LTN/Inlets and Nozzles Branch Overview
NASA/GE- Methods Development Review

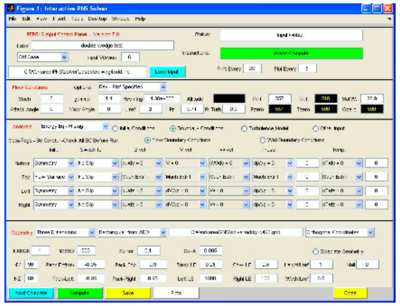
Mary Jo Long-Davis
Mary.J.Long-Davis@nasa.gov
06/01/17

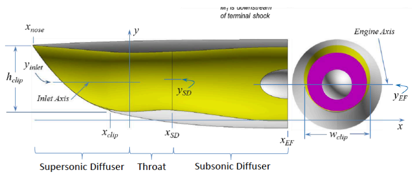
LTN/Inlets and Nozzles Branch

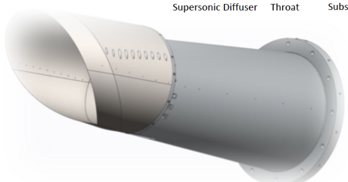
Overview

Computational and Experimental Research for Advanced Inlet and Nozzle Concepts

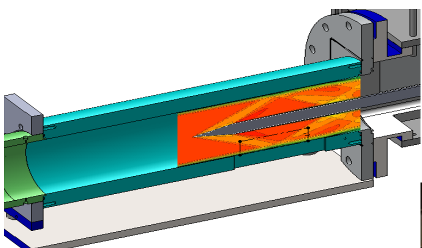
- •Design & Analysis Tools
- •Fundamental Flow Physics Investigations
- •Small & Large Scale Component Testing





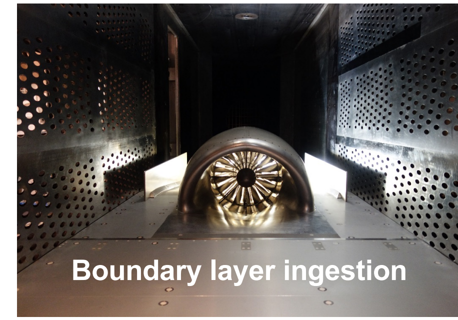


Inlet Design Tools

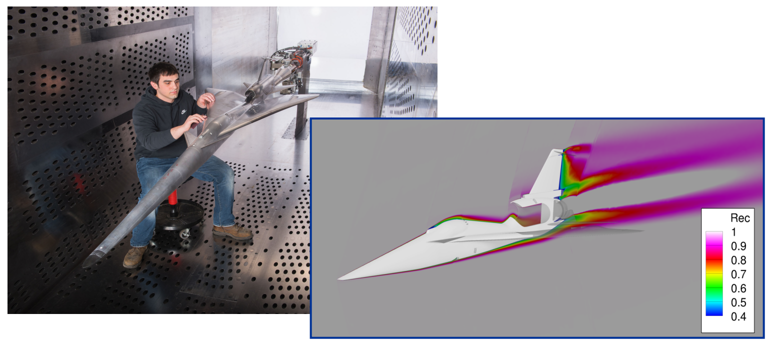


Axisymmetric Shock Wave/Boundary Layer Interaction

Turbulent CFD Validation Experiments



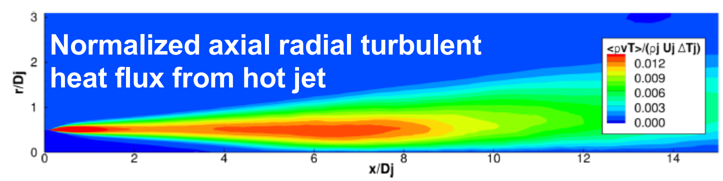
Boundary layer ingestion

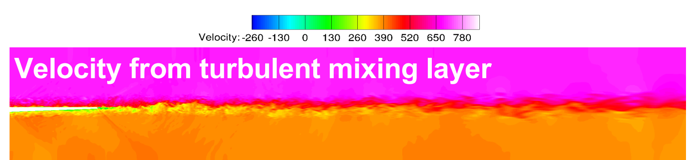


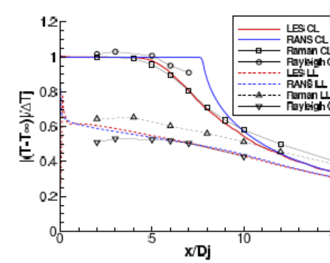
PAI effects on performance & sonic boom

Turbine-based combined cycles

Performance evaluation of advanced concepts



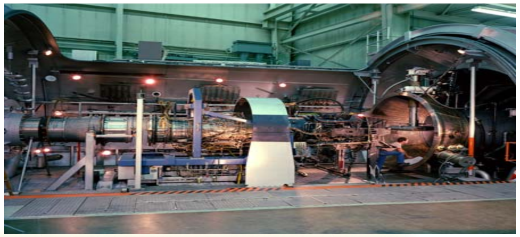
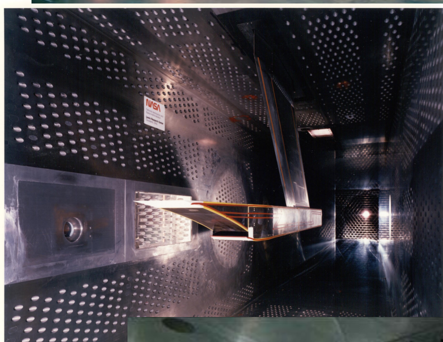
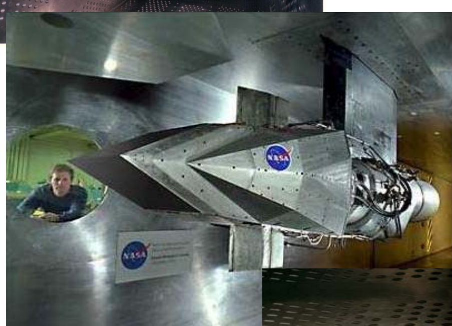




High-order large eddy simulation of shear layers, mixing layers and separated flows

Primary Experimental Facilities

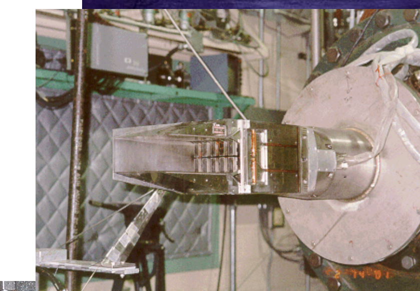
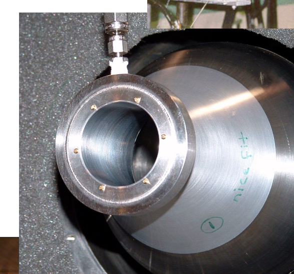
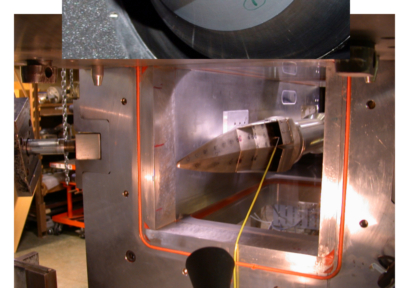
Inlets and Nozzles Branch

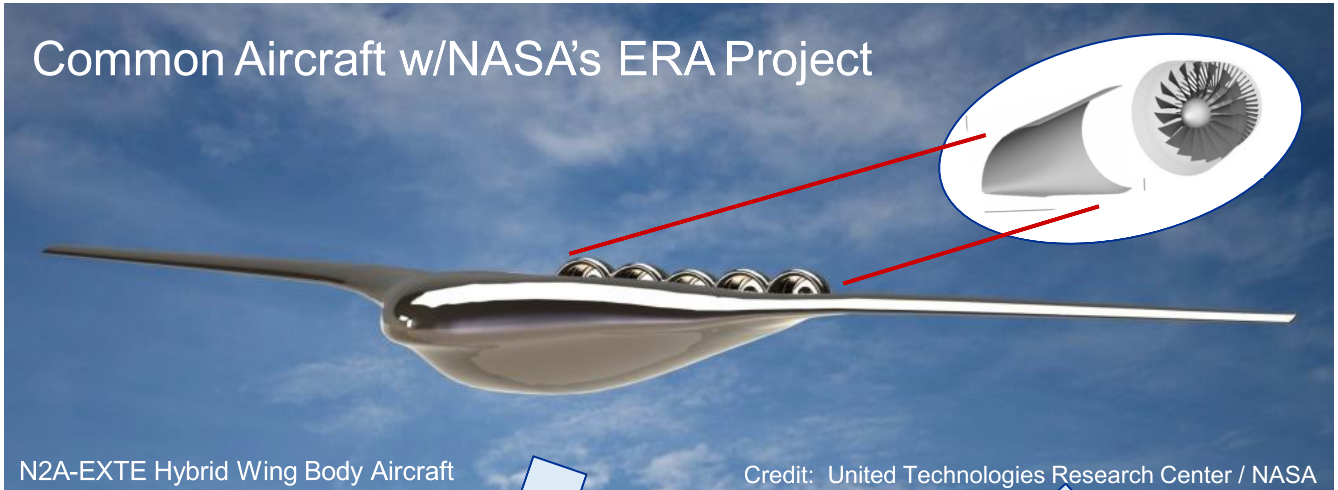

- CE-22: Nozzle Static Test Facility
- (CW-13, CW-17): Free Jet Facilities
- 1’x1’ Supersonic Wind Tunnel (SWT)
- 15cm x 15cm (and new 17-cm axisymmetric) SWT
- AeroAcoustic Propulsion Laboratory

- 10’x10’ Supersonic Wind Tunnel
- 9’x15’ Low Speed Wind Tunnel
- 8’x6’ Supersonic Wind Tunnel
- Propulsion Systems Laboratory

- NAS Supercomputing

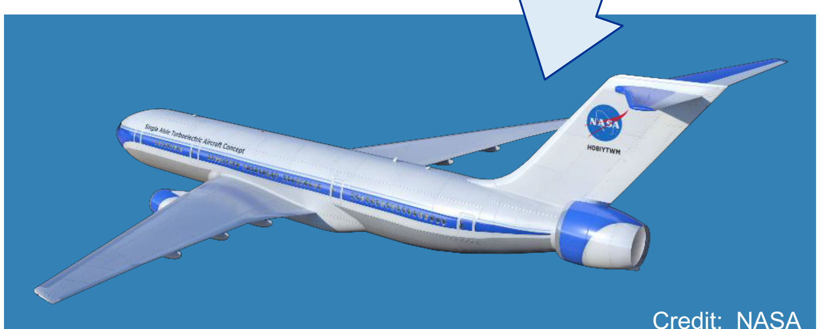
BLI Propulsor Technology Genesis and Applications



Credit: United Technologies Research Center / NASA

Credit: NASA / MIT / Aurora Flight Sciences

Credit: DZYNE Technologies / Brendan Kennelly



Credit: NASA

Future BLI Aircraft

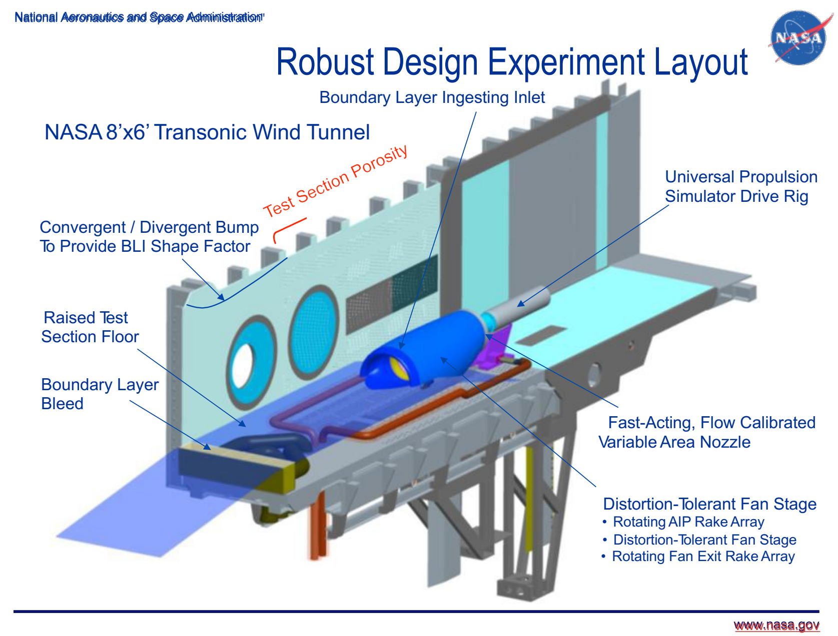
Boundary Layer Ingesting Inlet and Distortion Tolerant Fan (BLI²DTF)

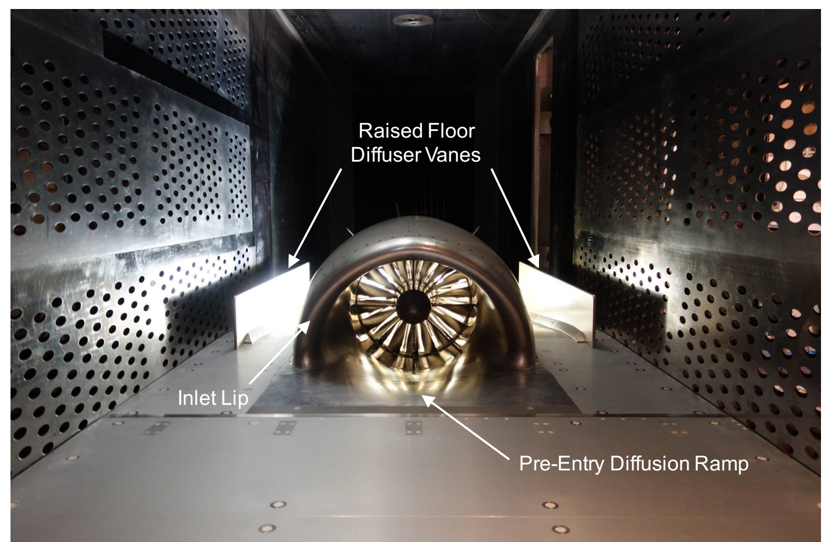
OBJECTIVES:

- Design an experiment which accurately simulates the integrated propulsion system Boundary Layer Ingesting Inlet and Distortion Tolerant Fan stage on a candidate Hybrid Wing Body/ Blended Wing Body vehicle.
- Validate that the performance and operability of the UTRC provided boundary layer ingesting inlet and distortion tolerant fan stage meets the design requirements & specified performance metrics.
- Utilize the experimental database obtained during this test activity for CFD validation and code development (i.e. both aerodynamics and aeromechanics/structural dynamics codes).
- Establish the aeromechanic behavior of the Distortion Tolerant Fan (DTF) and compare to pre-test predictions.

ACCOMPLISHMENTS:

- Very successful wind tunnel test –operability, research goals met
- Unique new BLI test bed infrastructure successfully demonstrated
- High quality data acquired –virtually no instrumentation failures
- Mach number range: 0.125 and 0.500 to 0.783
- Boundary layer thickness range: 3.65 to 5.00 inches
- Fan stage pressure ratio range: Windmill to 1.37
- Research results to be reported at the 2017 AIAA Propulsion and Energy Forum, Atlanta, GA and 2018 AIAA SciTech



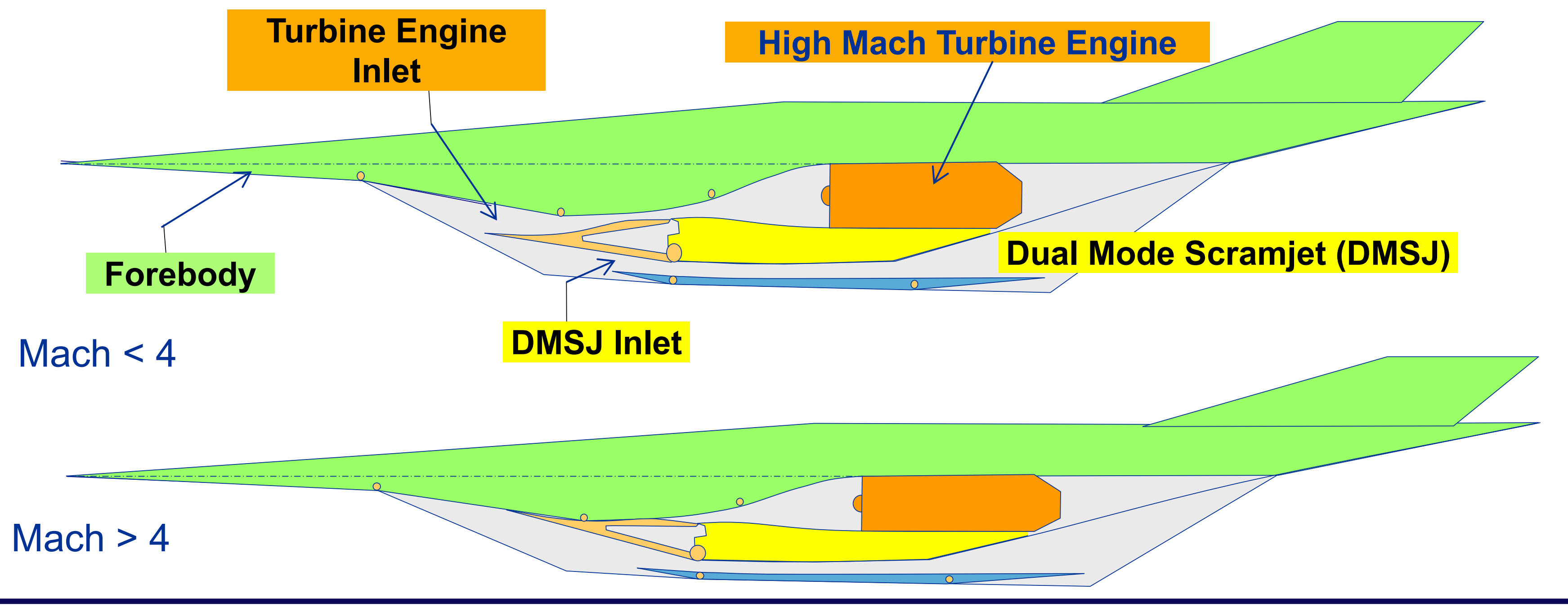


Credit: NASA / David Arend

POINT OF CONTACT:
LTN/Dave Arend, 216-433-2387, david.j.arend@nasa.gov

The TBCC Propulsion Concept

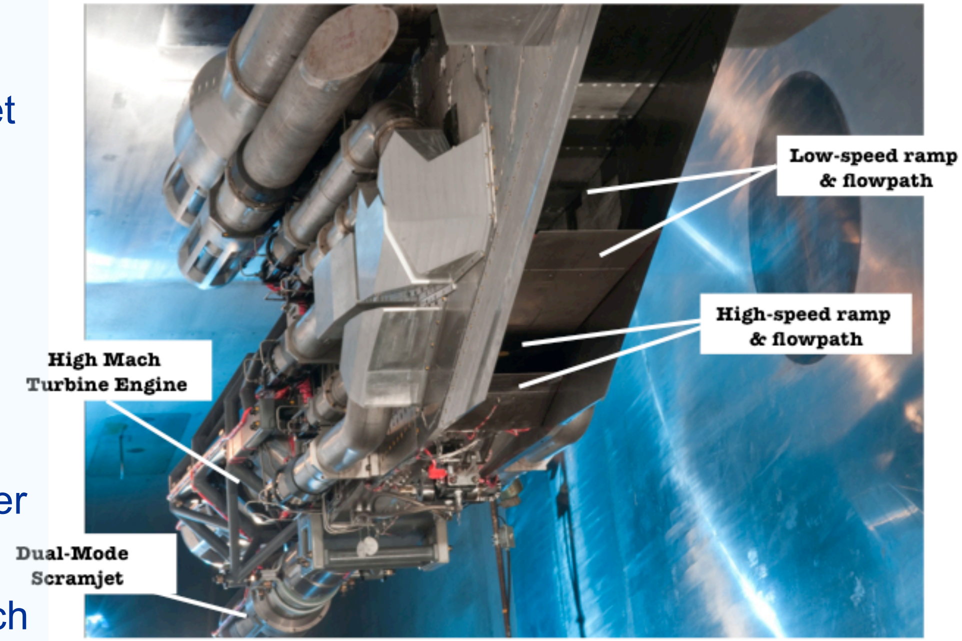
- The Turbine-Based Combined Cycle (TBCC) concept involves an over/under configuration with a split in the propulsion flowpath.
  - From take-off to Mach 4, a turbine engine provides propulsion.
  - Above Mach 4, a dual-mode supersonic combustion ramjet /scramjet (DMSJ) engine provides propulsion.
- Inlet Mode Transition is the procedure by which the flowpath split is achieved and controlled as the turbine engine shuts down and the dual-mode ramjet/scramjet engine takes over.



Turbine-Based Combined Cycle Engine Inlet and Controls research in the GRC 10’x10’ SWT

RESEARCH GOALS

- Proof of concept of over/under split flow inlet
  - Develop performance & operability database for both the turbine and scramjet inlets
  - Demonstrate mode transition and control
- Validate CFD predictions for both flowpaths
- Develop realistic distortion characteristics throughout the mode transition Mach number range
- Testbed for mode transition controls research & for integrated inlet/engine propulsion systems



TEST APPROACH - 4 Phases

1. Inlet performance and operability characterization
2. System Identification of inlet dynamics for controls
3. Demonstrate Control strategies for smooth & stable mode transition without inlet unstart
4. Add engines/ nozzle for integrated system test

Completed

IMPACT: Critical stepping stone to towards practical TBCC-powered aircraft

Axisymmetric Shock-Wave/Boundary Layer Interaction Experiment

Objectives:

- Obtain mean and turbulence quantities through a M=2.5 SWBLI of sufficient quantity and quality to be considered as a CFD validation dataset. Initial efforts will focus on a Mach 2.5 2-D (in the mean) interaction with follow-on efforts investigating 3-D interactions. Both attached and separated interactions will be considered.

Technical Challenge:

- Very few Shock Wave/Boundary-Layer Interaction (SWBLI) experiments reported in the open literature meet the rigorous criteria required to be considered as a CFD validation dataset. This is particularly true for experiments with detailed turbulence measurements.

Approach/Key Milestones:

✓ Completed flow surveys using conventional pressure probes

✓ Completed testing using dynamic surface shear stress film (S3F) and fast response Pressure Sensitive Paint (PSP) in collaboration with Innovative Scientific Solutions, Incorporated (ISSI)

✓ Completed flow surveys using normal hot-wire anemometry

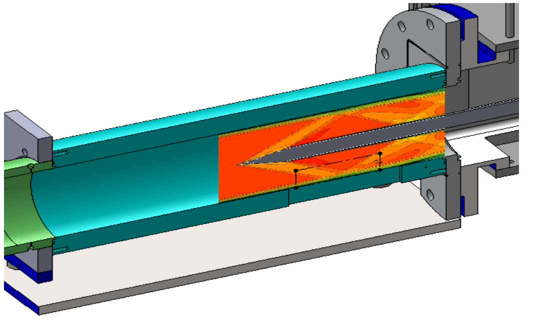
○ Complete flow surveys using slant hot-wire anemometry

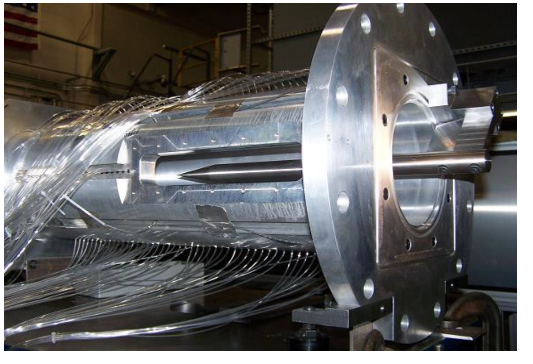
○ Complete flow surveys using particle image velocimetry (PIV)

POINT OF CONTACT:
LTN/Dave Davis, david.o.davis@nasa.gov

Deliverables/Products:

- Complete characterization of an axisymmetric shock-wave/boundary-layer interaction
- High quality CFD validation dataset including experimental uncertainty quantification
- New axisymmetric supersonic tunnel facility, including PIV measurement system





17cm Axi-Supersonic Wind Tunnel (in ERB W6B cell)

Turbulent Heat Flux (THX) Validation Experiment

OBJECTIVES

- Perform validation experiments to collect turbulent flow measurements in the region of an injection hole and at several locations downstream.
- Implement advanced non-intrusive diagnostics technique for measuring temperature fluctuations in a plane with high spatial resolution and high temporal response rates
- Data will be used to validate CFD methods (RANS, LES, DNS), to develop boundary conditions for film cooling, to develop more accurate models for turbulent heat flux, and to improve

TECHNICAL CHALLENGES

- Accurate measurement of fluctuating 3-component velocities and temperature at high speed and high temperature close to a surface and/or over an injection hole
- Temporal and spatial resolution capabilities of measurement techniques

APPROACH/KEY MILESTONES: 3-Steps

✓ STEP 1: Perform a flat plate test examining single hole and multiple hole patterns using SW-6 facility. Low speed (up to Mach 0.2) room-temperature crossflow with heated film injection flow (ΔT up to 100 degs F).

✓ STEP 2: Collect fluctuating temperature data from a heated convergent jet (up to 1300 degs F) tested on the Small Hot Jet Acoustic Rig using a new advanced RAMAN spectroscopy capability

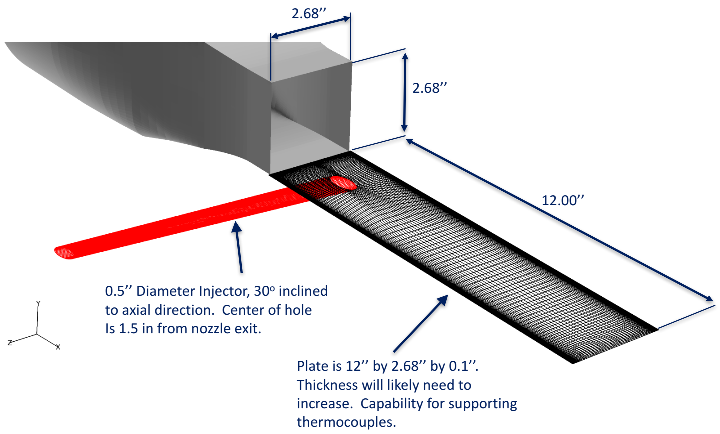
- STEP 3: Perform a flat plate test installed in a hot free-jet (either downstream of a nozzle in the SHJAR or the NATR) with ambient temperature film cooling flow injected through a single or multiple holes using a new advanced RAMAN spectroscopy capability.
- Complementary CFD simulations for each step at pre-test to help define the experiment and post-test to validate/improve codes and models

PRODUCTS

- High-quality flow field data including: Mean velocities and temperatures, kinetic energy, Reynolds stresses, surface temperature, dissipation rate, turbulent viscosity or length scales, and turbulent heat flux
- New advanced RAMAN spectroscopy to provide improved non-intrusive diagnostic capability at GRC for measuring mean and fluctuating temperatures in a plane at high spatial resolution

PARTNERS/CUSTOMERS: Boeing, P&W, AFRL

Step 3-Nozzle Geometry



POINTS OF CONTACT

CFD; Nick Georgiadis, 3-3958, georgiadis@nasa.gov
RAMAN/PIV: Mark Wernet, 3-3752, mark.p.wernet@nasa.gov

Tools and Methods

Inlets and Nozzles Branch

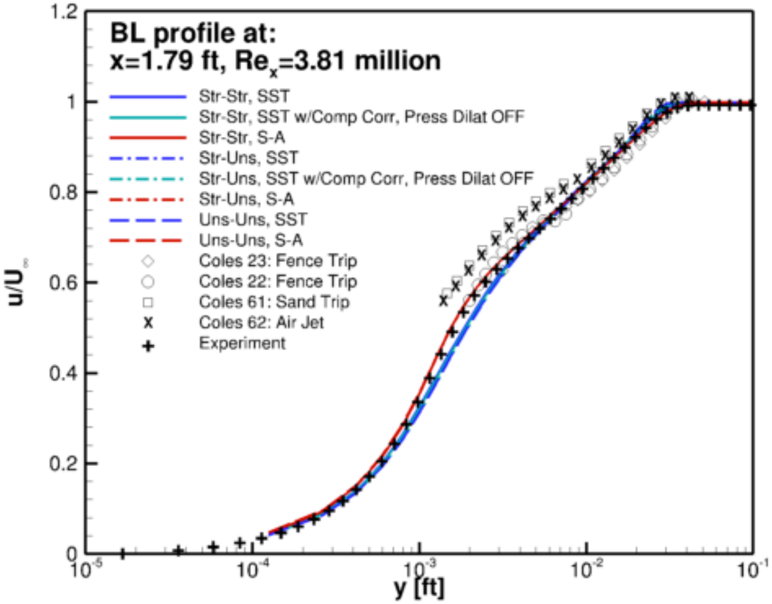
- **Method of Characteristics (MOC) design codes:** Several in-house computer programs for designing high performance supersonic inlets and nozzles.
- **Inlet and Nozzle Performance Analysis Code (IPAC & NPAC)** - predicts performance for a given geometric design, can be used to design preliminary inlet & nozzle systems and to make subsequent performance analyses.
- **SUPersonic Inlet design and analysis tool (SUPIN)-** uses analytical, numerical, and empirical methods to design the geometry and estimate the aerodynamic performance of supersonic inlets. Steps include:
  - establish the parametric design of the inlet
  - efficiently model the geometry and
  - generate the grid for CFD analysis with design changes to those parameters.
- **RANS Codes:**
  - **Wind-US** - a general purpose, applied CFD tool. It solves the Navier-Stokes equations of fluid mechanics for steady state and unsteady flows, along with supporting equation governing turbulent and chemically reacting flows. NASA GRC personnel partner with the USAF AEDC and Boeing to develop, support, and validate the code.
  - **FUN3D** – a general purpose, applied CFD. It solves the Navier-Stokes equations of fluid mechanics for steady state and unsteady flows, along with supporting equations governing turbulent flows. Developed and maintained by NASA Langley.
- **LES and Hybrid RANS/LES (Wave-Resolving LES (WRLES) & Glenn Flux Reconstruction (GFR codes))** – high-order codes for propulsion flow modeling.

What is Wind-US and NPARC?

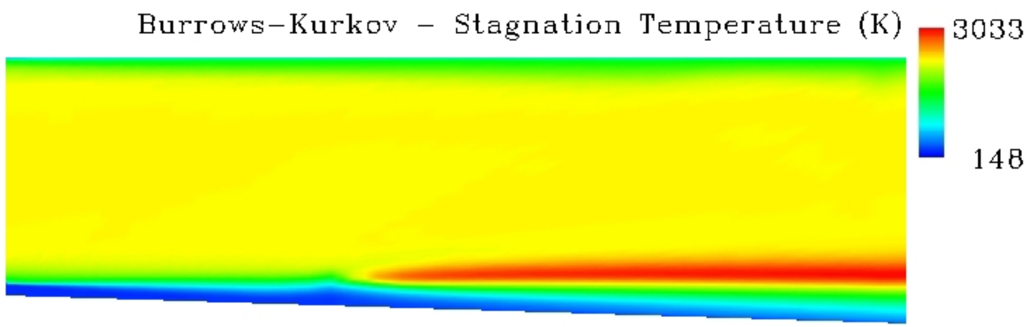
- The NPARC (National Project for Applications-Oriented Research in CFD) Alliance is a partnership of NASA GRC and USAF AEDC, with significant participation of the Boeing Company, to provide the state-of-the-art in proven CFD technology.
- The NPARC Alliance has been in existence for 20 years; significant evolution from PARC code in early 1990s to Wind-US Version 4.0.
- The primary product of the NPARC Alliance is the Wind-US (US=unstructured) code with accompanying utilities and extensive on-line documentation. It is made available to all U.S. organizations approved to receive export-controlled information.
- Wind-US is a general-purpose, multi-zone, compressible Navier-Stokes CFD code. It is designed to provide the state-of-the-art in proven CFD code technology.
- It is the workhorse flow solver for NASA GRC Inlets and Nozzles branch.
- Wind-US code distinguishing features: very detailed documentation and validation suite, broad range of physical models.

Validation of Combustion, Structured & Unstructured Solvers

Mach 4.5 Boundary Layer:



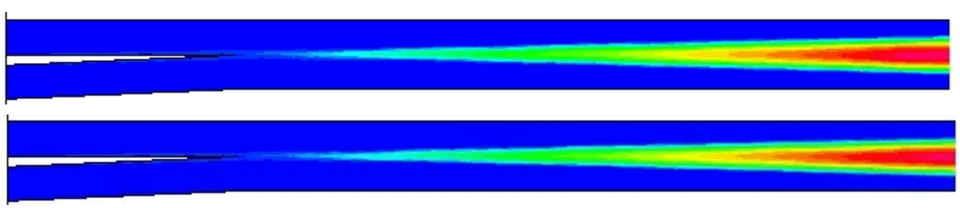
Burrows-Kurkov Reacting Mixing Layer:

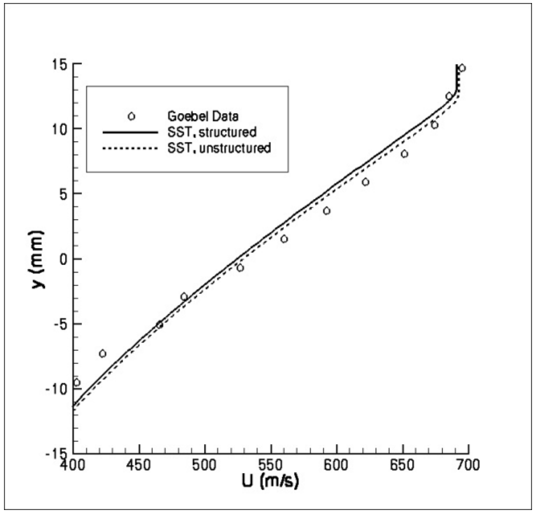


Dutton-Goebel Supersonic Mixing Layer:

Struct.

Un.-St.





Wind-US Validation Website

NATIONAL AERONAUTICS AND SPACE ADMINISTRATION

+ NASA Portal

Validation Web Site Search: + GO

NPARC Alliance CFD Verification and Validation Web Site

[Home] **Archive** [Resources] [History] [Contacts]

NPARC Alliance Verification and Validation Archive

Last Updated: Thursday, 22-Jan-2015 13:02:29 EST

The **archive** consists of ***cases***. Each case represents a geometric configuration and flow condition (i.e. ONERA M6 wing, MADIC 3D nozzle). A case can be classified as either a verification or validation case depending on whether the comparison data for the case is exact (analytical) or experimental, respectively. Each case contains one or more ***studies***. Each study consists of one or more runs of the Wind-US code (or another CFD code). Throughout some of the cases and studies are detailed examples that provide a step-by-step description or tutorial on using WIND-US and associated tools.

Downloading Files from the Archive

The cases of the archive include:

Mach	Grid	Gas	Description
0.00	SG	IG	Sod's Shock Tube
0.05	SG	IG	Incompressible Driven Cavity
0.10	SG	IG	Blasius Incompressible Laminar Flat Plate
0.13	SG	IG	Driver-Seegmiller Incompressible Backward-Facing Step
0.15	SG	IG	Fraser Subsonic Conical Diffuser
0.20	SG	IG	Incompressible, Buice Axisymmetric Diffuser
0.20	OG	IG	NLR Airfoil with Flap
0.20	SG,UG	IG	Incompressible, Turbulent Flat Plate
0.20	SG	IG	Laminar Flow over a Circular Cylinder
0.20	SG	IG	Ejector Nozzle
0.21	SG	IG	Low-Subsonic S-Duct
0.30	SG	IG	Steady, Inviscid Flow in a Converging-Diverging Verfication (CDV) Nozzle
0.30	SG	IG	Subsonic Annular Duct
0.40	SG	IG	Square Jet Injection
0.46	SG,UG	IG	Sajben Transonic Converging-Diverging Diffuser
0.73	SG	IG	RAE 2822 Transonic Airfoil
0.80	SG	IG	MADIC 2D Axisymmetric CD Boattail Nozzle
0.80	SG	IG	MADIC 3D CD Boattail Nozzle
0.84	SG	IG	Transonic, ONERA M6 Wing
0.97	SG,UG	IG	Acoustic Reference Nozzle with Mach 0.97, Unheated Jet Flow
1.30	SG	IG	Normal shock at Mach 1.3
1.82	SG, UG	CH	Hydrogen-Air Combustion in a Channel
2.00	SG, UG	IG	Mach 2.0 Flow over a 15-Degree Wedge
2.00	SG, UG	IG	Seiner Nozzle with Mach 2.0, Heated Jet Flow
2.22	SG	IG	Supersonic Axisymmetric "submerged" Jet Flow
2.35	SG	IG	Conical shock on a 10 degree cone at Mach 2.35
2.44	SG	CH	Burrows and Kurkov Supersonic Mixing/Combustion
2.50	SG, UG	IG	Oblique shock on a 15 degree wedge at Mach 2.5
2.50	SG, UG	IG	Prandtl-Meyer 15 Degree Expansion Corner at Mach 2.5
4.50	SG,UG	IG	Mach 4.5 Flow over a Flat Plate
5.00	UG	IG	Mach 5.0 Shock Boundary Layer Interaction
7.00	SG	IG	Mach 7, Laminar 15-degree Ramp
15.00	SG	IG	Hypersonic Cylinder

Tags:

SG = structured grid
OG = overset grid
UG = unstructured grid
IG = ideal gas
CH = chemistry

Unstructured jet excerpt:

Table 3. Unstructured, 3-D grids for unstructured solver

Name	Azimuthal Width (deg)	Azimuthal Cells	Centerline Scheme	Total Cells
Str-Uns_90deg-v1	90	20	triangular-prisms	1,006,847
Str-Uns_90deg-v2	90	20	triangular-prisms	1,957,622

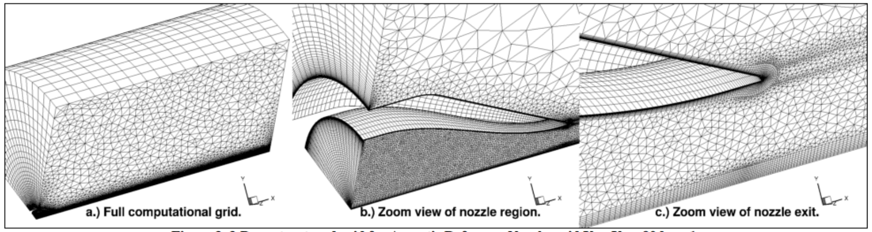


Figure 3. 3-D, unstructured grid for Acoustic Reference Nozzle, grid Uns-Uns_90deg-v1.

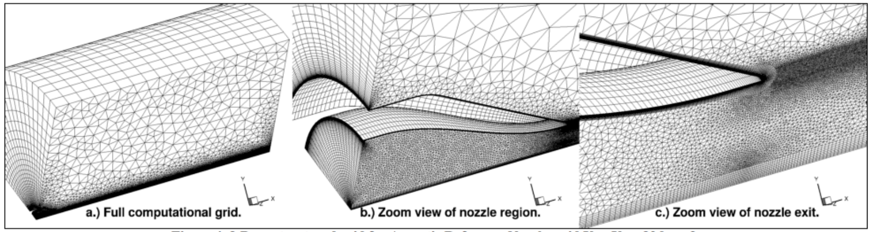


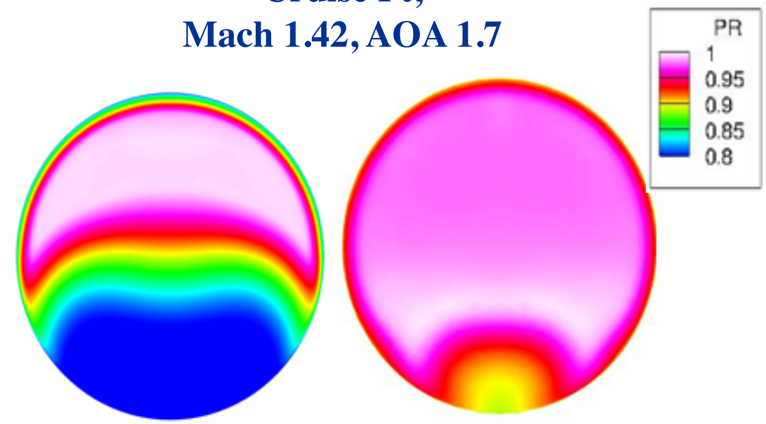
Figure 4. 3-D, unstructured grid for Acoustic Reference Nozzle, grid Uns-Uns_90deg-v2.

Propulsion Airframe Integration CFD

Potential need for additional validation data and CFD tool refinements

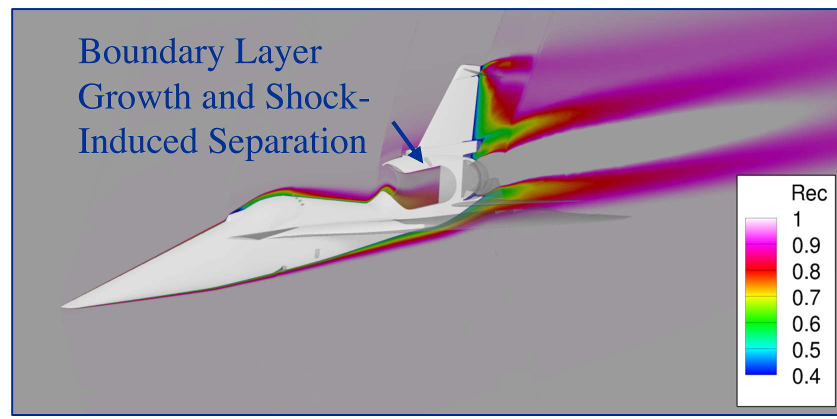
- NASA analysis of Quiet Supersonic Technology (QueSST) design was performed:
  - Large differences (>5%) in NASA FUN3D vs. LM CFD++ inlet recovery and distortion.
  - Differences attributed to predicted flow-field on top of vehicle.
- At GRC, the QueSST project marks the first time full vehicle aerodynamic analysis was performed using FUN3D on a mature vehicle design.
- The QueSST aircraft is unique in that it contains an aft-fuselage, top-mounted inlet installation, with complex shock-wave boundary-layer interactions that are sensitive to geometry and are difficult to predict.
- Depending on the success of this first test, more investment/validation data sets likely needed to bring PAI best practices up to the same maturity level as sonic boom analysis.

Cruise Pt,
Mach 1.42, AOA 1.7



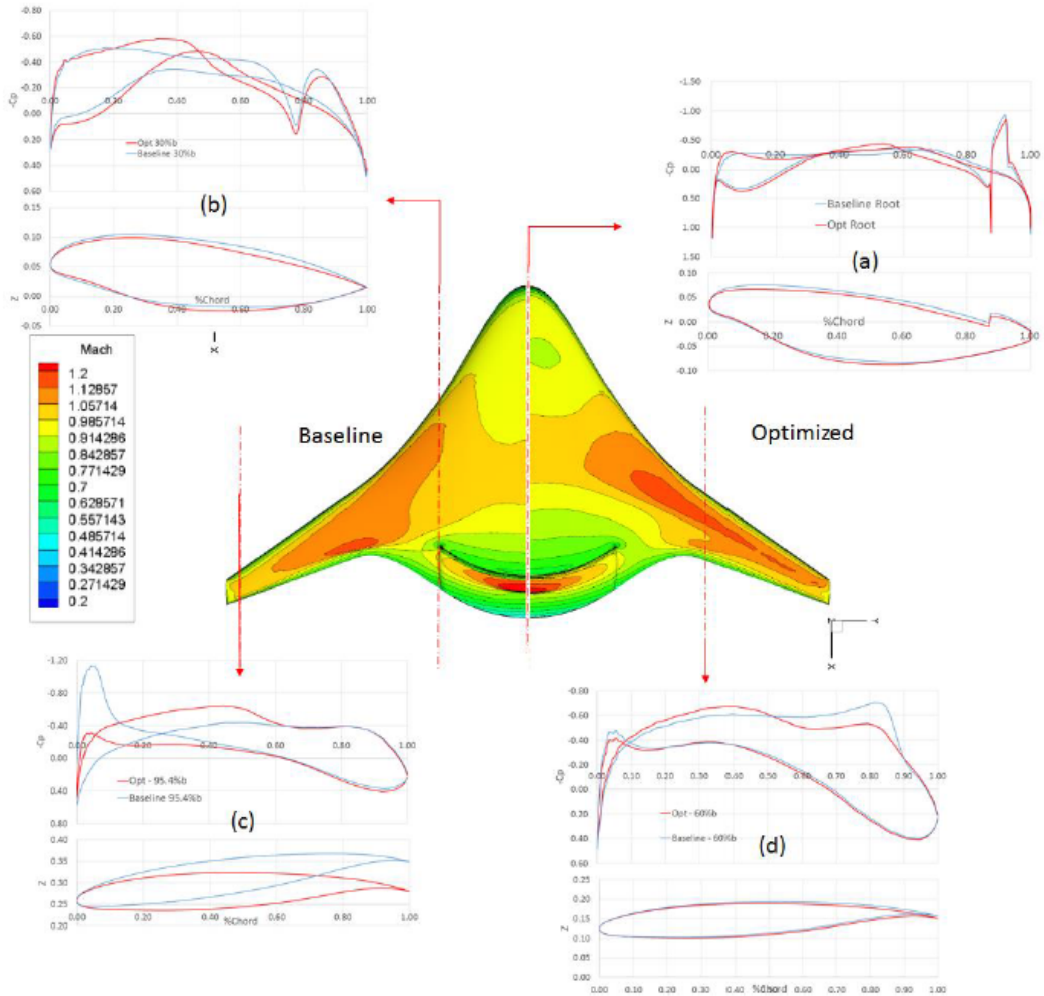
92% Recovery
Fun3D

97% Recovery
CFD++



HWB Propulsion Airframe Integration CFD

- Developed a design approach for hybrid wing body (HWB) integrated airframe and propulsion by parameterizing the planform and airfoils at control sections of the wingbody
- Used Go-Flow code, a 3D unstructured N-S flow solver, performed Euler analysis'; flow through propulsion represented by a simple block model and a proper static back pressure at the fan face
- Developed adjoint-based optimizer to improve aerodynamic performance
- Reduction in induced and wave drags were achieved, resulting in ~10 counts reduction in total drag under the trimmed condition
- Demonstrates mutual interference between airframe and propulsion system is significant and can be accounted for using this computational procedure



Optimized geometry comparison: Baseline (left, blue) vs Optimized (right, red) of the integrated configuration

LES Research for Propulsion

Objectives:

▪Develop tools and techniques for simulating propulsion flows using large-eddy simulation (LES) and hybrid Reynolds-Averaged Navier-Stokes/LES methods.

▪Create detailed databases of shear layer and shock-wave/boundary-layer interaction flows to guide new model development.

Technical Challenge:

While LES is a conceptually simple technique there are numerous issues that must be resolved before it becomes routine analysis tool; inflow turbulence specification, wall modeling, hybrid RANS/LES models, numerical scheme, etc.

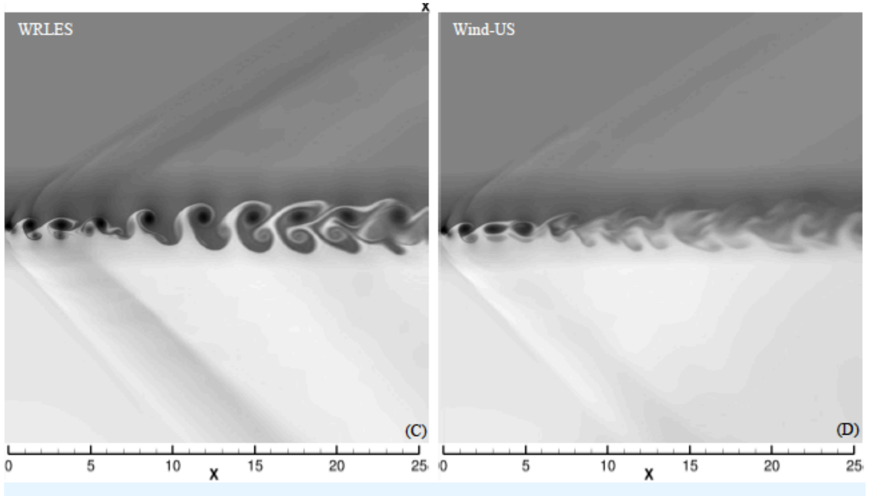
Approach/Key Milestones:

- ✓ Incorporate inflow turbulence generation methods
- ✓ Develop detailed flow field solutions for mixing layers
- ✓ Develop detailed flow field solutions for hot jets
- ○ Develop detailed flow field solutions of shock-wave/boundary-layer interactions
- ○ Incorporate wall models
- ○ Develop detailed flow field solutions of diffusers, separated flows

POINT OF CONTACT:
LTN/Jim DeBonis

Deliverables/Products:

- High quality LES databases for model development
  - 2D mixing layers
  - Hot jets
  - Shock-wave/boundary-layer interactions
  - Diffusers, separated flows
- Accurate turbulent inflow boundary conditions and best practices for simulations



Density contours of mixing layer (close-up view of splitter tip) for high-order LES (left) and low-order RANS (right)
[from AIAA2015-2939, Mankbadi & Georgiadis]

Informing Turbulence Models Using ILES

PROBLEM

Accurate prediction of shock wave/boundary-layer interaction (SBLI) remains a challenge using the current industry-standard turbulence models. This is largely due to the lack of understanding of SBLI characteristics like low-frequency motion of the shock-separation system, three-dimensionality, and corner separation where the Boussinesq approximation starts becoming suspect.

OBJECTIVES

Using the insights provided by Implicit LES (ILES) in the physics of low-frequency shock-separation oscillations, three-dimensionality, and corner separation, inform the current turbulence models to gain improved predictions and/or show the need for development of new higher order turbulence models.

APPROACH

Highly-resolved meshes will be used in SBLI simulations performed with ILES to, first, acquire statistical quantities to study the unsteady characteristic of an SBLI and second, to calculate budget of exact equations of turbulence kinetic energy (TKE), rate of dissipation, specific rate of dissipation, and stress transport and compare with their RANS counterparts. An 8-degree SBLI exhibiting evidence of three-dimensional shock-separation system will be investigated. Effect of corner separation will be explored by including a sidewall in the simulation. The code, FDL3DI, employs bandwidth and order optimized WENO with Roe scheme for inviscid flux and a sixth-order compact scheme for the viscous fluxes. The time integration will be performed using implicit Beam-Warming scheme. Digital filtering method will be used to generate inflow turbulent boundary layer and and the shock is imposed using the Rankine-Hugoniot relations by specifying pre- and post-shock conditions.

POC: Manan Vyas (GRC)

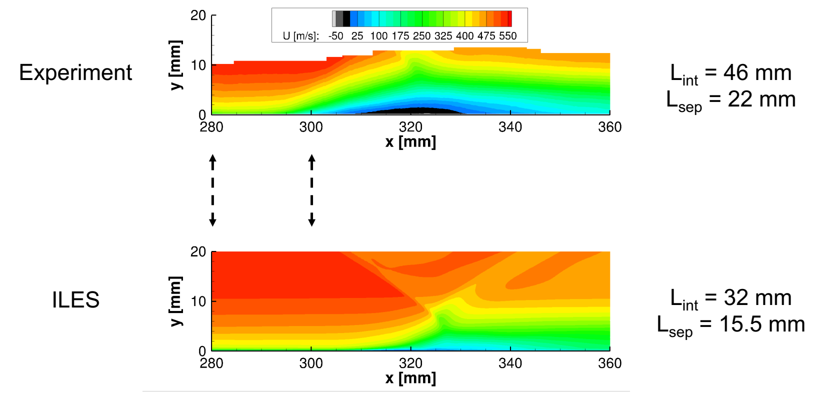
DETAILS OF PRELIMINARY WORK

Preliminary simulations using a flatplate have been completed and verified the suitability of turbulent boundary-layer generation for the present flow conditions and calculation of components of the budget terms like production, transport, dissipation, dilatation, etc. However, the SWBLI length is smaller in comparison to the experiment and future work is aimed at uncovering the source of the discrepancy.

SIGNIFICANCE

This two-pronged approach will provide physical insights on the role that unsteadiness and three-dimensionality play in SBLI and lead to improvements in our current models and spur the development of new models that incorporate the relevant flow physics.

Streamwise Velocity



- Separation is present in the ILES, but in the near-wall region

Glenn Flux Reconstruction (GFR) Navier-Stokes Code

PROBLEM

Current industrial CFD codes used for solving aeropropulsion flows with complex geometries generally use unstructured finite-volume methods and are typically no higher than second-order accurate. These methods constrain the accuracy and efficiency of current CFD codes.

OBJECTIVES

At Glenn, develop a high-order 3D CFD code based on the Flux Reconstruction (GFR) method for solving the Navier-Stokes equations on brick element meshes to demonstrate accuracy and efficiency improvements over standard methods.

APPROACH

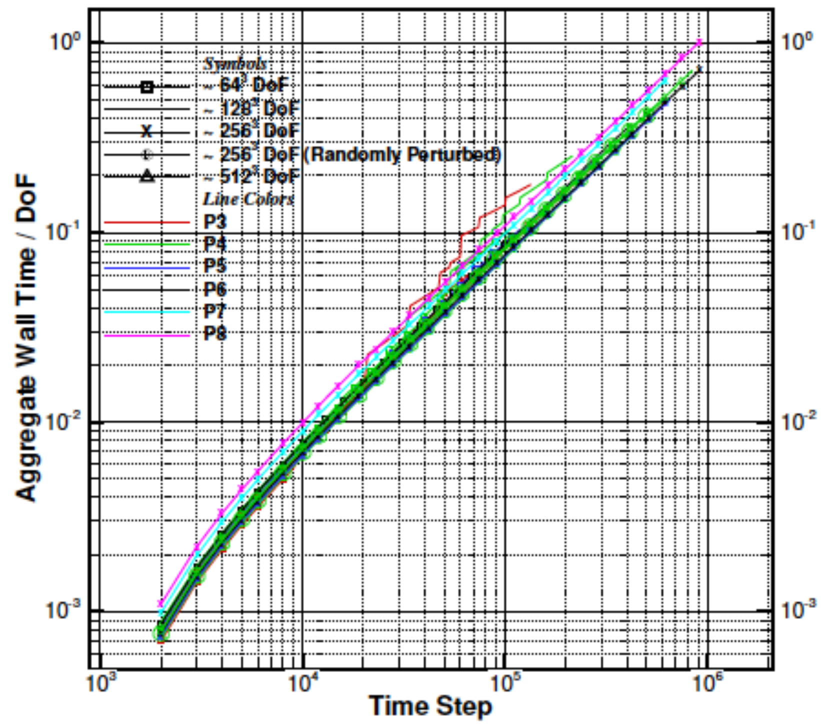
The GFR code provides a simple, efficient, and easy to implement method for solving fluid flow problems on mixed element unstructured grids and is accurate to an arbitrary order. The GFR code is to be used for solving aeropropulsion flows with complex geometries. This code has been expanded from 2D inviscid flows to 3D viscous flows on brick elements. While extending the 2D code to 3D, additional infrastructure has been embedded within the GFR code to easily allow for future additions such as 3D hybrid meshes and non-uniform P-adaptive meshes.

RESULTS

The isentropic Euler vortex problem was computed to verify the accuracy of the FR method for inviscid flows. Results of the study found the expected P+1 order accuracy through one period and super-accuracy through 100 periods demonstrating that vortical flow structures can be accurately predicted by GFR for long amounts of time with minimal unwanted numerical dissipation introduced into the system.

SIGNIFICANCE

This 3D Navier-Stokes GFR code uses the FR method to efficiently compute high-order solutions on brick element meshes. Its capabilities are being extended to allow for hybrid element meshes and more generalized aeropropulsion-oriented flows. Results indicate that the GFR code appears to be a viable candidate for accurate ILES of aerodynamic flows for complex geometries.



Comparison of the aggregate wall times per DoF versus time step for each of the vortex simulations. This shows that for each time step, the computational cost per DoF is essentially independent of P:

POC: Seth Spiegel (GRC)