

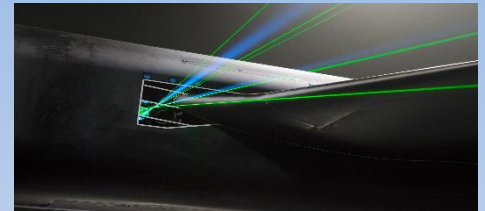
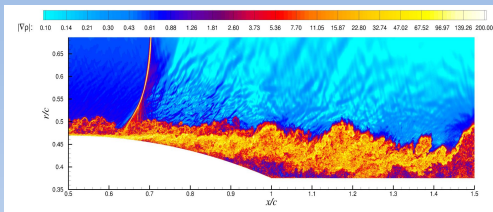
NASA's Revolutionary Computational Aerosciences (RCA) and CFD Vision 2030

Mujeeb R. Malik

Technical Lead, RCA

NASA Langley Research Center

Hampton, VA 23681



International Conference on Computational Fluid Dynamics (ICCFD10)

Barcelona, Spain

July 9-13, 2018



Outline



Revolutionary Computational Aerosciences (RCA) Transformational Tools and Technologies Project

- **Impact of Computational Fluid Dynamics (CFD)**
- **RCA Research Portfolio**
- **CFD Vision 2030**
- **RCA's 40% Technical Challenge**
- **Wall-resolved LES of RCA Test Cases**
- **CFD Validation Experiments**
- **Concluding Remarks**



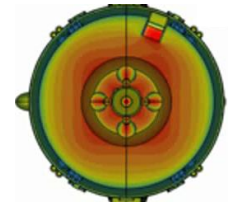
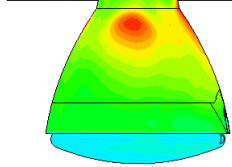
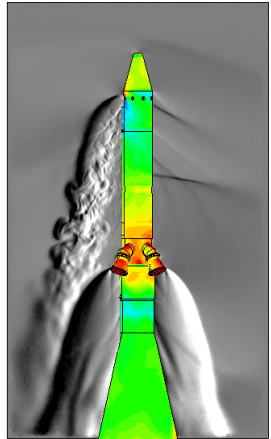
Acknowledgements



- **RCA Colleagues at NASA Ames, Glenn and Langley**
- **RCA Sponsored Principal Investigators at Universities, funded through NASA Research Announcements (NRA)**

Motivation

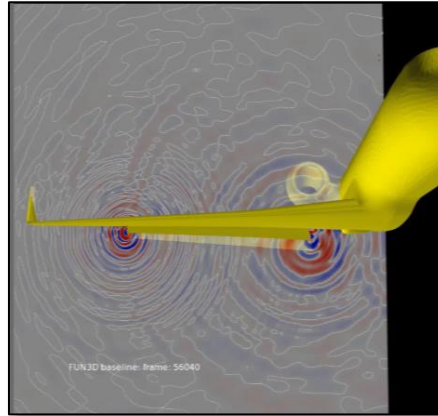
Computational fluid dynamics is used to tackle some of NASA's/nation's most complex aerodynamics problems



Ares 1-X



**NASA/Boeing
Truss-Braced Wing**



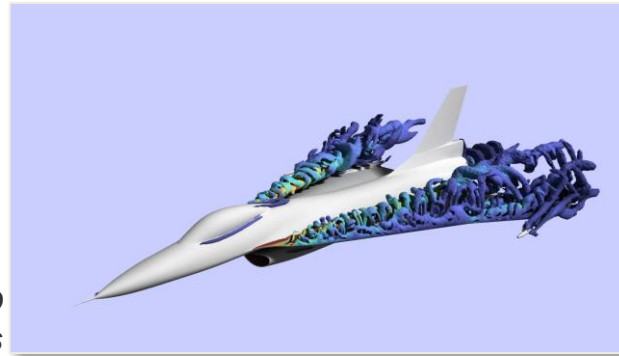
**Gulfstream
G550**



**Mars InSight
Lander**



**DoD
Applications**





Impact of CFD - 1



“Since the first generation of jet airliners, there has been about a 40% improvement in aerodynamic efficiency and a 40% improvement in engine efficiency ... **about half of that has come from CFD.”**

Robb Gregg, BCA Chief Aerodynamicist

During Panel Discussion Session “Transformative Aerospace System Analysis, Design and Certification: A Vision for CFD in 2030” at AIAA Aviation 2014

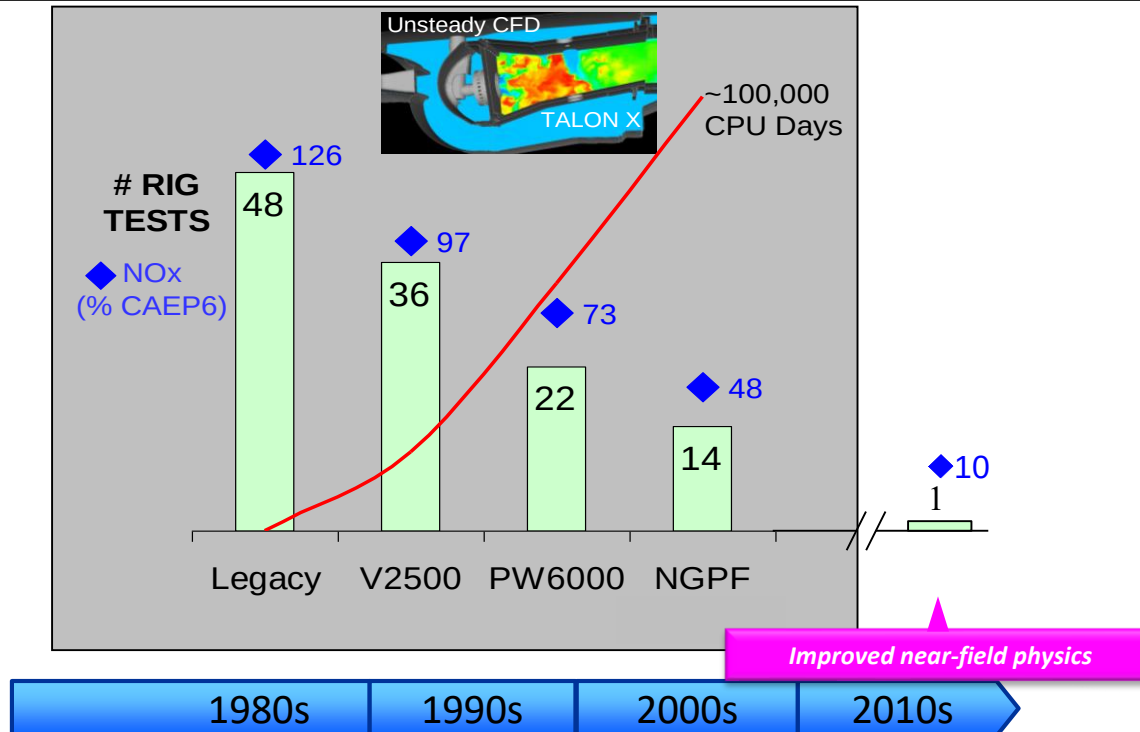


NASA ARMD Strategic Thrust 3: Ultra-efficient commercial vehicles

Impact of CFD - 2

“Using CFD in the design ... has resulted in 75% reduction in the number of tests to develop a combustor, and 60% reduction in emissions.”

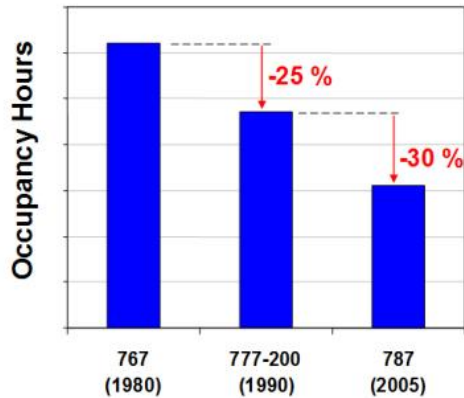
Stephen Morford, P&W Chief Engineer for Systems and Aerodynamics
During Panel Discussion Session “Transformative Aerospace System Analysis, Design and Certification: A Vision for CFD in 2030” at AIAA Aviation 2014.



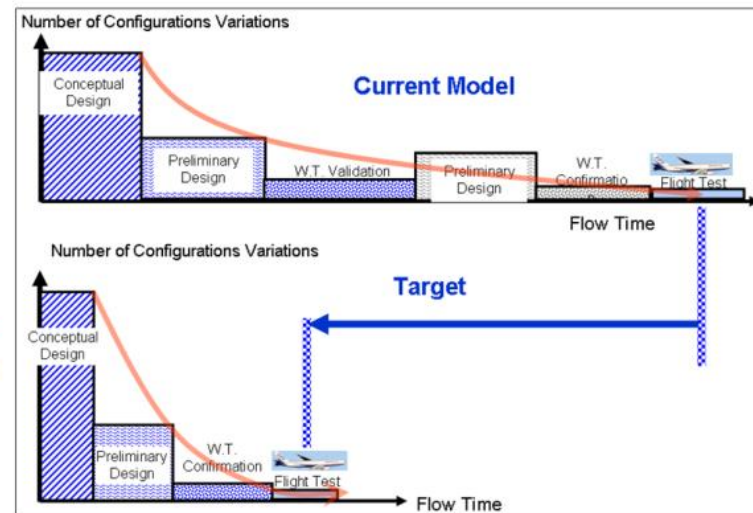
Impact of CFD - 3

CFD's Role

Lower testing costs, reduced time to market...



And promises to provide breakthrough cycle time reduction

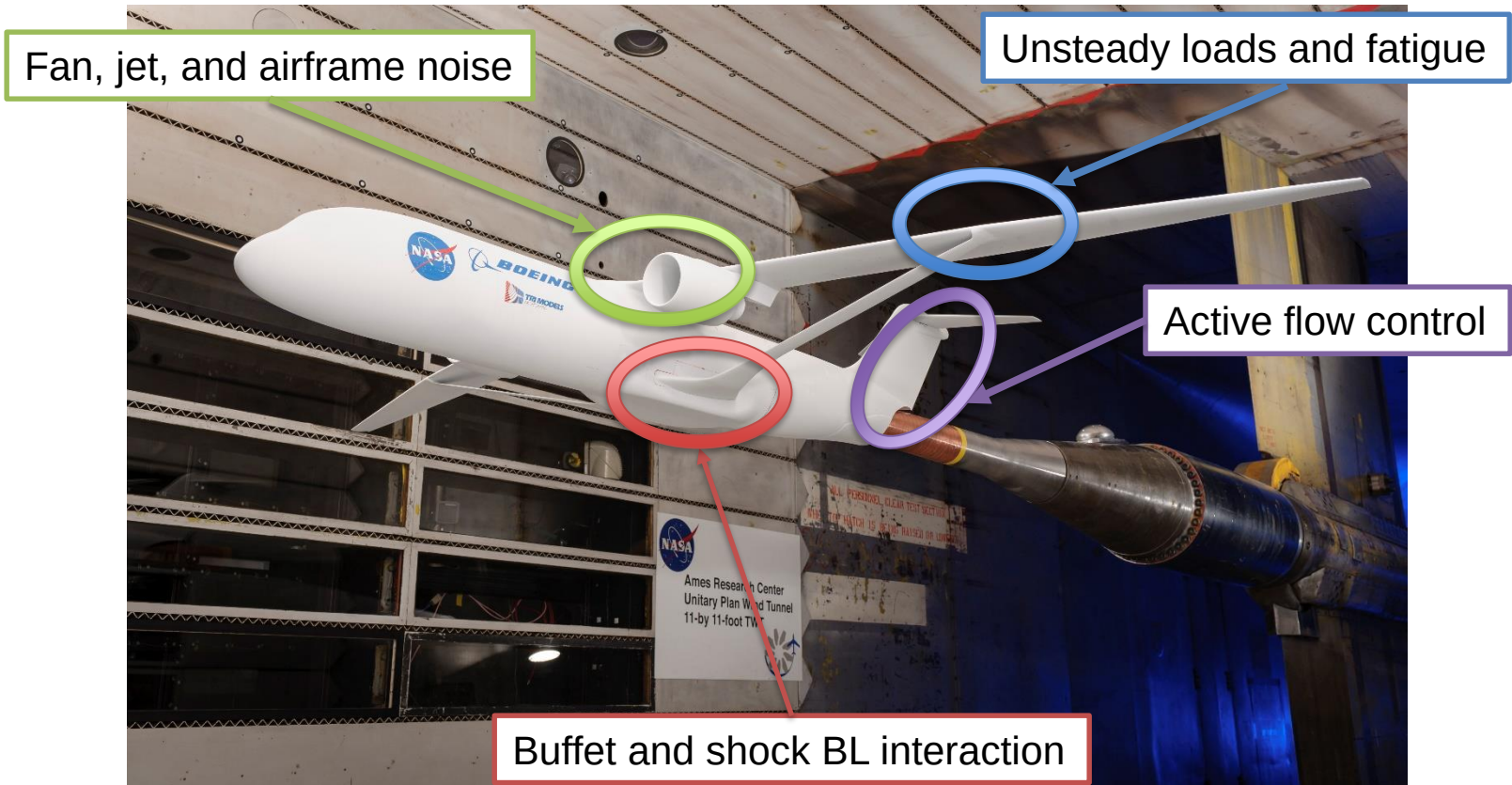


COPYRIGHT © 2015 THE BOEING COMPANY

Gregg, R. and Alonso, J. "Certification by Analysis," AIAA Aviation 2016

Challenge

- Increase the use of validated computational aerodynamic simulation tools for next generation airplane concepts which would reduce the need for physical testing (ground and flight) to confirm performance requirements
 - Develop **accurate**, **efficient** and **robust** computational techniques (including scale resolving simulations) to predict:





“Three Pillars” of **RCA** Research

- **Robustness/Reliability**

- *Ability to generate results with error bounds on every try, by a nonexpert user*
 - Robust solver technology/nonlinear stability
 - Uncertainty quantification

- **Cost/Efficiency**

- *Ability to compute faster by orders of magnitude compared to the current practice*
 - Exploit emerging HPC hardware capability
 - Numerical algorithms (e.g., solvers, adaptive grids)

- **ACCURACY**

- *Ability to accurately compute complex turbulent flows (e.g., transition, flow separation, free shear flows, shock/boundary-layer interaction)*
 - Numerical methods (e.g., HOMS), grids, boundary/initial conditions, etc.
 - Improved physical modeling and simulations
 - CFD validation experiments (including physics experiment for model development) a critical need

CFD technology with above attributes will enable “Simulation-Based Engineering”:

- Application to novel configurations, with confidence, for all NASA missions
 - Airplanes (fixed-wing , vertical lift, manned/unmanned)
 - Launch vehicles, airbreathing propulsion
 - Entry, Descent, Landing
- Aircraft certification by analysis



RCA Research Portfolio – 1

Turbulence Modeling and Simulations



OBJECTIVE

Develop new and improved turbulence models and simulation strategies that overcome the existing challenges in prediction of complex turbulent flows and significantly increase the accuracy and range of applicability of the models.

APPROACH

- **RANS Modeling**
 - Higher moment closures
 - Structure-based models
 - Explicit algebraic stress models
 - 2 eq model with improved length scale equation
 - Lag model for non-equilibrium effects
- **Eddy Resolving Methods**
 - Wall-resolved and wall-modelled LES, for high Reynolds number flows
 - Hybrid RANS/LES approaches
 - LES of mixing layers and jet flows
 - DNS of canonical flows to provide data for development/evaluation of turbulence models
- **Experimental Validation**
 - Provide flow physics data for model development/validation
 - Juncture flow experiment
 - Shock/BL interaction
 - Turbulent heat flux



RCA Research Portfolio – 2

Numerical Methods



OBJECTIVE

Develop numerical methods that significantly increase accuracy, efficiency, and robustness of turbulent flow computations for aerodynamic and propulsion applications.

APPROACH

- **Structured and Un-structured Grid Schemes with Low Numerical Dissipation and Dispersion.**
 - Flux reconstruction
 - Discontinuous Galerkin
 - Stabilized finite-elements (SUPG)
 - Conservation element/solution element (CE/SE)
 - High-order entropy stable formulations
- **Convergence Acceleration Strategies for Efficient Solution of Navier-Stokes Equations.**
 - Grid adaptation
 - Enhanced linear/nonlinear solvers
 - Temporal integration schemes
 - High Performance Computing
- **Error Estimation and Uncertainty Quantification to Increase Solution Credibility.**



RCA Research Portfolio – 3

Transition Modeling



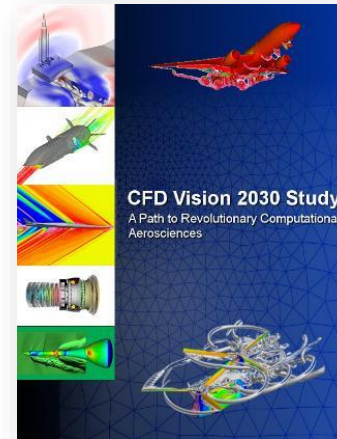
OBJECTIVE

Develop amplitude-based transition prediction methods for laminar flow wing design and develop reduced order transition prediction models for routine use in CFD codes.

APPROACH

- **Physics-Based Transition Prediction**
 - Receptivity to roughness
 - Linear and nonlinear parabolized stability equations
 - Subsonic/supersonic/hypersonic boundary layers
 - 2D and 3D [TS and crossflow modes]
 - Non-modal growth
- **Direct Numerical Simulation (DNS)**
 - Provide validation for lower fidelity prediction
 - Receptivity and late stage of transition
- **Validation Experiments**
 - Supersonic quiet tunnel
 - Roughness effects
 - Surface imperfections/steps
 - Crossflow-induced transition
- **Develop and Implement Reduced-order Transition Models in CFD Codes**

NASA CR 2014-218178

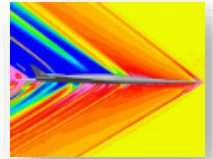
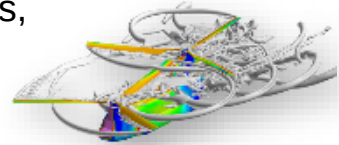


- **The study was conducted:**
 - “...to provide a **knowledge-based forecast** of the future computational capabilities required for **turbulent, transitional, and reacting flow simulations**...”
 - “...and to lay the foundation for the development of a future framework/environment where **physics-based, accurate predictions of complex turbulent flows, including flow separation**, can be accomplished routinely and efficiently in cooperation with other physics-based simulations to enable multi-disciplinary analysis and design.”

**An industry-universities team led this study with
CFD community input**

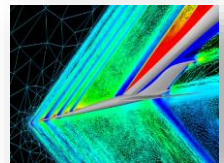
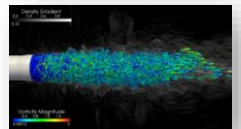
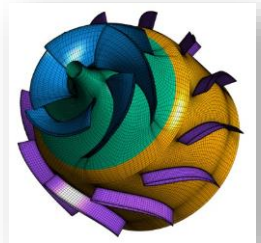
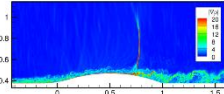
The CFD Vision 2030

- Emphasis on physics-based, predictive modeling**
 Transition, turbulence, separation, unsteady/time-accurate, chemically-reacting flows, radiation, heat transfer, acoustics and constitutive models, among others.
- Management of errors and uncertainties**
 Quantification of errors and uncertainties arising from physical models (epistemic), mesh and discretization, and natural variability (aleatory) and their effect on important engineering quantities of interest.
- Automation** in all steps of the analysis process
 Geometry creation, meshing, large databases of simulation results, extraction and understanding of the vast amounts of information generated with minimal user intervention.
- Ability to effectively utilize massively parallel, heterogeneous, and fault-tolerant HPC architectures** that will be available in the **2030 time frame**
 Multiple memory hierarchies, latencies, bandwidths, programming paradigms and runtime environments, etc.
- Flexible** use of HPC systems
 Capability- and capacity-computing tasks in both industrial and research environments.
- Seamless integration with multi-disciplinary analyses and optimizations**
 High fidelity CFD tools, interfaces, coupling approaches, etc.



Vision 2030 Study Findings

- The **accuracy of CFD** in the aerospace design process is severely limited by the **inability to reliably predict turbulent flows** with significant regions of **separation**
- **HPC hardware is progressing rapidly** and technologies that will prevail are difficult to predict
- **Revolutionary algorithmic improvements** will be required to enable future advances in simulation capability
- **Mesh generation and adaptivity** continue to be **significant bottlenecks** in the CFD workflow, and very little government investment has been targeted in these areas
- Managing the **vast amounts of data** generated by current and future large-scale simulations will continue to be problematic and will become increasingly complex due to changing HPC hardware
- Technology advances are required to enable increasingly **multidisciplinary simulations**, for both **analysis and design optimization purposes**





CFD 2030 Technology Development Roadmap



TRL ■ LOW
■ MEDIUM
■ HIGH

◇ Technology Milestone

★ Technology Demonstration

⊕ Decision Gate

2015

2020

2025

2030

HPC

CFD on Massively Parallel Systems

PETASCALE

Demonstrate implementation of CFD algorithms for extreme parallelism in NASA CFD codes (e.g., FUN3D)

Demonstrate efficiently scaled CFD simulation capability on an exascale system

30 exaFLOPS, unsteady, maneuvering flight, full engine simulation (with combustion)

CFD on Revolutionary Systems (Quantum, Bio, etc.)

Physical Modeling

RANS

Improved RST models in CFD codes

Highly accurate RST models for flow separation

Hybrid RANS/LES

Integrated transition prediction

Unsteady, complex geometry, separated flow at flight Reynolds number (e.g., high lift)

LES

WMLES/WRLES for complex 3D flows at appropriate Re

Combustion

Chemical kinetics calculation speedup

Chemical kinetics in LES

Unsteady, 3D geometry, separated flow (e.g., rotating turbomachinery with reactions)

Algorithms

Convergence/Robustness

Automated robust solvers

Grid convergence for a complete configuration

Multi-regime turbulence-chemistry interaction model

Production scalable entropy-stable solvers

Uncertainty Quantification (UQ)

Characterization of UQ in aerospace

Reliable error estimates in CFD codes

Uncertainty propagation capabilities in CFD

Large scale stochastic capabilities in CFD

Geometry and Grid Generation

Fixed Grid

Tighter CAD coupling

Large scale parallel mesh generation

Automated in-situ mesh with adaptive control

Adaptive Grid

Production AMR in CFD codes

Knowledge Extraction

Integrated Databases

Simplified data representation

Creation of real-time multi-fidelity database: 1000 unsteady CFD simulations plus test data with complete UQ of all data sources

Visualization

On demand analysis/visualization of a 100B point unsteady CFD simulation

On demand analysis/visualization of a 100B point unsteady CFD simulation

MDAO

Define standard for coupling to other disciplines

High fidelity coupling techniques/frameworks

Robust CFD for complex MDAs

MDAO simulation of an entire aircraft (e.g., aero-acoustics)

UQ-Enabled MDAO



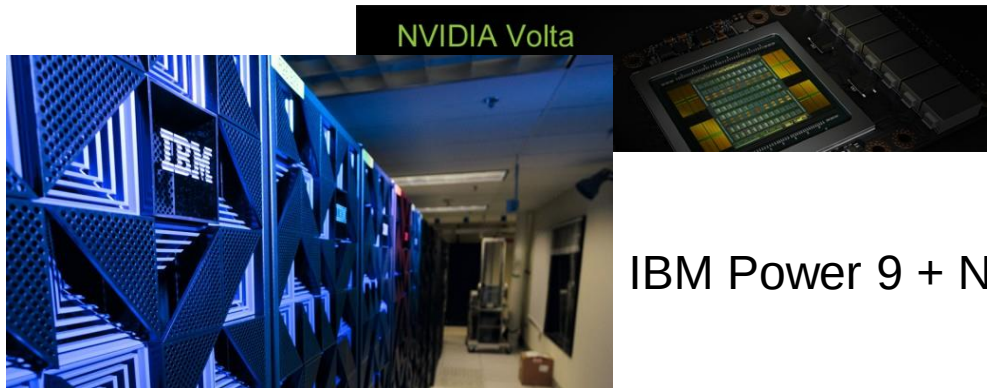
The Fundamental Computational Challenge!

- **Prediction of Smooth Body Flow Separation**
- **Unsteady Flows**
- **Boundary Layer Transition**
- **Speed of Computation/High Performance Computing**
- **Geometry Handling and Mesh Generation/Adaptation**

All these identified as technical challenges by CFD Vision 2030 Study

High Performance Computing (HPC)

- HPC hardware is changing rapidly, exascale in next 3-5 years
 - Improve performance of existing CFD solvers (e.g., FUN3D) on the new hardware
 - New solvers (GFR, *Eddy*) that more efficiently exploit emerging hardware capabilities



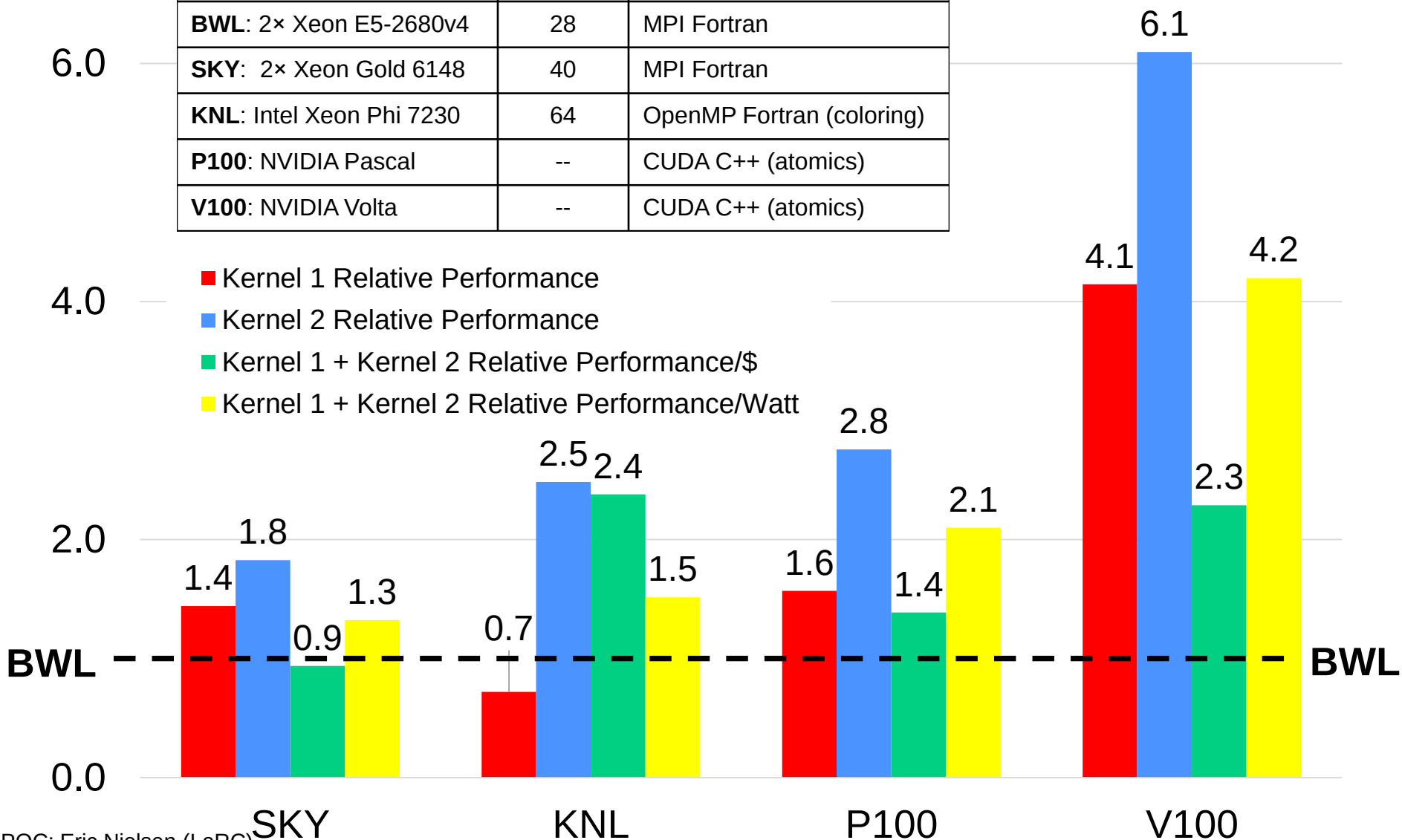
IBM Power 9 + NVIDIA V100

- Summit at ORNL: 200 pflops
 - 95% of Summit's compute is on the GPUs
 - 1,000 nodes = 6,000 GPUs $\approx$ 1.2 million Broadwell cores $\approx$ 5 Pleiades

Node-Level Performance Relative to Broadwell (BWL)

FUN3D Mini-Apps: Matrix Construction, Linear Solve Kernels

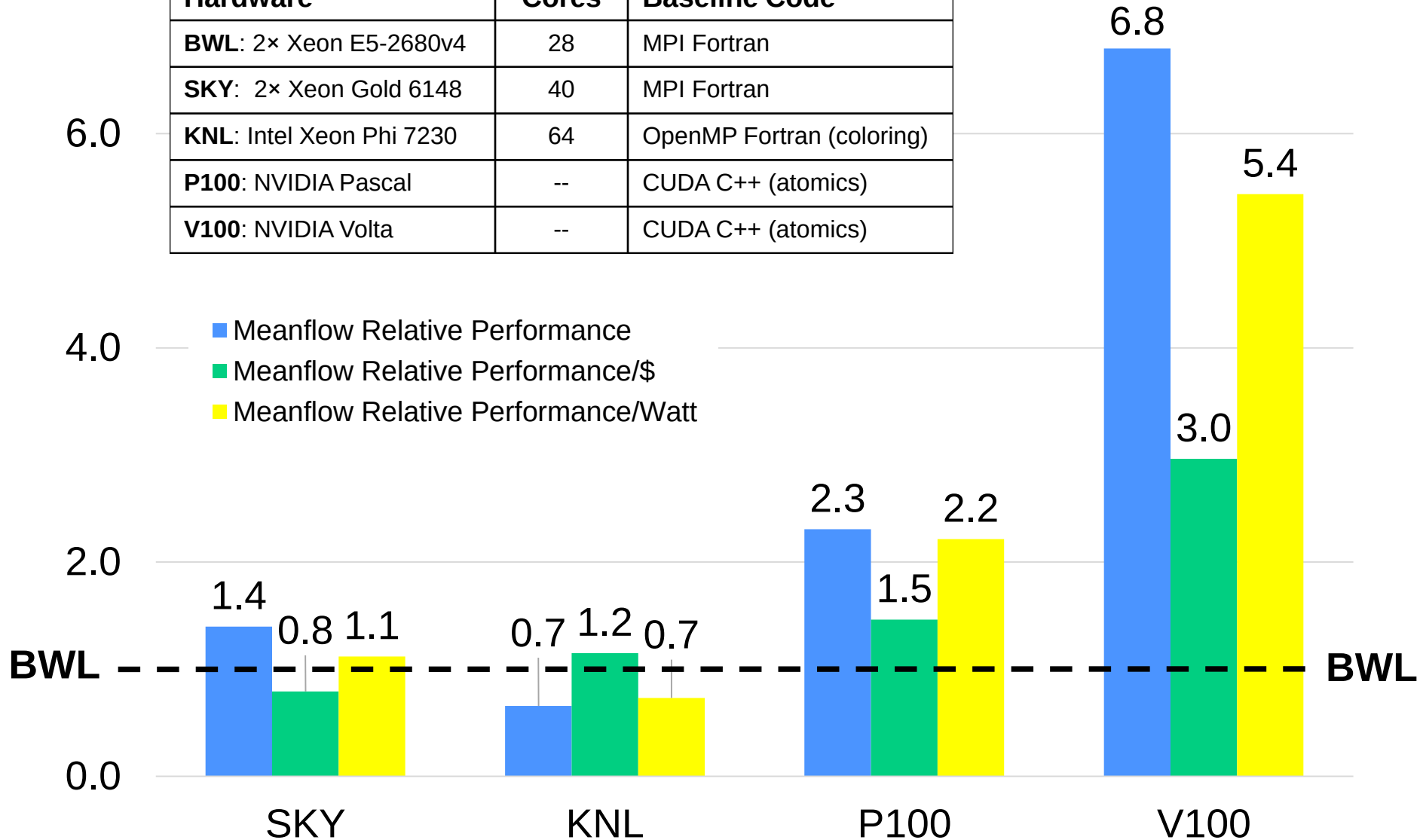
Hardware	Cores	Baseline Code
BWL: 2× Xeon E5-2680v4	28	MPI Fortran
SKY: 2× Xeon Gold 6148	40	MPI Fortran
KNL: Intel Xeon Phi 7230	64	OpenMP Fortran (coloring)
P100: NVIDIA Pascal	--	CUDA C++ (atomics)
V100: NVIDIA Volta	--	CUDA C++ (atomics)



Node-Level Performance Relative to Broadwell (BWL)

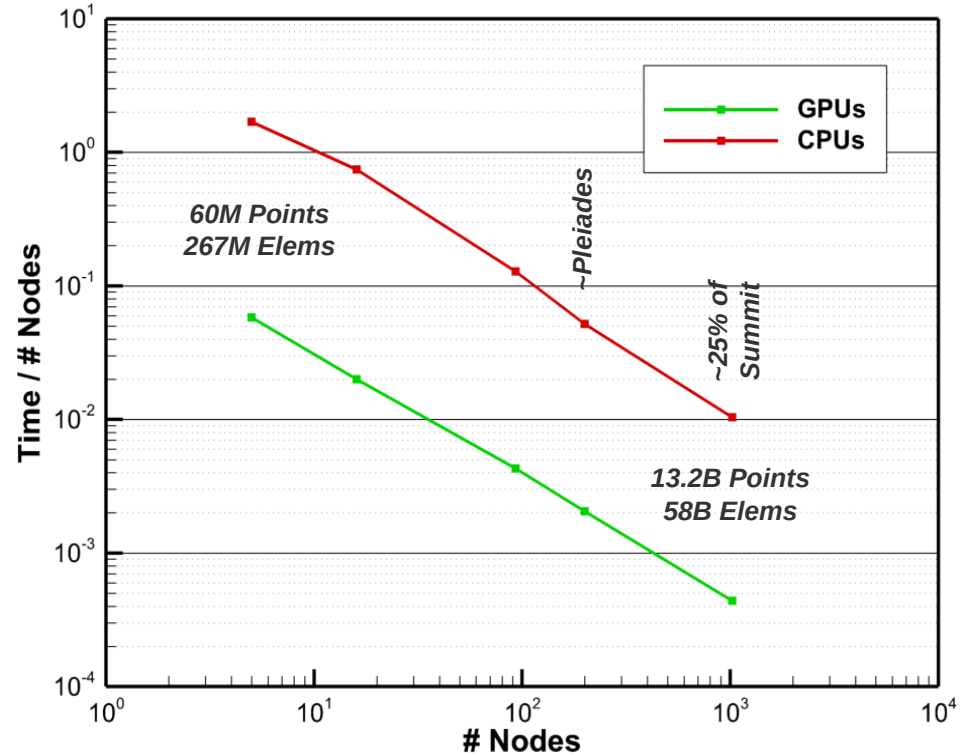
FUN3D: Laminar Navier-Stokes

Hardware	Cores	Baseline Code
BWL: 2× Xeon E5-2680v4	28	MPI Fortran
SKY: 2× Xeon Gold 6148	40	MPI Fortran
KNL: Intel Xeon Phi 7230	64	OpenMP Fortran (coloring)
P100: NVIDIA Pascal	--	CUDA C++ (atomics)
V100: NVIDIA Volta	--	CUDA C++ (atomics)



FUN3D Performance on Summit

- Plot captures weak scaling on new **ORNL Summit** system (IBM Power9 + V100)
- Each node solves ~12.8M grid points
 - 5 nodes: 60M pts/267M elms
 - 1,024 nodes: 13.2B pts/58B elms
- CPU curve** is MPI+OpenMP with 3 ranks/socket (total of 6 per node) with 168 total OpenMP threads per node (smt4)
- GPU curve** is MPI+CUDA:
 - 3 ranks/socket shepherding 1 GPU each (total of 6 per node); all MPI via GPU Direct
- Nearly linear performance for both
- GPU performance is 23x-37x faster, depending on data point (generally around 25x)

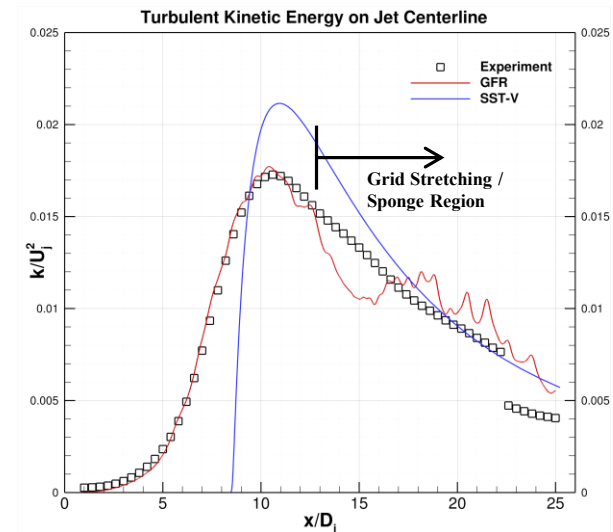
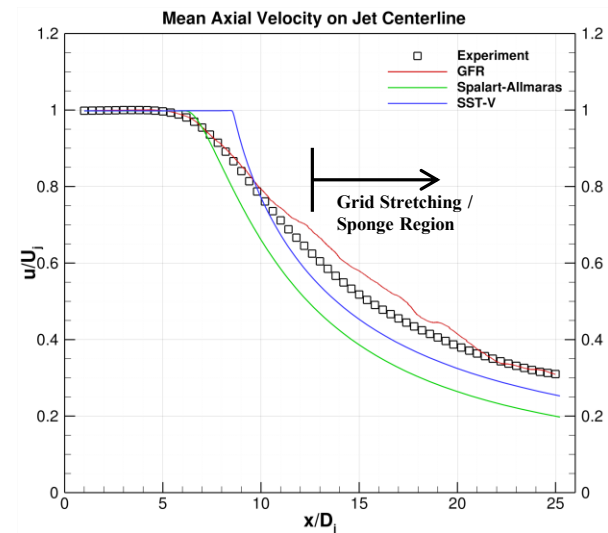
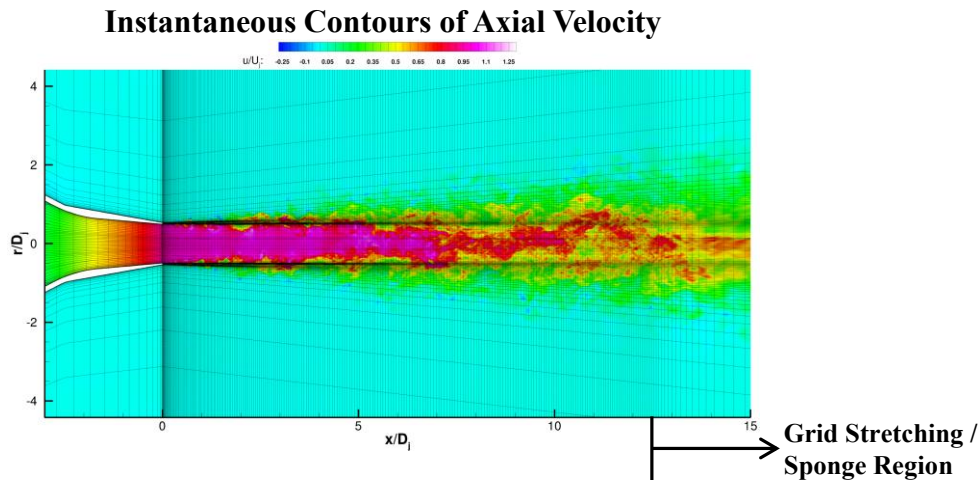


A potential 25-100x speedup in sight for CFD codes, from both hardware and algorithms

LES using GFR (Glenn Flux Reconstruction)

RCA Standard Test Case #4: Round Jet at Set Point 3 ($M_a=0.5$, $T_j/T_\infty=0.95$)

	HO Cells	DoF	FV Cells	FV Nodes
P4 Grid	1.0 M	130.0 M	66.5 M	66.7 M

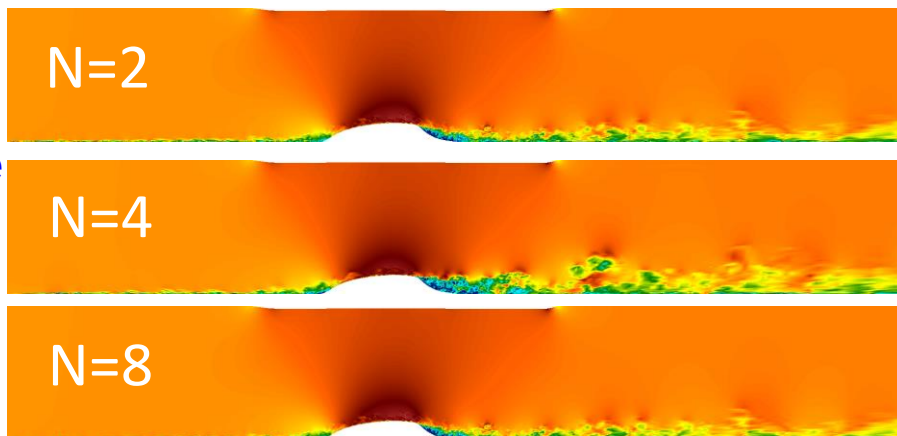


	Exp.	GFR ILES	% Err
Axial Location of $u/U_j=0.98$	6.221	6.353	2.12
Value of Max TKE	0.0173	0.0177	2.53
Axial Location of Max TKE	10.7	10.415	2.66

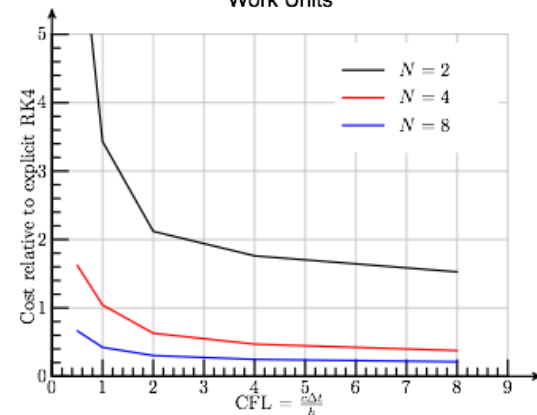
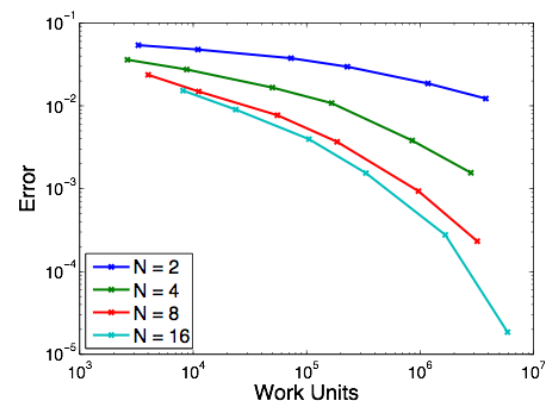
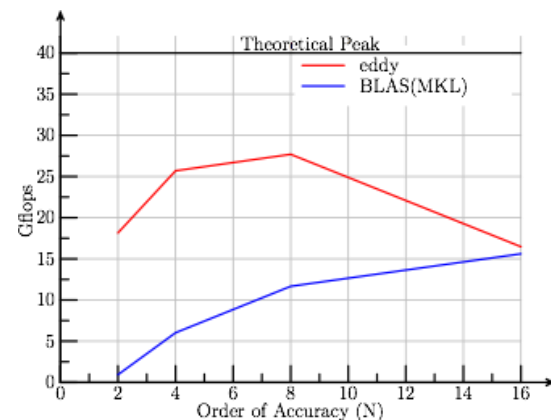
GPU performance is 20-25x faster
than 1 CPU core

- **Eddy** is an infrastructure based on the spectral-element method for scale resolving simulations

- Fully implicit space-time, entropy stable formulation
- 70% of machine theoretical peak on Intel hardware
- Better match for emerging HPC hardware
 - Less data movement, more flops for the same level of accuracy
- For high order, same error as low order, for orders of magnitude less CPU time
- Same memory usage and better efficiency than explicit schemes

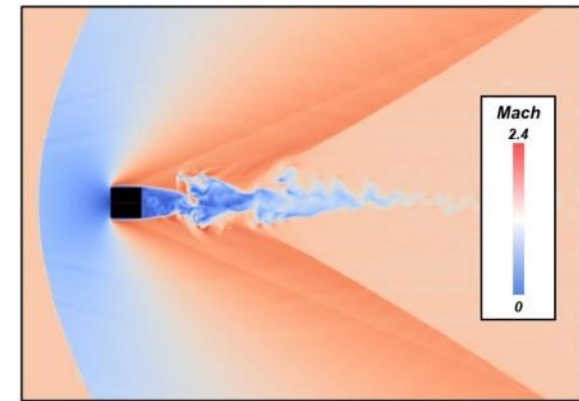


More fine scale structures with increasing N



- **Nonlinear Stability: “The Holy Grail of Numerical Analysis”**
 - RCA: Robustness/Reliability
- **High-Order Methods: Essential Tools for Scale Resolving Simulations**
 - Tantalizing Potential (e.g., P7-FEM)
 - 8-10 pts/wavelength vs 35-40 (2nd-order) $\rightarrow O(10^2)$ reduction
 - High arithmetic complexity $\Rightarrow$ 60-70% hardware utilization (GPU)
 - Frustratingly unreliable (very little previous theory)
 - Heuristic “fixes” include filtering, dealiasing, over-integration, ...
- **Developed Framework to Construct Nonlinearly Stable High-Order Discrete Operators for Compressible Navier-Stokes Equations**
 - Nonlinearly Stable Multi-block Finite-Difference
 - Weighted essentially non-oscillatory (WENO4 and WENO6)
 - Nonlinearly Stable DG spectral collocation
 - Curvilinear mapped hexahedral elements
 - Conservative (amenable to shock capturing)
 - Nonlinearly stable solid wall boundary conditions
 - Preserve the stability of interior operators
 - Re-inventing FEMs that are nonlinearly stable
 - Rich variations of collocation points
 - Nonconforming interfaces (hanging nodes)
 - H-refinement and P-refinement
 - Curvilinear mapped elements

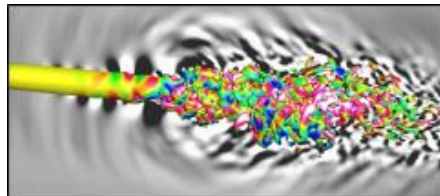
Marshal Merriam, NASA TM 101086 (1989)



Flow past 3D square cylinder at $Re = 10^4$ and $M = 1.5$; fourth-order accurate, **no explicit stabilization**.

Identify and downselect critical turbulence, transition, and numerical method technologies for 40% reduction in predictive error against standard test cases for turbulent separated flows, evolution of free shear flows and shock-boundary layer interactions on state-of-the-art high performance computing hardware.

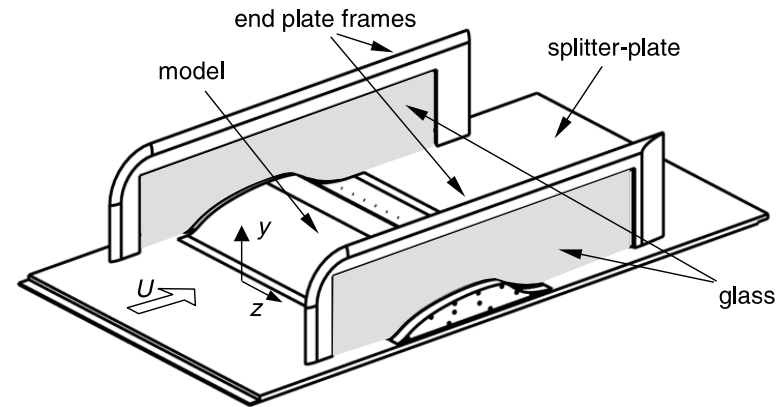
Completed May 31, 2018



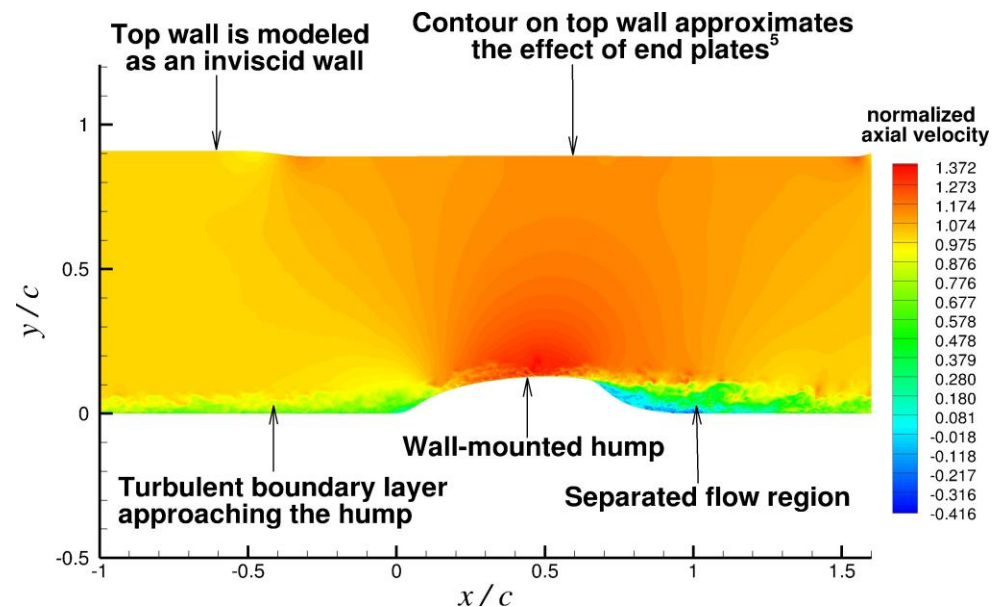
RCA Standard Test Cases

- **Selected after discussions within NASA and with AIAA Turbulence Modeling Benchmark Working Group**
 - Test cases possess the relevant flow physics
 - Simple enough to be useful (avoid complex geometries/additional uncertainties)
 - RANS fail to provide accurate solution
- **“Best” available test cases, but in no way “ideal”**
 - Case 1: “2D” NASA hump
 - Case 2: Axisymmetric transonic bump
 - Case 3: Compressible mixing layer
 - Case 4: Round jets (cold and hot)
 - Case 5: Axisymmetric compression corner
- **Issues**
 - Not necessarily free of WT/facility effects
 - Scale-resolving simulations require more detailed boundary info than RANS
 - Some cases too old → Exercise caution on level of trust put in the results
 - What quantitative metrics for progress assessment?
 - Define some relevant metrics, but allow some leeway and use judgement
 - Note: No test case for BL transition specified
- **Case for new standard/benchmark experiments**
 - Leverage advances in measurement techniques

- Experiment of Greenblatt et al. (AIAA J., 44(12), 2006).
 - $M = 0.1$
 - $Re_c = 936,000$
 - $Re_\theta = 7000$

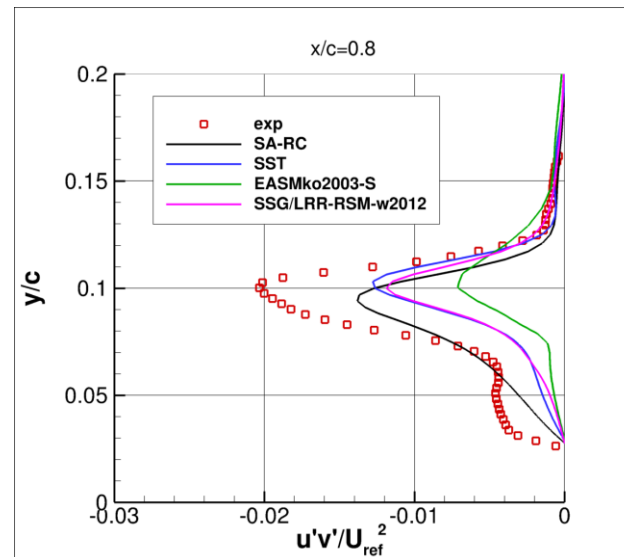
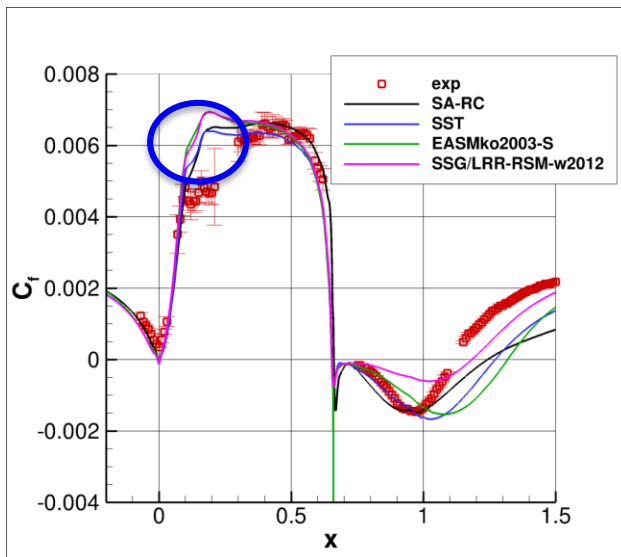


- In the computations, end plates omitted, with spanwise periodic assumption
 - Blockage effect modeled with contoured upper wall



• RANS Predictions

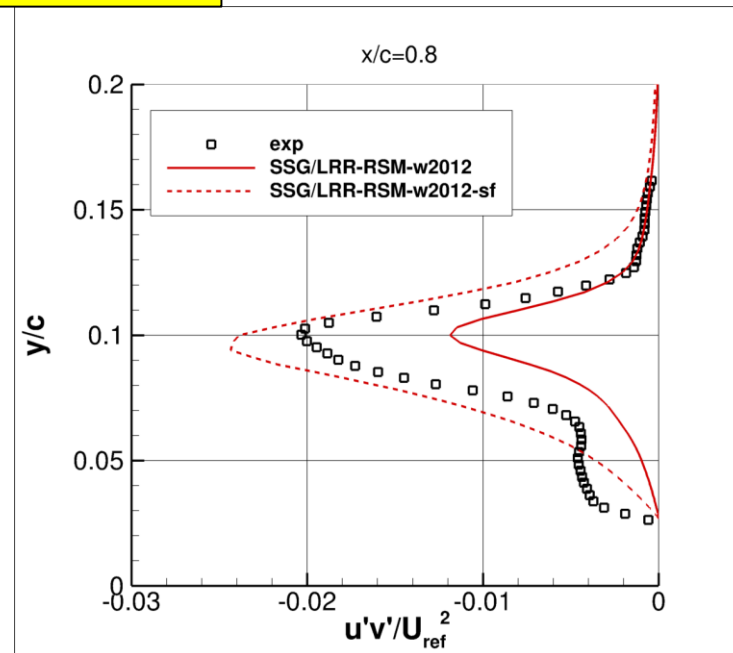
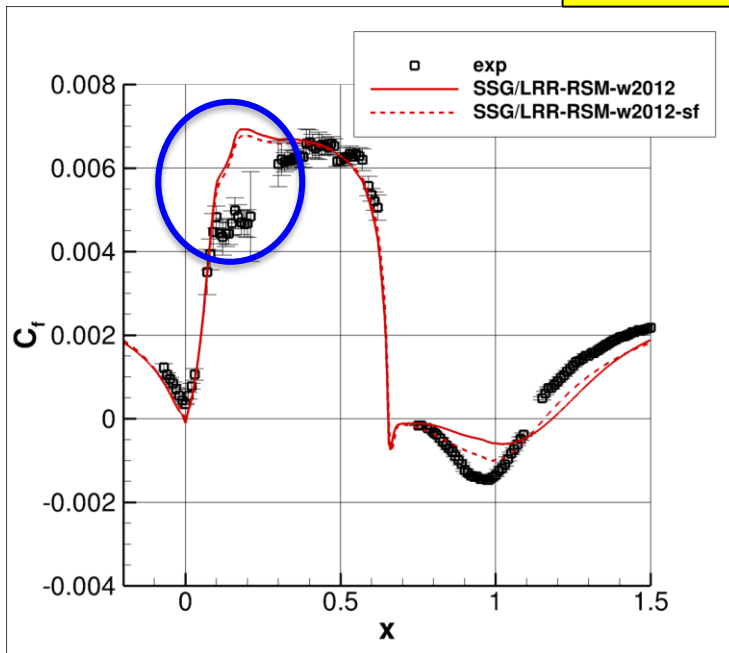
- Original predictions using SA, SST models were deficient
- Additional computations using Reynolds stress models did not improve predictions



	Separation error	Reattachment error	Bubble length error	Peak $u'v'$ error at $x=0.8$
SA-RC	-0.6%	12.9%	33.6%	-31.5%
SST	-1.7%	15.6%	41.9%	-36.2%
EASMko2003-S	-1.9%	19.8%	52.9%	-64.5%
SSG/LRR-RSM-w2012	-1.7%	7.5%	21.6%	-40.5%

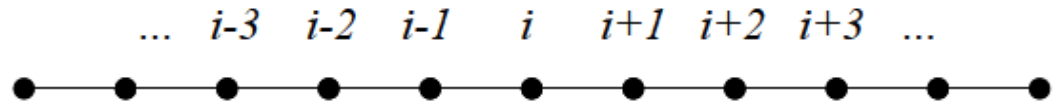
RANS Predictions

RSM separation fix

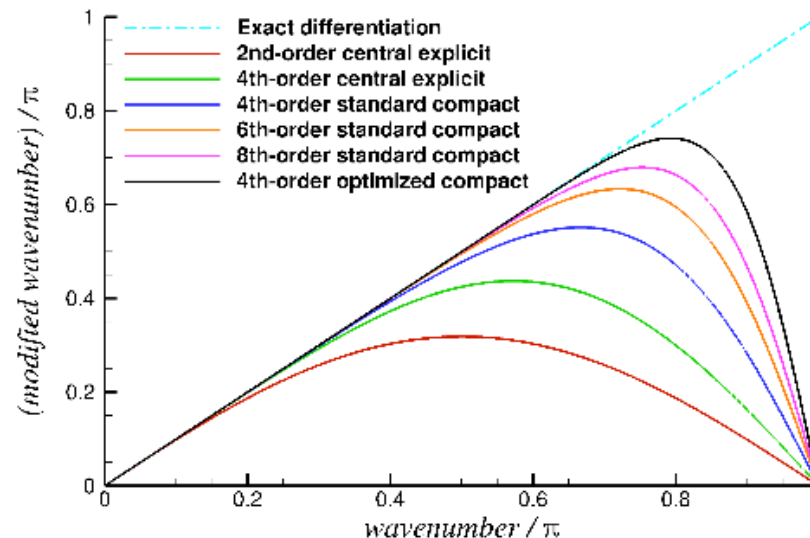


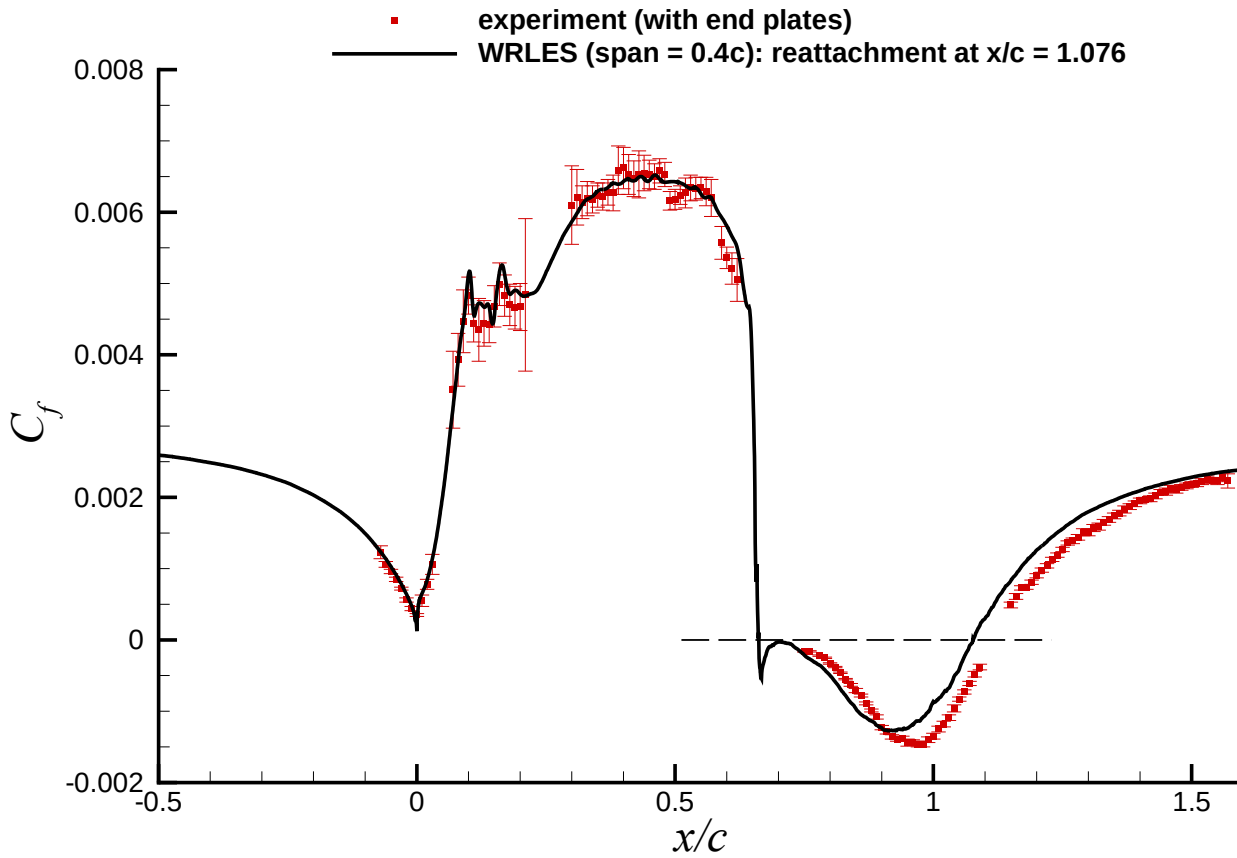
	Separation error	Reattachment error	Bubble length error	Peak $u'v'$ error at $x=0.8$
SSG/LRR-RSM-w2012	-1.7%	7.5%	21.6%	-40.5%
SSG/LRR-RSM-w2012-sf	-1.5%	5.1%	15.1%	21.5%

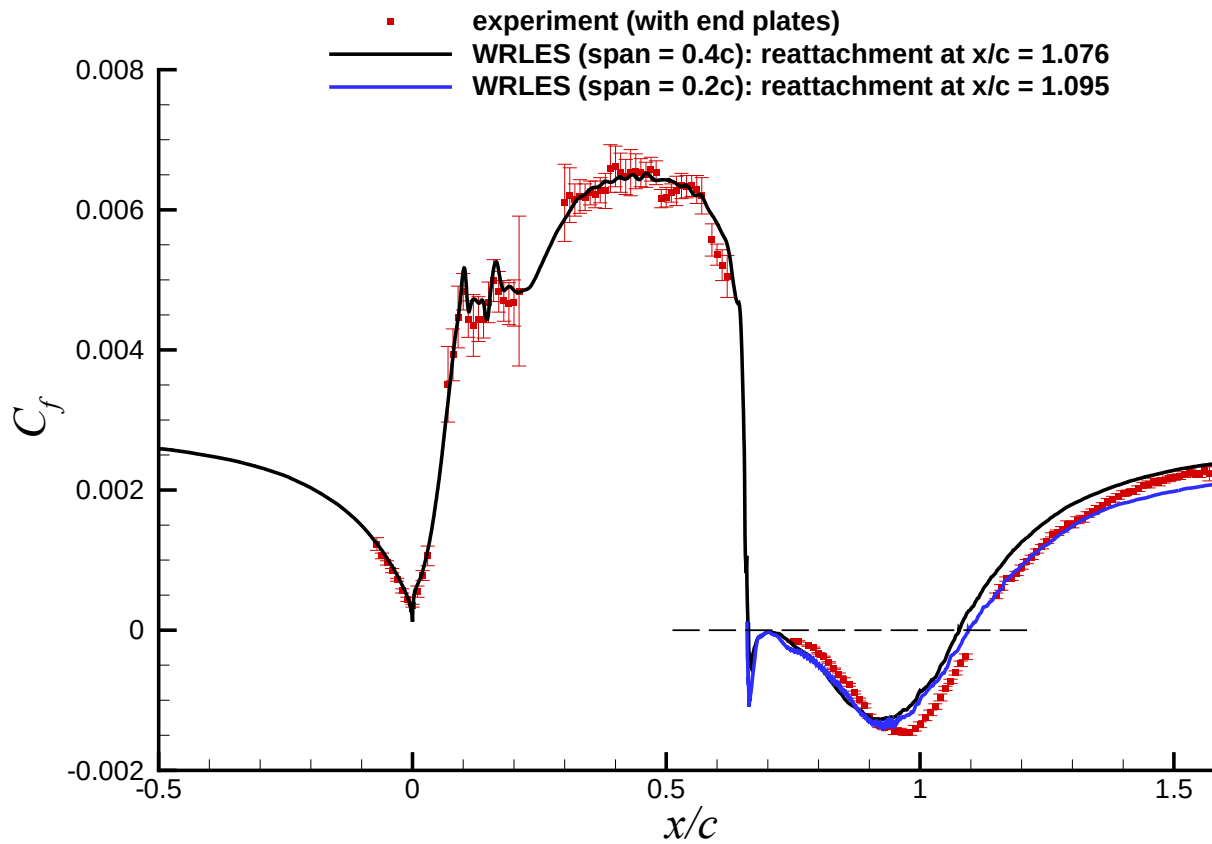
- Fourth order optimized compact scheme (*Ashcroft and Zhang, JCP 190, 2003.*)
- Second-order fully implicit time integration
- Spatial filter for stabilization (*Gaitonde and Visbal, AIAA J., 38(11), 2000.*)
 - Filter order = 2 N
 - $-0.5 < \alpha_f \leq 0.5$
 - $\alpha_f = 0.49$ used

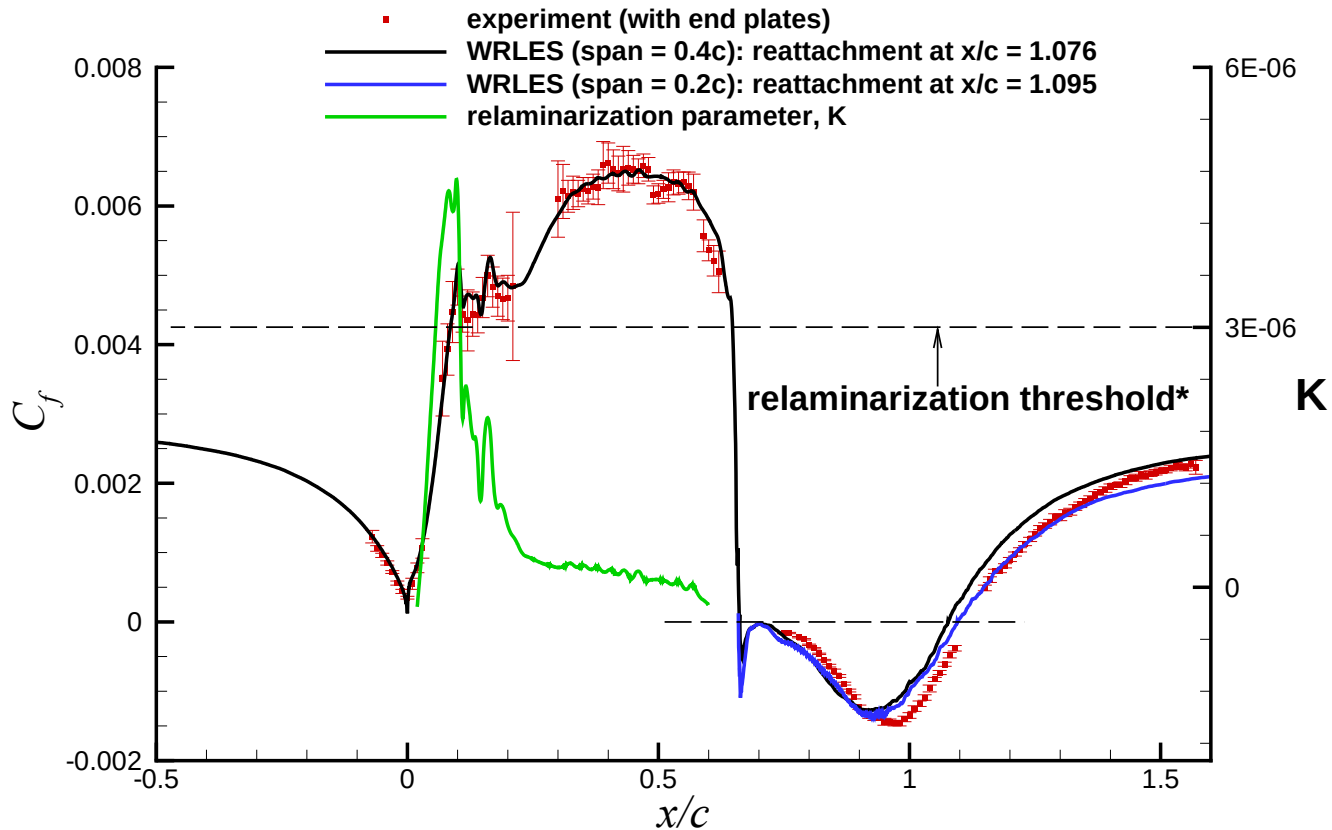


$$\alpha_f \bar{f}_{i-1} + \bar{f}_i + \alpha_f \bar{f}_{i+1} = \sum_{n=0}^N \frac{a_n}{2} (f_{i+n} + f_{i-n})$$



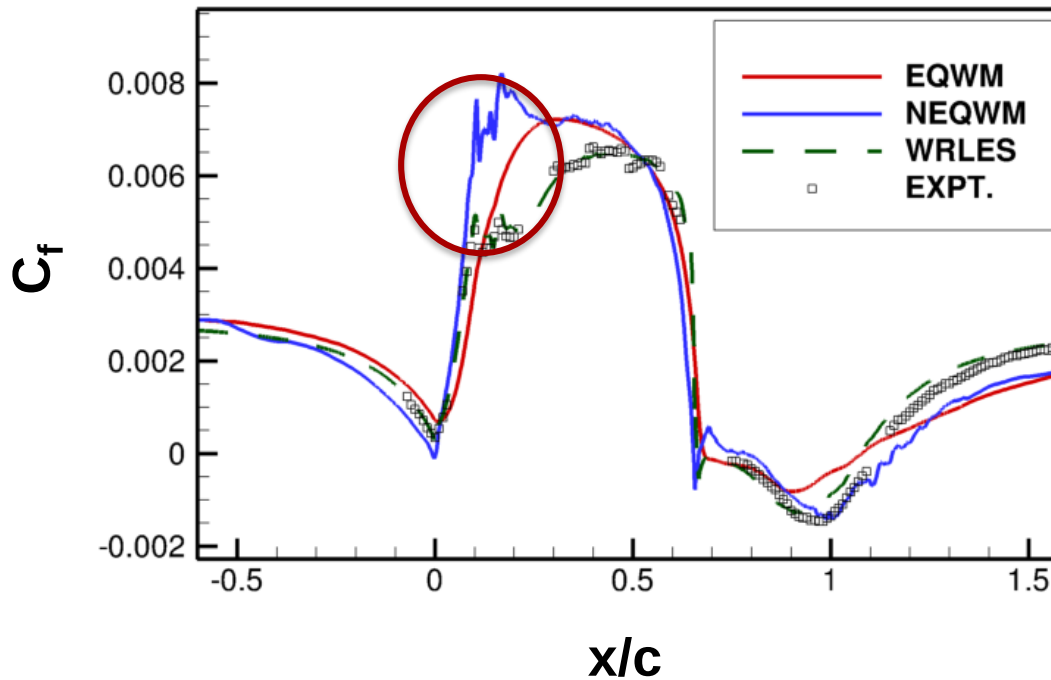




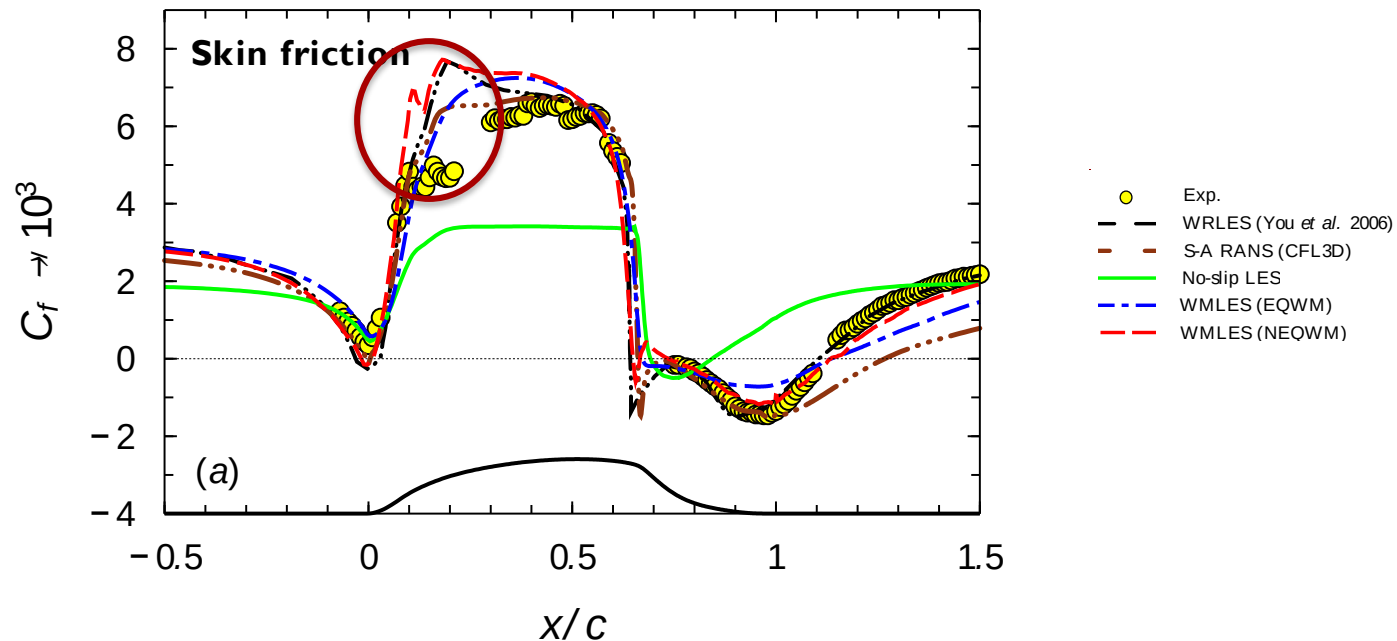


*Narasimha and Sreenivasan, "Relaminarization in Highly Accelerated Turbulent Boundary Layers," *JFM*, Vol. 61, 1973.

- Uses Charles Code (Cascade Technologies)
 - 2nd order low dissipation scheme, with unstructured grid
 - Equilibrium and nonequilibrium wall models
 - One to two orders more efficient compared to WRLES

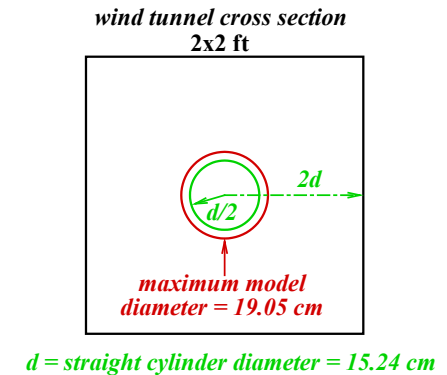
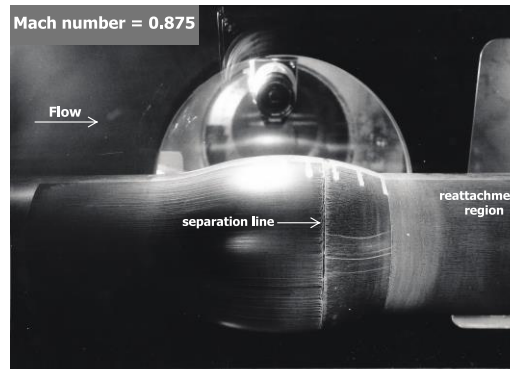


- Uses Charles Code (Cascade Technologies)
 - 2nd order low dissipation scheme, with unstructured grid
 - Equilibrium and nonequilibrium wall models
 - One to two orders more efficient compared to WRLES
 - Computations performed by Stanford-CTR



- WMLES for NASA Common Research Model (CRM) ongoing at Stanford-CTR under a NASA NRA

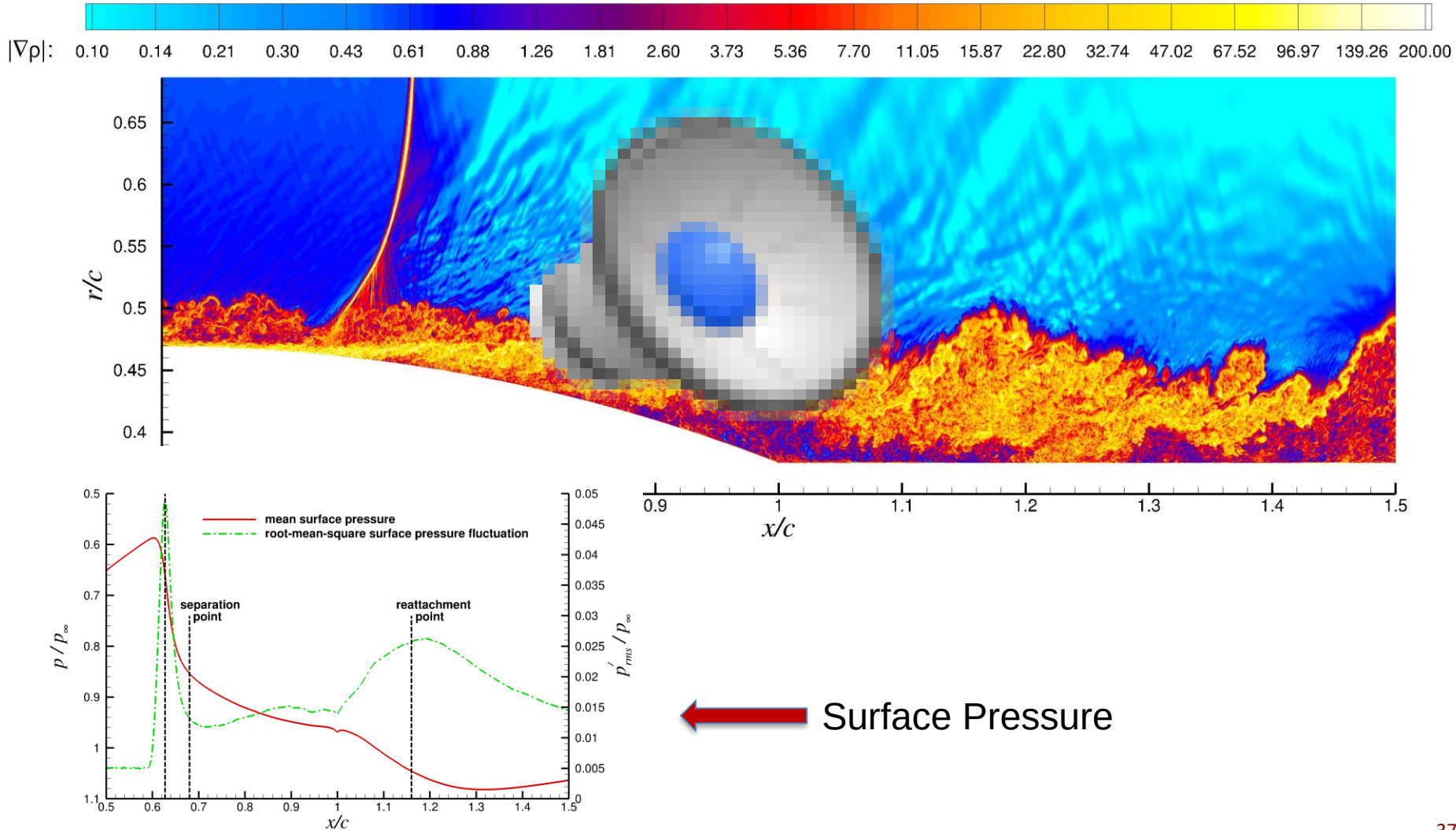
- Experimental set up
 - Bachalo and Johnson, AIAA J. 24(3), 1986.
 - $M = 0.875$
 - $Re_c = 2.763 \times 10^6$
 - $Re_\theta = 6600$



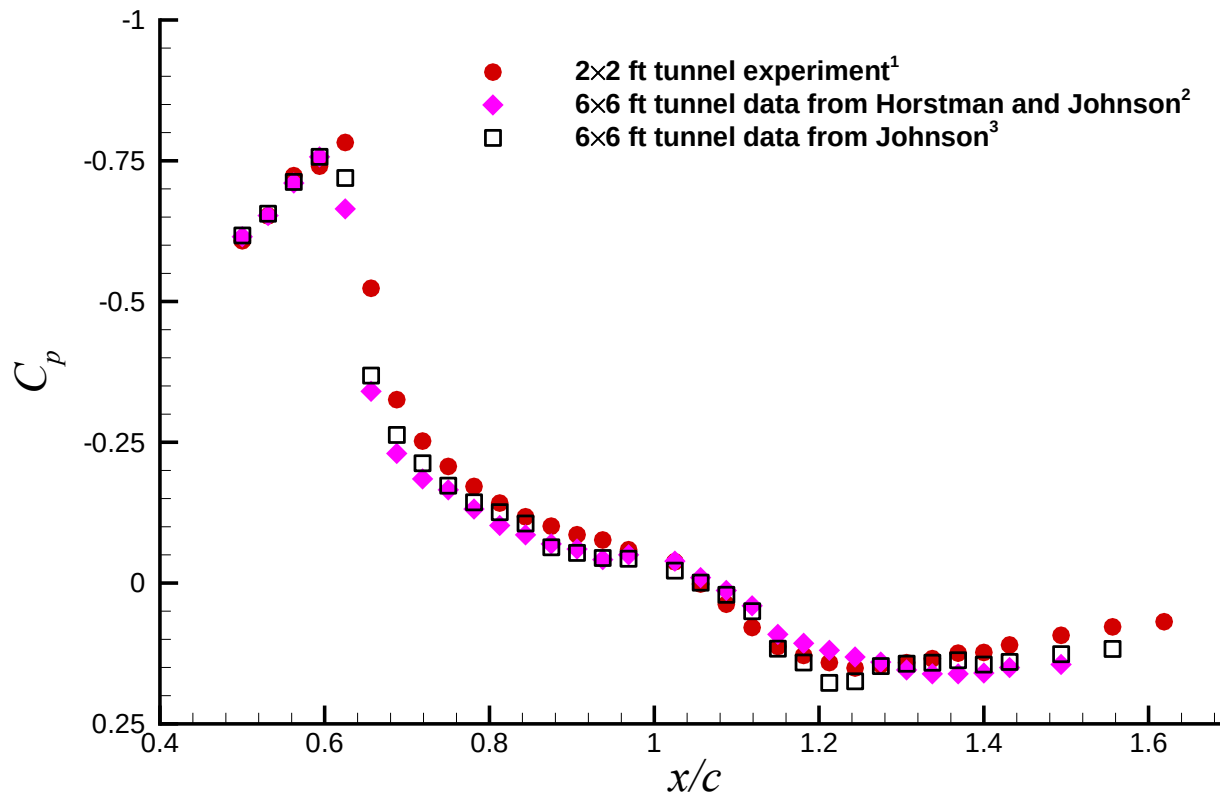
- Computations performed on Edison and Cori at NERSC (LBNL)
 - Edison: Cray XC30 with Intel Ivy Bridge nodes
 - 134K cores total, used up to 42K cores
 - Cori: Cray XC40 with Knights Landing (KNL) nodes
 - 660K cores total, used up to 170K cores
 - For 24 billion grid points and 10 chord flow time, about 18 days of run time
 - Computations assumed free stream BC at 10c
 - Uzun and Malik, AIAA Paper 2018-????



Density Gradient



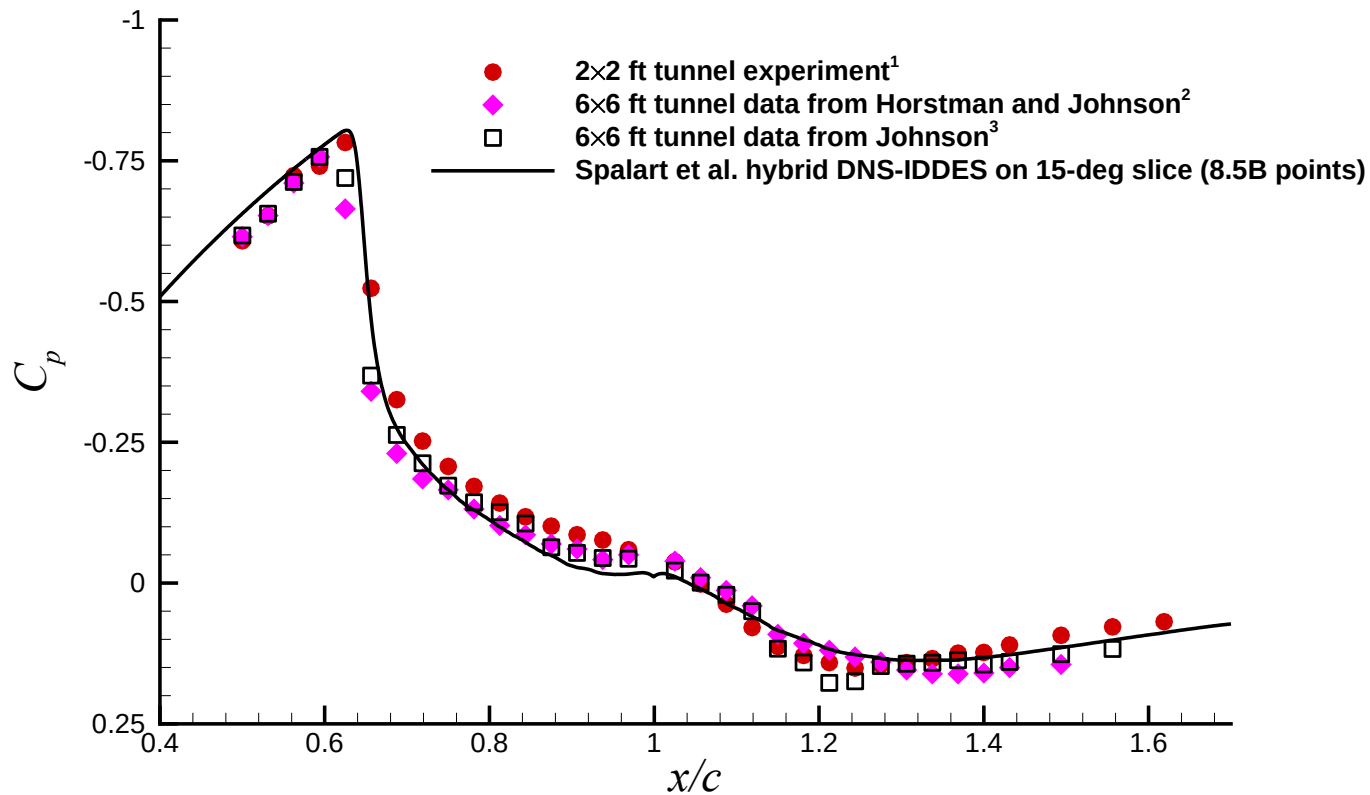
Surface Pressure Coefficient



¹Bachalo and Johnson, "Transonic, Turbulent Boundary-Layer Separation Generated on an Axisymmetric Flow Model," *AIAA Journal*, 1986, 24(3).

²Horstman and Johnson, "Prediction of Transonic Separated Flows," *AIAA Journal*, 1984, 22(7).

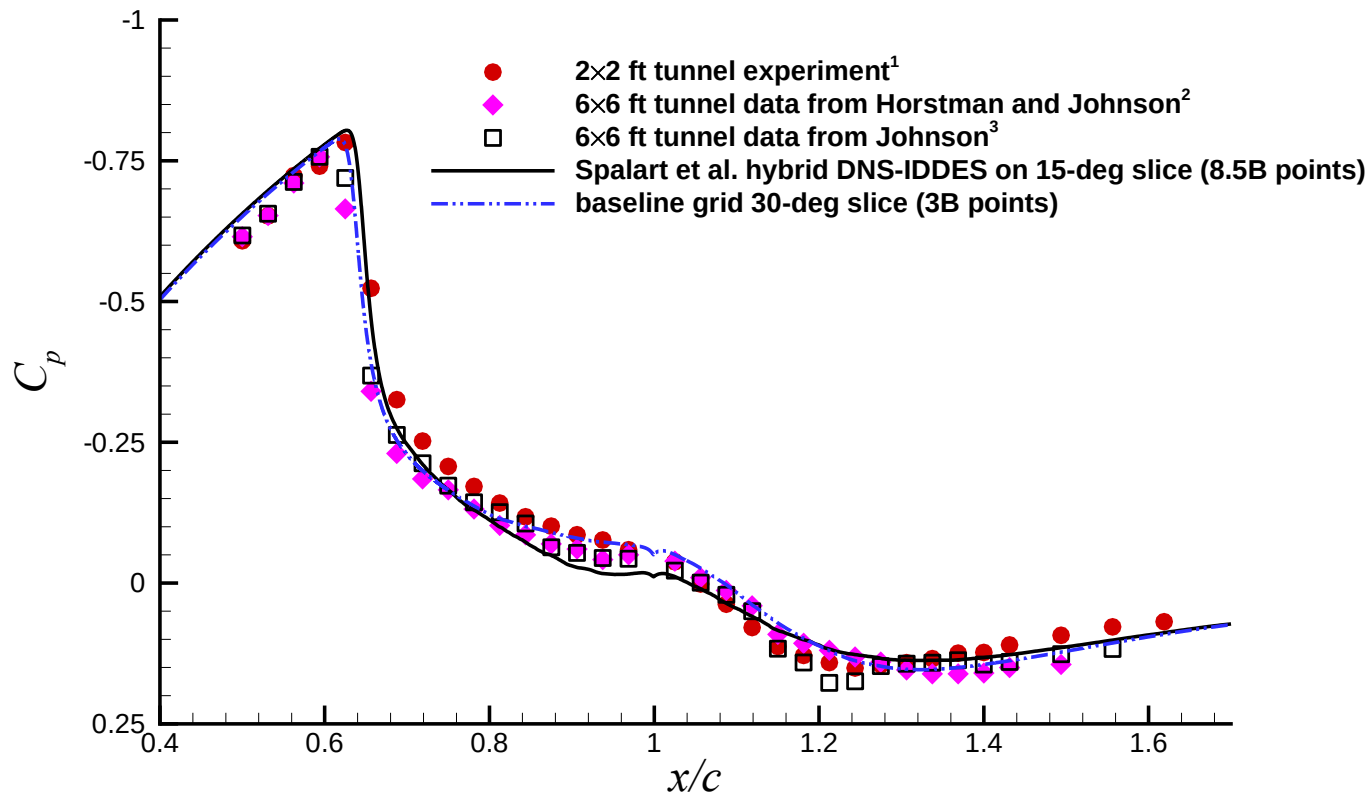
³Johnson, "Transonic Separated Flow Predictions with an Eddy-Viscosity/Reynolds-Stress Closure Model," *AIAA Journal*, 1987, 25(2).



¹Bachalo and Johnson, "Transonic, Turbulent Boundary-Layer Separation Generated on an Axisymmetric Flow Model," *AIAA Journal*, 1986, 24(3).

²Horstman and Johnson, "Prediction of Transonic Separated Flows," *AIAA Journal*, 1984, 22(7).

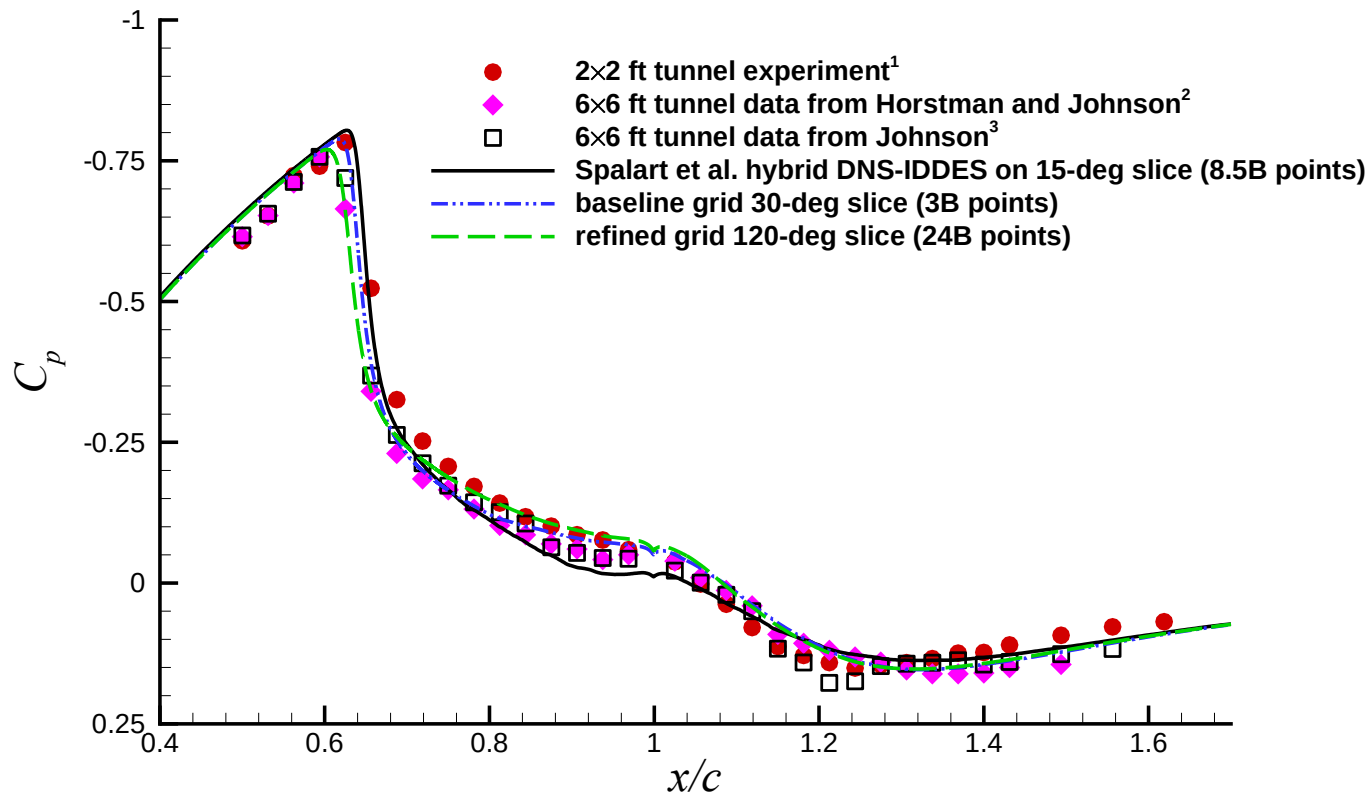
³Johnson, "Transonic Separated Flow Predictions with an Eddy-Viscosity/Reynolds-Stress Closure Model," *AIAA Journal*, 1987, 25(2).



¹Bachalo and Johnson, "Transonic, Turbulent Boundary-Layer Separation Generated on an Axisymmetric Flow Model," *AIAA Journal*, 1986, 24(3).

²Horstman and Johnson, "Prediction of Transonic Separated Flows," *AIAA Journal*, 1984, 22(7).

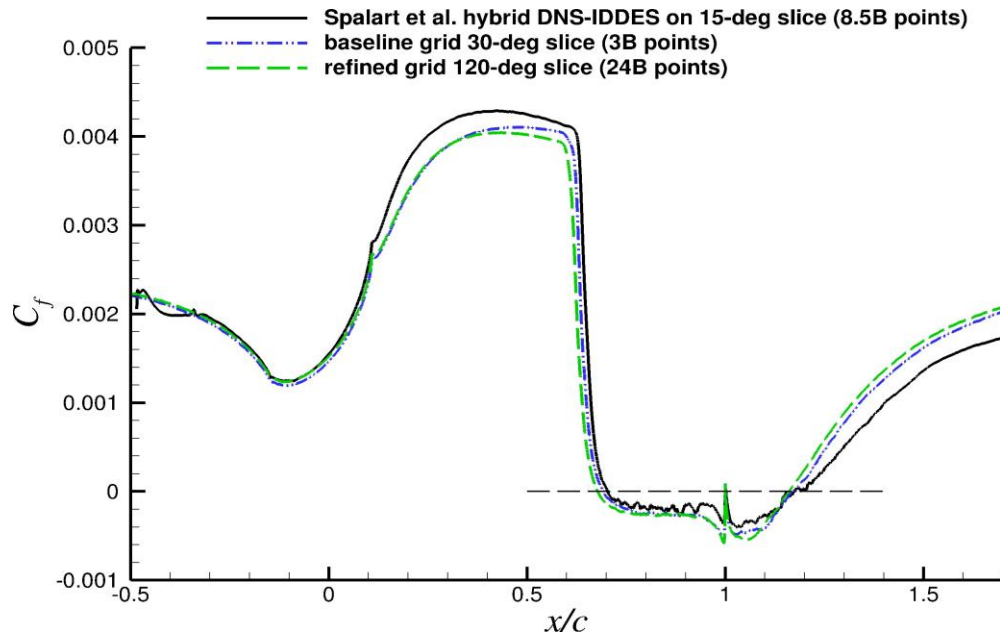
³Johnson, "Transonic Separated Flow Predictions with an Eddy-Viscosity/Reynolds-Stress Closure Model," *AIAA Journal*, 1987, 25(2).



¹Bachalo and Johnson, "Transonic, Turbulent Boundary-Layer Separation Generated on an Axisymmetric Flow Model," *AIAA Journal*, 1986, 24(3).

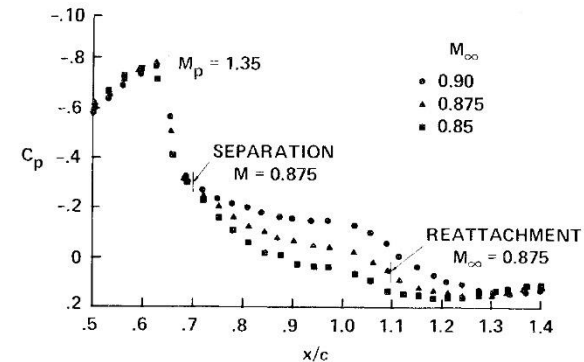
²Horstman and Johnson, "Prediction of Transonic Separated Flows," *AIAA Journal*, 1984, 22(7).

³Johnson, "Transonic Separated Flow Predictions with an Eddy-Viscosity/Reynolds-Stress Closure Model," *AIAA Journal*, 1987, 25(2).

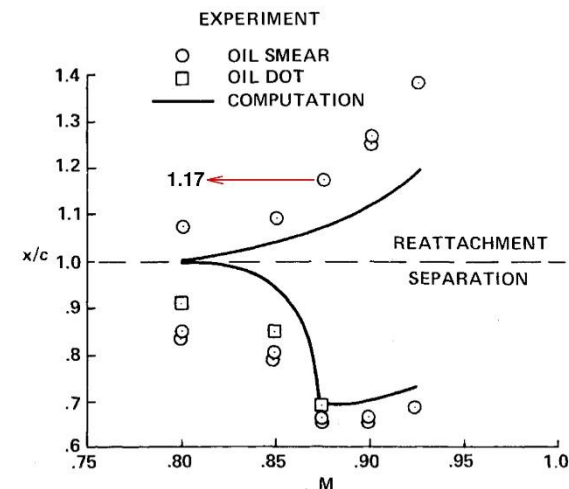


- Computed separation location, $x/c = 0.68 - 0.69$
- Computed reattachment location, $x/c = 1.16 - 1.17$

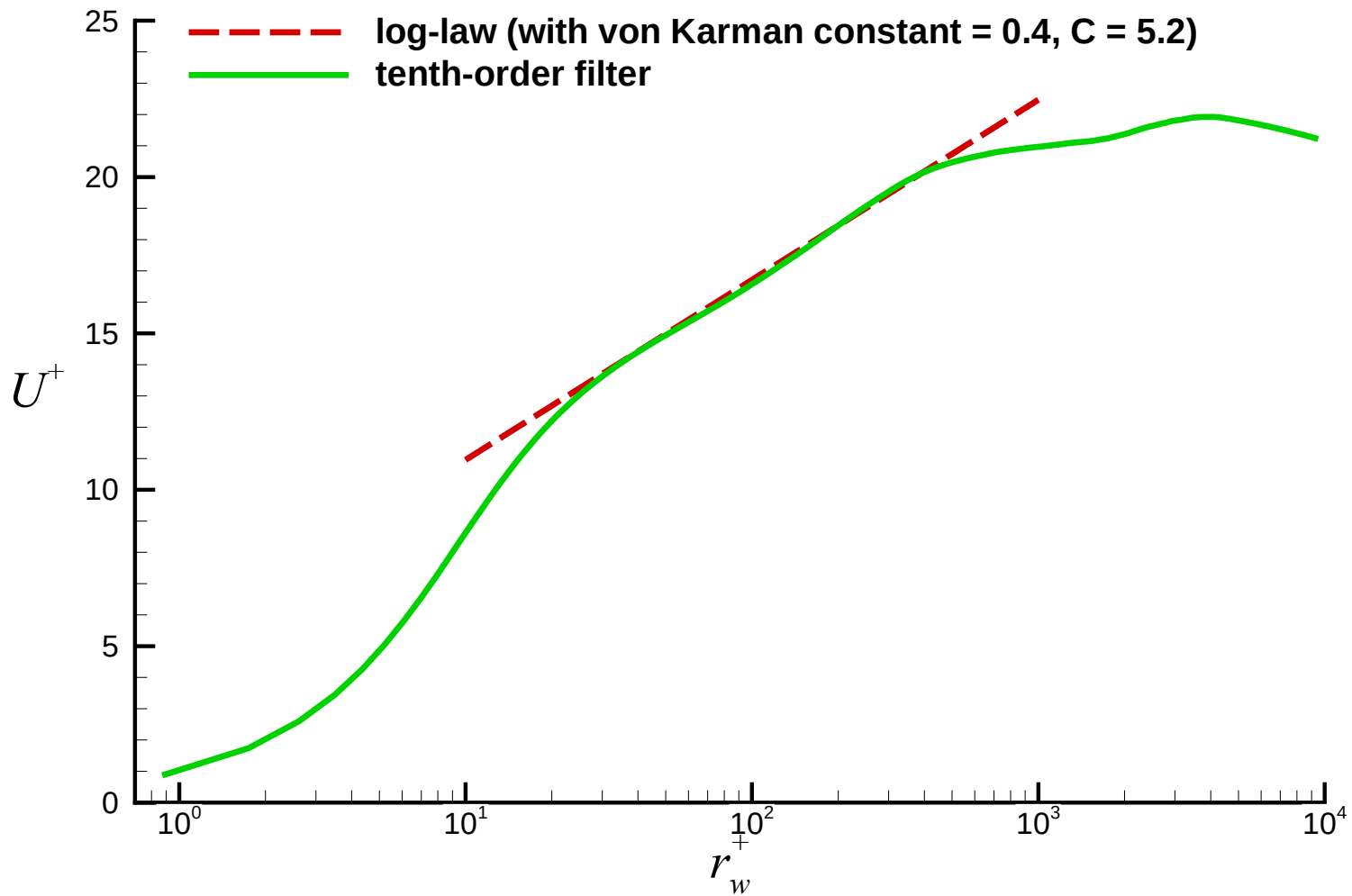
Small tunnel



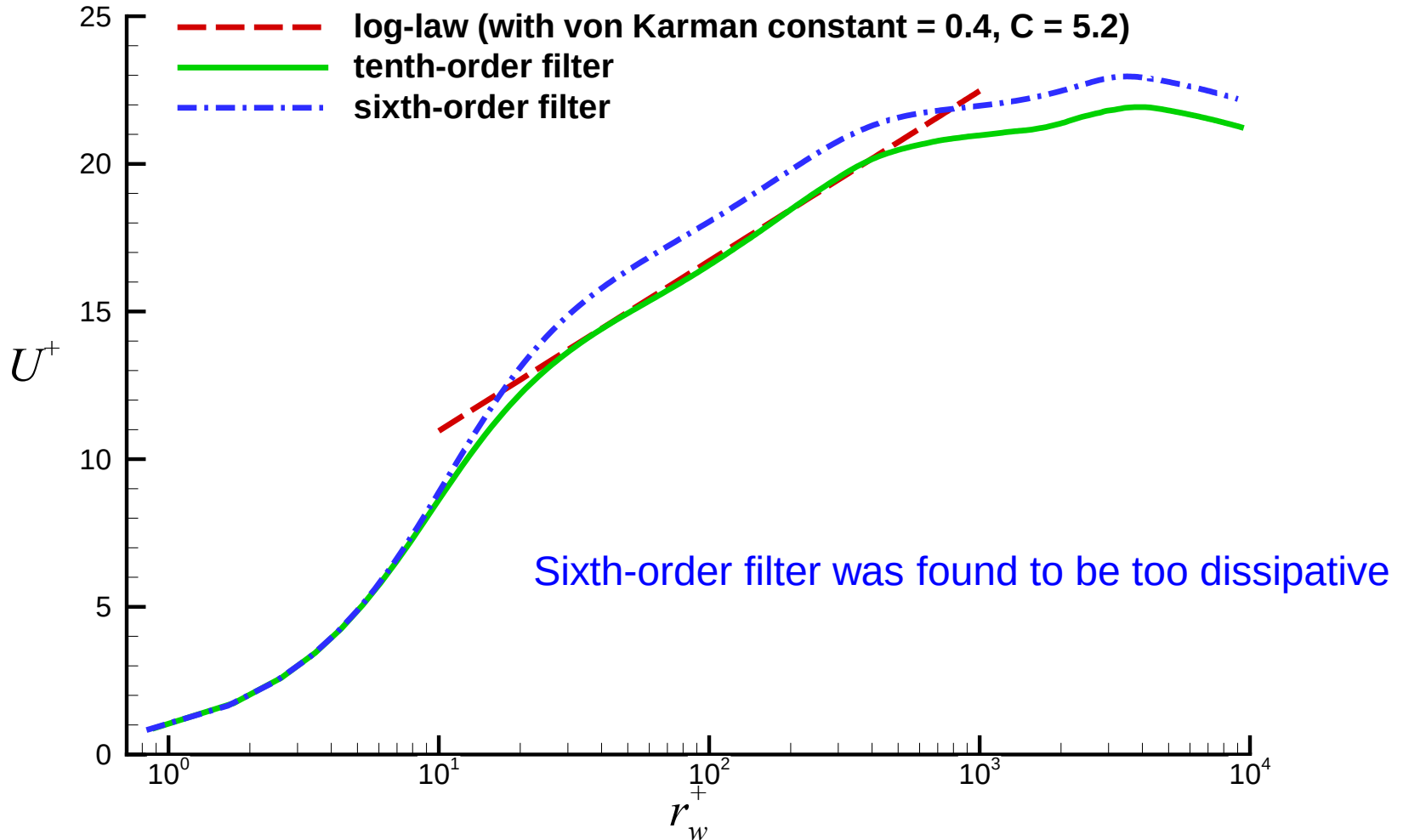
Large tunnel



Velocity Profiles in Wall Units at $x/c=0.5$



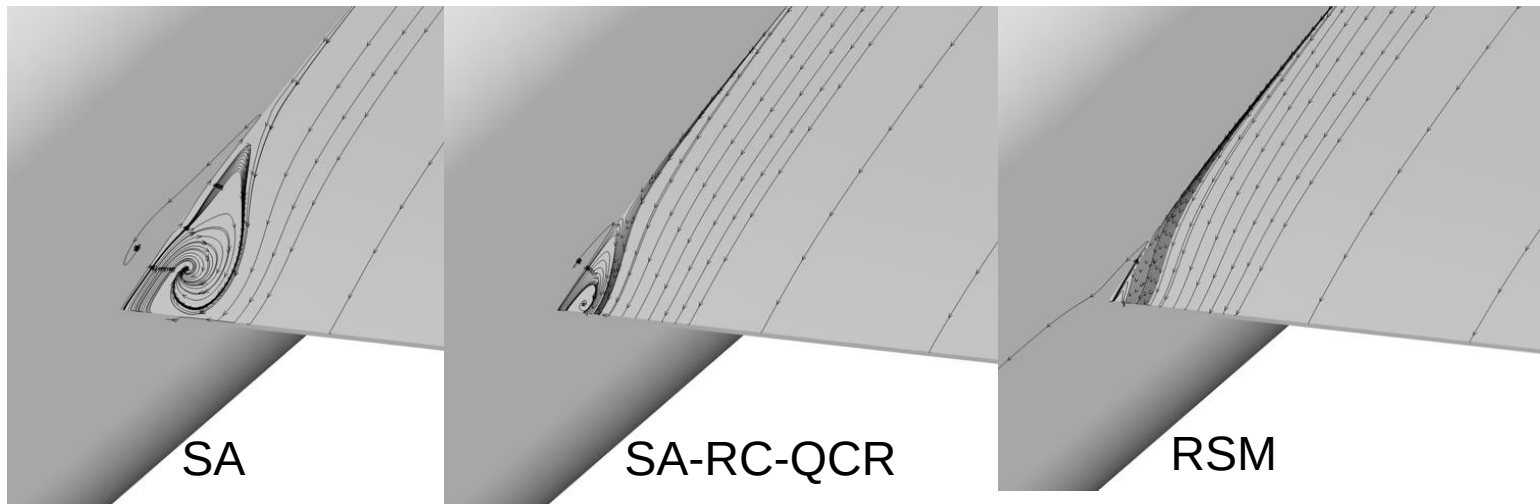
Velocity Profiles in Wall Units at $x/c=0.5$



- **NASA should lead efforts to develop and execute **integrated experimental testing and computational validation campaigns****
 - Experiments to provide data for development of advanced turbulence models/prediction capability is a critical need
- **Requirements for a “CFD Validation-Quality” experiment**
 - Experiment should include the measurements of all information, including boundary conditions, geometry information, and quantification of experimental uncertainties, necessary for a thorough and unambiguous CFD validation study See, e.g., Aeschliman & Oberkampf (AIAA J 36(5):733-741, 1998)
- **Experiments currently underway**
 - Junction Flow
 - Turbulent Heat Flux
 - Shock-Wave/Boundary-Layer Interaction
 - Smooth Body Flow Separation (NRA: U Notre Dame)
 - 2D ramp with adjustable ceiling contour
 - Mean velocity and turbulence stress profiles, oil film interferometry for shear stress measurements
 - Compressible Mixing Layer (NRA: UIUC)
 - $M_c = 0.19, 0.37, 0.54, 0.74, 0.86$ and 1
 - Stereo PIV (mean velocity and turbulence quantities), Schlieren, hot-wire measurements

Juncture Flow Experiment - 1

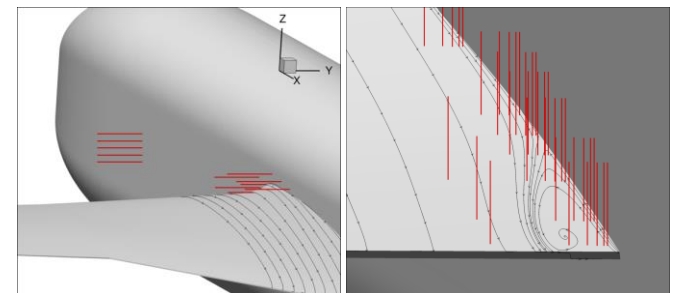
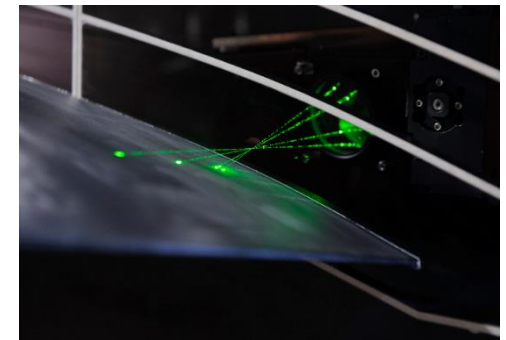
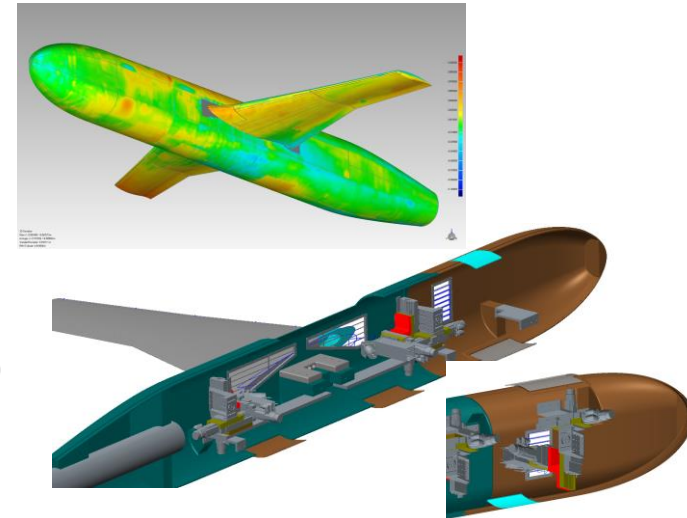
- Geometric junctures (corners) are common on aircraft
 - CFD predictive capability is currently uncertain
 - For example, in Drag Prediction Workshops (DPW), participants predicted a wide range of wing-body corner separation bubble sizes (none to very large)
- Because of the high degree of uncertainty in the CFD predictions, relevant separated corner flow experiments focused specifically on obtaining high-quality data for CFD validation are needed



Example juncture flow computations using 3 different turbulence models

Juncture Flow Experiment - 2

- Unique on-board Laser-Doppler Velocimetry (LDV) system specifically designed for measuring the near-wall juncture region flow field through windows
- Consistent CFD and wind tunnel experiment
- Data
 - Off-body velocities and moments (LDV and some PIV)
 - Model surface pressures, unsteady surface pressures and unsteady surface shear stress measurements
- Boundary conditions
 - Over the body: measurements at front window
 - Test section wall pressures/wall rakes
 - Infrared thermography for transition detection and trip effectiveness
 - Tunnel inflow (measured with Boeing's QWSS in separate tests)
- Geometry
 - Laser scans of tunnel walls, model, and mast/sting/cabling details
 - Photogrammetry to document wing deflections
- Documentation
 - Special session at AIAA SciTech 2019



Notional Measurement Locations

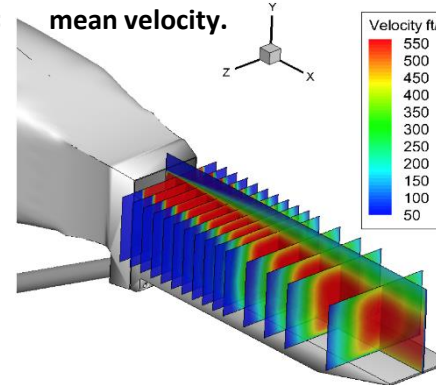
Turbulent Heat Flux (THX) Experiment

- Current CFD uses inadequate models when turbulent heat transport is important
 - Propulsion flow applications with wall cooling (e.g., turbine blades and exhaust nozzles)
 - Measure $u, u', T, T', u'T'$
- Raman scattering measurements of mean and *rms* flow temperature and PIV velocity measurements used to characterize a single hole injection cooling flow across a range of Mach numbers and blowing ratios
- Embedded thermocouples used to compute film effectiveness
 - decreases with Mach number
- Unique set of flow/heat measurement data
 - NASA TM - 2018-219739
- RANS CFD used to design the experiment
 - Scale resolving simulations planned
- THX Phase IV (3 sets of multiple cooling holes) just completed

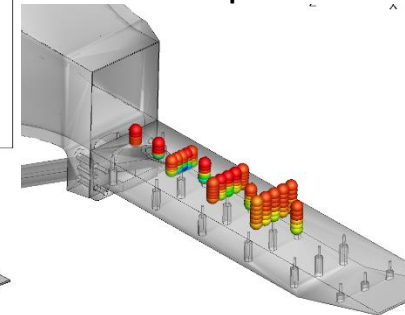
THX Phase 3 nozzle and plate (single injector hole) installed in NASA GRC AAPL facility.



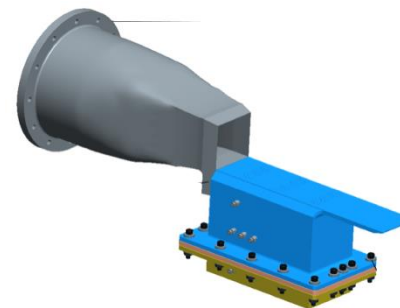
PIV measurements of mean velocity.



Raman measurements of mean temperature.

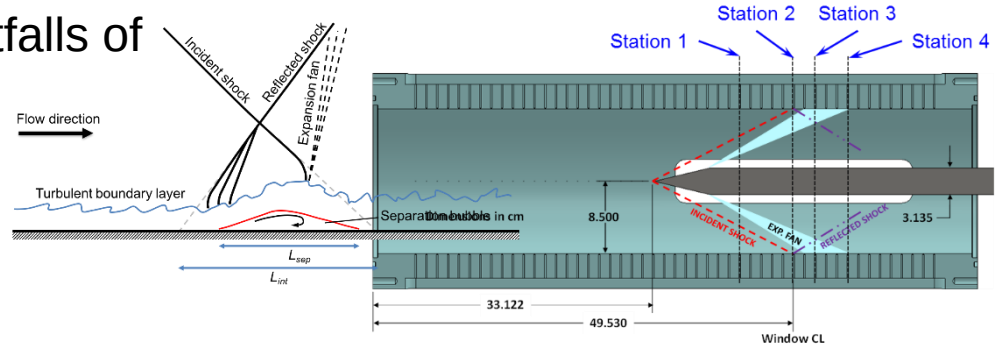


THX Phase 4 plate concept (3 arrays of cooling holes).

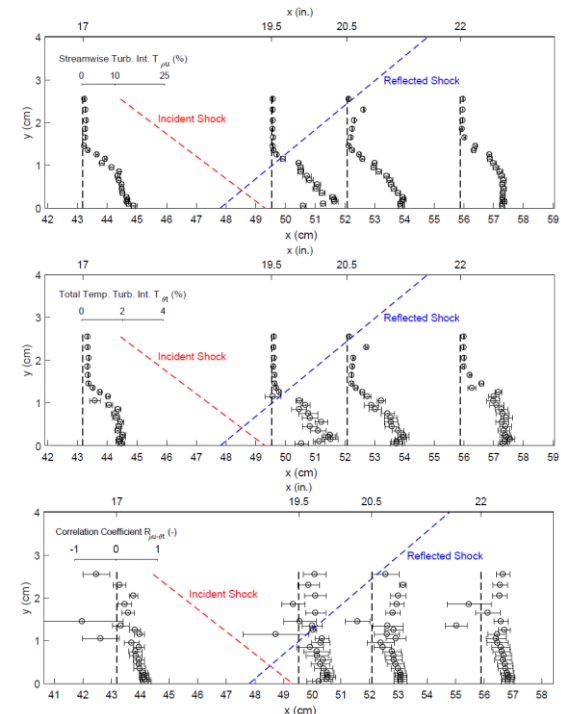


Shock-Wave/Boundary-Layer Interaction Experiment

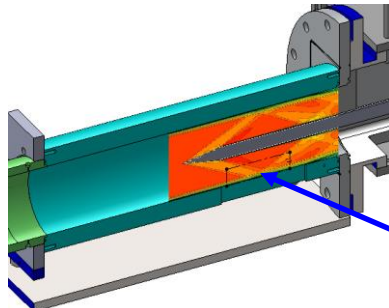
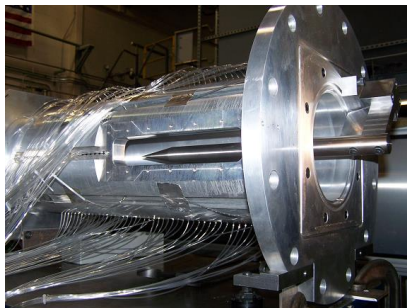
- Axisymmetric geometry to avoid pitfalls of rectangular configurations
 - Mach 2.5 developing pipe flow
 - Cone shock interacts with pipe BL
 - 10.0° cone, fully attached
 - 13.5° cone, incipient separation
 - 16.0° cone, separated flow
 - 2D in the mean, but 3D created by offsetting cone-cylinder
- Redundant measurements
 - Conventional steady-state and dynamic pressure taps and probes
 - Hot-Wire Anemometry (mean and turbulence)
 - Particle Image Velocimetry (PIV)
 - Dynamic Pressure Sensitive Paint (PSP)
 - Dynamic Surface Shear-Stress Film (S3F)



Cross-Section Depicting Interaction Region



Resulting quantities of Streamwise Turbulence Intensity (top), Total Temperature Turbulence Intensity (middle) and Correlation Coefficient (bottom)

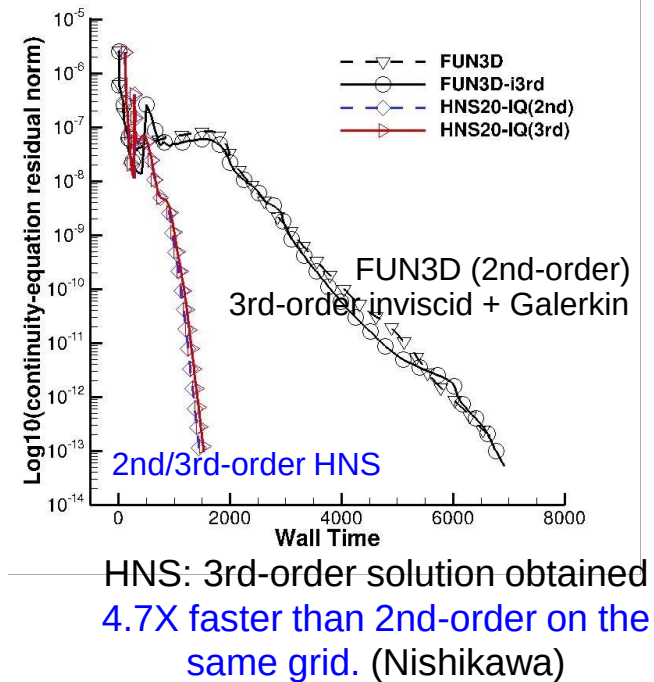


Region of Interest

- **Turbulence Modeling and Simulations**
 - Modified k-kL model(s) for wall-bounded and free shear flows
 - Lattice-Boltzmann method for unsteady flow simulation
 - Several hybrid RANS/LES approaches
 - Turbulent hot jets

- **Numerical Methods**
 - Stabilized finite-elements (SUPG)
 - Hyperbolic Navier-Stokes (HNS)
 - Kinetic energy preserving schemes
 - Space-time flux reconstruction
 - HANIM for convergence acceleration
 - Uncertainty quantification

- **Transition Prediction and Modeling**
 - LASTRAC: Langley Stability and Transition Analysis Code
 - Analysis and control of hypersonic boundary layer transition
 - Supersonic transition experiments with effect of roughness
 - Crossflow transition experiments to study the effect of steps





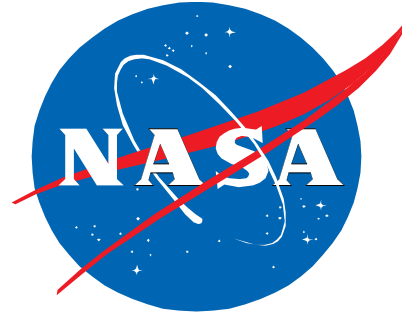
- **CFD Prediction Error Assessment Workshop, March 2018**
 - RSM did not improve predictions for RCA standard test cases
 - It might for the juncture/corner flow
 - k-kL based model(s) a bright spot!
- **CFD 2030 Decision Gate in 2019**
 - Fully shift development from RANS to hybrid RANS/LES methods, if RANS/RSM fails to predict flow separation
- **U Michigan-NASA Turbulence Modeling Symposium, July 2017**
 - NASA/TM-2017-219682
 - RANS development needed → Data-driven modeling, using machine learning tools
- **AIAA Turbulence Model Benchmarking Working Group Position Paper**
 - Recommends not to phase out of RANS research, and keep a balance between RANS and hybrid methods
- **AIAA CFD 2030 Integration Committee**



Concluding Remarks



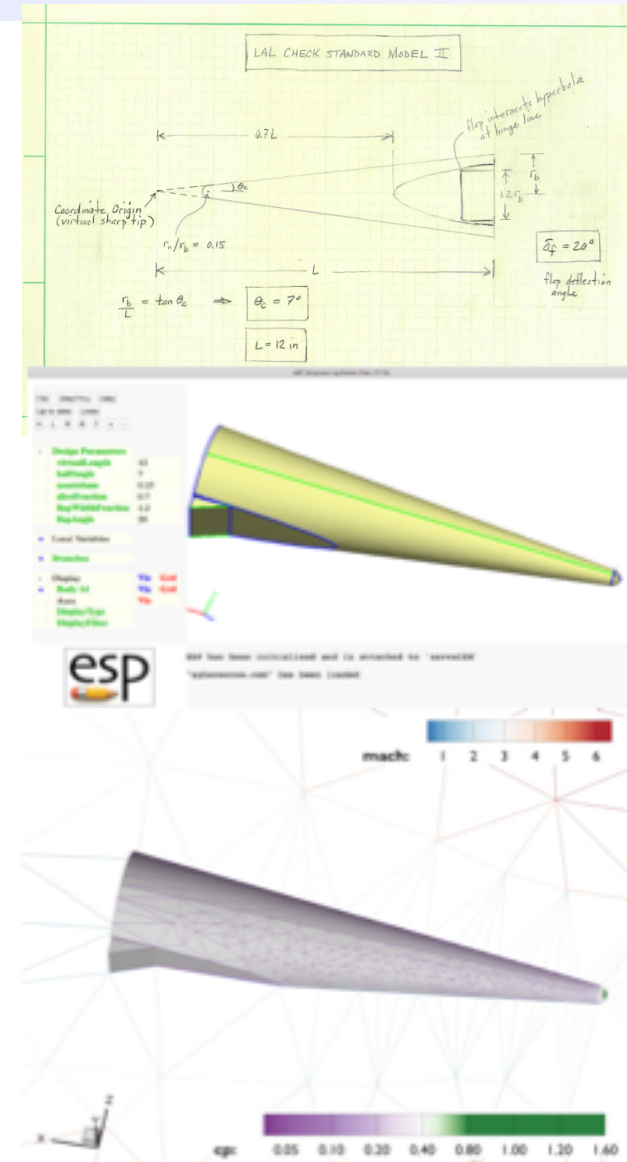
- **The CFD Problem is Hard!**
 - But, making steady progress toward RCA research goals
 - Robustness
 - Cost/efficiency
 - Accuracy
- **On the way to Certification by Analysis, to significantly reduce flight testing**
 - 100x speed up, with algorithms and hardware
 - Accurate prediction of smooth body flow separation, CL_{max} , buffet and aeropropulsion performance
- **New, high-quality CFD validation data being generated**
 - Will be shared with CFD community at large





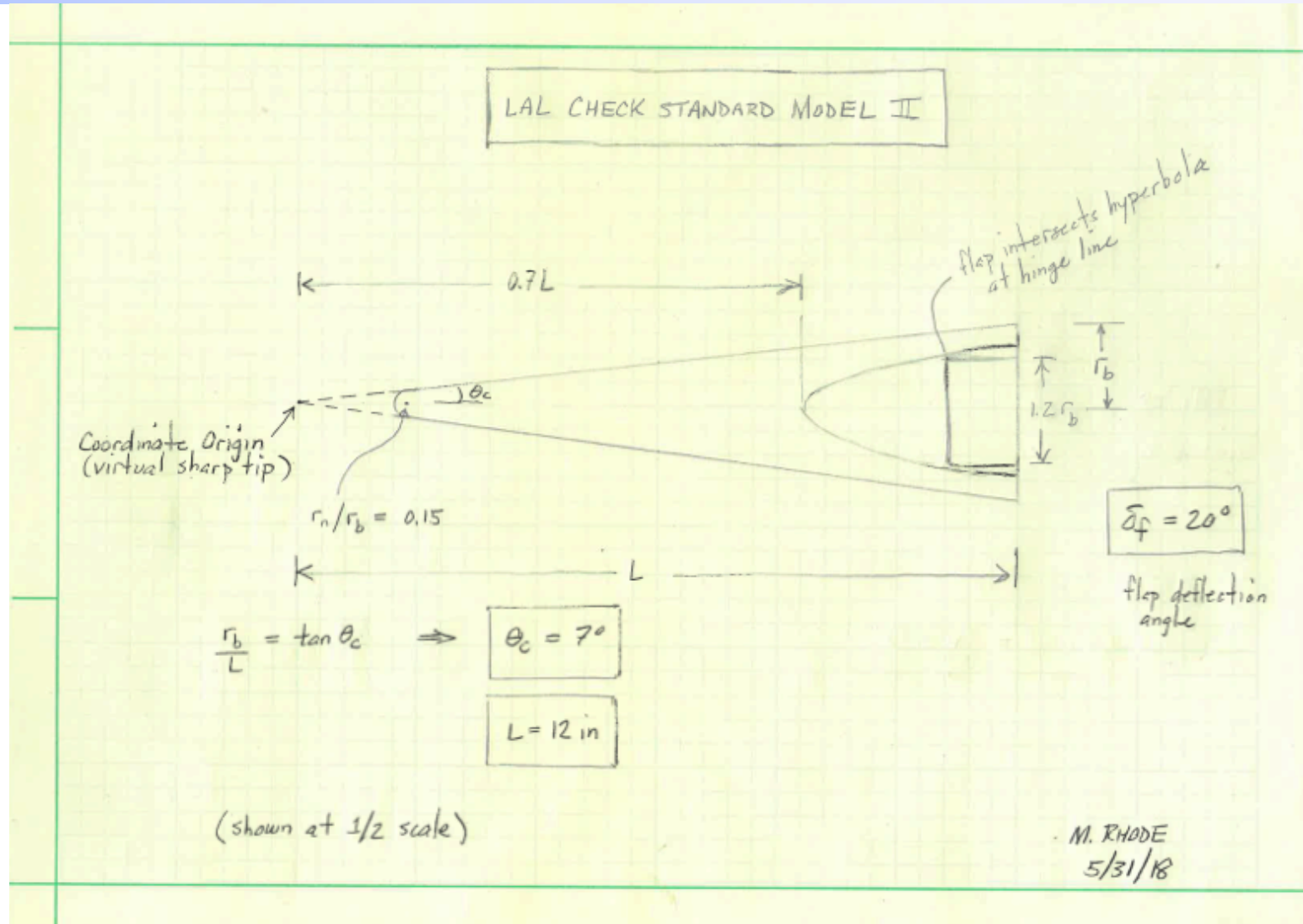
Riding the Vector Automated Grid Modeling and Adaptation

- Concept to solution in hours, as opposed to weeks.
- User moved from a sketch to an error-adapted CFD solution without ever seeing the grid.
 - Solution automatically adapted to shocks, boundary layers, and wakes.
- CFD Vision 2030 Study states, “ultimately **the mesh generation process should be invisible to the CFD user or engineer**. Given a suitable geometry representation and a desired level of solution accuracy, a fully automated meshing capability would construct a suitable mesh and adaptively refine this mesh throughout the solution process with minimal user intervention until the final accuracy levels are met. This process allows the user to focus on the final solution without concern for the construction and maintenance of the underlying mesh.”
- NASA presently under-emphasizes CFD automation in formulation of research and development goals.





Automated geometry modeling and adaptation





Automated geometry modeling and adaptation

ESP (Engineering Sketch Pad, V1.13)

File StepThru Help

Up to date Undo

H L R B T + -

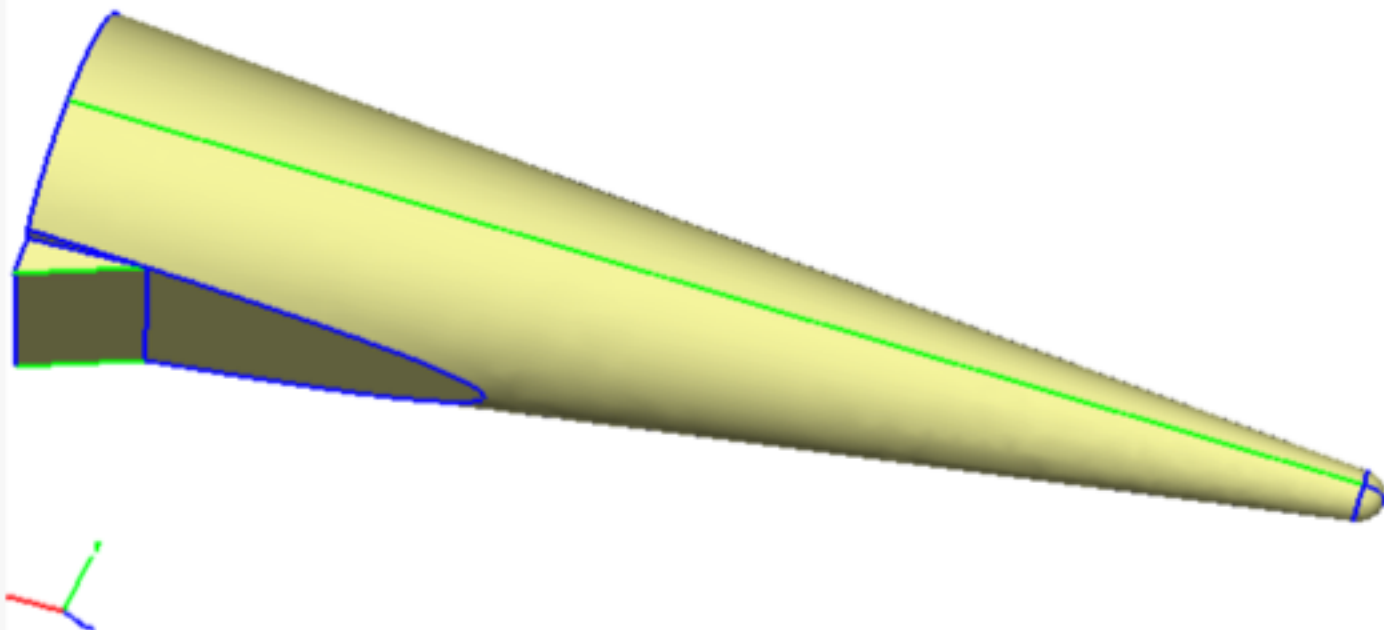
- Design Parameters

virtualLength	12
halfAngle	7
nosetobase	0.15
sliceFraction	0.7
flapWidthFraction	1.2
flapAngle	20

+ Local Variables

+ Branches

- Display	Viz	Grd
+ Body 14	Viz	Grd
Axes	Viz	
DisplayType		
DisplayFilter		

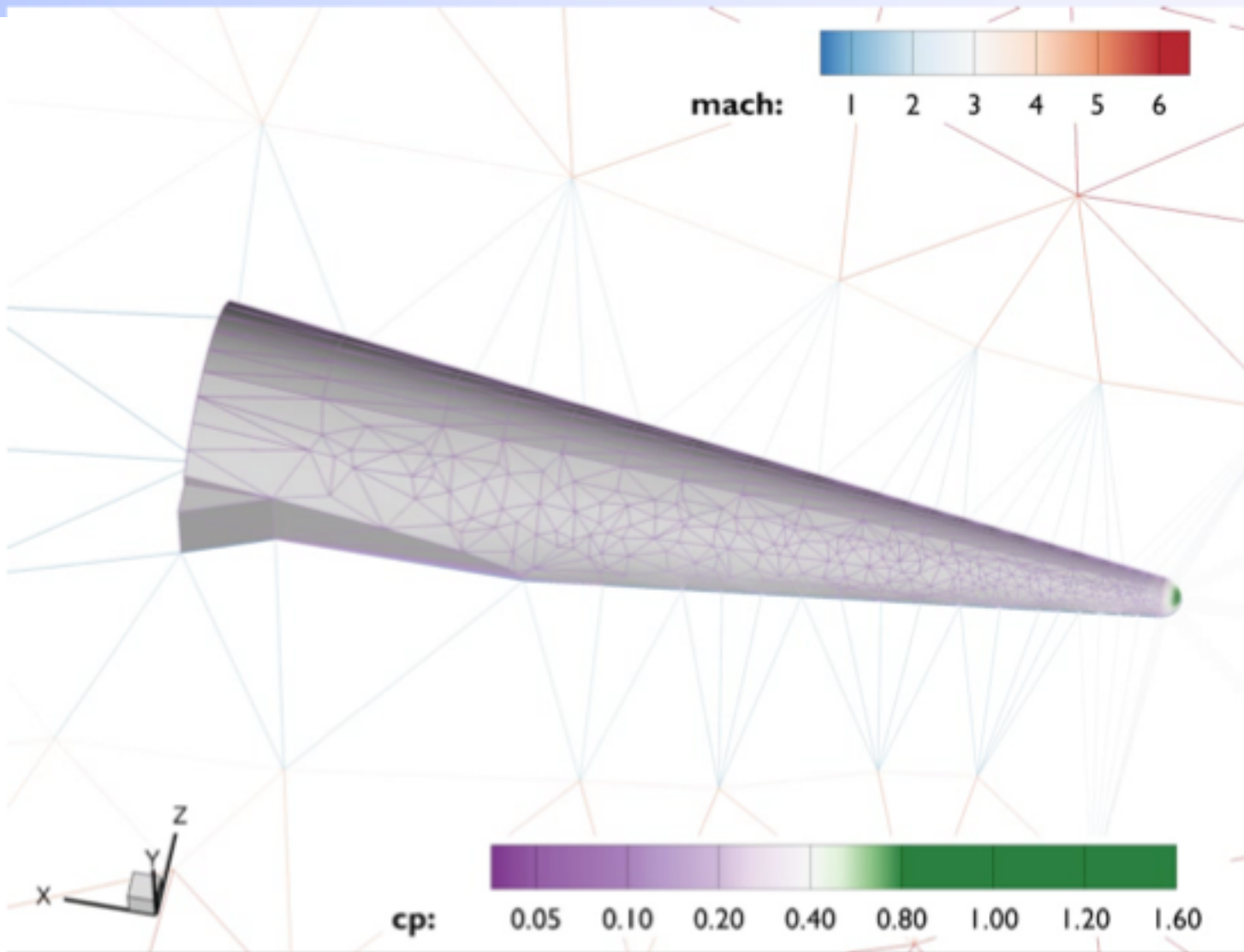


ESP has been initialized and is attached to 'serveCSM'

"spherecone.csm" has been loaded



Automated geometry modeling and adaptation





Automated geometry modeling and adaptation

- Time-to-solution significantly reduced (weeks/days to hours)
- Geometry to viscous solution w/o explicit grid generation step
- Grid adaptation tied to L2 Mach interpolation error
- Standardized, repeatable workflow process
- Toward moving viscous CFD from artisan product to commodity
- CFD Vision 2030 Study states, *“ultimately the mesh generation process should be invisible to the CFD user or engineer. Given a suitable geometry representation and a desired level of solution accuracy, a fully automated meshing capability would construct a suitable mesh and adaptively refine this mesh throughout the solution process with minimal user intervention until the final accuracy levels are met. This process allows the user to focus on the final solution without concern for the construction and maintenance of the underlying mesh.”*
- As William Gibson says, *“The future is already here — it's just not very evenly distributed.”*