Integrating a Rotating Detonation Combustor with a Power Generating Gas Turbine to Realize the Pressure Gain

Project #: FE0032077
2021 UTSR Project Review Meeting
Department of Energy

November 8, 2021

Ajay K Agrawal, PI, University of Alabama
Joseph Meadows, co-PI, Virginia Tech

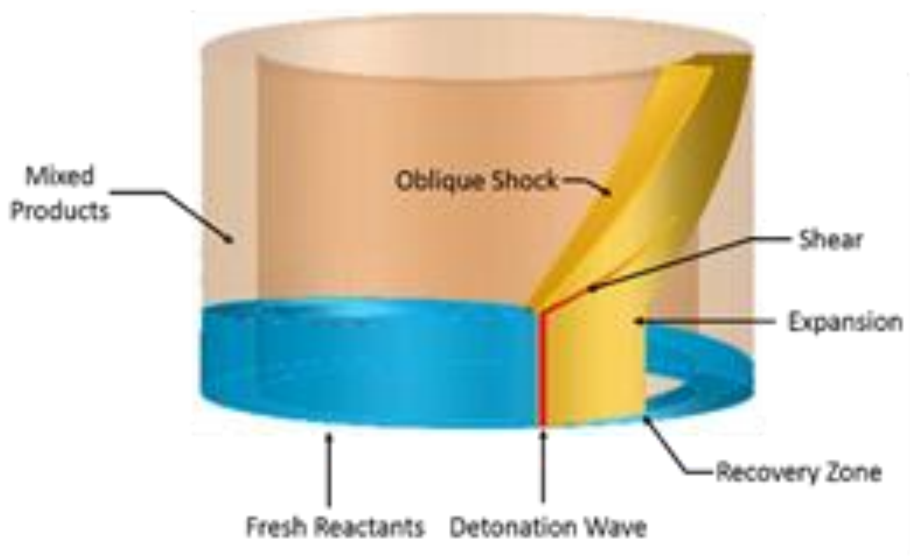
COMBUSTION LABORATORIES
THE UNIVERSITY OF ALABAMA
LOW EMISSIONS • HIGH EFFICIENCY • FUEL FLEXIBILITY

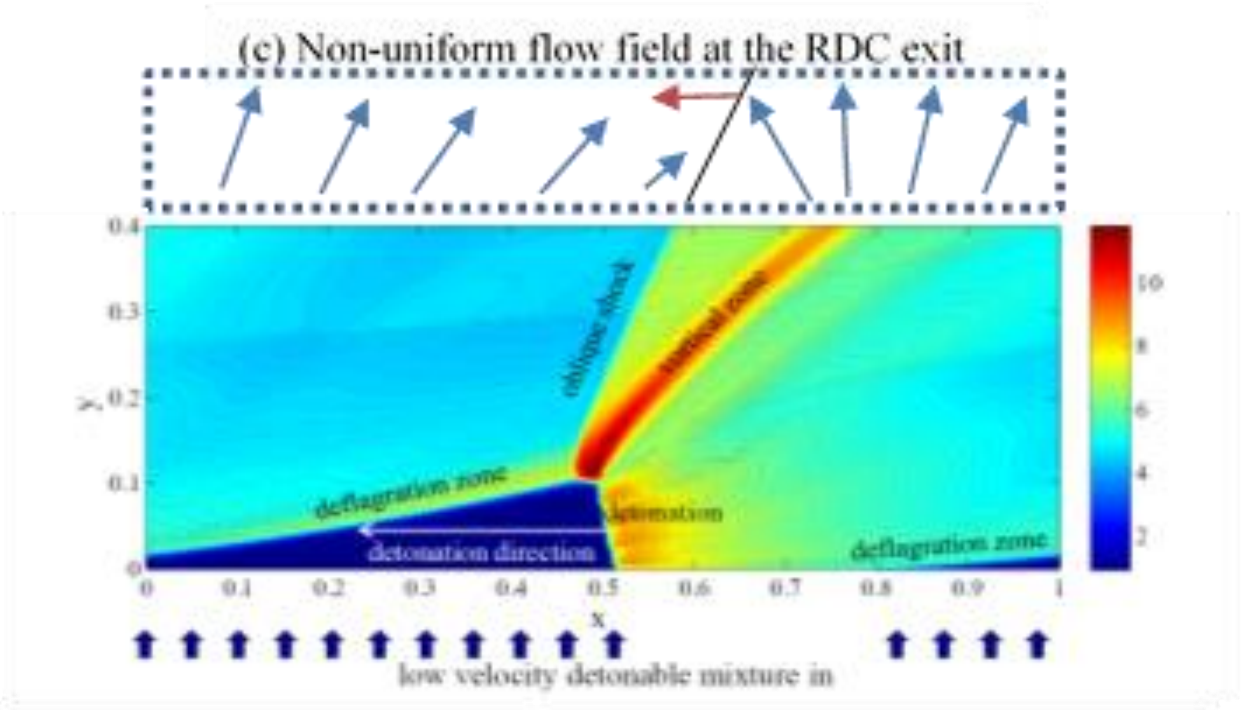
THE UNIVERSITY OF ALABAMA®

VIRGINIA TECH

Project Background

- Rotating detonation combustion (RDC) can result in pressure gain instead of the pressure loss typically experienced during the heat release process.
- However, the shock laden RDC exit flow is highly unsteady and spatially non-uniform (hydrodynamically and thermally) with a high degree of periodicity.
- In contrast, gas turbines are designed to operate with relatively small velocity and temperature at the inlet variations.
- For practical considerations, RDC flow must be properly conditioned to avoid the potentially disastrous negative impacts of flow oscillations on turbine operation, reliability, and maintainability in the long term, and to achieve the overall performance.





Background

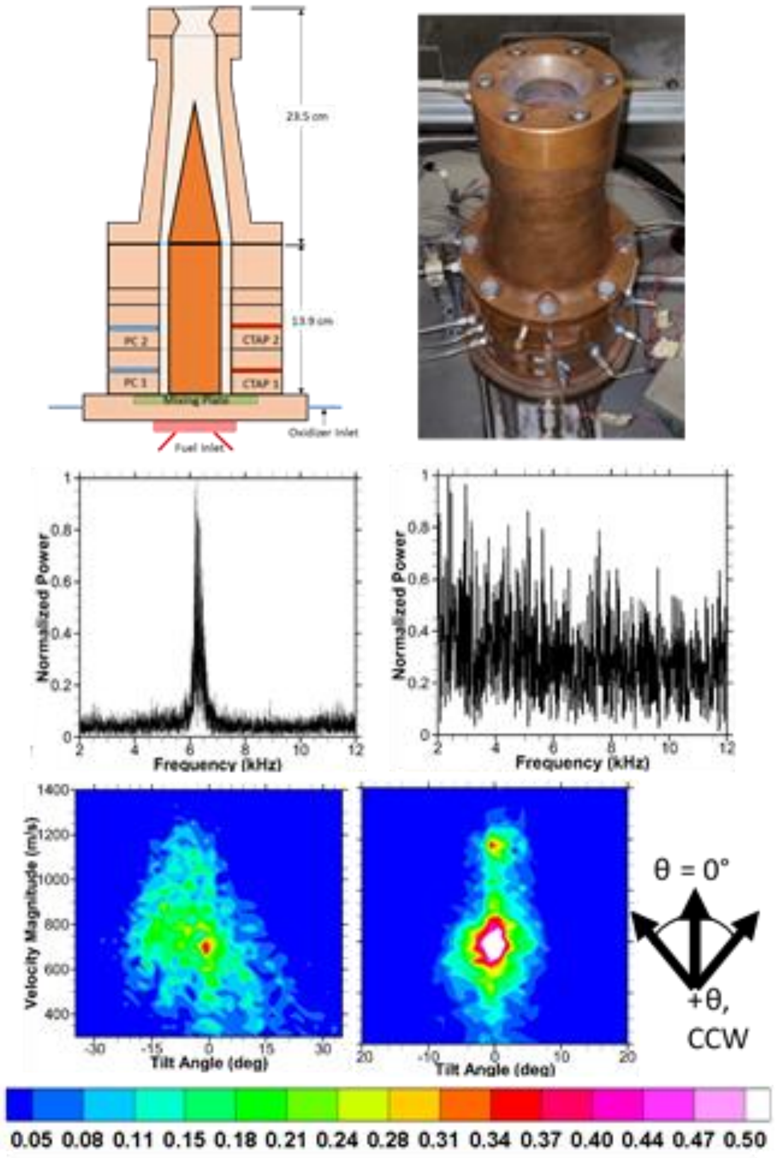


Fig. 4 Schematic (top-left) and photograph (top-right) of RDC with diffuser; FFT of axial velocity without (middle-left) and with diffuser (middle-right); 2D histogram of velocity without diffuser (bottom-left) and with diffuser (bottom-right) [20].

- We have employed PIV at 30 kHz to measure the flow field at the RDC exit without and with diffuser aimed at attenuating the unsteady flow. The integrated RDC-diffuser system was operated at high chamber pressures (about 450 kPa) by employing a CD nozzle to restrict the exit flow area.
- FFT plots of axial velocity show a sharp peak at 6.2 kHz (same as the detonation wave frequency) without the diffuser, but spectra with diffuser is incoherent indicating that the oblique shock wave and coherent flow structures or periodicity associated with it are eliminated by the diffuser.
- The 2D histograms show that the flow tilt angle decreases from about +/- 20 degrees without diffuser to less than +/- 8 degree with diffuser.
- However, this diffuser design did not mitigate the problematic axial flow oscillations; the axial velocity varied between 300 m/s and 1,200 m/s.

Project Objectives

The overall objective of this experimental/computational study is to develop validated strategies, guidelines, and designs for rotating detonation combustor (RDC)-turbine integration with the purpose of realizing pressure gain to extract maximum work in power generating gas turbines.

- The first phase will focus on area profiling of the RDC channel to minimize viscous losses by friction and oblique shock-induced phenomena.
- In the second phase, an integrated RDC-diffuser system will be developed for F-class gas turbines.
- In the third phase, the impact of loss mechanisms in the combustion process associated with inhomogeneous fuel/oxidizer mixing and parasitic combustion will be evaluated.

Pressure gain in the system will be quantified by measurements of

- CTAP, dynamic, and (possibly) total pressures and
- flow field by high-speed particle image velocimetry (PIV).
- In addition, ion-probes and schlieren imaging will be used to characterize the operation modes of the RDC at different test conditions.

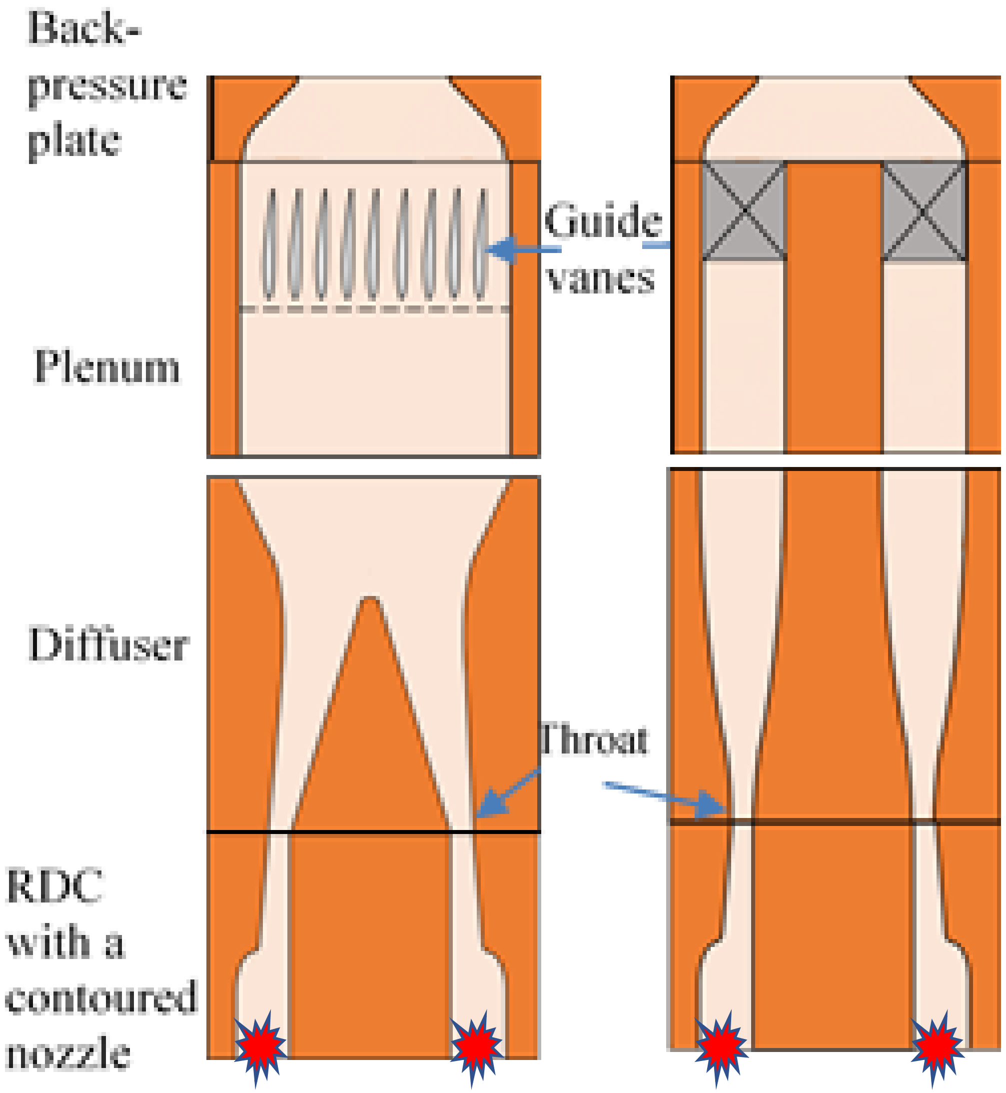


Fig. 1 Conceptual Diagram of RDC-turbine integration; F-class turbines (left) and aero-derivative turbines (right).

- In Fig. 1, the plenum with back-pressure plate is a versatile feature to simulate the turbine inlet conditions at high-pressures.
- The plenum is intended to impose the appropriate boundary conditions of the downstream turbomachinery, much like the reactant supply plenum (not shown) to represent upstream turbomachinery
- F-class gas turbines use a transition piece (circular to annular sector), which can be contained within the plenum, together with the inlet guide vanes, to fully replicate the hot gas flow path between the (can) combustor and the turbine inlet.
- Alternatively, for aeroderivative gas turbines, guide vanes can be placed in an annular configuration within the plenum.
- In this study, we will focus on the RDC and diffuser sections, which taken together constitute a converging-diverging (CD) geometry.

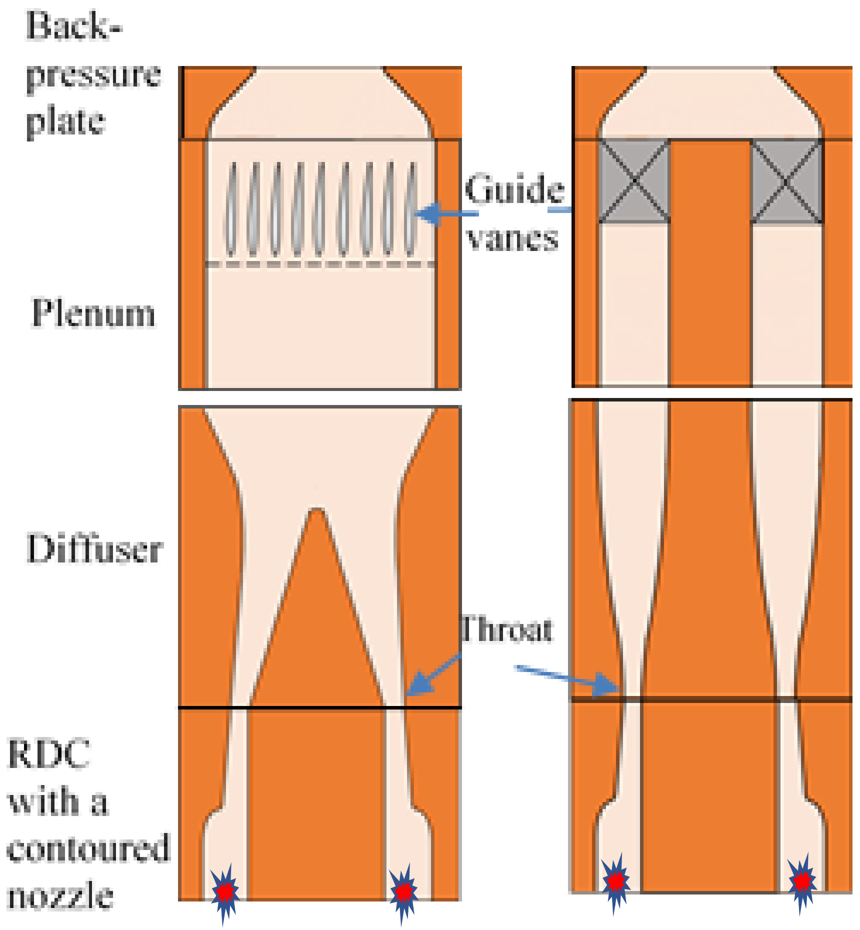


Fig. 1 Conceptual Diagram of RDC-turbine integration; F-class turbines (left) and aero-derivative turbines (right).

- Rankin et al. [2] used a CD nozzle to accelerate the flow to maximize the thrust for propulsion applications.
- However, for gas turbines, we must do the reverse, i.e., decelerate the flow in the diffuser downstream of the throat (where the area is the minimum) to recover the pressure.
- For this reason, the flow at the throat must ideally be sonic/subsonic at all or majority of the times.
- The rotating detonation inherently produces a supersonic flow, and hence, it must first be decelerated upstream of the throat which requires a converging section.
- However, the converging section will accelerate the subsonic portion of the flow emanating from the detonation region.
- The two types of flows also interact with each other in a complex 3D manner across the oblique shock wave, which by itself is a significant loss mechanism.
- Thus, *the supersonic and subsonic flows present opposing requirements, which must be balanced carefully to transition the flow in the converging section upstream of the throat*.

Technical Approach

- Unlike prior studies at low chamber pressures, we will utilize *a plenum with a backpressure plate* in both computational and experimental studies to impose the boundary conditions appropriate for power generating gas turbines. We expect to reach pre-detonation pressures of up to 12 atm.
- We will employ area profiling of the RDC channel to increase detonation stability, weaken the oblique shock wave and associated losses, and produce a more uniform and mostly subsonic (but close to sonic) flow at the throat region.
- Instead of a few test cases, large number of geometric and operating parameters will be investigated using URANS models with detailed chemical kinetics employing DoE approach to develop baseline and optimized designs of the RDC channel and diffuser.
- High-speed PIV at 100 kHz will be used to quantify the internal flow unsteadiness. PIV data will be acquired for the entire test duration to identify changes in mode characteristics during the experiments.
- Working with Kulite, we expect to implement advanced piezo-resistive pressure transducers to measure the total pressure in the RDC.
- Hydrogen and hydrogen blends with natural gas will be utilized, which will be a first, since most prior studies with RDC's have used either pure hydrogen or natural gas.

Technical Approach

Two steps

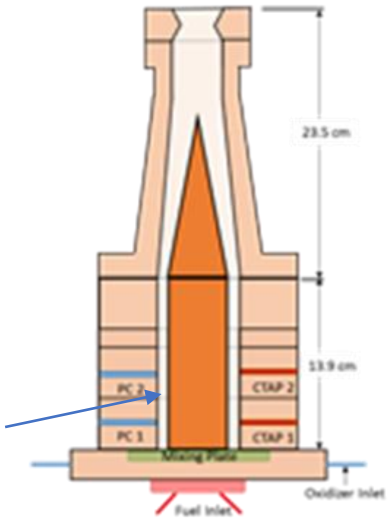
- First, optimize the cross-sectional area of the converging RDC channel to weaken the oblique shock structures and thus, achieve a more uniform and mainly subsonic but close to sonic flow conditions at the throat of the overall converging-diverging channel

- Second, profile the diverging diffuser section to decelerate the nozzle flow to achieve the Mach number desired at the turbine inlet.

In both cases, the downstream turbine inlet conditions are simulated by the high-pressure plenum with the capability to control the operating pressure.

RDC Channel Area Profiling

- Design of converging RDC channel is inspired by the CFD study of our recent work [Bell et al, 2021].
- Three area profiles along the RDC channel are shown: linear, gradual to rapid change (GTR), and rapid to gradual (RTG) change in the flow cross-sectional area along the flow path.
- Results show that the RTG case has much greater pressure gain (PG) compared to linear and GTR area profiles.
- *We plan to implement area profiling to design the annular RDC with improved detonation stability and performance.*



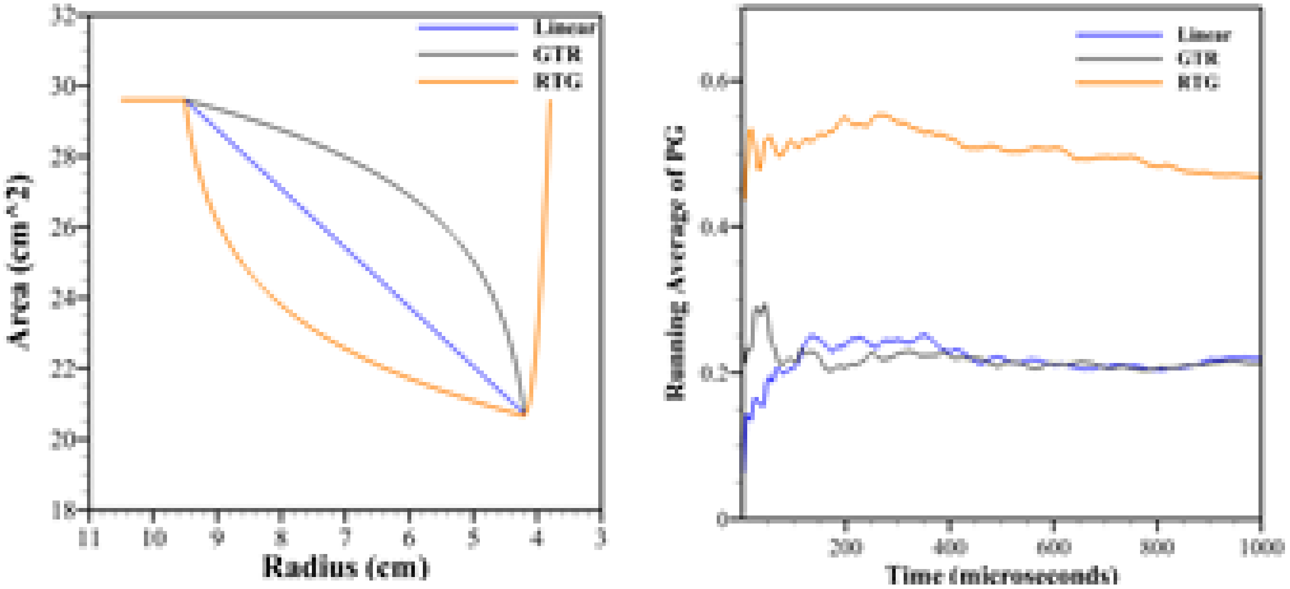


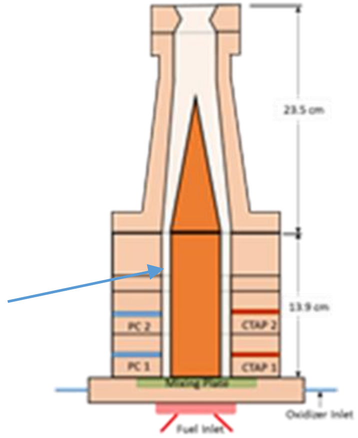
Fig. 8 Area variation with respect to radius in disk-RDC (top), and running average of pressure gain (bottom)

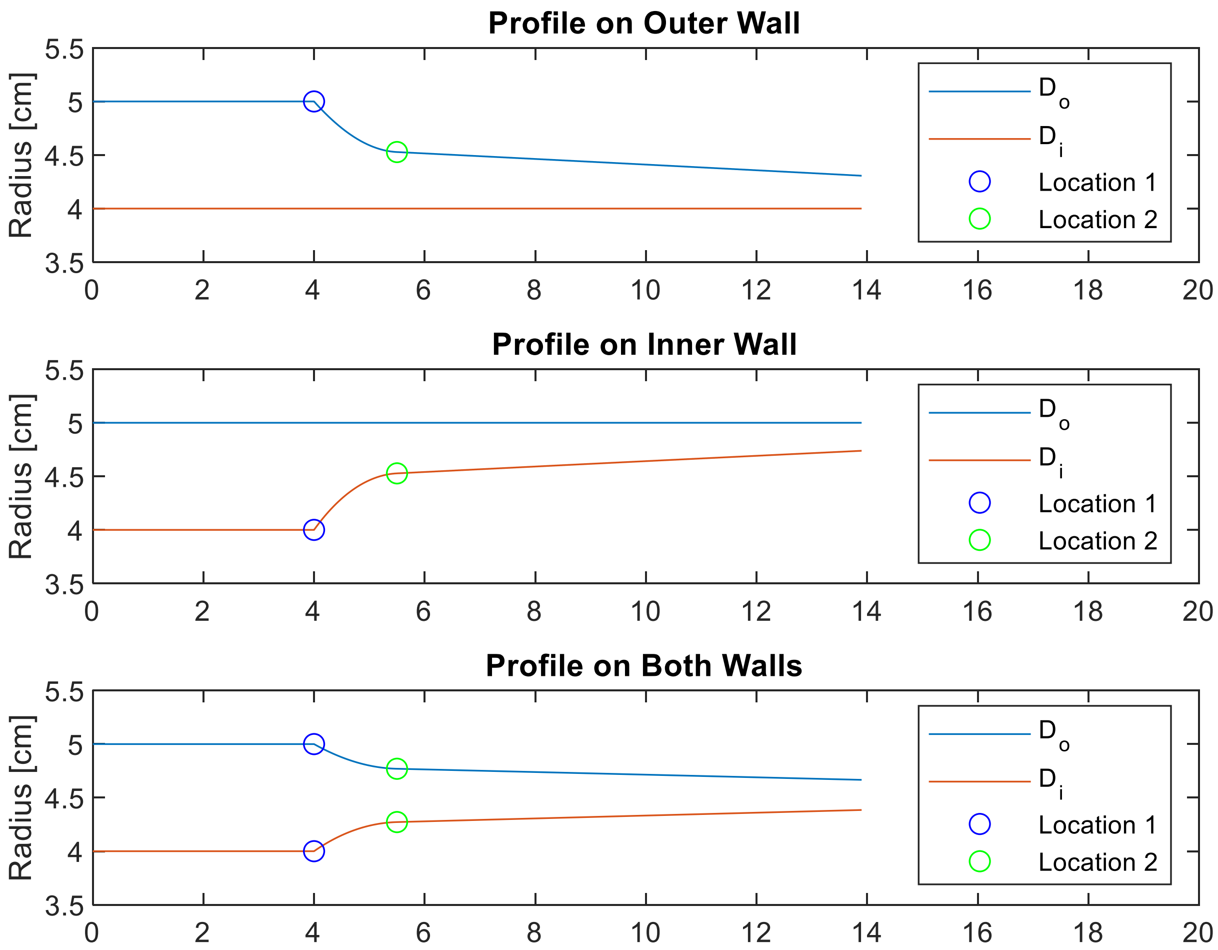
RDC Channel Profiling

Parametric CAD model for combustor annulus to do Design of Experiment (DoE) analysis.

Constraints:

1. Area profile between location 1 and location 2 is parabolic
2. Area profile between Location 2 and exit is Linear
3. Derivate on either side of the Location 2 (from parabolic and Linear) is the same.



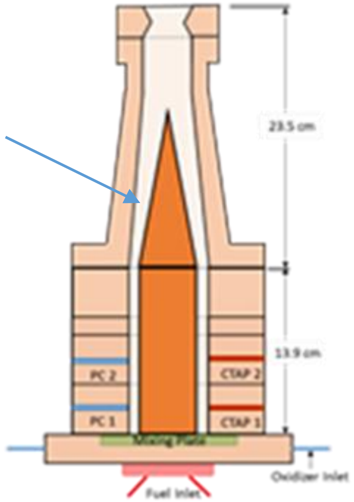


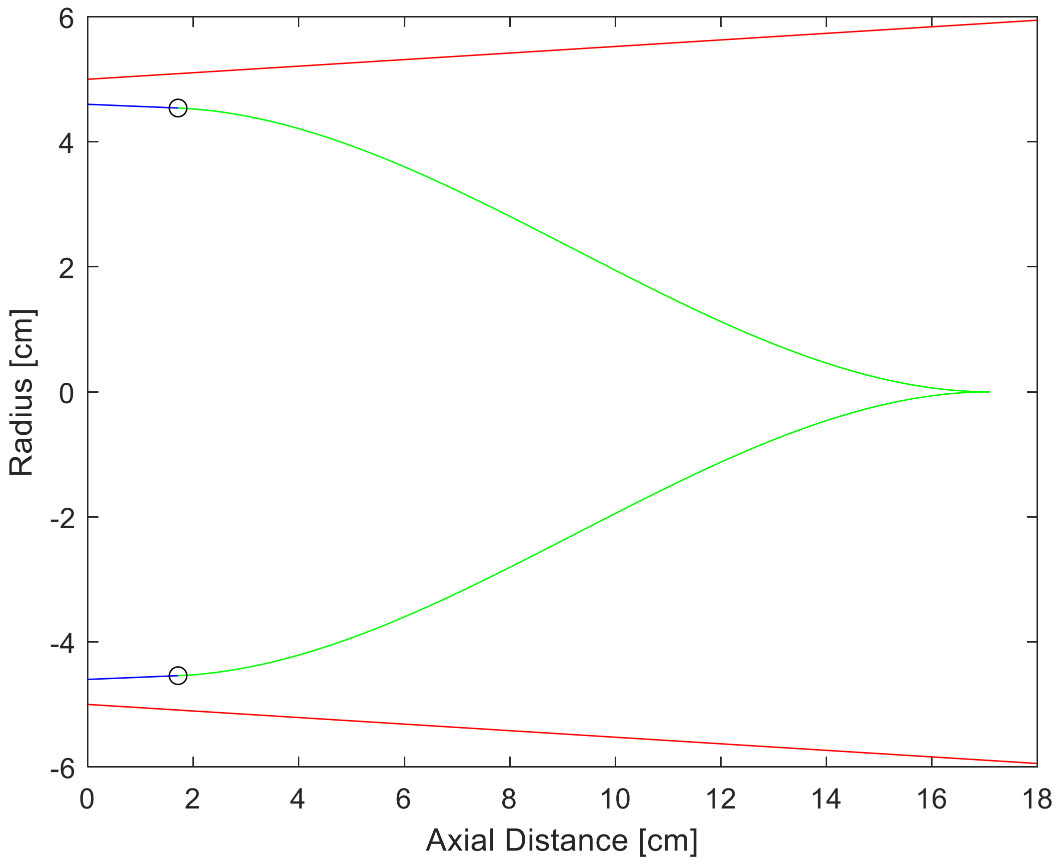
Diffuser cross-section Profiling

For parameterizing the inner and outer wall of the annulus following are the geometric parameters.

1. Location of transition from gradual to rapid in inner wall.
2. Initial Linear decay in inner wall.
3. Length of diffuser
4. Length of the inner wall

Constraints:
1. Outer wall diameter is varying linearly
2. Inner wall derivative at location of transition is the same on both sides.
3. Inner wall derivative at the tip is zero.





Preliminary Results

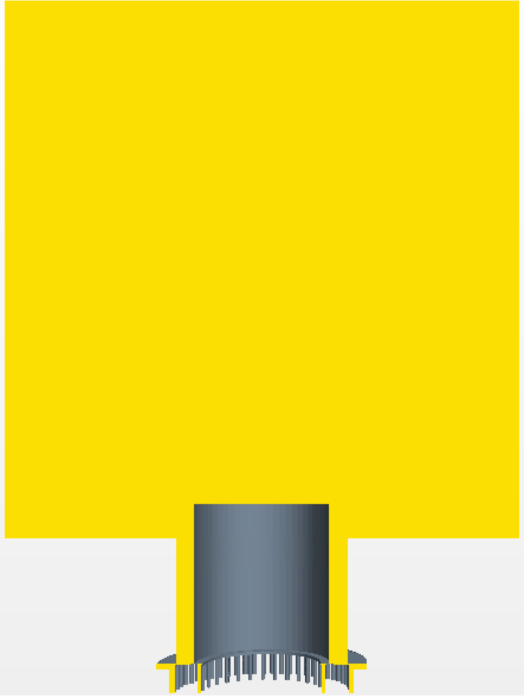
- CAD model of the RDE, based on the dimensions provided by UA.

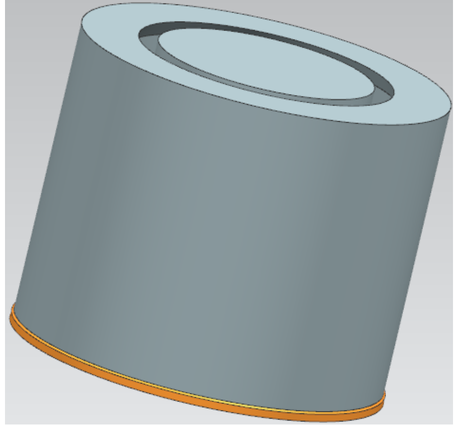
- The key features to consider in the CAD Model are:

1. The gap distance between the oxidizer manifold and the outer ring/wall.
2. The gap distance between the Fuel manifold and the Centerbody.
3. Nozzle dimension near the outlet of the combustor

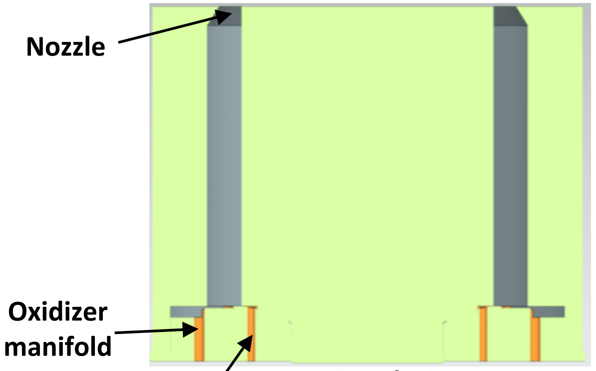
- Aero-solid model of the combustor for the 3D simulation.

- A plenum is created at the downstream section of the combustor (aero solid) to model the aspect of environmental (ambient) condition and to prevent the influence of the outlet boundary conditions on the flow exiting the combustor.

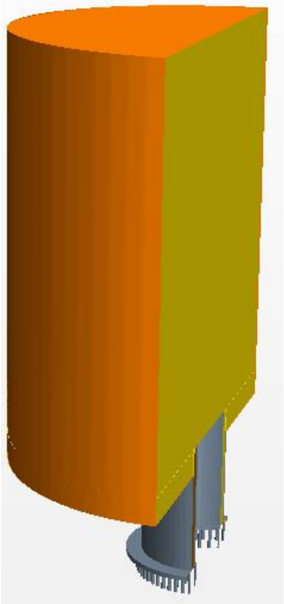




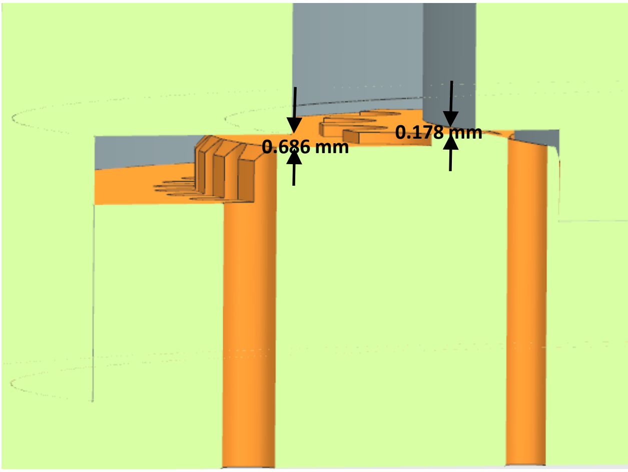
CAD Model



Sectional View



Aero Solid Model with Plenum



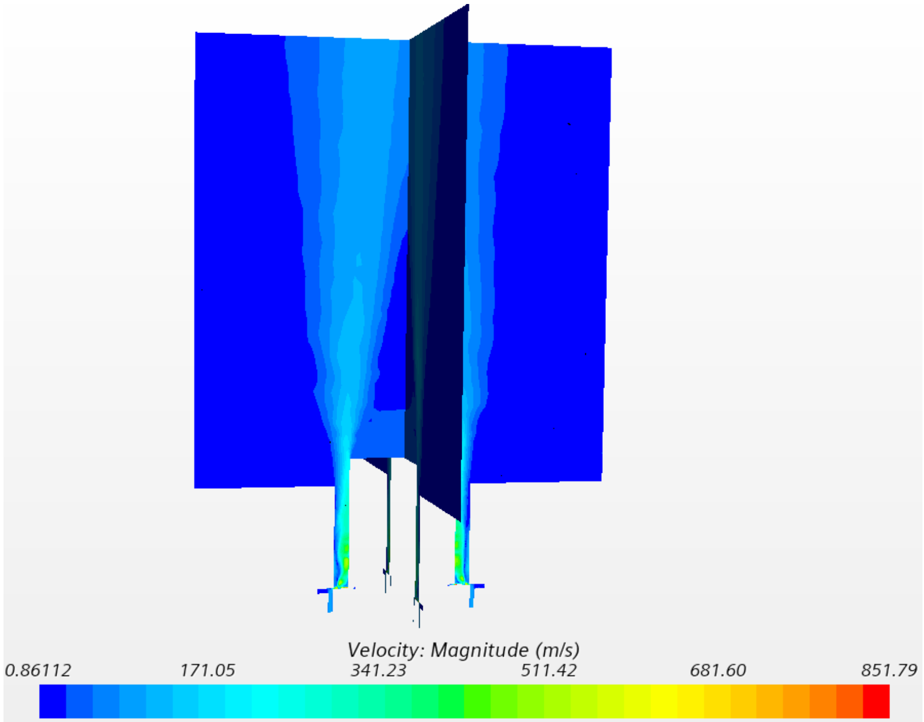
Zoomed Inlet View

Current Work

- 3D Coldflow simulation setup with downstream plenum.
- The outer walls and top surface of the plenum is given an ambient boundary condition.
- The operating conditions for the fuel and oxidizer inlet is selected based on the condition provided by UA

1. $\emptyset = 1$
2. Methane mass flowrate 0.073 kg/s
3. Oxygen mass flowrate 0.254 kg/s
4. Air mass flow rate 0.161 kg/s

- The inlet temperature is 300 K.
- REDRAM CH4/air mechanism file is used for the reaction mechanism along with GRI Mech 3.0 thermodynamic and transport data files.
- The base size is 2 mm. Mesh is refined in the injector and shock region to 0.3 mm.
- Mesh is refined to 0.4 mm near the interface connecting the RDC chamber to the plenum.
- To reduce the mesh count the mesh progressively get coarser in the plenum region. The total mesh count is $\sim 7.5\ million$.
- The geometry is divided into two halves strategically for reacting flow simulation.
- Converged Coldflow solution is obtained for the given operating condition.

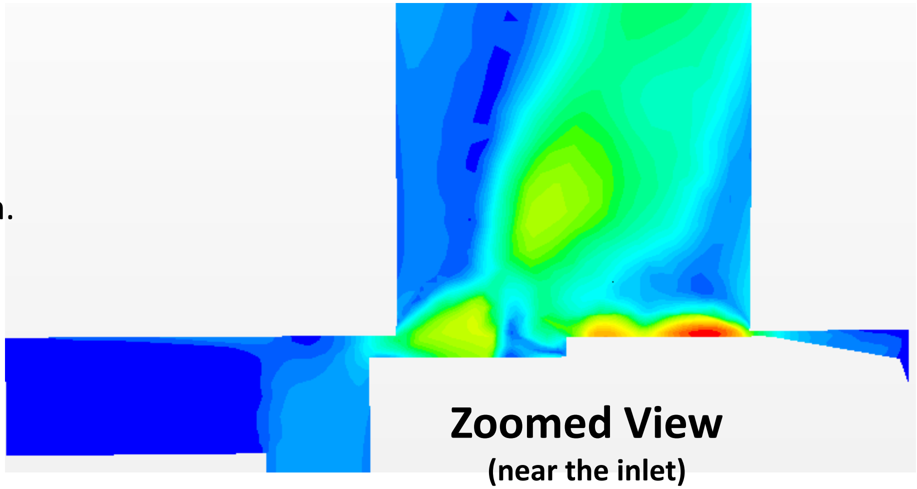


Coldflow solution

***Work in Progress**: 3D reacting simulation is currently running. Converged simulation is not yet achieved.

3D reacting simulation setup.

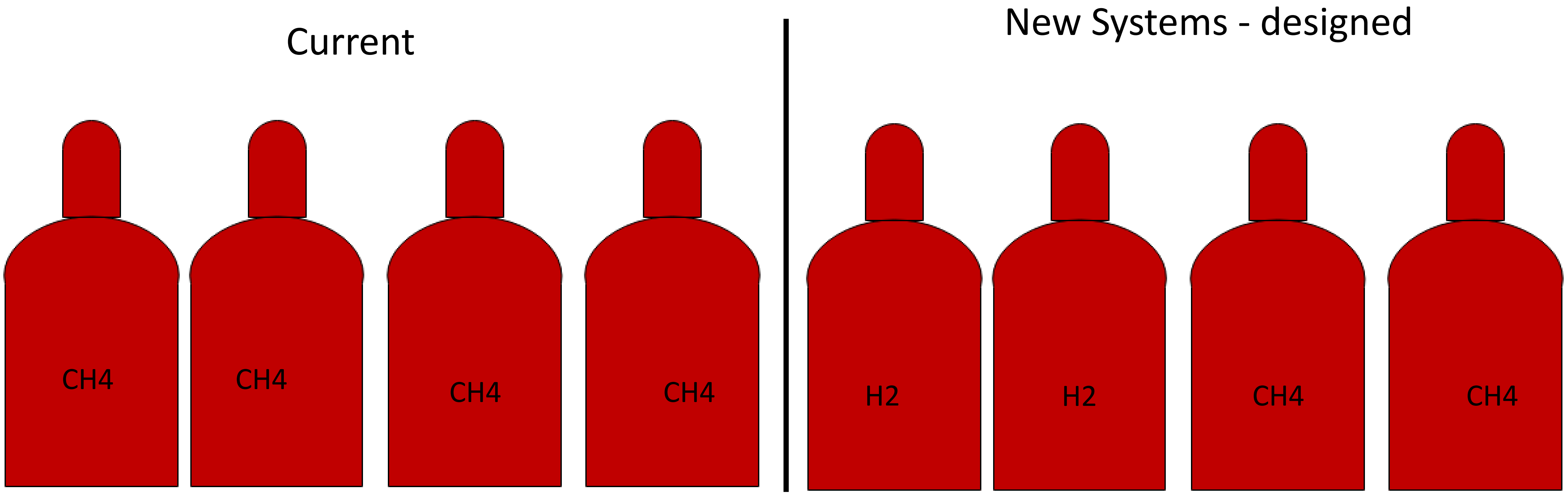
- The converged coldflow solution is utilized as an initial condition for the reacting simulation.
- Unsteady RANS with $k - \omega$ model is used.
- 3rd order MUSCL scheme is utilized for spatial discretization
- 2nd order implicit scheme is utilized for temporal discretization.
- Time step is 0.1 μs.



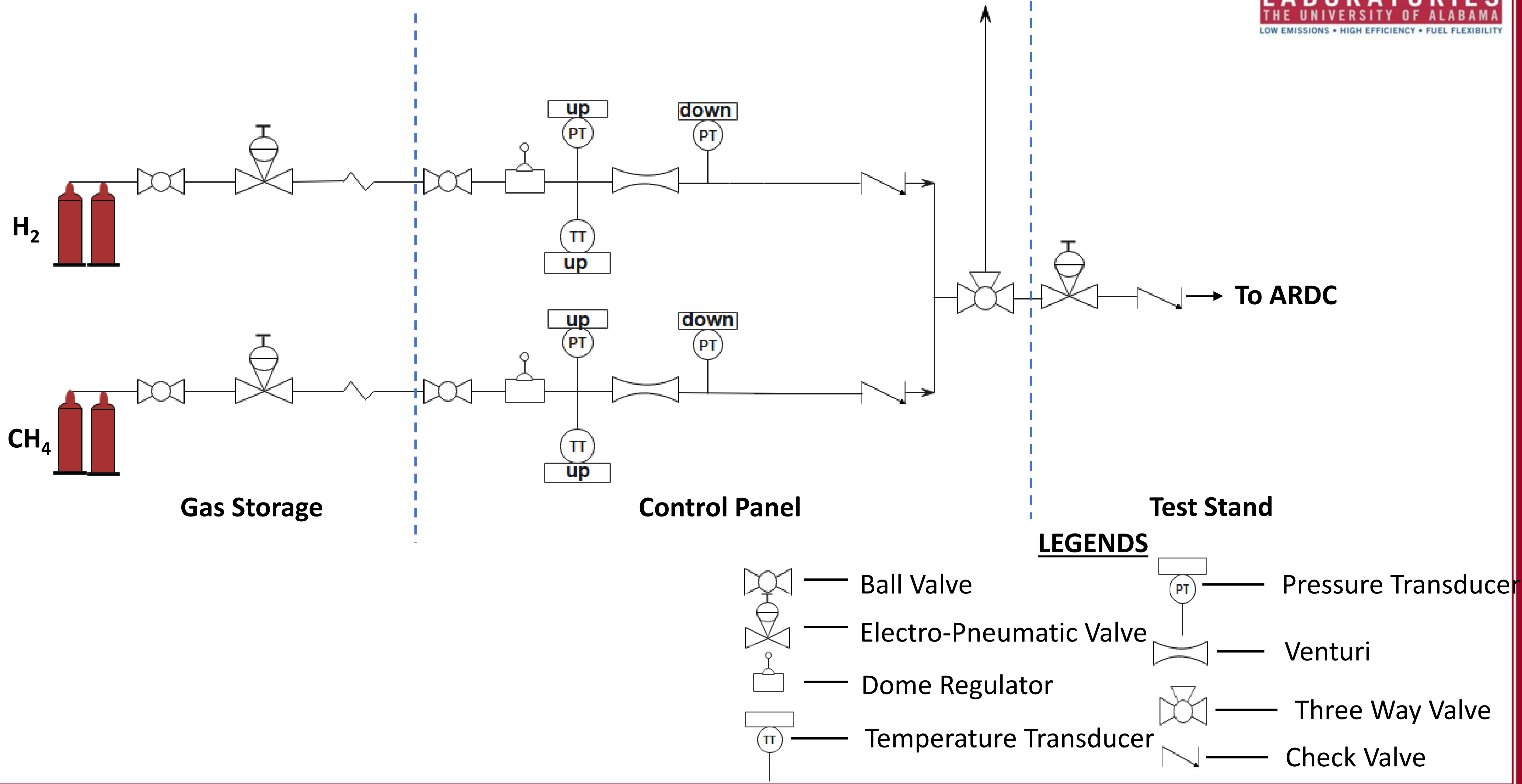
Zoomed View
(near the inlet)

Fuel Supply Systems

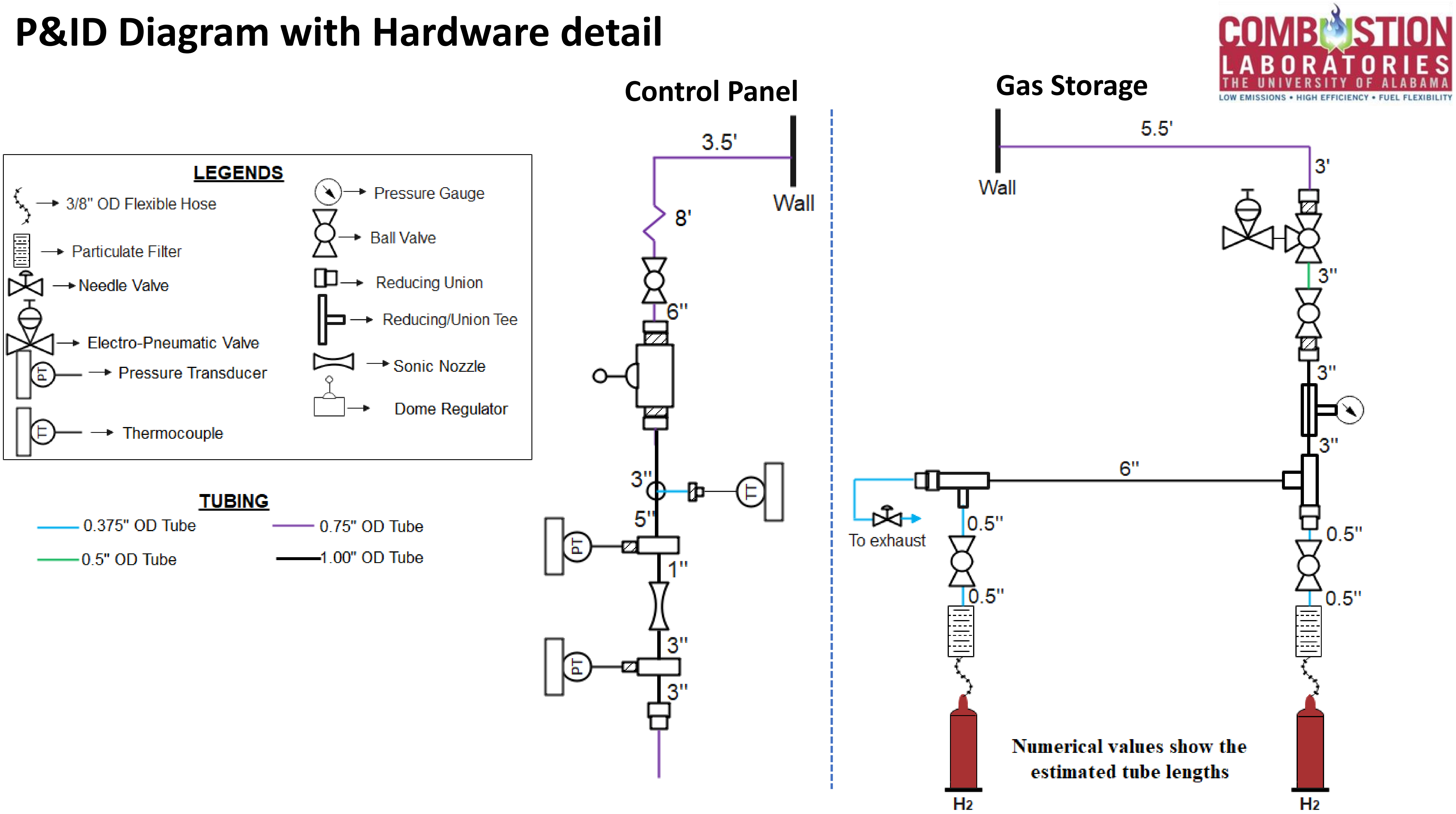
Cylinder Arrangement for Fuel Supply as a Mixture of CH_4 & H_2



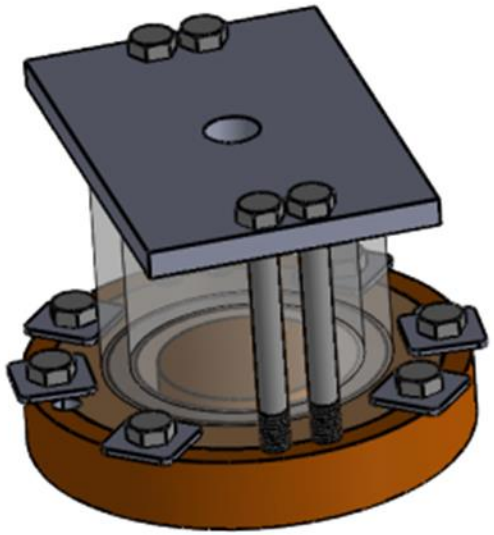
P&ID Diagram for Fuel (H_2+CH_4) Line



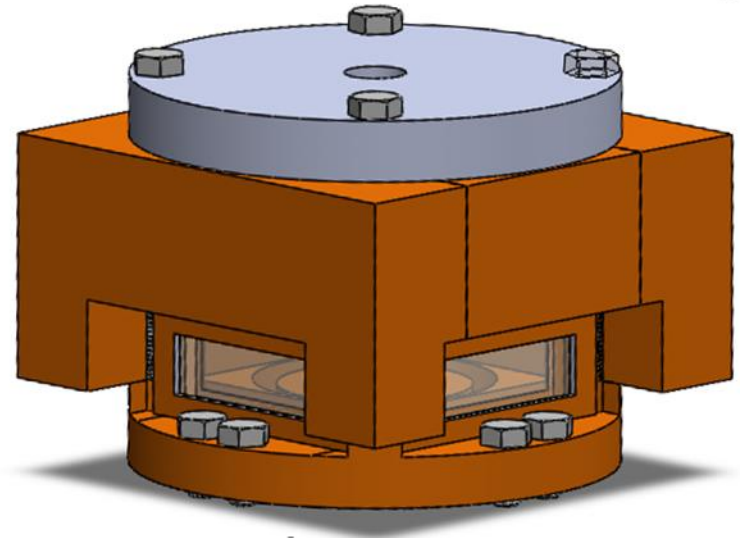
P&ID Diagram with Hardware detail



Back-Pressure Plenum – preliminary design ideas



Circular Optical Plenum



Square metal Plenum with rectangular optical windows

High-Speed PIV

- We have measured the unsteady flow field at the exit of an RDC with an aerospike.
- PIV measurements at 100 kHz were acquired at multiple views, but results are shown only at the RDC exit to characterize the flow unsteadiness.
- Similar Data will be acquired in this study but with the downstream plenum.

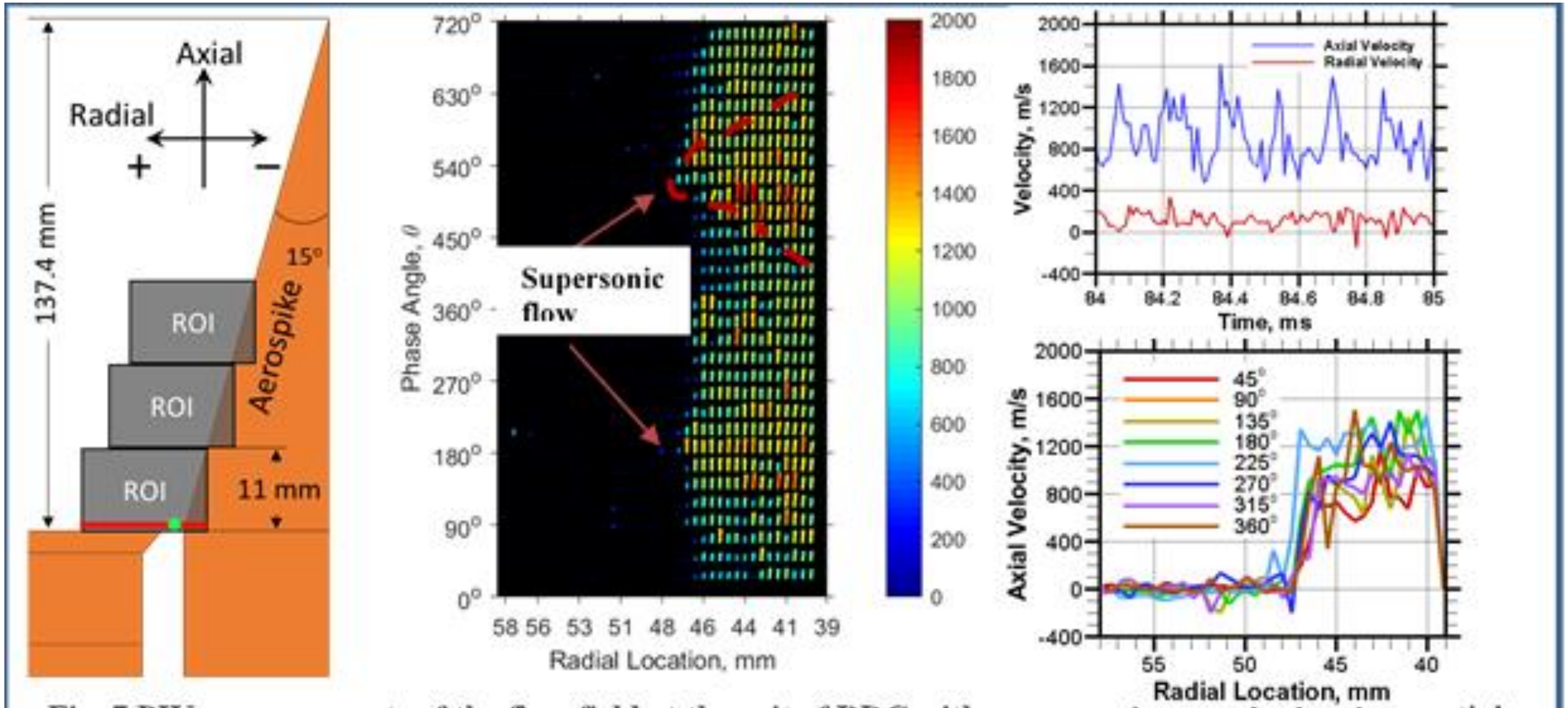
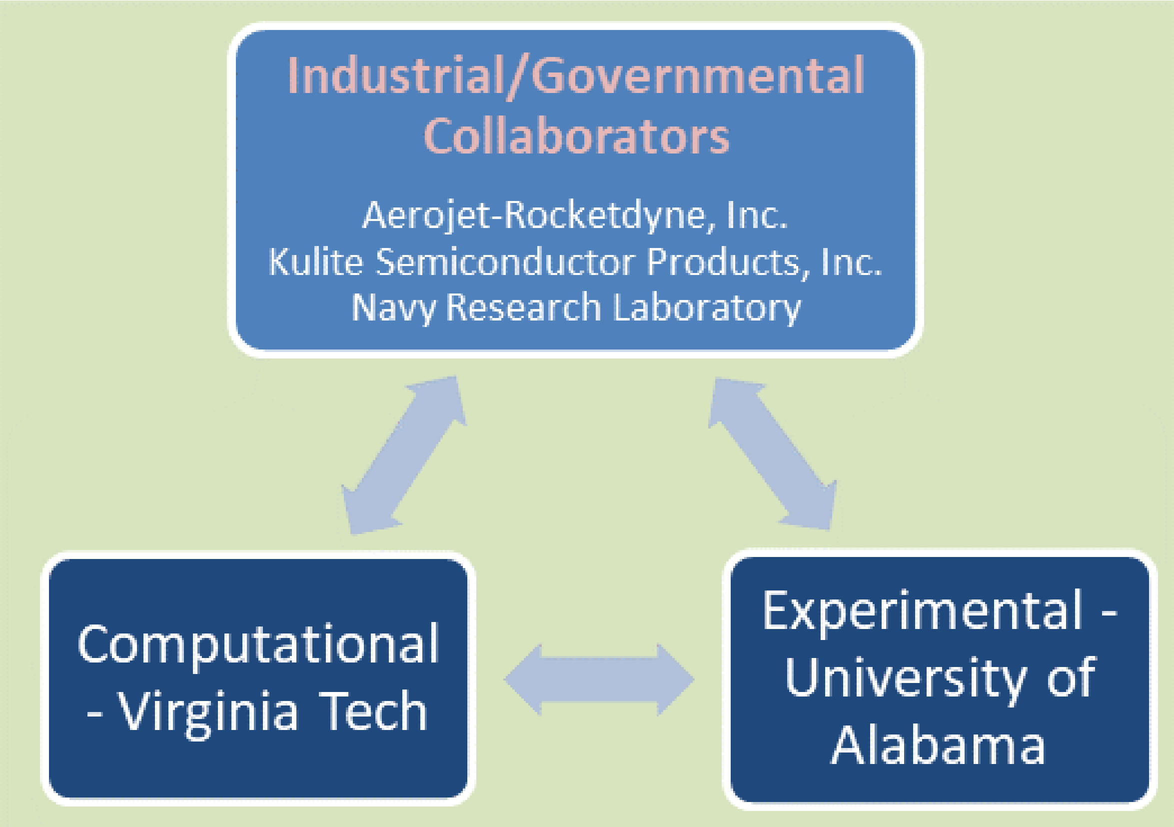


Fig. 7 PIV measurements of the flow field at the exit of RDC with a converging nozzle showing spatial non-uniformity and temporal unsteadiness near the throat. [UA, publication in preparation]

Project Structure

- The project is organized into experimental tasks to be performed at the University of Alabama (UA) and computational tasks conducted at Virginia Polytechnic Institute and State University (VT).
- Professor Ajay K. Agrawal, Robert F. Barfield Endowed Chair Professor at UA is the PI and coordinator of this project. He leads the experimental research
- Professor Joseph Meadows, Assistant Professor at VT, is leading the computational modeling tasks/subtasks (Task 2.0, 4.0, and 6.0) as a co-PI.
- Input and feedback from industrial collaborators will be shared with the appropriate team members, and all project related materials are kept on a secure, password protected website (UA Box).
- We will follow DOE guidelines to produce required reports, and other reports outlined in the SOPO.



Project Timeline: Milestone

Task		Year 1				Year 2				Year 2			
		Q1	Q2	Q3	Q4	Q1	Q2	Q3	Q4	Q1	Q2	Q3	Q4
1.0		→	→	→	→	→	→	→	→	→	→	→	→
	2.1	→											
	2.2		→										
2.0	2.3			→	→								
	3.1	→											
	3.2		→	→									
3.0	3.3			→	→	→							
	4.1					→	→						
4.0	4.2						→	→	→				
	5.1					→	→	→					
5.0	5.2							→	→	→			
	6.1									→	→	→	
6.0	6.2											→	→
	7.1									→	→		
7.0	7.2										→	→	
8.0													→

Milestone Chart

Task/ Subtask	Milestone Title & Description	Planned Completion Date	Verification method
2.2	CFD simulation of nominal RDC channel constriction	End of Y1 Q2	Nominal design configurations provided to experimentalists
3.2	RDC experiments with nominal RDC channel restriction	End of Y1 Q3	Experiments for the nominal channel constriction design and quantification of pressure gain and exit flow conditions.
4.1	CFD simulations of nominal RDC-diffuser configuration.	End of Y2 Q2	Nominal design configurations provided to experimentalists
4.2	Design space exploration of RDC-diffuser configuration	End of Y2 Q4	Most promising design configurations provided to experimentalists
5.2	Final optimized RDC-ejector-diffuser system	End of Y3 Q1	Experiments performed for the most promising designs from simulations and pressure gain quantified
6.2	CFD post processing and quantification of loss mechanisms	End of Y3 Q4	Quantification of the impact of loss mechanisms on RDC performance
8.0	Final recommendations for integrated RDC-turbine system	End of Y3 Q4	A report with relevant strategies/guidelines prepared