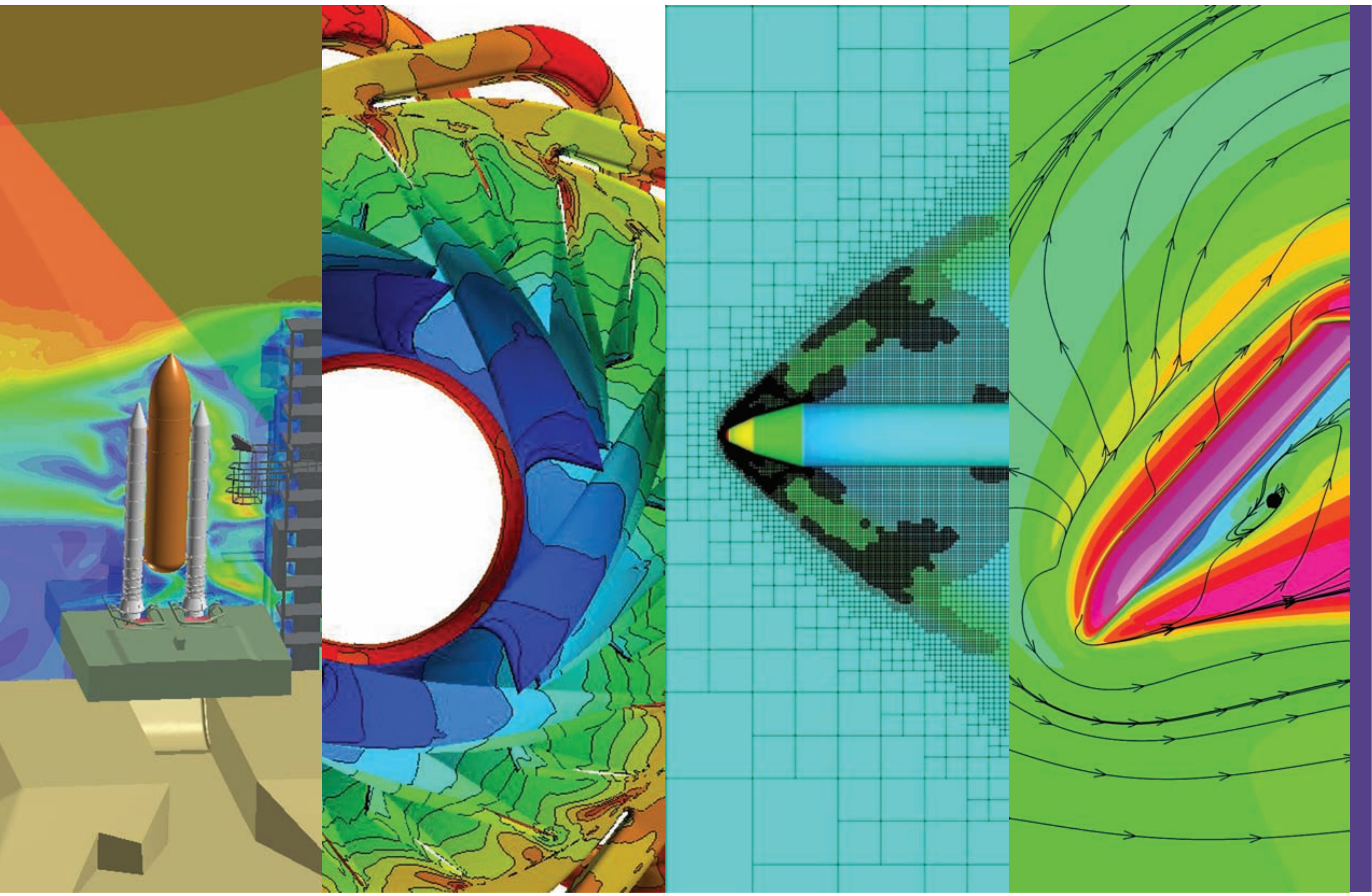


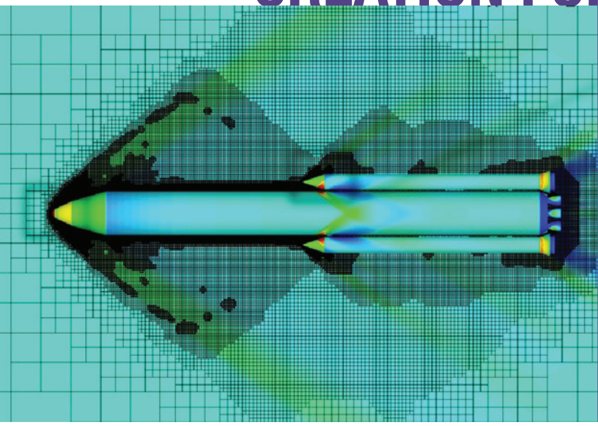
MISSION DIRECTORATE

SPACE OPERATIONS

The Space Operations Mission Directorate provides the Agency with leadership and management of NASA space operations related to human exploration in and beyond low-Earth orbit. Space Operations also directs low-level requirements development, policy, and programmatic oversight. Current exploration activities in low-Earth orbit are the Space Shuttle and International Space Station programs. The directorate is similarly responsible for Agency leadership and management of NASA space operations related to Launch Services, Space Transportation, and Space Communications in support of both human and robotic exploration programs.

WILLIAM H. GERSTENMAIER
Associate Administrator
http://www.nasa.gov/directorates/somd/home/index.html

AUTOMATED AERO-DATABASE CREATION FOR LAUNCH VEHICLES



SPACE OPERATIONS MISSION DIRECTORATE

MICHAEL J. AFTOSMIS
NASA Ames Research Center
(650) 604-4499
Michael.Aftosmis@nasa.gov
http://people.nas.nasa.gov/~aftosmis/cart3d/

◀ Detail of Figure 1.

Project Description: Computational fluid dynamics (CFD) is routinely used to analyze aerospace vehicle performance using high-fidelity methods at only a handful of critical design points. Recent progress in automated methods for numerical simulation of vehicle aerodynamics, however, now enables complete simulations to be performed with little to no human intervention. This progress coincides with the unprecedented increase in NASA's high-performance computing capacity afforded by the Agency's Pleiades and Columbia superclusters. These two concurrent developments place NASA in a unique position to explore the viability of developing fully automated aerodynamic performance databases for new aerospace vehicles. Such databases describe the aerodynamic performance of new vehicles throughout their entire flight envelope, and enable vehicles to be "flown" through the databases, to quantify vehicle performance for any candidate mission profile. These parametric studies vary both flight conditions as well as all permissible control surface deflections.

The aim of this project is to develop and deploy a prototype system for rapid generation of aerodynamic performance databases, and to use the system on real-world problems faced by NASA's Exploration Systems and Space Operations mission directorates. The project focuses on developing efficient, automated tools for generating aerodynamic data; and on establishing accurate, formal, and quantitative estimates of the uncertainty of this data. Such error estimates can be fed back into the simulations to improve their accuracy to customer-driven tolerances, yielding aerodynamic performance databases of certifiable quality.

These aerodynamic databases give a much broader picture of aerodynamic performance and can be used both in the preliminary design of new vehicles and for providing insight into the detailed aerodynamics of existing vehicles. The database can be used with six-degree-of-freedom (6-DOF) trajectory simulations coupled to guidance and control (G&C) systems. "Flying" a design through an aerodynamic database in faster-than-real time enables performance estimates for prototypes to be rapidly evaluated, and supports G&C system design. The broad utility of such simulation-based aero-performance data makes the accuracy of this data of paramount importance—underscored by NASA's development of an Agency-wide Standard for Models and Simulation, which outlines formal standards for quantitative estimation of errors in modeling and simulation data.

Relevance of Work to NASA: NASA's Exploration Systems Mission Directorate is currently faced with the challenge of developing the Orion Crew Exploration Vehicle (CEV), the Ares I Crew Launch Vehicle (CLV), and related exploration architecture systems to provide the nation's access to space after the planned retirement of the Space Shuttle. During vehicle development, NASA continues to fly and modify its fleet of Space Shuttle Launch Vehicles operated by the Space Operations Mission Directorate. With these challenges ahead, our need for advanced simulation technology, such as a system for automated aero-database creation, has never been greater.

Computational Approach: A typical CFD aerodynamic database contains on the order of 10^4–10^6 simulations, depending on the problem requirements. We employ Cart3D, a massively parallel, automated aerodynamic simulation package that scales linearly to thousands of processors. Automated tools drive this package to manipulate the geometry for control surface deflections; produce surface and volume grids; and manage the massive numbers of simulations that populate such large datasets. Error estimation tools shadow each simulation and automatically refine the computational grid to control numerical error and provide quantitative error bounds on the calculations. Further automation is used to harvest meaningful results and capture them in a performance database.

Results: This prototype database generation system has been developed largely in direct support of both the Space Shuttle and Constellation programs. Figures 1 and 2 show snapshots extracted from error-controlled simulations in performance databases of two different vehicles. Figure 1 shows supersonic aerodynamics on an early design of an Ares-based heavy-lift launch vehicle. The Ares simulations are part of a design effort to look for potential gains in aerodynamic performance through modifications to the vehicle's shape. This figure includes an overlay of the computational mesh, which the error-estimation module has refined to minimize errors in force coefficient predictions. Figure 2 shows simulations from large parametric studies—spanning multiple Mach numbers, angles-of-attack, and thrust settings—used to study the control effectiveness of forward-mounted jets on the Orion Launch Abort Vehicle (LAV).

The error-estimation module developed under this project has driven mesh refinement to resolve the intricate shock structures that occur when the (abort control motor) plumes (see Figure 2) first emerge from the vehicle, and to modify the shape of the plumes as they evolve downstream, affecting force and moments on the overall vehicle. The simulation tools developed under this project help assess such mechanisms with greater fidelity than ever before.

Role of High-End Computing: This unprecedented simulation capability is contingent upon high-end computing. Each simulation in a database typically has 15–50 million degrees-of-freedom, and a performance databases usually consist of 5–100 thousand such simulations. Calculations supporting error analysis and flowfield sensitivity approximately double this load. Using Cart3D and the prototype automation system, a dedicated node of the Pleiades or Columbia supercomputers can perform 10,000–20,000 simulations per hour. Given the low cost per processor-hour on these systems, this is by far the cheapest method available to obtain high-quality aerodynamic data.

Future: As NASA continues to develop vehicles for the Constellation Program, simulation requirements continue to grow. Not only is the number of designs to analyze increasing, but error-estimation and validation due-diligence mandate that we re-create wind tunnel test databases, as well. As Constellation evolves, these tools offer NASA an unprecedented ability to "fly" candidate designs through various mission profiles to gain insight into vehicle performance and carry out trade studies.

Co-Investigators

- Marsha J. Berger, Courant Institute, New York University
- Marian Nemec, ELORET Corp.

Publications

[1] Aftosmis, M.J. and Rogers, S. E., "Effects of jet-interaction on pitch control of a launch abort vehicle," *AIAA Paper 2008-1281*, Jan. 2008.

[2] Biswas, R., Aftosmis, M.J., Kiris, C., and Shen, B.-W., "2007: Petascale Computing: Impact on Future NASA Missions," *Petascale Computing: Algorithms and Applications* (D. Bader, ed.), Chapman and Hall /CRC Press, pp. 29–46, Dec. 2007.

[3] Mavriplis, D.J., Aftosmis, M.J., and Berger, M.J., "High resolution aerospace applications using the NASA Columbia supercomputer," *International Journal of High Performance Computing Applications*, 21(1), pp. 106–126, Jan. 2007.

[4] Nemec, M., and Aftosmis, M.J., "Adjoint error-estimation and adaptive refinement for embedded-boundary Cartesian meshes," *AIAA Paper 2007-4187*, 18th AIAA CFD Conference, Miami, FL, Jun. 2007.

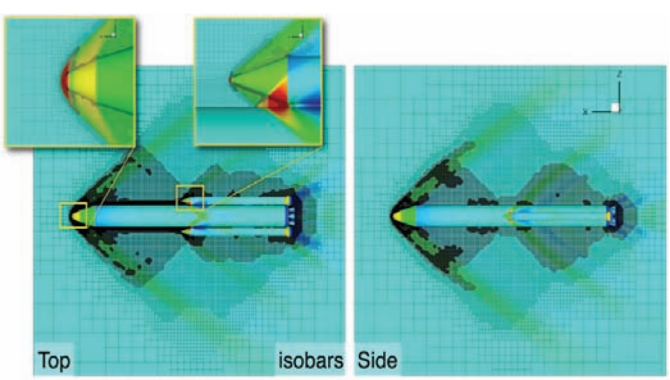


Figure 1: Top and side views of an Ares-based heavy-lift launch vehicle at supersonic conditions.

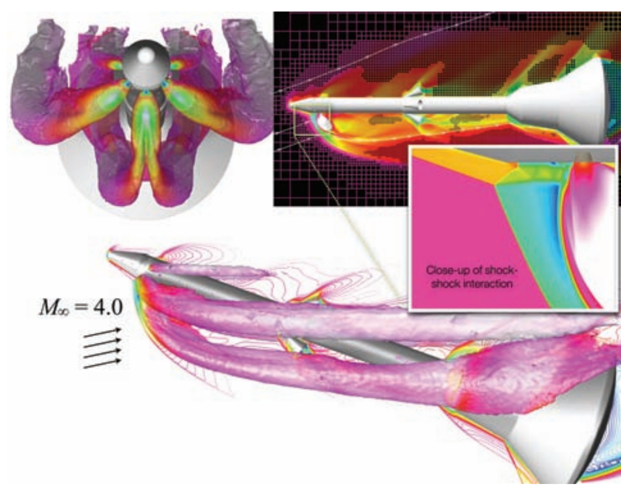
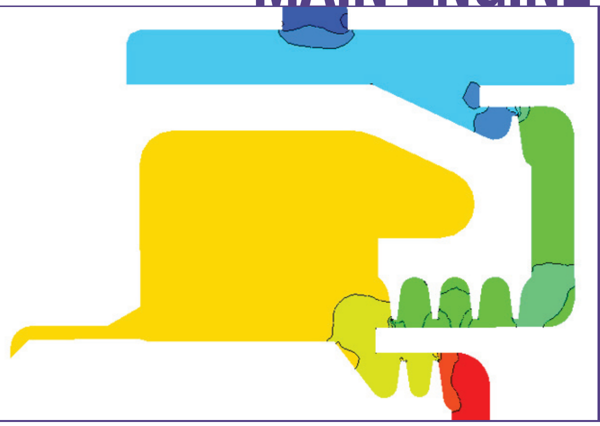


Figure 2: This set of figures shows front and side views of the Orion Launch Abort Vehicle at Mach 4, with the abort control motors firing. Cart3D was used with the adjoint-based error-estimation and mesh adaptation module to control numerical error in predicted aerodynamic forces and capture the many scales of the flow.

CFD ANALYSIS OF SHUTTLE MAIN ENGINE TURBOPUMP SEAL CRACKS



SPACE OPERATIONS MISSION DIRECTORATE

KRIS MCDOUGAL
Marshall Space Flight Center
(256) 544-0659
Kristopher.McDougal@nasa.gov

DAN DORNEY
Marshall Space Flight Center
(256) 544-5200
Daniel.J.Dorney@nasa.gov

◀ Close-up of Figure 1.

Project Description: The primary goal of this effort was to utilize computational fluid dynamics (CFD) models to characterize the flow drivers that were suspected as the root cause cracking mechanism on the Space Shuttle Main Engine (SSME) High Pressure Oxidizer Turbopump (HPOTP) Knife Edge Seals (KES) (Figure 1). The CFD analysis also provided detailed load characterization at discrete engine operating conditions.

The SSME HPOTP uses knife-edge seals to control leakage flows that pass between turbine rotating and stationary parts. The leakage flow on the backside of the HPOTP third-stage turbine is controlled by a series of seals. After hot-fire testing, two of these seals were found cracked or broken on several turbopumps. In this project, CFD models have been developed to provide insight into the root cause of these cracks.

Relevance of Work to NASA: This project provided analyses that contributed to identifying the root cause of the seal cracks. Furthermore, the analysis offered guidance for a new design, and was extended to provide dynamic fluid loads. Starting with shuttle mission STS-114, this new design has been incorporated in high-pressure oxidizer turbopumps that have successfully flown on the Space Shuttle.

Computational Approach: The code used in the numerical KES simulations was Loci-CHEM. Loci-CHEM (version 2) is a finite-volume flow solver (with combustion kinetics) for generalized grids. Developed at Mississippi State University, in part through NASA and National Science Foundation funding, Loci-CHEM uses high-resolution approximate Riemann solvers to simulate finite-rate, chemically reacting, viscous turbulent flows. Several turbulence models are available, including the Spalart-Allmaras one-equation model and a family of three-equation turbulence models. Loci-CHEM is comprised entirely of C and C++ coding and is supported on all popular Unix variants and compilers. Parallelism is supplied by the Loci15 framework, which exploits multithreaded and Message Passing Interface libraries.

Results: Pre-test three-dimensional (3D) steady and unsteady CFD predictions showed good-to-excellent agreement with the ensemble of steady and unsteady pressure data measured in a cold flow KES air-flow test model. The two-dimensional (2D) unsteady CFD predictions were conservative such that the predicted unsteadiness was greater than that observed in the experiments. Therefore, we have concluded that the 2D unsteady CFD tooth-loading predictions used in the structural response analyses of the redesigned seals (Figure 2) are conservative with respect to the acoustic loadings in the engine, and that the redesigned KES are sufficient.

Role of High-End Computing: Two-dimensional CFD grids containing on the order of 3 million grid points, as well as 3D grids containing on the order of 13 million grid points were run to model the SSME HPOTP KES for investigation. A steady simulation was first run for each condition, followed by an unsteady simulation initiated from the steady solution. Computations of this magnitude require intensive levels of processing. The simulations were run on 20–160 processors of SGI Altix Linux clusters located at NASA's Marshall Space Flight Center and Ames Research Center. Without NASA high-end computing (HEC) resources, this analysis could not have supported the timely redesign of the KES.

Future: While the SSME HPOTP KES project has been closed out with no immediate need for additional modeling, many CFD analyses and models are currently benefitting from the vast computing capabilities of the HEC Program. Examples include CFD analyses of the cracked STS-126 Flow Control Valve to support STS-119's launch, as well as J-2X turbomachinery CFD analyses which is on the J-2X engine design critical path.

Publications

[1] Dorney, D., "Comparison of Pre-Test Predictions and Experimental Data for Knife-Edge Seals," Joint Army Navy NASA Air Force (JANNAF) Meeting, 54th Propulsion Meeting, May 2007.

[2] Dorney, D., "Investigation of the Flow in a Series of Knife-Edge Seals," JANNAF, 53rd Propulsion Meeting, Dec. 2005.

[3] Hawkins, L., "SSME High Pressure Oxidizer Turbopump Find 072 Inner," and, 011 Outer Turbine Outlet Duct Seal *INCREMENTAL VERIFICATION COMPLETE REPORT (FINAL) VRS-0648-2*, Contract NAS8-01140, March 2007.

[4] McDougal, K., "NASA MSFC Knife Edge Seal Air Rig Post Test Analysis *Report Test Number P2499*, March 2006.

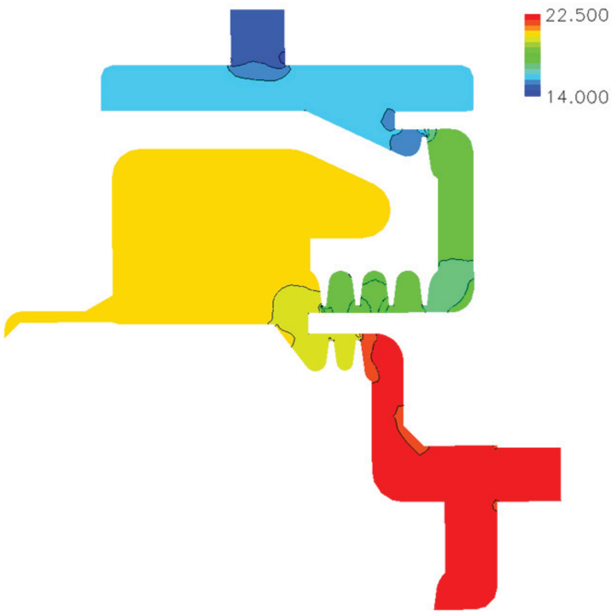


Figure 1: Cross-section of a representative design case of Space Shuttle Main Engine (SSME) Knife Edge Seal time-averaged static pressures at design conditions.

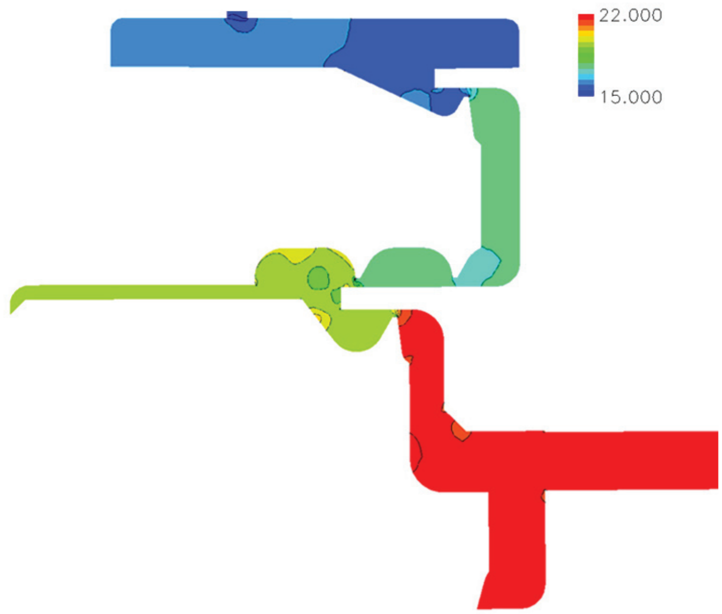
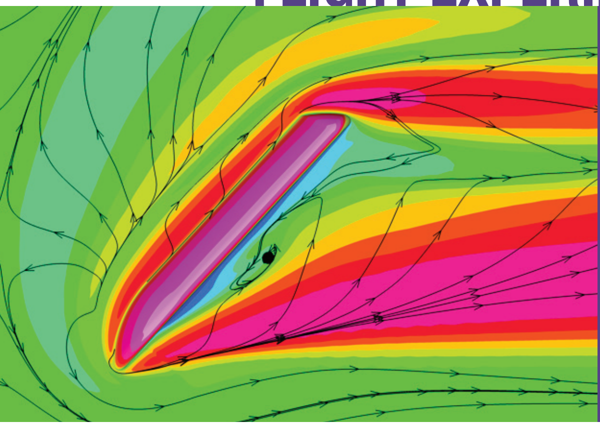


Figure 2: Cross-section of the SSME Knife Edge Redesign time-averaged static pressures at redesign conditions.

NUMERICAL ANALYSIS OF BOUNDARY LAYER TRANSITION FLIGHT EXPERIMENTS ON THE SPACE SHUTTLE



SPACE OPERATIONS MISSION DIRECTORATE

CHUN TANG
NASA Ames Research Center
(650) 604-3480
Chun.Y.Tang@nasa.gov

WILLIAM WOOD
NASA Langley Research Center
(757) 864-8355
William.A.Wood@nasa.gov

◀ Close-up of Figure 2.

Project Description: The objective of this work is to study the behavior of early flow transition due to the presence of discrete roughness elements. Since transition from laminar to turbulent flow can result in significant increases in heating on the thermal tiles of the Space Shuttle during reentry to Earth, a better understanding of this flow phenomenon is useful in estimating the impact that protuberances (such as protruding gap fillers and thermal blankets) may have on the safe return of the Orbiter. In addition, knowledge gained from the numerical simulations and experimental data will lead to better designs of thermal protection systems (TPS) for future spacecraft such as the Orion crew exploration vehicle and planetary probes.

A boundary layer transition (BLT) flight experiment, scheduled for launch in March 2009 onboard shuttle mission STS-119, consists of a 0.25-inch tall by 4-inch long protuberance placed on the windward surface of the Orbiter along with pressure sensors and calorimeters downstream from the protrusion. The goal of the experiment is to measure any elevated heating caused by flow transition. Numerical analysis of the BLT protrusion is used to predict peak heating rates on the surface of the protuberance and the surrounding acreage tiles during the Orbiter's reentry. These solutions are used to select the appropriate TPS material and shape of the protuberance.

Relevance of Work to NASA: This work is highly relevant to the safety of NASA's shuttle fleet and crews, since local protuberances such as protruding gap fillers may cause early transition from laminar to turbulent flow and result in higher heating on the Orbiter's heat tiles. The ability to accurately predict the aerothermal effects of these protrusions is important in real-time safety assessments during a shuttle mission. A better understanding of flow transition is also important in optimizing the design of future spacecraft for space exploration. Additional BLT flight experiments will include testing of Orion TPS materials in the turbulent zone created by the protuberance. Data and computer simulations from these experiments improves our understanding of the ablation process in a turbulent hypersonic flow environment.

Computational Approach: High-fidelity computational fluid dynamics (CFD) Navier-Stokes codes developed at NASA Ames and NASA Langley Research Centers are used to predict the aerothermal effects of a protuberance on the Space Shuttle. Instead of running the CFD solver on the entire vehicle, a "local" rapid analysis process (developed for real-time analysis of damages sustained by the shuttle during a mission) is used to evaluate the flow around the BLT protrusion. This approach uses existing solutions from the Return-to-Flight (RTF) database (for example, see Figure 1) to provide boundary conditions for a local simulation. The heating environment (as seen in Figure 2) is then fed into a thermal analysis model to predict the internal and bondline temperatures of the protrusion and surrounding heat tiles. This information is used to assess possible risks associated with the BLT flight experiment.

Results:

- The original BLT design plus seven geometry revisions were analyzed at flight conditions using two CFD codes, Data-Parallel Line Relaxation (DPLR, developed at NASA Ames) and LAURA (developed at NASA Langley).
- Arc jet experiments and corresponding arc jet CFD simulations were completed at NASA Johnson Space Center.
- BLT Flight Experiment #1 launched aboard STS-119 (March 2009).
- Additional laminar and turbulent CFD simulations on taller protuberances were computed in preparation for future BLT flight experiments.

Role of High-End Computing: NASA's Columbia supercomputer provided the necessary resources to quickly compute the protuberance simulations. For example, each protrusion calculation took approximately 8 hours of wall-time using 70 processors on Columbia. This rapid turnaround allowed us to run numerous simulations to optimize the protuberance shape for minimum surface heating. In addition, personnel at the NASA Advanced Supercomputing facility are optimizing the DPLR code for maximum performance. These improvements should result in better turnaround time for obtaining high-fidelity CFD solutions, which will be useful for real-time risk assessment during a shuttle mission.

Future: Data gathered from the first flight experiment will be used to assess accuracy of the numerical predictions. It is expected that post-flight CFD simulations will be needed to match the actual freestream conditions of the experiment. Two additional BLT flight experiments are planned for future Orbiter missions. These experiments will involve taller protuberances to induce earlier flow transition and the testing of different TPS materials to investigate the behavior of catalycity in a turbulent flow environment. Thus, more CFD simulations at flight and arc jet conditions will be needed to support these future flight experiments. Once again, supercomputers will play a critical role in providing the resources needed to quickly compute and post-process these calculations.

Co-Investigators

- Kerry Trumble, NASA Ames Research Center
- Victor Lessard, Genex Systems
- Charles Campbell, NASA Johnson Space Center

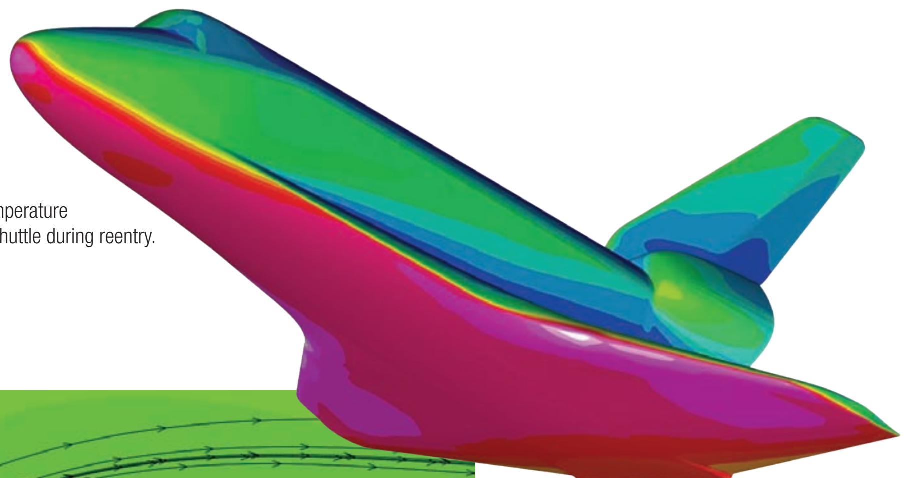


Figure 1: Computed temperature contours on the Space Shuttle during reentry.

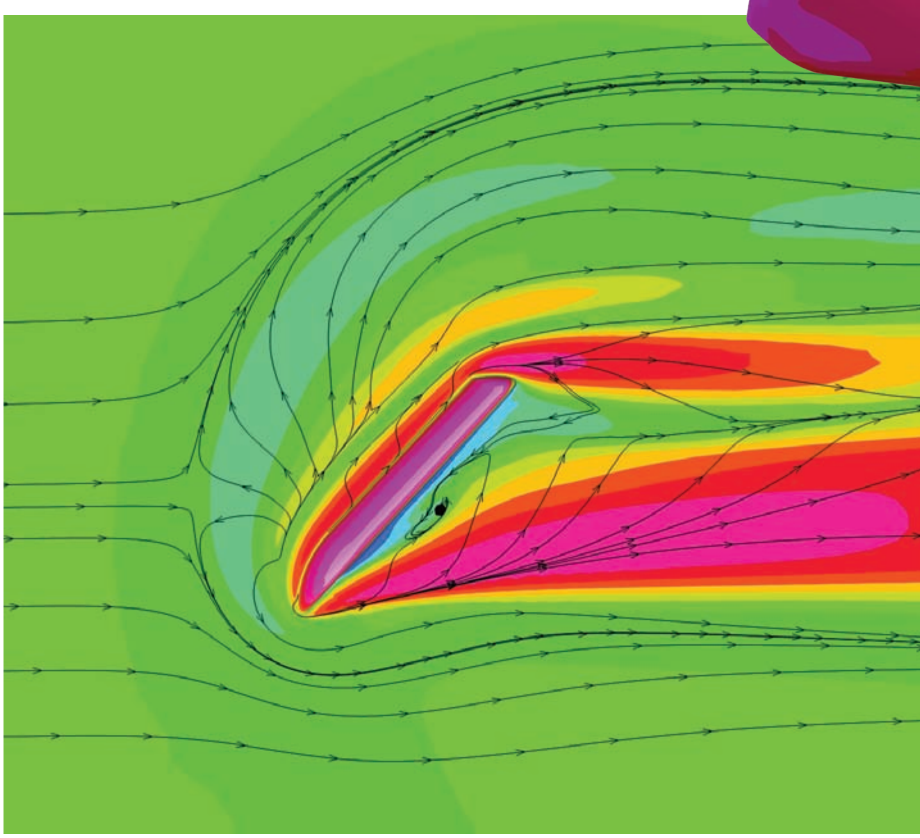


Figure 2: Computed streamlines and temperature contours on the boundary layer transition protuberance during shuttle reentry.

SPACE SHUTTLE ASCENT AERODYNAMICS AND DEBRIS TRANSPORT ANALYSES

SPACE OPERATIONS MISSION DIRECTORATE

REYNALDO J. GOMEZ III
NASA Johnson Space Center
(281) 483-6108
Reynaldo.J.Gomez@nasa.gov

◀ **Figure 1:** Surface pressure and shockwaves for the Space Shuttle at Mach 1.4.

Project Description: The Space Shuttle Program relies on detailed computational simulations to assess the wide range of aerodynamic and aerothermodynamic environments encountered during a shuttle mission. From redesigns to eliminate potential ice and foam debris, to inflight assessments used to clear the shuttle for entry and landing, the Columbia supercomputer plays a key role in NASA's Space Operations Mission Directorate. Probabilistic debris risk assessments use ascent computational fluid dynamics (CFD) flowfields along with probabilistic aerodynamic models to prioritize redesigns of the external tank and inspections on the Space Shuttle Orbiter.

This project supports Space Shuttle aerodynamic, aerothermodynamic, and debris transport assessments through the use of high-fidelity, unsteady CFD simulations. Redesigns of the shuttle's external tank have progressed so rapidly and involved such small details of the geometry that wind tunnel testing would be difficult to schedule and instrument. Computational models, anchored to previous wind tunnel tests, have been used to produce the various types of data required to assess these redesigns. Additionally, debris transport assessments use these flowfields to predict debris trajectories, and are key inputs for the probabilistic risk assessments used to prioritize external tank redesigns.

Relevance of Work to NASA:

- The last major outer mold line change to the Space Shuttle external tank was the replacement of four of the LO_2 feedline brackets with titanium (Ti), covered by a minimal amount of thermal protection system foam. The lower thermal conductivity of the Ti brackets enabled the shuttle program to reduce the amount of potential foam and ice debris that could be shed by the brackets, and increased clearance between the brackets and external tank. Detailed simulations of this configuration change, run on the Columbia supercomputer, were a key part of the redesign assessments and provided insight into flowfield details that would be difficult or impossible to extract from a wind tunnel test. Surface grids and a representative flowfield around the launch vehicle with these modifications are shown in Figures 1 and 2.
- Maximum allowable iceball maps, created using in-house debris transport tools, have also been assessed using Columbia. The primary result of these analyses is a map of the allowable ice-ball size that can be shed from the external tank without exceeding the impact-damage threshold for a given component. The result of this study has been incorporated as a launch commit criteria in the document "*Space Shuttle Ice/Debris Inspection Criteria.*"

Computational Approach: NASA's overset CFD flow solver, OVERFLOW, is the primary tool used to produce ascent airloads on the Space Shuttle. Typically, 64–128 processors are used in parallel to produce a set of solutions along an ascent or entry trajectory. NASA's Cart3D moving body Euler code is also used to produce probabilistic aerodynamic models for a wide range of debris shapes.

Results:

- *Iceball Allowables Debris Transport Analyses:* The results of these analyses are used during the prelaunch countdown to ensure that the shuttle is not launched with a potentially dangerous ice accumulation on the external tank [1].
- *OVERFLOW Simulations:* Improvements include the extension of transonic, slender-body performance to high-speed, complex configurations, and significant robustness enhancements for lower-speed flows with high-speed jets. The increased robustness has paid significant dividends for analyzing the shuttle and next-generation Orion space vehicle designs. The improved CFD code has also been applied to Space Shuttle Orbiter calculations. In a case at Mach 4, the modified code achieved a two-orders-of-magnitude reduction in processor time to converge the solution on Columbia [2].

- *External Tank ET-128 Ascent Airloads, Aerothermal, and Debris Transport Simulations:* This project used the OVERFLOW code to simulate the flowfield around the last major outer mold line change to the external tank. This design change replaced the aluminum LO_2 feedline brackets with Ti, decreasing the potential for ice and foam debris from these components, and increasing safety of the vehicle and crew. The detailed pressure distributions extracted from these solutions were used to assess changes to the aerodynamic and aerothermodynamic loads on the modified components. Further simulations are currently being used to simulate ascent debris environments and address inflight issues [3].

Role of High-End Computing: Producing an assessment database for a geometry as complex as the Space Shuttle Launch Vehicle is a computationally intensive task. Each unsteady ascent simulation of the new design required approximately 7,600 processor-hours and 46 gigabytes of disk space for the time varying results. Approximately 625,000 processor-hours were used to produce 80 solutions of the new design at a range of ascent flight conditions. The complex nature of the flowfield around this complex geometry requires the use of unsteady simulations significantly more computationally expensive than simpler steady-state solutions. Each of these time-accurate simulations is time averaged to create ascent flowfield environments.

Future: Currently, NASA is planning to retire the shuttle in 2010. Modifications to the external tanks and procedural changes have slowed production, and we have been using CFD tools to assess a number of reproducibility enhancements intended to reduce the time required to manufacture external tanks without affecting launch safety.

Co-Investigators

- Stuart Rogers, Scott Murman, and Edward Tejnil, all of NASA Ames Research Center
- Darby Vicker, NASA Johnson Space Center

Publications

[1] Rogers, S. E., "Hemisphere Ice Ball Debris-Transport Analysis," NAS Technical Report," NAS-07-004, June 2007.

[2] Murman, S.M., "Dynamic Stability Analysis using CFD Methods," in Experimental Determination of Dynamic Stability Parameters, von Karman Institute for Fluid Dynamics, February 2008.

[3] Tejnil, E., Rogers, S. E., and Gomez, R. J., "OVERFLOW CFD Analysis of SSLV with ET-128 Design Changes,"June 2008.

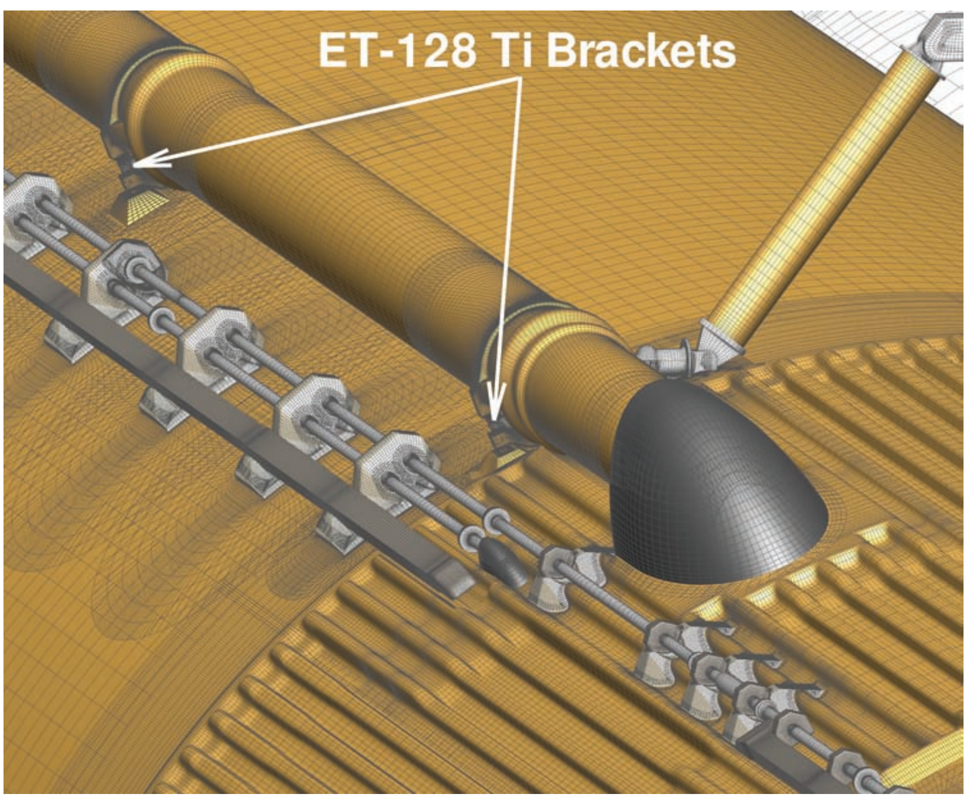
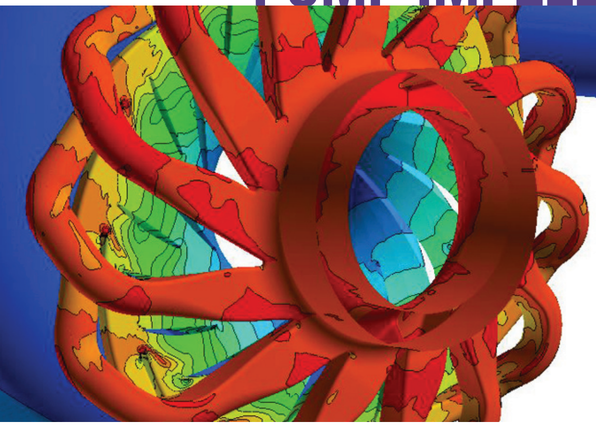


Figure 2: Surface grids for the ET-128 external tank redesign.

SSME HIGH PRESSURE FUEL PUMP IMPELLER CRACK INVESTIGATION



SPACE OPERATIONS MISSION DIRECTORATE

PRESTON SCHMAUCH
NASA Marshall Space Flight Center
(256) 544-1218
Preston.B.Schmauch@nasa.gov

◀ Close-up of Figure 1.

Project Description: In 2007, a liquid dye penetrant test revealed a crack on a Space Shuttle Main Engine (SSME) High Pressure Fuel Pump (HPFP) impeller mid-blade. In order to more fully understand the events leading to the impeller crack initiation, as well as the dynamic environment that caused the crack to grow, various unsteady computational fluid dynamics (CFD) simulations were performed at SSME power levels experienced by the cracked impeller. These simulations calculated unsteady blade loading and associated frequencies that were used to determine expected part life. The simulations were also run with multi-phase flow to determine at what power levels cavitation was likely to exist near the crack site. Additionally, running conditions from prior water flow testing on the HPFP were compared with unsteady CFD water testing simulation results to validate and correlate the computational model with the engine running conditions. Numerous simulations that varied the inlet guide vane trailing-edge thickness were also run to determine whether vortex shedding of the inlet guide vane could have been a contributing factor.

Relevance of Work to NASA: With safety as one of NASA's primary goals, it is imperative to fully analyze damage to any critical parts of the SSME. These simulations provided key information to the impeller crack investigation team so they could make educated decisions regarding the cause of the crack and the next course of action. Data used to support those decisions included the number of engine starts since crack initiation, mean and alternating stresses near the crack location, and expected margin of safety at that location. Those decisions led to the new requirements placed on the impeller related to part life and intervals between inspections. This process helps to ensure the safety of our astronauts, NASA employees and contractors, and helps to improve future designs.

Computational Approach: Phantom, a NASA code that has been anchored and validated for supersonic turbines, was used for the unsteady CFD simulations. Phantom uses three-dimensional, unsteady Navier-Stokes equations as the governing equations. The Baldwin-Lomax turbulence model is used for turbulence closure. For this simulation, H grid topology with moving grids was employed to simulate blade motion. In order to accurately resolve flow features for the impeller it was necessary to simulate the inlet scroll, inlet guide vane, impeller, and crossover diffuser (see Figures 1 and 2). Merlin, a code similar to Phantom but with two-phase flow capabilities, was used to simulate the unsteady CFD with cavitation effects.

Results: Simulation results have been used to help understand the original robustness of the impeller design and factors involved in crack initiation and growth. These simulations helped the investigation team to arrive at three conclusions based on the unsteady CFD solutions. First, all unsteady rotor-stator flow drivers appear to affect all flow channels, and a rotating stall or a planar disturbance driver will be experienced by each blade. Second, results indicated that cavitation would be present on the mid-length blade. Third, further investigation into the vibration phenomena seen in the CFD simulation and from experimental data is recommended to gain a better understanding of the loading impacts. It was determined that the unsteady rotor-stator loads seen in the CFD were the source for the cracking. The reasons that only one blade on one pump has experienced cracking—especially since other flight engines have higher cumulative run times than the affected engine—are still unknown. All simulations pertaining to this investigation have been completed.

Role of High-End Computing: Quick simulation turnaround time in this study allowed multiple suspected causes of the crack to be investigated. Suspected crack initiators were simulated, analyzed, and determined to be either a contributor or non-factor. HEC resources significantly reduced overall simulation time and allowed the team to focus more on pertinent areas of investigation. The processor power, processor time, and high-speed file transfers were important factors in getting the work done in a timely manner. The simulations run for this investigation totaled over 300,000 processor-hours on the Columbia supercomputer.

Future: Future work utilizing HEC resources includes investigations into vibratory loading effects on the impeller blade. Due to the high mass flow variation from passage to passage of the impeller, there is a corresponding velocity and incidence change at the leading edge of the blade. The resulting loading effect could be a factor in crack initiation. Other areas of interest involve adding secondary two-phase leakage paths to the current simulations for a better model of the SSME HPFP flow environments.

Co-Investigators

- Daniel Dorney, NASA Marshall Space Flight Center

Publications

[1] Marco B., Zabo R., Dorney D., and Zoladz T., "The Effect of Acoustic Disturbances on the Operations of the Space Shuttle Main Engine Fuel Flowmeter," 43rd AIAA Joint Propulsion Conference, *AIAA 2007-5534*, July 2007.

[2] Dorney D., Griffin L., Marcu B., and Williams M., "Unsteady Flow Interactions between the LH2 Feedline and SSME LPFP Inducer," 42nd AIAA Joint Propulsion Conference, *AIAA 2006-5073*, July 2006.

[3] Marcu B., Tran K., Dorney D., and Schmauch P., "Turbine Design and Analysis for the J-2X Engine Turbopumps," 44th AIAA Joint Propulsion Conference, *AIAA 2008-4660*, July 2008.

[4] Dorney D., Griffin L., and Schmauch P.,"Unsteady Flow Simulations for the J-2X Turbopumps," 54th JANNAF Propulsion Meeting, May 2007.

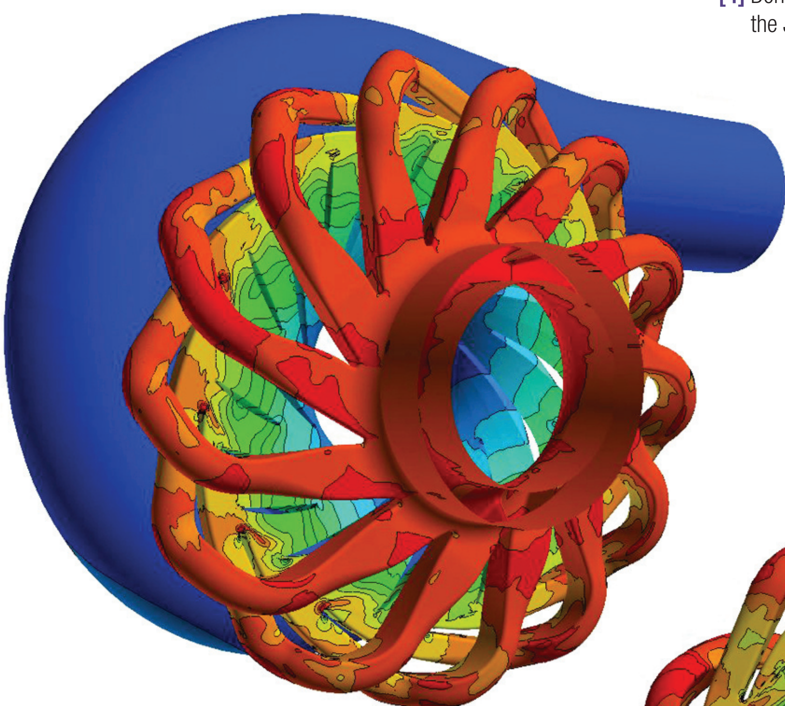


Figure 1: Space Shuttle Main Engine (SSME) High Pressure Fuel Pump static pressure. View is looking upstream from the first crossover diffuser exit.

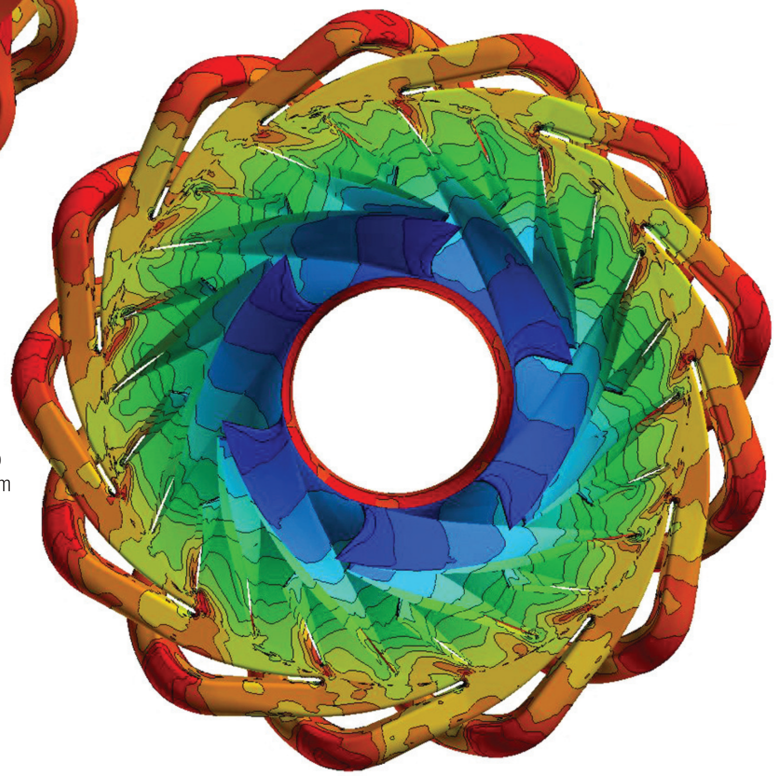
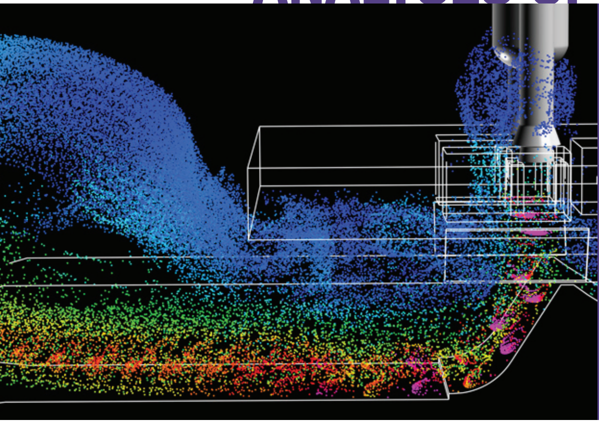


Figure 2: SSME High Pressure Fuel Pump static pressure. View is looking downstream from the impeller inlet.

TIME-ACCURATE COMPUTATIONAL ANALYSES OF THE LAUNCH PAD FLAME TRENCH



SPACE OPERATIONS MISSION DIRECTORATE

CETIN KIRIS
NASA Ames Research Center
(650) 604-4485
Cetin.C.Kiris@nasa.gov

◄ Detail of Figure 1.

Project Description: During ignition of rocket propulsion systems such as that of the Space Shuttle, interaction of the exhaust plume with the flame trench below the launch pad produces a series of strong pressure waves that travel back through the inlet of the trench, where they may affect the stability of the launch vehicle during takeoff. For this project, we have performed time-accurate computational fluid dynamics (CFD) simulations to characterize ignition overpressure (IOP) phenomena during liftoff of new and existing launch vehicles. In preparation for the launch of the Ares I-X flight test vehicle in 2009, we developed a computational model to investigate the feasibility of using the shuttle mobile launch platform (MLP) for next-generation Ares launch vehicles.

To validate the geometric model and CFD procedure, we first simulated IOP waves generated during ignition of shuttle mission STS-1, which did not use a water suppression system, and compared results with measured data (Figure 1). The model was then modified to study the effects of different MLP configurations on IOP waves for the Ares I-X vehicle, represented by a single, modified solid rocket booster (SRB) positioned over the left flame trench hole. Ares I-X ignition simulations were performed for three alternate configurations of the existing MLP (which has two exhaust holes with deflectors to direct flow into the flame trench): both holes open with both deflectors present (Figure 2); both holes open with right deflector removed; and right hole closed with right deflector removed.

The flame trench computational model has also subsequently been used to support repair efforts at NASA Kennedy Space Center after the launch of STS-124 damaged a large section of the trench wall. Time-accurate flow data and pressure values were provided for 21 points on the flame trench wall and at six inches out from each point.

Relevance of Work to NASA: This work has provided valuable insight into launch environments for NASA's current and future space vehicles. CFD analyses provide an efficient, cost-effective means of reassessing ground operations infrastructures to determine and plan modifications required for the Ares launch vehicles. This study has demonstrated the value of these efforts by showing that the MLP exhaust holes will not need to be modified for the Ares I-X vehicle, saving millions of dollars in engineering and construction expenses. Additionally, computed flame trench pressure values helped determine structural requirements for the critical repair of flame trench damage that occurred during the launch of STS-124.

Computational Approach: Using a grid generation script based on the Chimera Grid Tools library, structured viscous overset grid systems were built to model the different launch site configurations, including the flame trench, MLP, plume deflectors, support structures, launch vehicles, and surrounding terrain. The overset grid and scripting approach facilitated modifications to the grids to model different options for the launch vehicle, MLP, and deflectors. The Space Shuttle grid system contained 129 zones and 92 million grid points, and grid systems for the single SRB with various MLP options contained 92 to 120 grids, and 73 to 87 million grid points.

The NASA-developed CFD code OVERFLOW, an implicit, structured, overset, Reynolds-Averaged Navier-Stokes solver, was used to simulate the exhaust plume interaction with the flame trench. To capture the correct plume behavior with a single species model, the SRB chamber conditions were modified to obtain correct nozzle exit thrust, temperature, and Mach number conditions critical to simulation accuracy. Impulsive start conditions were used for STS-1 and Ares I-X simulations to assess the magnitude of the IOP waves, while a transient ramp-up of SRB chamber conditions was applied for the later wall damage simulations.

Results: For the initial validation study, good correlation was found between the predicted CFD results and the recorded flight data for STS-1. Peak pressure levels agreed closely, and qualitative behavior also compared well once the acoustic noise was removed from the flight data. Results of the Ares I-X MLP exhaust hole studies showed that, in each configuration, the peak IOP waves are reflected from the SRB's own exhaust hole, and that blocking the right hole and removing the deflector has no effect in reducing the IOP on the test vehicle. Comparison of the predicted pressure peaks with the STS-1 data showed that similar IOP behavior is predicted for all three of the alternate configurations, and that none of them will generate significantly larger peak pressures than those experienced by STS-1. With additional damping provided by the launch pad's current water suppression system, which was excluded from the STS-1 model, the shuttle MLP should provide an adequate launch platform for the Ares I-X vehicle without requiring costly modifications to the right-side hole.

Results of the STS-124 flame trench wall damage study show that the initial pressure wave reaches each of the 21 examined points according to the distance from the main deflector. Peak pressure magnitudes were found to be highest near the flame trench floor and lower near the top of the trench. These computed pressures were compared to STS-4 flight data and showed good correlation, although some differences were observed due to the presence of the water suppression system used for the STS-4 launch.

Role of High-End Computing: The HEC resources supported by the NASA Advanced Supercomputing facility were essential to performing the large-scale, time-accurate CFD simulations needed for these studies. Using 128 processors on the Columbia supercomputer, computation of two seconds of ignition conditions for the STS-1 and Ares I-X configurations required several weeks of runtime. For the time-critical wall damage support study, unsteady flame trench simulations were carried out to 1.15 seconds in four and a half days using 504 processors on Columbia.

Future: Significantly more modeling and simulation of ground operations infrastructures and environments will be required to prepare for the Ares I Crew Launch Vehicle and the Ares V Cargo Launch Vehicle in the coming years. In addition to the launch pad configurations, the Vehicle Assembly Building at NASA Kennedy is also being modeled to evaluate its potential future use for the Ares vehicles.

Co-Investigators

- Jeffrey Housman, Daniel Guy Schauerhamer, Marshall Gusman, all of ELORET Corp.
- William Chan, Dochan Kwak, both of NASA Ames Research Center

Publications

[1] Kiris, C., Chan, W., Kwak, D., Housman, J., "Time-Accurate Computational Analysis of the Flame Trench," Fifth International Conference on Computational Fluid Dynamics, Seoul, Korea, July 2008.

[2] Kiris, C., Schauerhamer, D.G., Housman, J., Gusman, M., Chan, W., Kwak, D., "Time-Accurate Computational Analysis of the Flame Trench Applications," 21st International Conference on Parallel Computational Fluid Dynamics, May 2009.

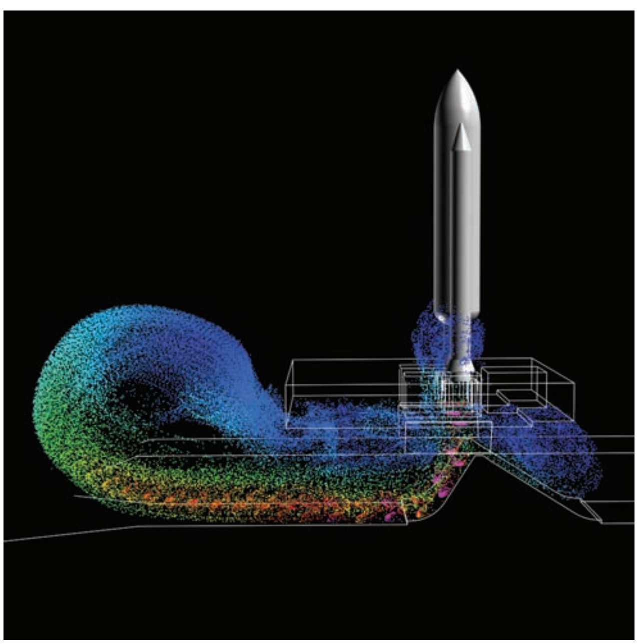


Figure 1: Visualization of shuttle flame trench CFD simulation showing instantaneous particle traces colored by Mach number.

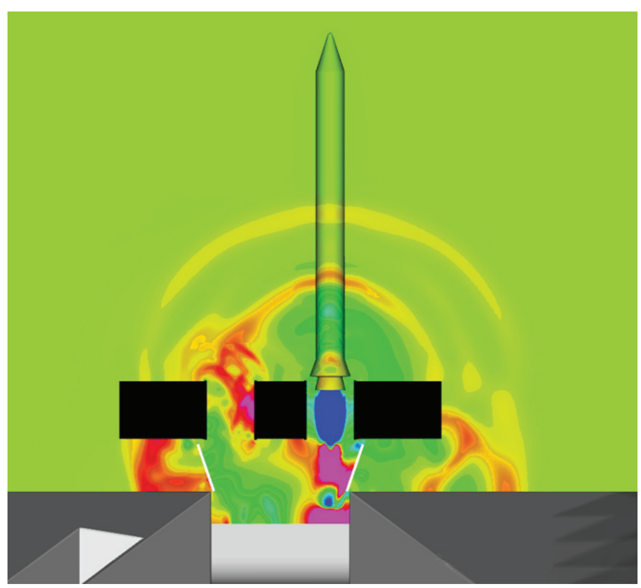


Figure 2: Instantaneous ignition overpressure waves for a single solid rocket booster ignition on the shuttle mobile launch platform configured with right hole open and right deflector present.