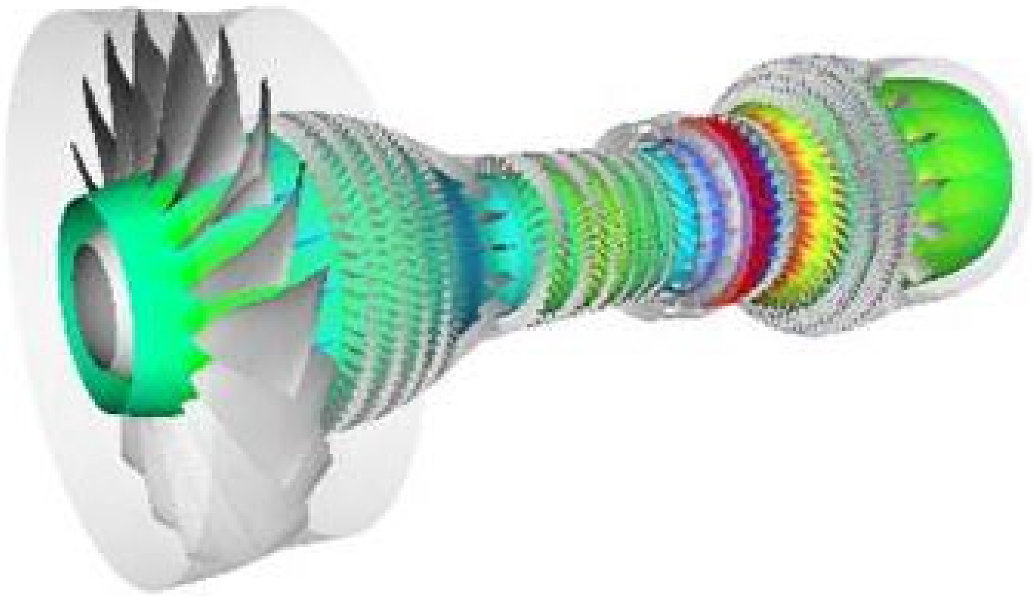
Revolutionary Computational Aerosciences (RCA) Research at NASA Glenn Research Center

Nicholas J. Georgiadis

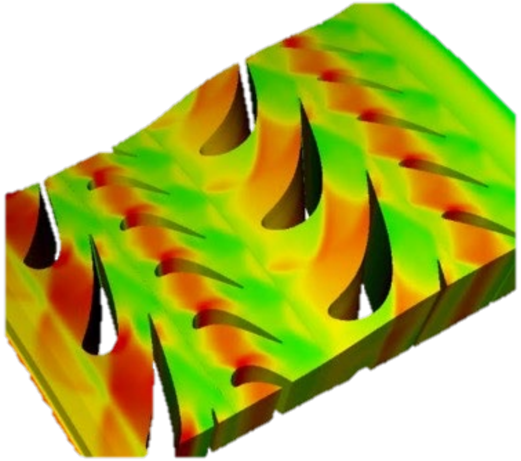
Outline

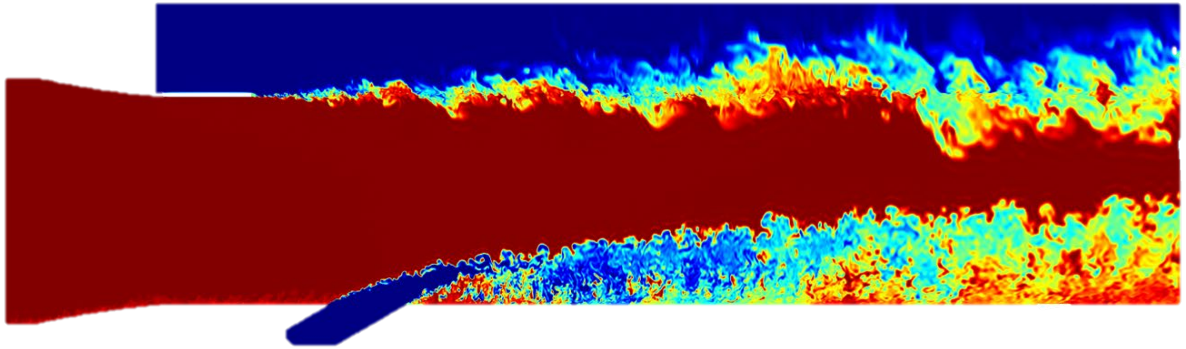
- RCA Goals for Improving Computational Modeling for Aerospace R&D
- Overall TTT-RCA Efforts and Research Portfolio
- Airframe and Propulsion Grand Challenge Problems
- Glenn Research Efforts
- Example – Turbulent Heat Flux Experiments and Computations
- Work Sponsored via Grants



RCA Goals for Improving Computational Modeling

- **Historically, aircraft and aerospace system R&D had been performed primarily with experimental and empirical techniques.**
- **In the past 2+ decades, computational methods (i.e. For aerodynamics, computational fluid dynamics (CFD)) have been used with ore frequency and greater trust.**
- **The chief technical challenges have been: (1) physical modeling – especially for turbulence in aerodynamics) and (2) computing resources.**
- **The Goal of RCA research is to improve computational modeling capabilities for aerospace flows – all speed regimes.**





Key Terminology

- CFD = Computational fluid dynamics
- RANS = Reynolds-averaged Navier-Stokes – the standard method of performing turbulent flow CFD, only provides the averaged flow field.
- SRS = Scale-Resolving Simulations – directly calculates the important time-varying turbulent motion. Includes DNS, LES, hybrids. Much more computationally costly than RANS.
  - DNS – direct numerical simulation – calculate ALL turbulent scales (not possible for most engineering flows).
  - LES - large eddy simulation - calculates the important time-varying turbulent motion. Very expensive.
  - Hybrid RANS-LES – use RANS in attached wall boundary layers; use LES in free shear layers, separations; remains an evolving method.

- CHT = Conjugate Heat Transfer = use heat transfer solver for solid, coupled to CFD solver

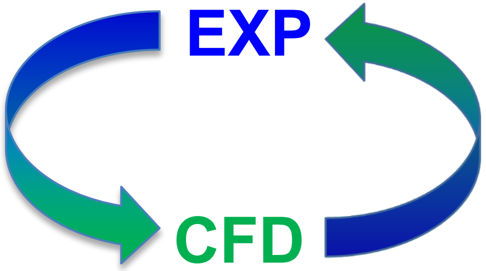
Overall RCA Research Portfolio

Technical Areas and Approaches

- **Physical Modeling and Simulations**
  - ***DNS/LES, WMLES, hybrid RANS/LES*** and Lattice-Boltzmann Method
  - Data driven (ML-assisted) turbulence modeling
  - Laminar-turbulent transition modeling, including ML
- **HPC Tools and Methods**
  - ***Effective utilization of emerging HPC hardware***
  - ***Accurate, efficient, and robust computational methods***
  - Grid adaptation
  - Uncertainty quantification
- **CFD Validation Experiments**
  - Juncture flow
  - Canonical flow separation (wall bumps/ramps)
  - High-lift experiments (including NTF)
  - ***Shock/boundary-layer interaction***
  - TDT aeroelastic experiment
  - ***Centrifugal compressor experiment***
  - ***Thermal mixing experiment***
  - ***Turbulent Heat Flux (THX) experiments***
  - Boundary layer transition

Research conducted at ARC/GRC/LaRC and through NRAs

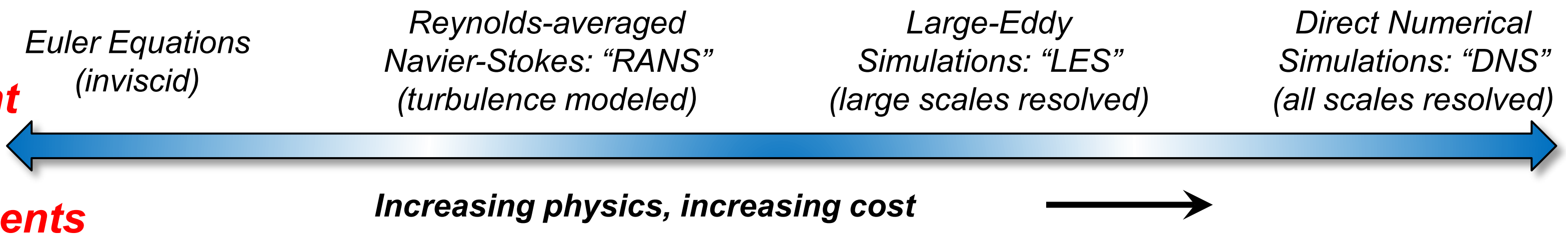
Foundational research aimed at solving technical challenges



EXP

CFD

Outcome: Accurate, fully validated CFD capability



Euler Equations (inviscid)

Reynolds-averaged Navier-Stokes: “RANS” (turbulence modeled)

Large-Eddy Simulations: “LES” (large scales resolved)

Direct Numerical Simulations: “DNS” (all scales resolved)

Increasing physics, increasing cost

Aircraft Grand Challenge Problem

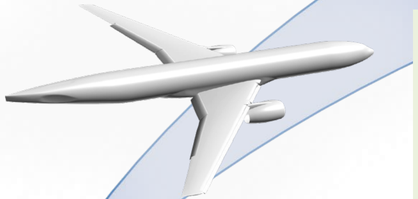
Advancing High Lift Aerodynamic Prediction
Series of Technical Challenges

Focus on key technical obstacles for specific time periods to make progress towards solving the grand challenge

Ground-Based Experimental Testing

FLOW PHYSICS PREDICTION

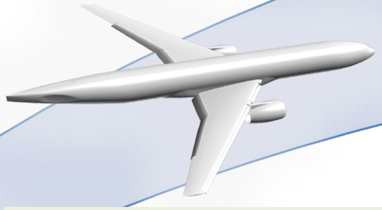
Sub-Challenge #1
1-3 years



Representative WT Geometry
Landing/TO configuration + nacelle/pylon
Re effects (atmospheric, pressurized, cryogenic environments)
Interactional flow physics (separation, vortex flow)
Static aeroelastics

CFD-generated data compared to WT data

Sub-Challenge #2
3-6 years



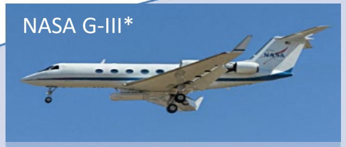
Representative WT Geometry
S&C (tail/control surfaces/trim)
Cross-flow effects
Acoustics (landing gear)
Engine propulsion effects
Ice effects

CFD-generated data compared to WT data

High Lift Common Research Model Ecosystem

Sub-Challenge #3
6-10+ years

LOW-SPEED STALL SPEED DETERMINATION



NASA G-III*

NASA AirSTAR*

Generic Flight Vehicle
Sub- or full-scale flight geometry
Flight Re
Steady flight
Basic maneuver
Dynamic structural response

CFD-generated data compared with flight-derived data

* Potential flight test vehicle configuration

Grand Challenge
15+ years

LOW-SPEED WIND-UP TURN

*

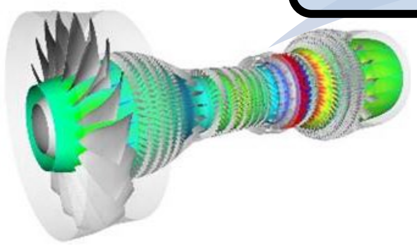
*

Generic Flight Vehicle
Full scale flight geometry
Flight Re
Dynamic, maneuvering flight
Dynamic structural/system response
Environmental effects
Engine power effects

CFD-based flight simulation (flight test used to verify flight simulation)

AIAA CFD 2030 Integration Committee

Propulsion Grand Challenge Problem



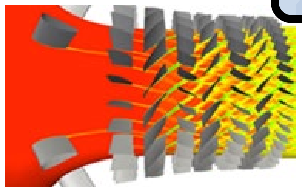
Integrated, Multicomponent

Combustor/Turbine & Compressor/Combustor

Inlet/Fan/bypass/Comp. & Turbine/Exhaust system

Full-engine including Secondary Air System

Aerodynamics, Aeromechanics, Aeroacoustics



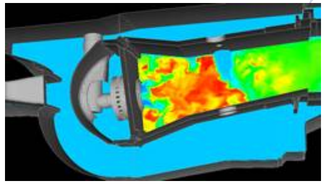
Fan, Compressor
(Inlet, Fan, OGV, Bypass, LPC, [IPC], HPC)

Single-Stage

Fan + OGV + Bypass

Multi-stage, Front Block

Multi-stage, Rear Block



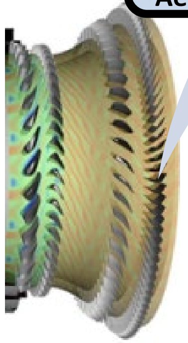
Combustor
(Diffuser, Fuel Nozzle, Liner)

Assumed Sprays, Improved Emissions/Soot

Radiation, Heat Transfer

Detailed Sprays. Advanced Emissions models

Full Annulus, Dynamics, LBO, Ignition, Relight



Turbine
(HPT, [IPT], LPT, Exhaust System)

Single-Stage Aerodynamics, Aeromechanics

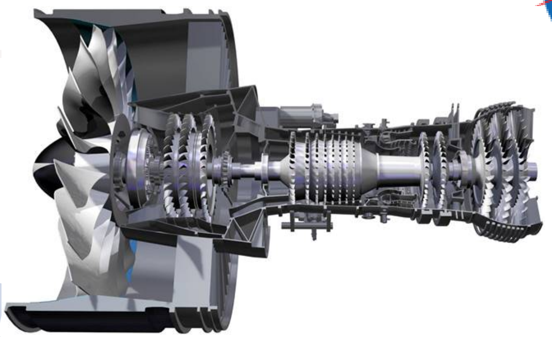
Single-Stage Heat Transfer

Multi-Stage Aerodynamics, Aeromechanics

Multi-Stage Heat Transfer

Challenges:
- Turbulence Modeling
- Wall-resolved/Near-wall modeling
- Multi-phase, Reacting, Heat Transfer…….
- Algorithms, pre-, post-process., Infrastructure

Enablers:
- Test Data (Common Research Models)
- HPC exploitation



A Full Engine Simulation in Less Than 1 Week

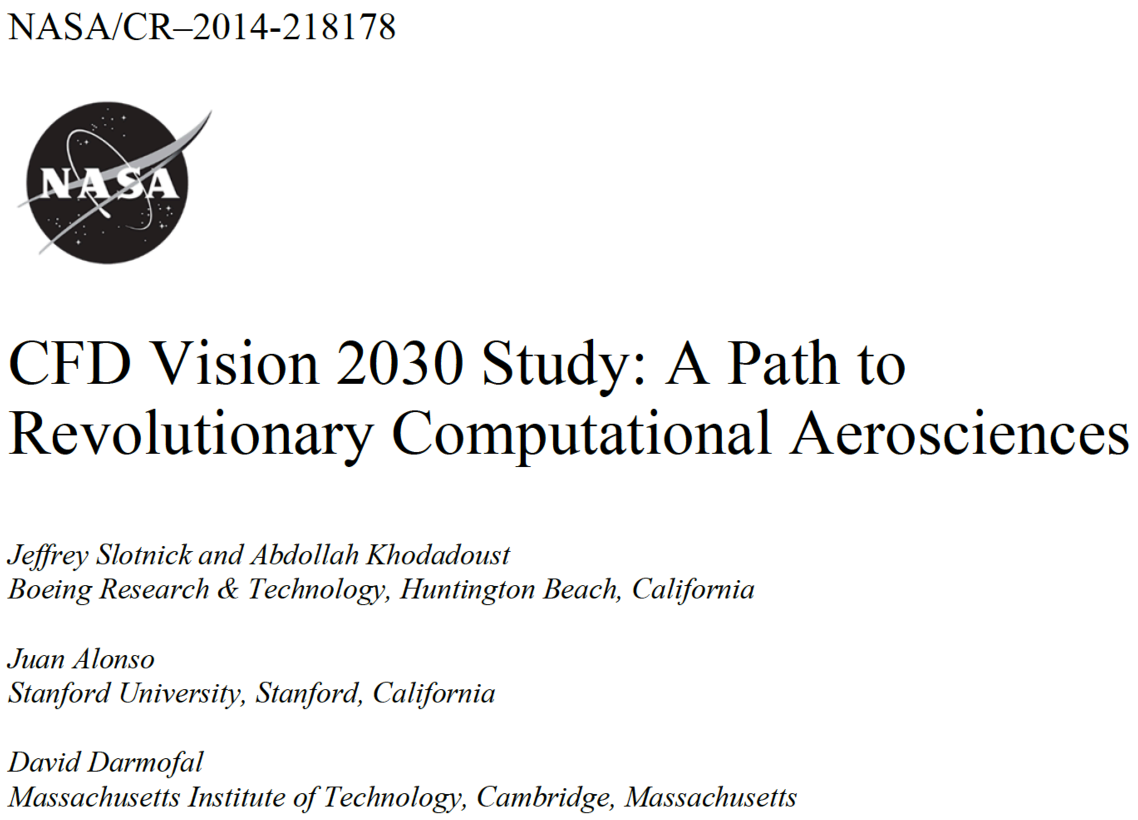
From model build to results enabling…
- meeting **performance, operability, emissions, durability metrics**
- with **geometric fidelity & accuracy of physics** to
- **reduce/eliminate testing**
- and deliver **billions of dollars savings** per year

2020 2025 2030 2035 2040

Long – Term Vision for Computational Modeling

- **Improve the accuracy / reliability of CFD using scale-resolving simulations for challenging turbulent flows.**

Vision 2030 Study



NASA/CR–2014-218178

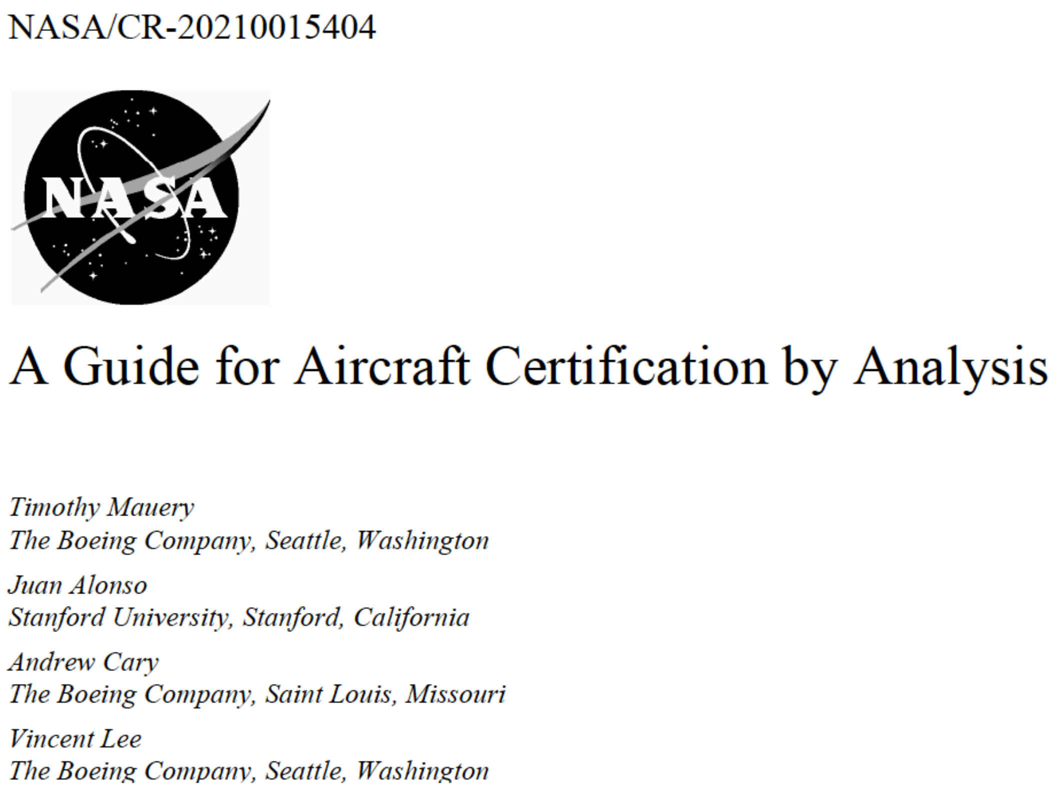
CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences

Jeffrey Slotnick and Abdollah Khodadoust
Boeing Research & Technology, Huntington Beach, California

Juan Alonso
Stanford University, Stanford, California

David Darmofal
Massachusetts Institute of Technology, Cambridge, Massachusetts

CbA Study



NASA/CR-20210015404

A Guide for Aircraft Certification by Analysis

Timothy Mauery
The Boeing Company, Seattle, Washington

Juan Alonso
Stanford University, Stanford, California

Andrew Cary
The Boeing Company, Saint Louis, Missouri

Vincent Lee
The Boeing Company, Seattle, Washington

GRC RCA Research Tasks

CFD Physical Modeling and Simulations

- High Fidelity – Simulations for Turbulent Propulsion Flows
- Glenn Flux Reconstruction Code
- Enhanced CFD Fan Aeromechanics
- Inlet Distortion Prediction

CFD Validation Experiments

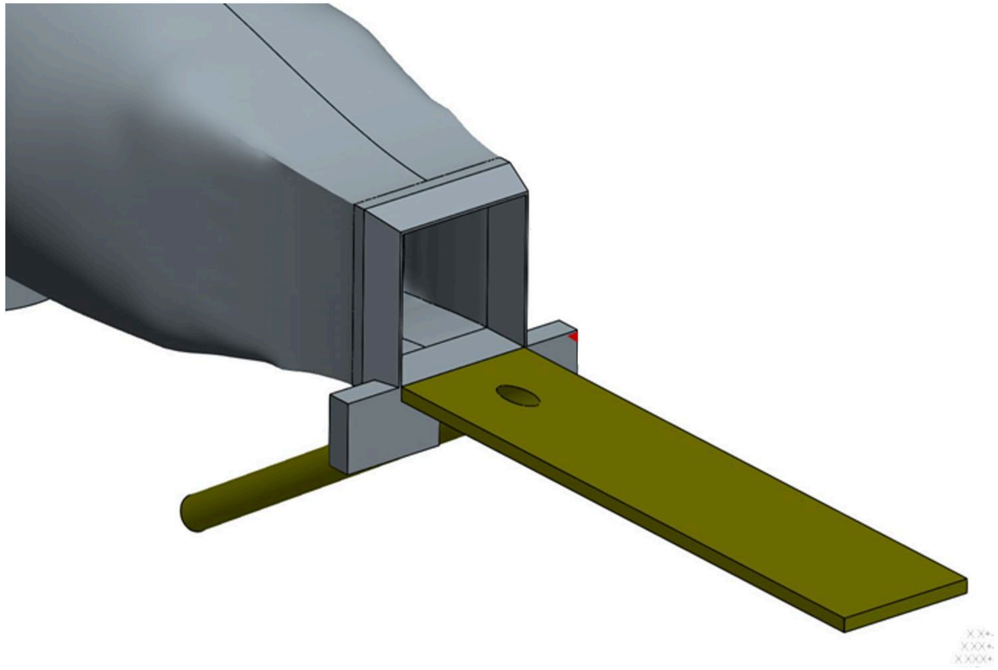
- Shock/boundary-layer interaction
- Centrifugal compressor experiment
- Thermal mixing experiment
- Turbulent Heat Flux (THX) experiments

- On the next set of charts, we show an example of an experimental effort followed by computational investigation – for the THX configurations.

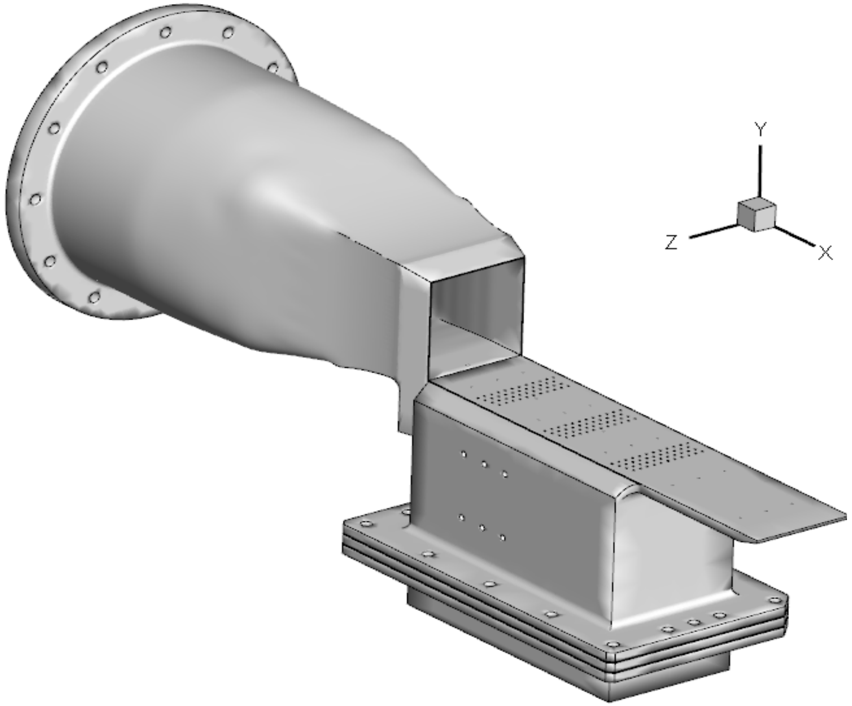
THX - Experimental Background (1 of 2)

- Turbulent Heat Flux (THX) Experiments – have completed 5 phases.
- THX3 (2017) and THX4 (2018) were experiments run in GRC AAPL, same nozzle, stainless steel hardware.
- <u>THX Experimental Goal:</u> Take measurements of velocities and temperatures, mean and rms (turbulent) for flows where turbulent transport of heat is important.
- <u>THX Computational Goal:</u> Determine capabilities of RANS and Scale-Resolving Simulations for the THX3 and THX4 configurations (API Milestone).

<u>THX 3</u>: Single Cooling Hole – fundamental aerodynamic & thermodynamic scientific study



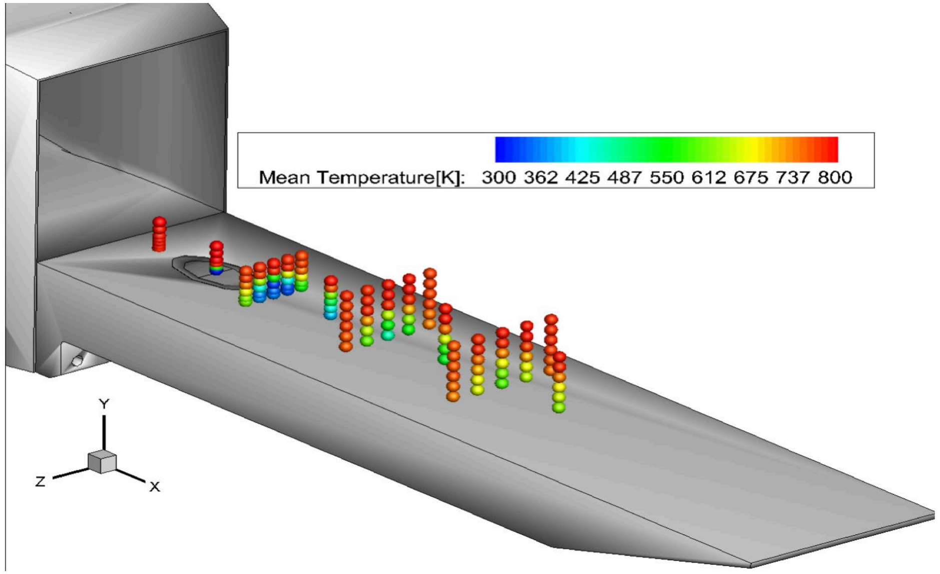
<u>THX 4</u>: Porous Plate – also detailed in-flow measurements, but more of an engineering configuration.



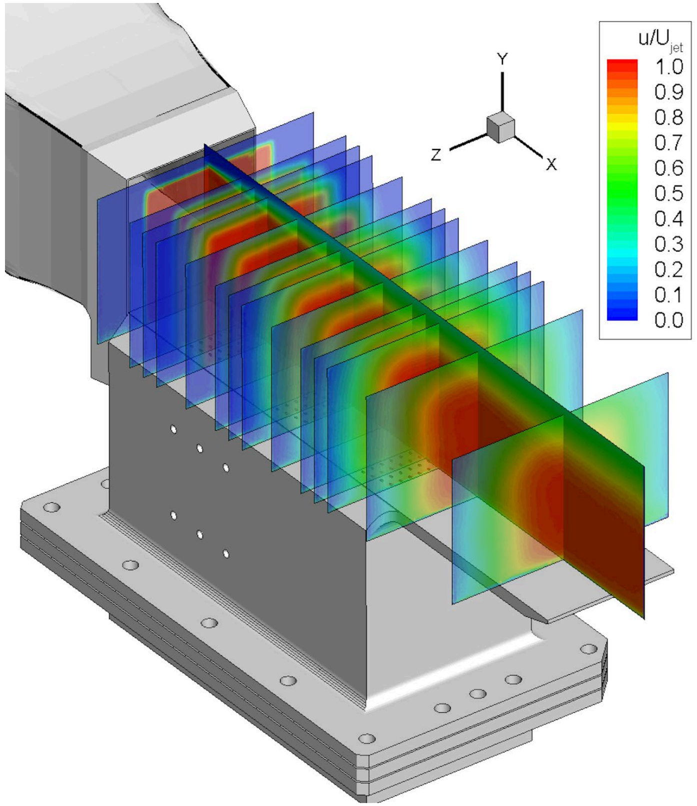
THX - Experimental Background (2 of 2)

- **Sample experimental measurements:**

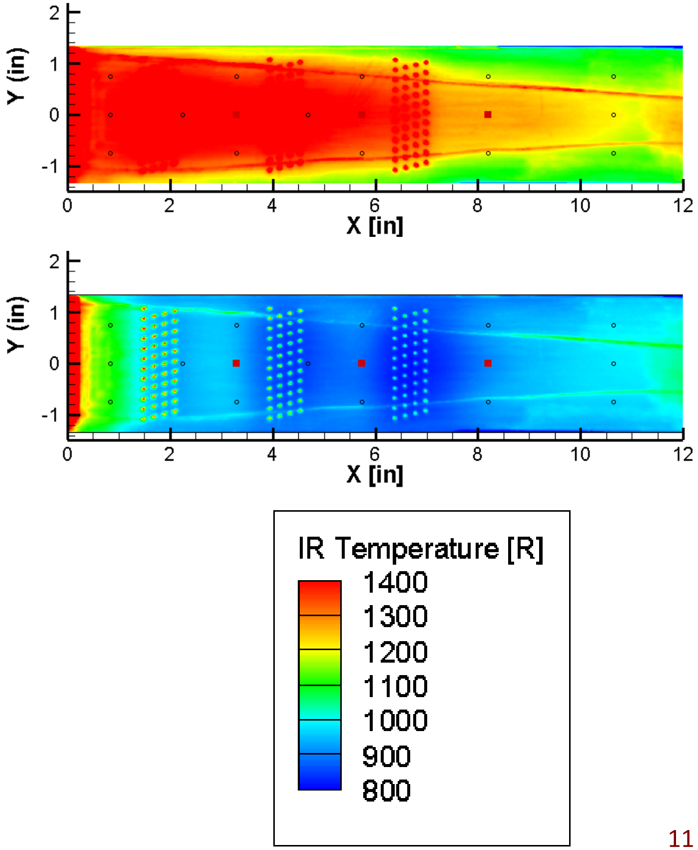
THX 3: Raman temperature measurements (mean and rms)



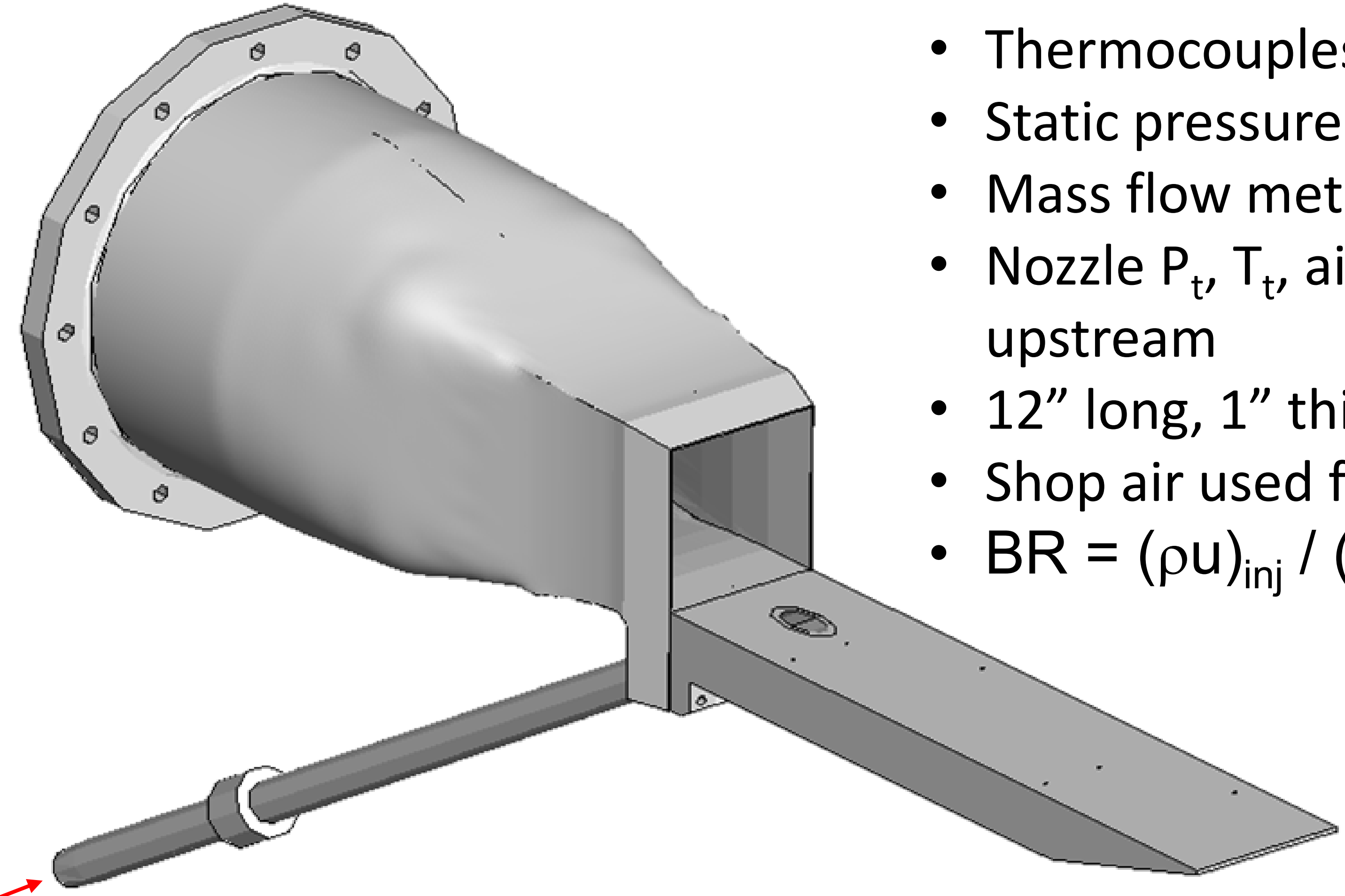
THX 4: PIV velocity measurements (mean and rms)



THX 4: Surface IR temperature measurements



THX Phase III

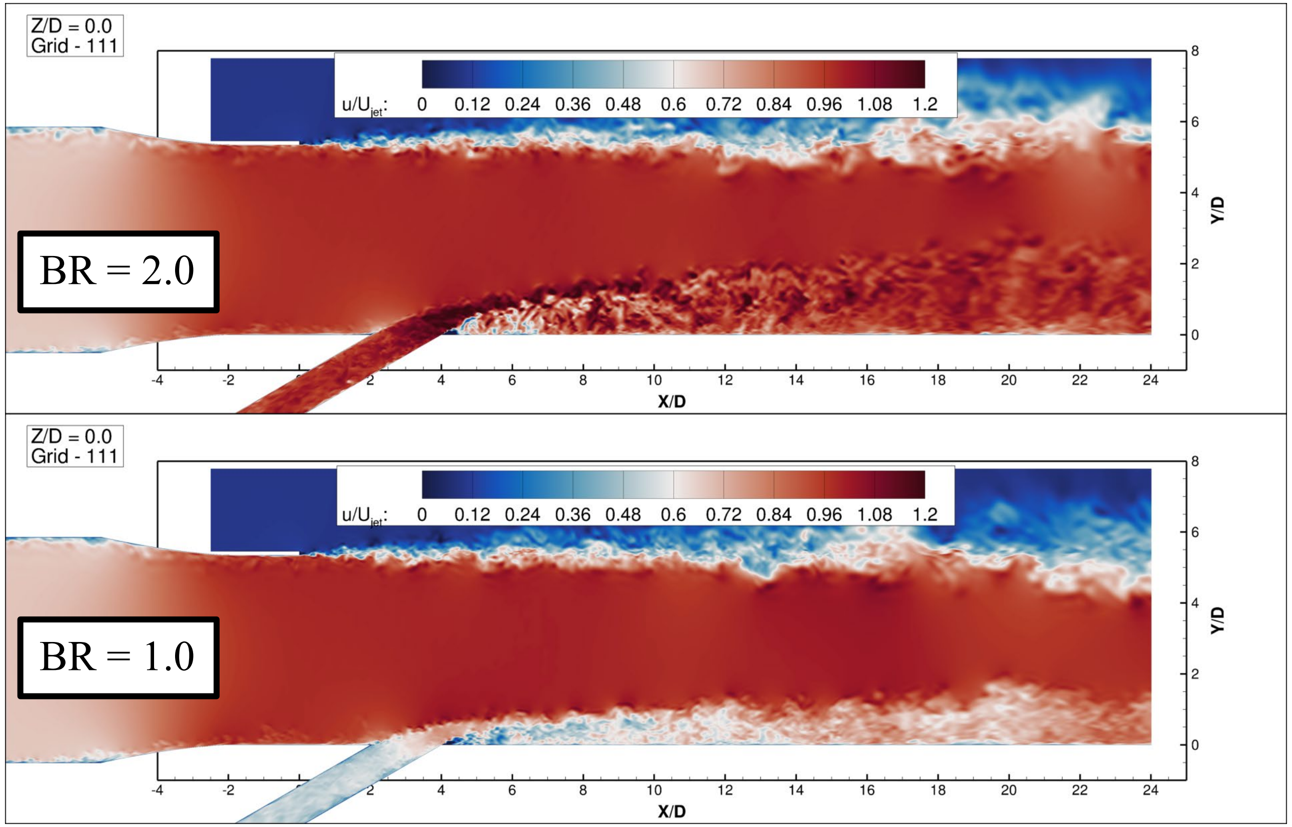


Shop air

- Thermocouples embedded in plate surface
- Static pressure measured in cooling flow tube
- Mass flow meter measured injector flow
- Nozzle P_t, T_t, air & fuel flow rates measured upstream
- 12” long, 1” thick SS deck
- Shop air used for cooling flow
- $BR = (\rho u)_{inj} / (\rho u)_{jet}$

THX Phase 3 LES using the FDL3DI (USAF AFRL) Code

Animation of U-velocity for BR=2.0 at Z/D = 0, and X/D = [3,5,7,9]



THX3 LES cases run with FDL3DI v2

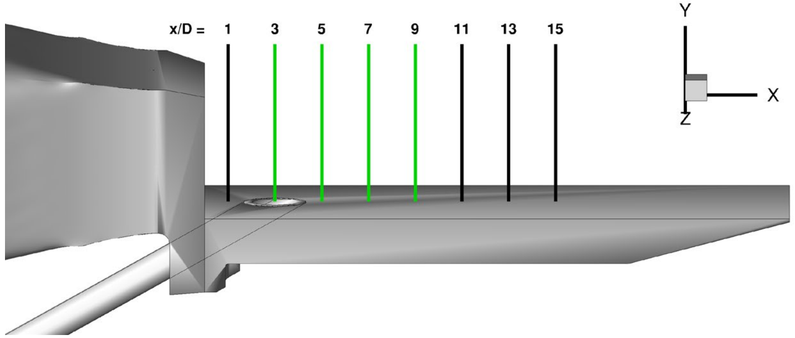
Blowing Ratio	Grid	Points/DOF	# Cores	dt (tau)	Time Scheme	Tau averaged	x+	z+
2.0	111	356,000,000	8,000	2.0e-4	Beam-Warming	232	30	18.75
2.0	010	1,520,000,000	16,000	2.0e-4	Beam-Warming	142	15	9.4
1.0	111	356,000,000	8,000	2.0e-4	Beam-Warming	253	30	18.75

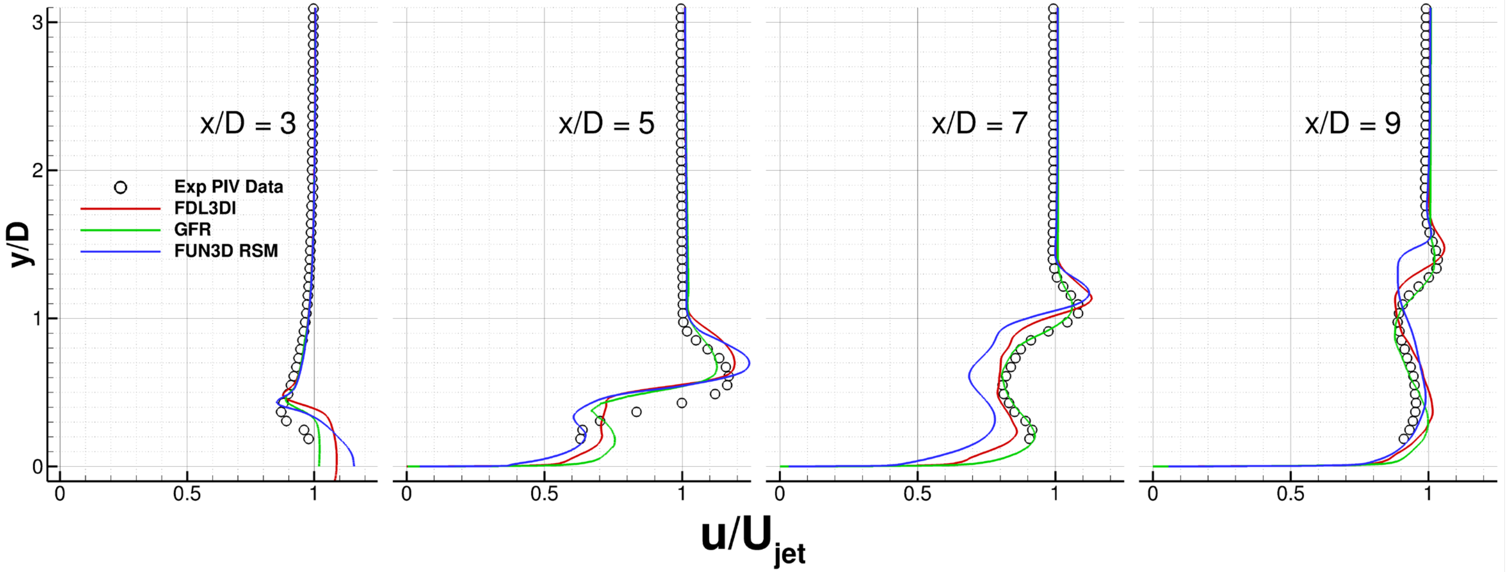
Numerical Results - Velocities

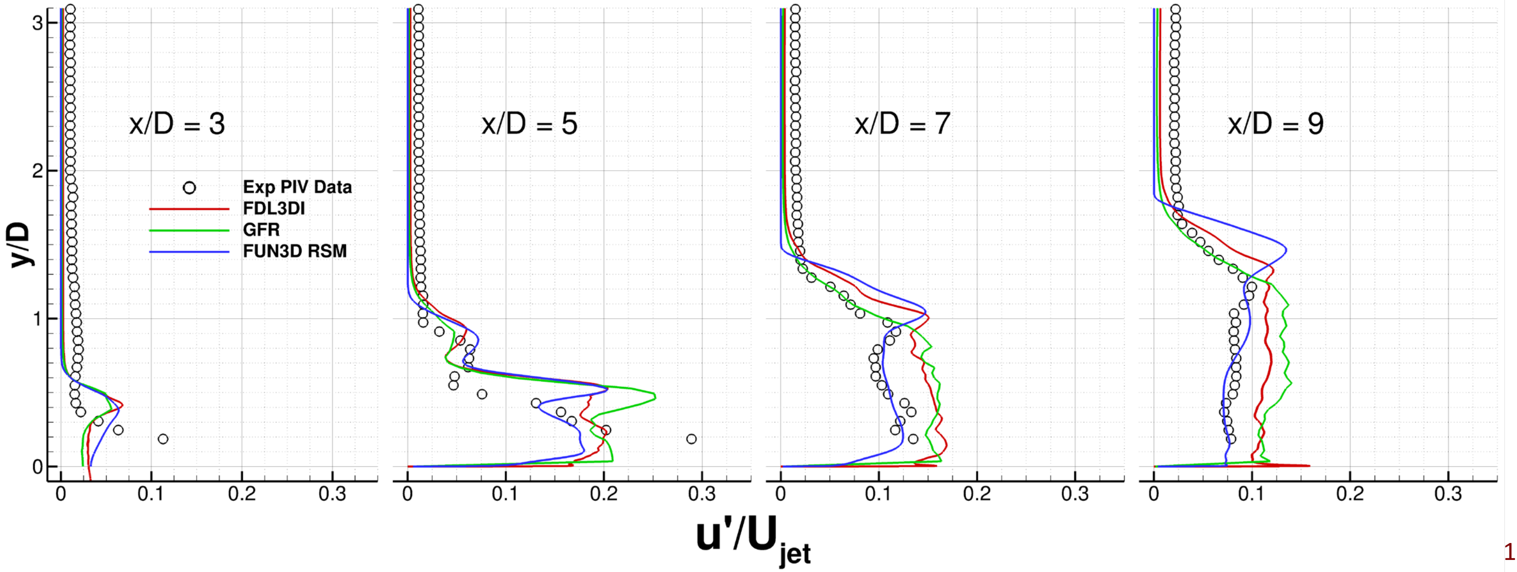
Set Point 23, Blowing Ratio = 2.0

Streamwise Mean and RMS (turbulent) Velocities

- Both LES approaches agree very well with each other and experiment for u.
- RSM enables good prediction of u’ near the wall (better than **eddy viscosity** RANS).
- RSM over penetrates downstream.





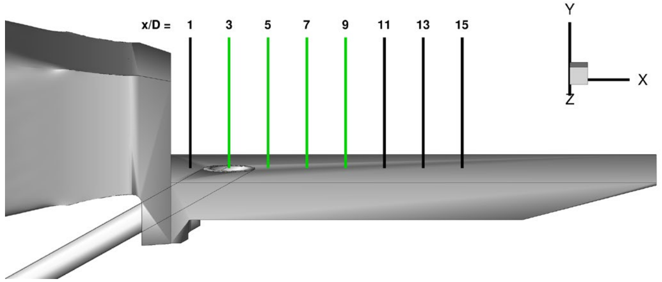


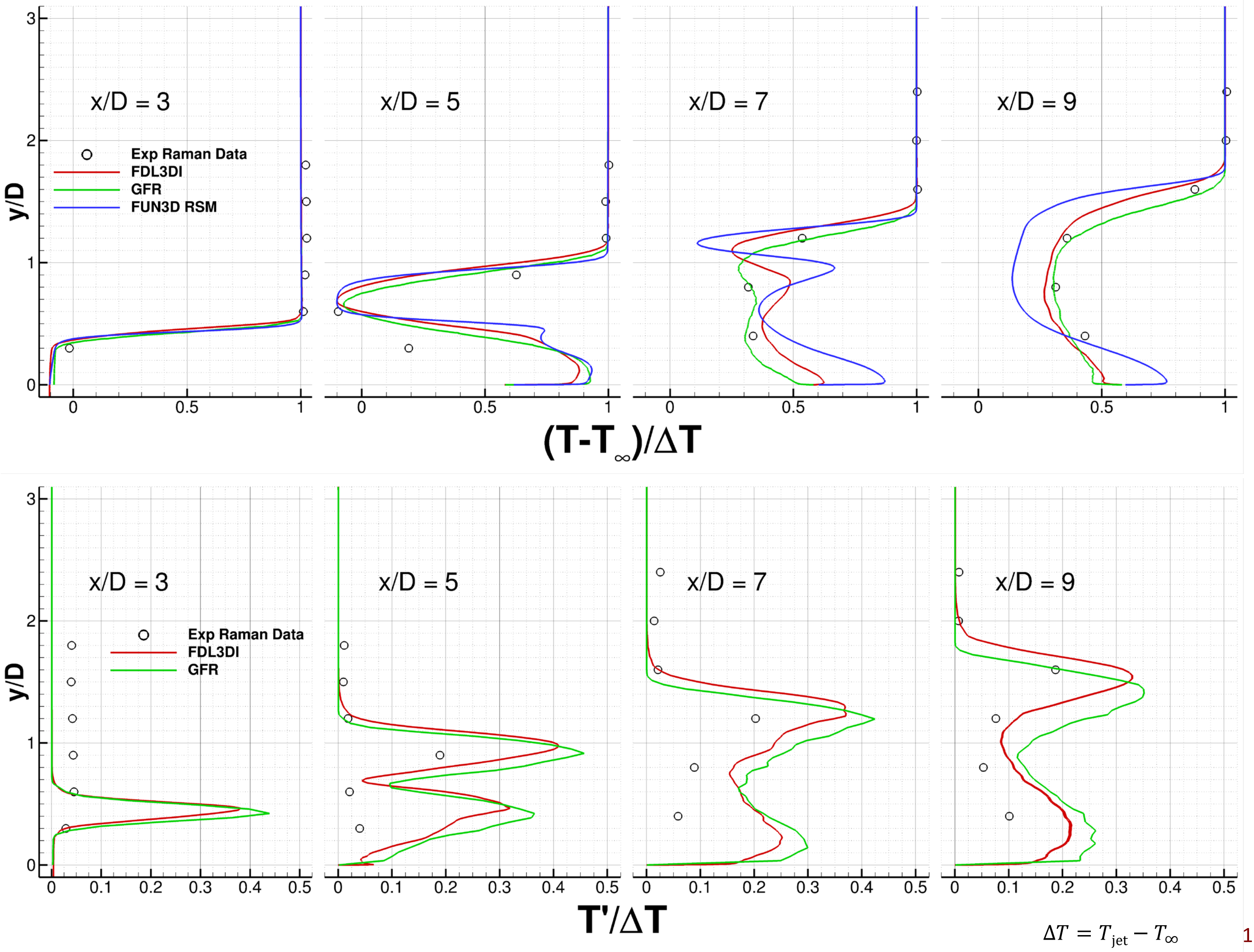
Numerical Results - Temperatures

Set Point 23, Blowing Ratio = 2.0

Mean and RMS (turbulent) Temperatures

- **Standard** RANS methods do not provide T’
- LES approaches follow similar trends
- LES predicts higher T’, but similar trends
  - This has also been seen in previous results comparing LES of heated jets to Raman T’ (THX II)

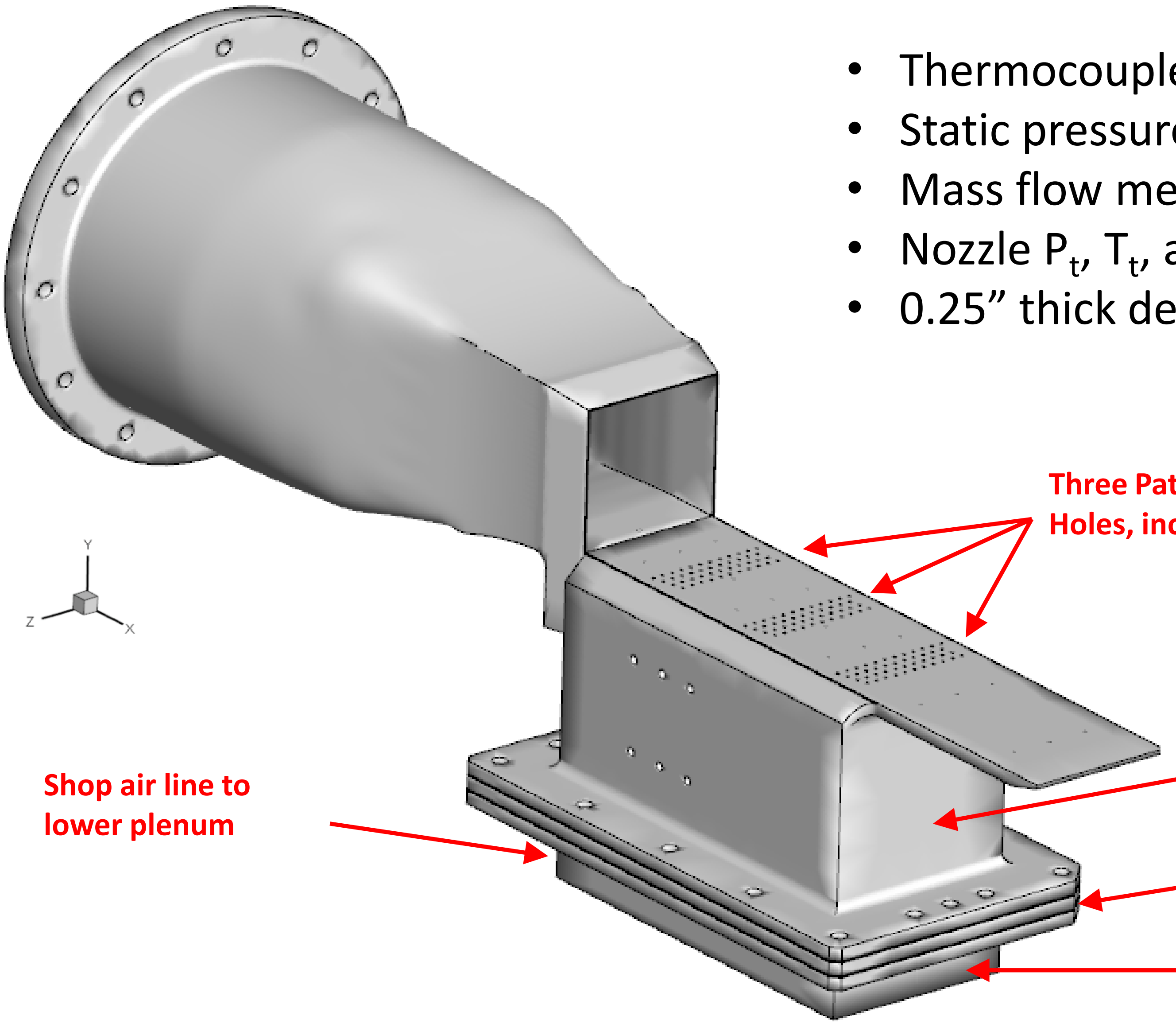




$\Delta T = T_{jet} - T_\infty$

THX Phase IV – Porous Plate Geometry

- Thermocouples embedded in plate surface
- Static pressure measured in top and bottom plenums
- Mass flow meter measured injector flow
- Nozzle P_t, T_t, air & fuel flow rates measured upstream
- 0.25" thick deck



Three Patches of Forty-Five 0.04" Cooling Holes, inclined at 30°

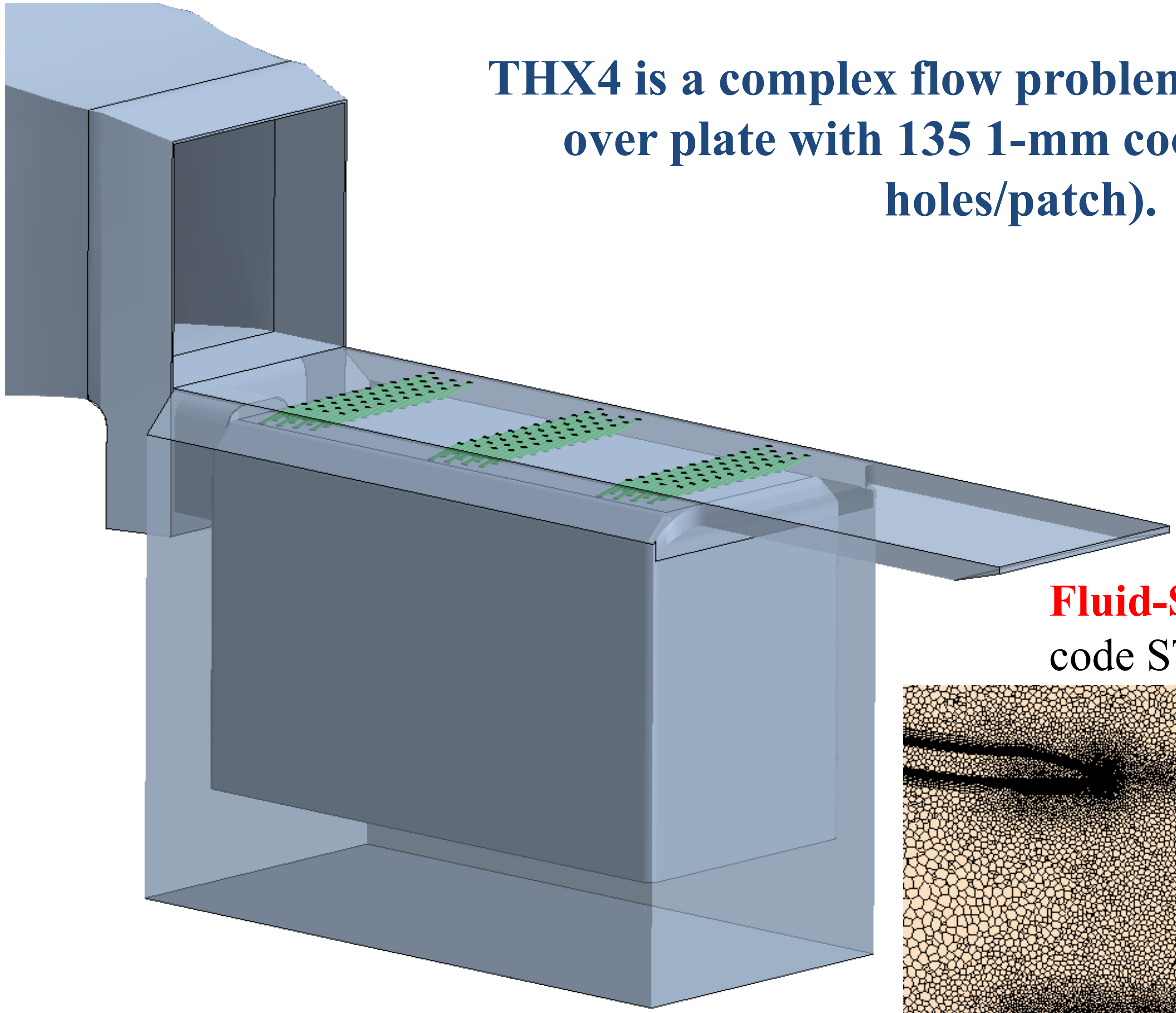
Top Plenum

Choke Plate

Shop air line to lower plenum

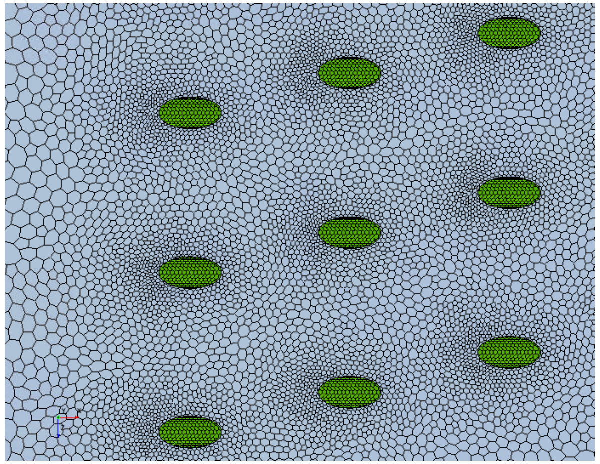
Bottom Plenum

THX4 Model Geometry & Flow Problem Setup

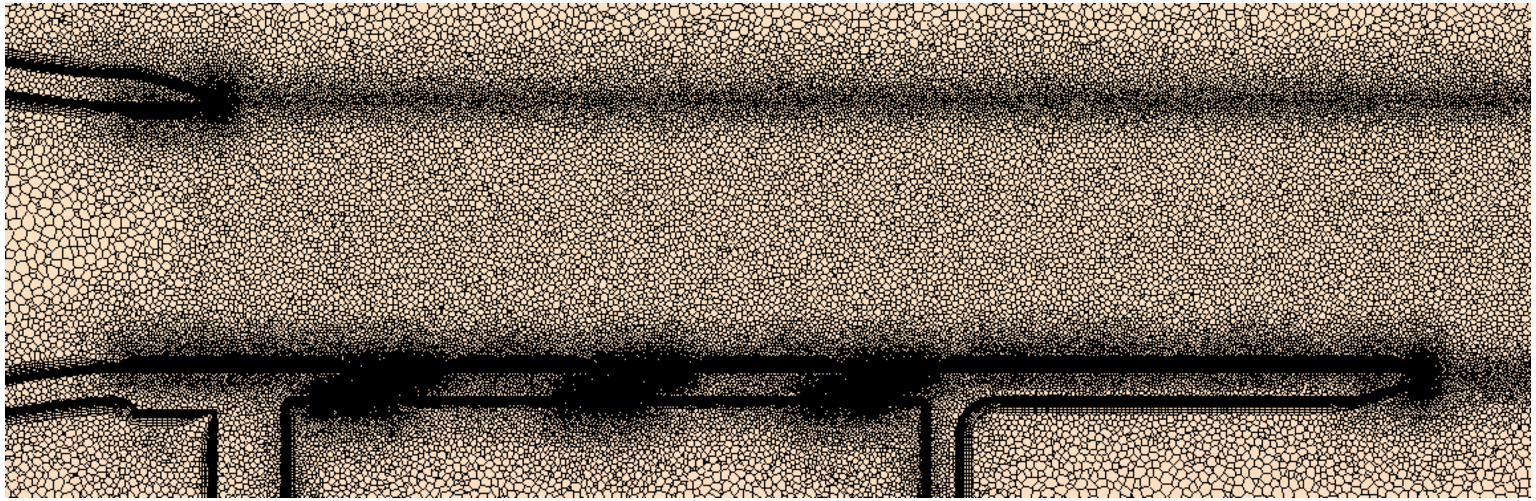


THX4 is a complex flow problem: hot jet exhaust over plate with 135 1-mm cooling holes (45 holes/patch).

Looking down on plate:



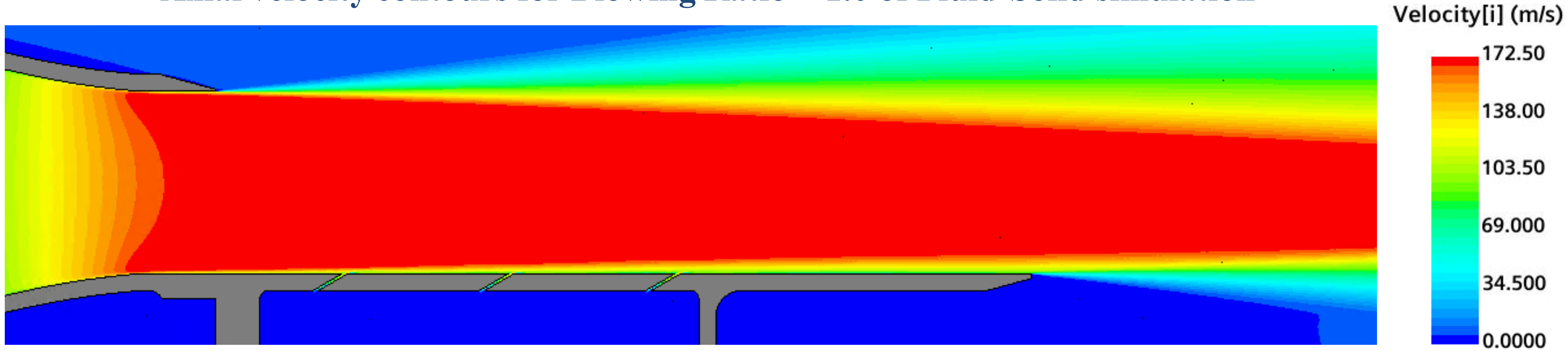
Fluid-Solid conformal polyhedral grid created in commercial CFD code STAR-CCM+ to solve the conjugate heat transfer (CHT)



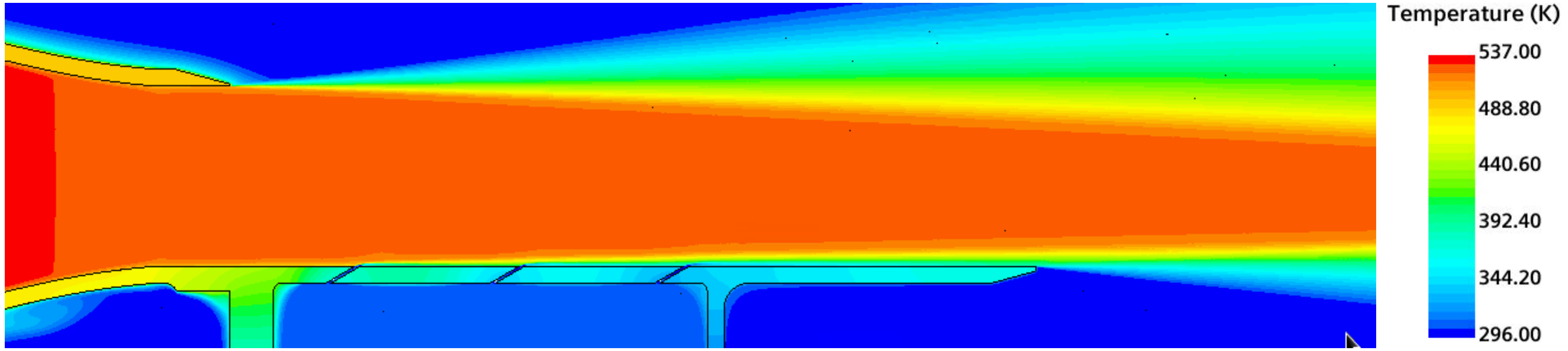
THX4: RANS Cases

- RANS SST simulation of **complete** Phase 4 configuration
- Modeling Approaches:
  1. Fluid-only (adiabatic wall)
  2. Fluid-Solid (CFD-CHT)
- CFD-CHT not a "typical" standard practice – complexity and cost.
- CFD-CHT examined here – due to potential to improve prediction of both (1) fluid flow and (2) temperatures of structure.

Axial velocity contours for Blowing Ratio = 1.0 of Fluid-Solid simulation

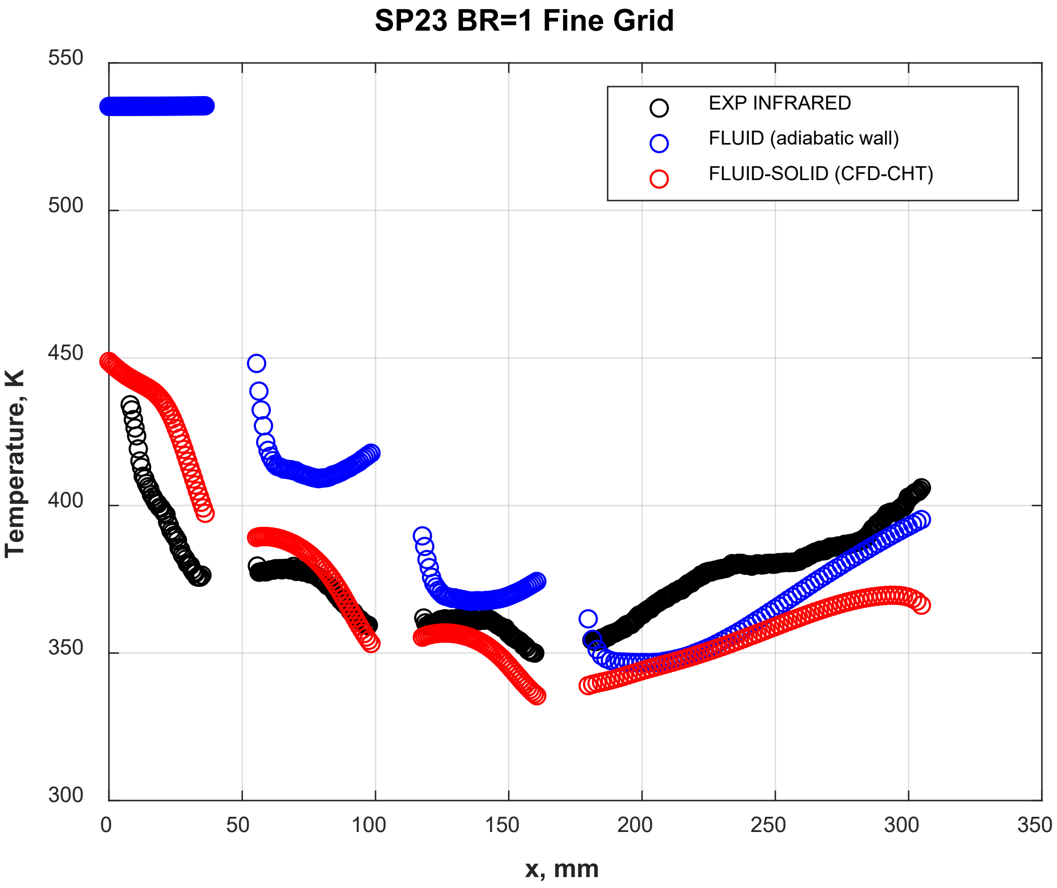


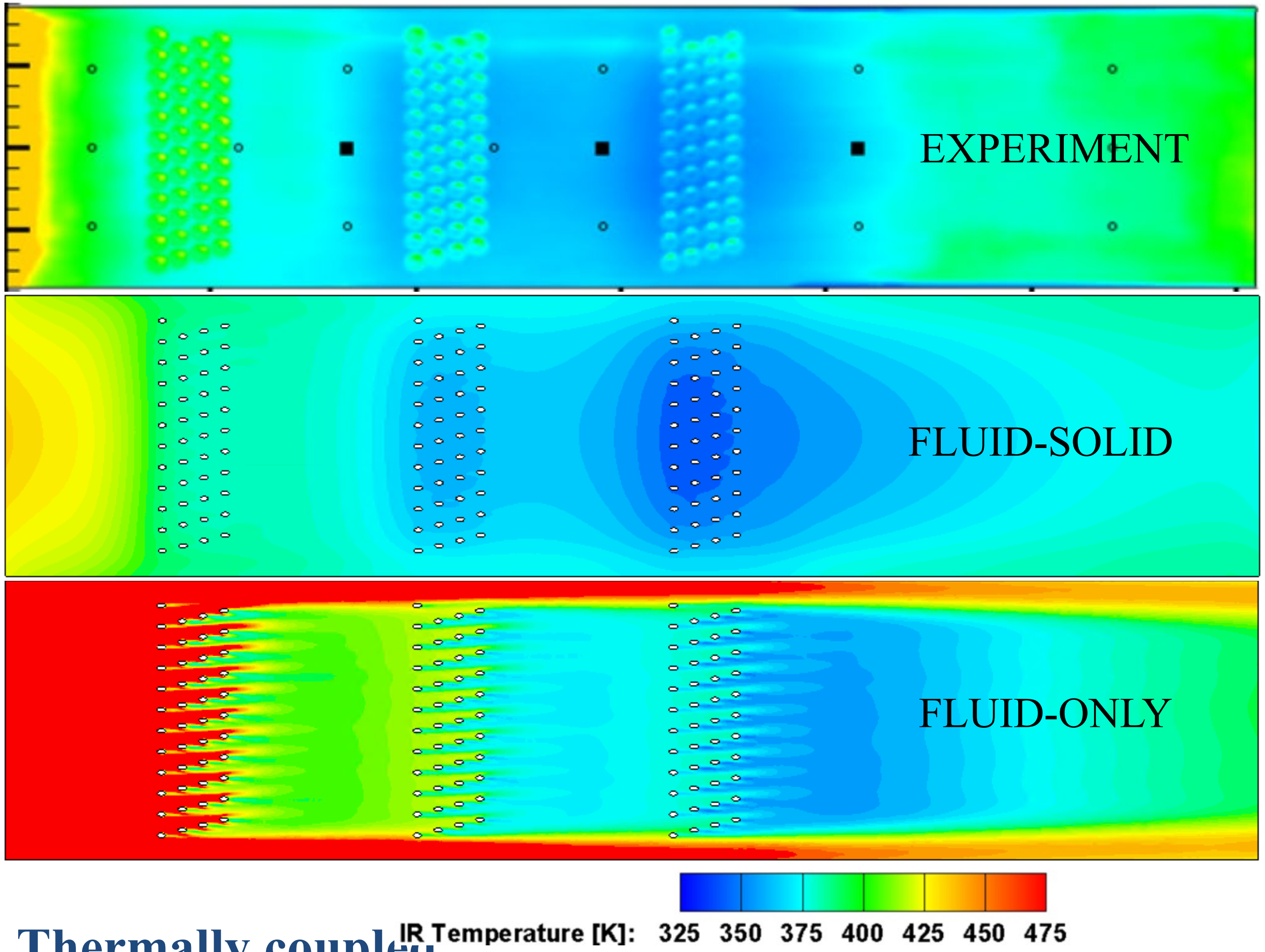
Temperature contours for Blowing Ratio = 1.0 of Fluid-Solid simulation



Grid	Fluid-only # cells	Fluid-Solid # cells	Blowing Ratio	# Cores	Numerical Scheme
Coarse	26.3 million	29.9 million	0.0, 1.0, 2.0	4,200	2nd order upwind
Medium	70.3 million	78.9 million	1.0	4,200	2nd order upwind
Fine	143.3 million	143.3 million	1.0	5,600	2nd order upwind
XFine	N/A	321.4 million	1.0	5,600	2nd order upwind

THX4 RANS Centerline Surface Temperature



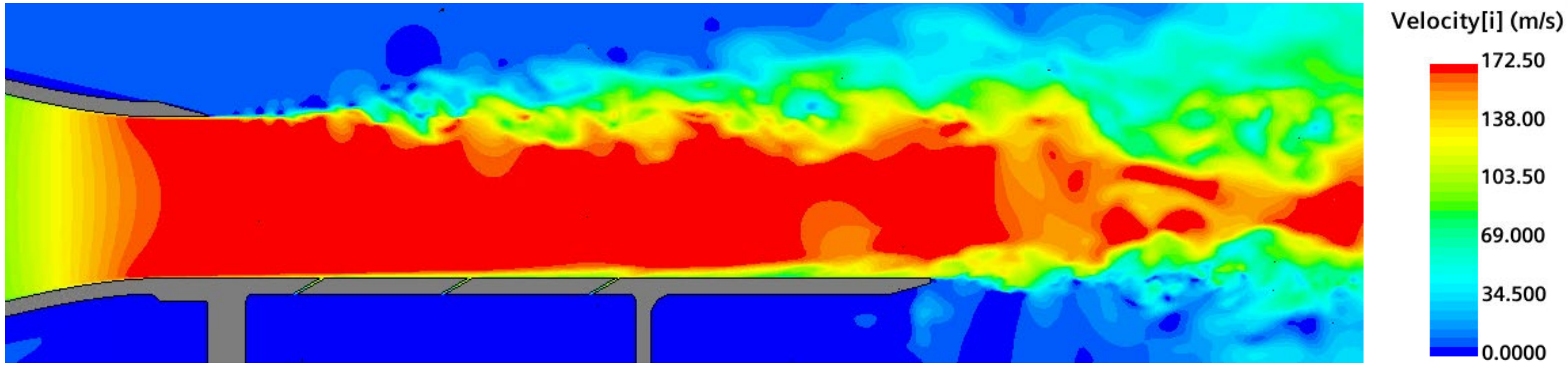


LESSON LEARNED / BEST PRACTICE: Thermally coupled CFD-CHT significantly improves surface temperature prediction.

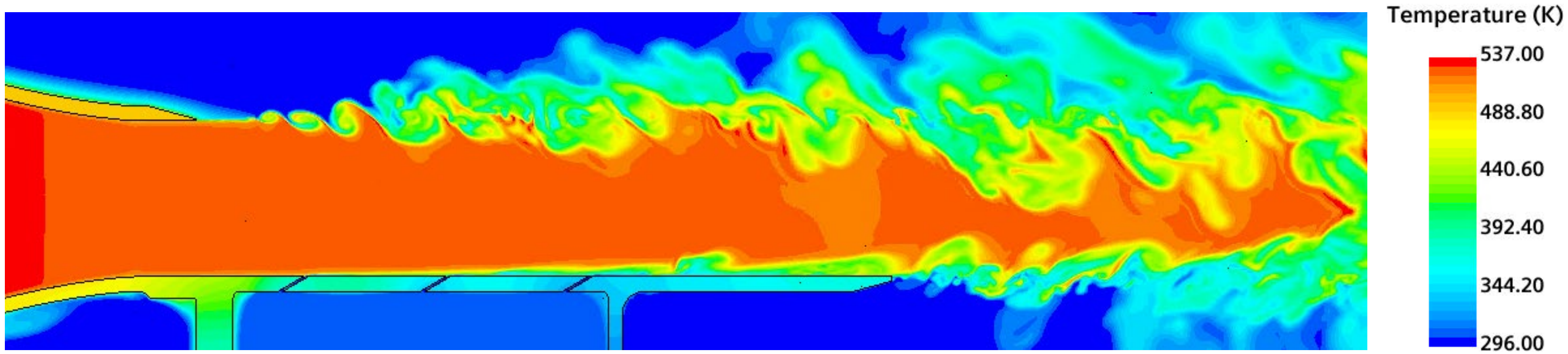
THX4: SRS (hybrid RANS-LES) Cases

- Per THX3 simulations (**20B-50B points required for wall resolved LES)** –not possible in this work.
- Scale-Resolved Simulation (SRS) of complete Phase 4 configuration completed using hybrid RANS-LES; 320 M points.
- Delayed Detached Eddy Simulation (DDES) technique successfully used within CFD-CHT model framework

Instantaneous axial velocity contours for Blowing Ratio = 1.0 of Fluid-Solid simulation



Instantaneous temperature contours for Blowing Ratio = 1.0 of Fluid-Solid simulation



Grid	Fluid-Solid # cells	Blowing Ratio	# Cores	dt	Total Tau Averaged*	Numerical Scheme
Fine	143.3 million	1.0	5,600	1E-6 s	41	Hybrid BCD
XFine	321.4 million	1.0	5,600	1E-6 s	42	Hybrid BCD

$$*\tau = \frac{U_{jet}}{L_{plate}} t$$

THX 5 – Supersonic Jet Experiments

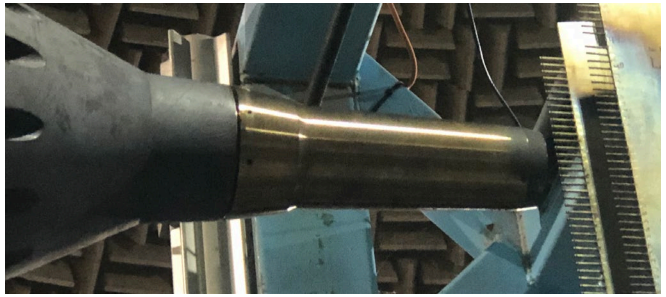
BACKGROUND AND OBJECTIVES:

- Test program conducted in GRC Aeroacoustic Propulsion Laboratory (AAPL).
- Objective was to create a comprehensive supersonic jet database using non-intrusive measurements.
- Test Matrix: Mach 1.4, Mach 1.65, and Mach 2.0 nozzles were investigated at temperatures from unheated up to $T_t = 1700^o$ R.
- Mach 2.0 test points replicated test points from experiments of Seiner et al (1992).

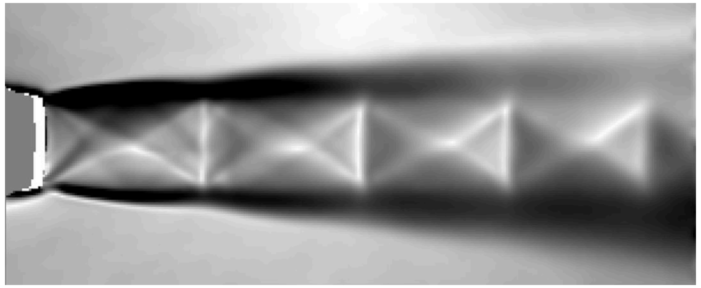
TEST PROGRAM RESULTS:

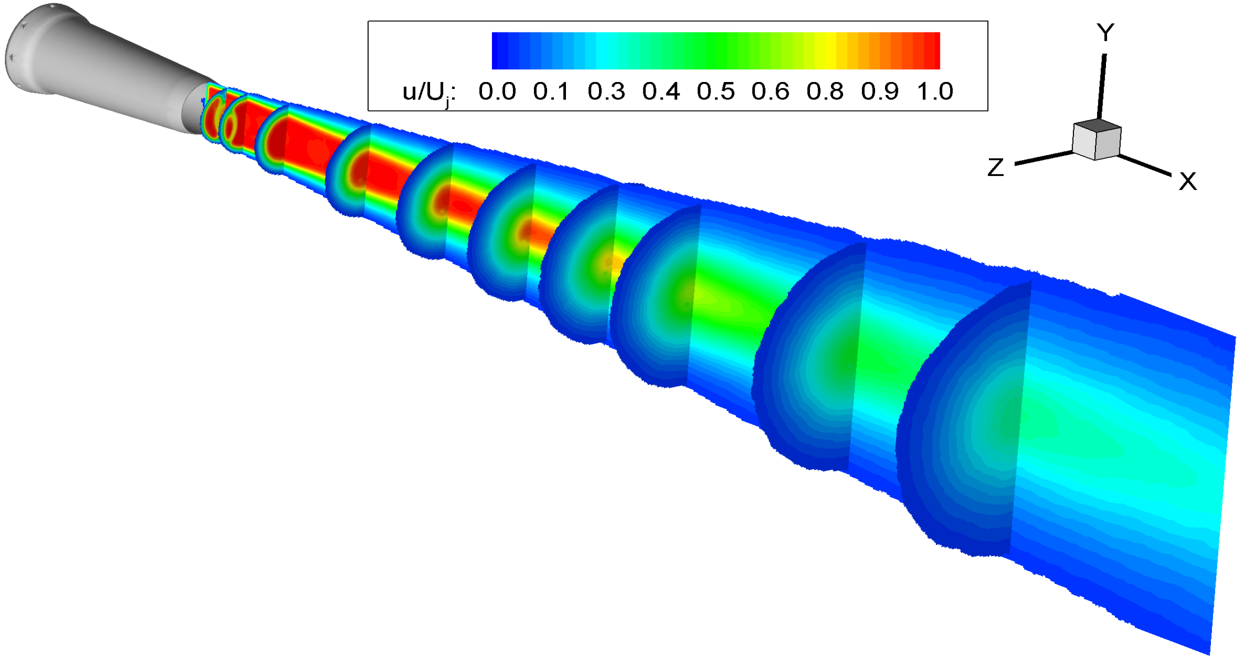
- Background-oriented Schlieren (BOS) and legacy probe measurements were obtained first.
- Streamwise and Cross-stream Particle Image Velocimetry (PIV) used to obtained velocities and turbulent stresses.
- Rotationally Resolved Raman Spectroscopy used to measure mean and rms velocities.
- Papers and data are available on results.
- AIAA Propulsion Aerodynamics Workshop Case for 2023.

Mach 2.0 nozzle installed in GRC AAPL



BOS image showing shock-cells at off-design test point





RCA Research Grants in Methods, Modeling/Simulations and Validation Experiments

NASA Research Announcements (NRAs) provide a mechanism to collaborate with academia and industry, a recommendation of the CFD Vision 2030 Study

Physical Modeling & Simulations:

- Improving the accuracy and efficiency of scale resolving simulations for favorable and adverse pressure gradient flows; U Colorado (Kenneth Jansen)
- Adaptivity in wall-modeled large eddy simulations of complex three-dimensional flows; U Maryland (Johan Larsson).
- Assessment of wall-modeled LES in nonequilibrium flows with emphasis on grid independency; U Pennsylvania (George Park)
- Scale-resolving turbulent simulations through adaptive high-order discretizations and data-enabled model refinements; U Michigan (Krzysztof Fidkowski)
- Wall-Modeled Large-Eddy Simulation of an Aircraft with Emphasis on High-Lift Configurations; Stanford (Parviz Moin)

HPC Tools & Methods:

- Scalable hierarchical CFD solvers for future exascale architectures; Stanford (Juan Alonso)
- Efficient and robust CFD solvers for exascale architectures; U Wyoming (Dimitri Mavriplis)
- A stochastic framework for computation of sensitivities in chaotic flows; U Pittsburg (Hessam Babaee)

CFD Validation Experiments:

- Smooth wall separation over bumps: Benchmark experiments for CFD validation; VA Tech (Kevin Lowe)
- Turbulent separated flow over a three-dimensional tapered bump [Joint DoD/NASA sponsorship]; Boeing/U Notre Dame – Not an NRA

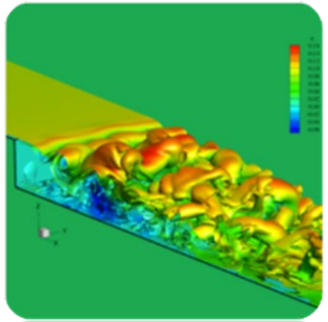
Certification by Analysis:

- Requirements for aircraft certification by analysis; Boeing (Vincent Lee) - Completed

- **Except for the Stanford NRA, all NRAs will end this year (some will go into no-cost extension)**
- **Consider new solicitation in 2022**

Summary

- **RCA research portfolio is aimed at making progress towards CFD Vison 2030 and meeting the challenge of predicting aircraft CL$_{max}$**
  - Physical modeling – especially turbulence
  - HPC
  - Numerical algorithms
  - Grid adaptation
  - CFD validation experiments
- **RCA's crosscutting computational tools will support:**
  - ARMD priorities of RLCC, UAM, EST and EAP
  - Other NASA missions (HEOMD, SMD, STMD)
  - Industrial competitiveness
- **Leverages collaborations with academia, industry, other government agencies and DoD, and the aerospace community at large**
  - High-lift prediction workshop
  - Aeroelastic prediction workshop
  - SBLI and Propulsion Aerodynamics Workshops
  - High order methods workshop
  - Transition modeling workshops



Development of Accurate and Efficient Computational Tools will Enable Aircraft Certification by Analysis and Design of New Aerospace Configurations