REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YYYY)* 02-09-2011	**2. REPORT TYPE** Briefing Slides	**3. DATES COVERED** *(From - To)*
4. TITLE AND SUBTITLE **Analysis of Coaxial Injectors Using CFD++**		**5a. CONTRACT NUMBER** FA9300-10-C-4002 **5b. GRANT NUMBER** **5c. PROGRAM ELEMENT NUMBER**
6. AUTHOR(S) Henry Vu		**5d. PROJECT NUMBER** **5f. WORK UNIT NUMBER** 484710HZ
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Advatech Pacific 560 E Hospitality Lane, Ste 400 San Bernardino CA 92408-3545		**8. PERFORMING ORGANIZATION REPORT NUMBER** AFRL-RZ-ED-VG-2011-370
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) Air Force Research Laboratory (AFMC) AFRL/RZS 5 Pollux Drive Edwards AFB CA 93524-7048		**10. SPONSOR/MONITOR'S ACRONYM(S)** **11. SPONSOR/MONITOR'S NUMBER(S)** AFRL-RZ-ED-VG-2011-370

12. DISTRIBUTION / AVAILABILITY STATEMENT

Approved for public release; distribution unlimited (PA #11666).

13. SUPPLEMENTARY NOTES

For presentation at the Metacomp Users Group Meeting in Long Beach, CA, 13-15 Sep 2011.

14. ABSTRACT

This is a presentation to the Metacomp Users Group meeting in Long Beach, CA, about analyzing coaxial injectors using CFD++.

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			**17. LIMITATION OF ABSTRACT**	**18. NUMBER OF PAGES**	**19a. NAME OF RESPONSIBLE PERSON** Tiffany Rexius
a. REPORT **Unclassified**	**b. ABSTRACT** **Unclassified**	**c. THIS PAGE** **Unclassified**	SAR	37	**19b. TELEPHONE NUMBER** *(include area code)* N/A

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std. 239.18

Analysis of Coaxial Injectors Using CFD++

Presenter:

Henry Vu, Ph.D.

This material is based upon work supported by AFRL/RZSA, AFRL/RZSE, and AFRL/RZST under Contract Nos. FA9300-06-D-0002 and FA9300-10-C-4002

Overview

- Company Background
- Current CFD projects
- Coaxial Jet Flow with Variable Density
- Coaxial Particle Laden Flow

Advatech Pacific, Inc. Background

- An Aerospace Engineering Research & Development Company Founded in 1995 primarily focused on:
  - Aerospace Vehicle Physics-based Modeling, Simulation and Analysis
  - Electronic Communications System Interoperability
  - Aerospace Engineering Design and Analysis Services

Contract Objectives

- Objectives
  - Modeling and analysis of fluid flows and heat transfer in support of experiments performed at AFRL Edwards
    - Complement experimental data by providing detailed visualization of fluid flow and thermal distributions inside experiments
    - Supplement experimental data by generating data at test conditions not performed in experiments
  - Provide independent verification and validation for CFD++ code development

Contract Objectives

- CFD Projects
  - Coaxial Particle laden flow dynamics to assist in design of an experimental apparatus
  - Coaxial gas/gas injector flow analysis for rocket fuel injector design
  - Mixing of jet in cross-flow for pre-burner studies
  - Evaluation of pipe flow conditioning devices
  - Conjugate heat transfer studies in pipe flow for experimental design.

COAXIAL JET FLOW WITH VARIABLE DENSITY

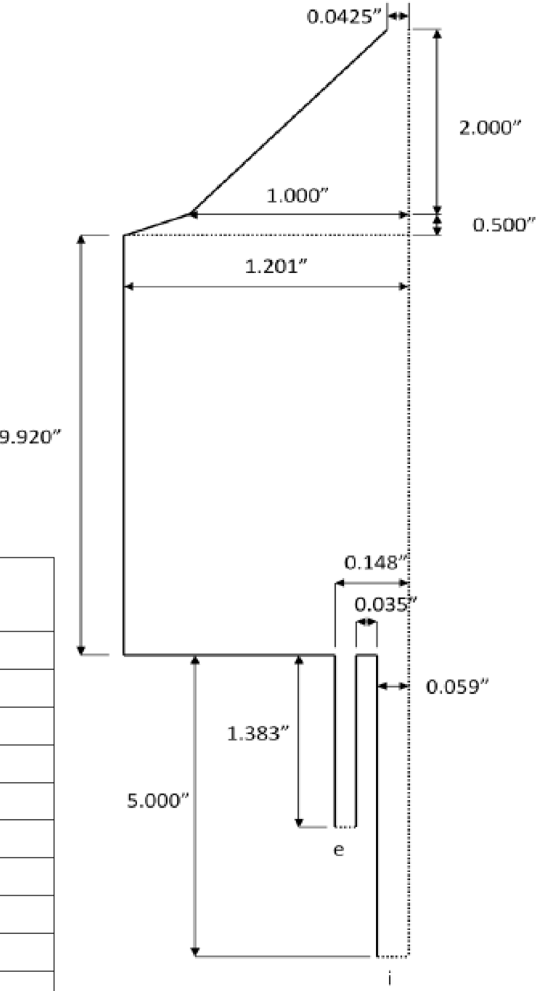
Collaborators: S. Alexander Schumaker, Ananda Himansu, Stephen Danczyk, Malissa Lightfoot

Motivation

- Liquid propellant rocket engine injectors characterized by low velocity, high density inner jet and high velocity, low density annular jet.
- Conservation equations linked through density variations.
- Need to determine accuracy of RANS predictions in flows with large density differences.
- Validate Metacomp CFD++ tool for relevant flows.

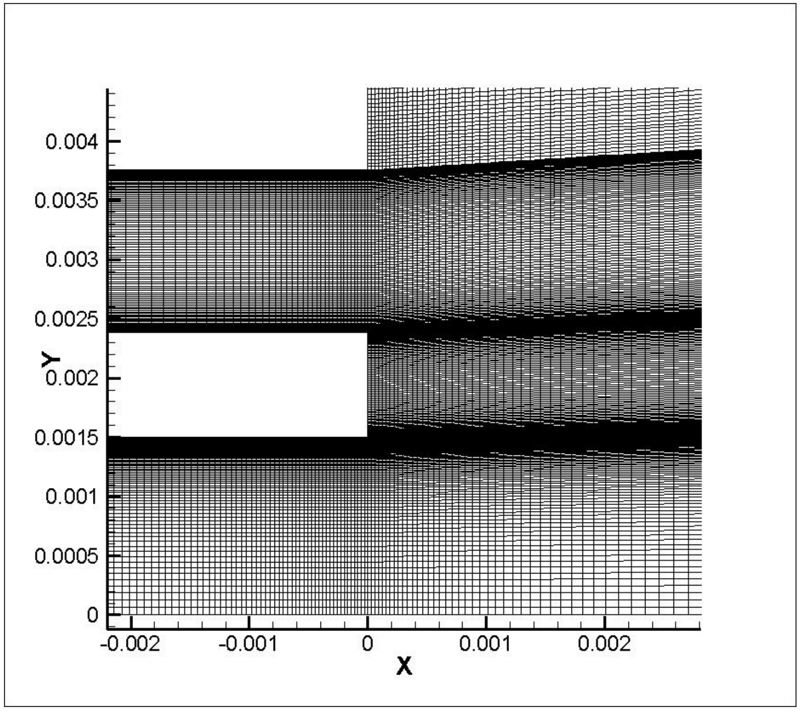
Michigan SEI Case

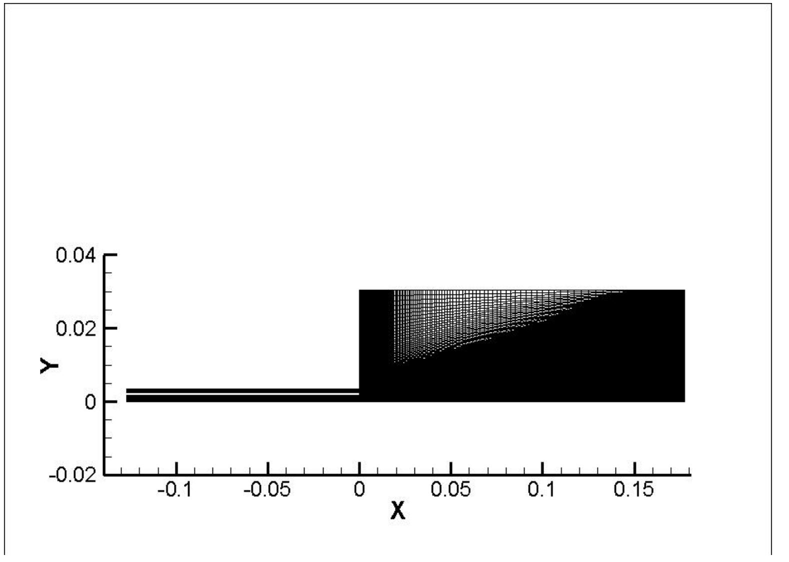
- PLIF concentration data for single element injector
- Low speed flow for ease of modeling
- Confined flow with nozzle at the exit
- Inner fluid (i) is air seeded with acetone
- Outer fluid (e) is helium
- $\rho_i/\rho_e=7.5$



Inner Fluid (i)	Air with 3.35% Acetone by Volume	
Molecular Weight (i)	29.95	Kg/kmol
Molecular Weight (e)	4.003	Kg/kmol
Chamber Pressure	543829.7	Pa
To (e)	293	K
To (i)	293	k
ρo (e)	0.894	Kg/m3
ρo (i)	6.685	Kg/m3
Uavg (e) in tube	63.559	m/s
Uavg (i) in tube	12.587	m/s
Mdot (e)	1.50136E-3	Kg/s
Mdot (i)	5.93253E-4	Kg/s

Michigan SEI Case





Case Conditions:

•Two species (He, Air/Acetone)

•Base Equation Type: Compressible Real Gas Navier-Stokes/Euler

•Equation of State: Ideal Gas

•Turbulence Simulation: RANS, realizable k-eps or SST

•Turbulence Intensity: 2% or 50%

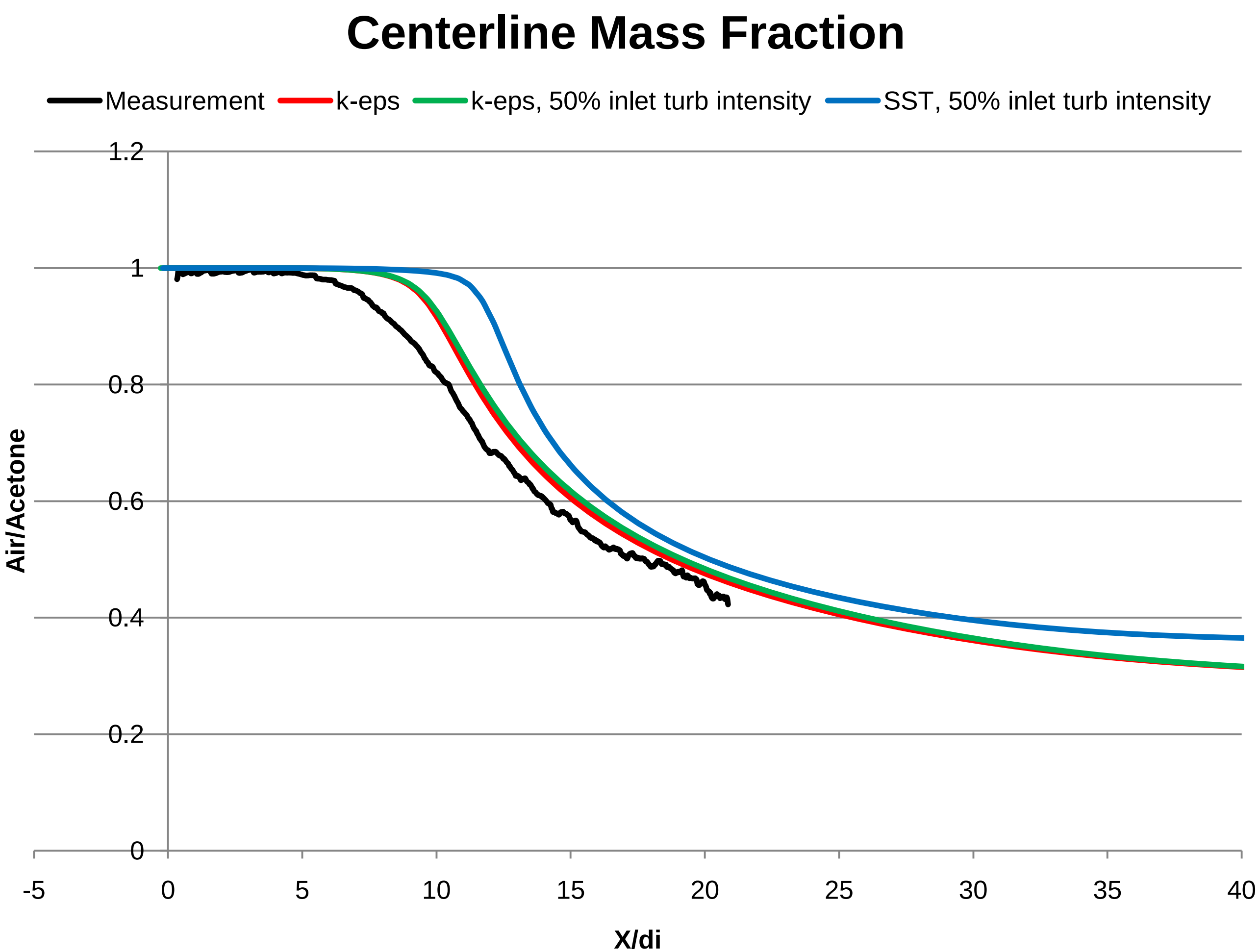
•nozzle geometry at the exit remove and replaced with back pressure imposition

Mesh:

2D axisymmetric

93386 quadrilateral cells, y+=1

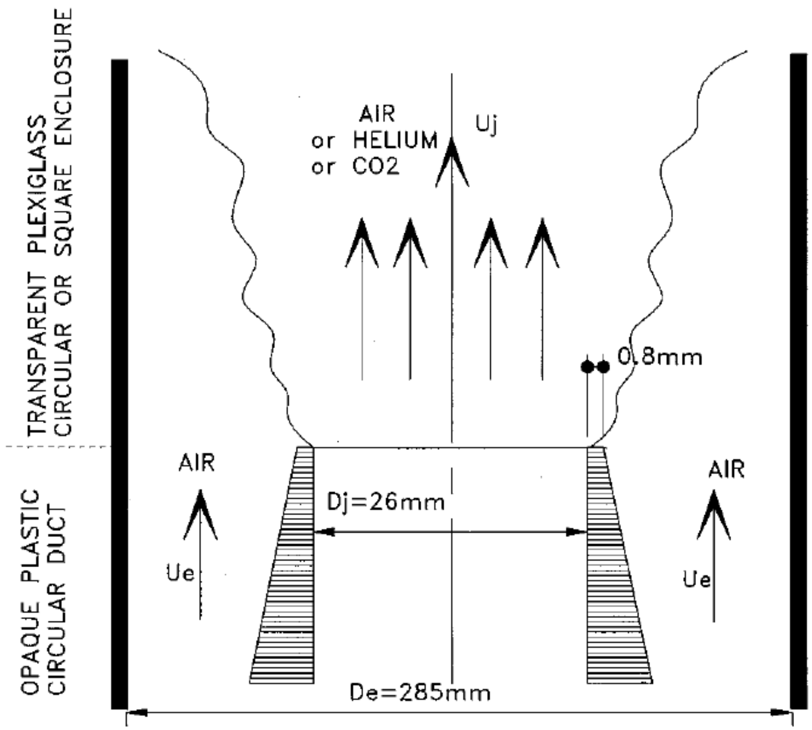
Michigan SEI Case



- At least 15% error in concentration along the centerline in the near-field
- No improvement by increase turbulence at inlet
- SST turbulence model makes result worse

Is error due to problem in modeling scalar or velocity field?

Validation Case



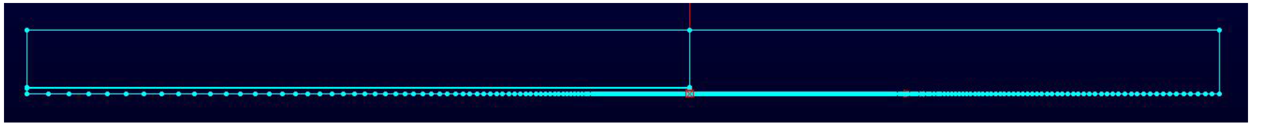
- 2D axisymmetric compressible RANS
- Density-based solver
- Realizable k-eps turbulence solved to the wall
- T=300 K and P=101325 Pa
- Center inlet boundary:
    - He normal velocity = 24.45 m/s
    - Air normal velocity = 10.5 m/s
    - CO_2 normal velocity = 9.0 m/s
- Outer inlet boundary: Air with normal velocity of 0.9 m/s
- Outlet: Simple back pressure outlet of 0 Pa.

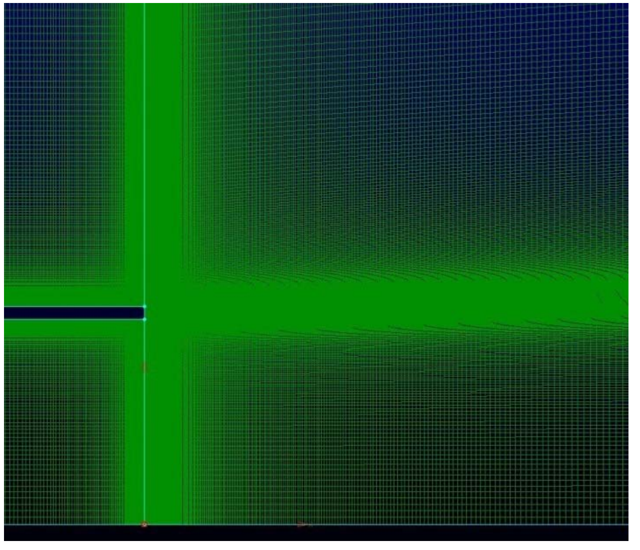
Gas	U_j [m/s] (Expt)	U_j [m/s] (CFD)	Re_j	ρ_j/ρ_e
He	32	31.4	7000	0.14
Air	12	12.68	21000	1
CO_2	10	10.75	32000	1.4

Amielh, M., Djeridane, T., Anselmet, F., & Fulachier, L. (1996). Velocity near-field of variable density turbulent jets. *Int. J. Heat Mass Transfer* , 2149-2164.

Djeridane, T., Amielh, M., Anselmet, F., & Fulachier, L. (1996). Velocity turbulence properties in the near-field region of axisymmetric variable density jets. *Phys. Fluids* , 1614-1630.

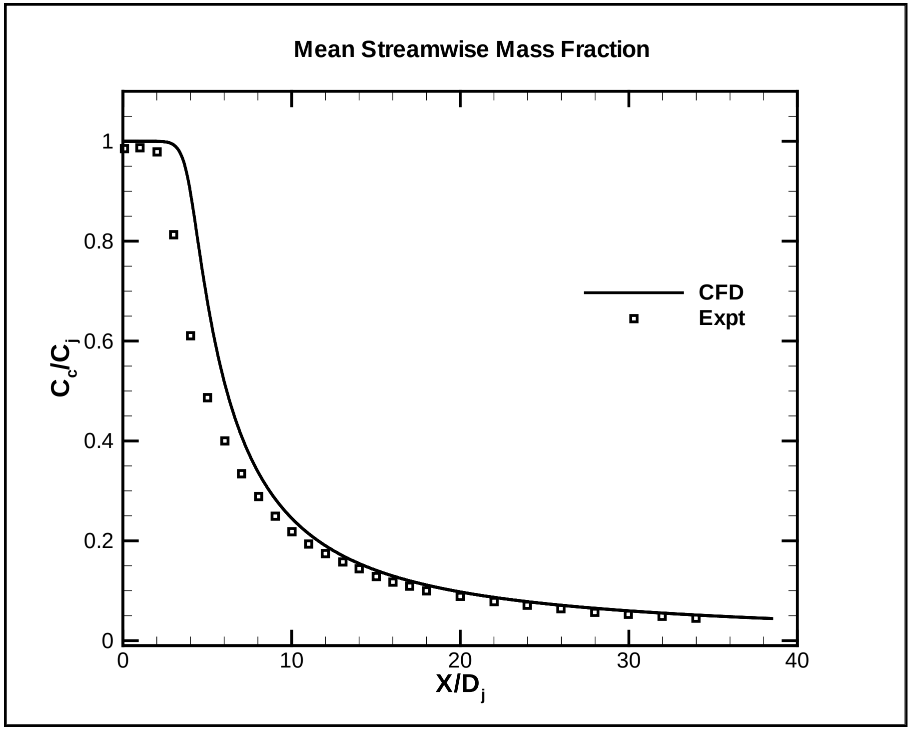
Validation Case



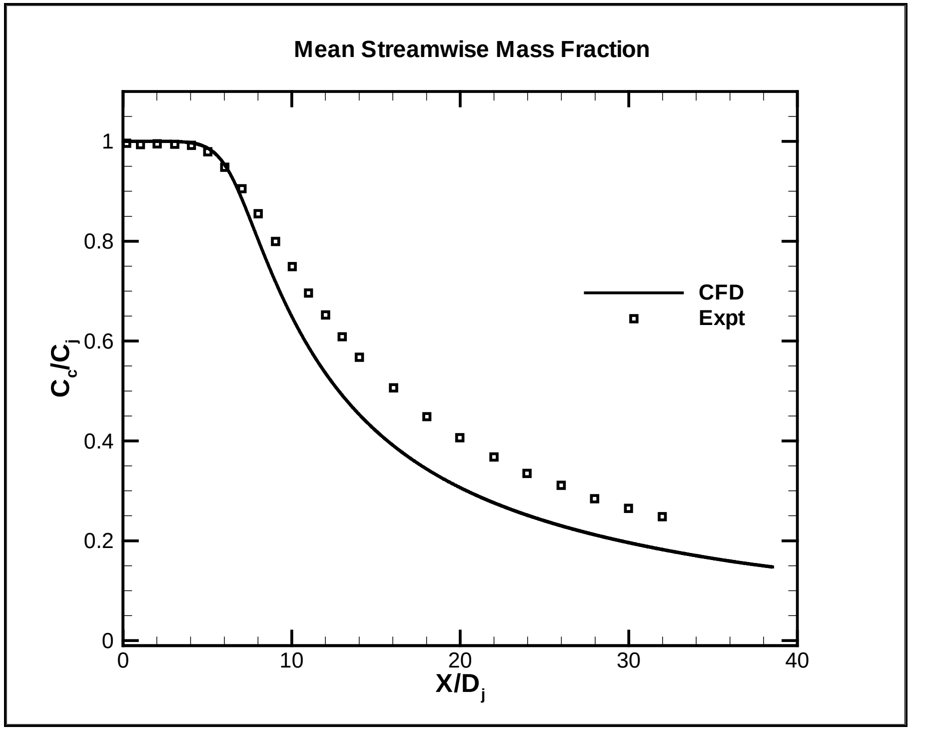


- 234880 quadrilateral cells with a y+ of 1 or less at all wall locations except the outermost wall.
- Run up length = 1.5 m
- Total domain length = 2.7 m

Validation Case Results

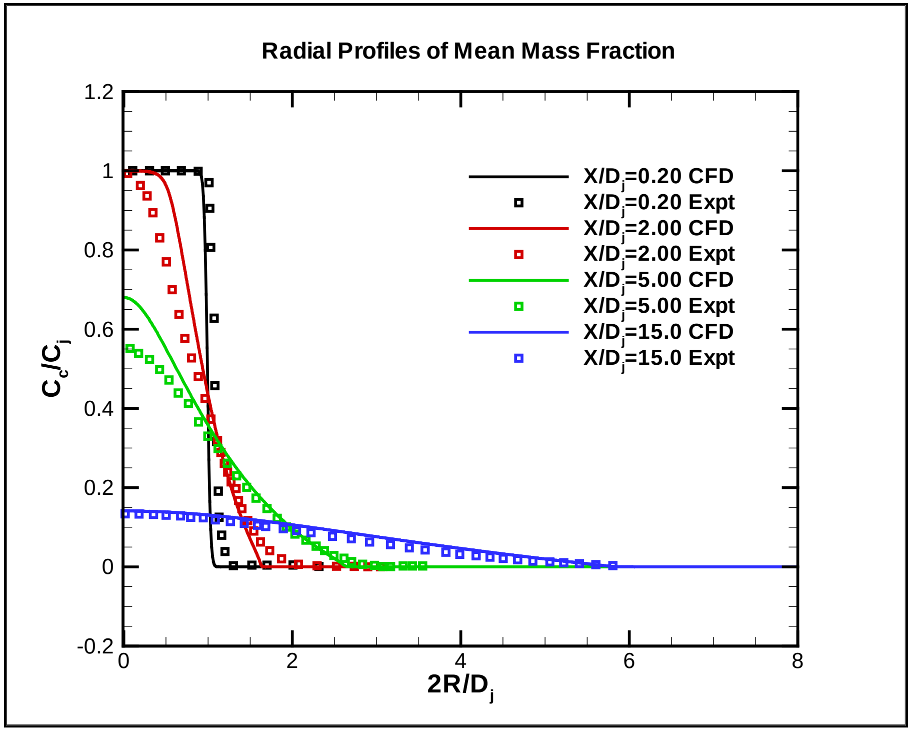


He

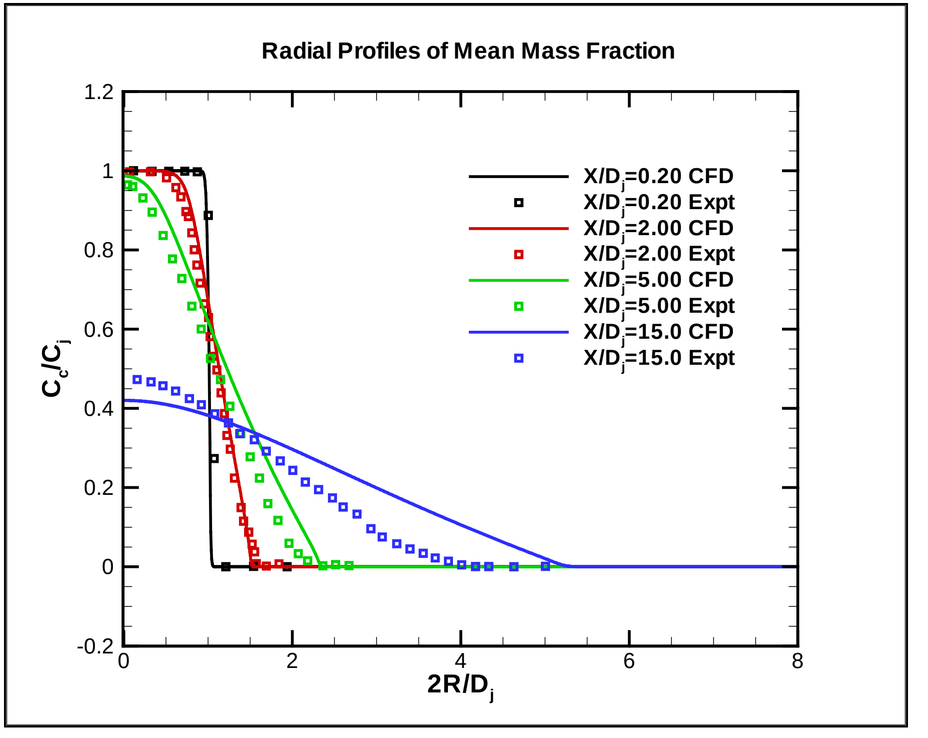


CO_2

Validation Case Results

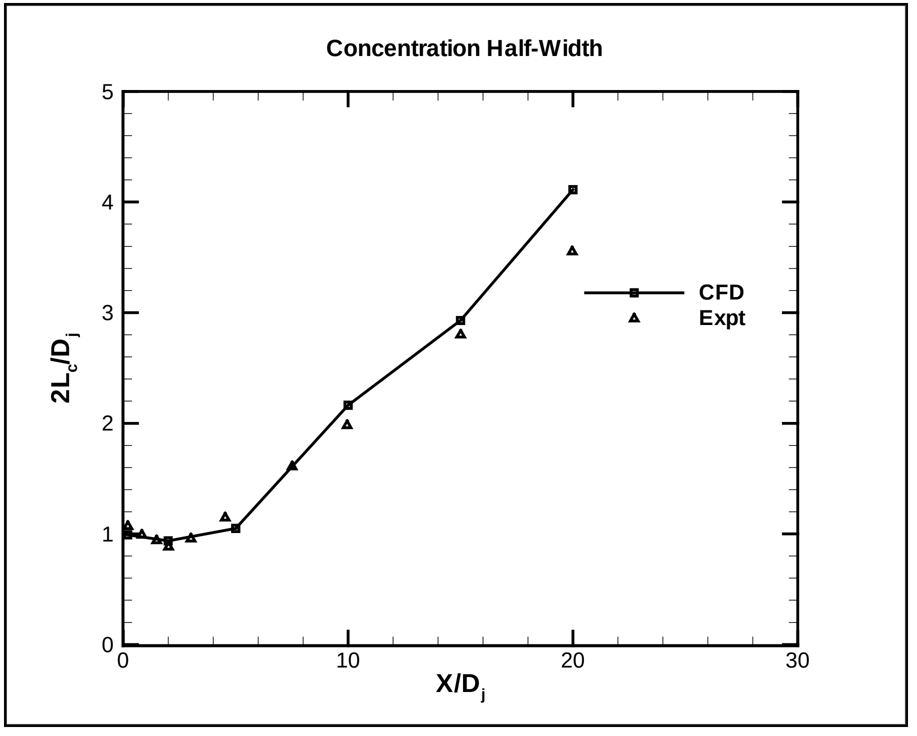


He

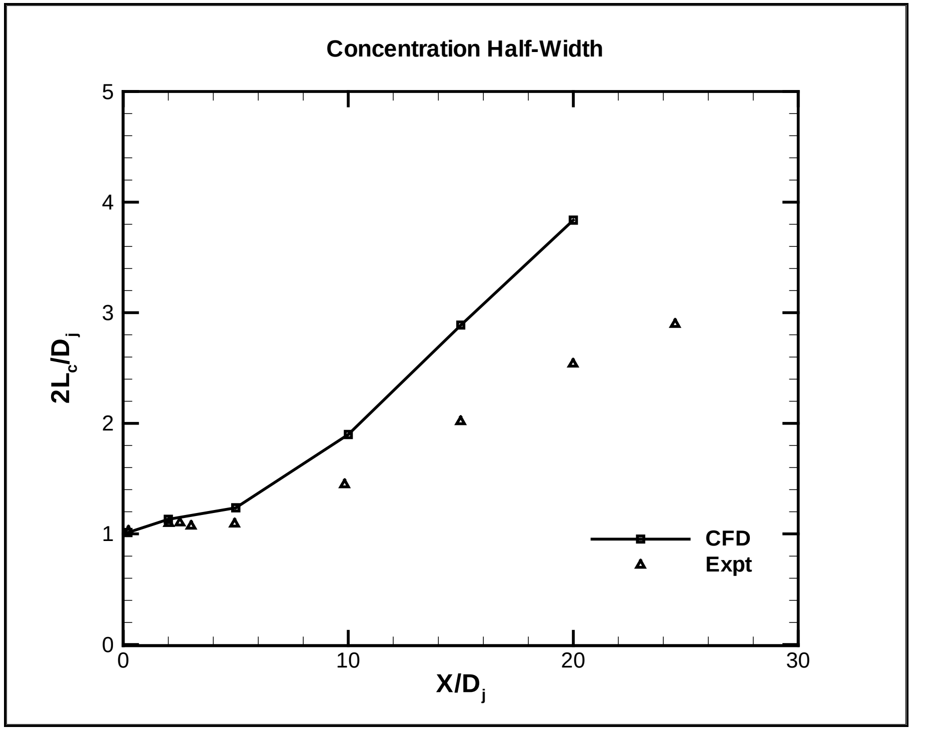


CO_2

Validation Case Results



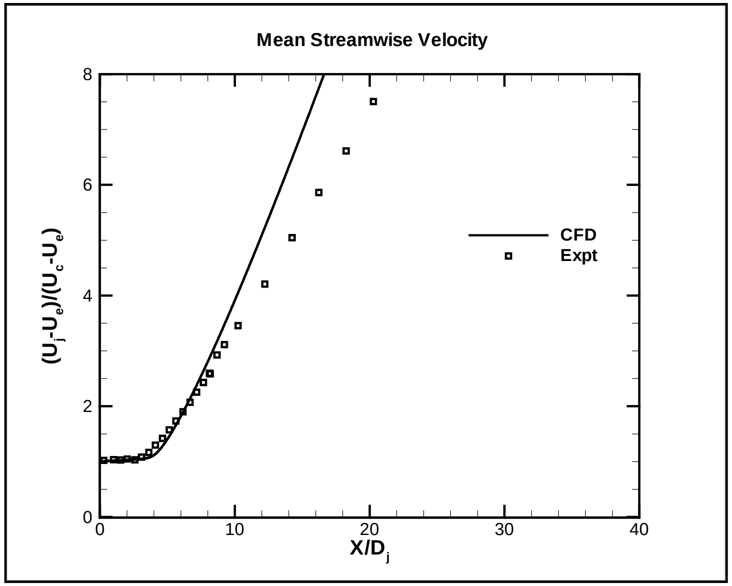
He



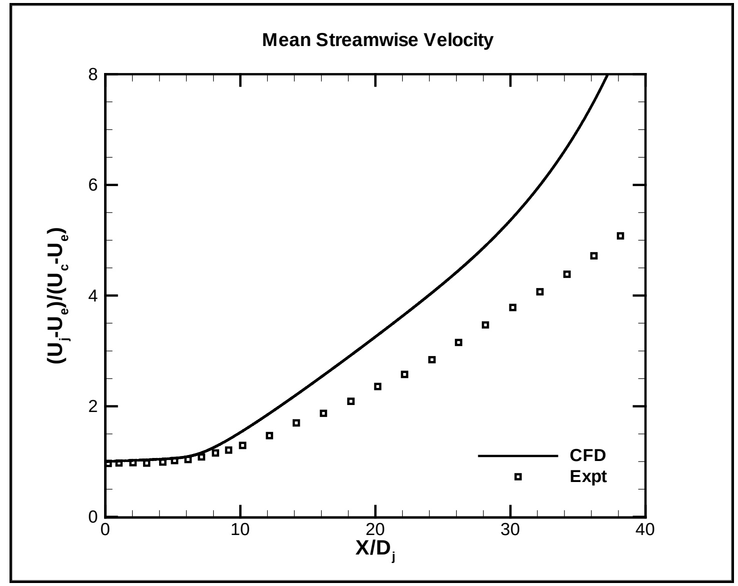
CO_2

Distribution A: Approved for Public Release (Pending)

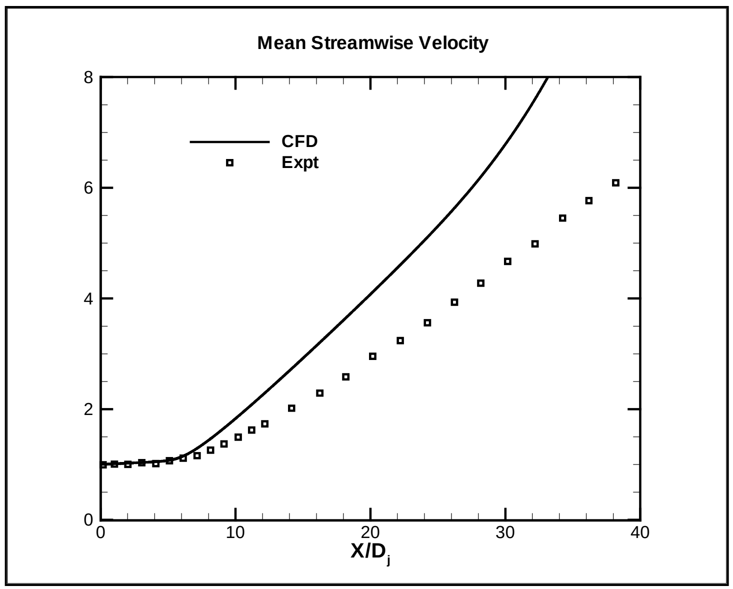
Validation Case Results



He



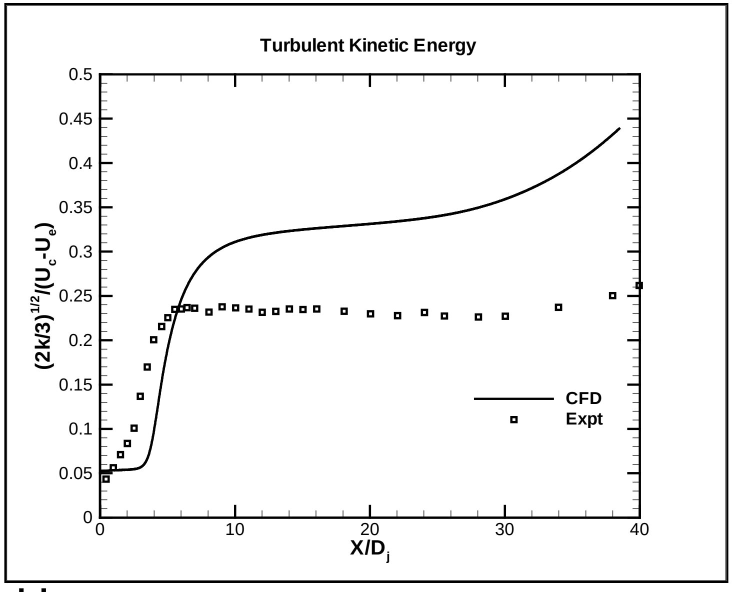
CO_2



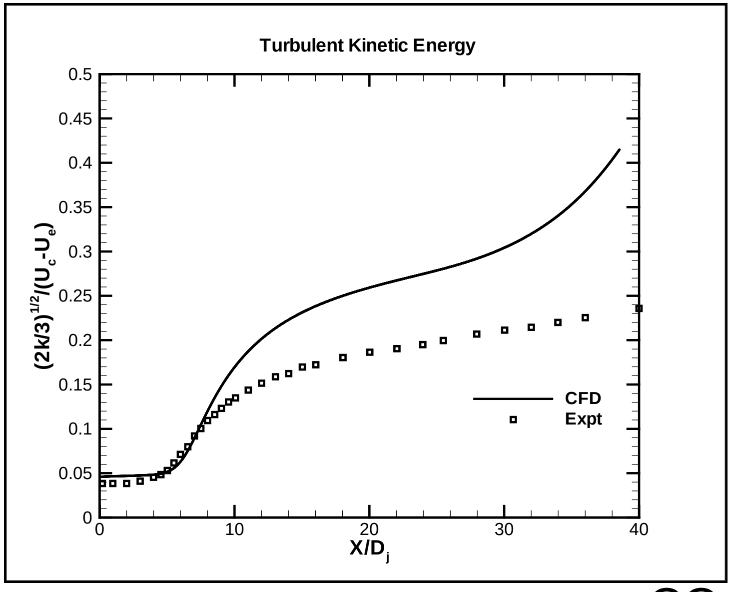
Air

Distribution A: Approved for Public Release (Pending)

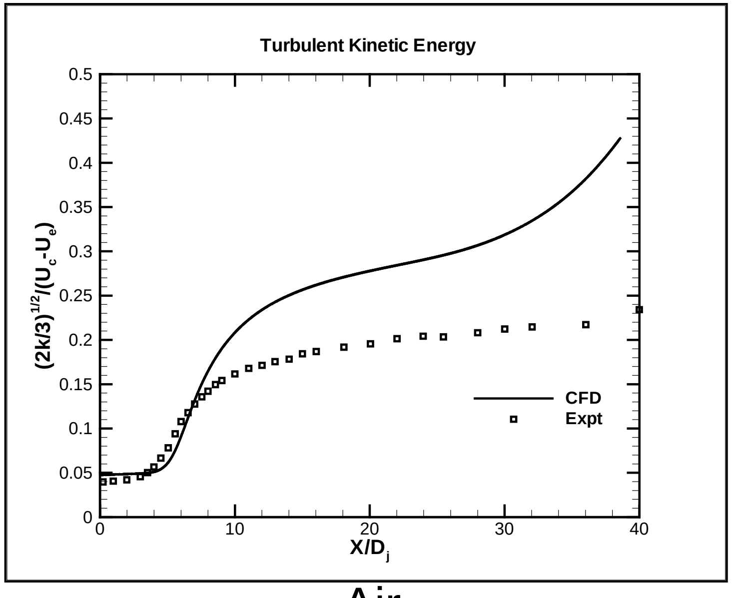
Validation Case Results



He



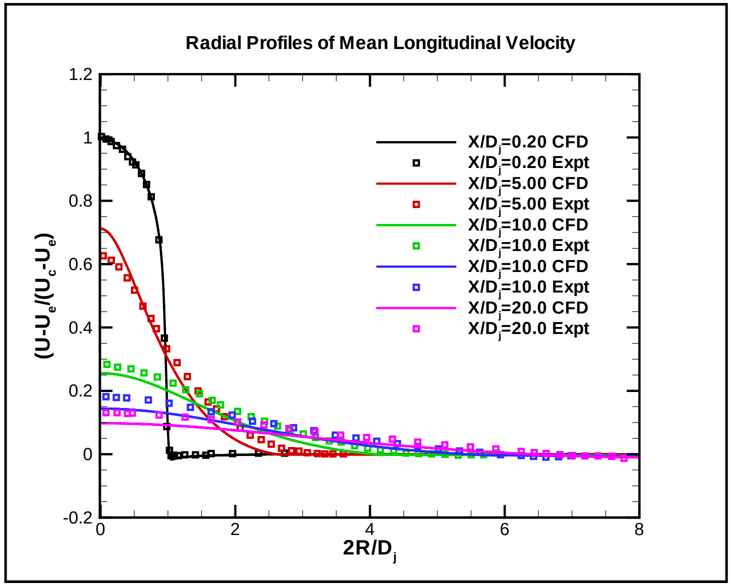
CO_2



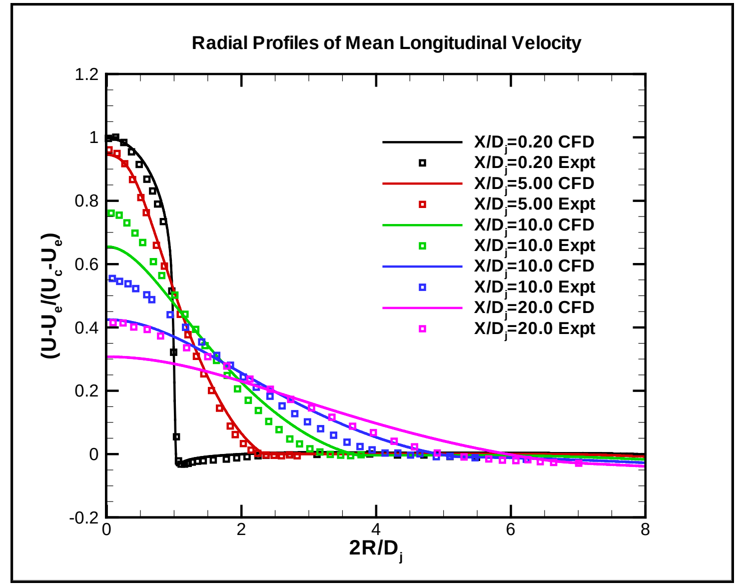
Air

Distribution A: Approved for Public Release (Pending)

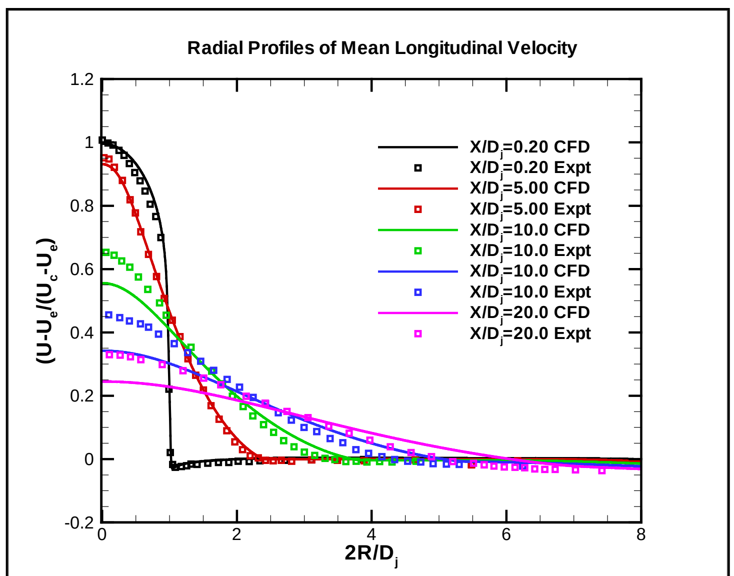
Validation Case Results



He

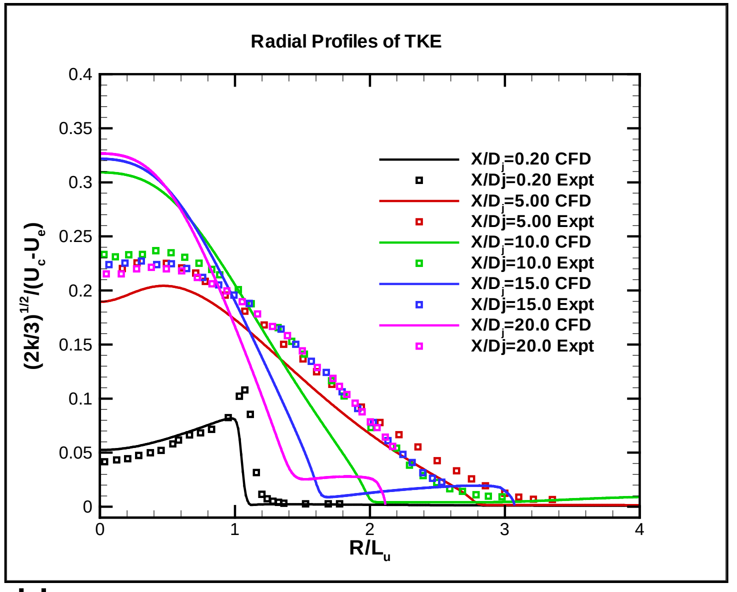


CO_2

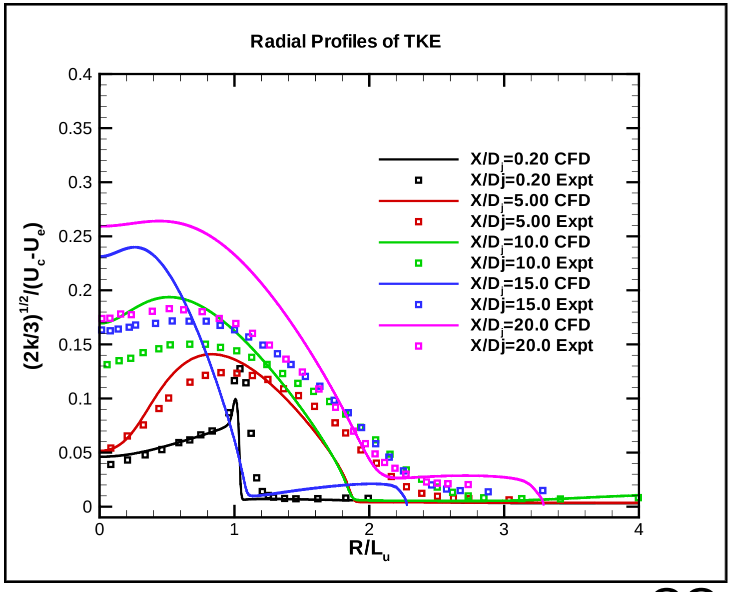


Air

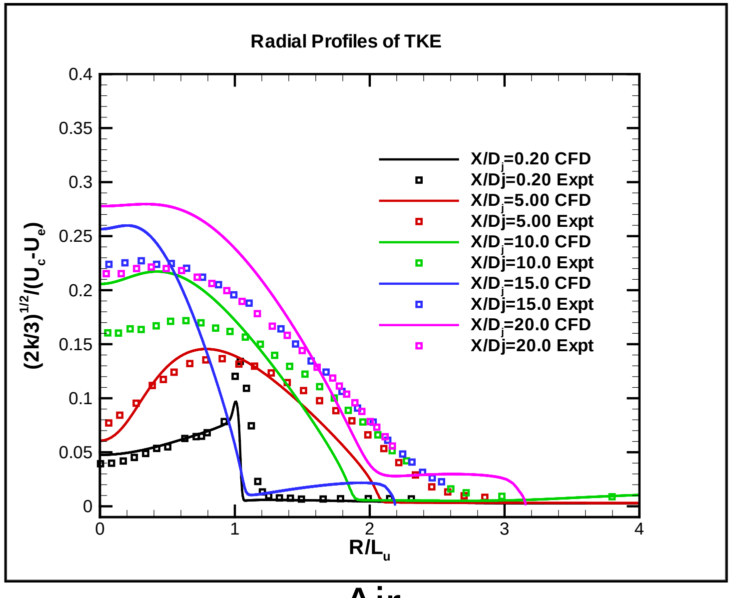
Validation Case Results



He

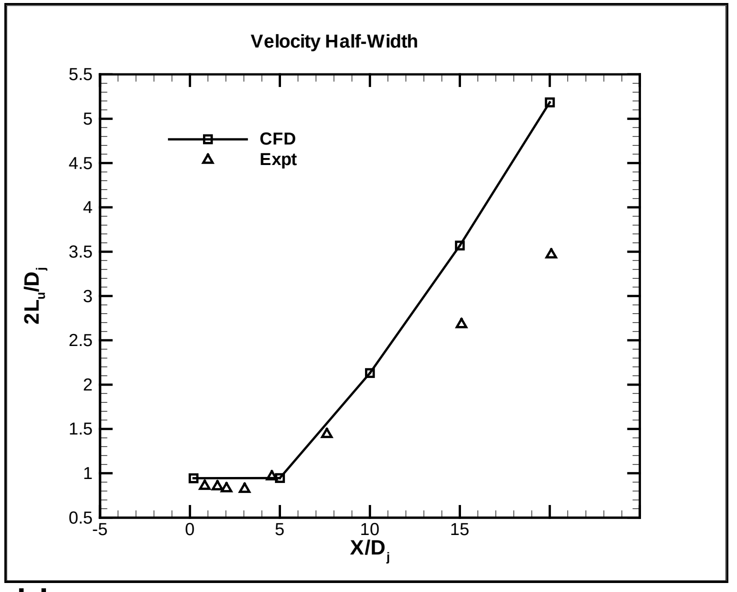


CO_2

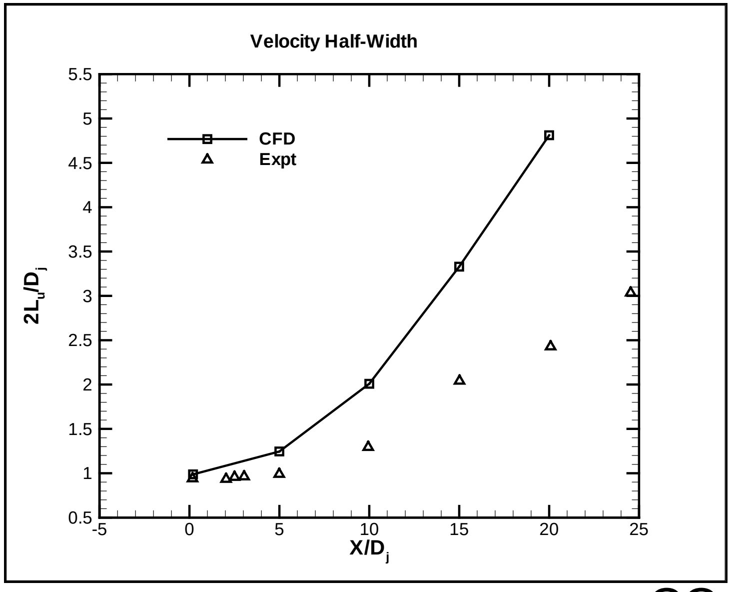


Air

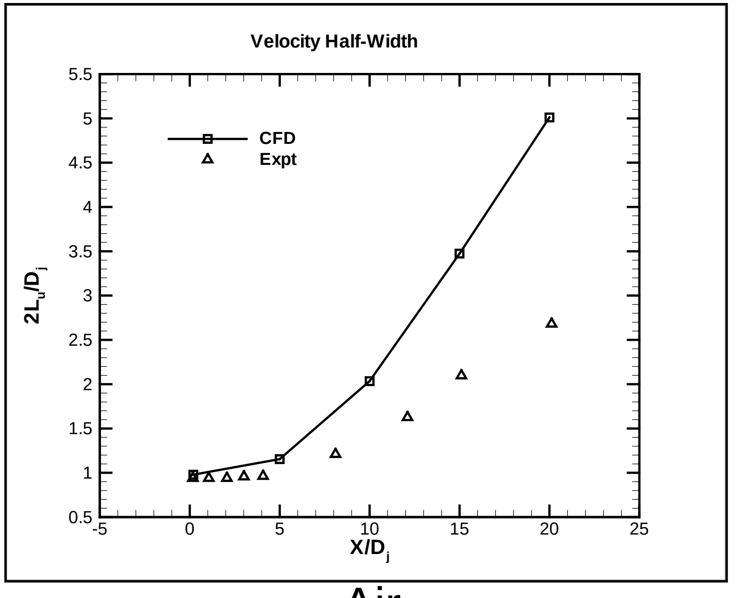
Validation Case Results



He



CO_2



Air

Distribution A: Approved for Public Release (Pending)

Validation Case Summary

- Case exhibits same deficiency in centerline mixing for He/Air in the near field while improving in the far-field.
- Radial profiles of mean quantities generally good, except at centerline.
- CO2/Air case shows accurate mixing in the near-field and worsens downstream.
- Both CFD and experiment show low sensitivity of density ratio on jet spreading. Mixing occurring by entrainment of surrounding fluid.
- Turbulence is generally over predicted everywhere except in the near field for He/Air.
- Deficiencies in centerline mixing may be correlated to turbulence.

Future Work

- Try difference turbulence models.
- Modify k-eps to compensate for low near-field turbulence.
- Try Large Eddy Simulation (LES).
- Proceed to more relevant flow conditions for rocket injectors.

COAXIAL PARTICLE LADEN FLOW

Collaborators: Ananda Himansu, Alireza Badakhshan, Stephen Danczyk

Motivation

- Experimental set up in lab to evaluate novel fuel ignition strategies
- USI device with a shroud to induce swirl in coflow
- Use CFD to help design apparatus and flow conditions
- Some earlier CFD work helped in redesign of shroud for better swirling
- A parametric study to test the effects of swirling flow showed some unusual results
- Need to validate dispersed phase modeling features in CFD++ to confirm results and determine important conditions to accurate modeling

Eulerian Dispersed Phase Modeling in CFD++

Dispersed Phase

Mass: $$\frac{\partial(\rho_{pi})}{\partial t} + \nabla \cdot (\rho_{pi}\vec{u}_{pi}) = \dot{m}_i$$

Momentum: $$\frac{\partial(\rho_{pi}\vec{u}_{pi})}{\partial t} + \nabla \cdot (\rho_{pi}\vec{u}_{pi}\vec{u}_{pi}) = \vec{F}_{D_i} + \vec{F}_{V.M._i} + \vec{F}_{T.D._i} + \vec{F}_{L_i} + \vec{F}_{P.G._i} + \vec{F}_{B_i} + \dot{m}_i\vec{u}_{pi}$$

Energy: $$\begin{aligned}\frac{\partial(\rho_{pi}e^0_{pi})}{\partial t} + \nabla \cdot (\rho_{pi}e^0_{pi}\vec{u}_{pi}) &= \text{Conduction source term} + \text{Radiation source term} + \dot{m}_i e^0_{pi} \\ &= \dot{Q}_{pi} + \dot{m}_i e^0_{pi}\end{aligned}$$

Number Density: $$\frac{\partial N_i}{\partial t} + \nabla \cdot (N_i\vec{u}_{pi}) = 0$$

Melting Fraction: $$\frac{\partial(\xi_i N_i)}{\partial t} + \nabla \cdot (\xi_i N_i\vec{u}_{pi}) = \text{Melting source term}$$

Material Density: $$\tilde{\rho}_{pi} = \tilde{\rho}_{pi_{liquid}}\xi_i + \tilde{\rho}_{pi_{solid}}(1-\xi_i),$$

Particle radius: $$r_i = \left(\frac{3\rho_{pi}}{4\pi\tilde{\rho}_{pi}N_i}\right)^{1/3}.$$

Eulerian Dispersed Phase Modeling in CFD++

Continuous Phase

Momentum source terms:

$$\sum_{i=1}^{NP} \left(\vec{F}_{D_i} + \vec{F}_{V.M._i} + \vec{F}_{T.D._i} \right) \;,$$

Energy source terms:

$$\sum_{i=1}^{NP} \left[\frac{\rho_{pi}\; C_{pi}\; f_{Ni}}{\tau_{T_i}} (T_f - T_{pi}) + \vec{u}_f \cdot (\vec{F}_{D_i} + \vec{F}_{V.M._i} + \vec{F}_{T.D._i}) \right] \;.$$

Relevant Source Terms

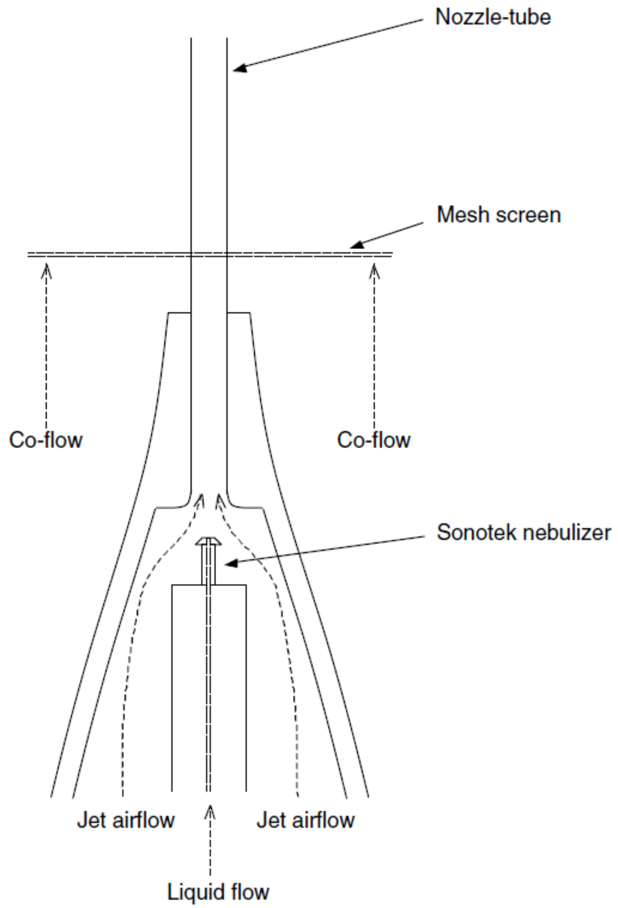
Interphase Drag:

$$\frac{\vec{F}_{D_i}}{\rho_{pi}} = \frac{f_{Di}}{\tau_{ui}} (\vec{u}_f - \vec{u}_{pi}) \;,$$

Turbulent Dispersion:

$$\frac{\vec{F}_{T.D._i}}{\rho_{pi}} = -\frac{f_{Di}}{\tau_{ui}} \frac{\nu_t}{\mathrm{Pr}_t} \left(\frac{\nabla \eta_{pi}}{\eta_{pi}} - \frac{\nabla \eta_f}{\eta_f} \right) \;,$$

Model Validation Case



Poly-dispersed turpentine droplets

d = 1-90 um

U_c = 2.4 m/s

Low turbulence intensity of 1.4%

Nijdam, J., Langrish, T., & Fletcher, D. (2008). Assessment of an Eulerian CFD model for prediction of dilute droplet dispersion in a turbulent jet. *Appl. Math. Model.*, 2686-2705.

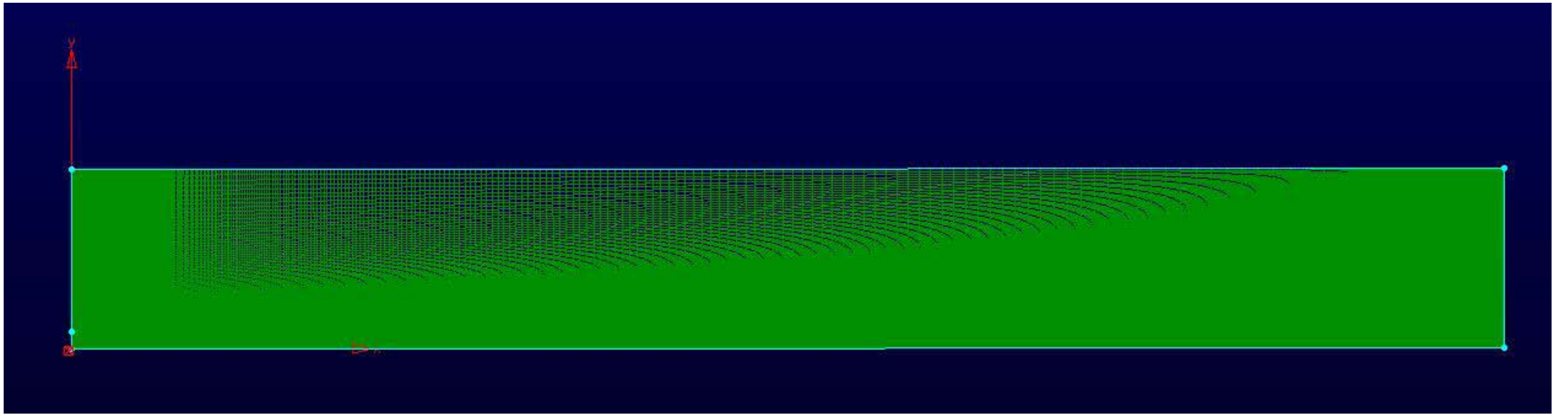
Model Validation Case

Droplet Inlet Boundary Condition (1D downstream)

Variable	Constants
Excess axial mean velocity (m/s) $U_e = U_{eo}\lfloor 1 - [\sin((R/R_{1/2U})^{n_2})]^{n_1} \rfloor$	(1) Peak excess axial mean velocity, U_{eo} $U_{eo} = -0.05148\,d + 21.97494$ where d is droplet diameter (μm) (2) $n_1 = 3.9320$, $n_2 = 1.222$, $R_{1/2U} = 4.978$
Radial mean velocity (m/s) $V = 0$	
Volume fraction $r = r_o \exp\lfloor -A(R/R_{1/2VF})^n \rfloor$	(1) Peak volume fraction r_o 4.32E−08 (5 μm), 6.91E−07 (15 μm), 5.07E−06 (25 μm), 9.50E−06 (35 μm), 1.47E−05 (45 μm), 2.16E−05 (55 μm), 2.04E−05 (65 μm), 7.43E−06 (75 μm), 2.50E−06 (85 μm) (2) $A = 0.6942$, $n = 2.1543$, $R_{1/2VF} = 3.9480$
Gas turbulent kinetic energy (m^2/s^2) $k = D\exp[-A(R - B)^n] + C$	(1) $A = 1.011$, $B = 4.904$, $C = 1.75$, $D = 8.73$, $n = 1.418$
Gas turbulent energy dissipation (m^2/s^3) $\varepsilon = \frac{k^{1.5}}{0.2D}$	(1) $D = 0.0098$ m

Nijdam, J., Langrish, T., & Fletcher, D. (2008). Assessment of an Eulerian CFD model for prediction of dilute droplet dispersion in a turbulent jet. *Appl. Math. Model.*, 2686-2705.

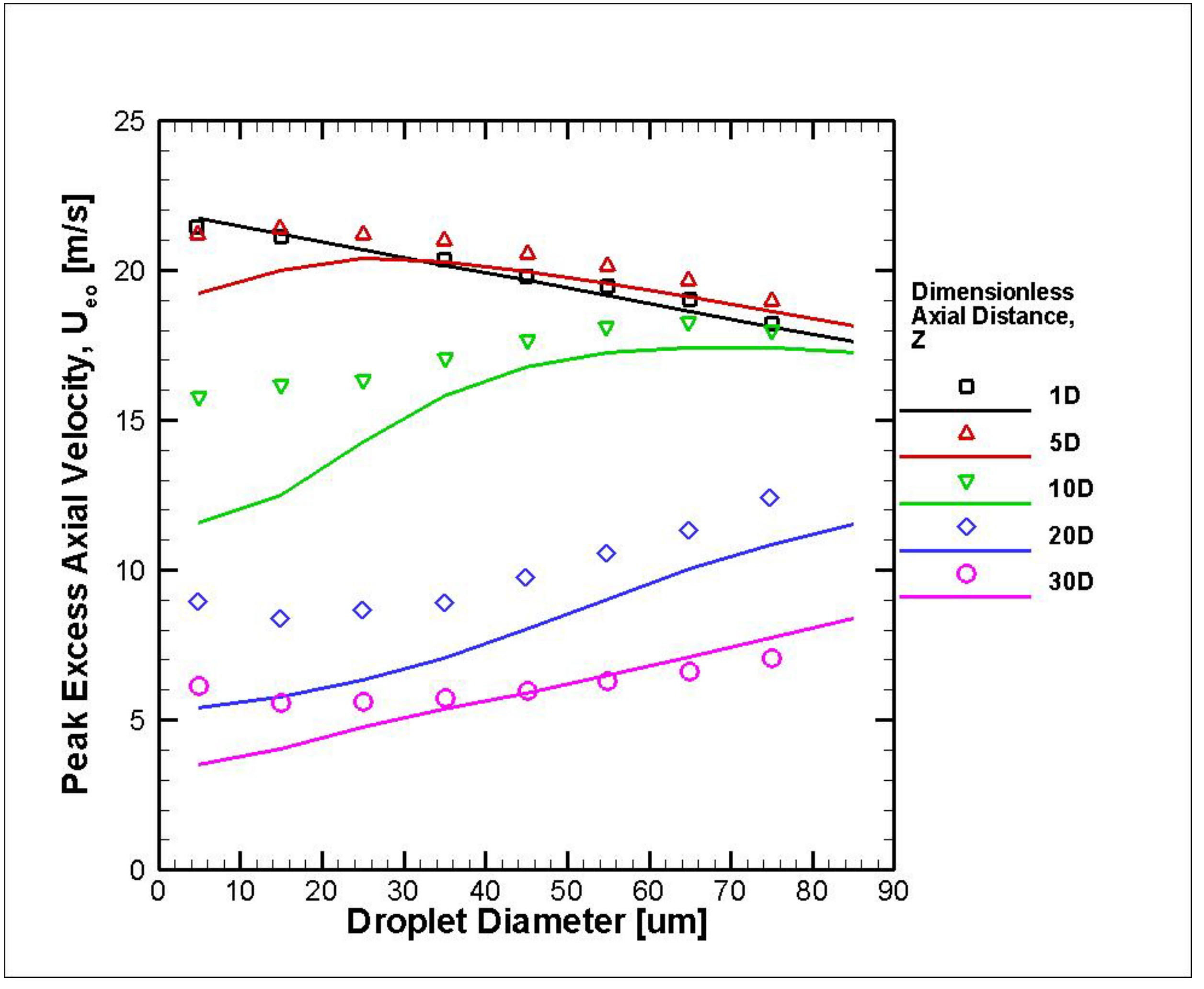
Model Validation Case



Case Conditions:
•Two species (Air, Turpentine)
•Base Equation Type: Compressible Real Gas Navier-Stokes/Euler
•Equation of State: Ideal Gas
•Turbulence Simulation: RANS, realizable k-eps or SST
•Turbulence Intensity: 1.4%
•10 um droplet size bins from 5-85 um diameter
•Temperature-based inlet profiles generated with Matlab script

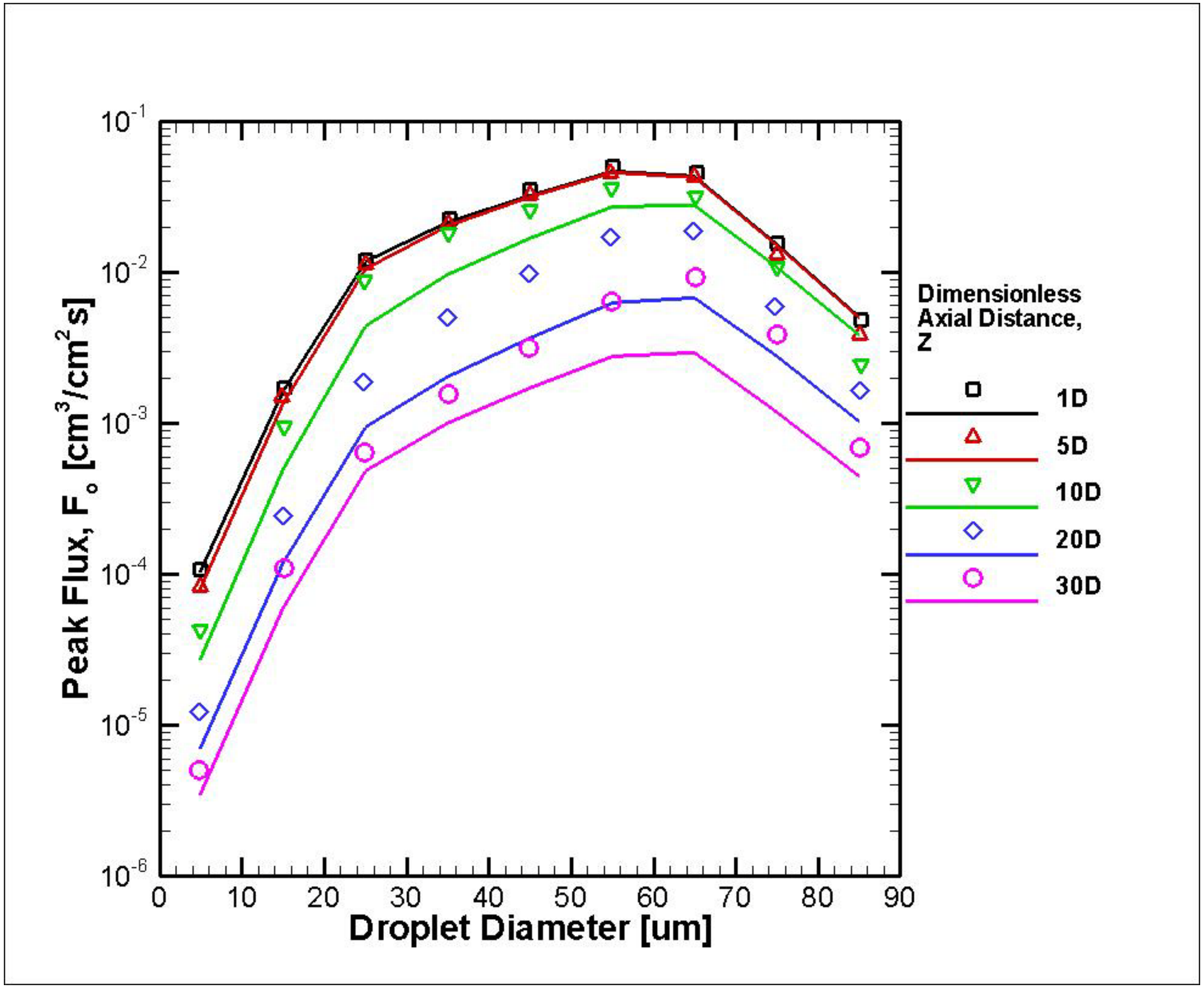
•79000 quadrilateral cells
•Total domain size = 0.05 m x 0.4 m

Results



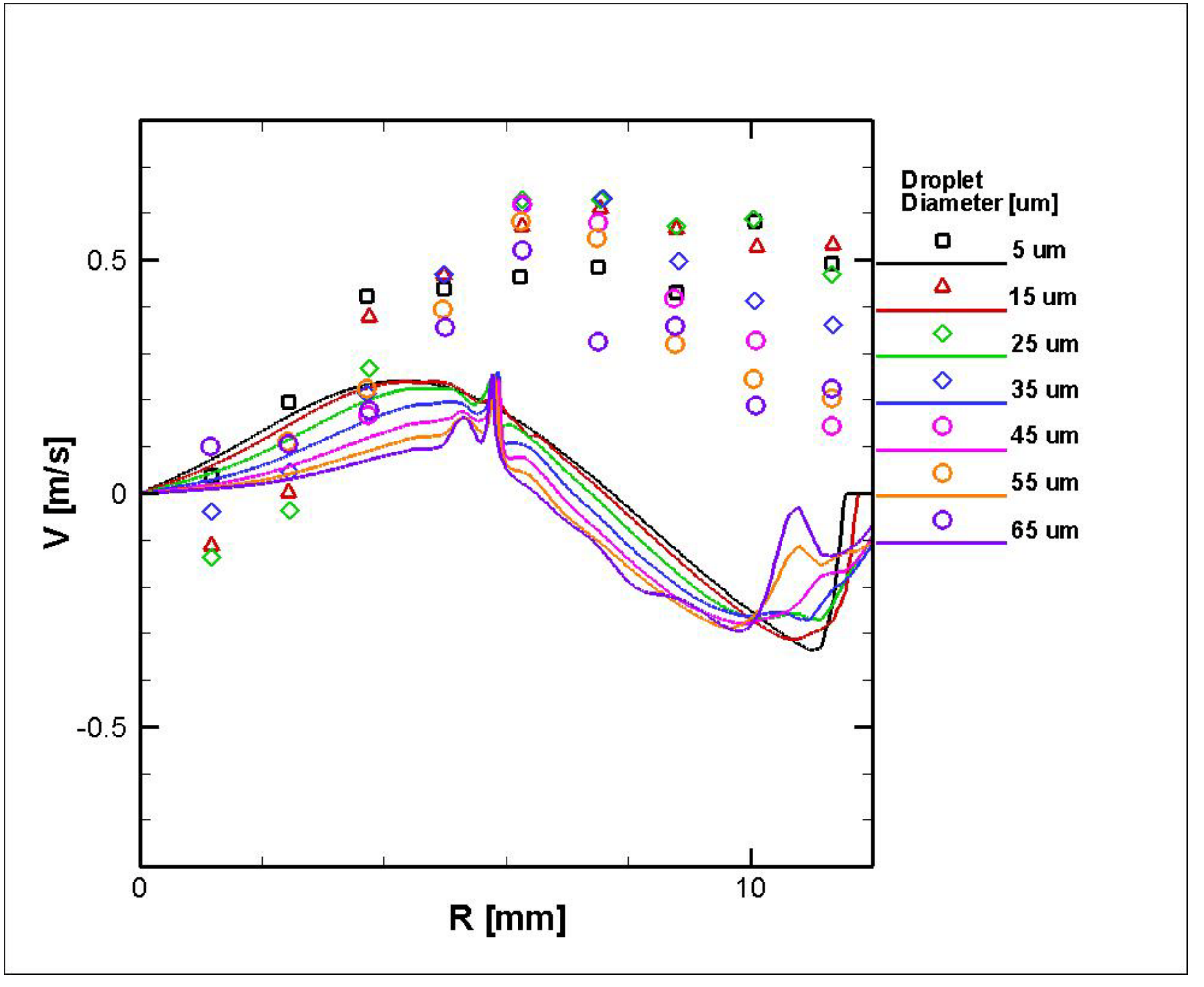
Distribution A: Approved for Public Release (Pending)

Results



Peak Flux, F_o [cm^3/cm^{2}s]
Droplet Diameter [um]
10^{-1}
10^{-2}
10^{-3}
10^{-4}
10^{-5}
10^{-6}
0
10
20
30
40
50
60
70
80
90
Dimensionless Axial Distance, Z
1D
5D
10D
20D
30D

Results



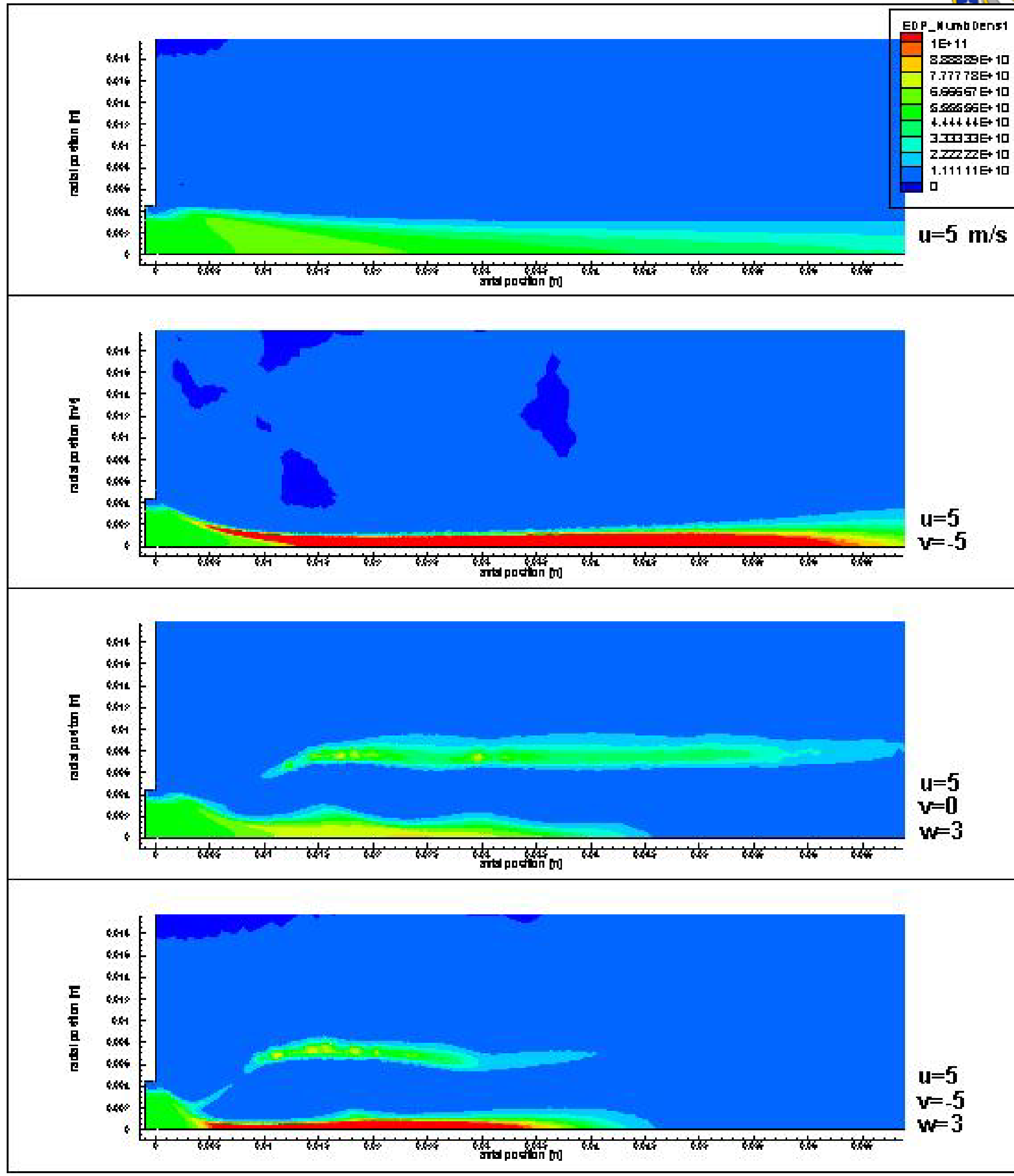
0.5
0
-0.5
V [m/s]
0
10
R [mm]
Droplet Diameter [um]
5 um
15 um
25 um
35 um
45 um
55 um
65 um

Summary

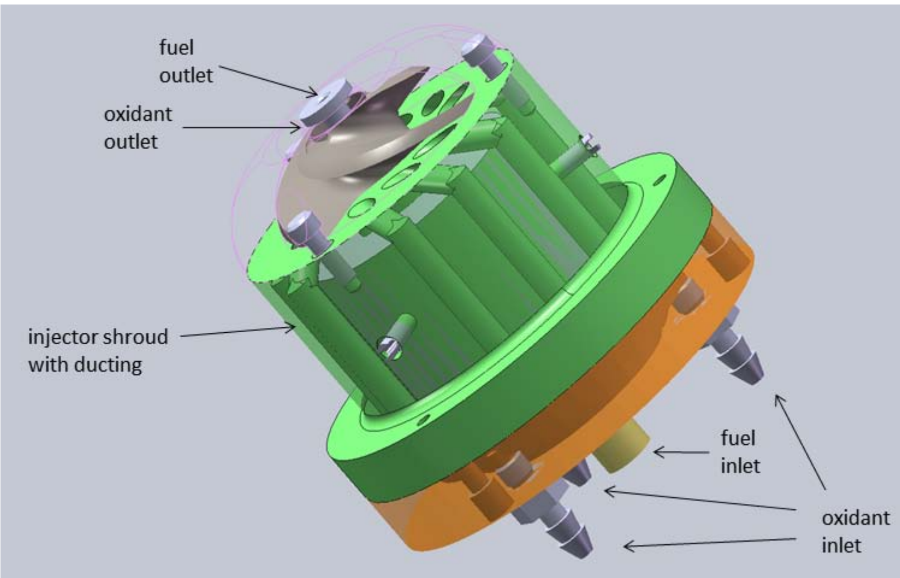
- Excess axial velocities of smaller droplets up to 50% slower than experimental while larger droplets show better agreement
- Error for peak volumetric fluxes higher for larger droplets– may be due to the fact that they carry more volume
- CFD agreement with radial velocities is poor– may be measurement error
- EDP features available in CFD++ may be missing important physics

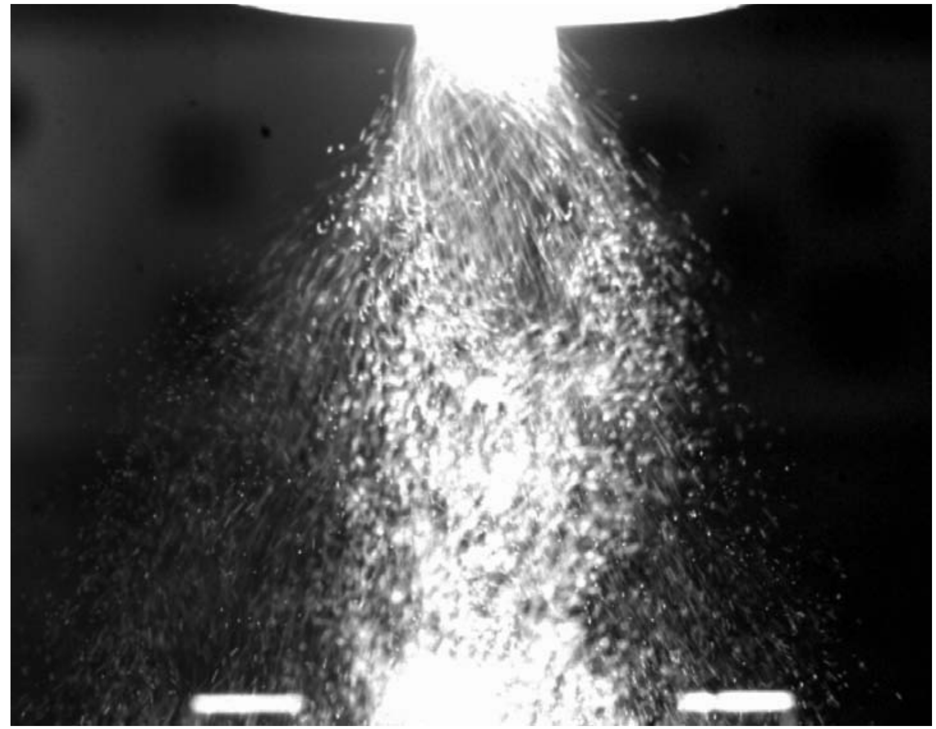
Future Work

Parametric study of effects of co-flow velocity components on droplet mixing



Future Work





Use CFD to design and determine conditions for experimental setup for testing combustion ignition strategies

Questions?