

High-Order Flux Reconstruction for the Implicit Large Eddy Simulation of an Axisymmetric Scramjet Combustor

T. Sailor Koeplinger¹, Caleb Hash², Jack Edwards³

North Carolina State University, Raleigh, NC, 27606, USA

Seth Spiegel⁴

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

The high-order flux reconstruction method is used to conduct large-eddy simulations of an axisymmetric inlet-isolator-combustor configuration experimentally tested in the University of Illinois's ACT-II arc-heated combustion tunnel and numerically investigated with North Carolina State University's REACTMB solver. A divergence-based indicating and modal filtering scheme is used for shock capturing, aided by a boundedness filter for additional solution stabilization. A 7-species 3-step reaction mechanism developed at the Air Force Research Laboratory is used in these calculations. An isothermal wall of 375 K is assumed. The flux reconstruction method, equipped with the correction functions that recover a discontinuous Galerkin discretization, was run at varying polynomial orders from linear P1 elements to quartic P4 elements. It is shown that P-scaling is an effective technique for the enhanced resolution of small-scale turbulent structures, with the present method comparing favorably to high-order finite volume methods while operating on a coarser discretization. The shock-capturing method presented is not found to scale well with polynomial order, and as such compares poorly with the previous results. The stable ram-mode combustion with an equivalence ratio of 1.24 was unable to be achieved, instead leading to unstart of the flowpath. At lower polynomial orders, P1 and P2, a rotating detonation wave is formed in the region downstream of the injectors prior to the unstart event. Higher polynomial orders, P3 and P4, do not form a detonation wave but show an intermittent combustion operation with multiple, smaller, pressure waves present in the powerhead region.

I. Nomenclature

A	=	amplitude of oscillation
a	=	speed of sound
h	=	width of an element
p	=	pressure
P	=	polynomial order of the solution
s	=	sensor or indicator value used for solution stabilization
u	=	solution vector
$\tilde{u}$	=	filtered solution vector
$\bar{u}$	=	mean of solution vector on an element
$\vec{V}$	=	velocity vector

¹ Research Assistant, Mechanical and Aerospace Engineering, Student Member AIAA

² Research Assistant, Mechanical and Aerospace Engineering, Member AIAA

³ Professor, Mechanical and Aerospace Engineering, Fellow AIAA

⁴ Aerospace Engineer, Inlets and Nozzles Branch, Member AIAA

Y	= species mass fraction
Ω	= Interior domain of an element
$\delta\Omega$	= Boundary of an element

II. Introduction

High-speed air-breathing propulsion (HSABP) engines, such as scramjets and ramjets, are of great interest for applications ranging from improved access to space to military uses. Their operation is difficult to achieve because of the extreme environments which they must withstand and the very short residence times available for the fuel and air to mix and burn. The premier tool for high-fidelity analysis of these flowpaths is large-eddy simulation (LES). These are scale-resolving simulations which aim to capture unsteadiness at scales which have a bulk effect on the flow and model the smaller, more universal, scales of turbulence. These simulations require the use of high-order numerical methods to resolve and propagate unsteadiness at the scale close to that of the grid itself. Typically, these methods are designed for structured grids and use large communication and discretization stencils. An alternative family of methods of recent interest are finite-element methods, such as the discontinuous Galerkin (DG), spectral element (SEM), and flux reconstruction (FR) methods. They are desirable for:

- The ability to use mixed-element unstructured grid elements.
- The ability to utilize a much coarser grid than that for an equivalent stencil-based reconstruction method with the same degrees of freedom.
- The ability of these methods to perform “P” scaling, which is to reduce the error of a method by increasing the polynomial order (P) of the solution.
 - For DG the order of accuracy for general nonlinear problems is $P+1$.
 - A “super-accurate” convergence on the order of $2P+1$ is observed theoretically and numerically for specific problems which are linear in nature and with a clear upwinding direction, e.g., acoustic propagation and density-wave advection.

There is a large body of work developing techniques for using these methods to analyze fluid dynamic flows [1-9]. They have been successfully developed and utilized for incompressible and subsonic compressible flows for both steady and unsteady analysis. Many challenges are present when extending these methods to supersonic flows because of shock induced instabilities. There has been much work focused on developing better techniques to stabilize these methods when applied to shock-dominated flows, and the development thereof is still an active area of research [9-16].

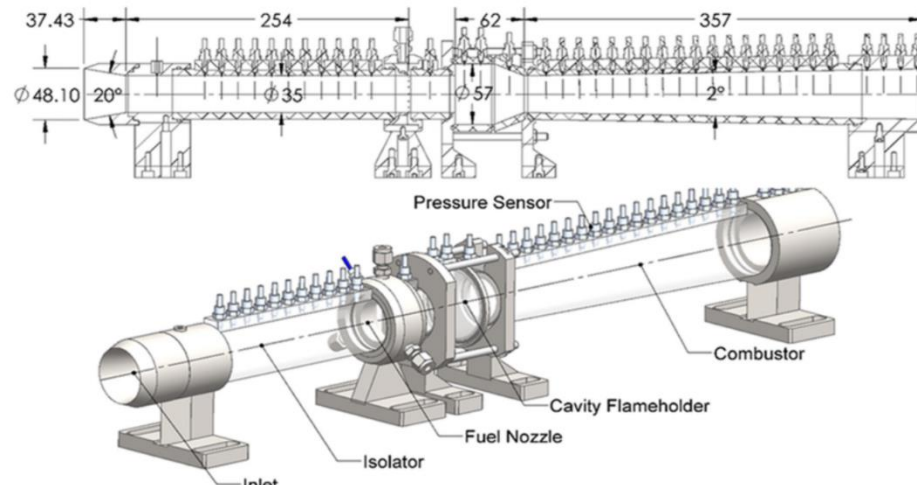


Fig. 1 Diagram of the axisymmetric cavity combustor tested in the ACT-II facility. This figure is taken from Ref. 24.

The ACT-II arc-heated scramjet test facility [17-23] has been used to experiment with different ethylene-fueled HSABP geometries and configurations. It offers pressure tap measurements in addition to a large amount of optical access enabling chemiluminescence and planar laser-induced fluorescence (PLIF) measurements. The ACT-II facility can reach high-speed flight enthalpy levels but only at relatively low stagnation pressures in the range of 1.0 to 3.0 atm. There is a body of recent and ongoing computational analysis of this facility [24-26]. These studies have

investigated the transition from stable ram-mode to unstart, the operational characteristics of the combustor in scram- and ram-mode, the effects of oxygen addition, and the effects of thermal nonequilibrium caused by the arc-heating process.

The case of interest is an axisymmetric cavity combustor in free-jet configuration shown in Fig. 1. Specifically, this work will be investigating stable ram-mode operation of this geometry and comparing it with the experimental and computational results presented in Ref. [24]. The inflow conditions for this case are Mach 4.5, a static pressure of 325 Pa, and a static temperature of 503 K, and it is fueled with ethylene to an equivalence ratio (ER) of 1.24. There are 16 circumferential injectors upstream of the cavity, each with a diameter of 0.75mm. Previous numerical studies of this configuration utilized a 6th-order hybrid central-difference/piecewise-parabolic method for discretization of the inviscid fluxes and used computational meshes containing 29 M and 55 M cells. This work will apply a high-order flux reconstruction method to discretize the case over a range of polynomial orders spanning from linear P1 elements (2nd-order and 3.4 M solution points) to quartic P4 elements (5th-order and 38.4 M solution points).

The FR method [5-7] unifies a wide class of high-order finite-element-type methods under a single framework. It utilizes the differential form of the governing equations, employs polynomial interpolation between solution points within a grid cell, and interaction between cells is accounted for by using a correction function together with common interface fluxes to reconstruct a continuous flux function. The choice of correction function determines the specific method under the FR framework, with common options being DG and several spectral-element methods [5,27-28].

III. Numerical Approach

A. Governing Equations

The present work solves the compressible Navier-Stokes equations extended to chemically reactive mixtures of thermally perfect gases by the addition of species continuity equations and reaction source terms. Species enthalpies are obtained using McBride/Chemkin-type polynomial curve fits. Subgrid viscous, diffusion, and heat conduction terms are modeled implicitly through the application of de-aliasing and solution stabilization techniques. The mixture molecular viscosity is calculated using Wilke’s law [29]. The Prandtl and Schmidt numbers are both assumed to be 0.72. The chemical source terms are calculated explicitly using a finite-rate kinetics model explained in Section III.E.

B. The Flux Reconstruction (FR) Method

The code used presently is a modified version of the Glenn Flux Reconstruction (GFR) code [30-32] being developed at NASA Glenn Research Center. It is being used with the appropriate correction functions to recover a nodal DG discretization. The solution point locations are chosen to be Gauss quadrature for the increased accuracy of that choice over other options. The interfacial fluxes are evaluated at a separate set of flux points located on the element boundaries. Interfacial inviscid fluxes are determined using Edwards’ Low Diffusion Flux Splitting Scheme (LDFSS) [33], while the viscous, diffusive, and conductive fluxes are handled via the Bassi-Rebay 2 (BR2) method [34]. Time advancement is done via explicit Runge-Kutta (RK) methods: the present investigation uses both a 3-stage/3rd-order RK3-SSP method and an 11-stage/3rd-order RK method depending on which is found to perform most efficiently for a given run case.

C. Shock-Capturing Scheme

As with all high-order methods, it is necessary to use a shock-capturing scheme to stabilize the solution in the presence of shocks and other discontinuities. The present approach consists of two components: an indicator which identifies regions with shocks, and a dissipative filtering mechanism for stabilizing the computation. The method presented here is a combination of the de-aliasing filter present in GFR [31] along with concepts from the shock-capturing scheme implemented in NCSU’s REACTMB solver [24].

First, for indicating troubled regions, a divergence-based indicator, $s_{divergence}$ (Eq. 1), is taken from the form of the Ducros switch used in REACTMB. The motivation behind this is that shocks will have a uniquely strong signal in the velocity divergence, and the magnitude of the divergence can provide a scale for how much dissipation to add for shocks of different strengths. Additionally, this indicator will not add excessive dissipation to contact discontinuities, such as those between species in a multi-component flow.

$$s_{divergence} = \left(\frac{2|\nabla \cdot \vec{V}|}{a} \cdot \frac{h}{P+1} \right)^4 \quad (1)$$

This divergence indicator is then multiplied with a jump indicator [10, 31], s_{jump} , based on the pressure jump at the cell boundary (Eqs. 2-4). It is a measure of how discontinuous the pressure field is along the boundary of an element. The parameters used in this study are $\Psi_{min} = -4.0$, $\Psi_{max} = -1.0$.

$$M_{jump} = \log_{10} \left(\max \left(\frac{1}{|\delta\Omega|} \int_{\delta\Omega} \left| \frac{(p_1 - p_2)}{0.5(p_1 + p_2)} \right| ds \right), 2^{-23} \right) \quad (2)$$

$$s_{jump} = \begin{cases} 0 & M_{jump} < \Psi_{min} \\ \frac{1 + \sin \xi}{2} & \text{else} \\ 1 & M_{jump} > \Psi_{max} \end{cases}, \quad \xi = \frac{\pi (M_{jump} - 0.5(\Psi_{min} + \Psi_{max}))}{\Psi_{max} - \Psi_{min}} \quad (3,4)$$

$$s = s_{divergence} \cdot s_{jump} \quad \text{if } s < 0.02, \text{ set } s = 0.0 \quad (5)$$

The filtering mechanism (Eqs. (6-8)) used in GFR [31] is inspired by Hesthaven [1]. It provides a way of adding dissipation by means of damping modes in the frequency space of the solution on an element, while leaving the cell average (P0 projection) alone. The specific modal damping coefficients, $\sigma(n)$, are determined according to an exponential formula, since aliasing errors and related instabilities tend to manifest in the higher frequency modes of a solution. One modification present in the current work is a means of scaling the filter dynamically in-situ. We introduce a parameter (s in Eq. 8) which interpolates between the filter and the identity operator. This is mathematically equivalent to interpolating between the default values of the modal damping coefficients and one. The shock sensor (s in Eq. 5) is used as this parameter, which provides a means of dynamically adapting the dissipation added by the filter.

$$\sigma(n) = \exp[-\alpha \eta^c], \quad \text{for } n = 1, \dots, P+1 \quad \text{where } \eta = \frac{n-1}{P} \quad (6)$$

$$F = V^{-1} \Sigma V, \quad \text{where } \Sigma = \delta_{i,j} \sigma(i) \quad \text{for } i, j = 1, \dots, P+1 \quad (7)$$

$$\tilde{u} = (s)uF + (1-s)uI \quad (8)$$

Here, c and α are parameters which control the filter's distribution and amount the highest mode is damped, respectively. The computations in this work use $c = 4$ and $\alpha = 16.0$. Finally, the indicator will only be used to filter elements if the s value is above a threshold, in this case 0.02. This was found to greatly decrease the dissipation of the scheme by moving many of the lower-intensity instabilities from the shock-capturing filter to the boundedness mechanism in the next section.

D. Other De-Aliasing / Stabilization

The shock-capturing scheme is not sufficient to ensure boundedness and positivity preservation for thermodynamic variables such as density, pressure, and temperature. As a way of stabilizing the solution, we have a separate filtering operation which is triggered by boundedness violations. It checks all solution and flux points for violations and tags them as a troubled cell if one is found. Troubled cells are then treated by having a filter successively applied to them until there are no boundedness violations, up to a maximum of 10 times. For the present study, the filtering mechanism used is to damp all the high-order modes by half, shown as Eq. 9. For successive filtering operations the $\tilde{u}$ from the prior filter application becomes the u for the next. While it is a very aggressive option, it was found to be more robust than weaker filtering options.

$$\tilde{u} = \frac{u + \bar{u}}{2} \quad (9)$$

The bounds used in this work are found in Table 1 for the standard thermodynamic variables. The lower bounds are set just below the lowest value they are expected to see. The pressure and density limits are low enough that the solution is only close to them in the inflow before the first shockwave; as such the added dissipation there should not interfere with the rest of the flow. The temperature limit is similar, with the addition of the near-injector region which experiences the cold fuel being accelerated through the injector. There are two upper bounds which have been imposed on the solution: Mach number and temperature. These are very seldomly activated but have been found to be necessary for the robustness of the method. The Mach-number boundedness limit helps to prevent issues in the region of strong expansion outside of the injectors; the high pressures and densities in this region, relative to the lower bounds of those

quantities, made the standard boundedness filter less effective. The temperature upper bound helps to limit unphysical reaction rates caused by the interaction of aliasing errors with the chemical kinetics in the turbulent mixing regions.

Table 1 Bounds used to trigger the boundedness filtering.

	Temperature	Density	Pressure	Mach
Lower Bound	250.0 K	0.0023 kg/m ³	325.0 Pa	N/A
Upper Bound	4000.0 K	N/A	N/A	5.0

There are also areas of constant filtering, termed sponge filters, defined to provide extra dissipation to particularly troublesome regions of the flow. The included regions are the injectors, a small region outside of the injectors, and the last 20 mm of the geometry before the outflow plane. The region outside of the injectors is defined by extending 2 mm before and after the injector orifice and 1 mm off of the wall. The sponge filter takes on the form of the de-aliasing filter in Eqs. 6-8 (with $s = 1$), and its parameters are: $c = 2$, $\alpha = 36.0$. Finally, as a mechanism for stabilizing the flow during startup, a weak global filter is used on every cell in the fluid domain, and with the parameters: $c = 6$, and $\alpha = 0.1$.

E. Chemical Mechanism

The chemical model used in the present study is a three-step, seven-species finite-rate mechanism, which has been tuned to match ignition delay in shock-tube experiments at the Air Force Research Laboratory [35]. This is significantly different than the previous computations of this geometry, which have used a 32-species skeletal mechanism. The reason for the change in chemical model is to balance the desire for representative chemistry with the computational cost of performing these simulations. The chemistry is being integrated explicitly, which is possible because of the restrictive Courant-Friedrichs-Lewy (CFL) conditions associated with explicit DG methods.

Table 2 Reactions and coefficients for the AFRL mechanism in Ref. [28]. Form: $AT^B e^{-\theta/T}$

Reaction	A	B	θ
$C_2H_4 + O_2 \leftrightarrow 2CO + 2H_2$	1.04E+13	0	21,511
$2CO + O_2 \leftrightarrow 2CO_2$	7.21E+07	1.92	16,478
$2H_2 + O_2 \leftrightarrow 2H_2O$	9.47E+16	-0.9	1.00

F. Grid

The grid used in this study is an unstructured voxel grid generated in Fidelity Pointwise. It consists of a near-wall T-Rex layer, a freestream of cubic voxels, and an unstructured tetrahedral/pyramidal region connecting the two. The voxel grid format is chosen because it facilitates the notion of a freestream length scale which is most comparable to that of the previous work, which was conducted on multi-block structured meshes. The freestream element width is 1.8mm, and the width of the first cell from the wall is 0.08mm. In total there are approximately 600,000 elements in the mesh. The mesh is elevated to 4th order using the high-order mesh capabilities of Pointwise to accurately represent the curved surfaces of the combustor geometry. The average distance between solution points is used as an effective length scale for a mesh given a particular P order. It can be calculated as $h / (P+1)$ for a mesh with a grid spacing of h . The number of degrees of freedom, and effective length scales are listed in Table 3.

Table 3 Number of grid solution points and effective length scales with increasing P order.

	P1	P2	P3	P4
Degrees of Freedom	3.4 M	9.5 M	20.4 M	38.4 M
Freestream Length Scale	0.9 mm	0.6 mm	0.45 mm	0.36 mm
Off-Wall Length Scale	0.04 mm	0.027 mm	0.02 mm	0.016 mm

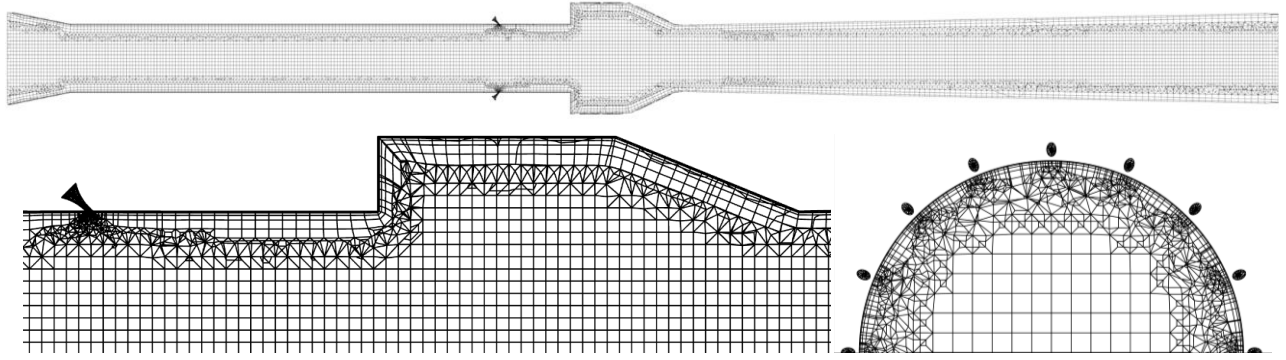


Fig. 2 Centerline views (top, bottom left) and an axial cut (bottom right) of the finite element grid used in this study.

G. Run Case and Boundary Conditions

The run condition we are targeting is the same as one studied in Ref. [24], which fuels the geometry at an ER of 1.24 and results in a stable ram-mode operation. The solution was initialized to the inflow value everywhere except for the injectors and cavity region, which were both initialized to their stagnation conditions. The inflow is a prescribed supersonic inflow which corresponds to case A from Ref. [24]. The modification of the inflow composition from that work is neglected, since the present chemistry does not include atomic oxygen or nitric oxide. The wall is treated as isothermal with a temperature of 375 K. This wall temperature was determined by examining the previous results which used a conjugate heat transfer model. The outflow is a supersonic outflow.

Table 4 Flow conditions specified at the inflow boundaries.

	Type	Mach	Pressure	Temperature	Y_{O_2}	Y_{N_2}	$Y_{C_2H_4}$
Main Inflow Conditions	Supersonic	4.5	351.9 Pa Static	502.9 K Static	0.233	0.767	0.0
Injector Conditions	Characteristic	0.01	49837.7 Pa Total	300.0 K Static	0.0	0.0	1.0

IV. Results

H. Initial results

The simulation was started with a P1 (linear) solution, chemical source terms disabled, and a global filter activated. After two flow-through-times (~ 1 ms), the chemical source terms were enabled, and the global filter was disabled. The very low pressures of this case require an aid to speed up the ignition of the fuel-air mixture. The ignition aid procedure used in this work is to modify the chemical source terms to calculate the rate of reaction as if the gas mixture was at a higher pressure and temperature, specifically 25 kPa and 2000 K. This was enabled until ram-mode operation was established, identified by a thermal throat forming at the end of the cavity, and an isolator shock train being propagated upstream of the injectors. From that point, the ignition aid was disabled and the solution evolved in a time-accurate manner.

That initial P1 (2nd-order) solution was run for 8.5 ms and resulted in the unstart of the flowpath by means of a helical pressure wave. This was an unexpected result, as the experiment and previous calculations did not indicate either an unstart or a helical flow response at this equivalence ratio. Figures 3 and 4 present snapshots of the centerplane pressure and Mach number throughout that process. A series of runs was then performed to explore the effects of increasing polynomial order, P . First a P2 (3rd-order) solution was restarted from just after the ignition source terms were disabled - that solution also unstarted with a strong helical pressure wave present. A P3 (4th-order) solution was also run but was started from a different condition. Instead of beginning just after ignition, the P3 solution was started from a P1 solution which has had a global filter applied to it that completely damped out the helical pressure wave and nearly all flow unsteadiness. The P3 run also unstarted, but the combustion intensity was much more intermittent, and a single unified pressure wave did not appear until much later in the unstart process. A P1 solution was then also restarted from the same damped/filtered solution to control for initial conditions, and that recreated the

original helical pressure wave behavior and subsequent unstart. Finally, a P4 solution was started from the P3 solution and run for long enough to observe the formation of a strong rotating pressure wave and that the flowpath was still unstarting (3.5 ms or approx. 7 flow-through-times). A chart of how different P orders were restarted from each other is shown in Fig. 5.

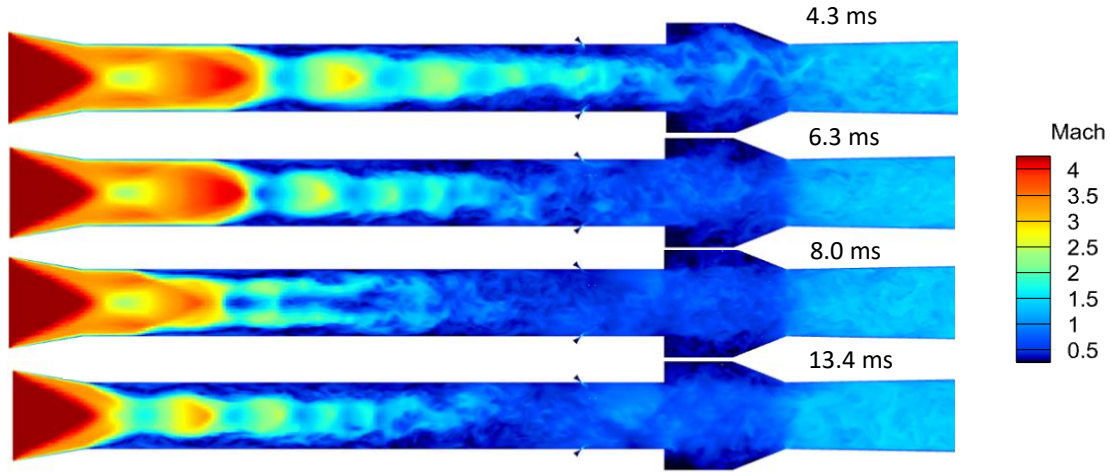


Fig. 3 Centerplane contours of Mach number throughout the formation of the helical pressure wave and unstart process for the P2 solution.

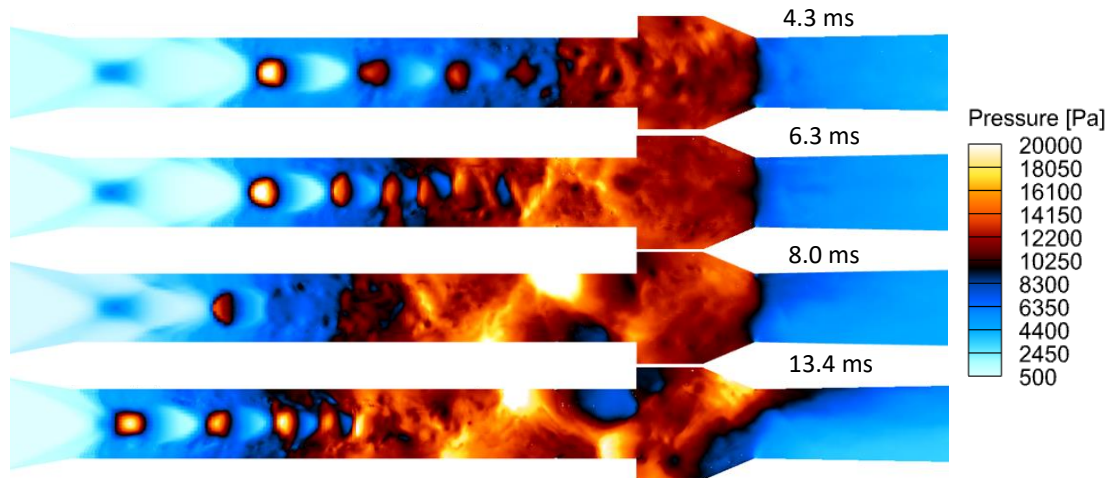


Fig. 4 Centerplane contours of pressure throughout the formation of the helical pressure wave and unstart process for the P2 solution.

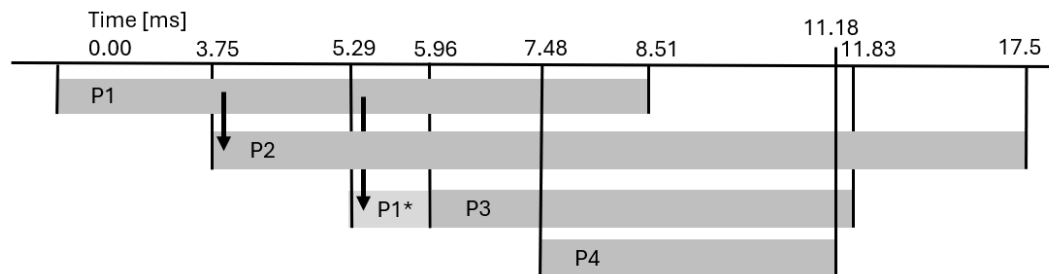


Fig. 5 Diagram of the restart history between different simulations. **P1*** indicates a P1 solution with a global filter.

A progression of wall pressures from the P2 case is presented in Fig. 6 alongside the experimental values taken at statistically steady state. The initial pressure distribution at 4.3 ms follows the trend well, overshooting somewhat in pressure downstream of the injectors. As the simulation runs and the helical wave forms, the isolator shock train moves upstream, and the pressure trace of the wave itself is seen as large spikes in the pressure value. A background pressure in the injector region of approximately 13 kPa is maintained, but the pressure wave itself can cause pressure spikes of more than 30 kPa.

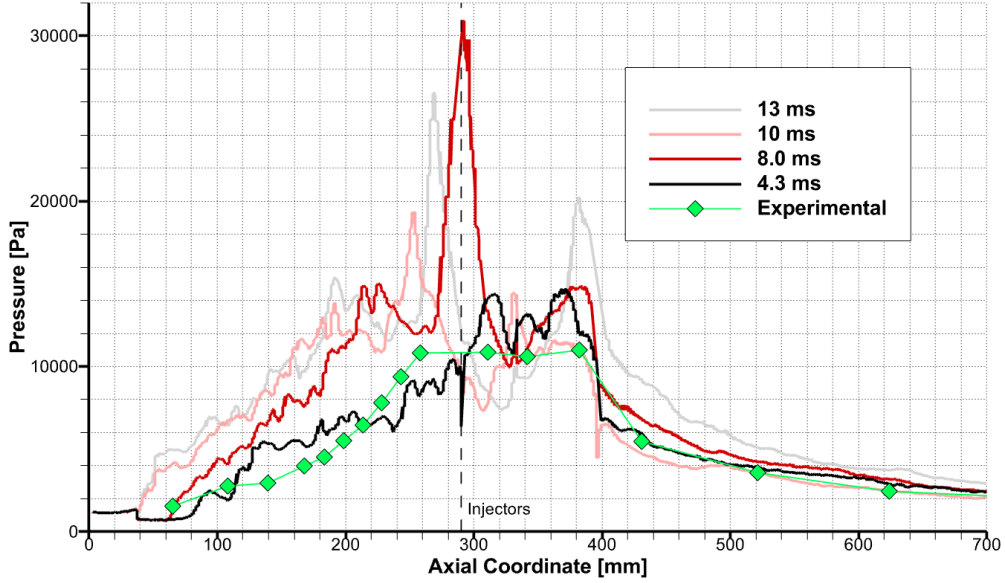


Fig. 6 Wall pressure traces of the P2 solution throughout the unstart process, coplotted against experimental pressure values for this case.

I. Spatial Resolution and P refinement

A goal of this work is to explore how increasing P-order affects the spatial resolution of the solution. The average distance between solution points on an element scale as $1 / (P + 1)$; thus, we can use that as heuristic for the smallest expected length-scale of flow features which the solution is able to capture. By that measure, a P3 solution should capture features half the size of a P1 solution. It is important to note that the distribution of solution points is not equally spaced; rather, they are placed at the Gauss quadrature points. For the P-orders presented in this study, the difference is small enough such that the average distance is a reasonable approximation for their spacing. That is not true for higher polynomial orders where the solution points will be increasingly clustered to the edges of the element.

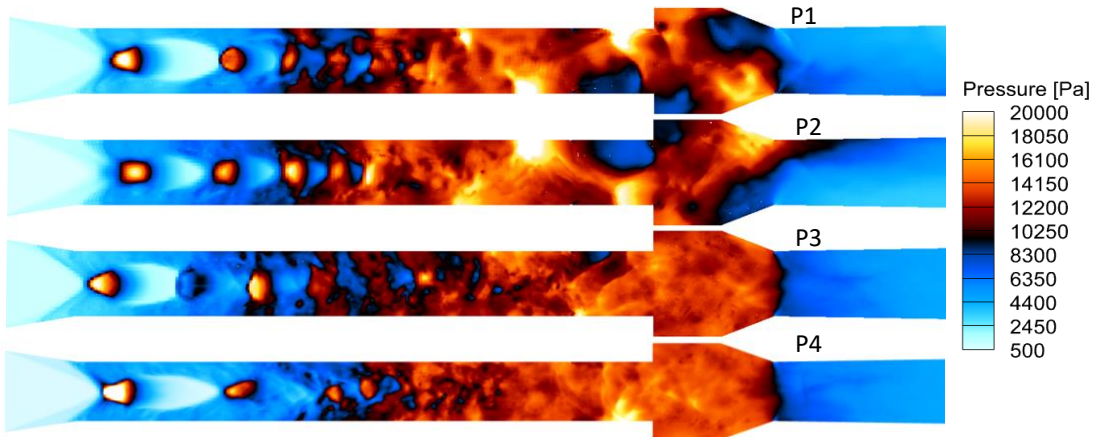


Fig. 7 Instantaneous pressure contours of different P-order solutions with the helical pressure wave present.

Figures 7 and 8 present midplane views of pressure and temperature during runs at different P orders. In Fig. 7 the pressure distributions for the P1 and P2 cases clearly show the midplane cross section of the helical pressure wave. It is strongest in the region between the injectors and the cavity and decreases in strength as it propagates upstream of the injectors. The wave is also present downstream in the cavity, impinging on the corner of the cavity closeout. It is unclear if the wave in the cavity is simply the downstream propagation of the pressure waves from the injector region, or if there is a coupling between the cavity and injector regions. The high-order solutions, P3 and P4, both show a different manifestation of the rotating pressure wave. As seen at the bottom of Fig. 7, the cavity is pressurized much more uniformly at around 14 kPa, and a unified helical wave does not form. Instead, there are smaller waves which appear intermittently and follow a similar helical motion. The high numerical dissipation of the lower orders is likely part of how the helical wave is able to persist without being broken up. A common thread between all the solutions is that the high pressures in the combustor are extended farther upstream in the isolator because of those waves. That extension of the high-pressure region pushes the shock train farther upstream and leads to the unstart of the flowpath.

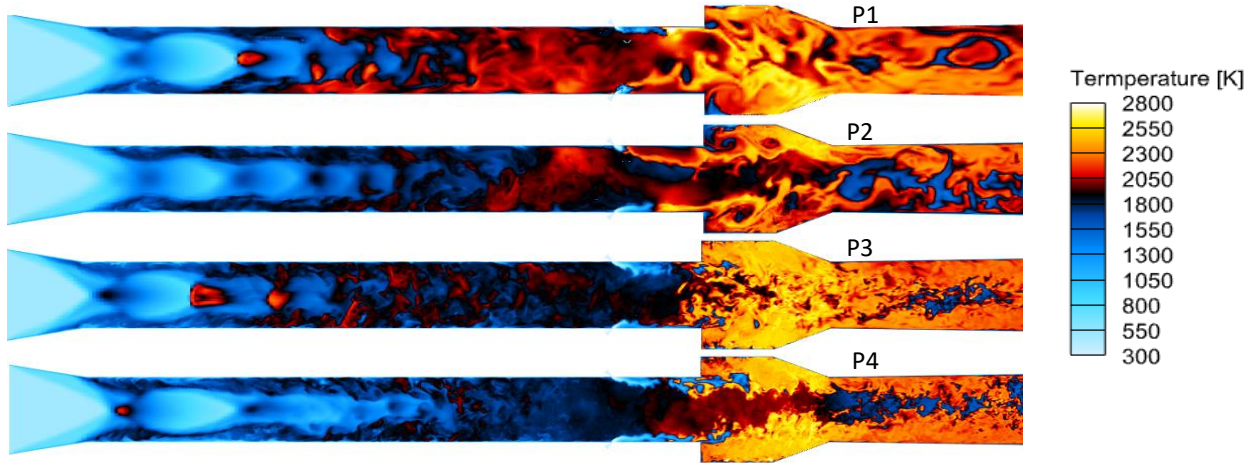


Fig. 8 Instantaneous centerline temperature contours of different P-order solutions after the formation of the helical pressure wave.

The temperature contours in Fig. 8 show a clear decrease in the numerical dissipation and increase in scale-resolving ability as the P-order is refined from P1 to P4. The transition from P1 to P2 and then to P3 all show significant changes in the turbulent mixing in the jet-wake of the injectors. The jump from P3 to P4 does show an increase in the scale of the resolved structures but not a change in the larger-scale flow characteristics. The P1 and P2 solutions show large scale flow mixing which is of the order of the width of the fuel stream being injected. P2 can resolve some extra detail in the cavity region, but it is still a fairly dissipative representation of the flowfield. Additionally, by cross referencing the pressure waves in Fig. 7 with the temperature in Fig. 8 both solutions show how the pressure wave is causing the temperature of the fuel jet to increase. That causes a diffusion flame in the wake of the wave observed opposite to the side of the instantaneous location of the pressure wave. It also occurs with a very sharp increase in temperature as seen in the fuel jet at the top of the P1 contour and bottom of the P2 contour. The P3 and P4 solutions resolve a smaller scale of turbulent breakdown of the fuel jets, leading to a higher degree of turbulent mixing and premixed combustion in the cavity region. These solutions show a very intermittent response, with the flame position shifting in the jet wake, varying from burning closer to the cavity region as shown in Fig. 8 to burning just downstream of the injectors.

As a benchmark for this method, a comparison is made with solutions previously obtained with REACTMB running the same case, with an ER of 1.24 [24]. Contours of their results are generated using data provided by the authors of that study. Fig. 9 contains a comparison of the temperature contours in the cavity region of the flowpath. The grid used by REACTMB is a structured multiblock mesh with an isotropic freestream spacing of approximately 0.3mm. At P4 (5th-order), the average distance between freestream solution points for the FR mesh is 20% larger at 0.36mm. The total number of degrees of freedom for the structured mesh is 55 M, whereas the P4 method is 30% smaller with only 38.4 M degrees of freedom.

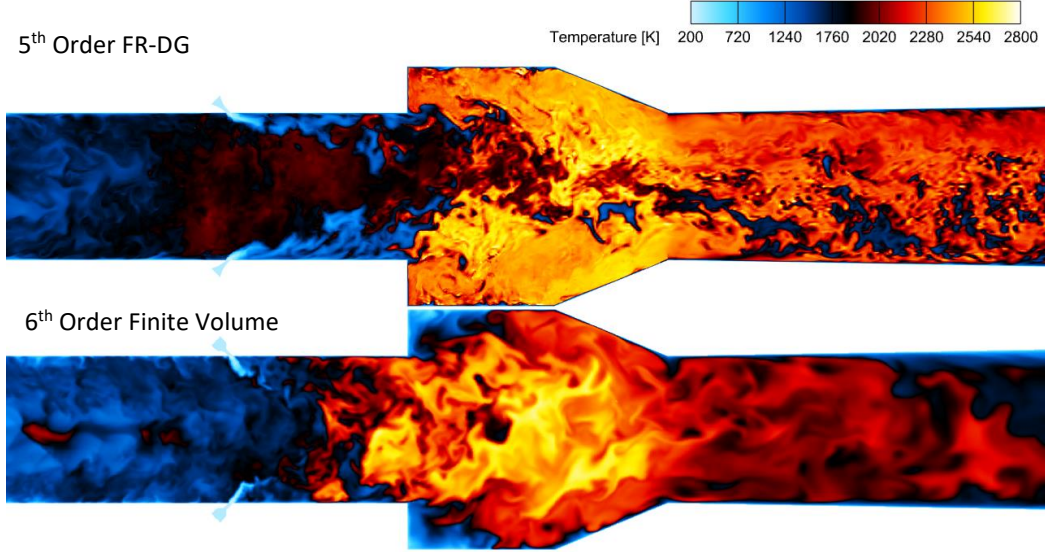


Fig. 9 Comparison of midplane temperature contours in the cavity region between the P4 FR-DG (top, 5th order, 38.4 M solution points, $\Delta X_{SP} \sim 0.36\text{mm}$) and REACTMB (bottom, 6th order hybrid PPM/central difference, 55 M cells, $\Delta X \sim 0.3\text{mm}$)

Despite being a coarser discretization, the FR scheme is recovering a finer resolution of small-scale turbulent flow structures. The same is not true for shocks which is demonstrated in Fig. 10 by the Mach number contours at the inflow of the geometry. There is a very significant difference in resolution of the inlet shocks and their interactions with the boundary layer. REACTMB can resolve the shock wave down to the width of one to two finite volume cells whereas FR is resolving them down to the width of one to two elements. Increasing the P-order of the solution drastically improves the resolution of small-scale turbulent structures, but same cannot be said for the resolution of shock waves with the current-shock capturing scheme. It shows an inability to resolve the inlet shocks with any degree of sub-element resolution. Attempts to fine-tune the shock capturing scheme to have better resolution of the shock train resulted in insufficient stabilization against the pressure waves downstream of the injectors and thus compromised the robustness of the method.

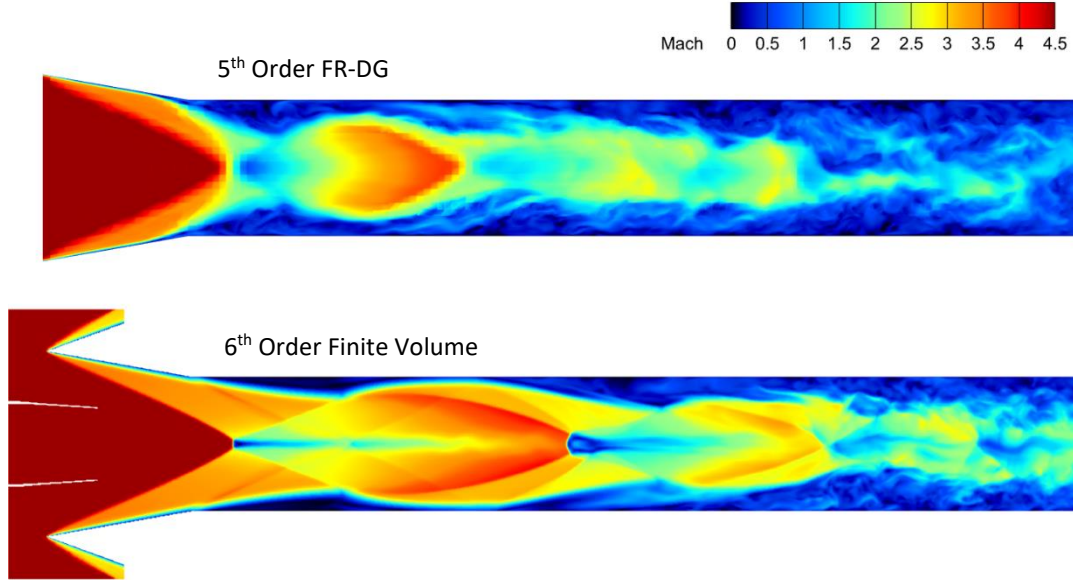


Fig. 10 Comparison of midplane Mach number contours between the P4 FR-DG and REACTMB solutions.

J. Examination of Rotating Wave Phenomenon

Although it has not been a response previously observed for this case, the helical pressure wave instability is a very interesting feature that warrants further examination. It appears to operate similarly to a rotating detonation wave. In this way, it enables the geometry to generate higher pressures and thus higher thrust with the same level of fueling. For that reason, an additional part of the present work is aimed at further studying this phenomenon.

One indicator to the presence of the rotating wave is that the mixing of the fuel into the core flow is greatly diminished. This can be seen in the temperature contours in Fig. 8 with the P2 and P4 runs showing the response most clearly. A comparison of the P4 solution from before and after entering this regime is presented in Fig. 11. The two contours presented are about 2.5 flow through times apart from each other. Before the helical pressure waves appear, the fuel jets penetrate into the core flow. This is marked by burning throughout the breadth of the cavity. Once the pressure wave(s) form, then penetration of the fuel becomes very limited, marked by burning near the wall downstream of the fuel injectors and the presence of a strong jet-wake-stabilized flame in the cavity shear layer. The core flow then has a much larger surface at which it interacts with the fuel jet, resulting in enhanced turbulent mixing compared to the standard operation of the combustor.

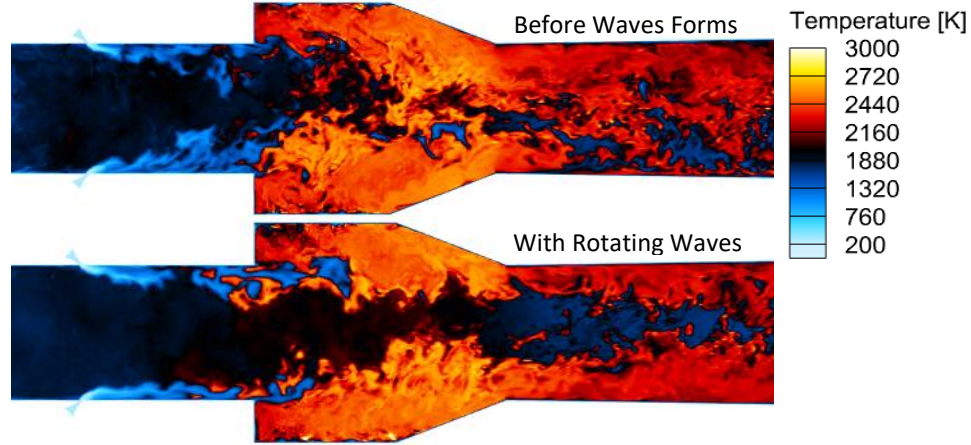


Fig. 11 Temperature of the P4 solution before and after the formation of the rotating pressure waves.

Figure 12 shows contours of temperature and heat release resulting from time averaging the P1-P3 solutions for a duration of 1ms after the formation of the pressure wave(s). All three solutions show the same type of behavior with very little jet penetration into the core flow. The P1 and P2 plots are the result of averaging the rotating detonation wave, which shows up instantaneously as alternating heat release signatures at the top and bottom of the plot. The P3 average is the result of a different process: rather than a detonation wave, it is the result of smoothing out an intermittent combustion process which jumps between burning in the cavity and flaring up to ignite the flow closer to the injectors. A peculiar observation is the presence of a heat release region at the end of the cavity closeout ramp. It is not certain what causes this, but it is plausibly an effect of the sharp corner in the mesh causing numerical issues and interacting with the solution filtering.

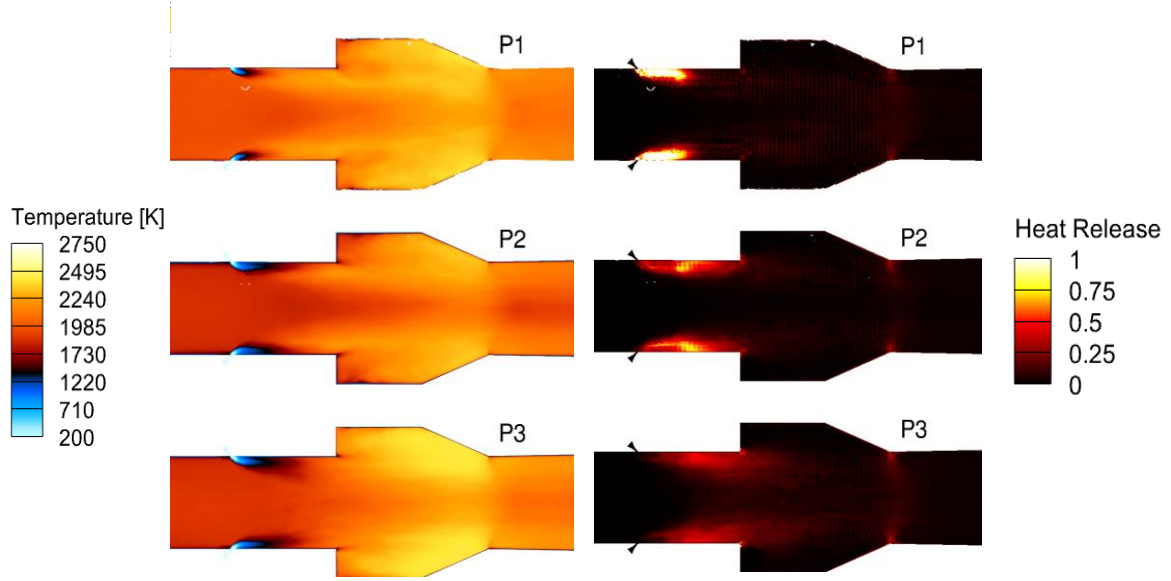


Fig. 12 Contours of temperature and heat-release ($\text{J} / \text{s m}^3$) for 1ms time averages of the P1-P3 solutions while the pressure wave phenomenon is present.

Figure 13 presents scatter plots of temperature against mixture fraction for the P4 solution. The dots are colored by their location and only one of every forty points is shown for clarity. The black dots are located in a region extending from just upstream of the fuel injectors to the cavity entrance plane. The red dots are axially located in the cavity region and only include points at a radius of less than or equal to the isolator wall radius. The flame is partially premixed in character, with the

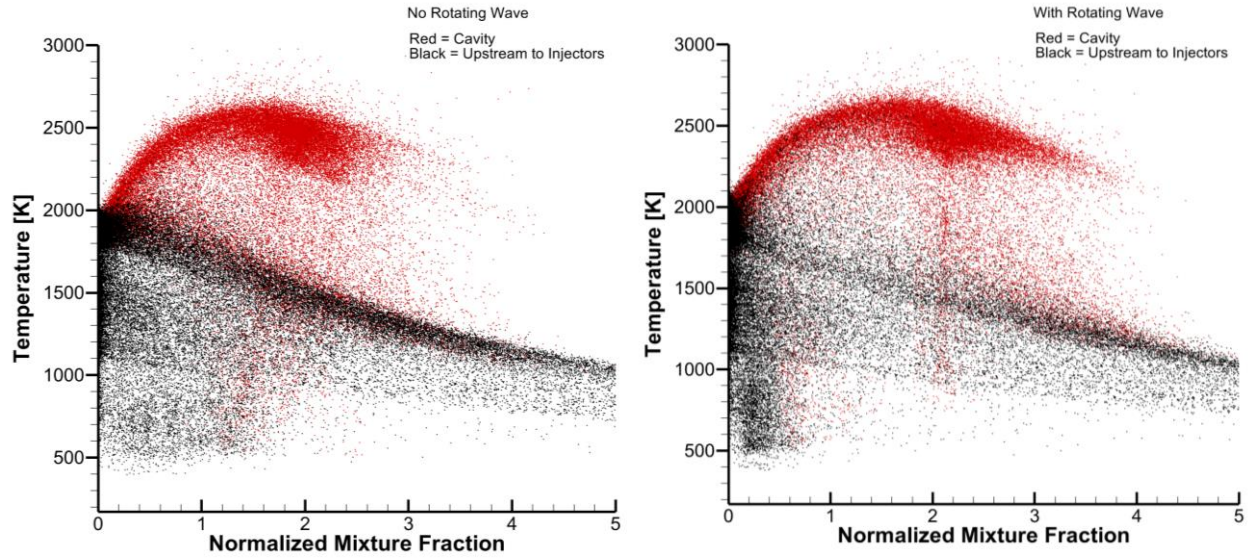


Fig. 13 Scatter plot of temperature vs normalized mixture fraction for points in between the injectors and cavity closeout in the P4 solution. All points which are at a larger radius than the isolator wall are excluded. The points are split between those in the isolator region (black), and those in the cavity region (red).

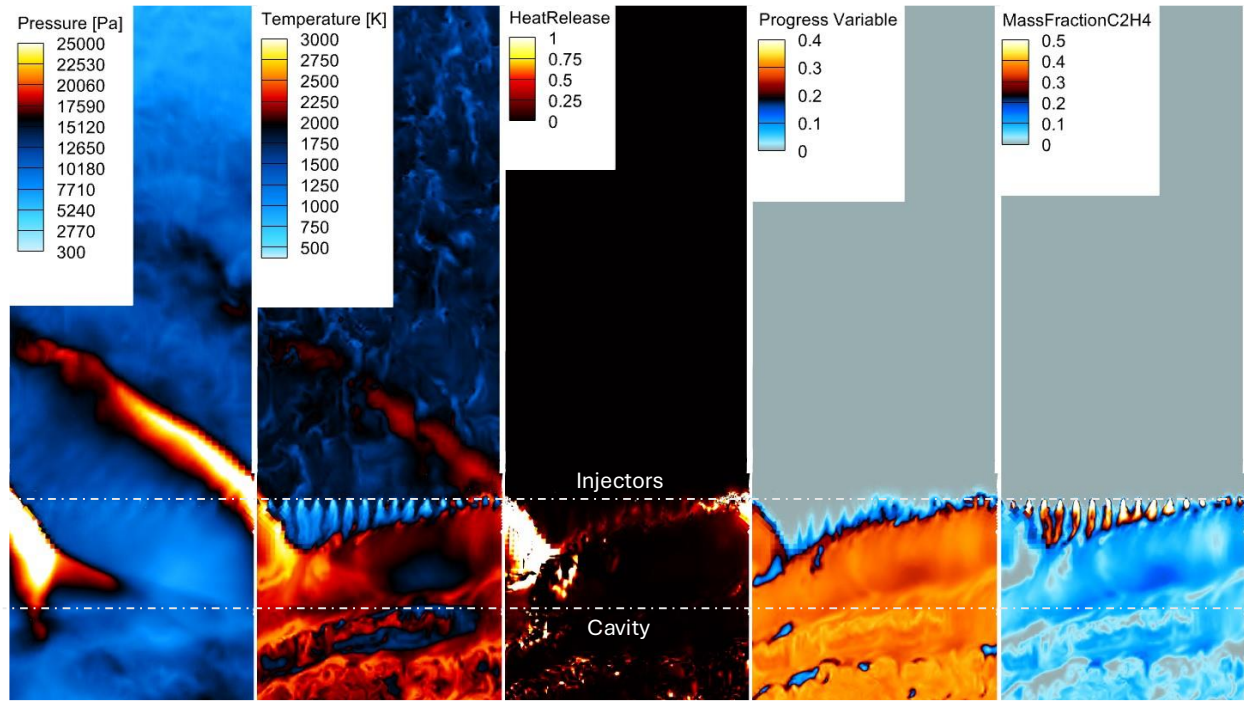


Fig. 14 Constant radius unwrapped view of the P2 solution during the pressure wave phenomenon. Taken at a radial value of 16mm, 1.5mm off the isolator wall.

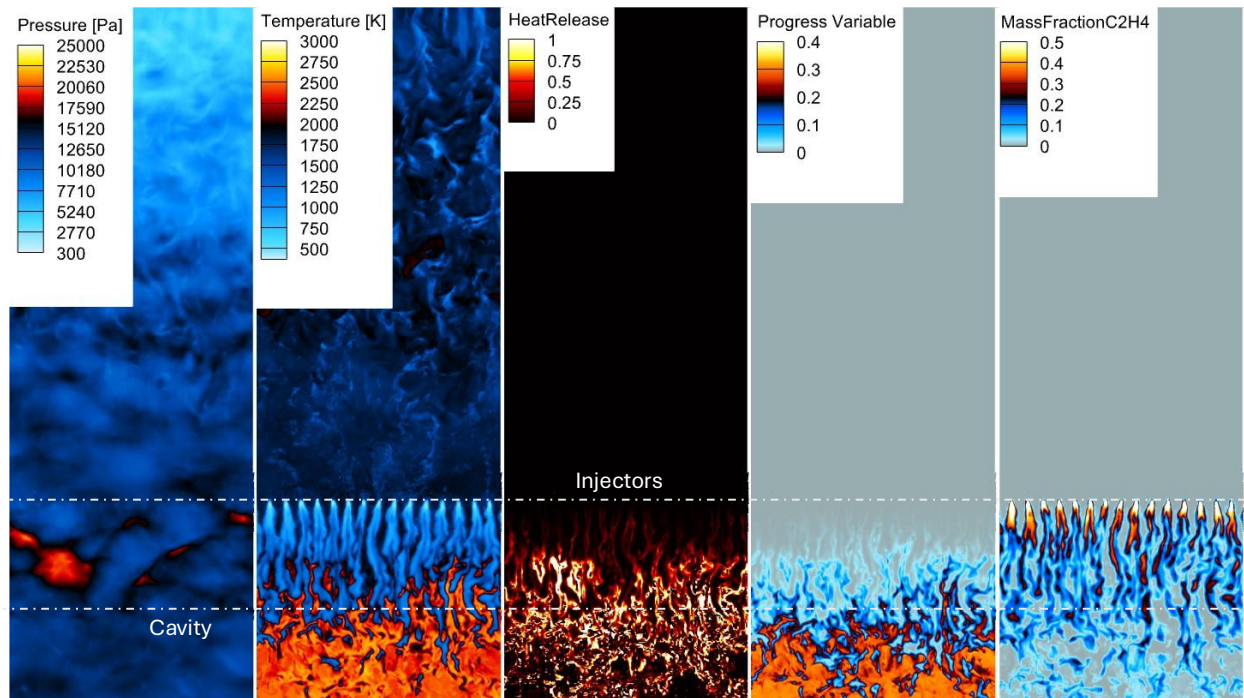


Fig. 15 Constant radius unwrapped view of the P4 solution during the pressure wave phenomenon. Taken at a radial value of 16mm, 1.5mm from off isolator wall.

regions just after the injectors being primarily a diffusion flame which transitions to a rich premixed flame downstream in the cavity region. The effect of the pressure waves seems to be the augmentation of the combustion intensity at fuel-rich conditions in both regions as seen in the increased temperature of the surveyed points. It also seems to be moving the solution away from laminar flamelet-like combustion by the deterioration of the linear relationship between mixture fraction and temperature in the injector jet region.

For a better look at the pressure wave, Figs. 14 and 15 show unwrapped views of the P2 and P4 solutions respectively. These solutions are from a point in the evolution of the flow where the wave is most pronounced. These are isosurfaces of constant radius which have been “unwrapped” or “unrolled” to be flat. A radius of 16 mm is chosen since it shows the strongest signal of the pressure wave. In this view P2 shows a response which is remarkably similar to the typical operation of a rotating detonation engine. The fuel and air begin to mix coming out of the injectors and a detonation wave is propagating left to right through the partially premixed region resulting in incomplete combustion. Some reactions continue downstream in the cavity as air is mixed with fuel that escapes the passage of the detonation wave. Upon refining the solution to P4, we see a very different response. The combustion is occurring more in line with the expectation of a partially premixed jet-wake stabilized flame. Instead of a single dominant pressure wave, there appears to be a few different helical pressure waves which transit throughout the combustor. These do not appear to be detonation waves. Rather they seem to resemble acoustic waves which could be sustaining their energy from the local augmentation of reaction rates in the regions of higher pressure.

The closest comparison to this phenomenon that the authors are aware of in the present literature on rotating detonation combustors (RDCs) is the flow-through RDC (FT-RDC) concept studied by Wiggins et. al. [36-37] and Schwer et. al. [38]. A large difference between these methods is the injection scheme. The present study utilizes 16 injectors around the circumference of a constant area section; the numerical study in Ref. [38] modeled an “injection surface” of premixed fuel and oxidizer which constituted an expansion in flowpath radius from the incoming core flow of inert gas; the experimental studies of Refs. [36-37] have an annular injection scheme and only a small portion of the forward surface of the combustor is open for a core flow of premixed fuel and oxidizer. The uniqueness of the present mode of operation is the fact that the core flow provides all the oxygen for combustion, and that the detonation exhausts into a subsonic cavity combustor instead of a plenum or nozzle. The fact that refining the order of the solution prevents the wave from forming suggests that the mixing provided by the numerical dissipation of the lower orders is essential to this process. Thus, a step towards realizing this phenomenon physically would be to explore configurations with enhanced mixing of the injectors and a more even circumferential distribution of fuel.

K. Computational Cost Scaling

Equally as important as the resolution is the cost of the method. The relative cost-scaling is included in Table 5. The behavior of this case is somewhat unusual, since the timestep would typically need to be decreased proportional to $1/P^2$ to satisfy the CFL stability condition. However, we notice that the limit for stable time stepping appears to be somewhat constant, at least for P2 through P3. This is a result of the fact that the limiting cells for time step size are being filtered, and their effective order of accuracy is not changing. These regions include the leading edge of the inlet, which has the largest gradients and fastest characteristics at the wall but is being filtered by the shock-capturing scheme, as well as the small cells at the orifice of the injector, which are filtered manually by a sponge filter. Despite the favorable scaling observed, the method is still very expensive to run. The cost per millisecond of flow time for the P1 discretization is 35,000 core-hours using 6,144 MPI processes across 32 compute nodes, with each node containing two AMD EPYC 9654 96-Core CPU processors. All the timing data in Table 5 was collected using the same number and type of processors.

Table 5 Cost as a function of P-order scaling.

	# of Solution Points	Relative wall-time per Runge-Kutta Stage	Non-Dimensional Timestep Size	Relative cost for a given flow time
P1	3.4 M	1	5.4 E-6 (11-Stage)	1
P2	9.5 M	2.14	1.6 E-6 (SSP RK3)	2.0
P3	20.4 M	3.97	2.0 E-6 (SSP RK3)	2.9
P4	38.4 M	7.33	1.6 E-6 (SSP RK3)	6.5

V. Conclusion

The present work has shown that the FR-DG method equipped with our particular stabilization method is able to simulate a HSABP combustor geometry with a high resolution of the turbulent structures present in the shock train and fuel jet regions of the flow. The shock-capturing method presented here is shown to stabilize the solution, but as it does so, it adds an excessive amount of numerical dissipation at weaker shock waves. When compared against NCSU's REACTMB code, a contemporary high-order solver which uses a stencil-based finite-volume reconstruction scheme, we observe a greater resolution of turbulent structures but a worse resolution of shockwaves.

The present work failed to recreate the stable ram-mode combustion observed in previous numerical and experimental analyses of the UIUC cavity combustor at an equivalence ratio of 1.24. All analyses of the flowpath using GFR showed unstable operation of the combustor eventually leading to unstart accompanied by the presence of helical pressure waves that appear to be akin to a rotating detonation wave under some conditions. Lower-order runs with higher numerical dissipation consistently capture a single rotating detonation wave present in the isolator aft of the injectors. Higher-order P3 and P4 runs did not form the same rotating wave and instead had a much more intermittent response which often contained multiple smaller waves appearing temporarily. Despite the refinement of the polynomial order, the flowpath would still unstart. Factors which have potential effects on the unstart are the solution stabilization method adding too much dissipation, the simple chemical mechanism which is operating well below the pressure it was designed for, and the coarse grid in the powerhead region which cannot resolve shocks particularly well. Future work will investigate these factors by exploring improved methods for solution stabilization, the addition of a turbulence chemistry interaction (TCI) model to alter the effective reactivity of the chemical mechanism, and grid refinement in the injector and fuel mixing regions.

Acknowledgments

We acknowledge the computational resources provided by the Department of War's High Performance Computing Modernization Program. Additionally, resources supporting this work were provided by the NASA High-End Computing (HEC) Program through the NASA Advanced Supercomputing (NAS) Division at Ames Research Center.

We acknowledge the use of the GFR software, provided by the Inlets and Nozzles branch at NASA Glenn Research Center, as well as the guidance and support of Dr. Seth Spiegel, the lead developer of GFR. A portion of this work was completed during an internship supported by the NASA Office of Stem Engagement at Glenn Research Center.

References

- [1] Hesthaven, J. S. and Warburton, T., Nodal Discontinuous Galerkin Methods: Algorithms, Analysis, and Applications, Vol. 54 of Texts in Applied Mathematics, Springer New York, 2008.
- [2] Bassi, F. and Rebay, S., A High Order Discontinuous Galerkin Method for Compressible Turbulent Flows," Discontinuous Galerkin Methods: Theory, Computation, and Application, edited by B. Cockburn, G. E. Karniadakis, and C.-W. Shu, Lecture Notes in Computational Science and Engineering, Springer, 2000, pp. 77-88.
- [3] Cockburn, B., Shu, C.-W., The local discontinuous Galerkin method for time-dependent convection-diffusion systems, SIAM J. Numer. Anal. 35 (6) (1998) 2440-2463
- [4] Cockburn, B., "Discontinuous Galerkin Methods for Computational Fluid Dynamics," *Encyclopedia of Computational Mechanics*, edited by E. Stein, R. Borst, and T. J. R. Hughes, Wiley, 2004. <https://doi.org/10.1002/0470091355.ecm053>
- [5] Huynh, H., A Flux Reconstruction Approach to High-Order Schemes Including Discontinuous Galerkin Methods," AIAA Paper 2007-4079, Jun 2007.
- [6] Huynh, H., A Reconstruction Approach to High-Order Schemes Including Discontinuous Galerkin for Diffusion," AIAA Paper 2009-403, Jan 2009.
- [7] Huynh, H., High-Order Methods Including Discontinuous Galerkin by Reconstructions on Triangular Meshes," AIAA Paper 2011-44, Jan 2011.
- [8] Arndt, D., Fehn, N., Kanschat, G., Kormann, K., Kronbichler, M., Munch, P., Wall, W.A., Witte, J., ExaDG: High-order discontinuous Galerkin for the exa-scale, in: Lecture Notes in Computational Science and Engineering, Springer International Publishing, 2020, pp. 189-224
- [9] Hoskin, D.S., Van Heyningen, R.L., Nguyen, N.C., Vila-Pérez, J., Harris, W.L., Peraire, J., Discontinuous Galerkin Methods for Hypersonic Flows. Progress in Aerospace Sciences, vol. 146, Apr. 2024
- [10] Barter, G. E. (Garrett E., "Shock Capturing with PDE-Based Artificial Viscosity for an Adaptive, Higher-Order Discontinuous Galerkin Finite Element Method," Thesis. Massachusetts Institute of Technology, 2008.

- [11] Yu, M. L., Giraldo, F. X., Peng, M., and Wang, Z. J., “Localized Artificial Viscosity Stabilization of Discontinuous Galerkin Methods for Nonhydrostatic Mesoscale Atmospheric Modeling,” *Monthly Weather Review*, Vol. 143, No. 12, 2015, pp. 4823–4845. <https://doi.org/10.1175/MWR-D-15-0134.1>
- [12] Moro, D., Nguyen, N. C., and Peraire, J., “Dilation-based Shock Capturing for High-order Methods,” *International Journal for Numerical Methods in Fluids*, Vol. 82, No. 7, 2016, pp. 398–416. <https://doi.org/10.1002/fld.4223>
- [13] Messaï, N.-A., Daviller, G., and Boussuge, J.-F., “Artificial Viscosity-Based Shock Capturing Scheme for the Spectral Difference Method on Simplicial Elements,” *Journal of Computational Physics*, Vol. 504, 2024, p. 112864. <https://doi.org/10.1016/j.jcp.2024.112864>
- [14] Dzanic, T., and Witherden, F. D., “Positivity-Preserving Entropy-Based Adaptive Filtering for Discontinuous Spectral Element Methods,” *Journal of Computational Physics*, Vol. 468, 2022, p. 111501. <https://doi.org/10.1016/j.jcp.2022.111501>
- [15] Trojak, W., and Dzanic, T., “Positivity-Preserving Discontinuous Spectral Element Methods for Compressible Multi-Species Flows,” *Computers & Fluids*, Vol. 280, 2024, p. 106343. <https://doi.org/10.1016/j.compfluid.2024.106343>
- [16] Huynh, H. T., “Shock Capturing via Limiting for High-Order Methods Including Discontinuous Galerkin and Flux Reconstruction,” *Computers & Fluids*, Vol. 300, 2025, p. 106743. <https://doi.org/10.1016/j.compfluid.2025.106743>
- [17] Smith, S., Gruber, M., Steiner, R., Collatz, M., and Mathur, T., “Development and Calibration of an Axisymmetric Direct-Connect Supersonic Combustion Flowpath” AIAA Paper 2009-5035, Aug. 2009. <https://doi.org/10.2514/6.2009-5035>
- [18] Gruber, M. R., Smith, S., and Mathur, T., “Experimental Characterization of Hydrocarbon-Fueled, Axisymmetric, Scramjet-Combustor Flowpaths,” AIAA Paper 2011-2311, June 2012. <https://doi.org/10.2514/6.2011-2311>
- [19] Landsberg, W., Vanyai, T., McIntyre, T., and Veeraragavan, A., “Experimental Scramjet Combustion Modes of Hydrocarbon Mixtures at Mach 8 Flight Conditions,” *AIAA Journal*, Vol. 58, No. 12, 2020, pp. 5117–5122. <https://doi.org/10.2514/1.J059856>
- [20] Baccarella, D., Liu, Q., Lee, T., Hammack, S., and Do, H., “The Supersonic Combustion Facility ACT-2,” AIAA Paper 2017-0103, Jan. 2017. <https://doi.org/10.2514/6.2017-0103>
- [21] Baccarella, D., Liu, Q., McGann, B., Lee, T., and Do, H., “Experimental Study of Heat Release Induced Choking in a Supersonic Circular Combustor,” AIAA Paper 2018-1139, Jan. 2018. <https://doi.org/10.2514/6.2018-1139>
- [22] Liu, Q., Baccarella, D., McGann, B., and Lee, T., “Dual-Mode Operation and Transition in Axisymmetric Scramjets,” *AIAA Journal*, Vol. 57, No. 11, 2019, pp. 4764–4777. <https://doi.org/10.2514/1.J058391>
- [23] Liu, Q., Baccarella, D., McGann, B., and Lee, T., “Cavity-Enhanced Combustion Stability in an Axisymmetric Scramjet Model,” *AIAA Journal*, Vol. 57, No. 9, 2019, pp. 3898–3909. <https://doi.org/10.2514/1.J058204>
- [24] Hash, C., Drummond, P., Edwards, J., Kato, N., Liu, Q., and Lee, T., “Numerical Simulation of Stable and Unstable Ram-Mode Operation of an Axisymmetric Ethylene-Fueled Inlet-Isolator-Combustor Configuration,” *Combustion and Flame*, Vol. 242, No. 8, 2022, Paper 112157. <https://doi.org/10.1016/j.combustflame.2022.112157>
- [25] Hash, C., Drummond, P., Edwards, J., Kato, N., Liu, Q., and Lee, T., “Numerical Simulation of Unstart Processes in an Axisymmetric Scramjet Combustor,” AIAA Paper 2022-1849, Jan. 2022. <https://doi.org/10.2514/6.2022-1849>
- [26] Richardson, S., Edwards, J. R., Hash, C. A., Drummond, P. M., Lee, T., and Kato, N., “Numerical Simulation of Scram-Mode Operation of an Axisymmetric Combustor in an Arc Tunnel,” *Journal of Propulsion and Power*, Vol. 41, No. 4, 2025, pp. 429–439. <https://doi.org/10.2514/1.B39754>
- [27] Castonguay, P., Vincent, P. E., and Jameson, A., “A New Class of High-Order Energy Stable Flux Reconstruction Schemes for Triangular Elements,” *Journal of Scientific Computing*, Vol. 51, No. 1, 2012, pp. 224–256. <https://doi.org/10.1007/s10915-011-9505-3>
- [28] Trojak, W., Watson, R., and Vincent, P., “An Extended Range of Stable Flux Reconstruction Schemes on Quadrilaterals for Various Polynomial Bases,” *Journal of Scientific Computing*, Vol. 96, No. 1, 2023, p. 12. <https://doi.org/10.1007/s10915-023-02216-3>
- [29] Wilke, C. R., “A Viscosity Equation for Gas Mixtures,” *Journal of Chemical Physics*, Vol. 18, No. 4, 1950, pp. 517–519. <https://doi.org/10.1063/1.1747673>
- [30] Spiegel, S. C., DeBonis, J. R., and Huynh, H. T., “Overview of the NASA Glenn Flux Reconstruction Based High-Order Unstructured Grid Code,” San Diego, California, USA, 2016. <https://doi.org/10.2514/6.2016-1061>
- [31] Spiegel, S. C., Borghi, M. R., and Yoder, D. A., “Large Eddy Simulations of a Single-Injector Cooling Flow Using the High-Order Flux Reconstruction Method,” San Diego, CA & Virtual, 2022. <https://doi.org/10.2514/6.2022-1813>
- [32] Spiegel, S. C., Yoder, D. A., DeBonis, J. R., Huynh, H. T., Heinlein, G., Borghi, M. R., and Georgiadis, N. J., “New Capabilities and Improvements to the High-Order Glenn Flux Reconstruction Code,” Orlando, FL, 2025. <https://doi.org/10.2514/6.2025-0061>
- [33] Edwards, J. R., “A Low-Diffusion Flux-Splitting Scheme for Navier-Stokes Calculations,” *Computers & Fluids*, Vol. 26, No. 6, 1997, pp. 635–659. [https://doi.org/10.1016/S0045-7930\(97\)00014-5](https://doi.org/10.1016/S0045-7930(97)00014-5)
- [34] Bassi, F. and Rebay, S., “A High Order Discontinuous Galerkin Method for Compressible Turbulent Flows,” *Discontinuous Galerkin Methods: Theory, Computation, and Application*, edited by B. Cockburn, G. E. Karniadakis, and C.-W. Shu, Lecture Notes in Computational Science and Engineering, Springer, 2000, pp. 77–88
- [35] Hassan, E. A., Hageman, M. D., Knadler, M. S., and Lucas, S., “Error In Shock Tube Ignition Delay Time Predictions Due to Bifurcation and Boundary Layer Effects,” Orlando, FL, 2025. <https://doi.org/10.2514/6.2025-2140>
- [36] Wiggins, R., Pritschau, T., and Gutmark, E., “Simultaneous Shadowgraph and CH* Chemiluminescence of a Flow-Through Rotating Detonation Combustor,” *AIAA Journal*, Vol. 63, No. 6, 2025, pp. 2261–2271. <https://doi.org/10.2514/1.J064341>

- [37] Wiggins, R., and Gutmark, E., “Dynamics of Detonation Wave–Core Combustion Interaction in a Flow-Through Rotating Detonation Combustor,” *AIAA Journal*, 2025, pp. 1–23. <https://doi.org/10.2514/1.J065988>
- [38] Schwer, D. A., Johnson, R. F., Kercher, A., Kessler, D. A., and Corrigan, A. T., “Numerical Investigation of Centerbody-Less Rotating Detonation Combustors,” Orlando, FL, 2020. <https://doi.org/10.2514/6.2020-2158>