domain using Eq. (7):

\[
\begin{align*}
 p_l &= \left[ \frac{1}{\int \rho u dy \, dz} \right] \left[ \int_{-30}^{30} \int_{-30}^{30} p \rho u dy \, dx \right] + \left[ \frac{1}{\int_{-30}^{30} dy \, dz} \right] \left[ \int_{-30}^{30} \int_{-30}^{30} p \rho dy \, dx \right] + \left[ \frac{1}{\int_{-30}^{0} dy \, dz} \right] \left[ \int_{-30}^{30} \int_{-30}^{0} p \rho dy \, dx \right] + \left[ \frac{1}{\int_{-30}^{0} dy \, dz} \right] \left[ \int_{-30}^{30} \int_{0}^{30} p \rho dy \, dx \right] \\
&= \left[ \frac{1}{\int \rho u dy \, dz} \right] \left[ \int_{-30}^{30} \int_{-30}^{30} p \rho u dy \, dx \right] + \left[ \frac{1}{\int_{-30}^{30} dy \, dz} \right] \left[ \int_{-30}^{30} \int_{-30}^{30} p \rho dy \, dx \right] + \left[ \frac{1}{\int_{-30}^{0} dy \, dz} \right] \left[ \int_{-30}^{30} \int_{-30}^{0} p \rho dy \, dx \right] + \left[ \frac{1}{\int_{-30}^{0} dy \, dz} \right] \left[ \int_{-30}^{30} \int_{0}^{30} p \rho dy \, dx \right]
\end{align*}
\]

(7)

where \( p_l \) is the local total pressure in Pa and the other variables are common with Eq. (4). The last three integrals on the right hand side of Eq. (7) account for the total pressure that is lost through the top and side (outflow) boundaries; if this is not accounted for the flow losses would be artificially high, since this total pressure would still be part of the flow if these boundaries were modelled as (inviscid) wall, which would be the case in a practical scramjet. The mass flow averaged total pressure calculated using Eq. (7) is then finally divided by the freestream total pressure on the inflow plane \( (p_{in} = 26346 022 \text{ Pa}) \) to calculate the total pressure loss through the domain.

3. Results

This results section will present trends of the one-dimensional performance parameter defined in Section 2.5 as well as several flowfield visualisations. For the performance parameter trends two types of graphs will be shown: the absolute trends through the domain and normalised trends through the domain. To obtain the latter the trends for the crescent cases are divided by the corresponding trend for the flat plate baseline case, i.e. the mixing efficiency for a crescent case at \( x \) is divided by the mixing efficiency for the baseline at \( x \). If the normalised value is higher than 1 there is more mixing in the crescent case, while there is less mixing in the crescent case if the normalised value is less than 1. This facilitates an easy comparison of cavity performance to baseline performance.

As will be discussed in Section 3.3 the flowfield inside the cavity is oscillatory in nature. Because these oscillations persist downstream in the fuel jet the one-dimensional performance trends also oscillate over time. All trends in this section will therefore be present with bounds of deviation that show how much the trends change over time; for each case the mean value of a performance parameter over the last 50 time steps in the simulation \((\tau = 2.265 \times 10^{-4} \text{ to } 3 \times 10^{-4})\) is presented along with the maximum deviation of that parameter over time. This indicates how much unsteadiness affects one-dimensionalised performance and demonstrates how strong the effects of unsteadiness are for each cavity case. All flowfield images in this section were also extracted from averaged flowfields, obtained by averaging the flowfield solutions of the last 50 time steps for each case.

Finally, before analysing the trends of the performance variable it is pertinent to examine the flowfield in one of the cavity cases to examine if the crescent design is operating as intended and to inform the discussion below. Fig. 10 shows the flowfield in the vicinity of the injector for case 3LD1.2 and the streamlines in the image clearly indicate that the recirculation vortex leaves the cavity and wraps around the injector, reorienting the vorticity in this vortex from the z-axis to the x-axis. This vortex will hereafter be referred to as the cavity vortex. A small horseshoe-type vortex is also observed to be present in front of the cavity, further increasing flowfield vorticity. The flowfield will be examined in more detail later on, but the image demonstrates that the crescent cavity is behaving as intended.

3.1. Performance parameters

To assess the impact of vortex reorientation on mixing Figs. 11 and 12 show the mixing efficiency and circulation, respectively, for all crescent cavity cases and the baseline. The crescent cavities clearly enhance mixing, with especially significant mixing enhancement in the hybrid injection cavity cases. At the domain outflow mixing is enhanced by 11%–22% for the single injector cavity cases and by up to 80%–90% for the hybrid injection cases. It is also clear that mixing varies quite significantly over time, especially for the hybrid injection cases and case 3LD0.8.

For all cavity geometries some mixing already takes place inside the cavity, especially in the hybrid injection cases, indicating that fuel is drawn upstream into the cavity. Immediately downstream of the injector mixing is enhanced the most in the crescent cavity cases, as
Fig. 10. Flowfield in the vicinity of the injector for case 3LD0.8. The black streamlines show shear stress on the combustor wall, while the ribbons represent fluid flowpaths. The combustor wall is flooded with contours of $H_2$ mass fraction, the ribbons are flooded with contours of absolute streamwise vorticity $\omega_x$. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

Fig. 11. Trends of mixing efficiency through the domain for all cavity cases and the baseline. The shaded area indicates the maximum variation of mixing efficiency over time.

Fig. 12. Trends of mixing efficiency normalised by the baseline through the domain for all cavity cases. The shaded area indicates the maximum variation of normalised mixing efficiency over time.

Fig. 13. Trends of circulation through the domain for all cavity cases and the baseline. The shaded area indicates the maximum variation of circulation over time.

For those geometries the rate of mixing continues to be higher than in the baseline relatively far into the domain, until $x/d \approx 25$, enhancing mixing by up to 125% with respect to the baseline. From $x/d \approx 25$ on the rate of mixing is lower than in the baseline for the hybrid injection cases, shown by the downward slope of the trends in Fig. 12, but this is because the fuel concentration gradients are higher in the more unmixed baseline case which promotes diffusion mixing.

Interestingly, mixing is observed to be most dependent on cavity depth in the near-field for the single injector cases; the cases with $D = 0.8$ show near identical mixing until $x/d \approx 5$, where the trends start to diverge and the performance of case 3LD0.8 is superior. Conversely, mixing directly downstream of the injector is observed to be higher for case 3LD1.2. One possible reason for this dependence on cavity depth is the influence of depth on the intensity of the cavity recirculation vortex. The vortex is larger and stronger for case 3LD1.2 than for case 3LD0.8 because of the increased cavity volume, and while the cavity is largest for case 4L.5D0.8 the vorticity is also spread over a larger area in that geometry, reducing its intensity. Furthermore, the cavity vortex exits the cavity farther away from the fuel plume for case 4.5LD0.8 because of the increased width of the cavity, reducing its effect on mixing.

Interestingly, mixing is observed to be most dependent on cavity depth in the near-field for the single injector cases; the cases with $D = 0.8$ show near identical mixing until $x/d \approx 5$, where the trends start to diverge and the performance of case 3LD0.8 is superior. Conversely, mixing directly downstream of the injector is observed to be higher for case 3LD1.2. One possible reason for this dependence on cavity depth is the influence of depth on the intensity of the cavity recirculation vortex. The vortex is larger and stronger for case 3LD1.2 than for case 3LD0.8 because of the increased cavity volume, and while the cavity is largest for case 4L.5D0.8 the vorticity is also spread over a larger area in that geometry, reducing its intensity. Furthermore, the cavity vortex exits the cavity farther away from the fuel plume for case 4.5LD0.8 because of the increased width of the cavity, reducing its effect on mixing.

For the hybrid cases the streamwise circulation in the cavity is also observed to be significantly higher than for the single injector cases, although the circulation in the farfield is relatively similar to the other cavity cases. Circulation is likely higher inside the cavity for the hybrid cases because the secondary fuel jets create additional recirculation regions inside the cavity, enhancing vorticity through shear and baroclinic torque production. For all cavity cases the increased circulation persists relatively far into the domain, until $x/d \approx 20$, where circulation for the cavity cases approaches the baseline value.

In the above figures performance oscillates the most over time in the hybrid injection cases, followed by case 3LD0.8 and 3LD1.2. This is expected, since the magnitude of the cavity oscillations are largest for those cases as discussed previously. The performance of 4.5LD0.8 is observed to vary the least over time, as is again expected because the oscillations inside the cavity are weak for this case. There is also no discernible change over time in the one dimensional performance of the baseline case, as expected.

To examine the mechanisms driving mixing enhancement in the single injector cavity cases, Figs. 14–16 show slices at several different streamwise locations for cases 3LD0.8, 3LD1.2 and 4.5LD0.8. The
baseline case has also been included on the left hand side of each figure for reference.

The slices show that in the cavity cases ‘lobes’ of fuel exist at the edge of the barrel shock at $x/d = 1$, where the cavity vortex shown in Fig. 10 is entraining fuel and transporting it to the wall. In the baseline case these lobes are absent, demonstrating the effect of the vortex reorientation on the fuel plume. As downstream distance increases the lobes grow larger and at $x/d = 3$ there is clearly more fuel on the cavity wall for cases 3LD0.8 and 3LD1.2 than in the baseline. Lateral penetration is also observed to be higher for these cases.

The fuel plume shape for case 4.5LD0.8 more closely resembles that of the fuel plume in the baseline case, presumably because the streamwise vortex is ejected out of the cavity at a greater lateral distance away from the centreline for this case and hence the cavity vortex entrains less fuel. As downstream distance and hence lateral jet penetration increases more fuel is drawn into the cavity vortex for case 4.5LD0.8 however and the plume shape is more similar to that in the other cavity cases.

The slices also clearly show that the peak fuel concentration is much lower for case 3LD1.2 than for the other cases, suggesting more mixing has taken place for this case as is indeed the case, Fig. 11. The higher peak fuel mass fractions (and hence higher fuel concentration gradients) in the baseline and cases 3LD0.8 and 4.5LD0.8 also explain why far field diffusion is higher for those cases than for case 3LD1.2, as shown in Fig. 12.

Figs. 17 and 18 show slices at the same streamwise locations as before, but for the hybrid injection cases. The baseline case is again included for comparison. The jet plumes for the hybrid injection cases are clearly fundamentally different from the plumes in the single injector cases; in the hybrid injection cases a significant layer of fuel is present on the combustor wall on the side of the main fuel plume. This fuel distribution is the result of fuel from the secondary injector that is entrained into the primary cavity vortex and then transported to the combustor wall when the cavity vortex is reoriented. This is visualised in Fig. 19, which compares contours of $H_2$ mass fraction on the combustor wall for both hybrid injection cases. The regions of increased fuel mass fraction are observed to coincide the path of the cavity vortex, demonstrating the role of the cavity vortex in creating the plume structures shown in Figs. 17 and 18. The fuel–air mixing layer is significantly larger for the hybrid cavity cases because of this fuel distribution, which together with increased circulation in the nearfield explains the significant nearfield mixing enhancement the hybrid injection cases offer with respect to the baseline.

While the fuel layer is present in both hybrid cases, the fuel distribution inside this layer is different for each case. For the front wall injection case the fuel is concentrated towards the main jet plume, while fuel is concentrated towards the edge of the fuel layer for the aft wall injection case. This is because fuel is entrained into the cavity vortex in a different location for each design. For the aft wall injection case fuel collects on the cavity front wall and is convected out of the cavity along the cavity front wall, away from the main fuel plume, whereas fuel collects on the cavity aft wall for the front wall injection case and as a result it is convected out of the cavity along the cavity aft wall, close to the main fuel plume. This is also shown in Fig. 19, where the fuel distribution inside the cavity is observed to be different for the two hybrid injection cases.

The difference in the fuel distribution in the lateral fuel layer becomes more pronounced as downstream distance increases. A ‘stem’ of fuel attaches the main fuel plume to the combustor wall for the front
Fig. 16. Slices of case 4.5LD0.8 and the baseline showing contours of H₂ mass fraction at several streamwise locations.

Fig. 17. Slices of the front wall hybrid injection case and the baseline showing contours of H₂ mass fraction at several streamwise locations.

wall injection case, while the fuel plume core is separated from the combustor wall for the aft wall injection case.

Furthermore, the shape of the lateral fuel layers becomes increasingly different as downstream distance increases. In the front wall injection case fuel from the lateral middle of the fuel layer (\(z/d = 3\) in Fig. 17) is sucked into the separation bubble at the base of the fuel jet, as can also be seen in Fig. 19. This draws fuel away from the middle of the lateral fuel layer, causing an almost separate lobe to form at the extrema of the lateral fuel layer.

In the aft wall injection case fuel is more evenly distributed in the fuel layer however, because fuel diffuses both inward and outward from the high fuel concentration region at the extrema of the lateral fuel layer, resulting in a more even fuel distribution in the far-field. This increases the fuel–air contact area and increases diffusion mixing in the farfield compared to the front wall injection cases.

A clear consequence of the cavity vortex drawing fuel to the wall is lower vertical penetration. This can be inferred from Figs. 14–18 and is clearly shown in Fig. 20, which shows jet penetration for all cases. The jet centre of mass is lowest for the hybrid injection cases, as is expected. The dynamic pressure ratio \(I\) of the primary injector is lower for these cases, reducing penetration, and the presence of the lateral fuel layer further lowers the jet centre of mass. For the front wall injection arrangement the jet centre of mass is also observed to be negative inside the cavity because the fuel inside the cavity is mostly concentrated towards the domain centreline and the cavity aft wall, which is the deepest part of the cavity.

Further downstream the jet centre of mass location is similar for cases 3LD1.2 and 4.5LD0.8, with penetration slightly lower for case 3LD0.8. Penetration is higher for case 3LD1.2 than for case 3LD0.8 because there is more shielding for case 3LD1.2, as will be discussed below, and penetration is higher in case 4.5LD0.8 than in case 3LD0.8 because the cavity vortex entrains less fuel in the latter case, as discussed before. A larger discrepancy exists between the hybrid injection cases however, where the jet centre of mass is markedly lower for the front wall injection case because of the presence of the high fuel concentration region at the base of the main fuel jet, i.e. the ‘stem’ in Fig. 17.

To assess the flow losses associated with the increase in mixing in the cavity cases Fig. 21 shows total pressure through the domain for all cases. All cavity cases incur a characteristic drop in total pressure inside the cavity, although this pressure drop is not very large because the cavities are relatively small. In the hybrid injection cases total pressure directly in front of the injector is higher than in the baseline case because the reduced dynamic pressure ratio \(I\) of the injector reduces the strength of the injector bow shock.

Directly downstream of the injector total pressure is higher than in the baseline case for the cavity cases Fig. 21 shows total pressure through the domain for all cases. All cavity cases incur a characteristic drop in total pressure inside the cavity, although this pressure drop is not very large because the cavities are relatively small. In the hybrid injection cases total pressure directly in front of the injector is higher than in the baseline case because the reduced dynamic pressure ratio \(I\) of the injector reduces the strength of the injector bow shock.

Further downstream the jet centre of mass location is similar for cases 3LD1.2 and 4.5LD0.8, with penetration slightly lower for case 3LD0.8. Penetration is higher for case 3LD1.2 than for case 3LD0.8 because there is more shielding for case 3LD1.2, as will be discussed below, and penetration is higher in case 4.5LD0.8 than in case 3LD0.8 because the cavity vortex entrains less fuel in the latter case, as discussed before. A larger discrepancy exists between the hybrid injection cases however, where the jet centre of mass is markedly lower for the front wall injection case because of the presence of the high fuel concentration region at the base of the main fuel jet, i.e. the ‘stem’ in Fig. 17.
and hence total pressure loss across the shock is lower. While there is also a small upstream shock in the baseline case, also highlighted in Fig. 22, the shock is much larger in the cavity cases and it originates farther upstream, allowing more freestream fluid to pass through it and shielding the injector barrel shock from the freestream significantly more.

Indeed, total pressure directly downstream of the injector is highest for case 4.5LD0.8 among the single injector cavity geometries, which is the geometry for which the upstream shock originates farthest upstream and hence the largest amount of freestream fluid passes through the upstream shock. Total pressure is also observed to be higher for case 3LD1.2 than for case 3LD0.8, because the cavity recirculation is larger for case 3LD1.2 and as a result it rises farther out of the cavity, providing more shielding than in case 3LD0.8. Downstream of the injector the rate of total pressure loss is slightly higher than in the baseline for all cavity cases. This is likely because of the enhanced mixing in these cases. At the end of the domain \(x/d > 20\) case 3LD1.2 has the lowest total pressure, followed by cases 3LD0.8 and 4.5LD0.8. Throughout the domain total pressure is highest for the hybrid injection cases, since the injection pressure in those cases is lower than in the baseline and single injector cases and hence the bow shock in front of the primary injector is weaker, reducing total pressure loss across the shock. Near the injector the total pressure is higher for case 3LD1.2 than for case 3LD0.8, because the cavity recirculation is larger for case 3LD1.2 and as a result it rises farther out of the cavity, providing more shielding than in case 3LD0.8.

 downstream of the injector the rate of total pressure loss is slightly higher than in the baseline for all cavity cases. This is likely because of the enhanced mixing in these cases. At the end of the domain \(x/d > 20\) case 3LD1.2 has the lowest total pressure, followed by cases 3LD0.8 and 4.5LD0.8. Throughout the domain total pressure is highest for the hybrid injection cases, since the injection pressure in those cases is lower than in the baseline and single injector cases and hence the bow shock in front of the primary injector is weaker, reducing total pressure loss across the shock. Near the injector the total pressure is higher for
the aft wall injection case, but at the outflow it is slightly higher for the front wall injection case. As will be discussed in Section 3.2 the upstream shock originates farther upstream in the aft wall injection case, reducing bow shock strength and increasing total pressure in the nearfield. Beyond \( x/d = 20 \) the far field mixing rate is higher in the aft wall injection case than in the front wall case however, as shown in Fig. 12, hence the total pressure loss is also expected to be higher.

At the domain outflow case 4.5LD0.8 and the hybrid injection cases have slightly higher total pressure than the baseline, with cases 3LD0.8 and 3LD1.2 having slightly lower total pressure. The absolute values are very close however, with relative differences on the order of 0.3%. In addition, for most cavity cases the bounds of deviation overlap the baseline total pressure trend, hence the differences in total pressure at the outflow are essentially within the margin of error. Only for the front wall injection case is total pressure consistently higher than in the baseline at the outflow, although the difference is again quite small.

It should also be noted that if the domain were longer (as would likely be the case for a practical combustor) this could change; as mentioned before the rate of total pressure loss in the farfield is higher in the cavity cases than in the baseline because of increased mixing, so in a longer domain total pressure could be lower in the cavity cases than in the baseline. This higher total pressure loss could be compensated for by increased combustion in the cavity cases, however this is beyond the scope of the present study.

While total pressure loss provides a measure of the flow losses in the flowfield, it does not directly account for wall drag. One of the main penalties of using cavities is that they incur pressure drag on the cavity walls, and it is therefore important to take this into account when evaluating cavity performance. For this purpose Table 4 compares the pressure, viscous and total drag incurred by each crescent cavity design and the baseline. For each case the mean values of drag over the last 50 time steps in the solution time is provided, along with the maximum deviation from this mean value over time.

Interestingly, total drag is lower for most cavity designs than for the baseline. Only case 4.5LD0.8 and the front wall hybrid injection case have higher drag than the cavity case, mostly because of the high pressure drag for these designs. For the front wall injection case the high pressure secondary fuel jet impinges on the cavity aft wall which significantly increases base drag. Conversely, for the aft wall hybrid injection case the jet impingement actually generates thrust and decreases total drag. For case 4.5LD0.8 the surface area of the cavity walls is larger than for cases 3LD1.2 and 3LD0.8, hence the pressure drag generated by the high pressure subsonic fluid inside the cavity is higher for case 4.5LD0.8 than for the other cases.

Besides pressure drag, the viscous wall drag is lower in the cavity cases than in the baseline case, particularly for the hybrid injection case. This is likely because there is more fuel on and near the combustor wall for these cases, as can be observed in Figs. 14–18; the viscosity of \( H_2 \) is roughly half that of air, which decreases viscous drag in the where there is fuel on the combustor wall.

As was the case for the one-dimensional performance parameters drag varies most significantly over time for the hybrid injection cases. This is because of the unsteady behaviour of the secondary fuel jet in these cases, discussed in more detail in Section 3.2, affecting the pressure drag component of the total drag.

To summarise the discussion on performance parameters Table 5 lists the difference in performance parameters with respect to the baseline at the domain outflow plane for all cases and performance parameters. The table shows that the cavity cases can significantly enhance mixing at minimal or no total pressure loss, which is an encouraging result. It does come at the penalty of reduced jet penetration and sometimes significant concentration of fuel on the combustor wall for the hybrid cases, however the effects of this on wall heat flux and combustion quality will only become evident when reacting flow simulations are performed.

### 3.2. Flowfield

The flowfield in several cavity cases is now discussed in some detail to better understand the mixing enhancement mechanisms and the factors driving the enhancement. The flowfield for case 3LD1.2 and the hybrid injection cases, followed by a discussion of the mechanisms driving the lateral oscillations inside the cavity.

#### 3.2.1. Case 3LD1.2

Fig. 23 shows an isometric view of the flowfield in the vicinity of the cavity for case 3LD1.2. The combustor wall has been flooded with contours of \( H_2 \) mass fraction and shear stress (black lines) and fluid streamlines (coloured ribbons) are visible.

There are several similarities between the flowfield in Fig. 23 and the flowfield around a rectangular upstream cavity, which was the topic of a previous study [41]. These similarities are now summarised.

In both cases the cavity recirculation rises out of the cavity and induces a shock at the cavity front wall, the upstream shockwave. For the rectangular case this shockwave is oblique, whereas it is a bow shock in the present study due to the finite width and curvature of the crescent cavities. As discussed before, this shock reduces the Mach number of the freestream combustor flow and reduces the strength of the bow shock in front of the fuel injector.

By rising out of the cavity recirculation partially shields the fuel jet from the freestream flow, allowing the barrel shock to stand more upright. The effect is not very strong for the crescent cavity cases in this study, since the crescent cavities are relatively short compared
to the rectangular cavity that was investigated previously and hence shielding is limited. For both cases the cavity recirculation entrains some fuel from the fuel jet into the cavity, which was shown to facilitate combustion and ignition enhancement inside the cavity for the rectangular cavity case [39].

Inside the cavity the flowfield in the present study is different from the rectangular cavity flowfield. For all crescent designs the large recirculation region inside the cavity wraps around the jet following the shape of the front wall, as shown by Fig. 10 and the red streamline in Fig. 23. This reorients the fluid (and vorticity) in the recirculation from the \( z \) to the \( x \) axis, enabling it to contribute to freestream mixing. This vortex, termed the cavity vortex, is the primary driver of mixing enhancement for the crescent cavity cases and the reorientation is the main design objective of the crescent cavities.

While the majority of the cavity fluid is entrained into the cavity vortex there is a pocket of fluid at the junction of the cavity front wall and cavity floor that has a rotation opposite to that of the cavity vortex, represented by the blue streamline in Fig. 23. This fluids forms a secondary cavity vortex that is also reoriented and merges with the primary vortex downstream of the injector. On the centreline the fluid in this pocket on the cavity front wall is not part of the secondary vortex, but it is entrained into the primary vortex instead as shown by the shear stress lines in region A.

In between the two cavity vortices a small layer of fluid exists that is part of neither vortex, represented by the white streamline in Fig. 23. This fluid approaches the cavity in the freestream and is convected towards the cavity floor on the edge of the primary cavity vortex or into the low pressure separation bubble behind the barrel shock, as demonstrated by the shear stress lines in region B in Fig. 23. The fluid in this region is also exposed to the region of high pressure, applying an adverse pressure gradient applied to the boundary layer upstream of the cavity. Finally, the primary cavity vortex is significantly larger for case 4.5LD0.8, as expected because of the increased cavity volume and aspect ratio [64], and as a result the secondary cavity vortex is relatively small and it is pushed out of the cavity approximately halfway along the cavity front wall.

For case 3LD0.8 region B in Fig. 23 is larger than for case 3LD1.2, likely because the cavity front wall is relatively low more fluid follows the path represented by the white streamline in Fig. 23.

It should be noted that the instantaneous flowfield differs to some degree from the average flowfield presented in Fig. 23. Specifically, the fluid in region B in Fig. 23 oscillates laterally over time as does the fuel jet, which causes slight differences in the \( H_2 \) distribution on the combustor wall. As mentioned before the macro flowfield behaviour is the same however, which is the subject of this discussion.

3.2.2. Hybrid injection cases

Figs. 24 and 25 show isometric views of the flowfield for the aft wall and front wall injection cases, respectively, which will now be analysed.

As before, the black lines represent shear stress lines and the coloured ribbons represent fluid flowpaths. The behaviour of the freestream fluid is largely similar to that of the single injector cases. Fluid from the cavity shear layer is entrained into the cavity recirculation or is pushed into the region of baroclinic torque production next to the barrel shock (represented by the green streamline in Fig. 24) and fluid in the wall boundary layer is entrained into the upstream horseshoe vortex. The discussion below will therefore mostly focus on the flowfield inside of and around the cavity.

Aft wall injection

For the aft wall injection case the separation region upstream of the cavity is clearly significantly larger than for the other cases. The impingement of the fuel jet on the cavity front wall locally creates a region of high pressure, applying an adverse pressure gradient to the boundary layer upstream of the cavity and causing it to separate farther upstream. This enhances both the size and strength of the horseshoe vortex that forms in this region, represented by the white streamline in Fig. 24. The horseshoe vortex also entrains some fuel from the cavity fluid, as represented by the orange streamline in the figure and the fuel mass fraction contours on the combustor wall.

Inside the cavity the fuel jet effectively bisects the cavity into two halves with similar flow physics. On both side of the fuel jet a cavity recirculation vortex develops that wraps around the fuel jet and creates the cavity vortex, represented by the blue and red streamlines in Fig. 24. Note that a large mount of fuel from the secondary is entrained into this vortex as expected and demonstrated by the blue and yellow streamlines in Fig. 24.

The cavity vortex is not fully contained inside the cavity for the aft wall injection case, instead it spills outside of the cavity over the cavity front wall, as marked in Fig. 24. This spillage of the cavity vortex

---

**Fig. 23.** Isometric view of the flowfield in case 3LD1.2. The combustor wall is flooded with contours of \( H_2 \) mass fraction. The black streamlines denote shear stress lines while the coloured ribbons represent fluid flowpaths. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)
is likely driven by the presence of the high pressure secondary fuel jet inside the cavity, which applies a pressure gradient to the cavity vortex that causes the vortex to spill out of the cavity. In addition, because the upstream bow shock stands farther away from the cavity front wall there is more room for this spillage to occur.

**Front wall injection**

This spillage effect occurs to a much lesser degree for the front wall injection case, for which the flowfield is shown in Fig. 25. As discussed before the curvature of the aft wall allows some of the fuel injected by the secondary injector to escape the cavity, lowering the pressure gradient applied to the cavity vortex. Some spillage still occurs however, as visualised by the white streamline in Fig. 25.

Because the secondary jet impinges on the aft cavity wall for the front wall injection case the size of the separation upstream of the cavity is similar to that in case 3LD1.2. The horseshoe vortex is therefore also smaller than for the aft wall injection case, which explains why the circulation for the aft wall injection case upstream of the cavity is higher than for all other cases.

The flowfield inside the cavity is quite complex for both hybrid injection cases, as is also shown by the flowpaths represented by the white, green and blue streamlines in Fig. 25; these streamlines showcase some of the detailed complex fluid behaviours inside the cavity, however the majority of cavity fluid is eventually entrained into one of the several vortices in and around the cavity and reoriented in the streamwise direction, contributing to streamwise vorticity. The specific behaviour of the fluid changes over time and has a relatively minor impact on the overall flowfield, hence these detailed fluid behaviours are not analysed in further detail.

### 3.3. Temporal stability

The preceding section has discussed the averaged one-dimensional performance and flowfield for the cavity cases, however it is clear that both are affected by unsteadiness. The nature of these oscillations is now discussed, first by describing the mechanisms driving the oscillation and then by characterising the harmonic frequencies of the oscillations for different cavity geometries.

**Oscillation mechanisms**

For the single injector cases the curvature of the cavity front wall drives the oscillations via the following mechanism: fluid in the cavity near the domain centreplane recirculates along the cavity floor and impinges on the front wall of the cavity. Because of the curvature of the wall this fluid is pushed inward, towards the cavity centreline which increases mass flow towards one side of the cavity, the +z side in Fig. 23 for example. This extra fluid mass then recirculates again, impinging on the cavity front wall and is pushed towards the other side of the cavity (the +z side in Fig. 23), increasing mass flow to that side of the cavity. The extra fluid will recirculate again, sustaining the oscillation.

By periodically adding and removing mass from one half of the cavity to the other the cavity oscillations affect how much fluid is expelled from the cavity by the reorienting cavity vortex. As a result the cavity vortex is stronger on the side of the cavity where mass flow is temporarily increased, causing the vortex to entrain more fuel from the jet plume on that side and laterally shift the jet plume. When the cavity vortex is then stronger on the other side of the cavity the same will happen, causing the fuel jet to oscillate laterally and transferring the cavity oscillations to the freestream.

For the hybrid injection cases the oscillation is driven by the secondary fuel jet, although the source of the oscillation is different for the front and aft wall injection cases.

In the aft wall hybrid injection case the secondary is effectively a pulsing jet: as fuel injected by the secondary fuel jet collects on the cavity front wall in Fig. 24 the jet back pressure increases and injector mass flow decreases. This decrease in fuel mass flow allows the fuel that has collected on the front wall to be distributed through the cavity, reducing the jet back pressure and increasing fuel mass flow again. Fuel then collects on the front wall again and the cycle restarts.

Because of the pulsing secondary jet the fuel mass flow entrained by the cavity recirculation vortex and convected downstream into the freestream is not constant, causing the oscillation observed in the one-dimensional performance trends.

For the front wall hybrid injection case the lateral oscillations are driven by impingement of the secondary fuel jet on the cavity aft wall. Because of the curvature of the aft wall the fuel jet does not impinge perfectly on the cavity secondary wall, causing more fuel to be injected in one half of the cavity, the −z half in Fig. 25 for example. As more fuel is injected the static pressure in that half of the cavity increases, pushing the fuel jet into the other half of the cavity where the process repeats, sustaining the oscillation.

The shape of the cavity walls and the direction of fuel injection allow some of the fuel that collects on these surfaces to escape into

---

1. Note that the average fuel mass flow is the same as the fuel mass flow in the single injector cases, i.e. global equivalence ratio is the same.
The mechanism driving the lateral oscillations relies on the recirculation of mass inside the cavity, a convection phenomenon that takes significantly longer than the propagation of pressure waves between the front and aft walls in the cavity and reduces oscillation frequency. The lateral transport of mass between the two halves of the cavity further increases the oscillation period, contributing to the decrease in harmonic frequencies for the crescent cavity cases.

Comparing the frequencies for the various cavity cases, the first harmonic frequency is similar for the single injector cavities but higher for the hybrid cavity cases. There are more differences in the second and third harmonic frequencies, although it should be noted that the third harmonic is weak for all cases and the second harmonic is only strong for the aft wall hybrid injection case and cases 3LD0.8 and 3LD1.2.

Interestingly, as cavity depth increases the second and third harmonic frequencies are observed to increase, which is somewhat unexpected. Observation of the flowfield reveals that the primary cavity vortex is larger for case 3LD1.2, hence the increased fluid mass in one half of the cavity is recirculated faster than in case 3LD1.2. As a result, the rate at which mass is exchanged between the two halves of the cavity is higher and the frequency of the oscillations is higher.

Conversely, as cavity length increases the second and third harmonic frequencies decrease, as is expected since the time required to recirculate the increased fluid mass in one half of the cavity is longer and the rate at which this mass is exchanged between the two halves of the cavity is lower.

The magnitude of the oscillations is largest for the hybrid injection cases, since the high-pressure secondary fuel jet drives the oscillations for those cases, followed by cases 3LD1.2 and 3LD0.8. In case 4.5LD0.8 the oscillations are very minor, likely because the front and aft walls are farther removed from each other and the curvature of the front wall is lower, reducing the exchange of mass between the two halves of the cavity and hence dampening the oscillations.

The above analysis demonstrates that a time step of $\Delta t = 1.5 \times 10^{-6}$ s is sufficient to temporally resolve the oscillations in the flowfield, with the first three cavity harmonics being sampled 4–24 times. It should also be noted that no large-scale unsteadiness was observed for the baseline case, as is expected when using URANS.

4. Conclusion

Unsteady RANS (URANS) simulations were used to compare the mixing performance of five crescent cavity geometries, two of which featured hybrid injection, to a no-cavity baseline and assess the temporal stability of the cavity flowfields.

It was found that all crescent designs enhance mixing with respect to the baseline, by 11%–22% at the outflow for the single injector cavity geometries and by 80%–90% for the hybrid injection geometries. The hybrid injection cases were also observed to offer more rapid mixing in the nearfield, with up to 125% mixing enhancement over the baseline in the nearfield.

The crescent cavity geometries were observed to enhance mixing by reorienting the vortex contained in the cavity to the streamwise direction and expelling this vortex out of the cavity, enhancing streamwise vorticity in the flowfield. For the hybrid cavity cases fuel from the secondary fuel injector was entrained into the reoriented cavity vortex which significantly increased fuel–air contact area.

Total pressure at the domain outflow was very similar for all cases, with the hybrid injection cases having slightly higher total pressure than the baseline and two of the three crescent cavity designs having slightly lower total pressure. The absolute differences in total pressure were very small however. Vertical jet penetration was also observed to be lower than in the baseline for all crescent configurations, especially for the hybrid injection schemes. This is mostly due to the stronger vortices transporting fuel along and towards the combustor wall for the cavity cases, lowering the jet centre of mass. Lateral penetration was
higher in the cavity cases however and the increased concentration of fuel on the wall also lowered viscous wall drag for the cavity cases. This caused total wall drag to be lower than in the baseline for some cavity cases, even when accounting for cavity pressure drag.

Analysis of the flowfield for the crescent cavity showed that there are two main vortices inside the cavity, both of which reorient to the streamwise direction and enhance streamwise vorticity. This achieves the primary design objective of the crescent cavity geometries. For all cases a horseshoe vortex was also present upstream of the cavity, although this vortex was significantly larger for the hybrid aft injection case than for all other cases. The flowfield inside the cavity was observed to be significantly more complex for the hybrid injection cases, although there were still two main vortices inside the cavity. In summary it was shown that crescent cavities can significantly enhance mixing in a high supersonic scramjet combustor flow, at a marginal increase in drag. Vertical jet penetration is lower for the cavity cases however, although lateral penetration is greater. The hybrid fuel injection arrangements that were studied are particularly promising, although more extensive studies are required to fully explore their potential as combined flameholding and mixing enhancement devices in high supersonic scramjet combustor flows.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgements

The authors would like to gratefully thank the support of the CRC-P50510 Hydrocarbon Fuel Technology for Hypersonic Air Breathing Vehicles for Hypersonic Air Breathing Vehicles and RMIT University, as well as the high performance computing resources and support from the National Computational Infrastructure (NCI) Australia. The CRC Programme support industry-led collaborations between industry, researchers and the community.

References


L.-Q. Li, W. Huang, L. Yan, Mixing augmentation induced by a vortex generator located upstream of the transverse gaseous jet in supersonic flows, Aerosp. Sci. Technol. 68 (2017) 77–89.


