ASPIRE – An Innovative Hypersonic Propulsion Paradigm

Christopher MacLeod, Matthew Murray
Engineering Section, University of the Highlands and Islands, Lews Castle College Campus, Isle-of-Lewis, HS2 OXR.
christopher.macleod@uhi.ac.uk

Abstract
This paper reports the results of a theoretical study into an innovative scramjet-like hypersonic propulsion system. The method, named ASPIRE (Air-breathing Supersonic Pellet Injection Rotary Engine), aims to overcome one of the main limitations of the traditional scramjet cycle – poor mixing and therefore inadequate combustion of air and fuel. The proposed system achieves this by redirecting the main airflow, injecting encapsulated or localised fuel throughout the engine-duct mixing volume and then switching the main airflow back on – this then engulfs and mixes with the fuel. The results presented here include design calculations, simulations and mathematical modelling. These show that the system performs well in simulation and gives excellent theoretical results for air-fuel mixing and airflow dynamics. A road-map to a working engine is also outlined.

Keywords: Scramjet, Pulse detonation engine, ASPIRE, hypersonic, propulsion, pellets, air fuel mixing

1. INTRODUCTION
There are many formidable engineering challenges to overcome in order to obtain a working scramjet engine. These have been catalogued extensively in the literature [1] - but perhaps the most pressing and difficult is that of achieving good air-fuel mixing (and as a result, combustion) [2]. Achieving this is the main aim of the method discussed here.

The system works by redirecting the airflow away from the main engine-duct and injecting the fuel in the form of encapsulated gas, liquid droplets/capsules or a pelletized solid (or a mixture of these forms). The air is then redirected back into the duct with the appropriate timing and engulfs the fuel, which vaporizes and so releases its load evenly throughout the mixing volume. The final stage is ignition and combustion. The engine concept has been named ASPIRE (Air-breathing Supersonic Pellet Injection Rotary Engine). The cycle is explained in the sections below.

The basis of this system has already been the subject of a peer-reviewed paper [3] and was also included in a review on air-fuel mixing [4]. A layman’s explanation was carried in the magazine “analog” [5] and it was featured in the magazine of the Institution of Mechanical Engineers (Professional Engineering), the New Statesman Magazine and by the BBC. The purpose of this paper is to present further proof-of-concept results which show that the idea is definitively worth pursuing to the experimental verification stage.

The paper outlines the basic cycle and engine structure in the section below. It then discusses the dynamics of the released fuel. Some alternative ideas for intake design are then reviewed. Finally,
some other applications are discussed and a roadmap for further development outlined before a concluding summary section.

2. THE BASIC ENGINE CONCEPT, CYCLE AND PARAMETERS

The original idea and its development is explained in previous papers – so only an overview will be covered here. An animation is also available on the internet which demonstrates the concept [6]. The basic idea is shown in figure 1.

Figure 1, The basic sequence of events during the engine cycle.

At the start of the cycle, the airflow is diverted away from the main engine duct as shown in figure 1b). One way this can be accomplished is using a rotary nose-cone valve, as discussed in the following sections. Next, fuel is injected or dropped into the duct as shown in 1c), the key principle of the system is that the fuel is in discrete packages – this may be in the form of solid-fuel pellets, liquid-fuel droplets, gas capsules or a mixture of these (these will all be referred to in the next sections as “pellets” for simplicity – unless it is necessary to discuss their actual nature). The injection sequence, dynamics and timing are arranged so that the fuel is evenly spaced in the duct volume prior to the next stage. The front valve is then opened as shown in 1d) and air enters, surrounding and accelerating the injected pellets as shown 1e). Because the pellets have a higher density than the airflow and are initially stationary, relative to that airflow, along the axis of the duct, their inertia means that the flow overtakes and engulfs them - but they are distributed correctly throughout the airflow volume. Finally, in 1f), the heat and friction of the impinging airflow heats the pellets and they vaporise, mixing their contents with the flow which then ignites. Both vaporisation and ignition can be assisted by other technologies.

A key advantage of this system is its flexibility. The pellets can be a mixture of different types and sizes, with different fuels (including hypergolic and pyrophoric types) – giving fine control of fuel distribution and composition. They can be of a composite nature, with a gaseous, liquid and solid component in a single unit or have, for example, a disruptive core which propels fuel outwards on activation. They can have a variety of coatings to control the timing of their load distribution or be disrupted by inductive
or radio frequency heating. They can be combined with more traditional scramjet continuous fuel injection from duct edges or struts. Finally, they can carry loads into the flow other than combustible fuels - like electromagnetically active materials for more advanced laser, microwave or magnetohydrodynamic propulsion concepts [4] and from this point of view the system outlined is an enabling technology for these.

For a conventional combustion system, fuel calculations are straightforward. The mass of any fuel required to completely burn in 1 m³ of air is:

\[ m_f = \frac{\rho_a}{S} \]

Where \( \rho_a \) is the air density under the selected flow conditions and \( S \) is the stoichiometric ratio by mass (important examples include: \( \text{CH}_4 = 17.2 \), Petrol (Gasoline) = 14.7, \( \text{H}_2 = 34 \)). This can be generalised for any volume by multiplying the result by that volume. The volume of fuel required for this mass is:

\[ V_f = \frac{\rho_a}{S} = \frac{\rho_a}{S\rho_f} \]

The volume of fuel per capsule for an air volume of \( V_a \) assuming \( n \) capsules is therefore:

\[ V_c = \frac{\rho_aV_a}{S\rho_f} \]

Or number of pellets required:

\[ n = \frac{\rho_aV_a}{S\rho_fV_c} \]

Expressions for volumes of common pellet shapes (for example, spheres, cylinders, ellipsoids) can be found in any appropriate reference.

Because the system is very flexible there is a large latitude in how it can be set up. Pellet numbers per second required for 1 MW of released combustion power, for two types of gaseous capsule (\( \text{H}_2 \) and \( \text{CH}_4 \)) at 1 bar internal pressure and one liquid type (paraffin or kerosene), are shown in table 1.

<table>
<thead>
<tr>
<th>Pellet Radius</th>
<th>1 mm</th>
<th>2 mm</th>
<th>5 mm</th>
<th>1 cm</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \text{H}_2 ) gas at STP</td>
<td>18.86M</td>
<td>2.36M</td>
<td>150k</td>
<td>19k</td>
</tr>
<tr>
<td>( \text{CH}_4 ) gas at STP</td>
<td>5.97M</td>
<td>747k</td>
<td>48k</td>
<td>6k</td>
</tr>
<tr>
<td>Paraffin / Kerosene</td>
<td>6.45k</td>
<td>805</td>
<td>52</td>
<td>7</td>
</tr>
</tbody>
</table>

Notes on table: Spherical pellets, M stands for million \((\times 10^6)\), k stands for thousand \((\times 10^3)\).

Table 1, number of fuel pellets per second required for one Megawatt of released combustion power.

As can be seen from the table, liquid or solid forms of fuel lead to far fewer required pellets (practical solid fuels have similar number requirements to liquids). This need not limit the effectiveness of the combustion system when it includes cryogenic and frozen fuel types. In a study of a similar system, Bates [8] for example, suggests a cryogenic hydrogen slurry as being a useful fuel form. The
information also suggests that a practical system might include different fuels and pellet sizes as already mentioned.

Obviously introducing the fuel into the mixing area involves some mechanical challenges. Liquid propellants would be the easiest to handle, as they can be sprayed into the duct from an injection plate. Solid pellets and gas capsules can be injected in using pressurised gas (which can also form part of the fuel system), using the differential pressures generated by combustion within the system or by other mechanical or electromagnetic means. It has also been suggested that the pellets could be projected ballistically into a continuous (unswitched) flow stream, but this has not been modelled fully yet.

Many parts of the system described above use established and well-tested technology (for example the expansion duct and exhaust). Others (like the fuel delivery system) are challenging, but nothing in them is outside standard mechanical engineering practice. However, there are two new key elements required to ensure that this system works. These are: a) That the pellet dynamics are such that the fuel will be well embedded into the flow and that b) the concept of the switchable front valve is feasible and achievable. The next sections investigate these key enabling technologies.

3. PELLET DYNAMICS

The dynamics of the pellets in the engine duct air-stream are critical to the operation of the system. The problem can be posed as a question: Can it be shown that the pellets become sufficiently embedded in the air-stream, so that their load is well-distributed throughout the mixing volume?

Some preliminary work was done on this question, and this was reported in our first paper [3] (which produced good results), but for a complete proof of concept (at least theoretically) a more rigorous approach was needed.

To establish the definitive answer to this question a four-pronged approach was adopted: 1) Simulating the pellet dynamics using Computational Fluid Dynamics (CFD); 2) Using published measured values of supersonic drag to predict their embedding; 3) Modelling them using theoretically determined drag values; 4) Finally, to calculate the “worst case” scenario based on the assumption that all the available fluid energy (kinetic, potential and thermal) is available as an accelerating pressure placed directly across the pellet. These approaches are considered one by one in the sections below.

1. Simulation using CFD.

Of all the approaches discussed here, simulation using CFD is likely to be the most accurate. This is because it uses the full Navier-Stokes model to capture a close approximation of fluid behaviour in all the various flow regimes that the pellet passes through. In the results presented below, the ANSYS Fluent commercial software package is used with a compressible flow model (a strongly coupled energy equation). Chemical reactions (hypersonic modelling) are not included as the flow being simulated in the mixing area is only mildly reacting at its upper velocity limit (Mach numbers of approximately 1.5 to 6.5) and combustion is not being modelled.

The pressure distribution is measured by splitting the surface of the pellet into areas equal to the pressure divisions generated by ANSYS and finding the pressure component in the $x$ (axial) direction by projecting this onto the $y$ (cross-sectional) plane. The corresponding force may then be found by multiplying the average pressure in the area by that projection. The total force on the pellet is then the difference between front and back faces. Side ($y$-direction) forces were assumed to cancel due to the symmetrical nature of the pellets. The acceleration of the pellet as it speeds-up is assessed by
simulating the conditions at 100 m/s velocity intervals. As an example of the data obtained using the CFD calculations, figure 2 below shows the total pressure on the surface of a pellet as it speeds up in a Mach 5 free stream engine mixing-section (approximately Mach 2 in the section in question). To confirm the results, full three-dimensional simulations of the pellets were also performed by an independent researcher, figure 3 shows examples of these simulations for pressure and pathlines around a pellet.

Figure 2, Simulations showing total pressure at four airflow velocities, of an accelerating pellet in a Mach 5 (free-steam) ASPIRE engine.

Figure 3, Examples of three-dimensional simulations used to confirm the basic results.
2. Calculations using published values of supersonic drag.

Knowing the measured simple (combined) coefficient of drag, one can use the drag equation to find the corresponding force produced:

\[ F_D = p_d C_D A \]

Where \( F_D \) is the drag force, \( p_d \) is the dynamic pressure, \( C_D \) is the drag coefficient and \( A \) is the frontal area.

Values for \( C_D \) have been published in several older papers [9 – 11] and more recent re-evaluations [12] as well as government, NACA and NASA reports [13, 14]. The results reported below use the worst case (lowest drag) figures reported. Almost all published values relate to stationary objects in a test stand (for example a wind or shock tunnel), rather than moving with the flow – this might conceivably lead to some discrepancies as in one case the drag is retarding movement and in the other causing it, however the results match up well with the CFD and other simulations. As with the other results, data points were taken at velocity intervals of 100 m/s throughout the acceleration curve.

3. Theoretically determined drag values.

A number of expressions for supersonic wave drag have been produced over the years – for example by Smith [15], Jones [16], Haack [17] and Sears [18]. These adopt similar strategies and give similar results. The wave drag thus calculated can be combined with known or estimated values of skin drag to produce an overall picture for simple symmetrical shapes like spheres and ellipses. Smith for example gives for an ellipse:

\[ C_D = \frac{128 \times \text{vol}^2}{\pi A l^4} K \]

Here \( \text{vol} \) is the volume of the object, \( A \) is the frontal area, \( l \) the characteristic length, \( K \) is a constant which can be calculated from the geometry of the ellipse, see references for further details. This is a semi-empirical approach and gives similar values to those obtained using measured values of drag and CFD.

4. Worst case scenarios.

In this case, we calculate the maximum possible acceleration, if all the available energy in the flow (kinetic and thermal) were converted into pressure and this were placed directly on the forward face of the pellet.

The stagnation temperature of a flow can be calculated from:

\[ T_s = T + \frac{v^2}{2C_p} \]

Where \( T \) is the normal temperature of the flow and \( C_p \) is the constant pressure heat capacity. The maximum velocity achievable by such a flow in an isentropic system is:

\[ v_{\text{max}} = \sqrt{2C_p T_s} \]

And we can convert this into an equivalent pressure:

\[ p_{\text{max}} = \frac{\rho v_{\text{max}}^2}{2} \]
As outlined in the previous sections, force can be calculated from this. Such an analysis could also be taken to its extreme by adding the static pressure to the result – which would be equivalent to there being a vacuum behind the pellet. Such an extreme scenario is impossible in practice, but serves as an exercise in *reductio ad absurdum* which is useful in proving the point.

As the pellet accelerates in the stream, its velocity relative to the flow decreases. Figure 4 shows what might be expected as it speeds up.

![Figure 4, expected velocity profile for pellet.](image)

To accommodate the constantly changing acceleration of the pellet, an iterative algorithm for calculating its velocity and displacement was used as shown in figure 5. This was supplied with the calculated or simulated accelerations from the four methods outlined above.

\[
\begin{align*}
\nu, \nu_r, t, s &= 0 \quad \text{\Delta t = iterative time step} \\
\text{Loop:} & \quad \text{Get current acceleration } a \text{ (from methods outlined)} \\
& \quad \nu_r = \nu_d - \nu \\
& \quad s = s + (v \times \Delta t) + (0.5 \times a \times \Delta t^2) \\
& \quad \text{Record and timestamp parameters} \\
& \quad v = v + (a \times \Delta t) \\
& \quad t = t + \Delta t \\
\text{End loop} \\
\text{Notes:} & \quad \nu = \text{current pellet velocity} \\
& \quad \nu_d = \text{flow velocity} \\
& \quad v_r = \text{relative velocity of pellet to flow} \\
& \quad s = \text{distance travelled by pellet}
\end{align*}
\]

![Figure 5, iterative parameter calculation algorithm.](image)

The parameter \( \Delta t \) was reduced until the results always converged on the same values – in practice a default value of 1 \( \mu s \) was used in most cases as this was an order of magnitude smaller than required.
A few simple extra lines can be added to this algorithm to calculate the progress of the main flow along the duct and its displacement relative to the pellet.

To calculate the pellet, intake and engine parameters in this research, a very typical standard engine-design used for reference calculations in many papers was used. This is the work published by Billig [7] and discussed by Anderson [19]. A table of this data is shown in table 2 (converted from imperial units and reformatted).

<table>
<thead>
<tr>
<th>Free stream Mach number</th>
<th>Altitude (km)</th>
<th>Mixing section Mach number</th>
<th>Pressure ratio</th>
<th>Mixing section pressure ($\times10^5$ Pa)</th>
<th>Mixing section temp (°C)</th>
<th>Mixing section temp (K)</th>
<th>Mixing section speed (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>14.6</td>
<td>1.53</td>
<td>8</td>
<td>1.0</td>
<td>140</td>
<td>413</td>
<td>620</td>
</tr>
<tr>
<td>4</td>
<td>17.5</td>
<td>1.95</td>
<td>16</td>
<td>1.28</td>
<td>244</td>
<td>517</td>
<td>879</td>
</tr>
<tr>
<td>5</td>
<td>20.0</td>
<td>2.36</td>
<td>25</td>
<td>1.37</td>
<td>339</td>
<td>612</td>
<td>1158</td>
</tr>
<tr>
<td>6</td>
<td>22.3</td>
<td>2.77</td>
<td>35</td>
<td>1.35</td>
<td>437</td>
<td>710</td>
<td>1454</td>
</tr>
<tr>
<td>7</td>
<td>24.4</td>
<td>3.14</td>
<td>47</td>
<td>1.31</td>
<td>533</td>
<td>806</td>
<td>1755</td>
</tr>
<tr>
<td>10</td>
<td>29.1</td>
<td>4.14</td>
<td>90</td>
<td>1.23</td>
<td>815</td>
<td>1088</td>
<td>2665</td>
</tr>
<tr>
<td>15</td>
<td>34.8</td>
<td>5.50</td>
<td>186</td>
<td>1.10</td>
<td>1327</td>
<td>1600</td>
<td>4239</td>
</tr>
<tr>
<td>20</td>
<td>42.0</td>
<td>6.65</td>
<td>314</td>
<td>0.69</td>
<td>1990</td>
<td>2263</td>
<td>5989</td>
</tr>
<tr>
<td>27</td>
<td>54.3</td>
<td>7.69</td>
<td>473</td>
<td>0.22</td>
<td>2609</td>
<td>2882</td>
<td>8292</td>
</tr>
</tbody>
</table>

Table 2, parameters in a common engine model by Billig [7].

The graphs below are for spherical pellets. It was found that, although irregularly shaped pellets exhibited different stability and tumbling characteristics, they did not show much difference in airflow penetration – this is almost certainly because the large Reynold’s number of the flow dominates the pellets behaviour. However, pellet size does make a difference - with penetration increasing with size. This is because flow-presentation area (and therefore force) increases as the square of pellet radius, but volume and therefore pellet-mass increases as the cube of radius. Therefore doubling the pellet radius increases frontal area by a factor of four, but mass by a factor of eight – and airflow penetration similarly scales (approximately doubles), ignoring the scaling-differences in skin and other drag components.

The graph in figure 6 shows the acceleration profile of a 2 mm radius pellet with a density of 850 kg/m$^3$ (this is typical of a heavier liquid fuel drop or a wax encapsulated light liquid or gaseous fuel) in a Mach 5 (free-stream) engine. A similar acceleration graph can be produced for all the other situations discussed. Only one graph is shown here out of the several dozen from the experiments, since they all show the same trends. The most likely values of acceleration are the three lines in the middle with the solid lines representing the limiting values (and the top - line A, being the absolute maximum theoretically possible). Line C terminates where reliable published figures for drag run out.
Figure 6, the acceleration profile of a typical 2 mm pellet in a Mach 5 free-stream engine.

Figure 7 shows the overall result for the whole flight-envelop for embedding of the same pellet in the air-stream. Here the y-axis coordinate shows the distance into the air-stream that the pellet has become embedded when the pellet itself has moved 10 cm from its original axial (x) position. It can be seen from this that even in the worst-case scenario the pellet is embedded 35 cm into the airstream (in other words when the pellet has moved 10 cm the leading edge of the airstream has moved 35 cm past it). Two lines are shown dotted because some parameters which they rely on are extrapolated from published data at lower speeds. In fact, even if we assumed that the entire initial (worst-case) pressure were across the pellet throughout its whole acceleration profile, the pellet would still embed itself in the stream sufficiently well – at Mach 3 this calculation yields an embedding of around 30 cm.

This exercise shows that the pellets will become completely engulfed in the airflow even in the most extreme scenarios and that the first of the two major issues discussed in the last paragraph of section two is (at least theoretically) not going to be a problem.
Figure 7, Embedding of 2mm radius liquid or encapsulated pellets in flow at flight Mach numbers.

Before finishing the discussion on pellets, a few other important issues would benefit from some attention. Some of these were not elucidated clearly in the first papers on the method, others result from audience questions arising when the system has been presented at seminars in other institutions, companies and universities.

One important aspect of the system revolves around the mechanisms of pellet break-up and vaporisation. Fortunately these important parameters can be controlled closely in a number of ways – adding greatly to the usefulness and flexibility of the system. Controlling the disintegration time of the pellet allows fine tuning of pellet penetration and therefore the whole combustion process.
The mechanisms of breakup and fuel dispersion in simple uncoated pellets has been studied and reported by Bates [8] in his work on a similar system (Bates suggested, in several papers, a system where pellets or strings of solid or semi-solid fuel are injected into a continuous stream from the duct walls). He has done extensive research, including experimental testing, on the topic and particularly pointed out the advantages of a solid/liquid slurry as a fuel (particularly studying He/H₂ cryogenic mixtures). His calculations from the latent heat of the H₂ slurries show that for small 1mm diameter pellets of this fuel only 1.2 J of heat was required for complete vaporisation. He also found that the following ratio is particularly important in the breakup process:

$$\frac{\rho_f v_f^2}{\rho_a v_a^2}$$

Here the subscript $f$ refers to the fuel pellet and $a$ to the airflow. The ratio must be greater than six for rapid effective breakup and vaporisation and these characteristics can be controlled using this ratio and pellet size. Bates found, in his experiments and calculations - which used pellets of similar dimensions to those already discussed - a complete vaporisation distance of around 30cm at flow conditions similar to the ones already mentioned. Aside from this simple scenario however, fuel can be encased in a wide variety of refractory coatings or shells – which allows breakup time to be tightly controlled.

The current authors [20, 21] have also done extensive research into the use of radio-frequency and microwave radiation in scramjet thermal systems including its use in aiding fuel release and mixing. Electromagnetically excitable material can be added to the pellets and this allows their disintegration to be controlled by a duct mounted radiation source (inductive heating could potentially be used in a similar way).

The use of pellets also allows innovative solutions to the ignition and combustion stability issues which traditionally plague scramjet design as a second-order issue. Pellets may be loaded with hypergolic or pyrophoric fuels, combustion catalysts or electromagnetically activated ignition agents which operate in a similar way to those mentioned in the previous paragraph. This potentially allows an even flame-front to propagate across the combustion space – something difficult with conventional mechanisms.

Let us now consider the second point highlighted in the last paragraph of section two – the design of the airflow switching mechanism.

4. INTAKE DESIGNS

Several options for switching the airflow into and away from the inner duct were discussed in the first paper [3]. These included using a system of shutters as a valve and a rotating wedge diverter and many other possibilities can be readily imagined. However, simulation and calculation has shown that one particular design seems to work well and is also a particularly elegant solution – a rotating nose-cone or rotor. The principle of this is shown in figure 8.
The nose-cone consists of a cone shaped external structure with standard design scramjet intake-ducts cut into it around its periphery. This structure continues internally within the body of the engine to form the isolator, mixing, combustion and expansion sections of the system as part of a rotor. Figure 9 shows a simple configuration for the front (external) portion of the rotor in 3 dimensions which was used in CFD simulations.

Within this basic idea there are many different configurations possible. These include the number and shape of the intake ducts. Some alternatives for these are shown in figure 10.
Figure 10, some of the many possible intake configurations, viewed from the front

Figure 10a shows a curved, gently changing duct-profile, where the top surface of the intake is part of the engine cowl. The idea behind designs like these is that, as they turn they provide a gradually changing profile to the incoming airflow and therefore avoid an abrupt disruption. However, their complex shape makes them difficult to design with traditional methods and it is planned to explore this using genetic algorithms at a later date. The intake compression surface may be a stepped topology as shown in figure 9 or an isentropic one. Figure 10b shows a similar structure to 10a, but with the entire intake build into the rotor (and not using the engine cowl). This allows the top of the duct to be flat, making design simpler. Figure 10c shows a very simple square duct (like that in figure 9 - of which there are many published designs). Figure 10d shows a staggered design which is a half-way house between b and c, but is easier to design than a gently curving structure. Figure 11 shows the cross-section of a design for the forward part of an intake similar to figure 9 and 10c on the left and a modified staggered design on the right.

Figure 11, cross-sections of some designs explored for external rotors.

An alternative to having the whole intake rotate, is to keep the outer (hypersonic) compression surface stationary and have only the inner part (the lower-speed, supersonic section) of the surface rotate. Alternatively, the isolator section of the engine could rotate. These options are shown in figure 12.
Similarly, the portion of the rotor internal to the engine has many possible configurations including partitioned or continuous annular ducts. It should be noted, that although these designs are presented here as axisymmetric, it is equally possible to create linear engine topologies using the same or somewhat modified ideas.

The power required to turn the rotor can be calculated from the aerodynamics of the situation. Consider an undiverted airflow represented by an average velocity vector $v_a$ and also another vector $v_b$ representing the diverted flow’s average velocity. The magnitude of the difference between these two velocities represents the effect on the airflow of the switching mechanism. The energy required to produce this change is therefore (assuming a loss-less system):

$$E = \frac{|v_a - v_b|^2 \rho V_D}{2}$$

Where $\rho$ is the average density of the flow and $V_D$ is the duct (or ducts) volume. The power used by the system depends on the speed of rotation of the nose-cone:

$$P = \frac{|v_a - v_b|^2 \rho V_D \omega}{4\pi} = \frac{|v_a - v_b|^2 \rho V_D R}{120}$$

Where $R$ is the rotational speed in r.p.m and $\omega$ is the angular velocity in rad/s. With careful design these values are in the tens to hundreds of kilowatts for practical engine powers in the region of megawatts to tens of megawatts.

In fact this analysis can also be extended to calculate the losses associated with a small rotation of the duct as air passes through it, as shown in figure 13. Here the velocity difference used in the equation represents the edge effect of the duct rotating. It may be seen intuitively from this, that ratio of the speed of the duct movement to the speed of an air particle is important – if the speed of a particle through the duct is much greater than the speed of the edge of the duct then the effect of duct rotation is minimised. This leads to the conclusion that, in this respect anyway, many ducts around a relatively slow-moving rotor would perform well.
Having discussed some of the many structures possible, the aim of this section is simply to show that such a system is theoretically plausible. In other words: Can a rotating ducted valve of this nature redirect the airflow in a manner consistent with the ASPIRE engine’s operation? This would show that the system is possible at the theoretical level – and serve as a basis for practical verification and design.

Fortunately there is a good way to describe the effect of a rotating intake duct on the airflow. As already stated above, if an incoming air-particle is moving much faster than the duct is rotating, then the duct appears nearly stationary to the particle (another way of looking at this is that the problem of flow through a moving duct is similar to a stationary duct on a yawing vehicle). Existing scramjet theory provides a way of assessing this, as figures can be calculated which give the area constriction required to cause the duct to unstart or stall. This happens when the constriction in duct size is sufficient for the intake to disgorge a normal shock-wave. The theory of this is covered in Segal [1] on pages 98 – 100 and in Heiser [22] on pages 250 – 251. In the case discussed here the constriction is caused by the inlet area decreasing in effective size due to its rotation.

In this analysis several assumptions are made. These include that the duct is smooth, uses isentropic compression and that edge effects do not cause a shock sufficient to disrupt the main airflow. Viscous effects (like the boundary layer) are also ignored. Segal provides several references which discuss these effects and reports a measured case where these other phenomena caused an error of approximately 27% in the calculated figures. The maximum area constriction is calculated by first calculating the Mach number of the post normal-shock flow:

$$M_s = \sqrt{\frac{1 + \frac{\gamma - 1}{2} M^2}{\gamma M^2 - \frac{\gamma - 1}{2}}}$$

Where $M$ is the Mach number of the duct we are considering and $\gamma$ is the ratio of specific heats. The restriction of area before unstart in an isentropic duct is then given by:

$$\frac{A_0}{A_R} = \frac{1}{M_s} \left( \frac{2}{\gamma + 1} \left( 1 + \frac{\gamma - 1}{2} M_s^2 \right) \right)^{\frac{\gamma + 1}{2(\gamma - 1)}}$$

Where $A_0$ is the normal duct area and $A_R$ is the restricted area. Table 2 gives the allowable constriction ratio over a range of operational Mach numbers. As can be seen there is an asymptotic progression towards a value of approximately 1.66.
Table 2, area constrictions necessary for unstart in an isentropic duct.

We can see from these figures that, as long as the allowable apparent duct-area decrease caused by rotation is less than around 28% at Mach 3 or 40% at Mach 25, during the time taken for an air particle to transverse the intake, unstart should not occur. Of course this is the limit, in a real system we would probably not wish to exceed 10% of this value.

A simplified topology of the system is visualised in figure 14. In this diagram $a$ is the duct width, its height is $b$ and its depth $c$. The letter $d$ designates the distance moved by the edge of the duct at velocity $V_d$ during the time $T$ that an air particle takes to transit the intake at velocity $V_a$. The duct is shown as rectangular because it is conditions at its mouth which limits its performance.

![Figure 14, stylised topology of duct](image)

We can see from this that $T = c/V_a$ and that the distance $d$ swept by the edge of the duct during the time taken for a particle to pass through is $TV_d$. The (shaded) area swept by the duct is therefore $bd = bV_aT = bV_d(c/V_a)$. From this and designating the ratio $A_0/A_s$ as $R$, for the duct to be operational:

$$ab - \frac{bcV_d}{V_a} > \frac{ab}{R}$$

Or rearranging:

$$V_d < \left(1 - \frac{1}{R}\right)V_a \frac{a}{c}$$

We can explore the effect of this by putting in some typical figures. An intake width to length ratio is around 0.06 for a long system, so, for example at Mach 5, $V_d$ must be less than 24.36 m/s. Now, as already stated, we probably do not wish to exceed 10% of this value, which is 2.4 m/s. In an engine...
with a 1m radius rotor this corresponds to an RPM of around 23. This figure, making the same assumptions, ranges from 10 RPM at Mach 3 to 134 at Mach 20. Table 3 gives the duct values, making the same assumptions and rounded down to the nearest m/s for the Mach numbers given in table 2. For practically sized engines with several ducts this is perfectly compatible with the fuel injection figures given in table 1 – for example at Mach 5, for 1 MW of power, approximately 800 2mm pellets need to be deposited within the swept duct area, if there were four ducts on the nose-cone, this would be around 200 pellets per 2.4 m of swept area.

<table>
<thead>
<tr>
<th>Mach number</th>
<th>3</th>
<th>5</th>
<th>10</th>
<th>15</th>
<th>25</th>
</tr>
</thead>
<tbody>
<tr>
<td>Duct velocity V_d (m/s)</td>
<td>10</td>
<td>24</td>
<td>61</td>
<td>100</td>
<td>197</td>
</tr>
</tbody>
</table>

Table 3, maximum duct velocity values for an a/c ratio of 0.06.

In many ways these figures are quite conservative. For example the output velocity of the intake has been chosen for the calculation above – in fact this only exists at its back-end and the modal average would be more accurate (and Heisler uses the free-stream velocity - since the duct does not technically stall until the normal-shock appears at the front). The derivation also ignores the progression of the duct to the right (in figure 14) which sweeps out an area which is equal to that occluded on the left – this would effectively half the area and double the allowed duct speed. However, the aim of this analysis is to show that the system is practical and so only the worst-case scenario is described – but these points should be enough to compensate for ignoring the viscous effects already discussed.

It is also worth pointing out at this juncture, that the above analysis gives some interesting pointers to the performance of the engine. For example: Both the intake and pellet calculations show that the engine works better at higher speeds – at lower Mach numbers there is less pellet penetration and the nose-cone must rotate more slowly (making injection timing more critical). This could be overcome by using larger pellets and more side-wall injection – and this is compatible with the lower airflow speed and better mixing at these speeds.

At the rotor speeds calculated above, transitive disruptive edge effects are minimal. During the passage of an air particle, the edge sweeps out a wedge (it hasn’t moved when the particle enters and has swept out a distance d when it leaves). This could cause a vertically orientated oblique shock emanating from the edge into the intake. However the wedge angle is very shallow and any shock therefore weak, close to the duct wall and insufficient to disrupt the main flow.

So that the dynamics of the ducts could be observed in more detail and some of the results discussed above could be confirmed, CFD simulations of the intake system were also undertaken. These used the same setup described in section 3. Given that the aim was a proof of principle, to simplify the problem a simple single speed duct was chosen. Mach 5 was selected as the free-stream speed because other published scramjet designs where available for comparison and this speed is achievable for testing on small sounding rockets. Only simple square and staggered intakes were designed and simulated – as shown in figure 10c and 10d. Finally, and again to keep the design simple, stepped (two or three stage) compression ramps were designed. Obviously this does not make for a practical engine, but serves to prove the main points.

Several simulations were used to observe operation of the duct – these included: a 3D simulation of the nose-cone system using setup similar to that already shown in figure 9; 2D sectional simulations where each section of the nose-cone was simulated slice-by-slice as it turned and then animated to
show flow in cross-section; finally moving-stream simulations were also attempted. Figure 15 shows one of the 2D cross-sections produced (half the cone shown) in the course of one of these investigations.

![Contour plot of Mach 5 open intake simulation](image)

**Figure 15**, simulation of Mach 5 open intake.

## 5. DEVELOPMENT ROADMAP

At this point there is little more theoretical proof-of-concept work that can be done on pellet dynamics – all our simulations and modelling indicate strongly that these will operate as expected. In the next stage of the project the system principles outlined above (pellet penetration) need to be confirmed experimentally.

Further theoretical work is desirable, though, to choose between intake topologies and investigate the action of these more thoroughly. In particular the aerodynamics of the different rotary duct types may be further illustrated with CFD simulations. It is intended to investigate this and publish the results in a follow-up paper.

Practical confirmation of both operational aspects may be done with two experiments. The first is to show that pellet embedding is as expected. This can be achieved simply in a standard shock-tunnel with a pellet dropping device and high-speed camera. The second experiment is to confirm the practical operation of the rotating nose-cone. It may be also possible to do this in a shock-tunnel – however only if the gas burst is of sufficient duration to observe the effect of rotation. One alternative would be testing in a high-temperature, high-speed flow (the outlet of a rocket system), several organisations offer this, although obviously the flow parameters are different in such a stream. Another alternative is using a small sounding rocket for in-flight testing – both these latter alternatives would be more expensive than tunnel testing.
Once these two aspects are confirmed, a single-speed functional demonstrator launched on a small sounding rocket would be a logical next step. Mach 4 or 5 are suitable and achievable speeds for such a test (and even some amateur rocket groups in the USA have achieved these velocities).

Several possible designs for these options have already been devised. Figure 16 shows a 3D printed rendering of a test nose-cone. These designs can be machined from the same software files using CNC for actual testing.

Finally, a full demonstrator for use with a larger vehicle could be designed and tested. A road-map for developments is shown in figure 17.

**Figure 16, 3D printed rendering of an example test design for the external nose-cone.**

**Figure 17, development road-map**
A projected full-price cost for phase A (the experimental validation of the key technologies discussed in this paper) would be in the region £100k - £200k at current costs. Phase B (testing a simple small design on a sounding rocket) would be around £250k. This is an important milestone, because at this point all the key operating principles of the engine will have been verified and tested. The cost of phase C is difficult to estimate as it requires much more serious development and testing of hardware and the involvement of government or large industrial bodies. In addition to this, a sustained on-going simulation and theoretical effort to refine current designs and produce new ones would be in the region of £40k per annum.

6. OTHER ADDITIONS AND APPLICATIONS

Numerous other variations and additions to the basic design have been investigated. A few of the more important ones, not described in the previous papers are outlined in this section.

It has already been mentioned that several other options for flow diversion are worth investigating (and many others are theoretically possible). One of these discussed above is switching the flow after the hypersonic intake section, at the lower-speed compression or at the isolator stage. This may have advantages, including less chance of flow instability and greater ease of implementing a variable topology inlet. The reverse of this philosophy is to switch the air before the (stationary) intake using a rotating plain or Busemann topology.

Another area of interest is the adaption of the design to use a variable cycle. For example, the nose-cone could be locked in place using a clutch system and the internal ducts used in a pure rocket mode for final transition to the space environment. A more complex option for multispeed operation would be an internal concentric rotor within that already discussed, which would rotate slowly as the engine passes through its flight envelop to expose a gradually changing compression surface through an open aperture in the outer rotor; the rotor assembly could also change its position relative to the cowl through the flight envelop as implemented in the Pratt and Whitney J58 engine in the Lockheed SR71.

The system described here may be used for innovative propulsion purposes other than fuel mixing - for example to facilitate the introduction into the airflow of [4]:

- MHD active substances - perhaps to aid ionisation or increase flow conductivity.
- Electrically active substances - to aid inductive or similar electrical heating or acceleration.
- Chemical catalysts to aid (for example) combustion.
- Compounds which lase.
- EMA active substances.

7. CONCLUSIONS

The aim of the ASPIRE engine is to overcome the main problem with scramjet operation – poor air-fuel mixing. However, the technique also offers an unparalleled degree of flexibility in the distribution and release of the fuel through the airflow. It is also an enabling technology for other advanced propulsion systems because it can distribute a wide variety of substances other than fuel into the flow.

This paper shows that the key untested parts of the system are at least theoretically feasible. In fact the flexibility of the idea is such that that there are a variety of possible topologies for the whole system which afford many options for overcoming technical issues. It is intended to follow this paper with another, analysing the intake section with more thoroughly through CFD and/or other appropriate modelling.
There is also a clear road-map available between the current research and results and a full experimental confirmation of the engine’s operation. The route outlined would be very cost-effective, with comprehensive verification only costing in the region of £300k - £500k.

Acknowledgements: Thanks to Callum MacDonald, who undertook the confirmatory three-dimensional pellet CFD simulations and Claire Gerrard and Murdo Smith for helpful suggestions and proof-reading.

References

17. W. Haack, Geschossformen kleinsten Wellenwiderstandes, Bericht 139 der Lilienthal-Gesellschaft fur Luftfahrt.