Computational Assessment of Inlet Backflow Effects on Rotating Detonation Engine Performance and Operability

Daniel E. Paxson¹ and Kenji Miki²

NASA Glenn Research Center, Cleveland, Ohio, 44135, USA

The performance impact of flow reversal at the inlet of an airbreathing rotating detonation engine (RDE) is investigated using 2 and 3-dimensional computational fluid dynamic (CFD) simulations. Flow reversal, or backflow, occurs in RDE inlets in the high-pressure region directly behind the rotating detonation front. This is also where most of the engine thrust or pressure gain is produced. The amount of backflow relative to throughflow depends on the inlet design. For the present work, a simple annular ‘slit’ design is used. The simulations are idealized in several ways, including that fuel and air are premixed, but prevented from reacting when within the inlet region. The results indicate that even with idealizations, the impact of inlet backflow on pressure gain can be substantial. The simulations also reveal an intriguing instability that develops in certain configurations. The mass flow rate into the RDE begins to oscillate at a regular frequency that is substantially less than the detonation rotational frequency. This is accompanied by oscillations in the detonation height. The oscillation amplitude grows over time until the detonation ultimately fails. Both the performance and instability results emphasize the need for carefully designed RDE inlets that provide low loss when flow is in the forward direction, but high resistance when the flow is reversed. Development of such high-diodicity inlets is critical to achieving pressure gain in airbreathing RDE’s.

Nomenclature

\[ A = \text{cross-sectional area} \]
\[ a = \text{speed of sound} \]
\[ C = \text{compressor characteristic} \]
\[ cw = \text{channel width} \]
\[ Dm = \text{mean diameter} \]
\[ l = \text{compressor characteristic duct length} \]
\[ L = \text{axial length} \]
\[ mfr = \text{mass flux rate} \]
\[ P = \text{total pressure} \]
\[ p = \text{static pressure} \]
\[ PG = \text{Pressure Gain} \]
\[ Rg = \text{specific gas constant} \]
\[ S = \text{valve characteristic} \]
\[ T = \text{temperature} \]
\[ V = \text{plenum volume} \]
\[ x = \text{normalized azimuthal distance} \]
\[ y = \text{normalized axial distance} \]

¹ Aerospace Research Engineer, Research and Engineering Directorate, AIAA Associate Fellow.
² Aerospace Research Engineer, Research and Engineering Directorate, AIAA Senior Member
I. Introduction

The design of an effective inlet for an airbreathing rotating detonation engine (RDE) is a critical technology challenge. Aside from the need to rapidly mix air with injected fuel, the inlet must provide low total pressure loss to fluid entering the annular channel while simultaneously providing a thrust surface and preventing backflow into the inlet manifold in the high-pressure region immediately behind the azimuthally propagating detonation. These competing objectives necessarily result in inlets which are a compromise. Compounding the challenge is the issue of separating and measuring the impact of the highly coupled forward and backflow losses. This is extremely difficult to do in the laboratory since existing measurement technology cannot isolate inlet losses from other loss mechanisms. Furthermore, experiments which attempt a parametric approach to optimizing inlet design require prohibitive resources since the parameters space is large. Meeting the inlet design challenge is nearly as difficult with high-fidelity computational fluid dynamic (CFD) simulations. These also require significant resources and time; so much so that they preclude practical design through parametric search. Moreover, the data generated from these simulations can be so detailed and voluminous that condensing it and utilizing it (i.e. synthesis) is prohibitively difficult.

In order to assess mission benefits for RDE applications and motivate further research (and develop the resources needed for the kinds of efforts just described), models and simulations must first reasonably quantify the performance impact that can be expected from a generic inlet. And, they must do so using modest computing resources. This necessitates a simplified approach to modeling inlets [1, 2]. The Ref. [1] effort used a thermodynamic model of an RDE with algebraic, discharge coefficient-type sub-models to assess forward flow loss and loss from backflow. The conclusion drawn from exercising the model was that these two losses alone were among the most significant in terms of lost performance relative to an ideal RDE. The Ref. [2] effort was based on a quasi-two-dimensional (Q2D) computational fluid dynamic (CFD) code. The inlet forward flow loss and backflow loss were assessed using instantaneous mixing sub-models that were implemented in the code boundary condition routines. This work compared the code output to experimental measurements and did not explicitly isolate the inlet losses from others (e.g. heat transfer, fuel and air mixing, etc). The experimental inlet area was also small in comparison to the RDE channel area. The amount of backflow was thus very small, while the forward flow losses were large. As such, the experimental results were only useful as a forward flow loss model test. Subsequent work [3, 4] considered the forward flow loss indirectly, but the simulations assumed a semi-ideal RDE where no backflow was allowed at all.

The present work seeks to maximally isolate the impact of the combined inlet backflow and forward flow loss mechanisms by using the Ref. [2] Q2D simulation to model a particular RDE configuration. The simulation is idealized in the sense that other loss mechanism sub-models are deactivated. The results are compared to the output from a full three-dimensional (3D) CFD simulation [5-7] of the same RDE configuration. The 3D computational domain includes an actual inlet design and an inlet manifold. The 3D simulation is run in an inviscid mode, with
simplified reaction equations, and a relatively coarse grid in order to arrive at converged solutions as quickly as possible. The comparisons include flowfield features and performance metrics. It is shown that Q2D inlet loss submodels produce reasonable results for a basic inlet design. The results also show that the impact of backflow on performance is indeed substantial for basic inlet designs.

A second focus of the present work is to document and validate by comparison a dynamic instability that occurs in the simulations when the inlet design allows substantial backflow. The instability was encountered unexpectedly during the present investigation and it influenced the direction of the work. The presentation of results in the paper follows the chronology of events. Thus, some limit-cycle performance results are first presented. These are followed by unstable results as they occurred. Finally, more limit-cycle performance results are shown from RDE configurations where the instability is mitigated. Although this instability is exhibited in simulations that are somewhat idealized (e.g. the air and fuel are assumed premixed, the flow is inviscid and adiabatic, and the combustion chemistry is greatly simplified), it is anticipated that it has potential real world relevance to operational laboratory RDE’s.

The Q2D and 3D codes are described in brief since they have been well documented in the literature. The particular configuration details are then presented. Results follow in the order mentioned above. Although the instability is not fully understood as of this writing, some discussion on its possible source and attributes are presented. The totality of the work demonstrates the criticality of high backflow resistant inlets with simultaneous low forward flow loss to the advancement of RDE technologies.

II. CFD Code Descriptions

Both of the codes were run in an inviscid mode unless otherwise stated in the paper. The incoming fuel (C₂H₄) and air were assumed to be premixed with an equivalence ratio of 1.0.

A. Q2D Code

The basis of the Q2D code is a high resolution algorithm that integrates the quasi-two-dimensional, two-species, reactive Euler equations with source terms [5-7]. The code adopts the detonation frame of reference and deliberately utilizes a coarse grid (i.e., it is diffusive) to eliminate the highest frequency unsteadiness (e.g., detonation cells, Kelvin-Helmholtz phenomena). The result is a flow field solution that is invariant with time when converged. The working fluid is assumed to be a single, calorically perfect gas. The source terms contain sub-models which govern the reaction rate, momentum losses due to skin-friction, and the effects of heat transfer to the walls. Only the reaction rate source term was active for this work. The finite rate reaction model is simple. The coefficient that multiplies the reactant fraction is a fixed value if the fluid temperature is above a prescribed ignition temperature, and zero otherwise. The fixed value of the coefficient is set to a high value if the pressure is more than twice the manifold pressure, and a lower value if the pressure is below this threshold. The high value is empirically determined such that detonation is assured. The low value is approximately 35 times less than the high value.

The governing equations are integrated numerically in time using an explicit, second-order, two-step, Runge-Kutta technique. Spatial flux derivatives are approximated as flux differences, with the fluxes at the discrete cell faces evaluated using Roe’s approximate Riemann solver. Second-order spatial accuracy (away from discontinuities) is obtained using piecewise linear representation of the primitive variable states within the cells. Oscillatory behavior is avoided by limiting the linear slopes.

Considering an ‘unwrapped’ RDE where the non-dimensional circumferential direction is x, and the axial direction is y, the following boundary conditions are imposed. At \( x=0.0 \) and \( x=1.0 \), periodic (aka symmetric) conditions are used. These ensure that the x-dimension of the computational space faithfully represents an annulus (which is continuous and has no boundary). At the exit plane, constant pressure outflow is imposed along with characteristic equations to obtain density and axial velocity for the image cells. If the resulting flow is sonic, or supersonic, then the imposed pressure is disregarded. If, in addition, the upstream flow is supersonic, then pressure, density, and axial velocity are extrapolated from the interior. The possibility for a normal shock solution whereby supersonic outflow jumps to subsonic is also accommodated. The x-velocity (azimuthal) component is extrapolated from the interior at each exit boundary location. At \( y=0.0 \) (the inflow face), partially open boundary conditions are applied as described and validated in Ref. [8]. This face is presumably fed by a large manifold at a fixed total pressure, and temperature. The manifold terminates at the face and is separated from it via an orifice. The ratio of orifice flow area to RDE annulus area, \( A/A_{th} \), is generally less than 1. If the interior pressure is less than the manifold pressure, \( P_a \) then inflow occurs. The boundary condition routine determines pressure, density, and axial velocity for the inflow face image cells subject to a momentum (total pressure) loss model which depends on the mass flow rate and the value of \( A/A_{th} \). It can accommodate a scenario where the orifice becomes choked. The x-velocity component prescribed during inflow,
and it is here that a reference frame change is implemented. Rather than specify zero velocity (i.e. no swirl) which is the laboratory or fixed frame condition, the negative of the detonation speed is prescribed instead.

If the interior pressure along the inlet face is greater than \( P_m \), as might be found just behind the detonation, then there will be backflow into the manifold through the orifice. The boundary condition routine can accommodate this as well.

In regions where inlet backflow occurs, the total mass and enthalpy of backward flow are averaged over the circumferential backflow span (recalling that in the steady detonation frame of reference, time is simply span divided by detonation velocity). When the interior pressure subsequently drops below \( P_m \) and forward flow resumes, all of the mass that flowed backward is sent back into the RDE at the same average enthalpy that it exited. Once this mass has re-entered, the prescribed manifold premixed air and fuel mixture enthalpy is used.

Although the model assumes that premixed air and fuel enter through the inlet, the reality of most RDE experiments is that fuel and air are injected separately. This raises the possibility that some finite time (and associated convection distance) is required to mix before they will react. To cursorily examine the effect of this reaction delay, the code allows a user specified number of axial computational cell rows near the inlet that do not react, even though the threshold temperature is reached.

### B. 3D Code

The 3D code is a version of the National Combustion Code, OpenNCC [5-7]. It is designed for unstructured grids (i.e., any mix of three-dimensional elements: hexahedral and tetrahedral mesh) and massively parallel computing. The numerical approach is a dual time-stepping procedure. The solution implicitly advances in physical time and the explicit four-stage Runge–Kutta scheme is called in pseudo time. The transport equations are spatially discretized using a cell-centered finite-volume method. The second-order Advection Upstream Splitting Method (AUSM⁺-up) scheme [9] is used to calculate the inviscid flux. Numerical dissipation is minimized using the minmod function as the limiter of the Monotonic Upstream-centered Scheme for Conservation Laws (MUSCL) scheme. The code has viscous and turbulence modeling capability, but it was not used in this work.

A thermally perfect gas mixture is assumed for the working fluid. A two-step, six-species global combustion mechanism was used [10]. However, due to the course nature of the grid, and the premixed assumption, it was found that a pressure factor had to be applied to the combustion mechanism. If the pressure in the computational domain was more than twice the inlet manifold total pressure, the reaction was allowed to proceed normally. For pressures below this threshold, the reaction rates were all reduced by a factor of 100. Without this factor present, deflagration processes would dominate the flowfield and any initialized detonation would soon be extinguished. Counterintuitively opposing this pressure limit, it was found that the reaction rate for the first step of the reaction needed to be increased by a factor of 5 in order to achieve stable detonation propagation. It is not known why this factor was necessary. However, it is noted that the reaction mechanism was developed to study conventional combustors and not detonations. Combustion reactions were disabled upstream of the inlet trailing edge in order to prevent propagation of a detonation into the inlet manifold. This also crudely simulates non-premixed configurations which, as mentioned, are the norm for RDE’s.

### III. RDE Configuration Description

The particular RDE simulated is shown to scale in Fig. 1. It has a mean diameter, \( D_m \) of 5.5 in. The channel length, \( L \) is 6.09 in. The channel width, \( c_w \) is 0.5 in. The channel exit area, \( A_e \) is 65% of the channel area, \( A_{c, \text{ch}} \). The transition from \( A_{c, \text{ch}} \) to \( A_i \) is a smooth one which commences at 83% of the channel length. The channel inlet area, \( A_i \) is 50% of the channel area unless otherwise stated. For the 3D simulation, the inlet length is 0.81 in. The Q2D simulation does not have an inlet length since the inlet is modeled as a simple orifice. The left side of Fig. 1 shows a slice through the left half of the device in the radial and axial directions. The right side of the figure shows the shaded surfaces that define the device in 3 dimensions. The figure also represents the 3D computational domain. The inlet and exit areas chosen are similar to those that have yielded significant pressure gain in previous simulation studies [4, 11].

The computational domain is a regularly spaced, 200 circumferential by 70 axial cell grid for the Q2D simulation. It is approximately 800 circumferential by 200 axial by 15 radial (in the channel region) for the 3D simulation. However, the coordinate system is actually Cartesian. The grid is structured in the channel and inlet regions, and unstructured in the manifold and entrance regions. A coarser grid is used in the manifold since high resolution is not required there.

The operating conditions for the RDE are a manifold total temperature of \( T_m = 540 \text{ R} \), an exit static pressure of \( p_e = 14.7 \text{ psia} \). For the Q2D simulation, whose inlet boundary conditions are applied at the trailing edge of the Fig. 1
inlet section (i.e. the Q2D computational boundary), a constant manifold total pressure, \( P_m = 290 \text{ psia} \) is imposed. For the 3D simulation, a constant mass flux rate, \( m_{fr} = 43.9 \frac{\text{lbm}}{\text{ft}^2 \cdot \text{s}} \) is applied at the furthest upstream face (aka, the head end). The constant mass flux boundary condition was used in the 3D simulation because this code is not equipped to handle the inevitable flow reversals that would occur if a constant pressure boundary condition were used. And even if a check valve-type procedure were implemented to prevent such flow reversals, a constant pressure boundary condition applied at this location was found to require exceptionally long run times in order to achieve a limit cycle. The Q2D simulation assumes a single calorically perfect gas with a gas constant, \( R_g = 53.67 \frac{\text{ft} \cdot \text{lbf}}{\text{lbm} \cdot \text{R}} \), and a ratio of specific heats, \( \gamma = 1.293 \). These values were obtained using the methodology described in Ref. [12]. The 3D simulation accommodates multiple species, as mentioned.

**IV. Results and Discussion**

**A. Preliminary Q2D Limit Cycle Performance**

The Q2D simulation limits inlet backflow through a user specified diodicity constant defined as follows.

\[
\delta = 1 - \frac{A_{\text{backflow}}}{A_{\text{forward}}} \quad (1)
\]

Here, the inlet area during backflow is assumed to be an effective one manifested fluidically by the design of the inlet. It is always less than or equal to the effective area of forward flow.

A typical limit cycle solution for \( \delta = 0.4 \) is shown in Fig. 2 using contours of temperature displayed in an unwrapped view of the RDE annulus. The reference temperature by which the data is normalized is \( T^* = 520 \text{ R} \). The axial and azimuthal directions, \( x \) and \( y \) have been normalized by RDE circumference at the mean diameter. The horizontal dashed white line in the figure at \( y = 0.015 \) represents the user-defined standoff distance below which chemical reactions are prevented. The effect of this standoff distance on computed performance is generally small. It is implemented in this work in order to ensure that when backflow does occur in the inlet, it is unburned fluid that is pushed into the manifold region rather than hot, post-combustion gas. As will be seen later, this is consistent with the 3D results at the leading edge of the inlet. It is not known if this unburned backflow is what actually occurs in RDE’s since no known experiments have focused on measurements in this region of the RDE. The computed detonation speed in Fig.
2 is 78% of the theoretical Chapman-Jouget (CJ) speed for a single calorically perfect gas and 86% of the CJ speed for a thermally perfect equilibrium gas mixture [13]. The Q2D code typically predicts detonation speeds below the CJ value due to its crude grid and simplified reaction model. A discussion of why such a simplification might closely match physical reality is presented in [14].

Also shown in Fig. 2 is a normalized mass flux rate distribution at the inlet (normalized by $\rho^*a^*$). Here it can be seen that a significant portion of the cycle is occupied with backflow, the backflow has been heated by an oblique shock, and the backflow must return to the interior. The backflow affects the cycle performance in at least three ways. First, it heats the unburned mixture which, after reentry and processing by the detonation yields a lower final pressure. Second, it essentially acts as a type of blockage to throughflow. The cycle response is more suction during the refill stage, which in turn yields lower pre-detonation pressure. Third, it reduces the size of the high-pressure region behind the detonation where a significant amount of the thrust is generated. Three colored streamlines are shown in the Fig. 2 contour to illustrate the backflow path. Fluid between the yellow and white streamlines enters the RDE from the manifold and passes directly through (i.e. is processed within a quarter of a wave revolution by) the detonation. Fluid between the red and yellow streamlines enters the RDE, is pushed back out the inlet, re-enters at an elevated entropy, and eventually passes through the detonation. Thus, there can be significant delay between initial entry of this fluid into the RDE and processing by the detonation.

The performance impact is shown quantitatively in Fig. 3 where pressure gain and backflow rate as a percentage of total flow rate are plotted as functions of prescribed diodicity. The pressure gain is measured using the Equivalent Available Pressure (EAP) methodologies described in Ref. [11]. EAP is a single exit total pressure meant to represent the availability of the exit flow for work or thrust production. Reference [11] presents two ways to calculate EAP. One is based on ideal specific thrust that could theoretically be developed (subscript i). The other is based on a measured thrust, and the often-valid assumption that the flow at the exit plane throat is choked. Since this method can be used with laboratory RDE’s, it is subscripted with exp. The experimental method is more conservative, but also easier to calculate in a computational setting. Both methods are shown here, but only the experimental method is used with the 3D results. Pressure gain is defined as follows.

$$PG = \frac{EAP}{\rho_m} - 1$$

The trends of Fig. 3 suggest that diodicity, and the associated backflow can have a profound impact on RDE performance. However, it is important to remember that both the forward flow total pressure loss and the backflow loss in the Q2D simulation are governed by sub-models of a more complicated flowfield. Part of the purpose of the 3D simulation effort was to either validate or modify these sub-models appropriately and to assign appropriate values of $\delta$ to specific inlet configurations. To date, based on comparison with experimental results [2, 15], a value of $\delta=0.4$ provides the best overall match for typical laboratory RDE inlets.
B. 3D Instability

The 3D simulation could not be initiated, reconfigured, or restarted as readily as the Q2D simulation. Changing the inlet area for example required generation of a new grid. And of course, there is no diodicity parameter to alter. Diodicity and forward flow loss are not modeled here, they are computed directly based on the inlet geometry.

All of the 3D simulation results to be shown were initiated by first establishing a particular steady flowfield in the computational domain where hot products filled the domain at the specified flow rate. Following this, fresh reactant mixture was introduced at the upstream wall of the manifold and allowed to fill the manifold, entrance, inlet, and a small length of the channel where chemical reactions were temporarily prevented. Then a small, high-pressure, high-temperature zone or ‘pocket’ of hot gas was inserted just downstream of the inlet trailing edge. This small simulated explosion soon grew into a regularly rotating detonation wave.

The simulation was initiated in the manner just described. A regular rotating detonation formed, and the inlet manifold began to pressurize as expected. However, approximately 10-20 wave revolutions into the simulation an oscillation in the area average manifold pressure was detected. The regular oscillation grew and after approximately 25 additional wave revolutions, the detonation failed. The manifold pressure during these last 25 revolutions is shown in Fig. 4. It can be seen that the period of the oscillations is approximately 2 wave revolutions.

Figure 5 shows the normalized mass flow rate at the leading edge of the inlet and in the exit plane over the last 5 oscillations before detonation failure. There are only approximately 6 data points per oscillation, but it is clear that the amplitude of the inlet oscillations is increasing over time. The inlet and exit oscillations are out of phase. This indicates dynamic activity within the channel (i.e. temporary mass storage). Also shown in the figure are two points in time (A and B) corresponding to minima and maxima in the inlet mass flow rate. Distributions of normalized inlet leading edge mass flux rate at these two times are shown in Fig. 6. The radial location of these traces is midway across the inlet width (see Fig. 1). It is seen that the low mass flow rate, A, corresponds to the moment when backflow is greatest. At this point in the instability development (i.e. very close to failure), the backflow is immense at 123% of the total flow. Correspondingly, it is seen that the high mass flow rate point (B) has no backflow at all.

Contours of temperature, as described in Fig. 2, at the times corresponding to points A and B are shown in Fig. 7. The radial location for the data is midway across the channel. It is clear from these contours that the detonation height varies markedly. It is highest when the inlet mass flow rate is the lowest, and lowest when the inlet mass flow rate is highest. This somewhat counterintuitive result may be because there is a convection delay between fluid flowing in the inlet plane and that same fluid manifesting as the height of the detonation. In other words, the low flow rate at time A yields the low detonation height at the later time B. Similarly, the high flow rate at time B will yield a high detonation height at the next revolution of the wave. Detonation failure appears to occur when the point B detonation height becomes so short that no detonation exists.
The instability just described was unexpected, and obviously not observed in the Q2D simulation of the same configuration used to obtain Figs. 2 and 3. This suggested that the instability was rooted in either: a numerical methods aberration associated with OpenNCC, interactions with the upstream manifold which is not modeled in the Q2D simulation, or a uniquely 3D phenomenon associated with flow variations in the radial dimension. Before exploring these possibilities, however, it was deemed prudent to investigate if some minor change to the Q2D approximation of the RDE configuration might produce the unstable behavior. It was found that when the channel was shortened by just 7% an instability resulted. Initially, as $\delta$ was reduced from 1 to 0.6 the behavior and performance were indistinguishable from Fig. 3. However, when $\delta$ was lowered further to 0.5, the flowfield developed the same regular oscillation in the total mass flow rate as the 3D simulation, and with the same period. Moreover, the amplitude increased over time until the simulation failed. The oscillation growth is seen in Fig. 8 where normalized mass flow...
rates at the inlet and exit of the RDE are plotted against time as measured in detonation wave revolutions. This data is taken from a point starting approximately 15 wave revolutions after the diodicity reduction was imposed. Similar
to Fig. 5, it is seen that the inlet and exit oscillations are out of phase and that the inlet amplitude is growing. The same minima and maxima are also shown in the figure as two points in time (A and B).

Figure 9 shows the distribution of normalized mass flux at the times corresponding points A and B. This should be compared with Fig. 6. The same phenomenon is observed whereby the low mass flow rate of point A is caused by the large amount of backflow taking place at that time.

Figure 10 shows contours of temperature at the times corresponding to points A and B. The similarity to Fig. 7 is clear. The detonation height varies markedly. It is highest when the inlet mass flow rate is the lowest, and lowest when the inlet mass flow rate is highest. It is noted here, though not shown, that in both the Q2D and 3D simulations of this instability, the detonation wave speed also oscillates slightly. It increases as the inlet flow rate increases, and vice-versa.

The similarities between Figs. 5-7 and Figs. 8-10, despite differences in simulation dimensionality, grid resolution, numerical methodology, boundary condition application, and combustion mechanism, strongly suggest that the
The instability observed is genuine. That is to say, the instability appears to be a valid dynamic solution to the governing equations used under the simplifying assumptions made. Whether these equations and assumptions are sufficiently reflective of real world RDE’s has yet to be determined. It is possible however, that laboratory RDE’s fabricated with inlets similar to Fig. 1, with \( \frac{A_i}{A_{ch}} \geq 0.5 \), could experience operability issues related to this phenomenon. Given that the Q2D simulation does not have a model for the inlet manifold but still shows an instability, it is unlikely that manifold dynamics play a significant role in the instability development.

D. Additional Instability Investigations

Though the cause of this instability is not well understood as of this writing, additional simulation results can provide insights. It is unlikely that the change in length required to initiate the instability in the Q2D simulation was related to some sort of wave related ‘acoustic’ tuning. It is more likely that forward running shock that reflects from the exit contraction, and which is visible in Figs. 2 and 10, was strengthened slightly. This, in turn, affected the dynamics of the incoming flow. As will be discussed in Subsection E, this may provide an avenue for instability formation. Further suggesting that RDE length is not a direct factor in the instability are the results from another simulated configuration. Here, the RDE was 14% longer than that in Fig. 1 and had no throat at the exit (i.e. uniform cross section). The value of \( \frac{A_i}{A_{ch}} \) was set at 0.6, with \( \delta = 0.2 \). The delayed reaction region was increased relative to Fig. 1 to \( y = 0.035 \). The simulation was initiated with the delay region set to \( y = 0.030 \) where it reached limit cycle behavior. The delay was then moved to \( y = 0.035 \), and approximately 25 wave revolutions later the instability had developed and grown to the point where backflow oscillated between 20% and 67% of the throughflow. The period of the instability varied slightly but was consistently between 2 and 2.3 wave revolutions. Contours of normalized entropy (relative to the inlet) are shown for the low and high flow moments in the cycle in Fig. 11. The entropy contours are truncated at a value of 8. The low entropy flow from the inlet manifold passing directly through the detonation is clearly distinguishable from the higher entropy backflow that gets recirculated. The variation in detonation height seen in the previous unstable simulations is also clear.

When this configuration was made 14% shorter than that of Fig. 1, the same instability resulted with the same period under the same conditions. The contours look similar to Fig. 11, and so are not shown. This result strongly suggests that the length of the RDE is not particularly important and that the origins of the instability lie in the detonation and inlet region.

Another unstable configuration was found using the same length as Fig. 1, but with \( \frac{A_i}{A_{ch}} = 0.75 \), \( \frac{A_i}{A_{ch}} = 0.60 \), and the reaction delay set to \( y = 0.0 \). This meant that the backflow was hot, post-detonative gas rather than fresh charge. The hot backflow was still recirculated through the system. The instability was initiated by changing \( \delta \) from 0.7,
where the simulation was stable to 0.6. Unstable behavior was evident within 6 wave revolutions. The inlet mass flow rate showed a regular oscillation that was 20% of the mean and increasing. The backflow varied between 11% and 23% of the throughflow. The period of the instability was approximately 1.5 revolutions; notably shorter than the previous cases of instability. Contours of normalized temperature at the low and high flow rate moments of the instability cycle are shown in Fig. 12.

Since the 3D simulation took more time to run, very few configurations could be examined. It is noted here however that simulations using the basic Fig. 1 geometry and boundary conditions, but with vastly different inlet designs, have shown this same unstable behavior. Details of the inlet designs are not appropriate for open publication, but it may be noted that all unstable simulations had values of $A_i/A_{ch}$ that were at least 0.5. A simulation of the Fig. 1 configuration with approximately double the grid resolution in every dimension is underway as of this writing. This was intended to examine if grid resolution impacted instability development. Due to an unforeseen computer hardware failure, it was not completed. However, small oscillations in the inlet mass flow rate have developed, they are growing, and they have a period of two detonation wave revolutions. Doubling the grid resolution in the Q2D simulation did not significantly alter the instability development.

E. Compressor Surge Analogy

The precise physics behind the onset of this observed instability are not yet understood. Because detonative cycles are so highly coupled, and because neither tracking individual particles nor monitoring physical locations in an RDE can fully describe their operation, explanations can be elusive. All that can be said definitively about the present instability are that it generally develops slowly over multiple wave revolutions, and that it only occurs with inlet configurations that allow significant backflow.

Some insight may result by drawing an analogy with a simple model for surge onset in compression systems [16]. This model considers a system consisting of a duct with a characteristic length $l$ and cross section $A_{ch}$, a pump with a linear pressure rise versus mass flow rate characteristic slope $C$, a plenum with a characteristic volume $V$, and an exhaust valve with a linear pressure drop versus mass flow rate characteristic slope $S$. This is basically a Helmholtz resonator that is highly modified by $S$ and $C$. The solution to the governing second order ordinary differential equation for plenum pressure or compressor mass flow rate takes different forms depending on the system characteristics. One form is oscillatory with a damping term having an exponential time constant of the form:
If \( \alpha \) is positive, the oscillation amplitude increases (i.e. the system is unstable). The value of \( \alpha \) can be positive only if the compressor slope is positive, and if \( \frac{scv}{a^2} > \frac{1}{A_d} \). The implication here is that there is a complex phase relationship between system energy capacitance, inlet duct inertia, and energy addition that can, under the right conditions, lead to instability.

If the RDE is imagined as an infinite number of axially aligned tubes or vessels arranged to form an annulus, and the rotating detonation is represented by the tubes undergoing sequential constant volume combustion cycles [12], then each of these vessels has all the elements needed for a crude surge model. The vessel here should be thought of as being only as long as the detonation height. The inlet duct and plenum volume are not entirely distinct from one another. They both conceptually comprise the vessel.

Figure 13 shows an example of the development of instability over time using the surge analogy. The governing ordinary differential equations were integrated numerically. It is a purely mathematical example such that the integrated quantities have no units. The pressure and mass flow rate shown represent perturbations from mean values. The necessary positive compressor slope \( C \) is replaced by one which is positive if the mass flow rate perturbation is negative, but is an impulse when the mass flow rate perturbation is positive. The impulse is delivered when the mass flow rate is at a cycle maximum. The impulse amplitude is proportional to the mass flow rate. This is intended to represent the constant volume combustion process. The constant of proportionality becomes the equivalent of the compressor slope. As expected, though not shown, growth or decay of the instability is quite sensitive to the system parameters described above. Interestingly, no growth is possible if reverse mass flow (i.e. backflow) is prevented. The example suggests that similar, though far more complex interactions between fluid inflow inertia, energy storage, and work exchange dynamics, may be involved with the RDE instability observed.

The period associated with the RDE instability is also not yet understood. It is reasonable however, that it exceeds the period of one wave revolution. Examining Fig. 2, considering the streamlines shown, and recalling that this is in the detonation frame of reference, it is seen that all of the fresh charge that enters the RDE between 0.8<x<1.0 and 0<x<0.05 will pass through (or be processed by) the detonation within 0.4 wave revolutions. Meanwhile, the fresh charge entering between 0.05<x<0.2 will not be processed for more than one wave revolution. The fact that there can

\[
\alpha = \frac{-\frac{1}{A_d} \frac{scv}{a^2}}{2s \frac{A_d}{k} \frac{a^2}{v}}
\]
be more than a one wave revolution difference in residence time between particles that enter the RDE at the same time suggests that any unstable phenomena involving its bulk flow is likely to have period that is longer than a single wave revolution as well. This being said, an explanation for the precise 2 wave revolution period for two of the configurations and 1.5 wave revolutions for the other, has not yet been found. More study and more validation by other CFD-based RDE simulations are needed.

F. 3D Limit Cycle Results With Q2D Comparison

It was found that if the Fig. 1 inlet was further restricted to $A_i/A_{ch} = 0.4$, a stable limit cycle could be obtained. It is interesting to note that this inlet restriction size was also stable for the Q2D simulation at all values of diodicity and with the reduced length configuration. Temperature contours at the channel midpoint at an arbitrary moment in time are shown in the upper half of Fig. 14. The computed detonation speed is 4.8% above the CJ speed for a thermally perfect equilibrium mixture. The lower half of the figure shows temperature contours computed with the Q2D simulation using $\delta$=0.6. The salient features (detonation height, oblique shock, reflection from the exit throat, shear layer, etc.) are remarkably similar.

Normalized mass flux distributions at the inlet leading edge and channel midpoint are shown in Fig. 15, at 3 arbitrary moments during one wave revolution. The profiles have been shifted in x so that they align. Despite this being a stable limit cycle, it is evident that there is significant high frequency unsteadiness. This results from the finer grid resolution and, to a lesser degree, the three dimensionality of this simulation as compared to the Q2D simulation (see Fig. 2). The profiles all look similar, but they vary significantly in the percentage of backflow to total flow. A more accurate accounting of this variation is shown in Fig. 16. Here is shown the instantaneous backflow rate as a percentage of total flow rate across the annular plane at the leading edge of the RDE inlet over 5 wave revolutions. The average backflow rate percentage is 16.5%. The standard deviation is 6.8%. Unfortunately, even though the average backflow percentage is modest, the forward flow losses arising from such a severe restriction yield a computed pressure gain for this configuration of just $P_{GAPexp}=2.5\%$.

The average backflow and pressure gain as computed with the 3D simulation are compared with the Q2D simulation in Fig. 17. It is seen that the pressure gain value compares well with $\delta$=0.6. And the backflow rate is only 3.5% higher in the Q2D simulation than in the 3D simulation. This result suggests that the Q2D simulation sub-model for forward flow loss and backflow loss provides reasonable performance estimates for simple inlets if a value of $\delta$=0.6 is used. Smaller slit widths were not simulated for comparison in 3D or Q2D since it is evident that the performance would be impractically poor in terms of pressure gain.
Fig. 14  Contours of temperature at a moment in time for the stable operation with $\Lambda_i/\Lambda_{ch}=0.4$ as computed by the: (upper) 3D simulation at the channel midpoint; (lower) Q2D simulation with $\delta=0.6$

Fig. 15  Normalized inlet leading edge mass flux distributions around the RDE annulus at the channel midpoint at three moments in time for $\Lambda_i/\Lambda_{ch}=0.4$
The impact of flow reversal, or backflow at the inlet of an airbreathing rotating detonation engine (RDE) was investigated using quasi-two-dimensional (Q2D) and three-dimensional (3D) computational fluid dynamic (CFD) simulations. A simple annular 'slit' inlet design was used. The simulations were idealized to isolate the effects of backflow from other loss-inducing RDE phenomena. The Q2D simulation contained a simplified sub-model for the inlet (which is highly 3-dimensional) that utilized a tunable diodicity parameter to capture the physics of the slit. The results showed that even with significant idealizations, RDE inlets that allow significant backflow suffer substantial performance loss. Both simulations also exhibited a novel instability that developed in certain RDE configurations.

V. Conclusion

The impact of flow reversal, or backflow at the inlet of an airbreathing rotating detonation engine (RDE) was investigated using quasi-two-dimensional (Q2D) and three-dimensional (3D) computational fluid dynamic (CFD) simulations. A simple annular 'slit' inlet design was used. The simulations were idealized to isolate the effects of backflow from other loss-inducing RDE phenomena. The Q2D simulation contained a simplified sub-model for the inlet (which is highly 3-dimensional) that utilized a tunable diodicity parameter to capture the physics of the slit. The results showed that even with significant idealizations, RDE inlets that allow significant backflow suffer substantial performance loss. Both simulations also exhibited a novel instability that developed in certain RDE configurations.
with large backflows. The instability development was detailed, though its origins are not yet fully understood. Although the instability could not be formally proven as a legitimate solution to the governing equations, it was shown to be a reasonable one. Comparison of the Q2D and 3D results also established a reasonable value for the Q2D diodicity parameter. This in turn gave added confidence to the output of the Q2D simulation which, being far less resource intense than 3D, can be readily used for parametric optimization and mission analysis. Both the stable performance and instability results from the investigation show that sophisticated RDE inlets designs are needed which provide low loss when flow is in the forward direction, and very high resistance when the flow is reversed. Development of such high-diodicity inlets is critical to achieving pressure gain in airbreathing RDE’s.

References