University of CINCINNATI

Department of Aerospace Engineering

Fundamental Processes and Design Considerations for the NexGen Fire Test Burner: Experiments and High-Fidelity Simulations

Prashant Khare, PhD
Assistant Professor
Department of Aerospace Engineering
University of Cincinnati

Presented at
International Aircraft Systems Fire Protection Forum Virtual Meeting
April 22, 2021

Acknowledgments
- This work is supported by FAA with Steve Summer as the Program Monitor
- Steve Summer, Rob Ochs, and other researchers at FAATC for inputs

Team: San Mou Jeng (Co-PI), Ryan Hasselbeck (staff), Manu Kamin (PhD student) and Prashant Khare (PI)

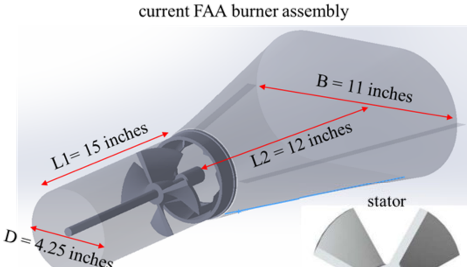
University of CINCINNATI

Department of Aerospace Engineering

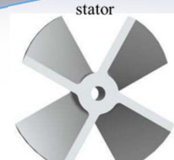
Computations: Overall Goals

- **Identify the detailed flow physics in the current and modified FAA NexGen burner systematically using high-fidelity LES computations**
  – cold flow without fuel spray
  – cold flow with fuel spray
  – "hot flow" with vaporizing fuel spray
  – reacting flow
- **Establish a reference database developed using high-fidelity LES simulations for the above conditions**

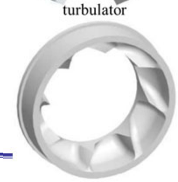
current FAA burner assembly



stator



turbulator



Objectives of this presentation

- Cold flow computations without fuel spray in current geometry
  – Identify the detailed flow physics for 2 inlet velocities
  – compare our results with measurements from Dr. Ochs' thesis
- Preliminary analysis with fuel sprays
  – liquid jet in crossflow configuration

* geometry dimensions source - https://www.fire.tc.faa.gov/pdf/materials/NexGenPlans_4_2016.pdf

University of CINCINNATI
Department of Aerospace Engineering

Approach: Large Eddy Simulation (LES)

Salient features of the in-house LES framework:

- Compressible finite volume solver
- Multi-block structured grid based solver with Message Passing Interface (MPI) for inter-process communication
- LES with dynamic Smagorinsky model for sub-grid scale modeling
- Up to fourth order accurate in space and third order in time
- Scalar or matrix artificial dissipation to assure numerical stability
- All Mach number with preconditioning schemes for steady and unsteady flows

University of CINCINNATI
Department of Aerospace Engineering

LES: Gas Phase Formulation

Favre-filtered conservation equations for gas-phase flowfield

mass $$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}_i}{\partial x_i} = \tilde{\dot{\rho}}_s$$

momentum $$\frac{\partial \bar{\rho}\tilde{u}_i}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}_i\tilde{u}_j}{\partial x_i} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial(\bar{\tau}_{ij} - \tau_{ij}^{SGS})}{\partial x_i} + \tilde{\dot{F}}_{s,i}$$

energy $$\frac{\partial \bar{\rho}\tilde{E}}{\partial t} + \frac{\partial[(\bar{\rho}\tilde{E} + \bar{p})\tilde{u}_i]}{\partial x_i} = \frac{\partial(-\bar{q}_i + \tilde{u}_j\bar{\tau}_{ji} - \sigma_i^{SGS} - H_i^{SGS})}{\partial x_i} + \tilde{\dot{Q}}_s$$

species $$\frac{\partial \bar{\rho}\tilde{Y}_k}{\partial t} + \frac{\partial(\bar{\rho}\tilde{Y}_k\tilde{u}_i)}{\partial x_i} = \frac{\partial(-\bar{\rho}\tilde{Y}_k\tilde{V}_{i,k} - Y_{i,k}^{SGS} - \theta_{i,k}^{SGS})}{\partial x_i} + \tilde{\dot{w}}_k + \tilde{\dot{S}}_{s,k}$$

Closure requirements

- Subgrid-scale (sgs) turbulence interaction

$$\tau_{ij}^{sgs},\ D_{ij}^{sgs},\ H_i^{sgs},\ \sigma_{ij}^{sgs},\ \Phi_{k,j}^{sgs},\ \theta_{k,j}^{sgs}$$

- Chemical reaction source and thermophysical properties & constitutive laws

$$\bar{\dot{\omega}}_k,\ Z,\ C_p,\ \mu,\ \lambda,\ D_{im}$$

Li, H. G., Khare, P., Sung, H. G., & Yang, V. (2016). A large-eddy-simulation study of combustion dynamics of bluff-body stabilized flames. CST, 188(6), 924-952.

University of CINCINNATI
Department of Aerospace Engineering

LES: Dispersed Phase Formulation

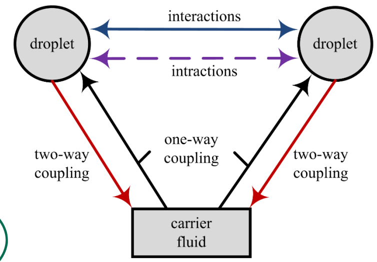
Spray Dynamics

$$\frac{d\boldsymbol{x}_d}{dt} = \boldsymbol{u}_d \qquad m_d \frac{d\boldsymbol{u}_d}{dt} = \boldsymbol{F}_d$$

Basset-Boussinesq-Oseen (BBO) equation

$$\boldsymbol{F}_d = \underbrace{\frac{\pi}{12} d_d^3 \rho (\dot{\boldsymbol{u}} - \dot{\boldsymbol{u}}_d)}_{\text{virtual mass}} + \underbrace{\frac{\pi}{8} d_d^2 \rho C_d |\boldsymbol{u} - \boldsymbol{u}_d| (\boldsymbol{u} - \boldsymbol{u}_d)}_{\text{drag force}} + \underbrace{\frac{\pi}{6} d_d^3 (-\nabla p + \nabla \cdot \boldsymbol{\tau})}_{\text{buoyancy}}$$

$$+ \underbrace{\frac{3}{2} d_d^2 \sqrt{\pi \rho \mu} C_B \left[\int_{t_0}^{t} \frac{\dot{\boldsymbol{u}} - \dot{\boldsymbol{u}}_d}{\sqrt{t - \xi}} d\xi + \frac{(\boldsymbol{u} - \boldsymbol{u}_d)_{t_0}}{\sqrt{t}} \right]}_{\text{Basset force}} + \underbrace{m_d \boldsymbol{g}}_{\text{gravity}} + \underbrace{\boldsymbol{F}_L}_{\text{lift}}$$



one-way: effect of carrier fluid on droplets
two-way: mutual coupling between droplets and carrier fluid
four-way: droplet-droplet intractions and interactions (e.g., collision & coalescence)

Mass and Heat Transfer

$$\frac{dm_d}{dt} = -\dot{m}_d$$

$$m_d C_l \frac{dT_d}{dt} = \dot{Q}_{conv} - \dot{m}_d L_v = h_d \pi d_d^2 (\tilde{T} - T_p) - \dot{m}_d L_v$$

Spray breakup models:

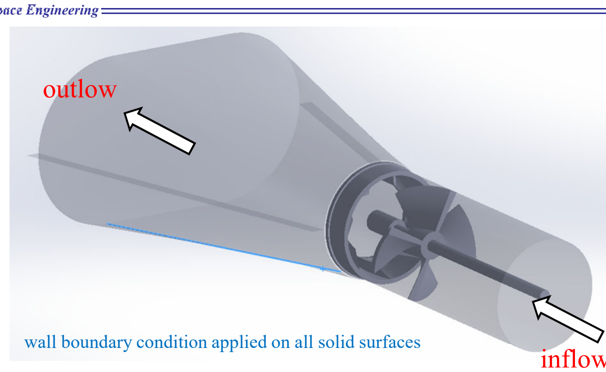
- K-H wave model for primary atomization
- Taylor Analogy Breakup (TAB) model for secondary atomization

Khare, P., Wang, S., & Yang, V. (2015). Modeling of finite-size droplets and particles in multiphase flows. Chinese Journal of Aeronautics, 28(4), 974-982.

University of CINCINNATI
Department of Aerospace Engineering

Cold Flow: FAA Burner Geometry

Case I: inlet airflow @ 10m/s

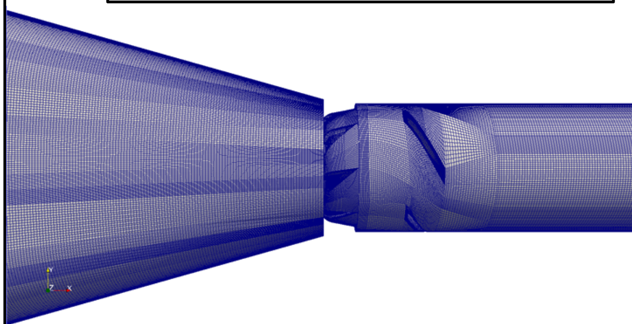


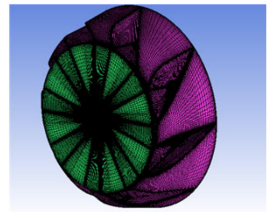
Inflow velocity : 10 m/s
Inlet air temperature : 300 K
Inlet air pressure: 1 atm
Reynolds number : 66,200

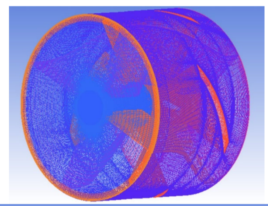
Computational Grid

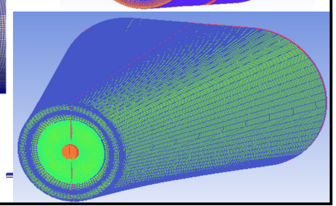
- Block structured grid with only hexahedral elements.
- Multi-block grid for massively parallel computing
- Smallest grid size based on y+ = 5 ≈ 0.14mm

Total number of grid points : 21.2 million
Total number of grid blocks : 1960
Smallest grid size: 0.4 mm







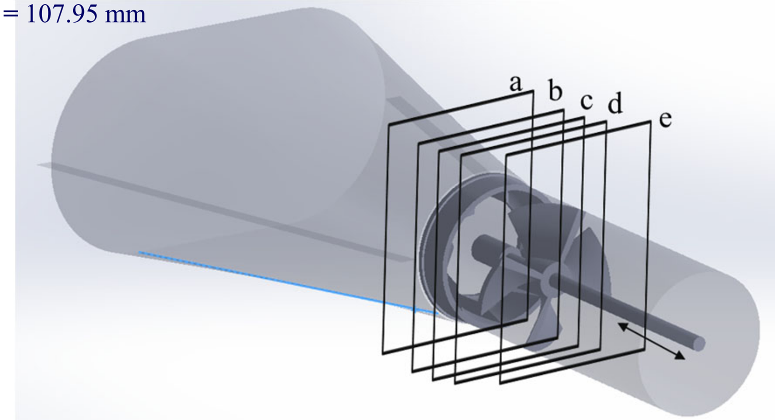


University of CINCINNATI
Department of Aerospace Engineering

Locations of cross sectional planes for instantaneous visualization

Reference position: turbulator plane

d = 107.95 mm



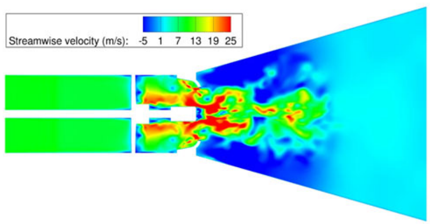
Locations a, b, c, d and e (combustion chamber, after turbulator, turbulator plane, after stator and stator plane respectively)

a : x/d = -0.5, b : x/d = 0, c : x/d = 0.1, d : x/d = 0.5, x/d = 0.65

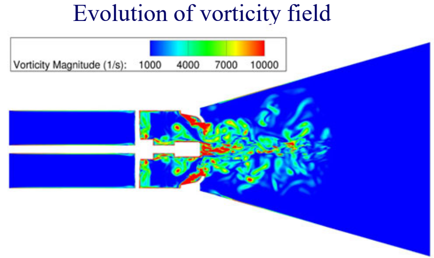
Streamwise Velocity (video)

Detailed Flow Dynamics

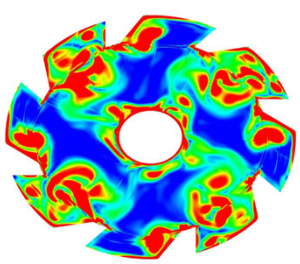
streamwise velocity evolution

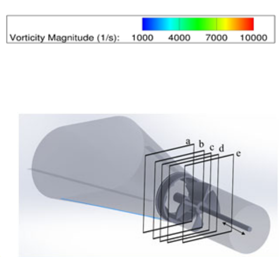


Evolution of vorticity field

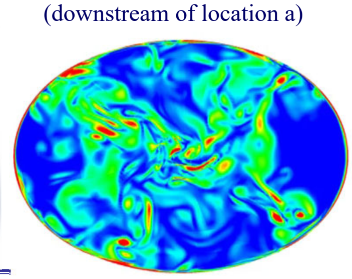


vorticity field - turbulator (location c)



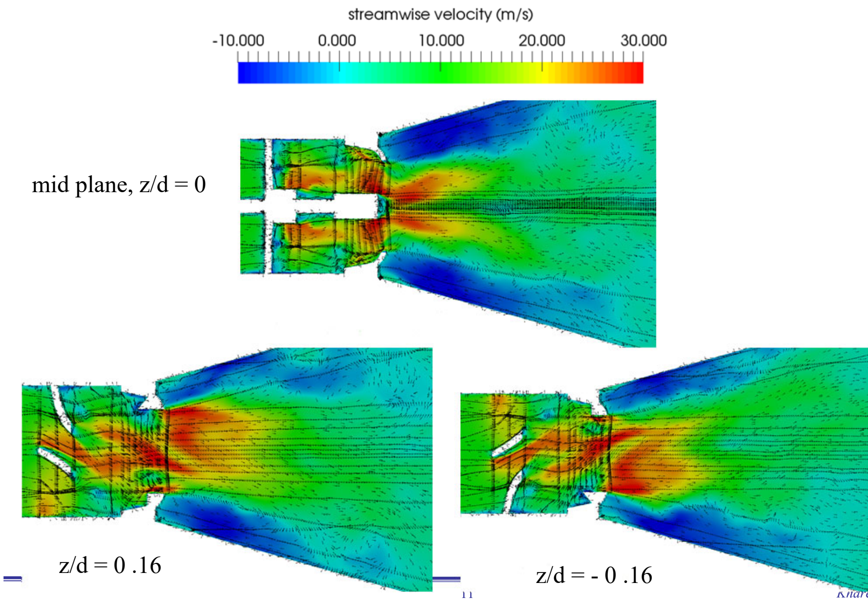


vorticity field - burner cone (downstream of location a)



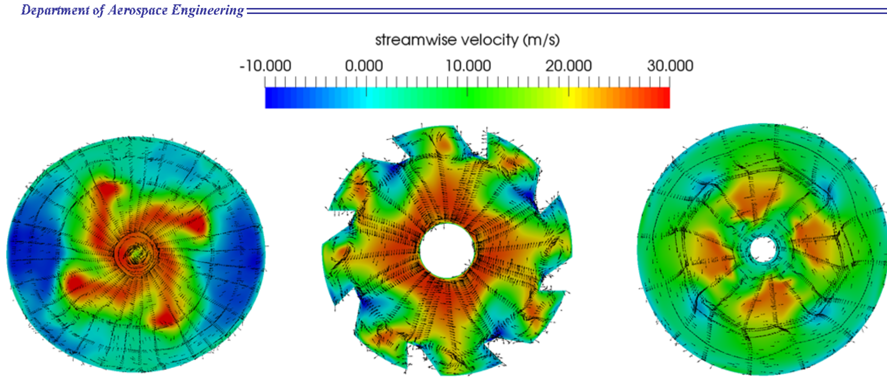
Mean Velocity
Center and Off-Center Planes

University of CINCINNATI
Department of Aerospace Engineering

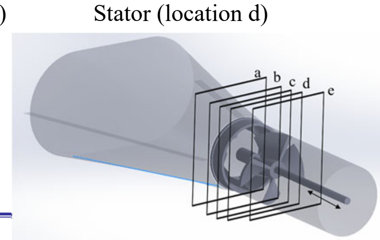


Mean Velocity
Streamwise Sectional Planes

University of CINCINNATI
Department of Aerospace Engineering

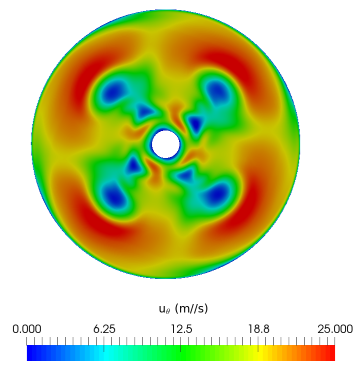


Burner cone (location a) Turbulator (location c) Stator (location d)

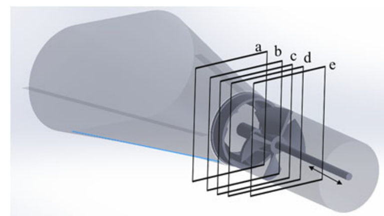


University of CINCINNATI
Department of Aerospace Engineering

Swirl Number



Contour of mean azimuthal velocity downstream of the stator (location d)

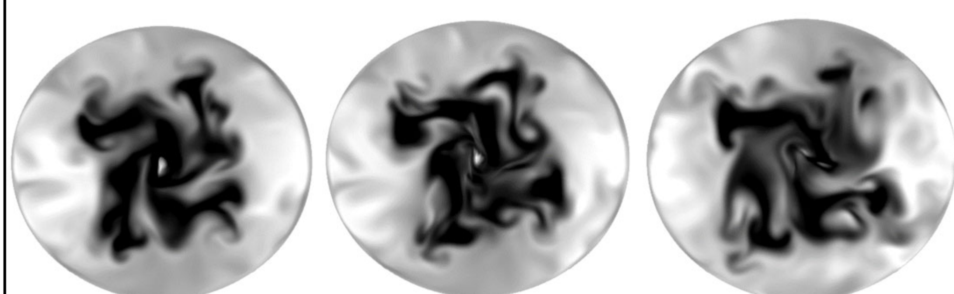


- The azimuthal velocity component suggests that stator is seen to impart swirl to incoming flow
- Swirl number S = 0.76 based using the expression below:

$$S = \frac{\int \rho U_x U_\theta r dA}{R \int \rho U_x^2 dA}$$

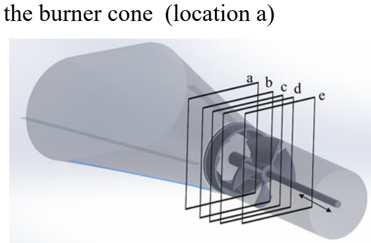
University of CINCINNATI
Department of Aerospace Engineering

POD Analysis using Method of Snapshots



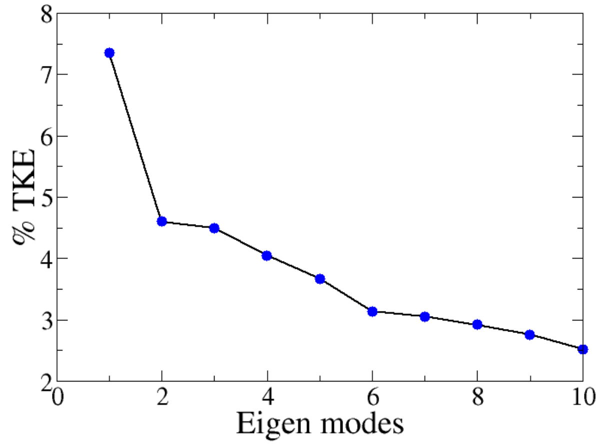
Sample instants of streamwise velocity contours in the burner cone (location a)

- Snapshots of instantaneous velocity fields is obtained at the fixed cross sectional location in the burner cone.
- A few sample instants are as shown in figure above, which is at a cross sectional plane in the burner cone



University of CINCINNATI
Department of Aerospace Engineering

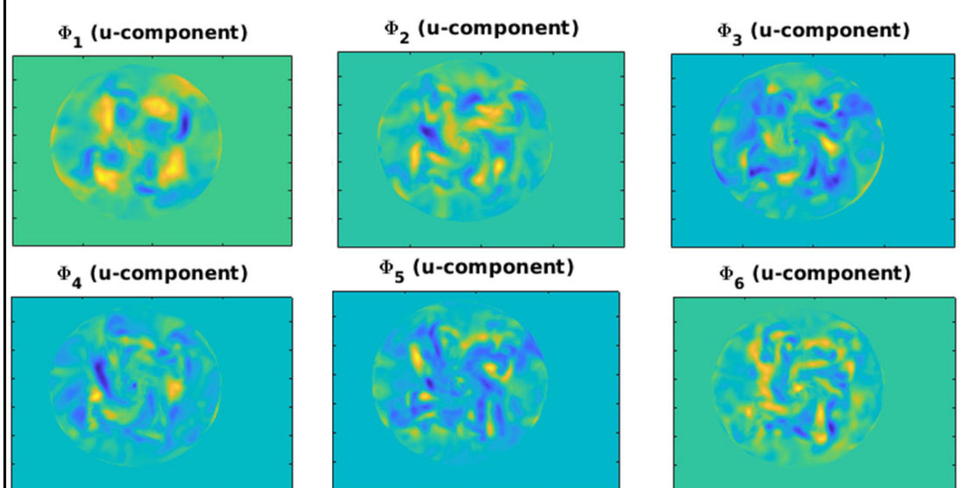
Distribution of Energy Content



- Above plot shows the energy content per eigen mode.
- Only first 10 eigen modes (out of 83) that have significant percentage of total energy content are shown here.

University of CINCINNATI
Department of Aerospace Engineering

POD Eigen Modes

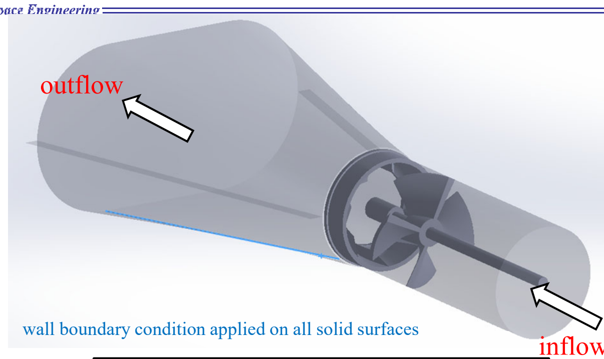


visualization of the energy content of the first 6 modes

University of CINCINNATI
Department of Aerospace Engineering

Cold Flow: FAA Burner Geometry

Case II: inlet airflow @ 3.86m/s



Inlet air temperature : 283 K
Pumped air pressure : 5.15 bar
Inlet air density: 1.2474 kg/m^3
Mass flow rate : 0.0384 kg/s
Equivalent inflow velocity : 3.86 m/s
Reynolds number: 30623

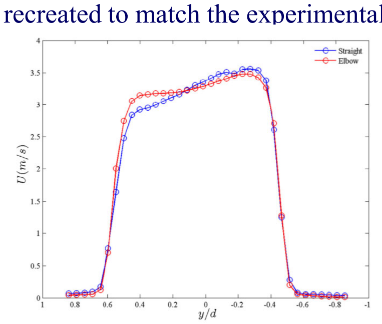
Note: the experiment was conducted the burner cone. A difference in some flow features can be expected as a result.

Ochs, R. I. (2013). Design and Analysis of the Federal Aviation Administration Next Generation Fire Test Burner, Ph.D Thesis, Rutgers University.

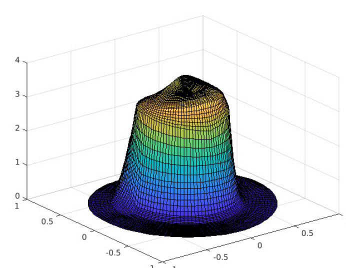
University of CINCINNATI
Department of Aerospace Engineering

Inlet Velocity Profile

- Asymmetry in the inflow profile within the draft tube
- The asymmetry is therefore accounted for, and a velocity profile is recreated to match the experimental profile.



Inlet velocity profile reported in the experiments



Inlet velocity profile reconstructed for simulation

University of CINCINNATI

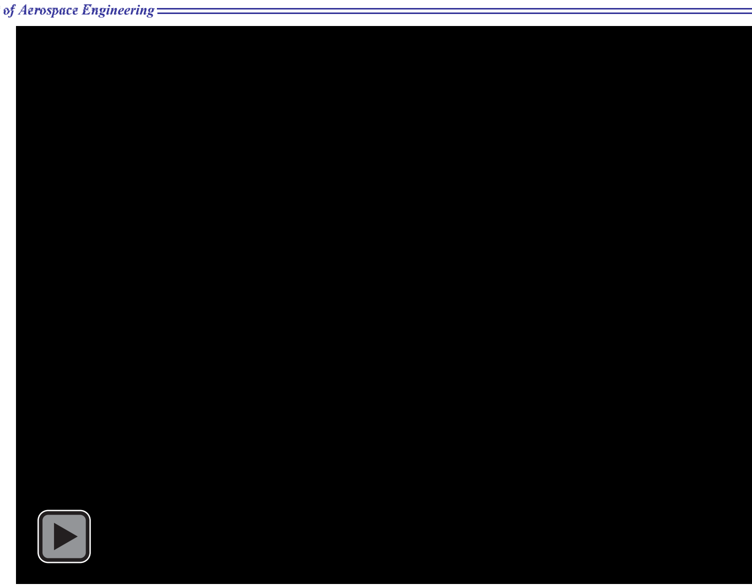
Department of Aerospace Engineering

Streamwise Velocity Dynamics (video)

University of CINCINNATI

Department of Aerospace Engineering

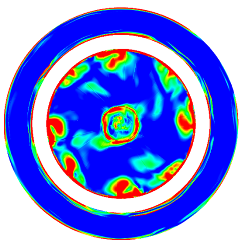
Vorticity Dynamics (video)



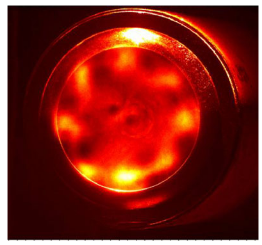
Preliminary Comparison with Measurements

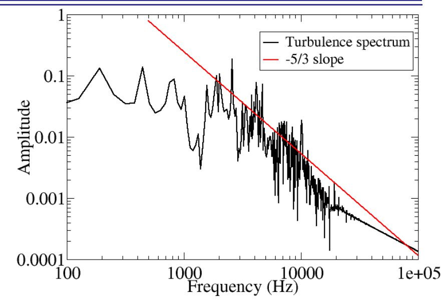
University of CINCINNATI
Department of Aerospace Engineering

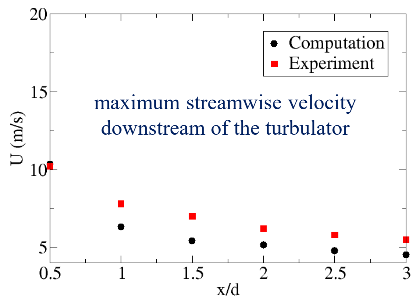
present



experiment



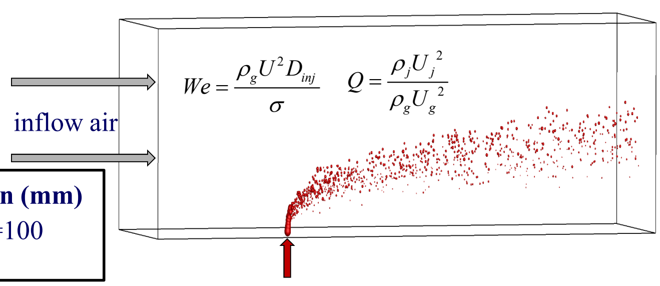




- experiment was conducted the burner cone and the simulations are conducted with the cone
- the recirculation zones at the burner cone lead to reduction in maximum sectional velocities

Fuel Spray: Code Validation I

University of CINCINNATI
Department of Aerospace Engineering



$$We = \frac{\rho_g U^2 D_{inj}}{\sigma} \qquad Q = \frac{\rho_j U_j^2}{\rho_g U_g^2}$$

computation domain (mm)
H=25.8, W=28.9, L=100
D_{inj} = 0.254 mm

Case A (air at room temperature)	Case B (air at 573 K)
Inflow Air - U_g=116m/s, 1 atm, 300 K	**Inflow Air -** U_g=166m/s, 1 atm, 573 K
Liquid jet - T=298K, U_j=12.01 m/s	**Liquid jet** - T= 298K, U_j=12.36 m/s
We = 68 **, Q** = 9	**We** = 68.3 **, Q** = 9

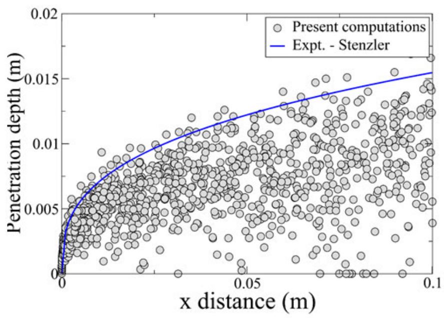
Mesh size : 1.6 million
Minimum grid size = 0.02 mm

*Stenzler et al., Atomization and Sprays , 2006

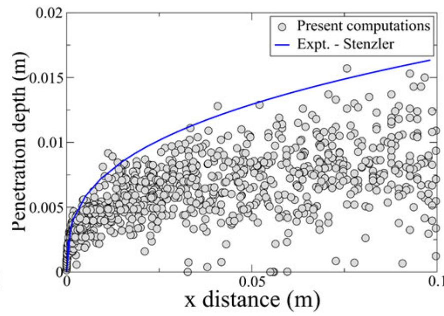
University of CINCINNATI
Department of Aerospace Engineering

Comparison with Experiments

liquid penetration trajectory



Case A : air temperature = 300 K



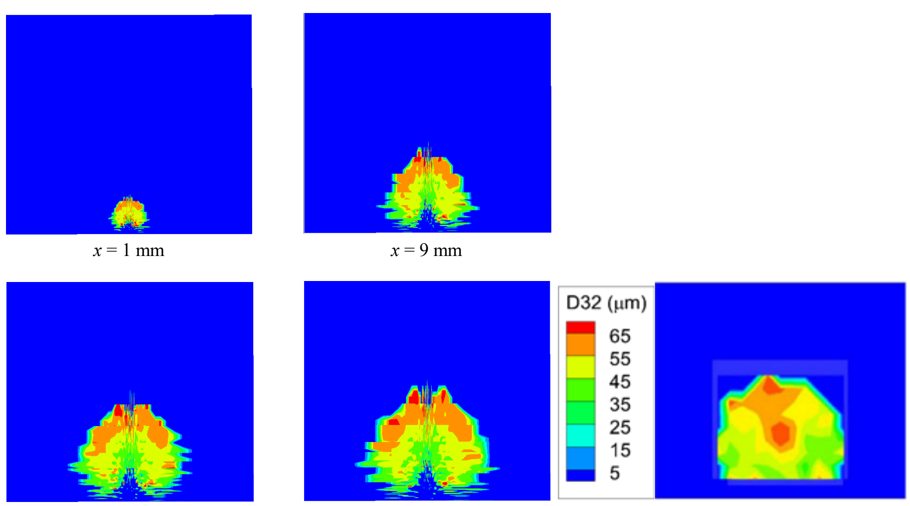
Case B : air temperature = 573 K

University of CINCINNATI
Department of Aerospace Engineering

Comparison with Experiments

sauter mean diameter

Case A : SMD distribution at various streamwise locations



x = 1 mm; *x* = 9 mm; *x* = 18 mm; *x* = 25.4 mm; *x* = 25.4 mm, experiment

University of CINCINNATI
Department of Aerospace Engineering

Conclusions & Upcoming Tasks

- Conducted high-fidelity cold flow simulations for two different flow conditions on the current FAA burner geometry using LES
- Analyzed the resulting flow field by visualizing the flowfiled at different spatial locations
- Conducted POD analysis to identify the dominant modes
- Validated the spray formulation using liquid jet in crossflow configuration

Upcoming Tasks

- Continue analysis of the second case
- Conduct cold flow simulations on the geometry modification informed by experiments
- Conduct cold flow simulations with fuel spray for the above two geometries

University of CINCINNATI
Department of Aerospace Engineering

Thank you for your attention.

Prashant Khare, PhD
Assistant Professor
Department of Aerospace Engineering
University of Cincinnati
Tel: (513) 556-5248
Email: Prashant.Khare@uc.edu
Web: http://khare.uc.edu