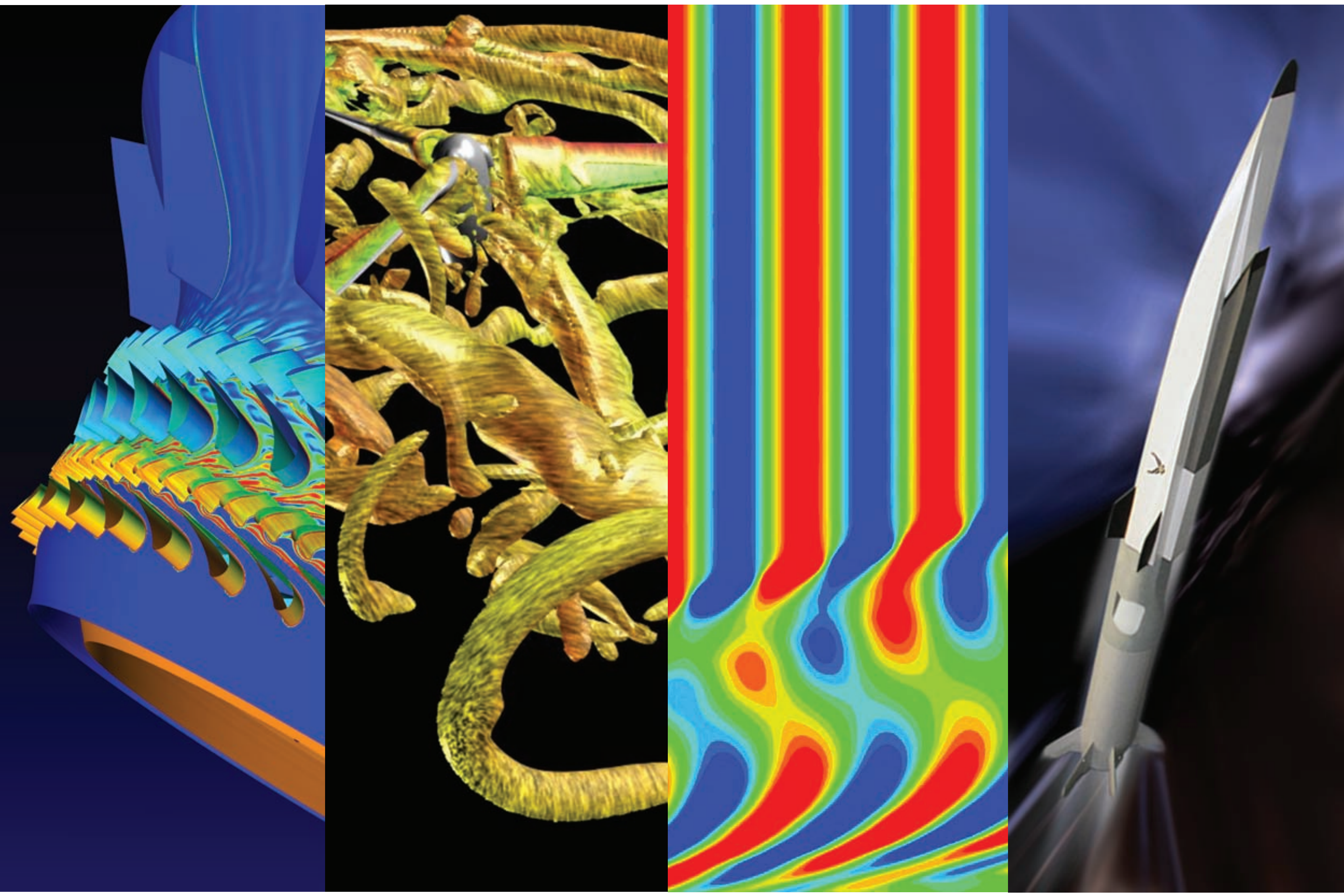


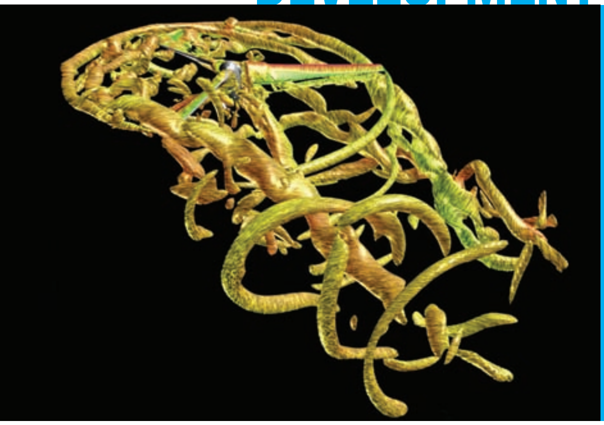
MISSION DIRECTORATE

AERONAUTICS RESEARCH

The Aeronautics Research Mission Directorate conducts cutting-edge, fundamental research in traditional and emerging disciplines to help transform the nation's air transportation system, and to support future air and space vehicles. Our goals are to improve airspace capacity and mobility, improve aviation safety, and improve aircraft performance while reducing noise, emissions, and fuel burn. Our world-class capability is built on a tradition of expertise in aeronautical engineering and its core research areas, including aerodynamics, aeroacoustics, materials and structures, propulsion, dynamics and control, sensor and actuator technologies, advanced computational and mathematical techniques, and experimental measurement techniques.

DR. JAIWON SHIN
Associate Administrator
http://www.aeronautics.nasa.gov

HIGH-RESOLUTION NAVIER-STOKES CODE DEVELOPMENT FOR ROTORCRAFT AEROMECHANICS



AERONAUTICS RESEARCH MISSION DIRECTORATE

NEAL M. CHADERJIAN
NASA Ames Research Center
(650) 604-4472
Neal.Chaderjian@nasa.gov

Figure 1: Improved rotor tip vortex resolution using high-order spatial accuracy and grid adaption.

Project Description: Helicopters and tiltrotor aircraft provide many crucial services, including emergency medical and rescue evacuation, security patrols, offshore oil platform access, heavy-lift capability, and military operations. Some of the phenomena associated with rotorcraft flight include aerodynamic performance and noise, vortex wakes generated from the rotating blades, and rotor blade flexibility and vibration. Blade-Vortex Interaction (BVI) also occurs when a rotor blade interacts with, and in some cases slices through, the vortices generated by other rotor blades. This not only affects the aerodynamic performance of the vehicle, but it is responsible for much of the noise generated by the rotor blades.

Many of these phenomena are poorly understood and difficult to accurately predict. One of the goals of the Subsonic Rotary Wing (SRW) Project, part of NASA's Fundamental Aeronautics Program, is to develop improved physics-based computational tools to address these issues. The long-term objective of this effort is to develop a more accurate aeromechanics computational tool that couples computational fluid dynamics (CFD) and computational structural dynamics (CSD) flow simulation tools with a rotor blade trim code. A trim code prescribes blade motions such that the resultant forces and moments are in balance for a desired flight condition.

This project combines the efforts of several NASA Ames researchers into two broad categories: code development and application support. The latter uses current CFD/CSD/trim capability to support wind tunnel tests. This report focuses on the code development portion of this work, where the primary objective is to improve accuracy of the OVERFLOW-2 Reynolds-averaged Navier-Stokes (RANS) flow solver and to explore new methods of unsteady flow visualization.

Relevance of Work to NASA: This code development effort directly supports the SRW Project's goal to conduct long-term, cutting-edge research in the core competencies of the subsonic rotary wing regime. More specifically, the focus is on improving our prediction capability in rotorcraft aeromechanics through research and development of physics-based, high-fidelity computational tools. Specific project deliverables and milestone metrics are met by validating these new tools with wind tunnel measurements.

Computational Approach: Flow simulations have been carried out for an isolated V-22 Osprey rotor. This simple, rigid, three-blade rotor geometry, along with the spinner hub, is an ideal case for exploring new methods for improving simulation accuracy of the generated vortex wake. A Rotor Grid Assistant (RGA) script is used to automate the generation of overset structured computational grids and OVERFLOW-2 input files so that different grid resolutions can be readily explored. Current state-of-the-art CFD methods use algorithms that are second-order accurate in time, third-order accurate in space, and grid resolution in the vortex wake region with grid spacing on the order of one vortex core diameter. Total grid size typically consists of tens of millions of grid points. Straightforward refinement of the grid in the wake region (to achieve ten grid cells across a vortex core diameter) would result in a grid system consisting of several billion grid points. This is not practical with current supercomputers, so the approach adopted here is to improve OVERFLOW-2's spatial accuracy up to sixth order, and use grid adaption to locally improve the vortex wake resolution by four to six times. This will result in a grid system that consists of a few hundred million grid points rather than billions of grid points. Since rotor flows are inherently unsteady, new concurrent visualization methods have been used to identify flow structures and determine cause and effect for predicted quantitative values.

Results: Preliminary results have been obtained for a V-22 isolated rotor in hover with a 14° collective and a rotor tip Mach number of $M_{tip} = 0.625$. The OVERFLOW-2 CFD code was modified to include up to sixth-order spatial accuracy and grid adaption using higher-resolution Cartesian grids embedded into the uniform coarser Cartesian background grid.

A sensor function based on vorticity magnitude is used to identify vortex cores and position the embedded Cartesian grids to better resolve these vortices. Figure 2 shows the predicted vorticity magnitude contours using second- and fifth-order spatial accuracy. It is also apparent that the vortex strength is much stronger using the fifth-order method. Second-order methods cause too much dissipation and dispersion errors. Figure 2 also shows an even greater improvement in the vortex strength using a third-order accurate method with Cartesian grid adaption.

It is anticipated that combining grid adaption with high-order spatial accuracy will improve overall accuracy of the CFD code while controlling the computer time required for a solution. Figure 1 shows a concurrent visualization of the V-22 rotor in forward-descending flight, where the advance ratio is 0.1 and the angle of descent is 6º. The vortex wake is visualized using iso-surfaces based on the Q-criterion (the second invariant of the velocity gradient tensor). Strong BVI is evident as the rotor blades slice through the rotor tip vortices of other blades. The vortices also roll up, forming two super-vortices similar to wing-tip vortices found on fixed-wing aircraft. A time-dependent animation produced during this project reveals the complex nature of the flow. The NASA Advanced Supercomputing (NAS) Division's Visualization group added functionality to the OVERFLOW-2 flow solver so that visualization extracts were written out at every time step, while the solution was evolving in time. The old paradigm of writing out the grid and solution files every 20 or so time steps for later post-processing would result in tens of terabytes of disk usage.

A texture mapping technique is also employed to indicate the instantaneous velocity field on the vortex iso-surfaces.

Role of High-End Computing: The NAS facility provides state-of-the-art computational resources needed to address this compute-intensive problem. The forward flight case was run on the Columbia supercomputer using 20 million grid points, and required 11 hours of wall-clock time per revolution using 64 processors. A hover case was run on the Pleiades supercomputer using 150 million grid points, and required 17 hours of wall-clock time per revolution using 512 processor-nodes. A rotor simulation typically takes 10 revolutions from impulsive start conditions to achieve dynamic equilibrium.

Future: During the next 18 months, further improvements to the grid-adaption method will be implemented. Furthermore, a Subsonic Rotary Wing milestone will be met by coupling this improved CFD tool with a flexible rotor (CSD) code and a rotor-blade trim algorithm.

Co-Investigators

- Thomas Pulliam, Terry Holst, Jasim Ahmad, David Kao, Guru Guruswamy, Ethan Romander, and I-Chung Chang, all of NASA Ames Research Center

Publications

[1] Holst, T. L. and Pulliam, T. H., "Overset Solution Adaptive Grid Approach Applied to Hovering Rotorcraft Flows," To be presented at the 27th AIAA Applied Aerodynamics Conference, *AIAA Paper No. 2009-3519*, San Antonio, TX, June 22–25, 2009.

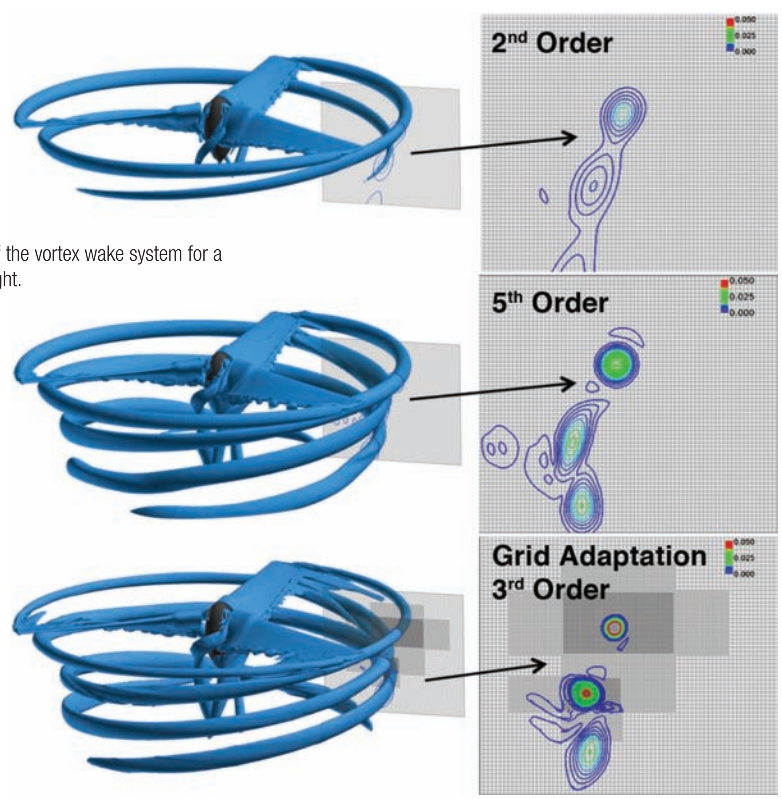
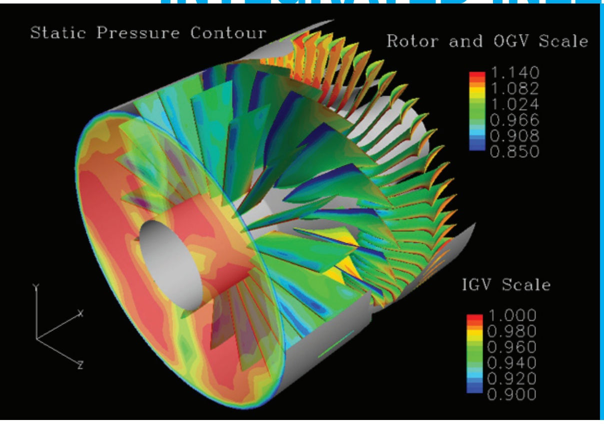


Figure 2: Concurrent visualization of the vortex wake system for a V-22 rotor in forward-descending flight.

INTEGRATED INLET/FAN SIMULATION (IISIM)



AERONAUTICS RESEARCH MISSION DIRECTORATE

MICHAEL HATHAWAY
Army Vehicle Technology Directorate
NASA Glenn Research Center
(216) 433-6250
Michael.D.Hathaway@nasa.gov

Figure 1: Instantaneous "snapshot" in time from an unsteady, 3D, full-annulus TURBO simulation of the 1½ stage fan. The case shown here captures results of a simulation which includes a two-per rotor revolution total pressure distortion induced by two diametrically opposed rods placed in the flow upstream of the fan inlet. The simulation results were used to predict the unsteady blade pressure loading from which the blade response characteristics where calculated and compared to measurements.

Project Description: This project was a collaborative effort among NASA, the Army Vehicle Technology Directorate, Honeywell Aerospace, and AVETEC, Inc. (Springfield, OH), to support development of Advanced Virtual Test Cell computational tools for simulating operation of a complete gas turbine engine with multi-component and multidisciplinary interactions. Traditionally, gas turbine engine subsystems (e.g., inlet and fan) have been designed, analyzed, and tested as isolated components. With trends toward more compact, higher-power, higher-density engines and future vehicle concepts with embedded engines and more compact, low-observable inlets, significant flow distortions are generated that the fan/engine must accommodate. With engine components becoming more closely coupled, conventional means of accounting for component interactions may be inadequate. Consequently, decreased performance, stall margin, and even life of the fan (and engine) may result if interacting are not properly addressed.

Development and validation of a consistent computational methodology for integrated inlet-fan/engine simulations enables analysis of the interaction and component matching effects between the inlet and fan/engine. For example, the effect of the fan/engine flow distribution can be included in the inlet analysis, as it can directly affect the inlet performance if the components are not properly matched, especially during flight operations wherein the bypass ratio can change significantly. Flow distortions generated by the inlet or ingested by the inlet are passed to the fan/engine, affecting performance, stability, stage, component matching, and fan aeromechanical response. Aerodynamic response of the fan/engine to the inlet distortion conversely impacts the inlet. Inlet distortions can comprise not only total pressure distortions, but also thermal and swirl distortions, as well as constituent-based distortions such as those resulting from steam ingestion. All of these distortions can potentially be analyzed with a validated integrated inlet-fan/engine simulation capability to better understand the impact on the inlet-fan/engine system, thereby leading to design improvements.

The objectives of this work are to:

- Develop a capability to simulate an integrated inlet/fan geometry, both with and without flow control, by coupling state-of-the-art computational fluid dynamics (CFD) codes for the inlet (Wind-US) and fan (TURBO).
- Validate capability of the fan simulation to support fan aeromechanical response analysis using predicted aero forcing function and fan damping based on the results of a modal analysis of blade motion from an ANSYS analysis to prescribe blade deflections for the TURBO aeromechanical simulations.
- Demonstrate a capability to generate coupled inlet (Wind-US) and fan (TURBO) simulations of an integrated inlet/fan geometry, both with and without flow control.

Relevance of Work to NASA: Inlet/fan interactions have been identified as a challenging problem for all vehicle classes within the NASA Fundamental Aeronautics Program (FAP). In addition, FAP's research philosophy is to develop technology and capabilities to enable validated, multi-component, multidisciplinary analysis-leading simulations of complete engine and vehicle systems. This work was performed under the Subsonic Fixed-Wing Project and supports level 2, integrated methods and technologies to develop multidisciplinary solutions.

Computational Approach: The primary computational approach of this work was to couple the Wind-US and TURBO CFD codes. Both are capable of simulating unsteady, 3D, turbulent flows by solving the compressible Reynolds-Averaged Navier-Stokes equations. Wind-US was developed to simulate inlet flows, while TURBO was developed to simulate turbomachinery flows. The reason for this choice of codes is that they were specifically developed and validated for their respective components, and both solvers use similar numerical approximations and algorithms, which simplify the integration effort. Both solvers are launched simultaneously in simulating the integrated inlet/fan flow. Parallel execution of the two solvers requires an interface that allows transference of information between Wind-US and TURBO. This interface, called the Aerodynamic Interface Plane (AIP), lies at a convenient distance upstream from the fan. It divides the integrated inlet/fan computational domain into an inlet domain and a fan domain. Wind-US computes flow through the inlet while TURBO computes flow through the fan. At the end of a time step, flow information at the interface is exchanged between the two solvers via Message Passing Interface (MPI) libraries. A simulation of a compressor with a long upstream duct was used to test the Wind-US and TURBO coupling capability.

A suitable inlet/fan geometry for which test data were available was selected for independent validation of Wind-US and TURBO predictions of the inlet and fan geometries. The geometry selected for simulation is a Lockheed inlet coupled to a Honeywell fan tested under the Air Force-sponsored Versatile Active Highly Integrated Inlet/Fan for Affordability Performance and Durability (VAIIPR) Program. The Lockheed inlet incorporated flow control to produce a uniform total pressure at the AIP between the inlet and fan. A known disturbance was then imposed at the AIP by inclusion of two diametrically opposed rods inserted in the flow path to generate a two-per revolution disturbance to the fan for which the blade response was measured (Figure 1). Fan performance and blade response were measured for two different orientations of the two-rod disturbance generator. Unsteady full-annulus TURBO simulations of the Honeywell fan stage were then conducted with the measured total pressure distortion resulting from both orientations of the two-rod disturbance prescribed at the fan inlet boundary, the AIP plane. Comparisons were made of the measured and predicted performance and blade response.

Results:

- Completed simulations of the baseline fan geometry at high- and low-speed conditions without the two-rod distortion.
- Completed simulations of the baseline fan geometry at low-speed conditions with the two-rod distortion at two orientations, with the second rotated 180 degrees from the first (Figure 1), and completed forced response analyses from the results.
- Completed coupling of the Wind-US and TURBO codes and subsequently demonstrated the code coupling capability on a compressor stage with an extended inlet (Figure 2).

Role of High-End Computing: These simulations would not have been possible without the computational resources provided by the NASA Advanced Supercomputing (NAS) Division. Due to contractual obligations with industry special resources (Toucan), high-priority queues (armd_spl) were provided to accomplish the large simulations (335 gigabytes of memory, 291 processors, 65 hours per rotor revolution) in a timely manner and to accommodate the significant storage requirements (1 terabyte of temporary storage each for three researchers) for multiple cases. Responsiveness of the NAS team in supporting our computational requirements was especially important in enabling adequate progress to meet our contractual obligations.

Future: This project was terminated due to reprioritization of research funding.

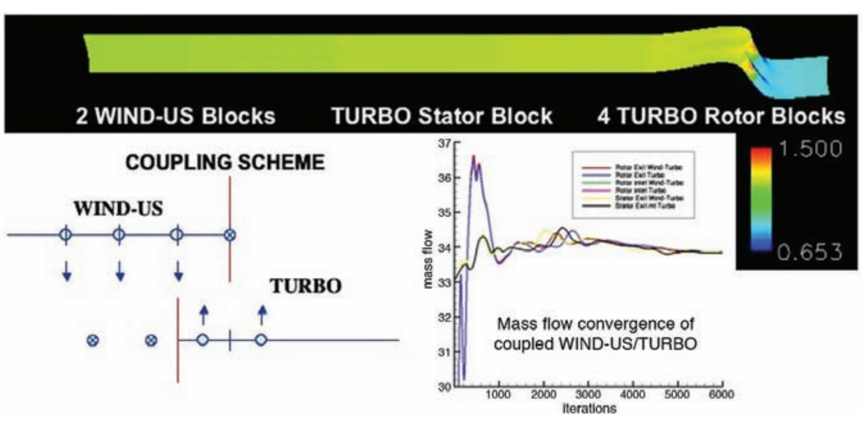
Co-Investigators

- Wai Ming To, University of Toledo
- Ambady Suresh, General Electric Company Global Research Center
- Rakesh Srivistava, Honeywell Engine Co.
- Milind Bakhle and John Lytle, NASA Glenn Research Center
- T. S. Reddy, University of Toledo
- Jeff Dalton, AVETEC, Inc.

Publications

[1] To, WM., Hathaway, M. D., “TURBO Simulation of a Honeywell Fan,” to be published as a NASA TM with limited distribution.

[2] Reddy, T. S., Bakhle, M. A., “Forced Response Analysis of a Low-Aspect Ratio Fan,” NASA internal publication.



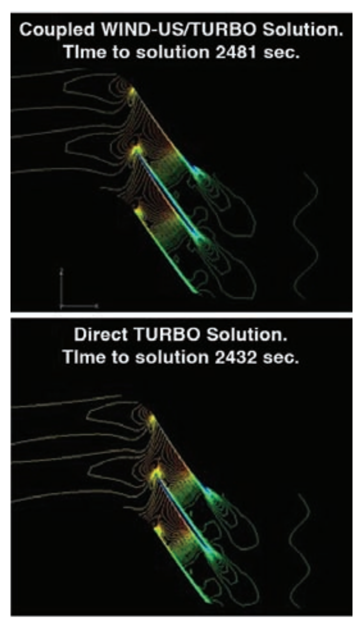
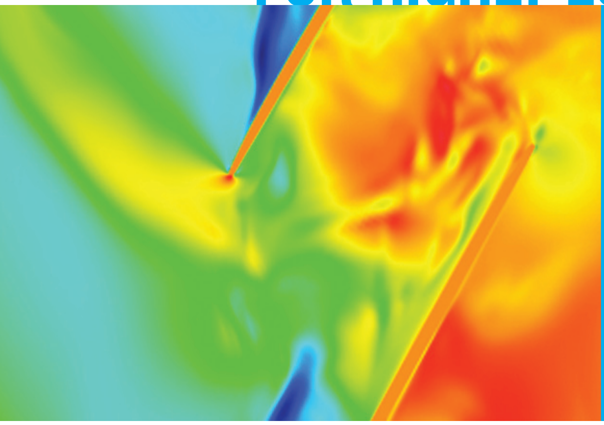


Figure 2: Demonstrated integrated inlet/fan simulation capability coupling of the Wind-US code, for simulation of an engine inlet duct, and the TURBO code for simulating a fan stage. Massflow convergence and comparison to direct TURBO simulation is shown.

LARGE EDDY SIMULATION FOR HIGHLY LOADED TURBOMACHINERY



AERONAUTICS RESEARCH MISSION DIRECTORATE

CHUNILL HAH
NASA Glenn Research Center
(216) 433-6377
Chunill.Hah-1@nasa.gov

◄ Close-up from Figure 1.

Project Description: Modern high-speed compressors operate with increasing aerodynamic blade loading. With this increased aerodynamic loading, it is critical to maintain a suitable stall margin to avoid engine stall during operation. Detailed unsteady flow structures in compressors operating near stall are not well understood, and it is generally believed that the conventional Reynolds-Averaged Navier-Stokes (RANS) approach does not predict the flowfield adequately. This fundamental aeronautics research project focuses on development and validation of a Large Eddy Simulation (LES) for this type of flowfield. The developed simulation tools can be applied to calculate detailed unsteady flow features so that advanced compressor designs can be studied to maintain a wide stall margin.

To date, the LES has been successfully applied to study self-induced flow instability and resulting non-synchronous blade vibration in axial compressors. The flow instability originates from interactions among tip vortex oscillation, vortex shedding, and passage shock (Figures 1 and 2). The simulation created with the LES tool clearly explains physics of flow instability in axial compressors for the first time. Further developments of the simulation are aimed at performing simulations of unsteady flow interactions between inlet and compressor stages with various flow control devices. A detailed understanding of the unsteady flowfield can contribute to better design of the compressor and any possible flow control devices.

Relevance of Work to NASA: This work is funded by NASA's Aeronautics Research Mission Directorate and supports NASA Fundamental Aeronautics Program goals to develop methods of subsonic fixed-wing simulations, specifically pertaining to inlet/fan interaction under supersonic conditions. The developed tool will be used to optimize both advanced fan design and flow control in the supersonic inlet/fan.

Computational Approach: A LES module has been integrated into the H3D code, a turbomachinery flow analysis code developed at NASA Glenn Research Center. The code employs a third-order accurate interpolation scheme for the convection terms and a central-differencing method for diffusion terms. A standard two-equation turbulence closure is used for the steady and unsteady RANS analysis. H3D is widely used throughout the U.S. aeronautics research community for the analysis of compressors and turbines. A Smagorinsky-type eddy-viscosity model is used for the sub-grid stress tensor for large eddy simulations. Dynamic models by Germano and Vreman are also implemented in the code. H3D has been parallelized for large-scale computations—up to 250 million grid-nodes have been used to simulate unsteady flowfields in an isolated compressor rotor.

Results: The H3D code was successfully applied to calculate combined flowfield in a fan stage with an ultra-compact inlet. The numerical results fairly accurately represent major flow characteristics including influence on stall margin.

The code was also applied to study flow instability in a transonic compressor at near-stall conditions. H3D calculated measured frequency of flow instability very well (Figure 3), and the underlying flow physics were explained with the simulated flowfield. This was the world's first-ever successful calculation of flow instability phenomena in a transonic compressor.

The code has been applied to investigate flowfield in a transonic rotor known as NASA Rotor 37. Numerical results from the LES show many improvements from the RANS simulations. It is thought that the LES module calculates flow interaction between the passage shock and tip leakage vortex much more realistically. Results from LES match the measured data very well, especially near the casing where flow interaction is strong.

Role of High-End Computing: The current LES of unsteady flowfields in highly loaded turbomachinery requires large-scale computations. Experts at the NASA Advanced Supercomputing (NAS) facility performed code optimization and parallelization on the H3D code to maximize computational efficiency on NAS supercomputers. Timely execution of the numerical analyses require parallel processing with many processors available.

Future: The H3D code will be extended to conduct unsteady flow simulations in multi-stage compressors and turbines. The code will be applied to simulate and develop optimum flow control strategy in a compact inlet/fan stage. Further improvement in computational efficiency is necessary to complete this work.

Co-Investigators

- Haoqiang H. Jin, NASA Ames Research Center

Publications

[1] Hah, C., Bergner, J., and P. Schiffer, "Tip Clearance Vortex Oscillation, Vortex Shedding and Rotating Instabilities in an Axial Transonic compressor Rotor," ASME paper, *GT2008-50105*, 2008.

[2] Hah, C., "Aerodynamic Study of Circumferential Grooves in a Transonic Axial compressor," ASME paper, *FED-55232*, 2008.

[3] Hah, C., "Self-Induced Flow Unsteadiness and Non-Synchronous Vibrations in Axial Compressors," ISOROMAC paper, *ISROMAC12-2008-20032*, 2008.

[4] Mueller, M., Schiffer, H., and Hah, C., "Interaction of Rotor and Casing Treatment Flow in an Axial Single-Stage Transonic Compressor," ASME paper, *GT2008-50135*, 2008.

[5] Hah, C., Bergner, J., and Schiffer, P., "Short Length-Scale Rotating Stall Inception in a Transonic Axial Compressor," ASME paper, *GT2006-90045*, 2006.

[6] Hah, C. and Lee, Y., "Unsteady Tip Leakage Vortex Phenomena in a Ducted Propeller," *International Journal of Transport Phenomena*, Vol. 9, No. 3, 169–176, 2007.

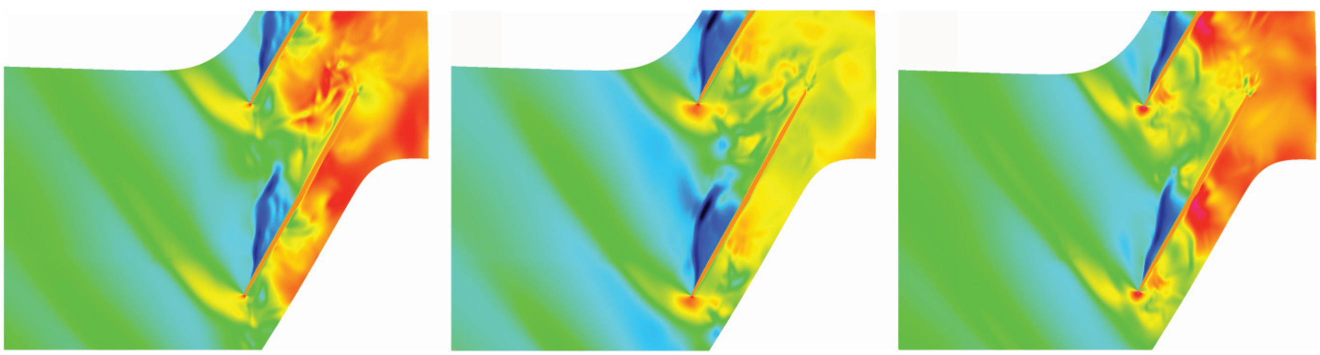


Figure 1: A Large Eddy Simulation (LES) calculates the oscillation of tip clearance vortex, as the measurements show. The synchronized tip vortex oscillation across several blade passages creates flow instability that can cause structural failure of the blade at certain operating conditions.

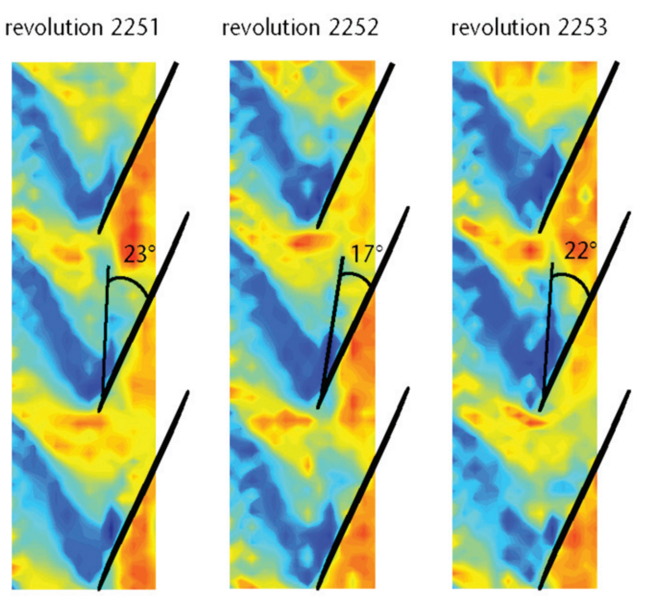


Figure 2: This measured instantaneous pressure field on the casing reveals oscillation of tip clearance vortex, due to interaction between this vortex and the passage shock.

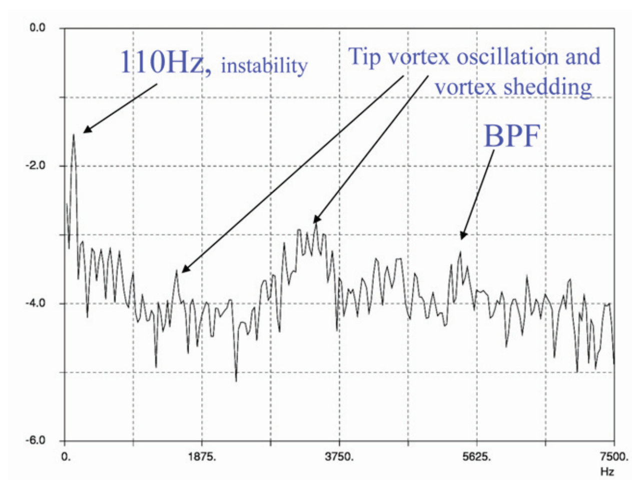
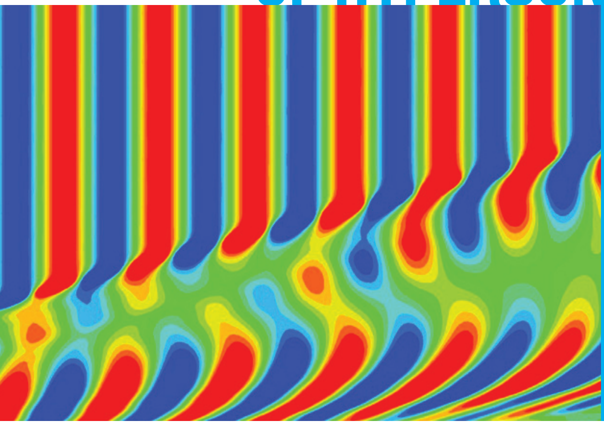


Figure 3: Wall pressure spectrum from LES shows dominant frequencies. The 110 Hz represents flow instability due to flow interaction between the passage shock and tip clearance vortex. The calculated frequency agrees well with the value from measurement.

RECEPTIVITY AND STABILITY OF HYPERSONIC BOUNDARY LAYERS



AERONAUTICS RESEARCH MISSION DIRECTORATE

PONNAMPALAM BALAKUMAR
NASA Langley Research Center
(757) 864-8453
Ponnampalam.Balakumar-1@nasa.gov

◀ Close-up of Figure 2.

Project Description: Accurately predicting transition onset and transition end-points, modeling this transitional region, and modeling the turbulence region are major challenges in accurately computing the aerodynamic quantities using computational fluid dynamics codes. The transition process depends primarily on the boundary layer characteristics and on the frequency and wave number distributions of the disturbances that enter the boundary layer. The difficulty is computing, predicting, or prescribing the initial spectral, amplitude, and phase distribution of the disturbances inside the boundary layer. In any new transition prediction strategy, one should quantify these two quantities and determine the minimum amount of information necessary to predict the transition onset accurately. The objectives of this research are to overcome some of these difficulties, and to eventually come up with an improved transition prediction method. Accurate transition onset prediction will help compute the heating and skin friction loads on the vehicle accurately and will improve design of thermal protection systems and structural components.

To understand and quantify receptivity coefficients and stability characteristics of hypersonic boundary layers, interaction of acoustic waves with hypersonic boundary layers over sharp and blunt flat plates, wedges, and cones were numerically simulated at different freestream and wall conditions. The importance of slow and fast acoustic waves, unit Reynolds number effects, the bluntness effects, and wall cooling effects on the receptivity and stability were systematically investigated. The receptivity coefficients, stability properties, and the transition Reynolds numbers were obtained for different cases.

Relevance of Work to NASA: One of the NASA Fundamental Aeronautics Program's objectives is to develop physics-based models to predict transition in hypersonic flows. Understanding the transition process from the first principle will lead to improved predictive capabilities in flows over hypersonic vehicles. NASA's interest in space exploration requires development of vehicles that fly through the hypersonic regime. Efficient, reliable, and reusable hypersonic vehicles will benefit NASA's space exploration mission.

Computational Approach: Three-dimensional compressible Navier-Stokes equations are solved using the fifth-order-accurate weighted essentially non-oscillatory (WENO) scheme for space discretization, and using the third-order total-variation-diminishing (TVD) Runge-Kutta scheme for time integration. These methods are suitable in flows with discontinuities or high-gradient regions. After the steady mean flow is computed, acoustic disturbances are superimposed at the outer boundary of the computational domain and time-accurate simulations are performed.

Results: A number of fundamental studies have been numerically performed to evaluate the effects of different parameters such as slow and fast acoustic waves; bluntness; wall cooling on the receptivity; and stability of hypersonic boundary layers over plates, wedges, and cones. Findings include:

- *Receptivity of Hypersonic Boundary Layers Over Cones and Wedges to Acoustic Disturbances:* The receptivity coefficient of the instability waves generated by the slow acoustic wave is about four times the amplitude of the freestream acoustic waves. The amplitude of the instability waves generated by the slow acoustic waves is about 60 times larger than that for the case of fast acoustic waves (Figure 1).
- *Effects of Nose Bluntness on Receptivity and Stability of Hypersonic Boundary Layers Over Cones:* The bluntness has a strong stabilizing effect on the boundary layers. This is due to the entropy layers that persist for longer distances with increasing bluntness. The receptivity coefficients for large bluntness are much smaller—on the order of 10^{-3} (Figure 2).
- *Effects of Wall Cooling on Receptivity and Stability of Hypersonic Boundary Layers Over Cones:* Wall cooling stabilizes the first mode and destabilizes the second mode and shifts the

transition onset further upstream. The fast mode is not affected by the wall cooling, and the receptivity coefficient of the fast wave is about 50 times larger than for the slow wave (Figure 3).

Role of High-End Computing: Performing a time-accurate simulation of receptivity and stability processes in hypersonic boundary layers is computationally demanding. One computation for a two-dimensional case takes about one week of computer time on 24 processors. Performing parametric studies for several cases requires Columbia supercomputer resources to obtain the results in a reasonable amount of time.

Future: Continuation of this work includes simulating the transition process in three-dimensional hypersonic boundary layers such as flow over cones at angles of attack and ellipsoids. Another goal is to simulate the roughness-induced transition in hypersonic boundary layers. These are three-dimensional simulations and require months of computational time for one case on 64 processors.

Co-Investigators

- Kursat Kara and Osama A. Kandil, Dept. of Aerospace Engineering, Old Dominion University

Publications

[1] Kara, K., Balakumar, P., and Kandil, O. A., "Effects of Wall Cooling on Hypersonic Boundary Layers Receptivity over a Cone," 38th AIAA Fluid Dynamics Conference and Exhibit, Seattle, WA, *AIAA 2008-3734*, 2008.

[2] Kara, K., Balakumar, P., and Kandil, O. A., "Effects of Nose Bluntness on Stability of Hypersonic Boundary Layers Receptivity over a Blunt Cone," 37th AIAA Fluid Dynamics Conference and Exhibit, Miami, FL, *AIAA 2007-4492*, 2007.

[3] Kara, K., Balakumar, P., and Kandil, O. A., "Receptivity of Hypersonic Boundary Layers Due To Acoustic Disturbances over Blunt Cone," 45th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, *AIAA 2007-0945*, 2007.

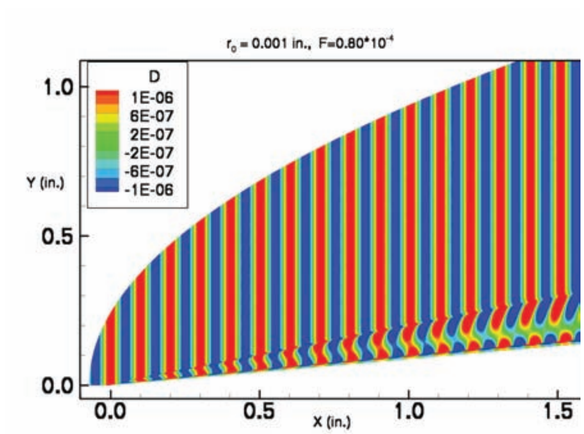


Figure 1: Contours of the unsteady density fluctuations due to the interaction of a slow acoustic wave over a 5-degree half-angle sharp cone at a Mach number of M = 6.

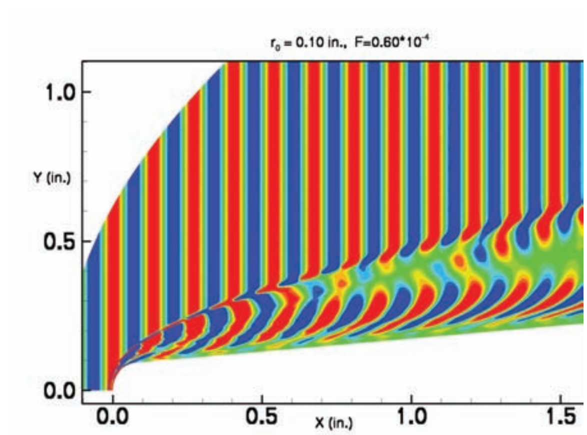


Figure 2: Contours of the unsteady density fluctuations due to the interaction of a slow acoustic wave over a 5-degree half-angle blunt cone.

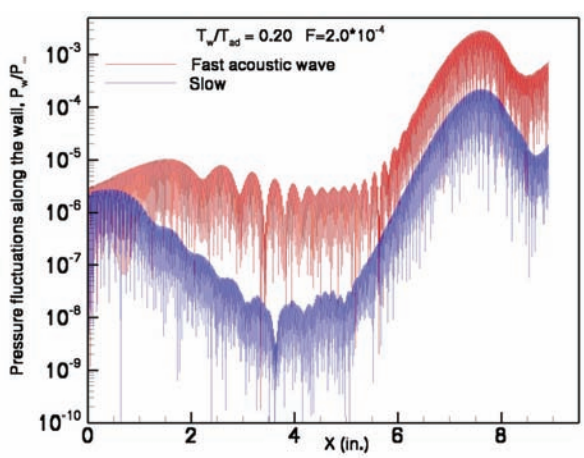
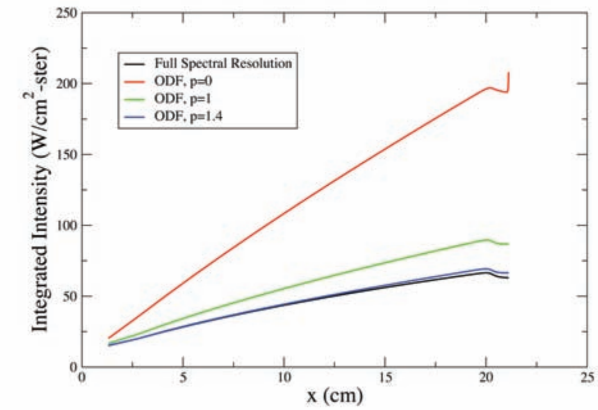


Figure 3: Unsteady pressure fluctuations along the wall in a log scale generated by the slow and fast acoustic waves in the cold wall case.

TOWARD IMPROVED RADIATIVE TRANSPORT IN HYPERSONIC REENTRY



AERONAUTICS RESEARCH MISSION DIRECTORATE

ALAN WRAY
NASA Ames Research Center
(650) 604-6066
Alan.A.Wray@nasa.gov

◀ **Figure 1:** Comparison of simulations of radiation along the peak radiative heating ray in the reentry of a crew exploration vehicle type of body into Earth's atmosphere. The black curve required 20 million points in wavelength space, and the blue curve only 10 million points, using an opacity distribution function method that will be incorporated into the HyperRad program.

Project Description: Large bodies reentering planetary atmospheres undergo substantial heating from radiation produced by high-temperature gas that results as the body decelerates. This heating must be accounted for in designing spacecraft to be both safe and efficient in carrying out their missions. Current software to compute such effects is hampered by large uncertainties resulting from various approximations in both physical modeling and numerical methods. These approximations were necessary in an earlier era of computational power, but now can be largely removed.

Our new software program, dubbed HyperRad, will bring the computation of radiative effects in hypersonic flow to a new level of accuracy and quantifiable uncertainty consistent with current computer hardware. Design robustness and confidence levels will thereby be increased and costs associated with wind tunnel and flight testing will be reduced.

Relevance of Work to NASA: Spacecraft design engineers must balance the requirements of safe mission completion with payload size and mass. This means that the weight and design of the thermal protection system (TPS) must be optimized to ensure integrity of the spacecraft on reentry without an excessive allocation of mass. Such designs require both experimental and computational studies to ensure accurate engineering specifications. Traditional fluid mechanics simulations must be augmented by radiative transport effects in order to gain complete knowledge of the thermal environment of the spacecraft.

Computational Approach: Several ongoing computational efforts are required to accomplish the project goals. First, accurate chemical databases must be constructed to obtain the radiative emission and absorption properties of the fluid, as functions of its composition, history, and thermodynamic state. These databases are obtained both from experiment and from quantum and classical computational algorithms. Second, these databases are being put in a form that can be efficiently utilized at runtime of the HyperRad radiative transport code (Figures 1 and 2). Finally, HyperRad will be optimized to compute the radiation throughout the flowfield with enough speed that the total runtime of the simulation is not greatly increased from that of non-radiative cases.

Results: A large database of molecular and atomic states, energy levels, lifetimes, collision cross sections, and coefficients required for radiative transport has been assembled from the literature and from massive new quantum chemistry calculations done on the Columbia supercomputer. The database includes information about the ground and excited states of molecular, atomic, and ionic species found in the atmospheres of Earth and Mars. This data forms the foundation for the tables required for efficient simulation of the radiation. These run-time data structures have been designed and are being programmed into Fortran 90. Other results include the following:

- Techniques for reducing the cost of carrying large numbers of points in wavelength-space have been designed and tested by comparison to existing experimental and simulation data.
- Techniques to allow efficient transport of radiation over many angular directions have been tested and compared to theoretical results and to existing approximations.
- Effective parallelization of these methods has been designed and tested.

Role of High-End Computing: The Columbia supercomputer has been essential for the quantum chemical calculations used to update and extend chemical databases required to complete the development of HyperRad. It is also on the high-end computing (HEC) systems that large-scale radiation-hydrodynamic simulations of spacecraft reentering the atmosphere will ultimately be done; thus, all program design considerations are based on such hardware. Testing of the components in HyperRad has been and will continue to be done on the Columbia and Pleiades systems.

Future: We are moving into the testing phases of the most computationally intensive portion of the radiative transport algorithm. Columbia, Pleiades, and future HEC systems will be essential to completing this project in a timely manner. Large computational meshes will be required for the computational fluid dynamics component, and equally fine, but different, meshes will be used for the radiation computation. Conversion of data between these two mesh classes will also be an important programming and execution challenge in the coming year.

Co-Investigators

- David Schwenke, Richard Jaffe, Yen Liu, Duane Carbon, Galina Chaban, all of NASA Ames Research Center
- Winifred Huo, Huo Consulting LLC
- Dinesh Prabhu, NASA Ames, ELORET
- Thierry Magin, Stanford University

Publications

[1] Jaffe, R., Schwenke, D., and Chaban, G., "Theoretical analysis of N2 collisional dissociation and rotation-vibration energy transfer," AIAA Aerospace Sciences meeting, January 2009.

[2] Jaffe, R., Schwenke, D., Chaban, G., "Theoretical analysis of N2 collisional dissociation and rotation-vibration energy transfer," AIAA Aerospace Sciences meeting, January 2009.

[3] Magin, T., Caillault, L., Bourdon, A., and Laux, C., "Nonequilibrium radiative heat flux modeling for the Huygens entry probe", *Journal of Geophysical Research*, 111, 2006.

[4] Huo, W., "Electron Recombination and Collisional Excitation in Air," Lecture Series, von Karmen Institute for Fluid Dynamics, 2008.

[5] Bourdon, A., Panesi, M., Brandis, A., Magin, T., Chaban, G.,Huo, W., Jaffe, R., and Schwenke, D., "Simulation of flows in shock-tube facilities by means of a detailed chemical mechanism for nitrogen excitation and dissociation," Proceedings, Stanford-NASA Center for Turbulence Research, 2008.

[6] Graille, B., Magin, T., and Massot, M., "Modeling of reactive plasmas for atmospheric entry flows based on kinetic theory," Proceedings, Stanford-NASA Center for Turbulence Research, 2008.

[7] Wray, A., Prabhu, D., and Ripoll, J-F., "Opacity Distribution Functions Applied to the CEV Reentry," AIAA Thermosciences Meeting, 2007.

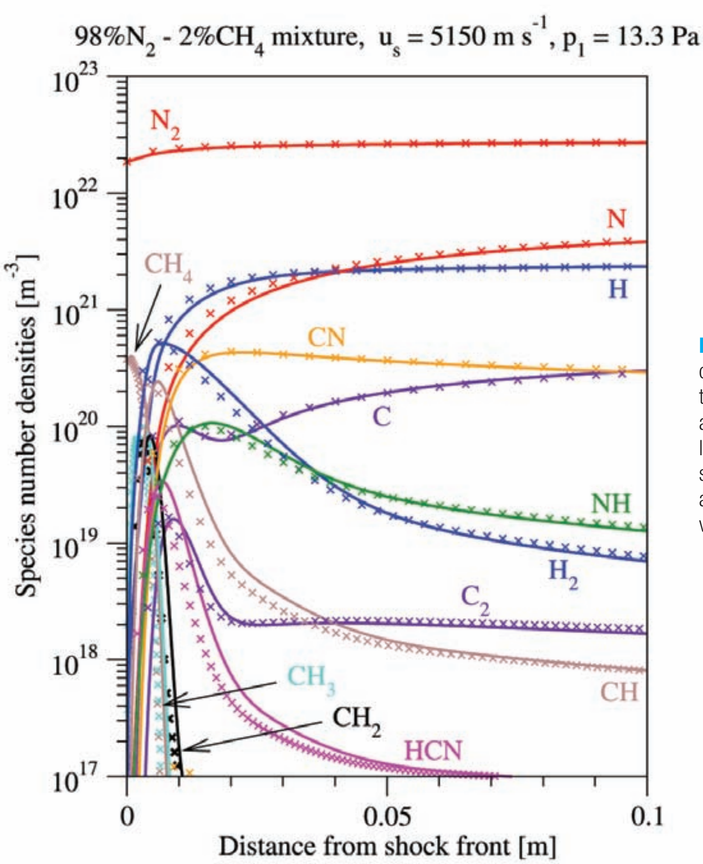
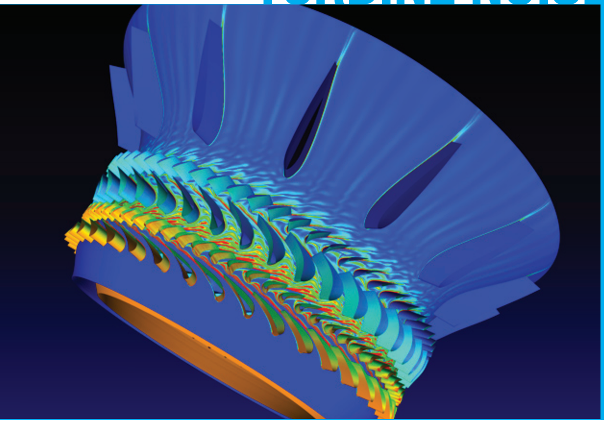


Figure 2: Results (Magin et al., 2006) comparing an Ames simulation of a shock-tube experiment (solid lines) and a time-accurate collisional-radiative model (xxxxxx lines) for species number densities downstream of a reentry shock in a Titan-like atmosphere. Such simulations are used to validate the models used in HyperRad.

TURBOMACHINERY AEROACOUSTICS: TURBINE NOISE GENERATION IN TURBOFAN ENGINES



AERONAUTICS RESEARCH MISSION DIRECTORATE

DALE VAN ZANTE
NASA Glenn Research Center
(216) 433-3640
Dale.E.VanZante@nasa.gov

◀ **Figure 1:** Instantaneous view of the five-blade-row, high-pressure turbine coarse mesh simulation with flow colored by vorticity to show velocity non-uniformities. The blade surfaces are colored by static pressure. The interaction of velocity non-uniformities with the blade surfaces is one noise generation mechanism in turbines.

Project Description: As aircraft engine bypass ratios continue to increase, the relative contribution of the turbine to the overall aircraft noise signature increases. Due to the dominance of fan and jet sources in moderate- and high-bypass ratio engines, turbine noise has typically been ignored, resulting in minimal effort focused on noise reduction technology development for turbines. Additionally, the desire for lower-cost, lower-weight, and higher-performance engines has resulted in turbine design changes that have typically increased turbine noise.

Understanding the relative importance of various turbine noise generation mechanisms and the characteristics of turbine acoustic transmission loss are essential ingredients in developing robust models for predicting the turbine noise signature. Typically, turbine noise models in use today are semi-empirical in nature and not suitable for detailed design and analysis studies. A computationally based investigation has been undertaken to help guide development of a more robust turbine noise prediction capability that does not rely on empiricism and is capable of addressing general design changes.

The Acoustics Discipline of NASA's Fundamental Aeronautics Subsonic Fixed Wing Project is pursuing technologies to reduce aircraft noise, with the ultimate goal of containing objectionable noise from aircraft to within the airport boundary. Aircraft noise is an amalgam of propulsion and airframe sources whose relative contributions depend on the aircraft type and operating condition. Generally speaking, propulsion noise (that is, engine noise) is a significant contributor to the total aircraft noise signature. Of the various sources of engine noise, fan and jet sources have received much attention in the past, but with the advent of ultra-high-bypass ratio engines (like the Pratt & Whitney geared turbofan), turbine noise is emerging as an important source of noise that must be mitigated in future low-noise propulsion systems. The Turbine Noise Project's objective is to develop an understanding of how and where noise is produced inside the turbine. Based on this knowledge, noise models can be formulated that will aid in evaluation of new engine designs and in the development of turbine noise mitigation technologies.

Relevance of Work to NASA: NASA is responsible for maintaining and improving an aircraft system noise prediction code called Aircraft NOise Prediction Program (ANOPP). This code is used by the Federal Aviation Administration, and engine and aircraft manufacturers to assess the impact of noise from contemporary aircraft on communities, as well as to evaluate how changes in design of the aircraft system will alter noise impact. Results from the Acoustics Discipline research activities are used to continuously improve models used in the ANOPP tool.

Computational Approach: The NASA turbomachinery aerodynamics solver TURBO is used to calculate the time-varying pressure field inside a turbine (Figures 2 and 3). A portion of this unsteady pressure field will propagate through the turbine blade rows and will emerge as noise from the engine exhaust. Frequency, modal content, and other characteristic information may be extracted from the pressure data by postprocessing and then used to construct reduced-order models for noise generation and propagation. To properly capture important pressure wave features inside the turbine, the numerical mesh must be about ten times denser than what is typically used for aerodynamic performance calculations. Such dense meshes, required by the wave propagation physics, result in very large computational resource requirements.

Results: The turbine noise work supports the Subsonic Fixed Wing Project goal of developing and/or improving and validating the next-generation multi-fidelity component and aircraft noise prediction capability. Accomplishments to date include:

- A converged time-accurate solution for a single turbine stage (two blade rows) with a mesh density appropriate for capturing twice the blade-passing-frequency (2xBPF) tone has been completed. The computational domain contains 80 million nodes and the solution dataset is 965 gigabytes in size. The simulation required 11 days of run-time on 200 processors of the RTJones supercomputer at NASA Ames Research Center to converge.

- Spectral and modal analysis of unsteady pressure data from the single-stage simulation shows that the simulation captures the anticipated acoustic modes properly. These results have been reported to joint working groups involving industry, academia, and government.

Role of High-End Computing: The availability of computing resources including large numbers of processors, high-speed networks, and parallel visualization capabilities have all been enabling technologies for the turbine noise effort. Prior to recent additions to NASA high-end computing resources (namely RTJones), a simulation of this scope would not have been practical.

Future: With the proof-of-concept phase nearly complete, the next phase is to complete a simulation of an entire aircraft engine high-pressure turbine (HPT) consisting of five blade rows. The additional solution domain will permit a more comprehensive assessment of the upstream and downstream traveling acoustic waves within the turbine. Preliminary coarse mesh simulations of the HPT are already complete and will help guide setup for the fine mesh simulation (Figure 1). The fine mesh case will require 200 million nodes and 600 processors to converge the solution in a reasonable period of time.

Co-Investigators

- Edmane Envia, NASA Glenn Research Center

Publications

[1] Van Zante, D. and Envia, E, "A Numerical Investigation of Turbine Noise Source Hierarchy and Its Acoustic Transmission Characteristics," Invited presentation at AeroAcoustics Research Consortium Turbine Noise Workshop, Vancouver, BC, May 2008.

[2] Van Zante, D. and Envia, E., "A Numerical Investigation of Turbine Noise Source Hierarchy and Its Acoustic Transmission Characteristics: Proof-of-concept progress," Acoustics Technical Working Group, Williamsburg, VA, Sept. 2008.

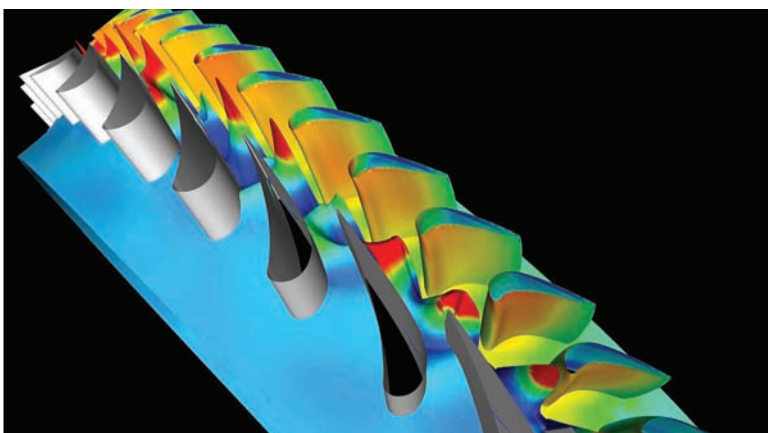


Figure 2: Pressure wave formation: In this view of the first stage of the turbine, the rotor (colored by static pressure) cuts through the wake of the vane (gray blade row) and forms a series of pressure waves shown by the color shaded "wavy" surface. These pressure waves propagate upstream and downstream from the rotor. Only the upstream propagating wave is shown here. A portion of this fluctuating pressure will emerge from the engine nozzle as noise.

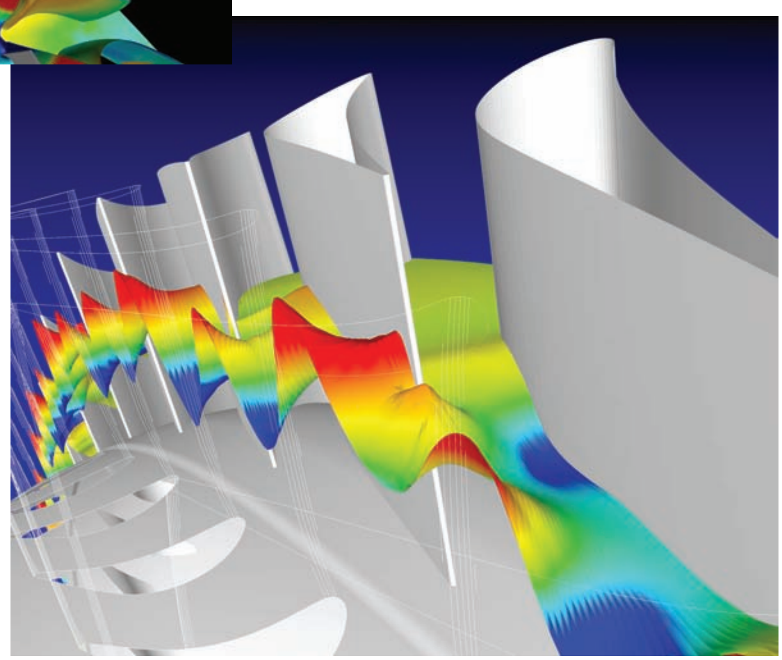
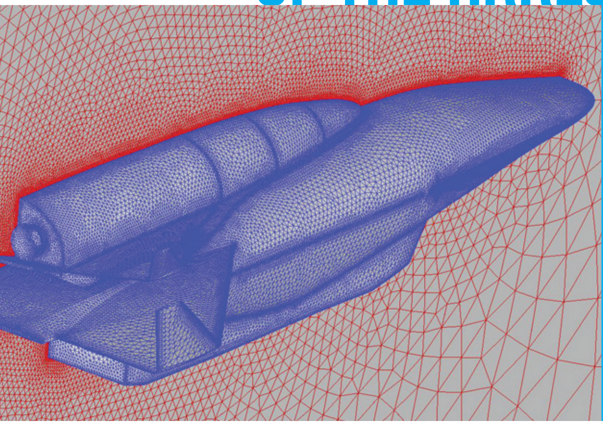


Figure 3: View of the pressure waves propagating upstream through the vane. The wave is attenuated as it moves upstream against the high subsonic Mach number flow coming through the vane passage.

USM3D ANALYSIS OF THE HRRLS CONFIGURATIONS



AERONAUTICS RESEARCH MISSION DIRECTORATE

JENN LOUH PAO
NASA Langley Research Center
(757) 864-3765
Jenn.L.Pao@nasa.gov

Figure 1: Surface grid of the Highly Reliable Reusable Launch Systems configuration.

Project Description: The leading roles of the Multi-Disciplinary Analysis Optimization (MDAO) team within the Hypersonics Project of NASA's Fundamental Aeronautics Program are to develop and analyze reference vehicle concepts to determine potential system capabilities, and to establish research and technology goals and requirements. One of the primary reference missions for MDAO is the Highly Reliable Reusable Launch Systems (HRRLS). The objectives of the MDAO system studies for the HRRLS mission are to provide: reference concepts for project disciplines to analyze/exercise tools and apply technologies; a means to exercise and evaluate MDAO tool development progress; and reference concepts for technology assessment and investment guidance. The specific objective of this task is to compute a longitudinal aerodynamic database for the HRRLS reference vehicle, from subsonic to hypersonic Mach numbers. This database was used in the trajectory analysis and optimization for the HRRLS mission.

To develop and analyze the HRRLS reference vehicle concepts, extensive computational fluid dynamics computations were performed for both mated and first stage configurations from subsonic to hypersonic speeds. These computations include: preliminary coarse grid solutions; grid convergence studies to determine final grids with adequate grid density; final grid solutions; and analysis of grid density impact on the longitudinal force and moment coefficients.

Relevance of Work to NASA: The work presented here is closely aligned with two of the Aeronautics Research Mission Directorate's primary goals: to gain advanced knowledge in the fundamental discipline of aeronautics, and to develop access to space technologies for safer long-range, high-speed aerospace transportation systems.

Computational Approach: In the HRRLS reference vehicle study, a longitudinal unpowered aerodynamic matrix was created for both mated and first stage configurations. The Tetrahedral Unstructured Software System (TetrUSS) and Navier-Stokes flow solver USM3D, developed at NASA Langley Research Center, were used to generate unstructured grids and compute the aerodynamic matrix. The surface grid for the HRRLS mated configuration is depicted in Figure 1. The final aerodynamic matrix contains more than 100 points over a Mach number range of 0.5–8.0, with an angle of attack range of -2–12 degrees to cover the three sigma trajectory dispersions. The flow simulation was performed using full viscous calculations with the Spalart-Allmaras turbulence model.

Results: In the early stages of the HRRLS configuration analysis, a set of preliminary subsonic and supersonic USM3D solutions were computed using a coarse grid. Soon after, extensive USM3D grid convergence studies were performed for the HRRLS mated and first stage configurations at subsonic, supersonic, and hypersonic Mach numbers. At the conclusion of the grid convergence study, the final subsonic, supersonic, and hypersonic grids were determined from a group of 34 grids. By applying these final grids, the USM3D aerodynamic database was generated for the HRRLS mated and first stage configurations. The flowfield solution for a representative case of the HRRLS mated configuration at Mach 0.8 and 8 degrees angle of attack is shown in Figure 2. Mach contours on the plane of symmetry are depicted in Figure 2a, the two-dimensional streamline plot on the plane of symmetry at the aft end is shown in Figure 2b, the three-dimensional isosurface of zero axial velocity in the external nozzle region and behind the base is shown in Figure 2c, the surface pressure contours on the leeward and windward sides are depicted in Figures 2d and 2e, and the surface pressure distribution along the centerline is shown in Figure 2f.

Role of High-End Computing: All computational results were generated on the Columbia supercomputer. Each job used 64 processors and required 60–70 gigabytes of system memory to execute all computations. Each subsonic case took 30–48 processor-hours to complete, while each supersonic case took 96–144 processor-hours to complete, and each hypersonic case took 160–200 processor-hours to complete, due to oscillatory behavior. NASA high-end computing experts helped to meet this challenge by making the most efficient use of the computer time allocations and supporting the largest volume of data transfer in the NASA Advanced Supercomputing (NAS) Division's history across the wide-area network between Columbia and NASA Langley. The NAS Division's high-end computing capability was a key component to the success of the HRRLS reference vehicle development task.

Future: The HRRLS reference vehicle was developed with a wing of fixed size to generate a longitudinal aerodynamic matrix. To achieve longitudinal stability for the HRRLS configuration, a parametric study of wings with common planform shape but with different areas to trim the HRRLS vehicle is planned. In addition, an assessment of turbine engine flow impact in the external nozzle region of the HRRLS configuration is also planned.

Publications

[1] Pao, J. L., "Presentation: Preliminary USM3D Results for the TSTO-mated and TSTO-1st-Stage Configurations at Subsonic and Supersonic Mach Numbers," TN 08-517 (NNL07AA00B), NASA Langley Research Center, August 2008.

[2] Pao, J. L., "USM3D Grid Convergence Study for the TSTO-mated and TSTO-1st-Stage Configurations at Subsonic, Supersonic and Hypersonic Mach Numbers," TN 08-518 (NNL07AA00B), NASA Langley Research Center, August 2008.

[3] Pao, J. L., "USM3D Computed Aerodynamic Database for the TSTO-mated and TSTO-1st-Stage Configurations at Subsonic, Supersonic and Hypersonic Mach Numbers," TN 08-520 (NNL07AA00B), NASA Langley Research Center, September 2008.

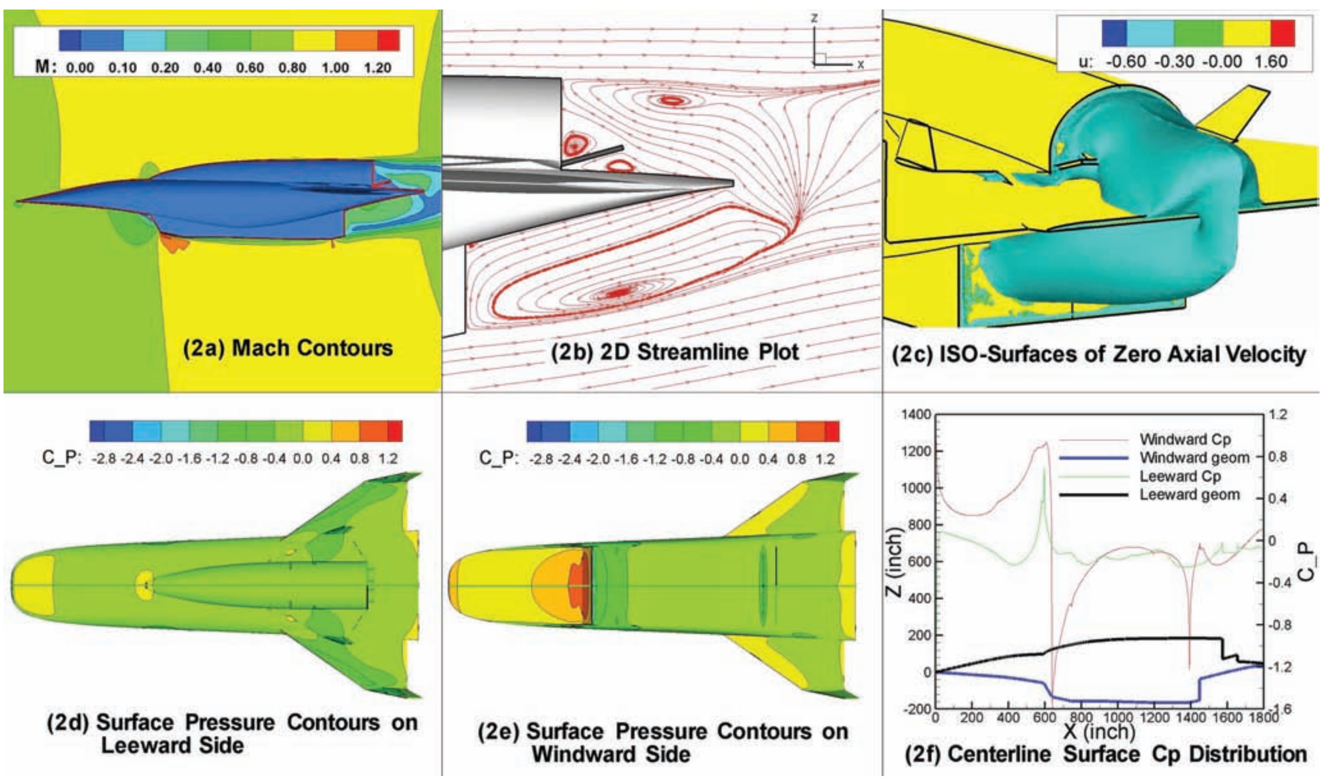
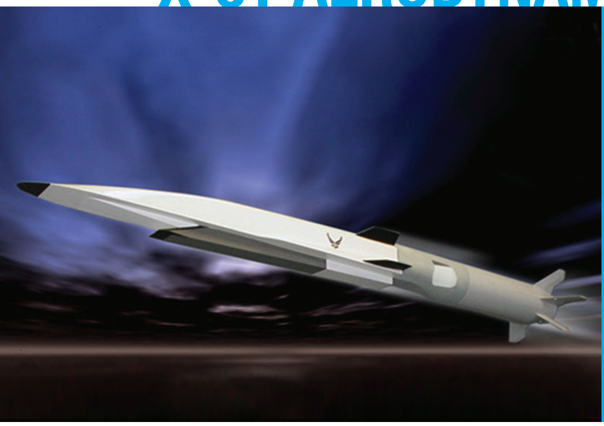


Figure 2: Flowfield solution for the Highly Reliable Reusable Launch Systems configuration at Mach = 0.8 and angle of attack = 8 degrees.

X-51 AERODYNAMICS



AERONAUTICS RESEARCH MISSION DIRECTORATE

TODD MAGEE
Boeing Research & Technology
(714) 896-1134
Todd.Magee@boeing.com

Figure 1: The goal of the X-51 Program is to flight-demonstrate an endothermic hydrocarbon-fueled scramjet engine. The work is sponsored by the Air Force Research Laboratory (AFRL) and Defense Advanced Research Projects Agency. Many team members are involved in the project, including individuals from Boeing, Pratt & Whitney, and NASA.

Project Description: The goal of this project is to utilize high-end computing to assess aerodynamic characteristics of the X-51 vehicle in preparation for its first flight, scheduled for fall 2009. The X-51 vehicle is an Air Force Research Laboratory- and Defense Advanced Research Projects Agency (DARPA)-sponsored flight demonstrator of the first hydrocarbon-fueled scramjet engine (see Figure 1). The flight profile for the X-51 covers several phases and flight regimes. The vehicle will be dropped from a B-52 carrier aircraft at Mach = 0.8, boosted to the scramjet starting condition (Mach ~4.5) using an Army ATACMS booster, then separate from the booster and ignite the scramjet engine, and accelerate up to a Mach 6+ cruise. After the vehicle runs out of fuel, it will decelerate to the mission ending point. Aerodynamic data is required over all flight regimes to support performance, stability and control, and loads analyses, as well as fin actuator and structure sizing. Utilizing wind tunnels to generate all of this data would be very impractical and costly—which ties in with a secondary project goal: to minimize the use of wind tunnel data by leveraging the use of computational fluid dynamics (CFD) and high-end computing.

This work entailed the generation of CFD results (Euler and Navier-Stokes) for both the X-51 hypersonic cruiser and stack vehicles over a flight condition range of Mach = 0.6 to 7, angles-of-attack = -10 to 25 degrees, sideslip angles = -4 to 4 degrees, and fin deflections = -25 to 25 degrees. The cruiser is the portion of the vehicle that contains the scramjet engine while the stack consists of the cruiser, inter-stage, and booster combined. The solutions were then processed to generate force and moment data for the cruiser and stack to populate various databases. The databases included skin friction versus altitude, general vehicle force and moment, fin forces and moments, beta vane calibration, B-52 influence, and stage separation. Many of the cases were utilized to fill in where wind tunnel data was not available. In addition, CFD pre-test analysis was conducted on several wind tunnel models to support wind tunnel tests at the Arnold Engineering Development Center (AEDC) Von Karman Tunnel B and the NASA Langley Research Center Unitary Plan Wind Tunnel (UPWT).

Relevance of Work to NASA: This project is supported by the Hypersonics Project under NASA's Fundamental Aeronautics Program. It leverages and extends the work completed by NASA on the X-43 Program in pursuit of practical hypersonic flight, and is fundamental to the X-51 Program meeting Critical Design Review delivery dates, and the subsequent hardware manufacturing/procurement. Therefore, it supports technology development for future access to space, hypersonic atmospheric vehicles, and high-performance weapons advancement.

Computational Approach: Three CFD tools were used in this project: the NASA Cart3D (Euler) code, the NASA OVERFLOW (Navier-Stokes) code, and the Boeing BCFD (Navier-Stokes) code. Integrated force and moment, surface pressure, viscous shear stress, and various flowfield data were utilized from the CFD simulation results.

Results:

The project generated a number of databases:

- *Skin friction database*: The skin friction database captures the effect of altitude on skin friction drag. Navier-Stokes CFD was conducted for both the cruiser and stack at various altitudes and Mach numbers (see Figure 2).
- *Flight polars for X-51 cruiser thrust/drag bookkeeping*: This database provides a determination of what forces are attributed to aerodynamics or propulsion. Navier-Stokes CFD was conducted at several supersonic and hypersonic Mach numbers and angles of attack (see Figure 2).
- *B-52 captive/carry and launch database*: This Euler CFD-based influence database was generated for conducting 6-degrees of freedom (6-DOF) separation analyses. The X-51 stack vehicle was analyzed at a matrix of locations below the B-52, and at various flight conditions (see Figure 3).
- *B-52 viscous captive/carry and launch cases*: Navier-Stokes CFD was utilized at several conditions that were also analyzed with Euler CFD to assess the effects of viscosity on the results (see Figure 3).

- *Pre-test predictions for AEDC VKF wind tunnel test*: Euler CFD was used to generate pre-test predictions for an AEDC Von Karman Facility (VKF) Tunnel B wind tunnel test. These pre-test predictions were used to ensure quality of the test data (see Figure 2).
- *Additional fin deflection data*: Euler CFD was utilized to provide fin effectiveness data for flight conditions that were not acquired through wind tunnel testing (see Figure 2).
- *Beta Vane calibration matrix*: CFD was utilized to generate a database to calibrate the beta vane sensor on the X-51 vehicle.
- *Stage separation CFD*: Navier-Stokes CFD was utilized to construct an aerodynamic database for use in 6-DOF separation analyses. Several flight conditions and various separation locations between the cruiser and interstage/booster sections were analyzed (see Figure 2).
- *Boat-tail CFD analyses*: Utilized Navier-Stokes CFD to determine the effect on aerodynamics by boat-tailing the back-end of the X-51 cruiser vehicle.
- *Sting Effects CFD*: Navier-Stokes CFD was utilized to determine effects of the sting, used in several wind tunnel tests, on the wind tunnel force and moment results (see Figure 2).
- *Pre-test predictions for NASA UPWT wind tunnel test*: Pre-test predictions using Euler and Navier-Stokes CFD were generated to ensure quality of the test results.

A total of 5,000 CFD runs were completed over the course of the project—approximately 4,500 Euler runs and 500 Navier-Stokes runs.

Role of High-End Computing: The X-51 Program would not have been able to complete the aerodynamic database work (within its time and budget constraints) without NASA's Columbia supercomputer. Many of the grids utilized in the work were too large to run on Boeing's systems. In addition, Boeing's systems did not have enough processors to provide the throughput for completing the work before the X-51 Critical Design Review. A total of ~1,400,000 processor-hours were consumed on Columbia to complete the project tasks.

Future: Future project goals include generation of CFD data to support the manufacturing and flight test phases of the project.

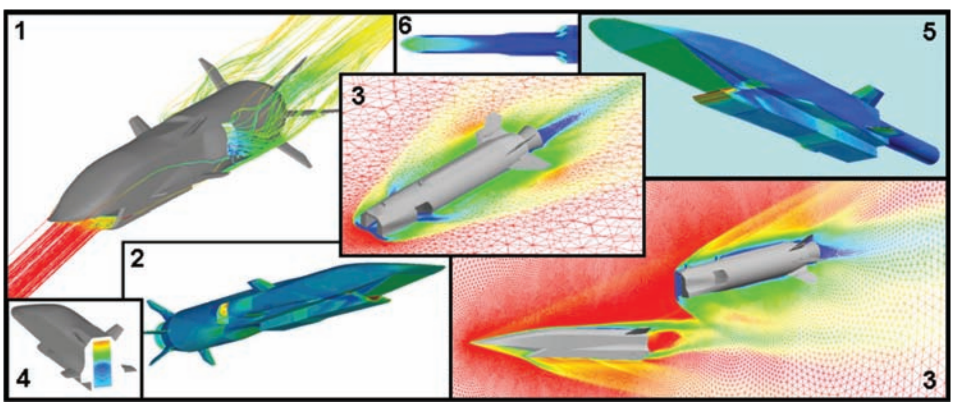


Figure 2: The figure shows a montage of CFD results that have been computed for the X-51 aerodynamics project using Columbia. Starting at top left: 1) OVERFLOW Navier-Stokes result for the X-51 stack at Mach 5; 2) Pressure coefficient (Cp) contours for the X-51 stack computed using the Cart3D code; 3) Stage separation results using the Boeing BCFD code; 4) Exit pressure contours using the OVERFLOW code, which were used in determining the proper thrust/drag bookkeeping; 5) BCFD results to determine influence of the sting on wind tunnel force and moments; 6) OVERFLOW N-S results for the cruiser at Mach 5.

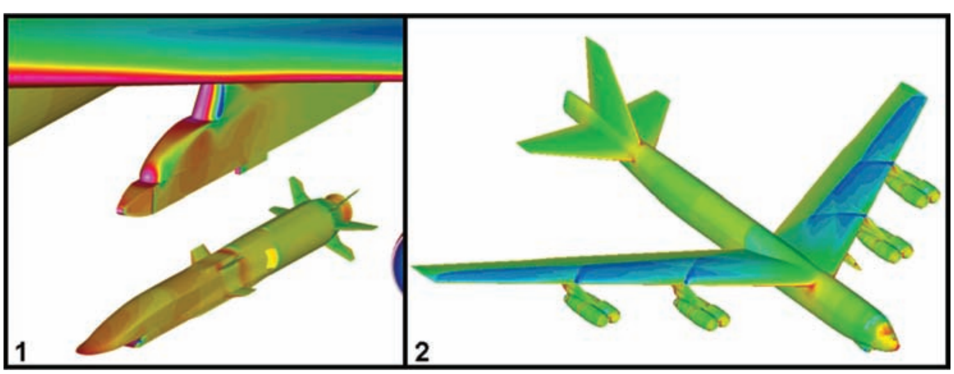


Figure 3: Two B-52 captive/carry and launch CFD cases are shown for the work conducted for the X-51 aerodynamics project: 1) Shows an OVERFLOW Navier-Stokes solution for the X-51 stack, 36 inches below the pylon at the Mach = 0.8 drop condition; 2) Shows the Cart3D-generated Cp contours for the B-52 with the X-51 stack attached at the Mach = 0.8 flight condition.