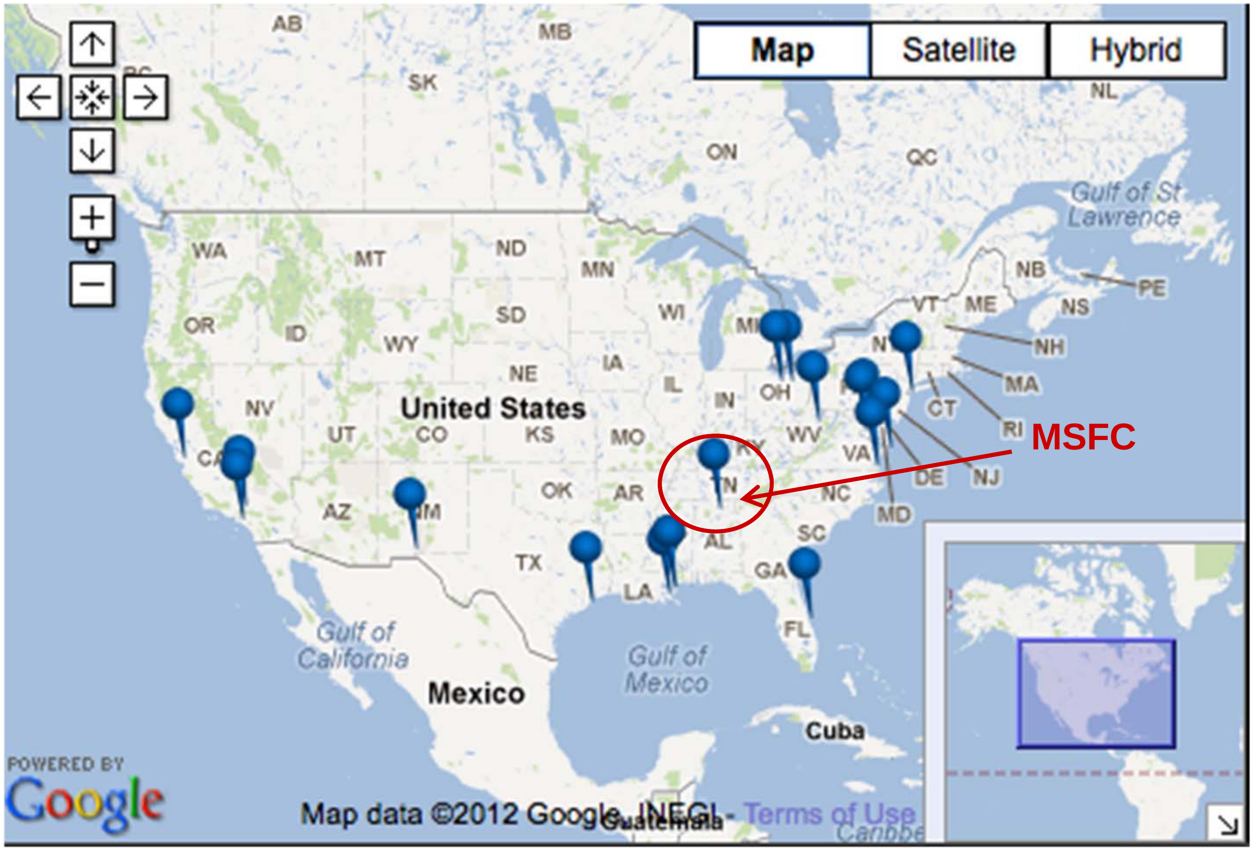
The Direction of Fluid Dynamics for Liquid Propulsion at NASA Marshall Space Flight Center

Presented by Lisa W. Griffin
Chief, Fluid Dynamics Branch – ER42
NASA Marshall Space Flight Center
Lisa.W.Griffin@nasa.gov

Advances in Rocket Engine Modeling and Simulation, and its Future
Tokyo, Japan
September 26 – 27, 2012

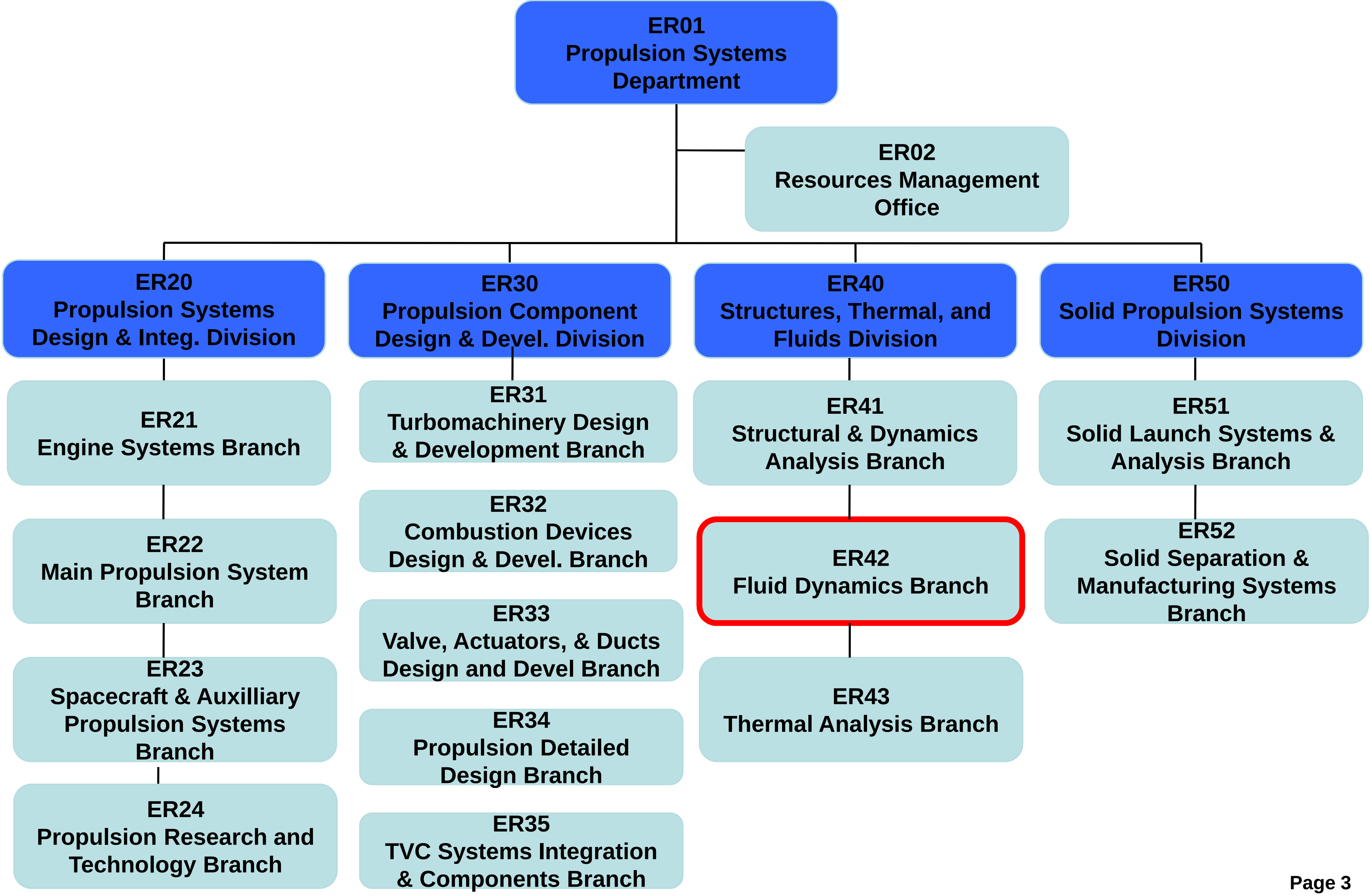
NASA MARSHALL SPACE FLIGHT CENTER

Marshall Space Flight Center (MSFC) is one of ten NASA field centers. MSFC supports the Agency goals of lifting from Earth, living and working in space, and understanding our world and beyond by providing propulsion, space transportation, space systems, and scientific research.



MSFC is the NASA-designated center for the development of space launch systems. The center is particularly well-known for propulsion system development

PROPULSION SYSTEMS DEPARTMENT



- ER01 Propulsion Systems Department
  - ER02 Resources Management Office
  - ER20 Propulsion Systems Design & Integ. Division
    - ER21 Engine Systems Branch
    - ER22 Main Propulsion System Branch
    - ER23 Spacecraft & Auxilliary Propulsion Systems Branch
    - ER24 Propulsion Research and Technology Branch
  - ER30 Propulsion Component Design & Devel. Division
    - ER31 Turbomachinery Design & Development Branch
    - ER32 Combustion Devices Design & Devel. Branch
    - ER33 Valve, Actuators, & Ducts Design and Devel Branch
    - ER34 Propulsion Detailed Design Branch
    - ER35 TVC Systems Integration & Components Branch
  - ER40 Structures, Thermal, and Fluids Division
    - ER41 Structural & Dynamics Analysis Branch
    - ER42 Fluid Dynamics Branch
    - ER43 Thermal Analysis Branch
  - ER50 Solid Propulsion Systems Division
    - ER51 Solid Launch Systems & Analysis Branch
    - ER52 Solid Separation & Manufacturing Systems Branch

FLUID DYNAMIC BRANCH STRUCTURE

Fluid Dynamics Branch

Branch Chief – Lisa Griffin
Assistant Branch Chief – Tom Nesman
Technical Assistant – Denise Chaffee
Computer System Administrator – Dennis Goode

Propellant Delivery CFD
Team Leader: Jeff West

Experimental and Unsteady Flows
Team Leader: Tom Zoladz

Acoustics and Stability
Team Leader: Vacant

Combustion Driven CFD
Team Leader: Kevin Tucker

ER42 is comprised of four teams of approximately forty-five employees

FLUID DYNAMICS BRANCH APPLICATIONS

The Fluid Dynamics Branch (ER42) is responsible for all aspects of the discipline of fluid dynamics applied to propulsion or propulsion-induced loads and environments. This work begins with design trades and parametric studies, and continues through development, risk assessment, anomaly investigation and resolution, and failure investigations. Because of the skills in the branch, ER42 also works non-propulsion items such as for telescopes and payload racks on an as needed basis.

Main Propulsion System

- Tank Dynamics
- Cryofluid Management
- Feedline Flow Dynamics
- Valve Flow and Dynamics

Turbopumps

- Pump Dynamics
- Turbine Dynamics

Liquid Combustion Devices

- Injection Dynamics
- Chamber Acoustics
- Combustion Stability
- Nozzle Dynamics

Solid Rocket Motors

- Motor Dynamics
- Nozzle Dynamics
- Combustion Stability

Coupled Systems

- Feed System Dynamics
- Coupled Pump/MPS Dynamics, e,g,, Pogo
- Thrust Oscillations and its Impact on the Vehicle
- Tank Slosh and its Impact on Vehicle Stability and GN&C

Launch, Separation, and Plume-Induced Environments and Debris

- Liftoff Acoustics
- Separation Acoustics
- Overpressure
- Inflight Plume Generated Noise
- Noise Mitigation
- Hydrogen Entrapment
- Liftoff Debris Transport

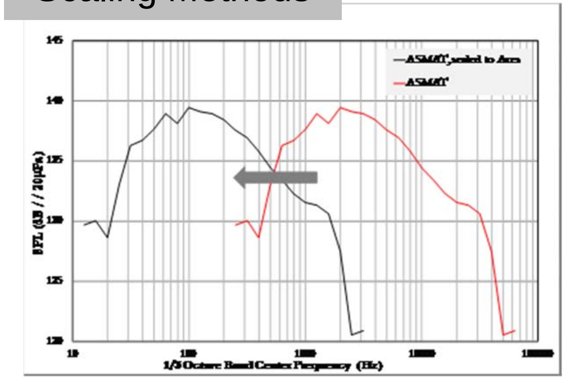
ER42 is a Discipline-Centric branch, not analysis-centric or test-centric. Integration of all discipline methods into one branch enables efficient and accurate support to the projects.

FLUID DYNAMICS ANALYSIS

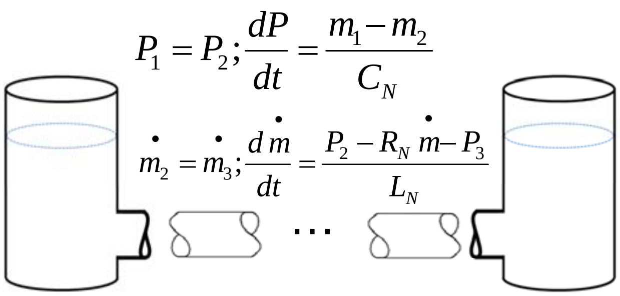
Propulsion Systems Department

Engineered to Excel–Driven to Explore

Scaling Methods



Lump Parameter Modeling



$$P_1 = P_2; \frac{dP}{dt} = \frac{\dot{m}_1 - \dot{m}_2}{C_N}$$

$$\dot{m}_2 = \dot{m}_3; \frac{d\dot{m}}{dt} = \frac{P_2 - R_N \dot{m} - P_3}{L_N}$$

ER42 conducts all levels of fluid dynamics analysis from scaling methods through 3D Unsteady CFD

Chug Stability Code v1.7

Geometry

Chamber Diameter (in)	3.5	Throat Diameter (in)	1.882
Cylinder Length (in)	14.9	Convergent Section Length (in)	2.68

Hydraulics

Oxidizer Pressure Drop (psi)	242	Fuel Pressure Drop (psi)	559
Oxidizer Flow Rate (lbm/sec)	7.5	Fuel Flow Rate (lbm/sec)	8.9
Oxidizer Exponent, a (mo=CdPDpo^a)	0.5	Fuel Exponent, b (mf=CdPDpf^b)	1.0

Combustion Parameters

Oxidizer Time Lag (msec)	0.93	Fuel Time Lag (msec)	0.93
Chamber Pressure (psia)	1246	Chamber Temperature (deg R)	1560
c* (ft/sec)	6717	Slope of c*(MR) (ft/sec-MR)	1980
Gas MW (lbm / lbmol)	3.715		

Feed System Parameters ● Feed Sys. Off ○ Feed Sys. On

Ox. Injector Inertance (lbf*s^2/lbm-in^2)	0	Fuel Injector Inertance (lbf*s^2/lbm-in^2)	0
Ox. Injector Compliance (lbm*in^2/lbf)	inf	Fuel Injector Compliance (lbm*in^2/lbf)	inf

Ranging Information

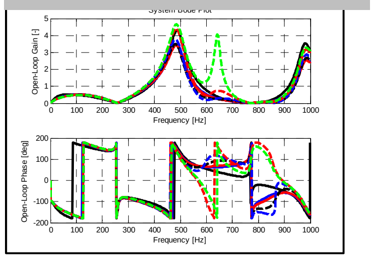
Max Frequency (Hz)	1000	Frequency Resolution (Hz)	0.01
Dpo/Pc Axis Max	0.5	Dpf/Pc Axis Max	0.5

$$x(\omega) = \frac{\bar{X}\sin\left(\omega(\bar{\tau}_{T,o} - \bar{\tau}_{T,f})\right)}{\sin(\omega\bar{\tau}_{T,f}) + \theta_g \omega \cos(\omega\bar{\tau}_{T,f})}$$

$$y(\omega) = \omega$$

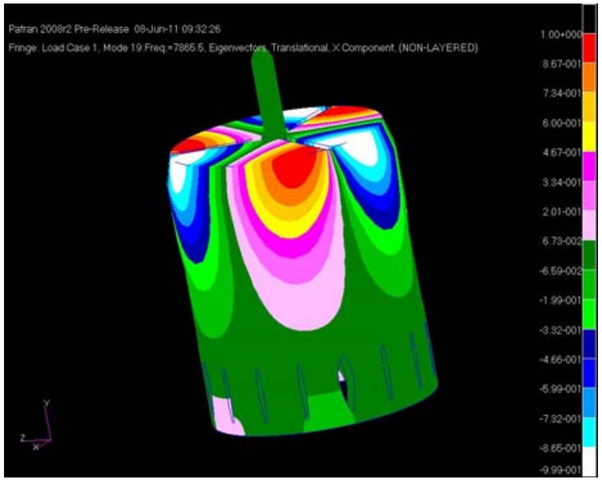
$$z(\omega) = \frac{\bar{F}\sin\left(\omega(\bar{\tau}_{T,f} - \bar{\tau}_{T,o})\right)}{\sin(\omega\bar{\tau}_{T,o}) + \theta_g \omega \cos(\omega\bar{\tau}_{T,o})}$$

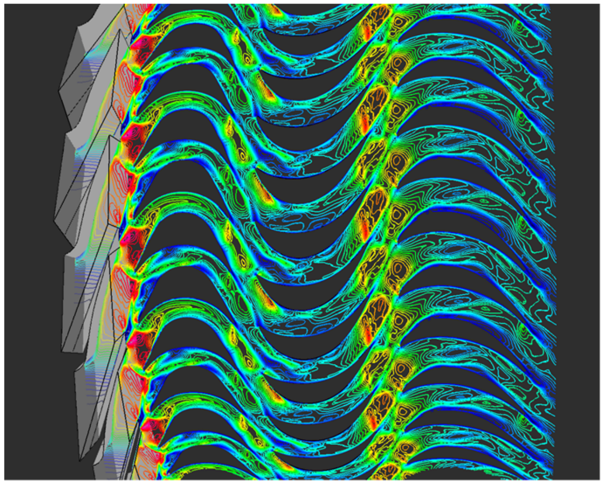
Gain / Phase Plots



System Stability Modeling

Finite Element Modeling



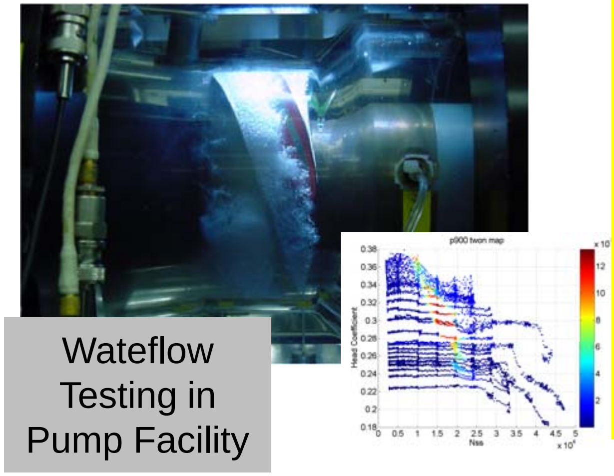


Computational Fluid Dynamics

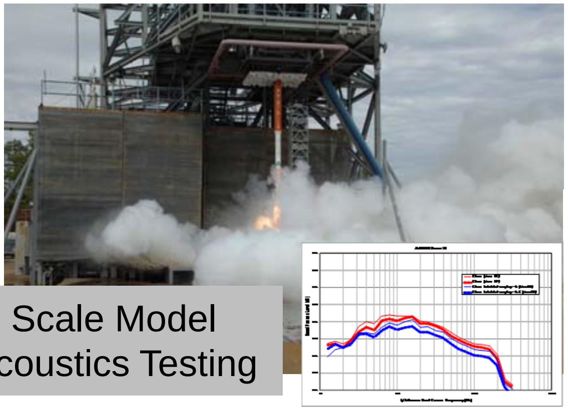
FLUID DYNAMICS TESTING

ER42 conducts and supports testing for hardware and technology development and verification, and analysis validation

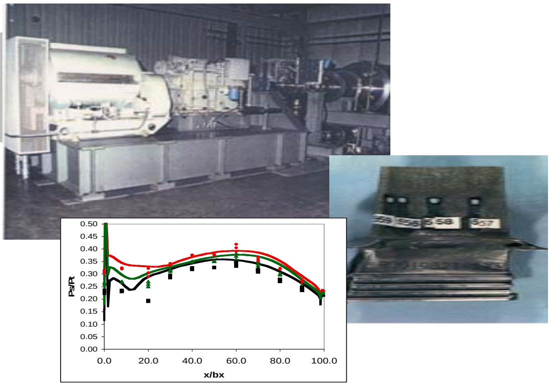
- Primary responsibility for cold flow and scale model acoustics tests
- Secondary responsibility for hot system and component testing



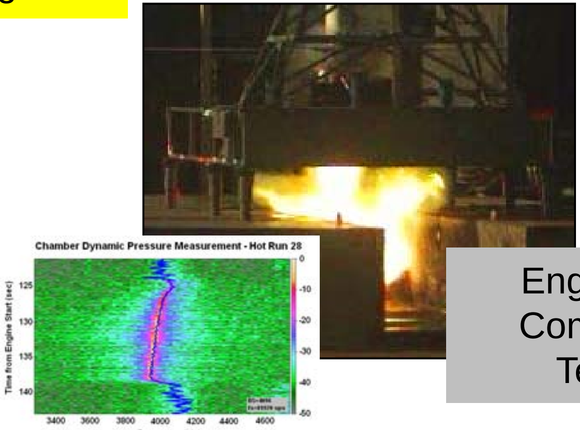
Wateflow Testing in Pump Facility



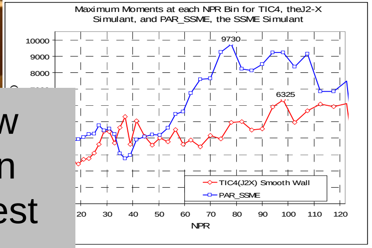
Scale Model Acoustics Testing



Airflow Testing in Turbine Facility



Engine and Component Testing



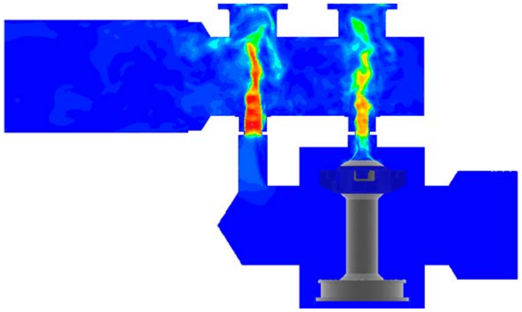
Cold flow testing in Nozzle Test Facility

Solid Rocket Testing

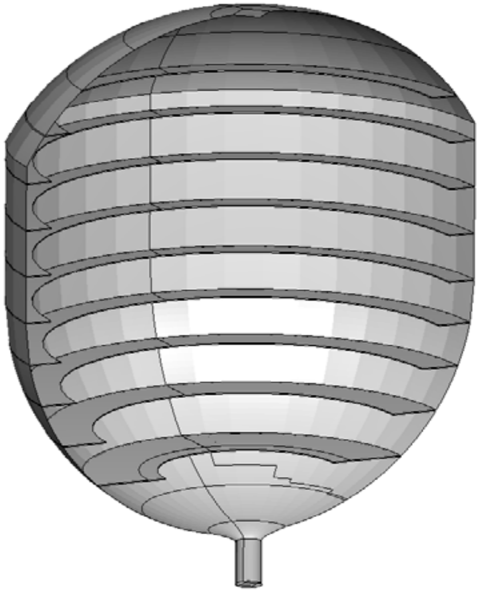
MAIN PROPULSION SYSTEM

- **The Main Propulsion System (MPS) is defined as the propellant delivery system from Tank to Engine Interface.**
  - Tank with all of its internal components
  - Valves
  - Feedlines with all of its internal components
- **ER42's primary analysis tool for MPS is CFD**

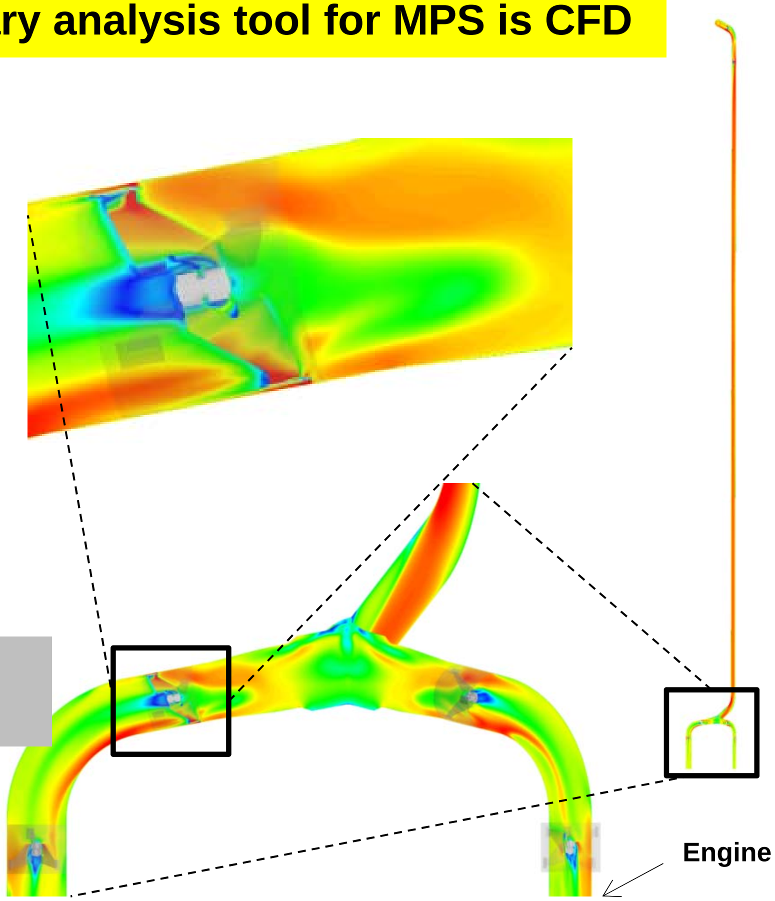
Tank Ullage Pressure Valve



LOX Tank



Articulating Feedline



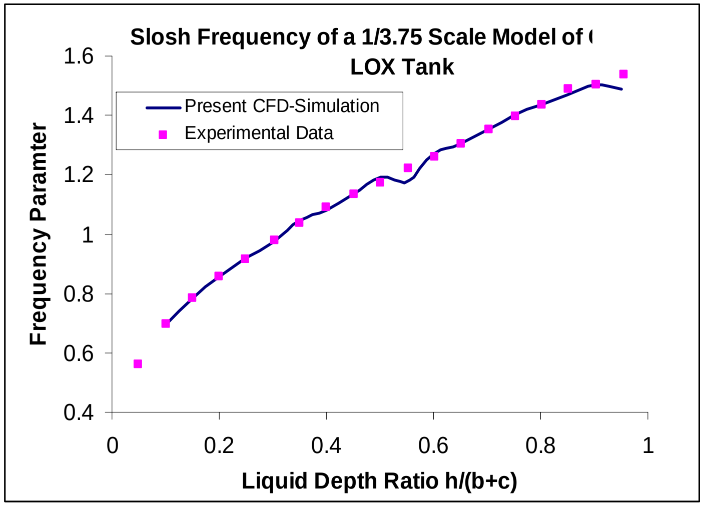
Tank Exit

Engine Interface

LIQUID PROPELLANT TANKS - SLOSH

ER42 performs high fidelity CFD analysis of complex geometry and/or complex accelerated propellant tank sloshing to determine slosh modes and their respective frequencies, amplitudes, and damping characteristics

Earth to Orbit Simulation

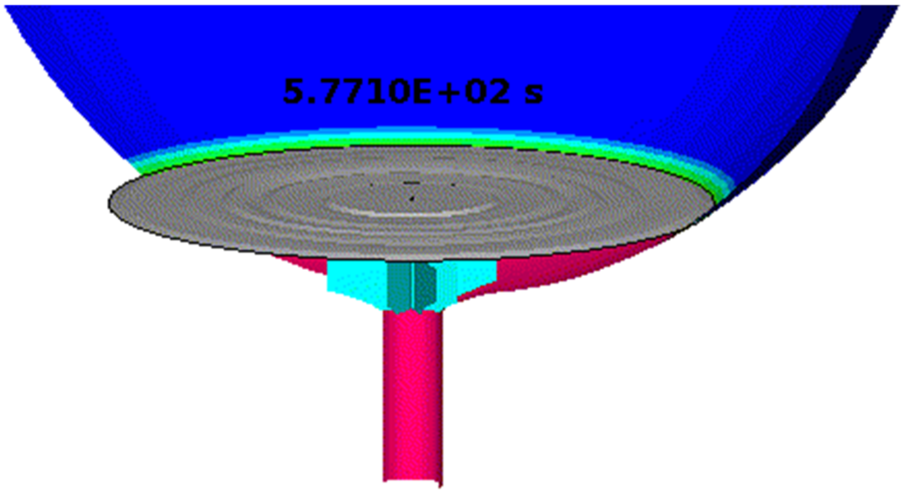


Improvement to Classic Mass-Spring Model

Next challenges with future simulations include implementation of massively parallel gas-liquid interface tracking methods and efficient hybrid implicit/explicit methods to address disparate time-stepping requirements

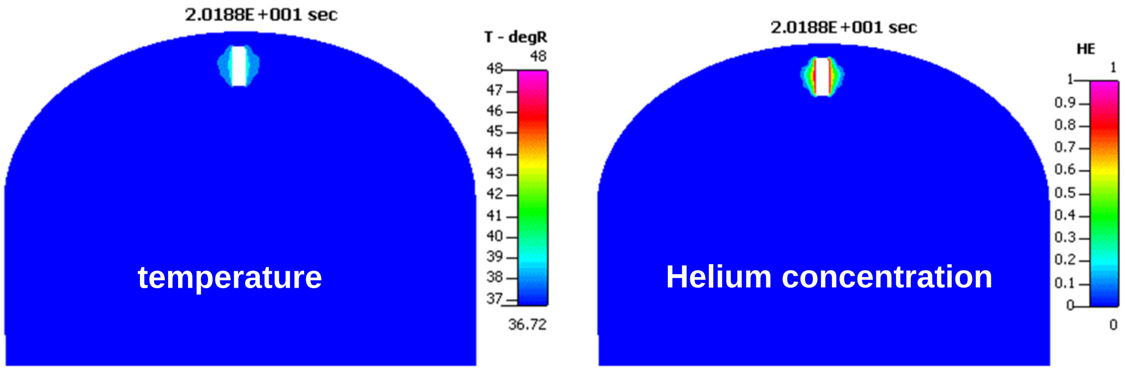
LIQUID PROPELLANT TANKS – PRESSURIZATION AND DRAIN

Assessment of Anti-Vortex Baffle Design



- **Tank Pressurization**
  - Flow through diffuser
  - Interaction of ullage gas with propellant surface (mass transfer, multiphase heat transfer, surface evaporation, chemical species)
- **Tank Drain**
  - Analysis of vortical flow in pipe
  - Assessment of anti-vortex baffle efficiency

- **Near Term Work**
  - Validation of robust method for simulating mass transfer across the gas-liquid interface

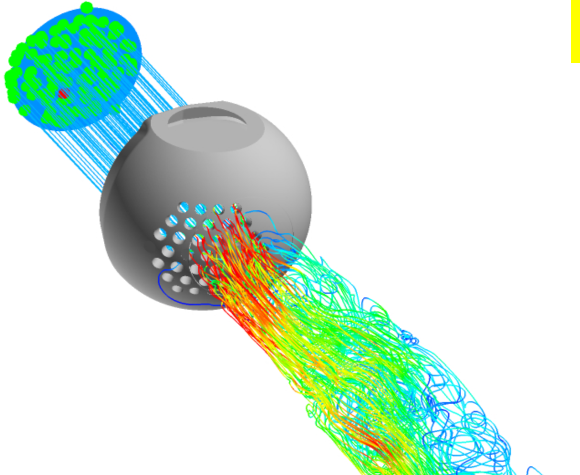


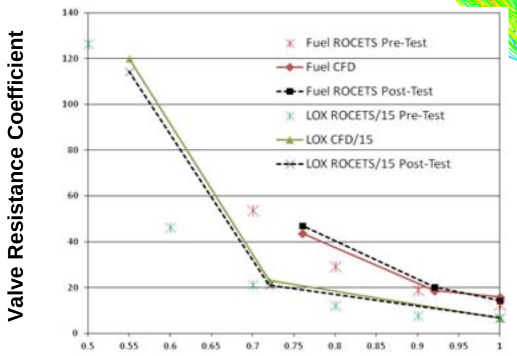
LH2 Tank Pre-press Analysis

VALVES

Partially Open Liquid Fuel and Oxidizer Ball Valves

ER42 conducts high fidelity CFD simulations of valves to predict fluid flow patterns, mean pressure drops, and unsteady fluid environments

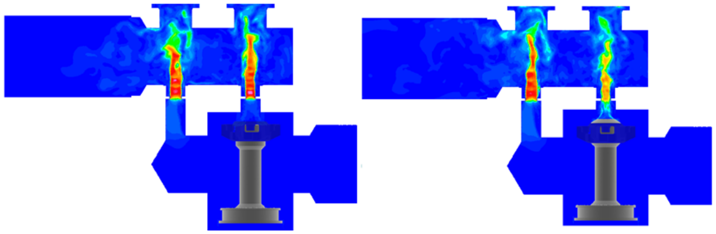




Valve Position [Valve Open Fraction]

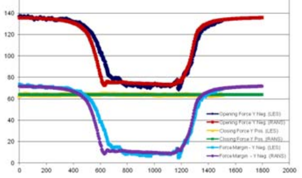
Tranisent 3D Simulation of Poppet Valve

Still Images of Mach Number as Valve Closes

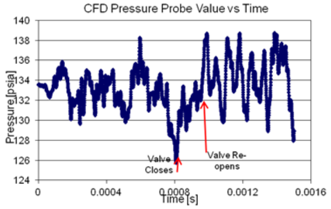


Still Images of Mach Number as Valve Re-opens

Time-accurate Forces on Poppet During Valve Stroke

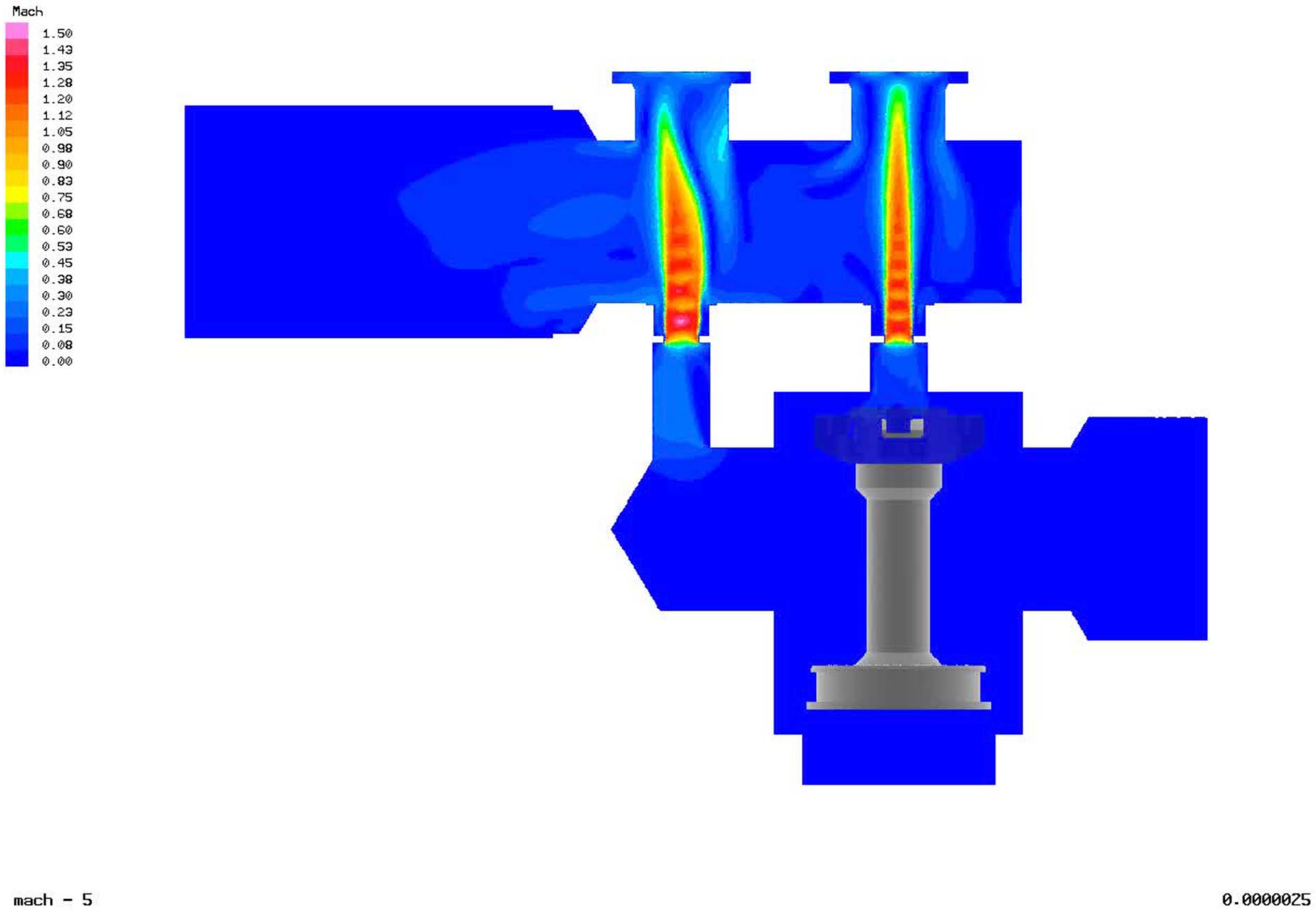


Time-accurate Pressure Oscillations During Valve Stroke



Future work aimed at implementation of valve component force and friction models

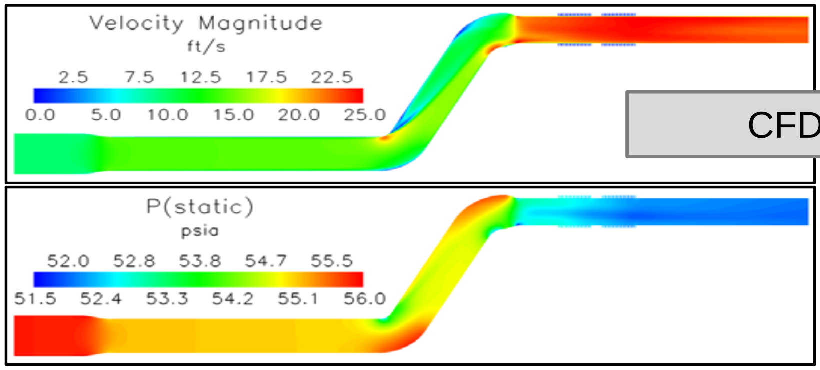
POPPET VALVE ANIMATION



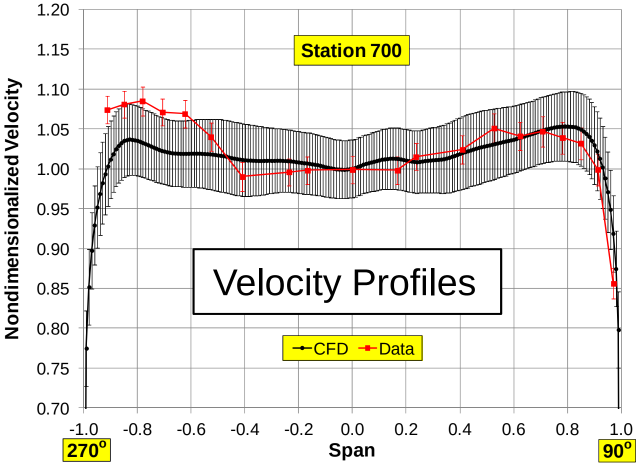
FEEDLINES

ER42 performs high fidelity CFD simulations of liquid propellant feedlines to predict pressure drops through bends, articulating joints, and splits, flow uniformity dues to bends and wakes, and unsteady pressure environments

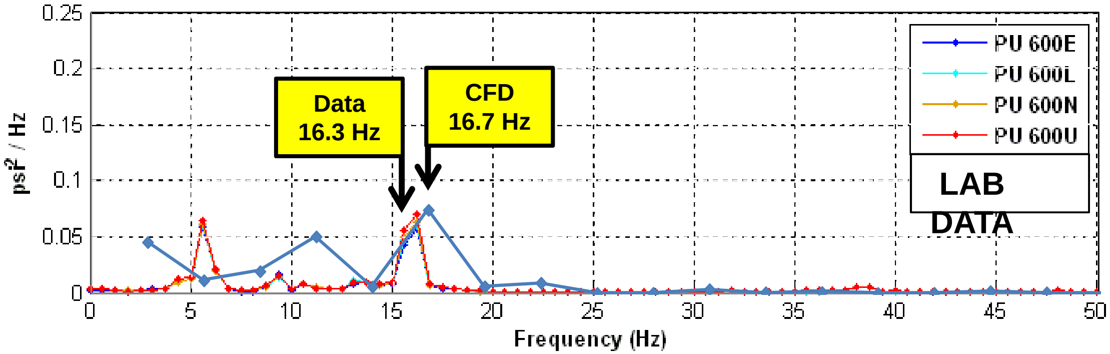
Waterflow Test Article



CFD Predictions



Velocity Profiles



Harmonic Analysis of Pressure Tap Data

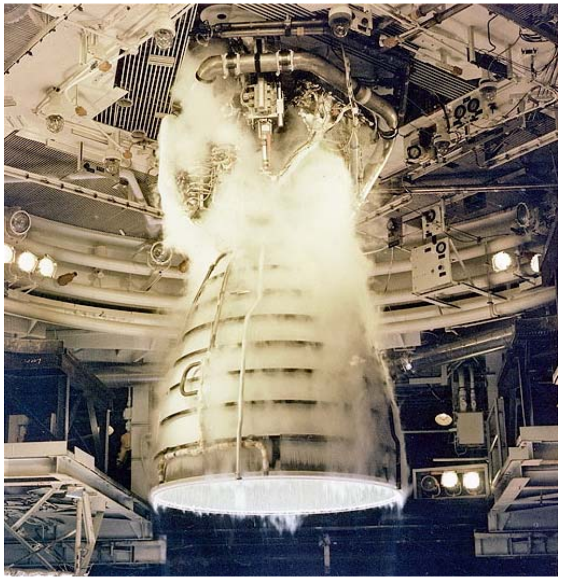
TURBOPUMPS

ER42 supports the design, development, and certification of high-speed turbomachinery

- Quick turnaround CFD design parametrics
- Time-accurate rotor-stator CFD analysis
- Highly instrumented pump waterflow test
- Component and engine test support

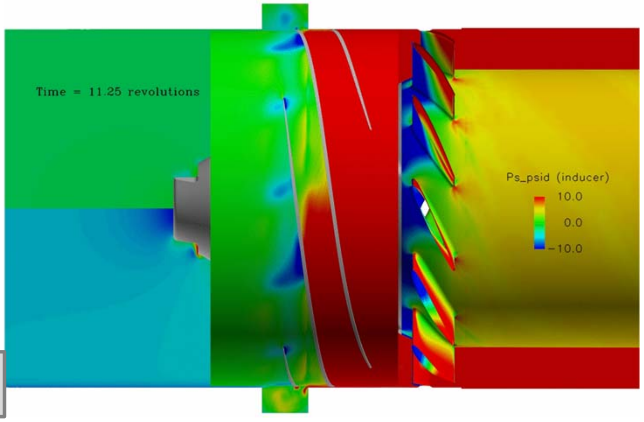
Turbine Airflow Rotating Assembly

Pump Waterflow Test Article



Hotfire Engine Test

Turbine Unsteady CFD Analysis

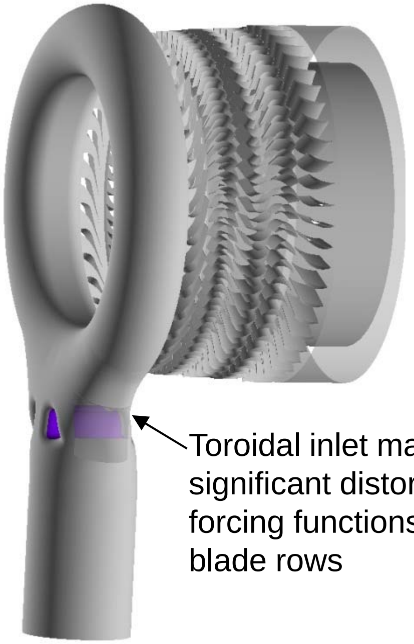


Pump Unsteady CFD Analysis

TURBOPUMPS – TURBINE ANALYSIS

Spatially Resolved First Rotor

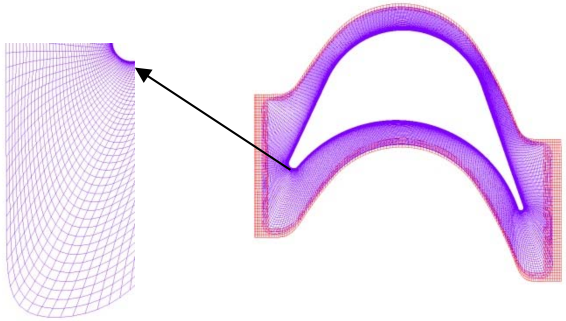
~550 Million Grid Cells



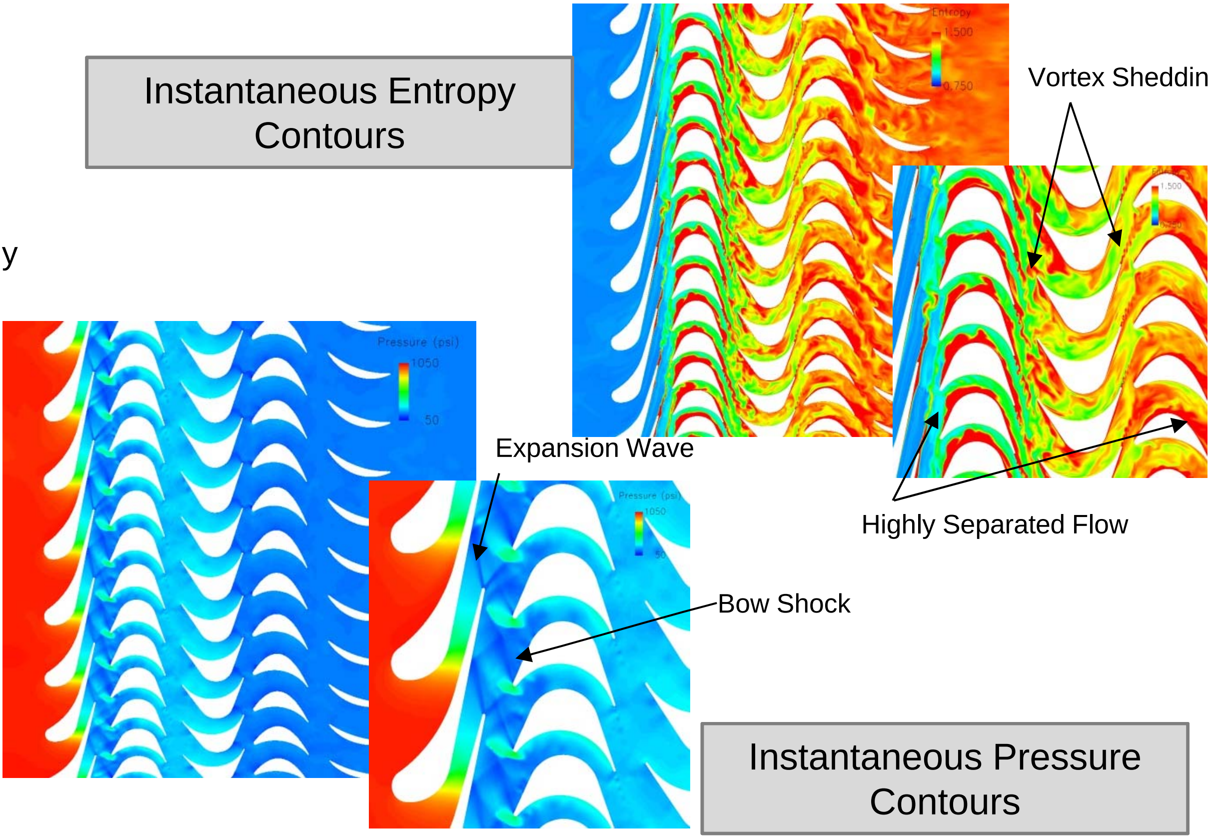
Toroidal inlet manifold causes significant distortions of unsteady forcing functions in downstream blade rows

Unsteady Loads Development

- All flow features which significantly modify fluid forcing functions of interest must be modeled
- Must show spatial and temporal resolution of unsteady forcing functions.
- Full 360 degrees models are necessary for most rocket turbines due to large regions of separated flow. Periodic models corrupt the unsteady forcing functions and are not sufficient.



Fuel Turbine Computational Domain



Instantaneous Entropy Contours

Vortex Shedding

Expansion Wave

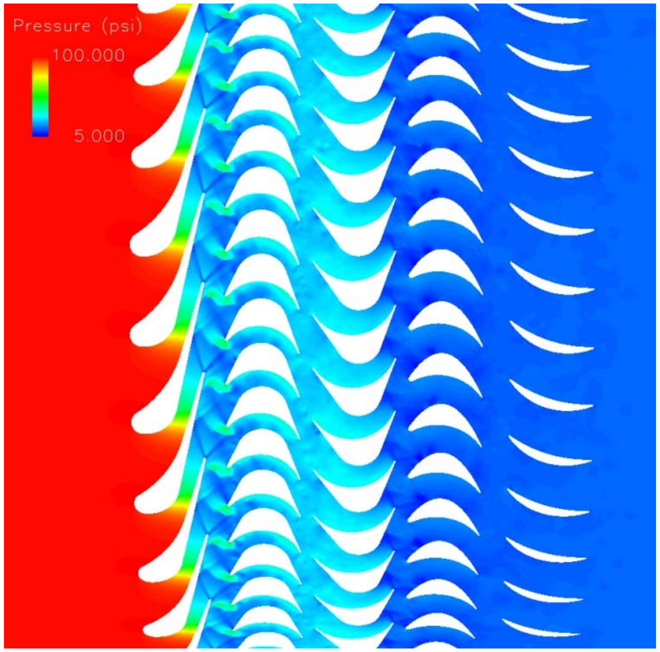
Highly Separated Flow

Bow Shock

Instantaneous Pressure Contours

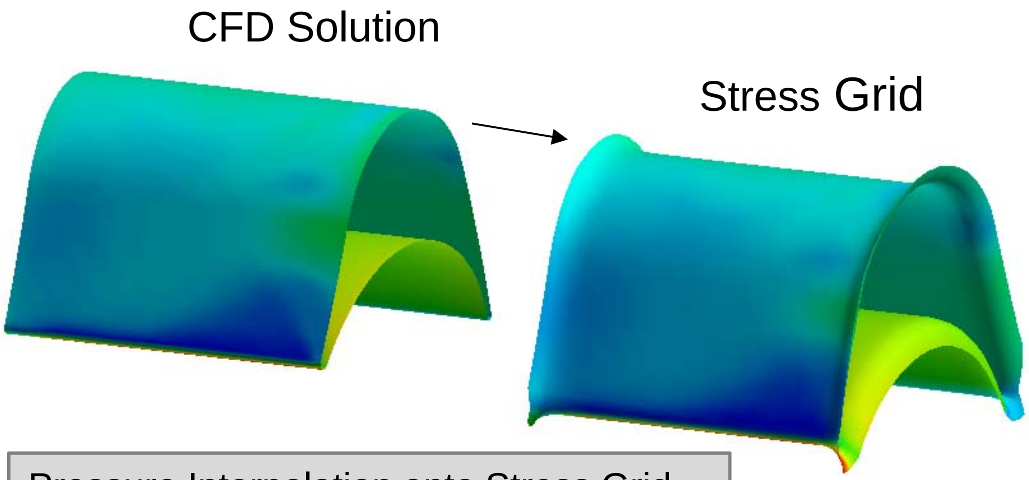
TURBOPUMPS – TURBINE ANALYSIS

Instantaneous Unsteady Pressure
Fuel Turbine



Unsteady Loads Delivery

- Unsteady pressure history saved at all points of all blade surfaces Must show spatial and temporal resolution of unsteady forcing functions
- Unsteady pressure histories from blade surfaces are interpolated onto stress grids for structural analysis. All blades must be used if rotor-rotor or stator-stator effects are to be captured
- Unsteady pressures may be delivered in temporal or frequency domains

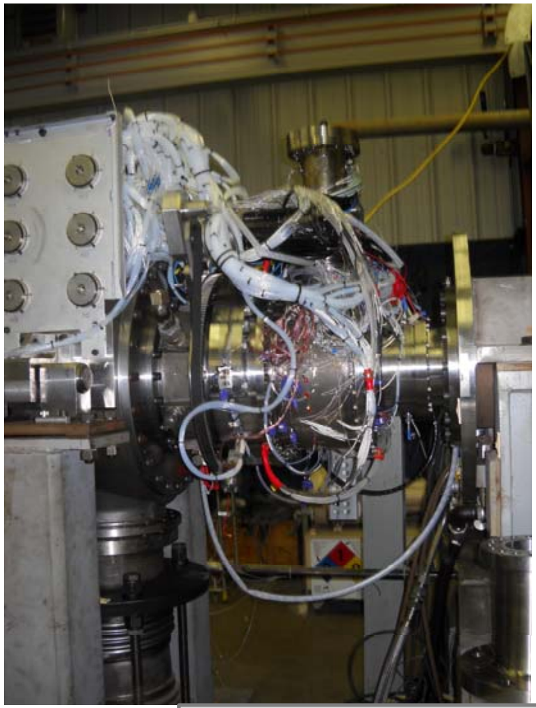


Pressure Interpolation onto Stress Grid

TURBINE AIRFLOW TESTING

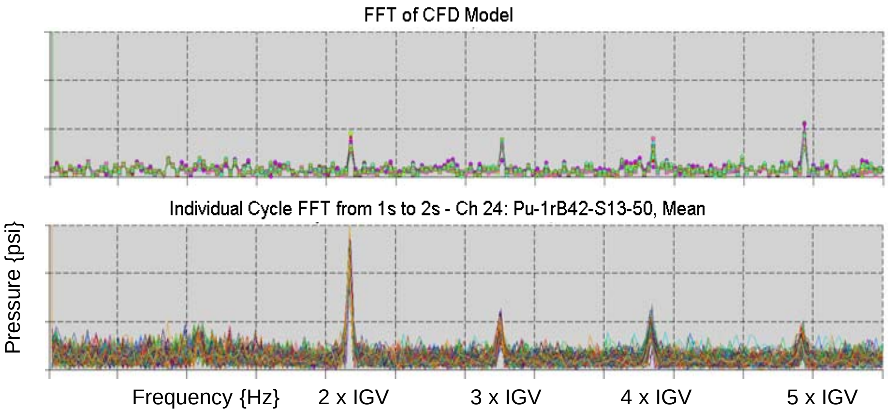
Testing of Highly Instrumented Turbine Models in Scaled Air Conditions

- Steady and unsteady pressure loadings
- Interstage cavity pressures
- Performance mapping over a wide range
- CFD validation



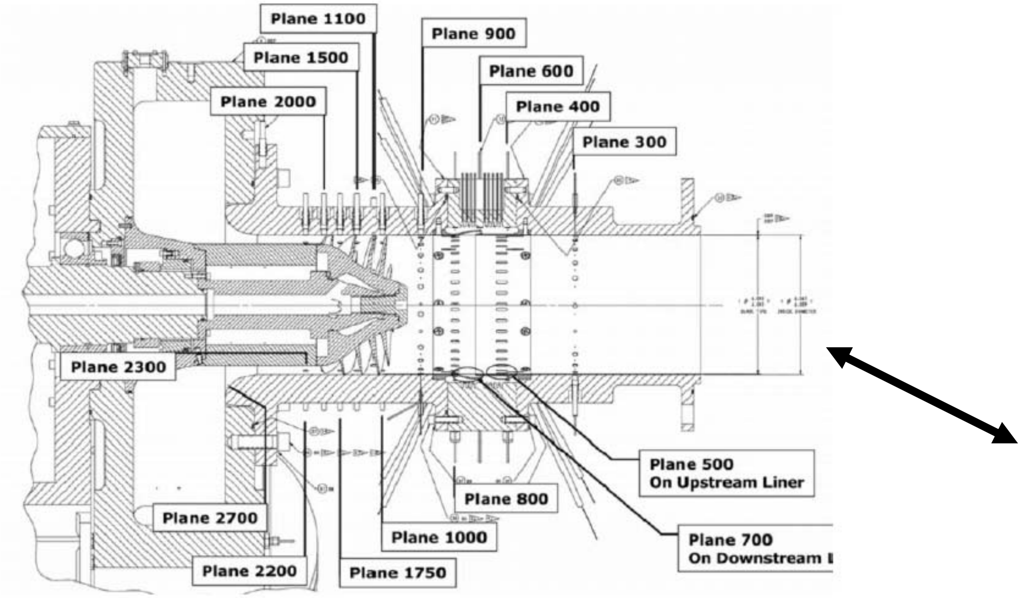
Highly Instrumented Turbine Test Article

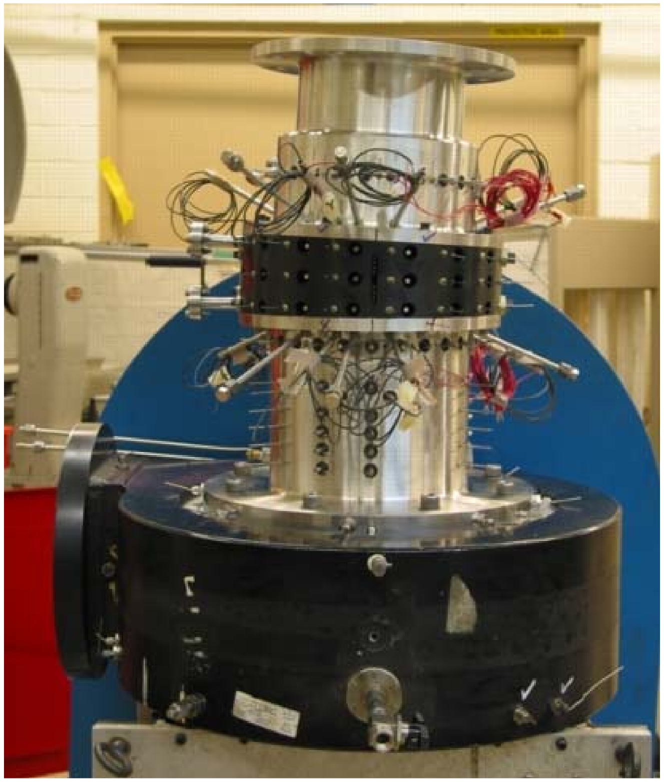
Fourier Transforms of First Stage Blade Suction Side at 13% Axial Chord and 50% Span Location



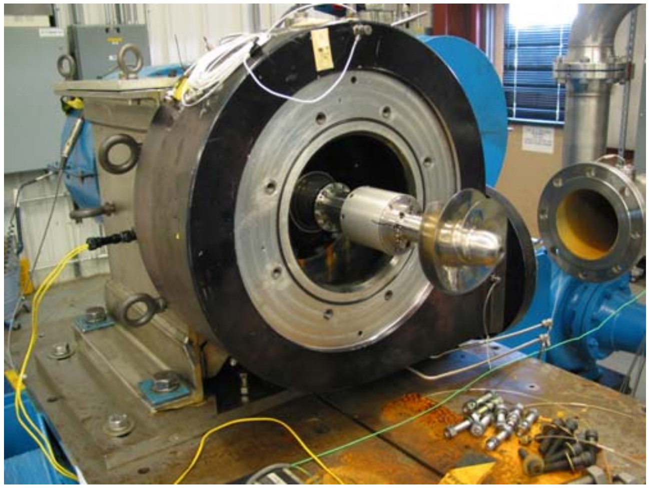
On Shaft Data Acquisition System

PUMP WATERFLOW TESTING





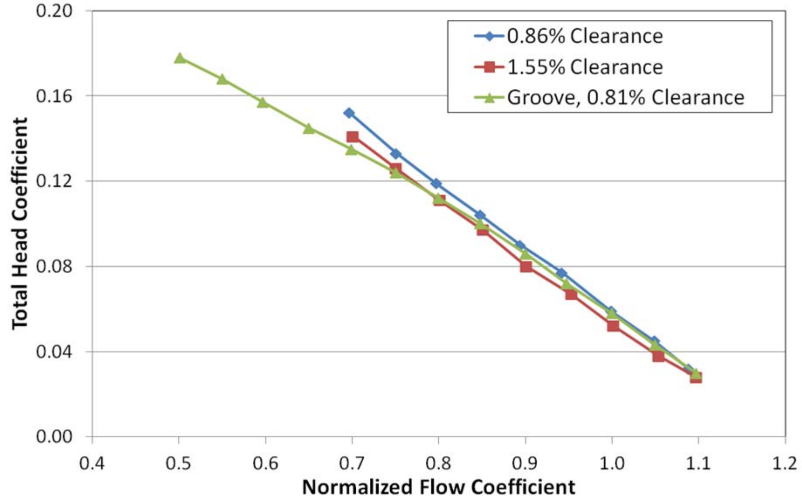
Low pressure pump with upstream main propulsion system element simulation

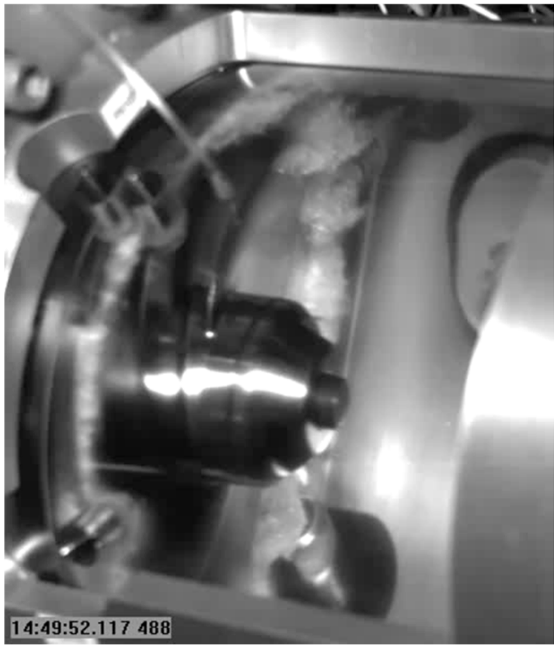


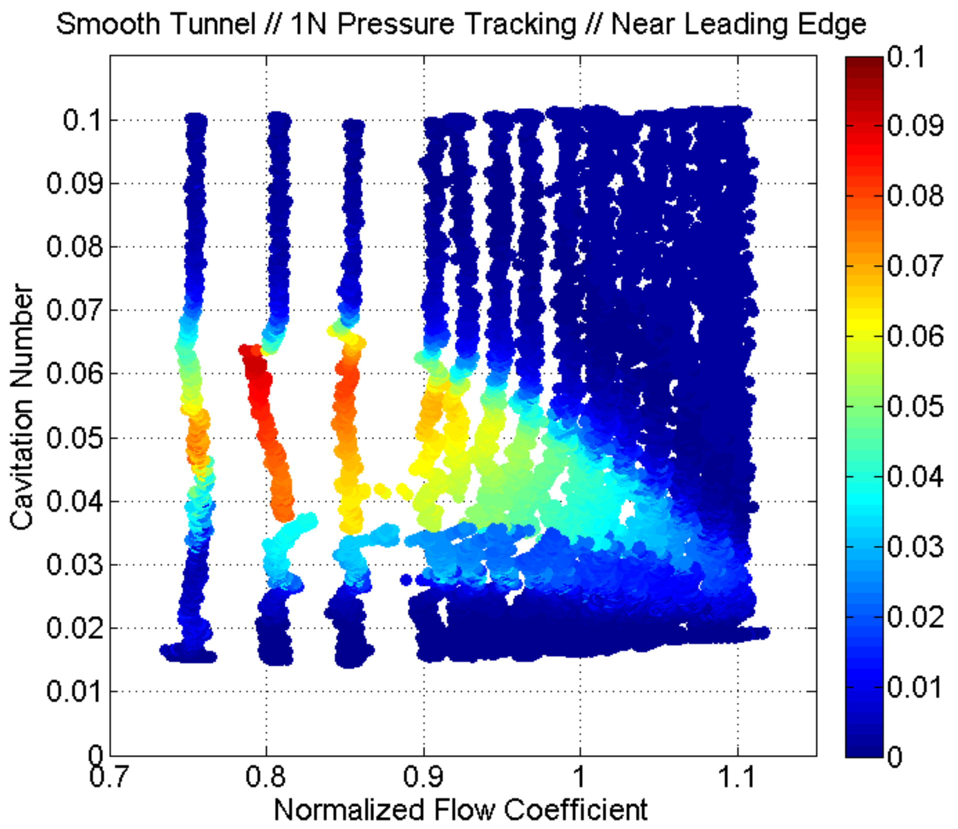
2-blade inducer with on-rotor dynamic force measurement system

Comprehensive steady and unsteady pump performance is evaluated at scaled engine operating conditions. Dense instrumentation suites, velocimetry, and flow visualization are utilized in mapping pump characteristics.

PUMP WATERFLOW TESTING





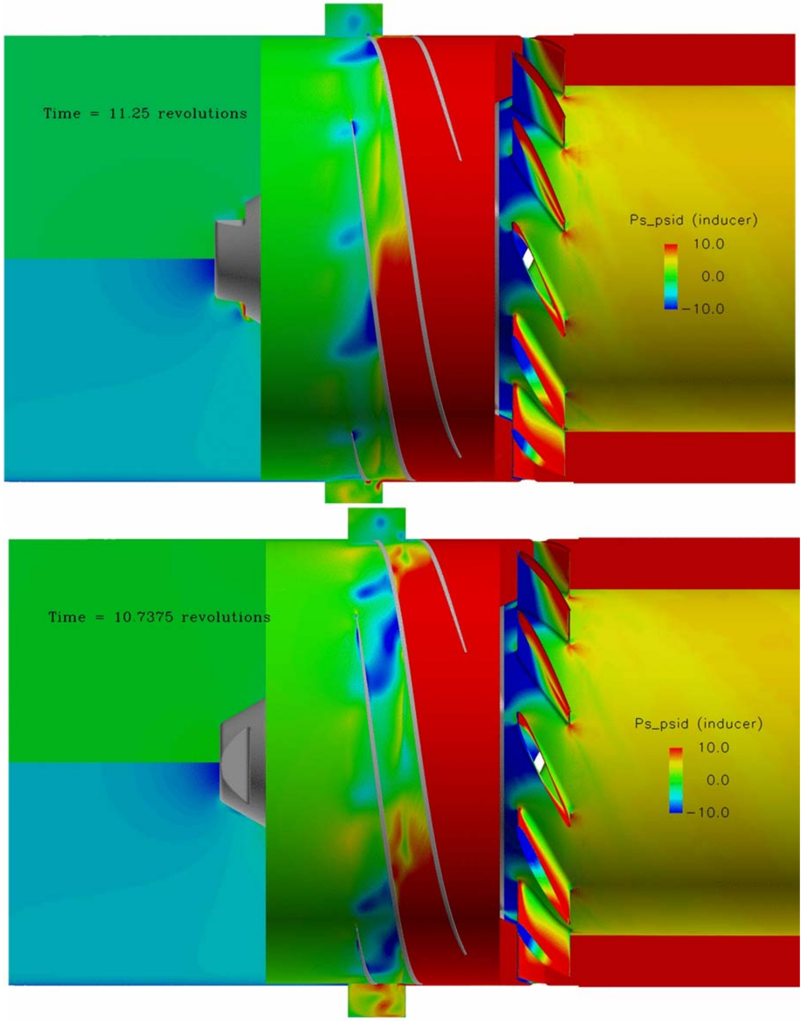


Evaluation of steady pump performance parameters, cavitation oscillation trends, and high-speed flow visualization provides early risk reduction for a turbopump during its preliminary design cycle. Sometimes, comprehensive waterflow is used to identify unsteady loadings and/or performance deficits within certified flight pumps during anomaly investigations.

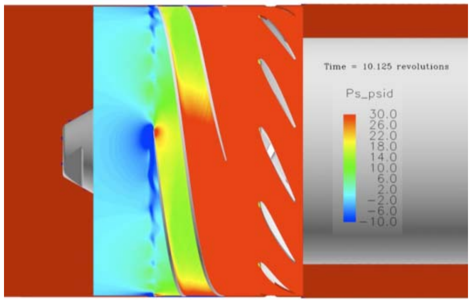
PUMP CFD

Non-cavitating CFD is used to identify critical unsteady flow interactions between inducer blades and cavitation suppression grooves. These interactions are thought to promote higher order cavitation oscillations within the cavitating turbopump. The time-accurate CFD predicts slowly rotating/high cell count progressions very similar to higher order cavitation instabilities measured in waterflow test.

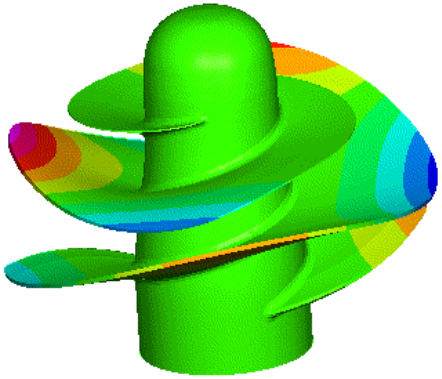
Time accurate CFD provides insight into the complex flow field behind higher order cavitation. Higher order cavitation is a potential forcing function for primary inducer bending modes.



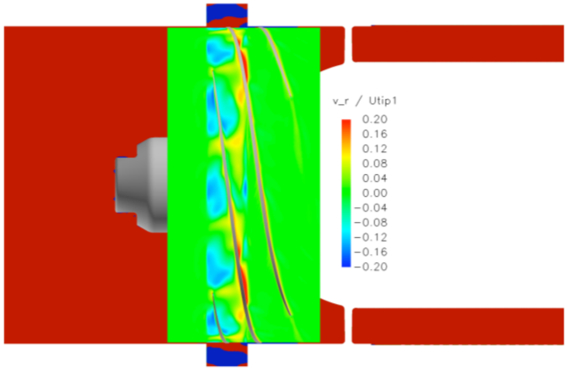
CFD calculations effectively capture tip vortex dynamics for inducers operating with minimal tip clearance (without cavitation suppressor).

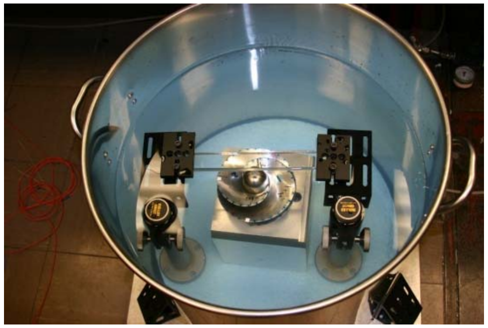


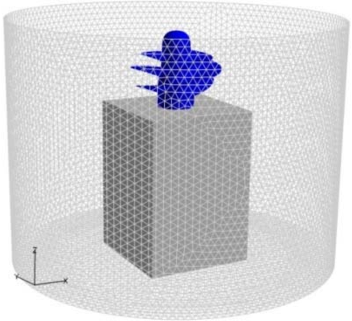
IMMERSED DAMPING



Higher order cavitation oscillation coincides in frequency and modal shape with primary vibration modes of inducers. Can immersed blade damping provide us fatigue margin?



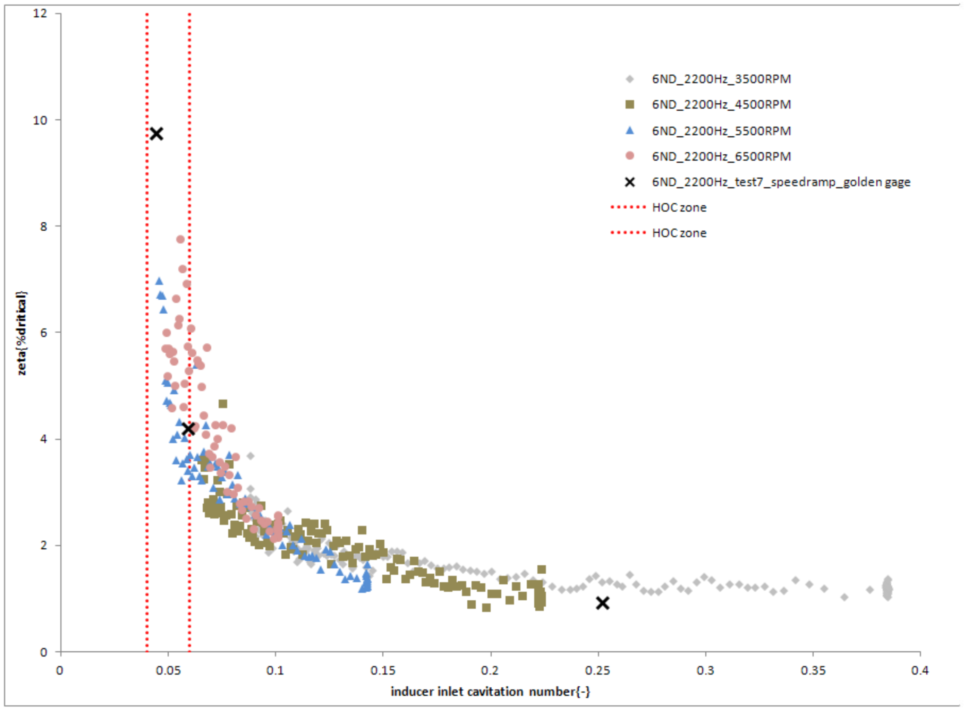




Immersed damping is evaluated under no-flow conditions by experiment and with CFD under modally accurate blade motion. System damping is measured via modal test, and the oscillatory damping forces are extracted from the CFD prediction.

The suppression of lower order cavitation oscillations may bring about higher order flow instabilities which can resonate primary inducer blade modes of vibration. A combination of CFD and experiment is being used to understand the significance of immersed blade damping. Our system damping prediction capability is evolving in a rigorous manner validated by experiment.

IMMERSED DAMPING



Fraction of critical damping increases drastically as inducer blade tip displacement increases during waterflow experiment. Y-axis above is damping, and X-axis is inducer inlet cavitation number. With decreasing cavitation number, random cavitation noise loads increase and deflect blades. Damping was extracted from high frequency strain gages mounted on inducer blade root.

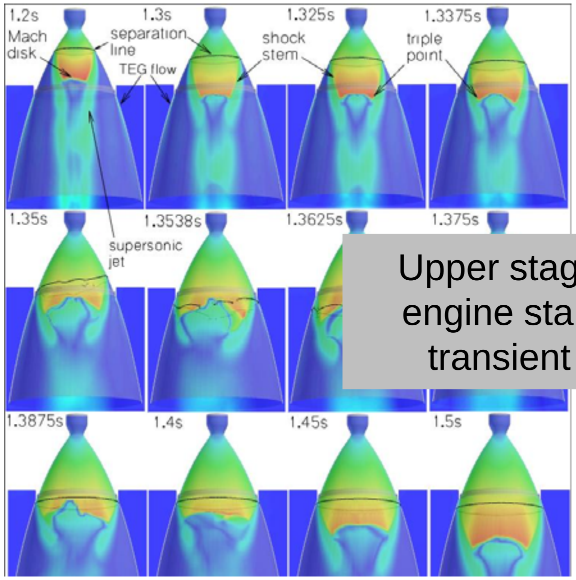
CFD-based simulation of 2-blade inducer displacing water at high frequency. Damping is developed via the formation of flow vortices near the inducer tip.

This combined Fluid Mechanics-Experimental effort showcases our disciplined penetration of complex propulsion system dynamic environments.

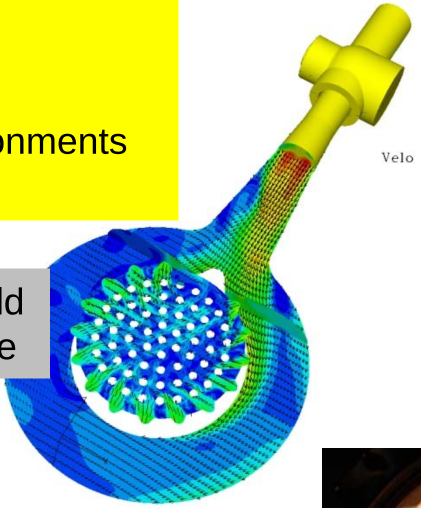
COMBUSTION DEVICES

Scope of branch responsibility in support of liquid rocket engine thrust chamber assembly design & development

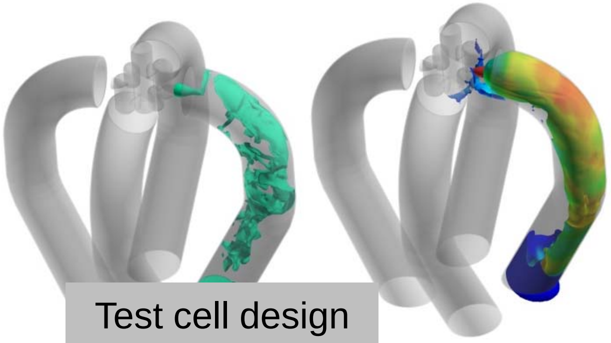
- Large and small engines
- Analysis and testing
  - Performance
  - Pressure, acoustic and thermal environments
  - Combustion stability



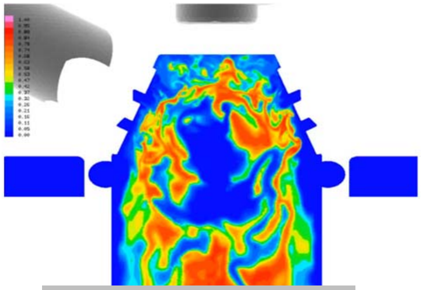
Upper stage engine start transient



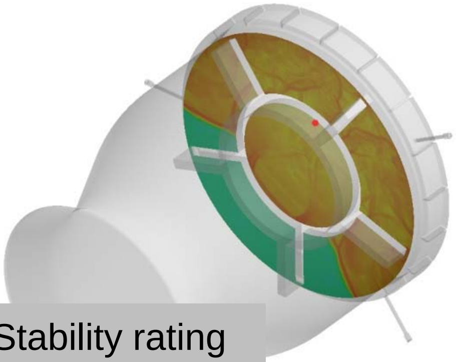
Manifold & Valve



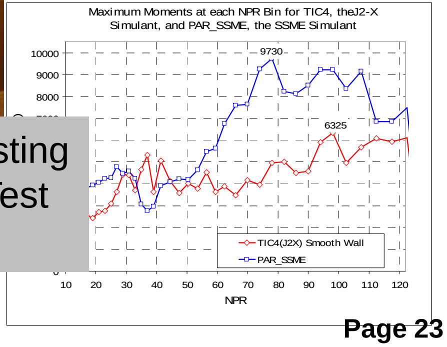
Test cell design & operation



Igniter



Stability rating bomb test simulation



Cold flow testing in Nozzle Test Facility

COMBUSTION STABILITY ASSESSMENT APPROACH & DISCIPLINES LEVERAGED BY ER42

- Branch asked to assess the combustion dynamics / stability of an engine design
  - Chug
  - Acoustic
  - Other oscillation modes (e.g., buzz from upstream supply system)
- Common to all three generic stability types are two main assessment questions:
  - What is the margin associated with the stability type?
    - Requires accepted definition of stable, unstable, and marginal
  - What margin is acceptable for a given engine design?
- Assessment comes from a combination of two approaches:
  - Analytical
    - Linear: system stability approaches; energy based approaches
    - Non-linear: limit cycle waveform evaluation
  - Testing
    - Non-linear: waveform characterization of damp times and amplitudes

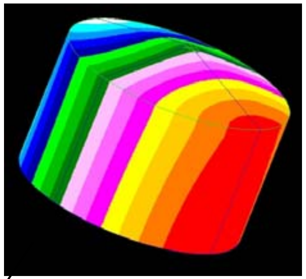
- **Disciplines**
  - Unsteady Fluid Transients and Dynamics
  - Heat Transfer and Thermodynamics
  - Acoustics
  - System Dynamics and Linear Analysis (Stability Theory, State Space, Transfer Matrix)
  - Electronics (Fluid Circuit Analogies, Linear Analysis)
  - Mathematics (DDEs, Model Development, Linear Analysis)
  - Control Engineering (System Identification, Nyquist Plots, Bode Plots)
  - Stability Theory (Nyquist Criterion, et al.)
  - Signal Analysis (Data Characterization and Reduction)
  - Instrumentation and Data Acquisition
  - Combustion Devices and Propulsion
  - Combustion Processes (Spray and Flame Dynamics, Mixing, Atomization, Vaporization, etc.)

- **Tools**
  - PC-Signal, ROCCID, NASTRAN, in-house lumped parameter / state space models, in-house transfer matrix models, in-house impedance models
  - Loci-CHEM, Loci-STREAM, ANSA

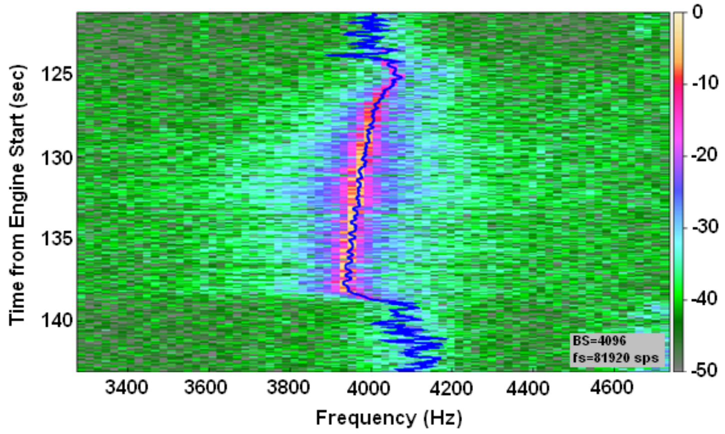
COMBUSTION STABILITY ASSESSMENT: MODE SHAPE IDENTIFICATION

- **Example engine test showed 1T mode during program**
    - Good sensor installation made tangential mode assessments reliable
    - Allowed for mode spatial decomposition

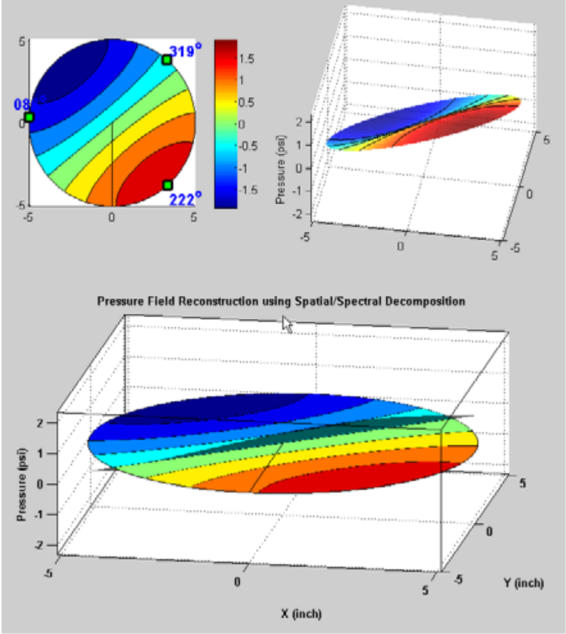
Engine 1T Mode Shape Predicted by FEM



Chamber Dynamic Pressure Measurement - Hot Run 28

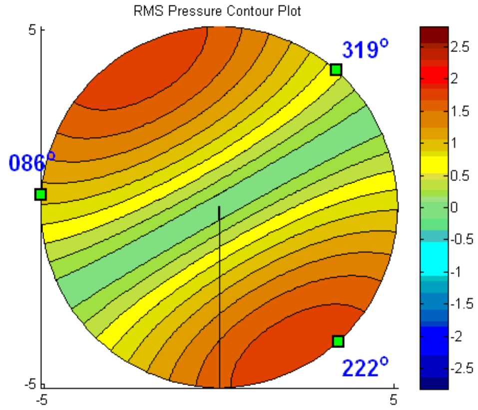


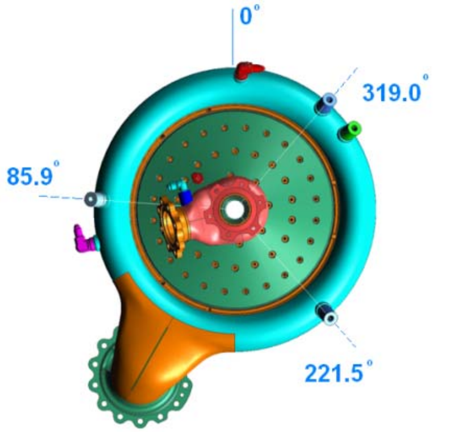
Test 21 at time 141 seconds



$$p(r,\theta,t)=\frac{J_1(\lambda_{1,0}\cdot\frac{r}{R})}{J_1(\lambda_{1,0})}\cdot(\psi_1\cdot\cos(2\pi f_n t+k_1\theta+\phi_1)+\psi_2\cdot\cos(2\pi f_n t+k_2\theta+\phi_2))$$

RMS Pressure Contour ***Kulite Instrument Orientation***





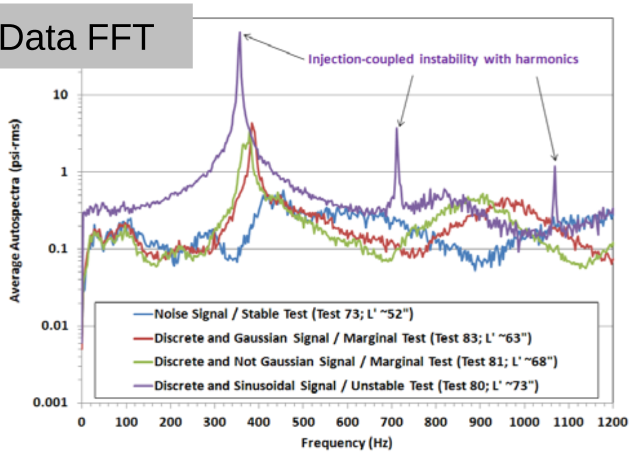
$$\hat{p}(r,\theta)=\frac{\sqrt{2}}{2}\cdot\frac{J_1(\lambda_{1,0}\cdot\frac{r}{R})}{J_1(\lambda_{1,0})}\cdot\sqrt{(2\cdot\psi_1\cdot\psi_2\cdot\cos(2\cdot k\cdot\theta+\phi_1-\phi_2)+\psi_1^{\,2}+\psi_2^{\,2})}$$

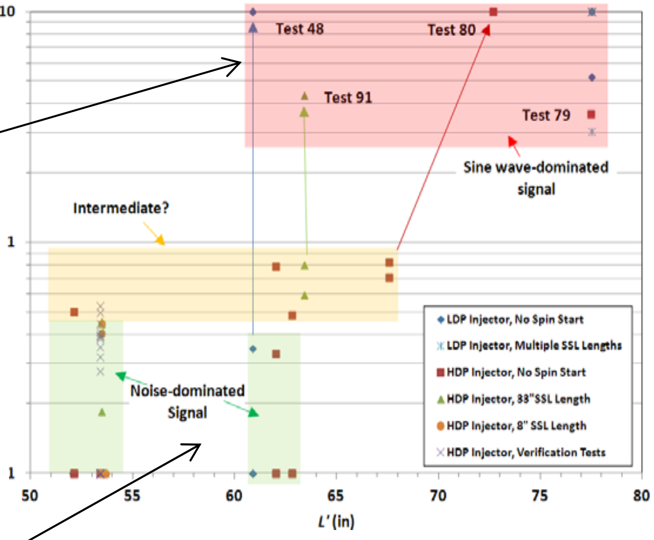
COMBUSTION STABILITY ASSESSMENT: EMPIRICAL STABILITY ASSESSMENTS

Example engine test data - 1L mode instability exhibited during testing program

  - ~300 – 400 Hz stable to unstable signal
- **New methods created to judge spontaneous stability**
  - Offered new way to approach characterizing signal via statistics and frequency variability
  - Gave metrics on how to divide stable vs. unstable
- **New methods created to judge dynamic stability**
  - Assess statistical character of data prior to bomb
  - Track when amplitudes reach back within ‘statistically significant limits’

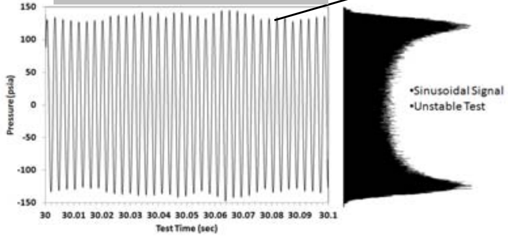
Data FFT



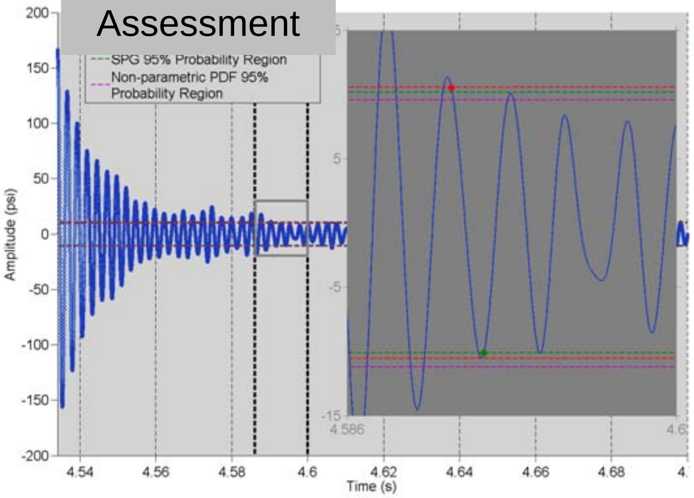


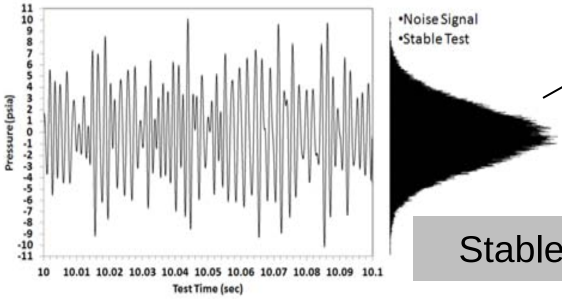
Stability Map

Unstable Test



Damp Time Assessment





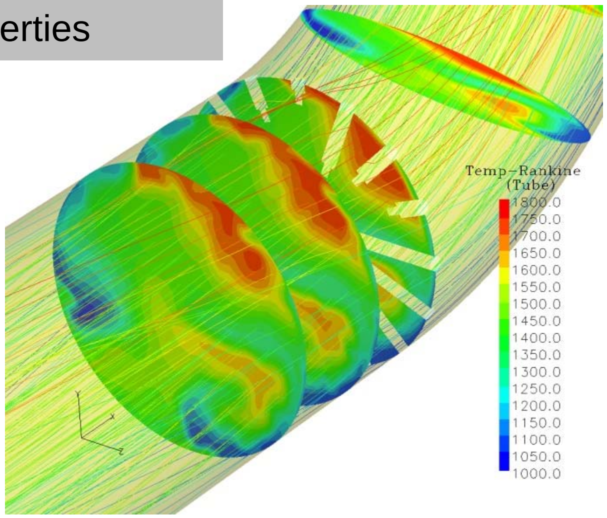
Stable Test

COMBUSTION STABILITY ASSESSMENT: ANALYTICAL ASSESSMENTS

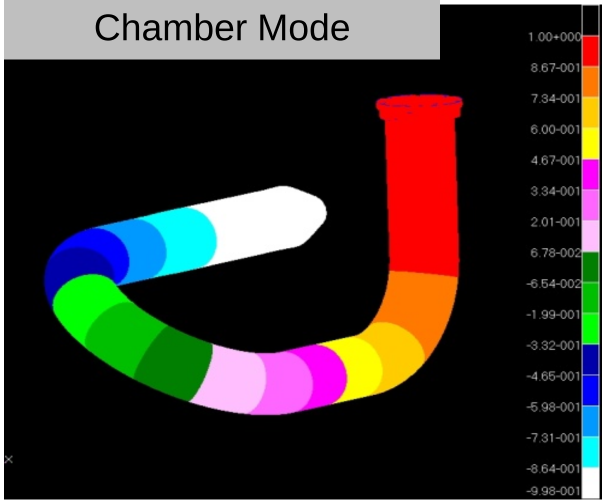
Branch analytical models encompass:

  - Classical linearized stability models
  - Computational Fluid Dynamics (CFD)
  - Finite element modeling (FEM)
- **Linearized models are used for chug and acoustic mode evaluations**
  - State-space and impedance models
- **CFD and FEM used to better characterize complex flowfields and geometries**
  - Accounts for distribution of fluid properties
  - Coupled acoustic modes better evaluated using CAD geometries and CFD inputs

CFD of Chamber Fluid Properties



Acoustic FEM of 1L Chamber Mode



Geometry

Chamber Diameter (in)	3.5	Throat Diameter (in)	1.882
Cylinder Length (in)	14.9	Convergent Section Length (in)	2.68

Hydraulics

Oxidizer Pressure Drop (psi)	242	Fuel Pressure Drop (psi)	559
Oxidizer Flow Rate (lbm/sec)	7.5	Fuel Flow Rate (lbm/sec)	6.9
Oxidizer Exponent, a (mo=CdPDpo^a)	0.5	Fuel Exponent, b (mf=CdPDpf^b)	1.0

Combustion Parameters

Oxidizer Time Lag (msec)	0.93	Fuel Time Lag (msec)	0.93
Chamber Pressure (psia)	1246	Chamber Temperature (deg R)	1560
c* (ft/sec)	6717	Slope of c*(MR) (ft/sec-MR)	1980
Gas MW (lbm / lbmol)	3.715		

Feed System Parameters ● Feed Sys. Off ○ Feed Sys. On

Ox. Injector Inertance (lbf*s^2/lbm-in^2)	0	Fuel Injector Inertance (lbf*s^2/lbm-in^2)	0
Ox. Injector Compliance (lbm*in^2/lbf)	[illegible]	Fuel Injector Compliance (lbm*in^2/lbf)	[illegible]

Ranging Information

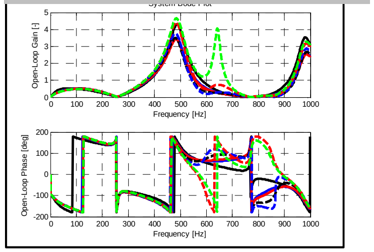
Max Frequency (Hz)	1000	Frequency Resolution (Hz)	0.01
Dpo/Pc Axis Max	0.5	Dpf/Pc Axis Max	0.5

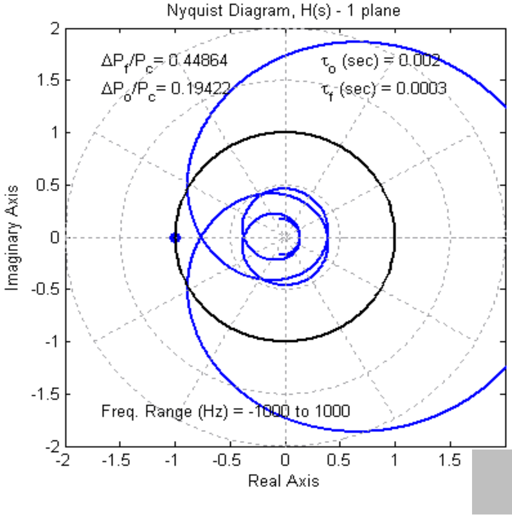
$$x(\omega) = \frac{\bar{X}\sin\left(\omega(\bar{\tau}_{T,o} - \bar{\tau}_{T,f})\right)}{\sin(\omega\bar{\tau}_{T,f}) + \theta_g \omega \cos(\omega\bar{\tau}_{T,f})}$$

$$y(\omega) = \omega$$

$$z(\omega) = \frac{\bar{F}\sin\left(\omega(\bar{\tau}_{T,f} - \bar{\tau}_{T,o})\right)}{\sin(\omega\bar{\tau}_{T,o}) + \theta_g \omega \cos(\omega\bar{\tau}_{T,o})}$$

Gain / Phase Plots





Nyquist Stability Plot

COMBUSTION STABILITY ASSESSMENT: IMPROVING THE STATE-OF-THE-PRACTICE

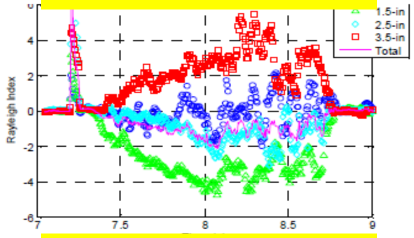
Objective of Improvements

- Advance the predictive capability of current, state-of-the-practice tools and methodologies used in combustion stability assessments
- Facilitate
  - Confident identification & characterization of combustion instabilities
  - Successful & efficient mitigation during propulsion system development
- Minimize development costs &improve hardware robustness

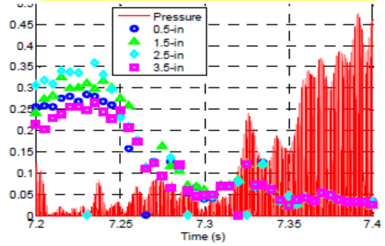
Approach to Improvements

- Improve state-of-the-practice stability assessment capability by use of higher-fidelity, physics-based information either integrated into the engineering tools or used separately in the assessment process
- Extract physics-based models/information from focused state-of-the-art CFD simulations
- Validate new capability by exercising the improved capabilities on relevant experiments

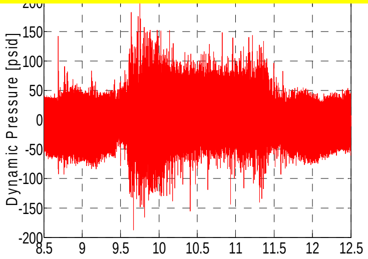
Rayleigh Index*



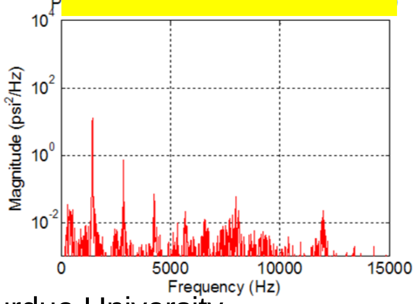
Oscillation Dec.



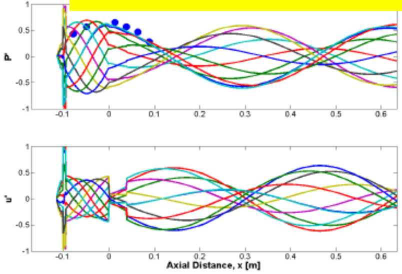
Fluctuating Pressure



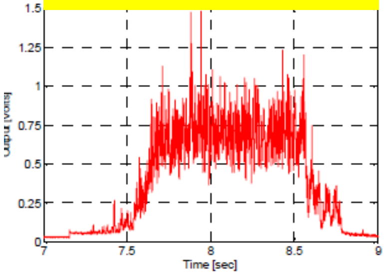
PSD



Mode Shape



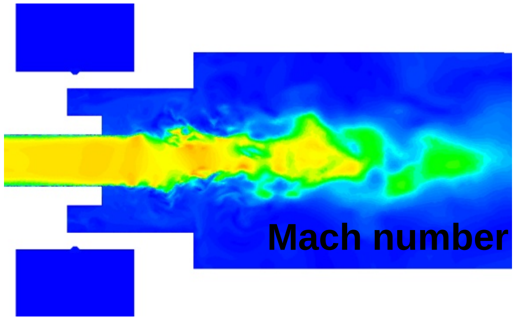
Heat Release

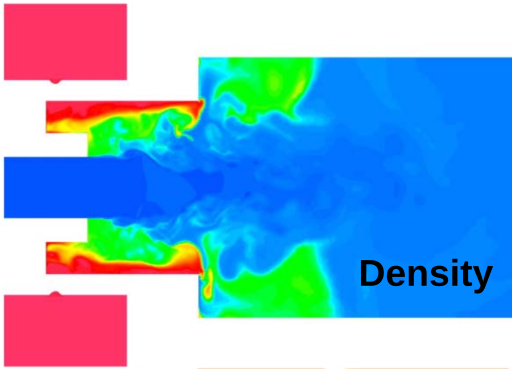


*Courtesy of W. Anderson/Purdue University

COMBUSTION STABILITY ASSESSMENT: IMPROVING THE STATE-OF-THE-ART

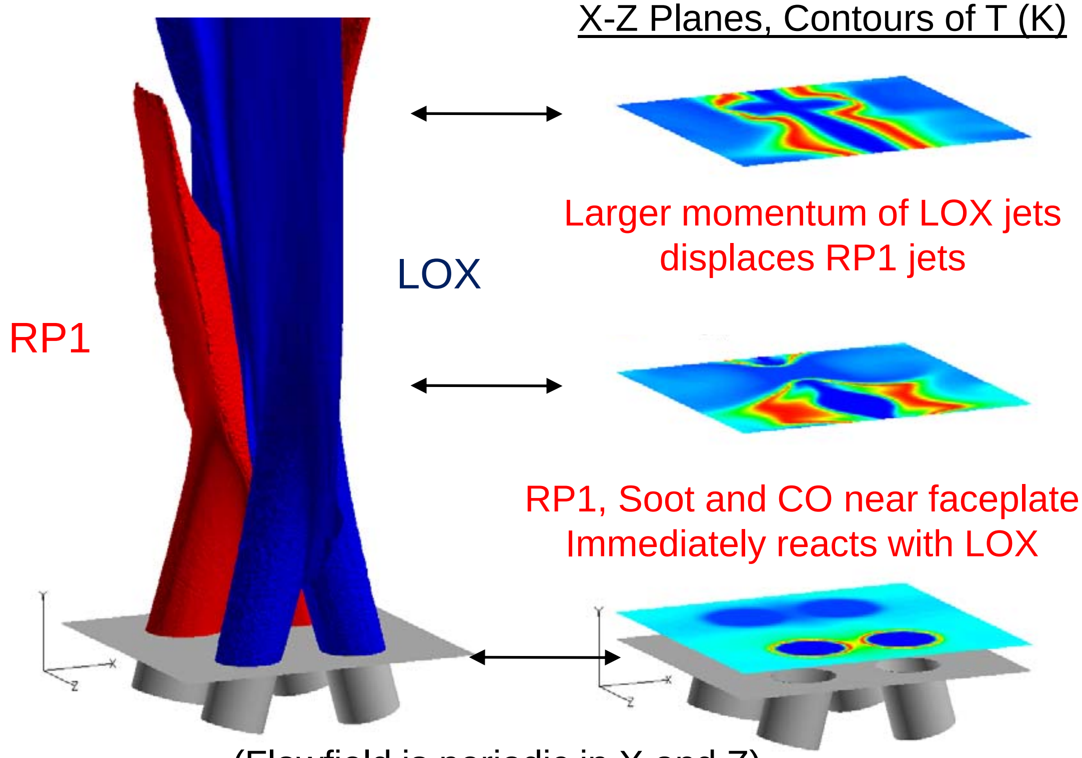
Instantaneous 2-D snapshots from a 3-D non-reacting simulation of a gas-centered swirl coaxial element





Pressure in fuel manifold

RANS simulation of a reacting like-on-like impinging doublet element



X-Z Planes, Contours of T (K)

Larger momentum of LOX jets displaces RP1 jets

RP1, Soot and CO near faceplate Immediately reacts with LOX

(Flowfield is periodic in X and Z)

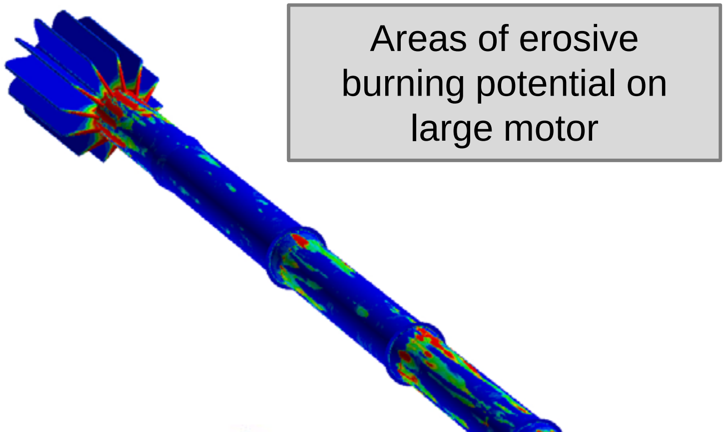
Ongoing improvements for injector CFD

- Flamelet formulation for efficient simulation of reacting flows
- VOF & atomization for 2-phase flow
- Low dissipation schemes better resolving turbulence & acoustics

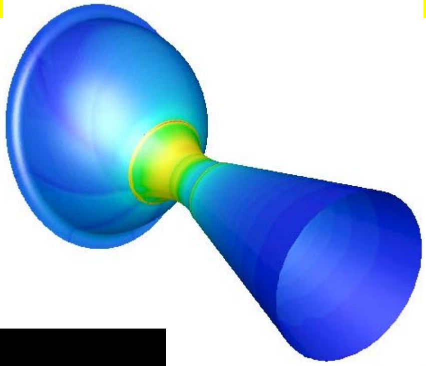
SOLID ROCKET MOTOR OVERVIEW

Scope of branch responsibility in support of solid rocket motor design & development

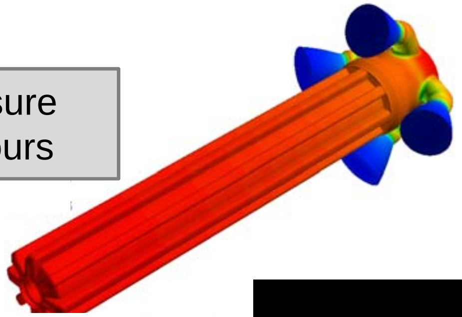
- Large booster-class motors
- Small motors-ullage settling, booster separation & launch abort
  - Performance
  - Environments-pressure, acoustic & thermal
  - Stability



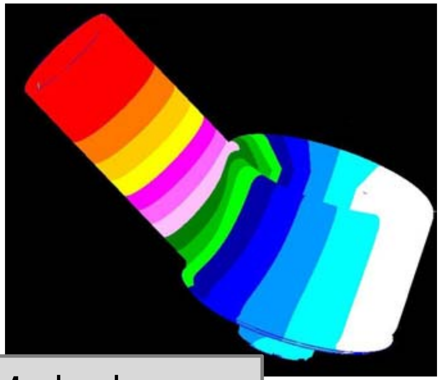
Areas of erosive burning potential on large motor



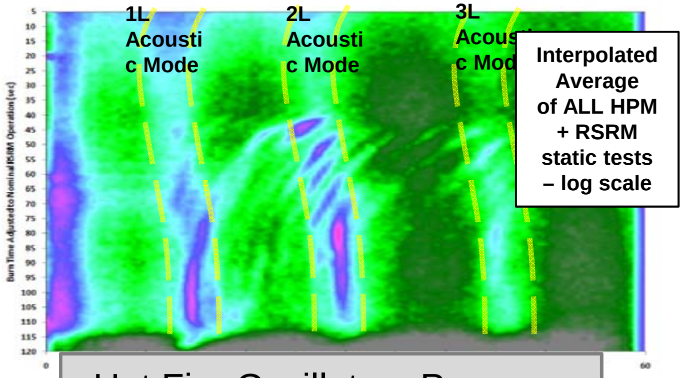
Aft dome heat transfer coefficients



Pressure contours

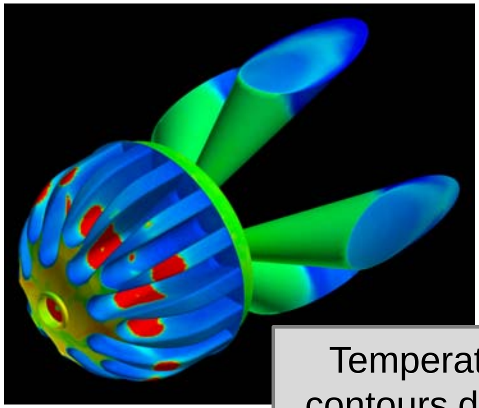


Mode shapes from finite element analysis



Interpolated Average of ALL HPM + RSRM static tests – log scale

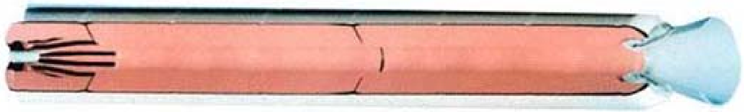
Hot Fire Oscillatory Pressure Characteristics

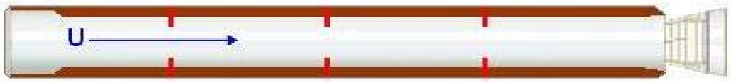


Temperature contours during ignition

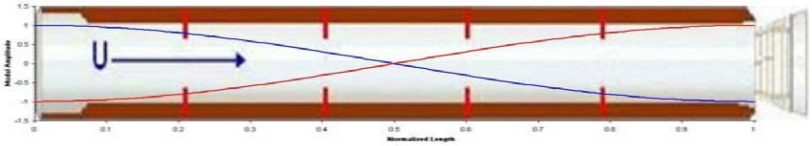
SOLID ROCKET MOTOR THRUST OSCILLATIONS: WHY ARE THEY A CONCERN?

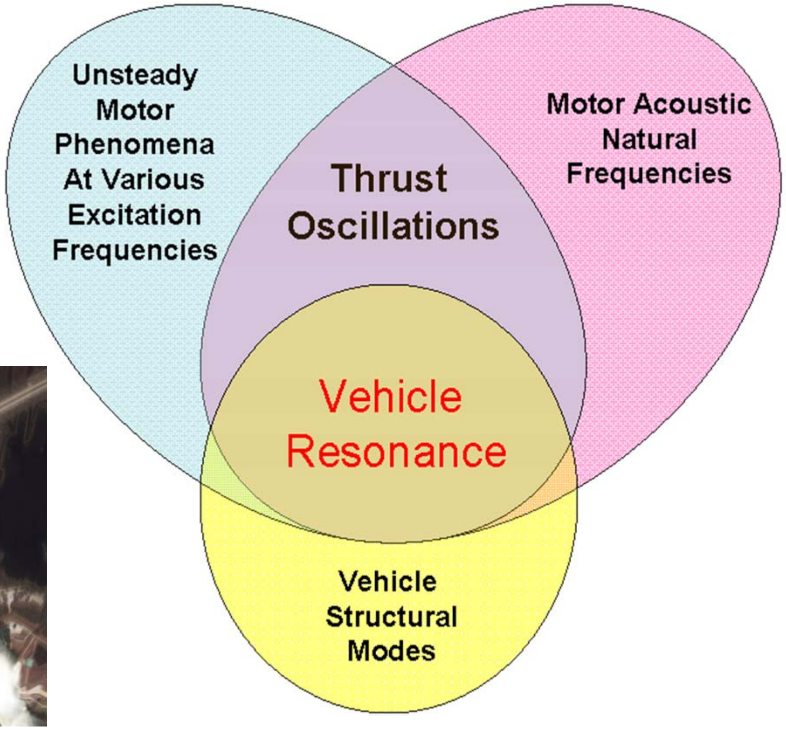
- **SRM thrust oscillations during flight can deliver forced accelerations to vehicle structure and acoustic mode frequencies**
  - **Space Shuttle System**
  - **Arianne 5**



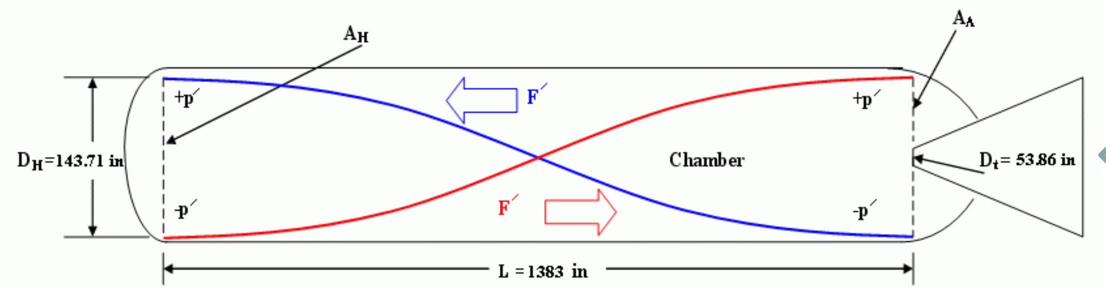


- **If these forced accelerations match appropriate vehicle structural modes, then vehicle resonance can occur**
  - **Ares I**

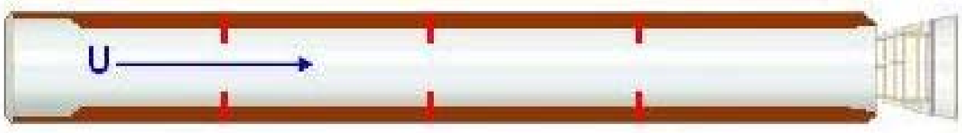


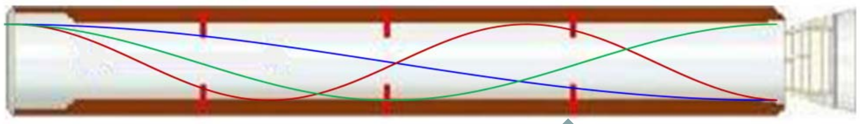


SOLID ROCKET MOTOR THRUST OSCILLATIONS: CFD INPUTS TO INCREASED UNDERSTANDING OF FLOWFIELD

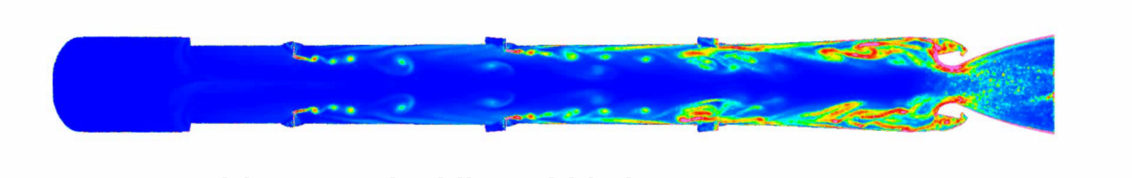


(3) 1L,2L,3L acoustic mode shapes create subsequent thrust oscillations





(2) Wave generation rate tunes with SRM 1L, 2L, 3L acoustic modes

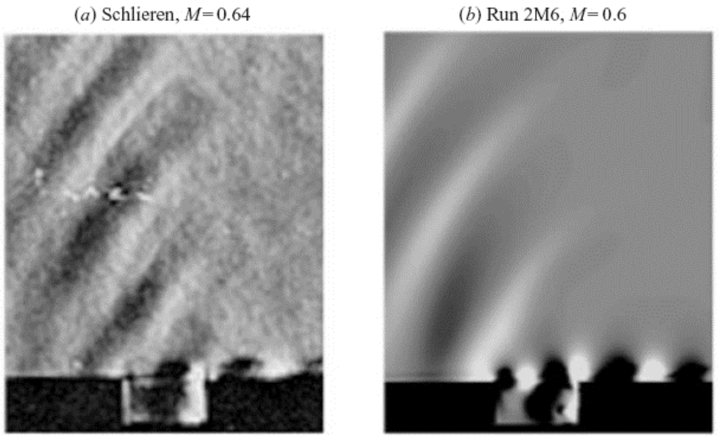


(1) Vortex shedding within internal SRM flow field causes pressure perturbations

Ongoing Improvements

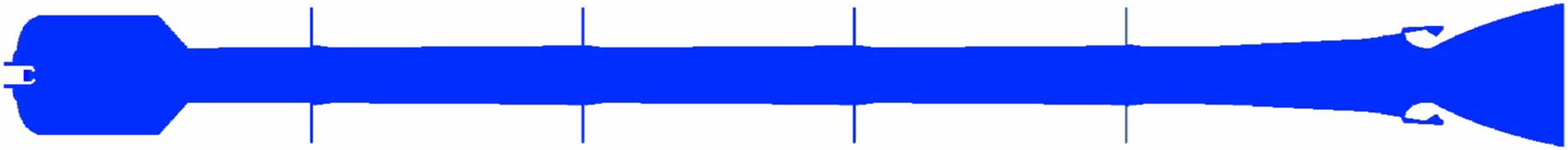
- Efficient LaGrangian particle tracking
- 2-phase capability to model slag dynamics
- Acoustic source location and mode extraction from CFD results

Oscillations in flow over rectangular cavities

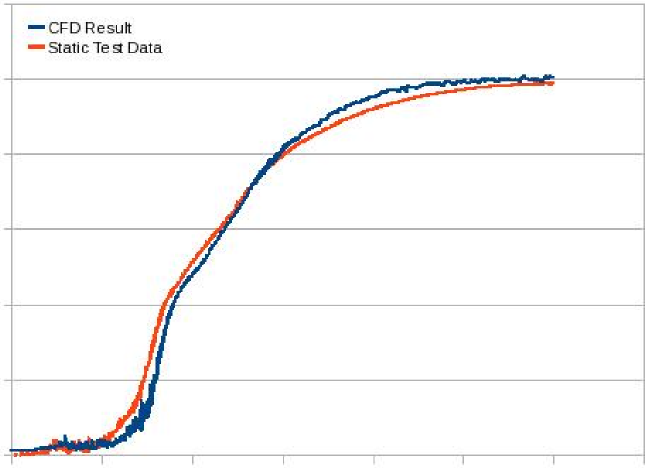

(*a*) Schlieren, $M=0.64$
(*b*) Run 2M6, $M=0.6$

SOLID ROCKET MOTOR IGNITION

- The ignition transient is a critical part of motor operation
- Elevated thrust rise rate is too high threatens vehicle structural integrity
- CFD ignition simulation
  - As-cast motor geometry mesh with ~ 150M cells
  - Simulation execution complete on 2400 CPUs in less than 2 weeks
  - Results are being used to help understand test stand dynamics issues



Pressure field during first ~ 0.6 s of large motor ignition transient



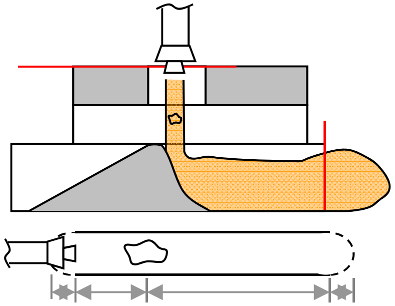
CFD results compared to head end pressure trace from static test

Ongoing Improvement Efforts

- Efficient LaGrangian particle tracking
- Propellant grain recession capability to enable appropriate propellant geometry during longer transient simulations

LAUNCH ENVIRONMENTS

1D Linearized Physics Models



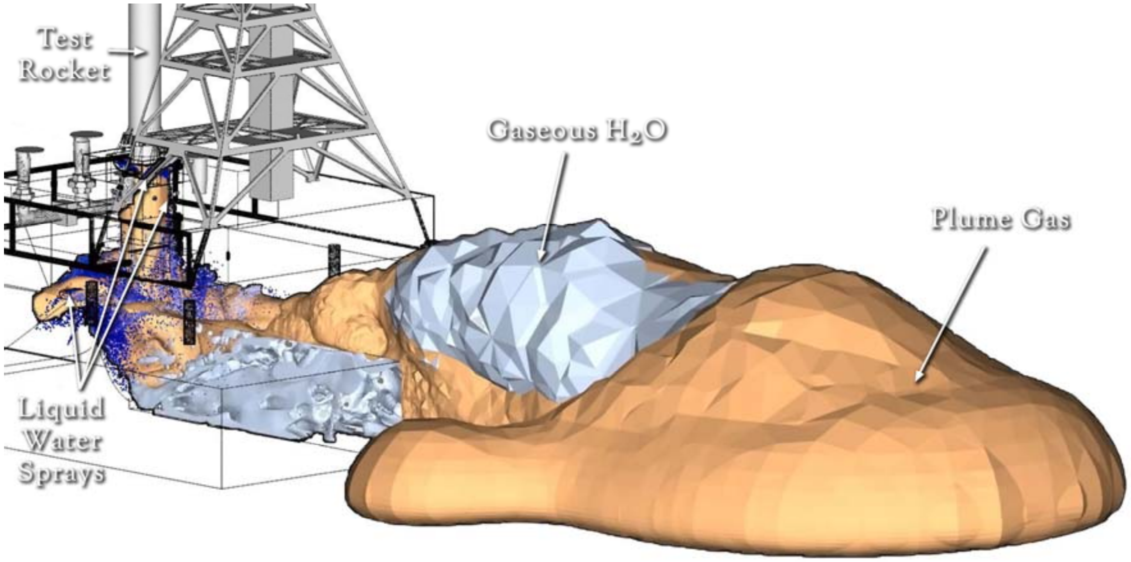
ER42 Develops the Fluid and Acoustic Environments for Launch

- Liftoff Acoustics
- Overpressure
- Sound Suppression
- Liftoff Debris Transport
- Hydrogen Entrapment

ER42 Uses Multiple Levels of Analysis and Testing to Accomplish this Work

Flight Tests

Scale Model Tests

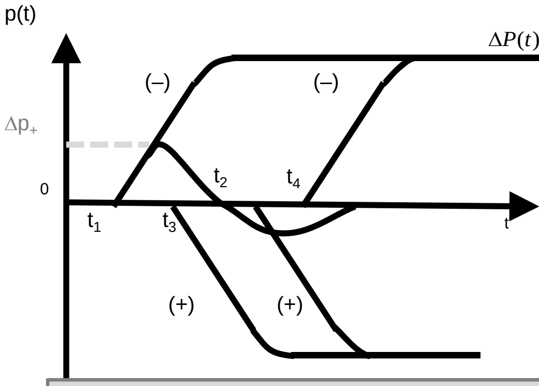


CFD

OVERPRESSURE – ANALYTICAL MODEL

Overpressure Predictions Using Analytical Models

- *Broadwell &Tsu Model*: Linearized 1-D physics-based model for overpressure in a ducted launcher
- *4-wave model*: Acoustic modification to incorporate resonant conditions
- *Attenuation Model*: Empirically based on Shuttle data or other motor/ engine correlations
- *Knockdown Factors for water suppression or pressure wave diffraction*: Empirically-based or CFD simulation-based
- *Margin*: Technical agreement based on CFD simulations and unknown
- Improvement – Continually improve models based on CFD, Test data, and Flight data



$$\Delta P(t)^{\pm} = \left[\frac{\gamma_0 P_0 \dot{m}_e}{C_0 \rho_{e0} 2A} k_m \pm \frac{F_T}{2A} k_f \right]$$

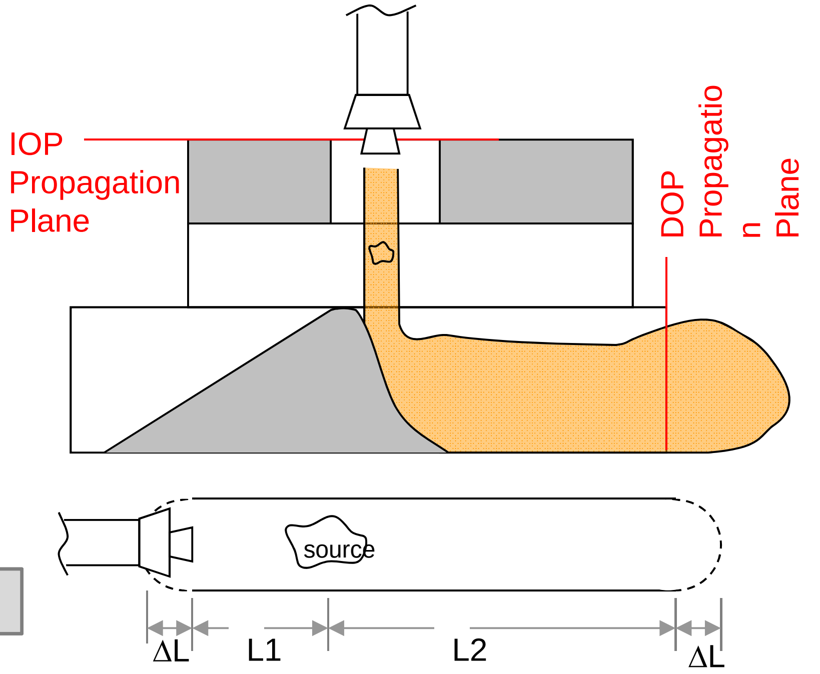
$$t_1^{-} = \frac{L_1}{c_0}$$

$$t_2^{+} = \frac{2(L_2 + \Delta L) + L_1}{c_0}$$

$$t_3^{+} = \frac{2(L_1 + \Delta L) - L_1}{c_0}$$

$$t_4^{-} = \frac{2(L_1 + \Delta L) + 2(L_2 + \Delta L) - L_1}{c_0}$$

4-Wave Physics Model



Broadwell and Tsu Model Application

OVERPRESSURE – CFD

CFD has recently shown to represent overpressure very accurately without the inclusion of water

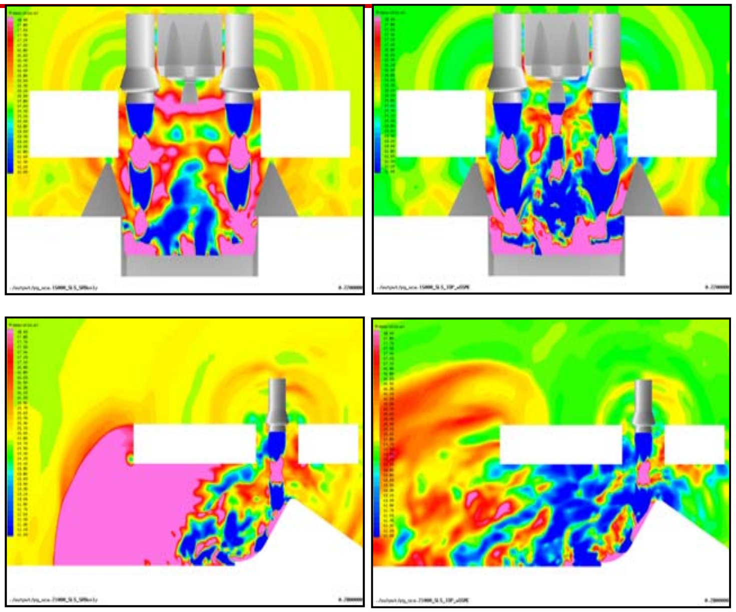
- Demonstrated ability to capture IOP and
- DOP waves at several locations for dry tests

Provides ability to address limitations of Analytical models

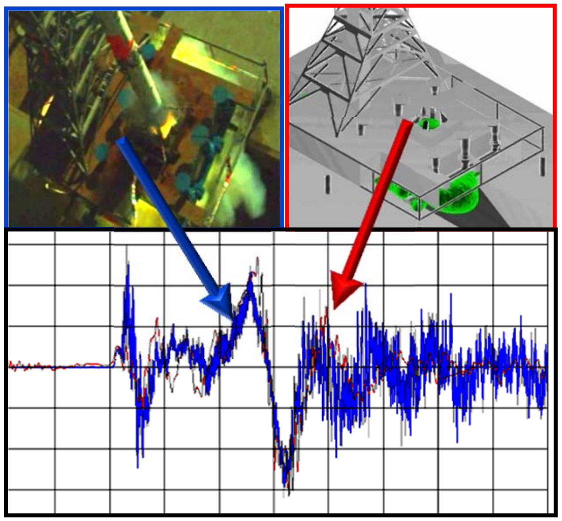
- Accounts for complex flow scenarios and three-dimensional launch pad geometry

Provides parametric studies where unknowns currently exist

Ongoing improvements include modeling water suppression systems, multiphase solid booster effluent, and capture higher frequency spectral content

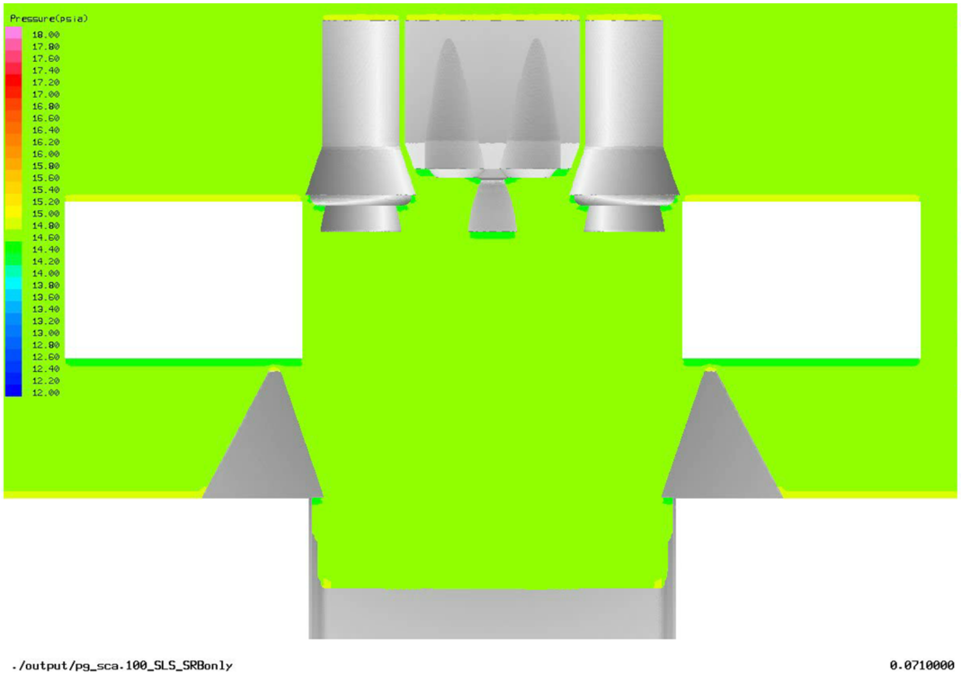


CFD simulations with (right) and without (left) liquid engine plumes

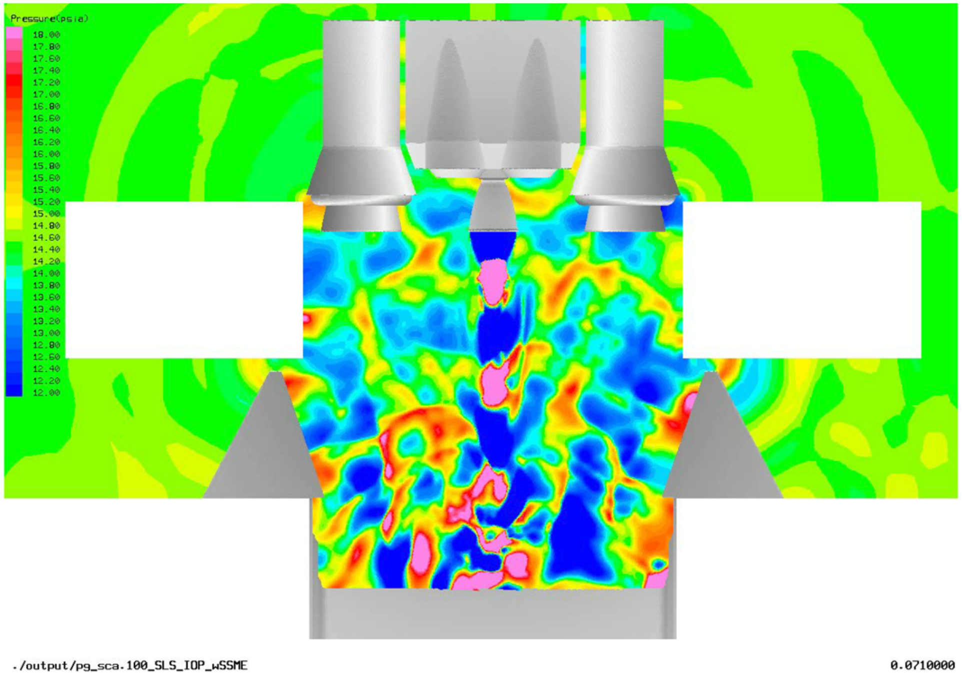


CFD simulations with (right) and without (left) liquid engine plumes

OVERPRESSURE – CFD ANIMATION



SRB Only



SSMEs Full Power

LIFTOFF ACOUSTICS

DESIGN NEW LAUNCH VEHICLE

DERIVE LIFTOFF ENVIRONMENTS

- Liftoff noise is generated by the mixing of rocket exhaust flow with the surrounding atmosphere and its interactions with surrounding launch pad structures.

- ER42 creates initial liftoff acoustic environment derived from Saturn V, Space Shuttle flight data, and Ares I-X flight test data. for the development of Ares I and the proof-of-concept vehicle, Ares I-X. Parametrics and identification of sources from CFD

VALIDATESCALE MODEL ACOUSTIC TEST

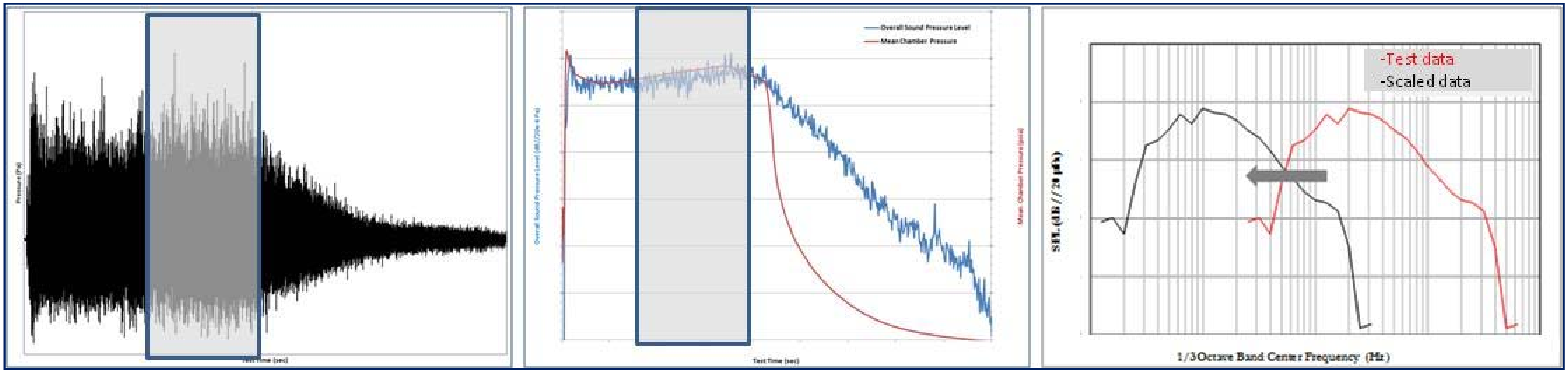
- Use acoustic scale model test to validate liftoff acoustic environments and water sound suppression system design.

SCALE MODEL ACOUSTIC TESTING

- **Determine model scale using Strouhal Number**

$$St = \left(\frac{f_1 d_1}{V_1}\right) = \left(\frac{f_2 d_2}{V_2}\right)$$

- **Design test article to this scale; fire; acquire data.**
- **Data Processing**

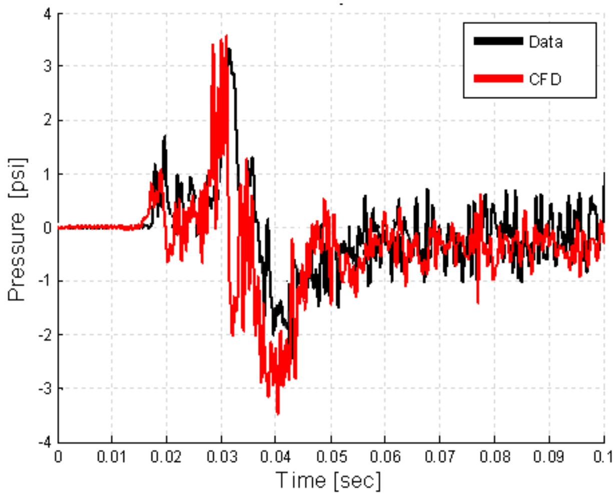


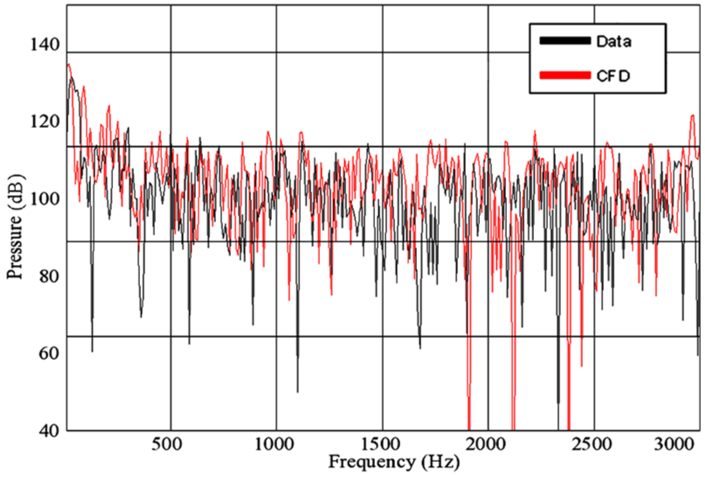
Typical pressure time history with analysis window (a) and analysis window overlaid on chamber pressure measurement and RMS OASPL time history (b) and a one third octave plot for the test data compared to the scaled data (c).

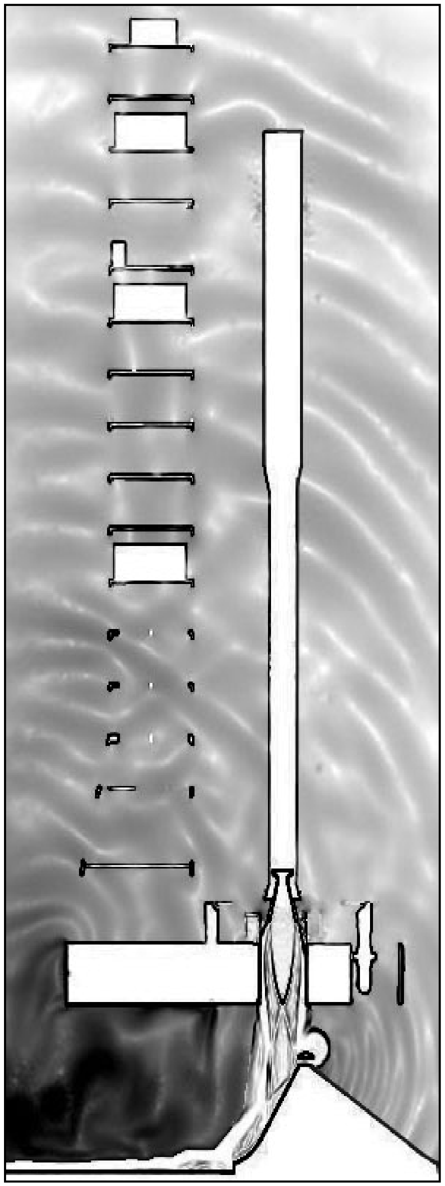
SCALE MODEL TEST MOVIE

ASMAT VALIDATION OF CFD
(COMPARISONS OF FREQUENCY WITHIN DUCT)

- Simulations of 5% scale rocket to model transient startup of motor
- Validated pressure temporal/spectral accuracy of CFD vs test data.
- Simulations showed good correlation with test data.
  - Matched pressure content above deck to 1000-1500 Hz
  - Matched pressure content below deck to 2000-3000 Hz
- Provided rationale and confidence to use CFD to predict environments for full-scale vehicles (up to ~ 150 Hz)



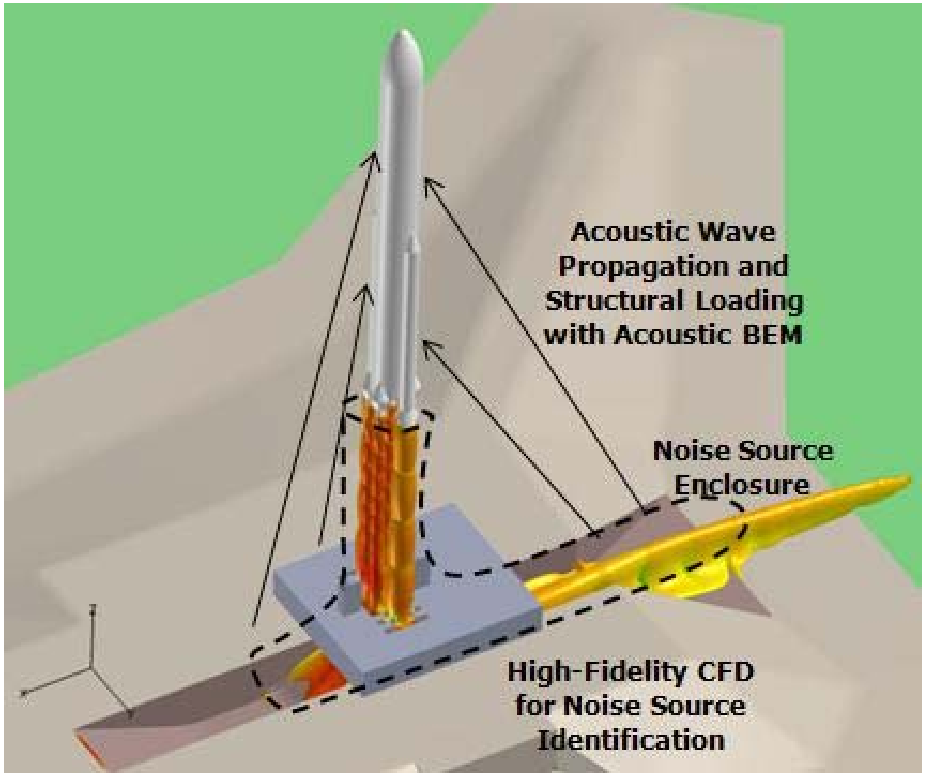




APPROACH TO ACOUSTICS PROPAGATION CHALLENGE

Solution: Implement hybrid approach of CFD + Computational Aero Acoustics (CAA) for liftoff acoustic fields

- Use high-fidelity CFD modeling to capture important plume physics (multi-phase plume, plume mixing and impingement, gas-water phase effects from deluge, etc.)
- Capture acoustic sources originating from plumes, impingement, capture water suppression effects
- Propagate using CAA from acoustic source surfaces enclosing noise source regions

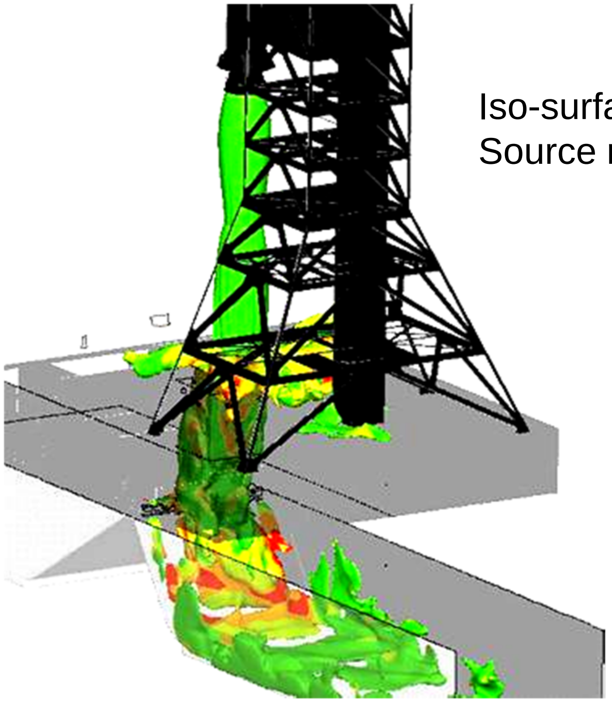


Which CAA method is best suited for this application?

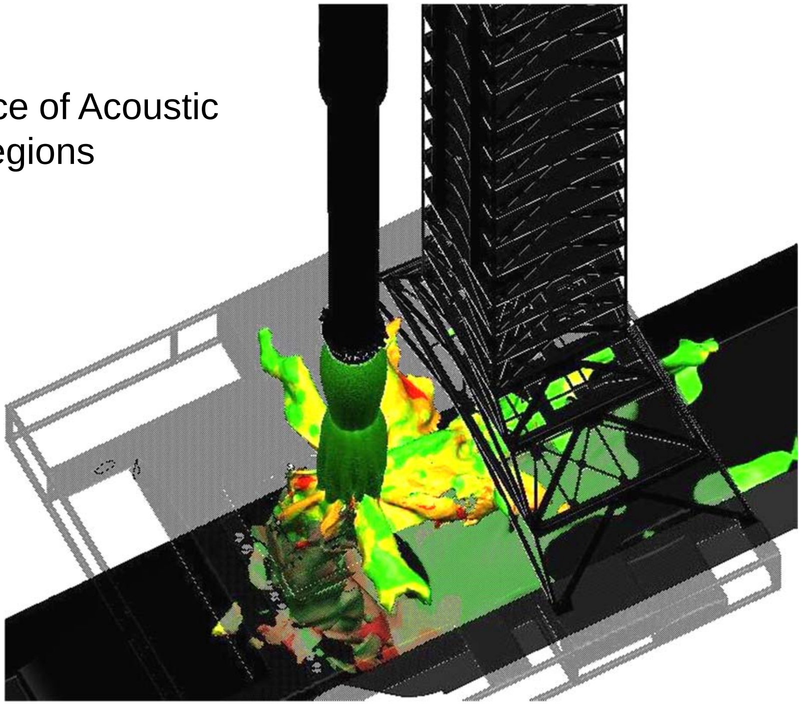
- CAA acoustic field propagation method must be able to resolve reflections, refraction and attenuation from interaction with structures such as launch platform and tower
- Two approaches under evaluation:
  - Boundary Element Method (BEM)
  - Farfield high-order Euler solution

CHALLENGE: IDENTIFICATION OF THE ACOUSTIC SOURCE REGIONS

- Major challenge arises in defining envelope of source regions for handover from CFD to CAA
- Plume boundary shape is quite complex due to interaction with launch pad
- Example: Visualization of Noise Source regions for ASMAT Plume Impingement

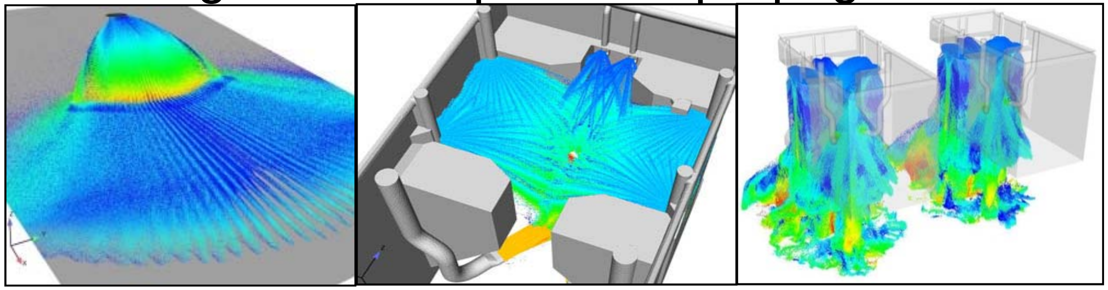


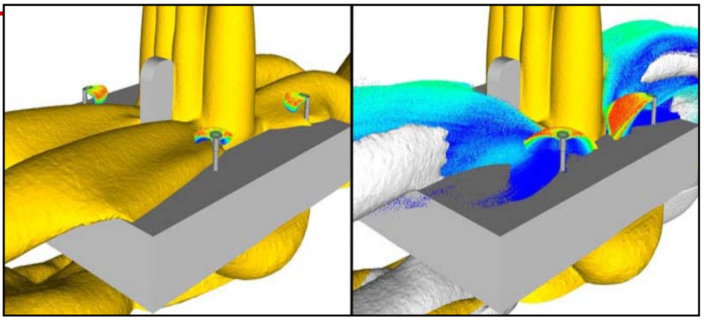
Iso-surface of Acoustic Source regions



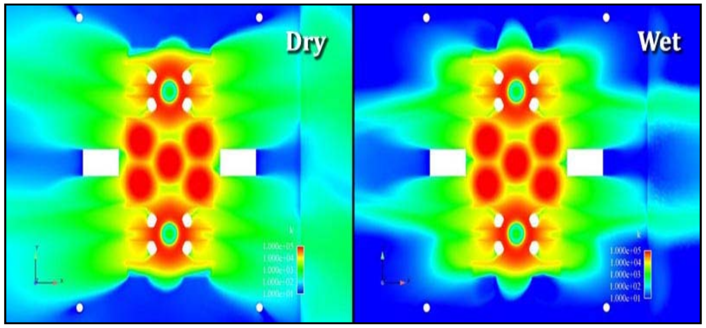
CHALLENGE: SIMULATION OF WATER MITIGATION IN CFD

- Using Lagrangian Particle model to simulate water injection into launch pad plume environment for SLS concepts, Space Shuttle, and scale tests.
- Injecting water at up to 200,000 gal/min
- Simulating up to 30M active particles
- Liquid drop emission from booster holes, trench deflectors, or from rainbird systems
- Modeling water break-up and phase change
- Considerable changes shown in turbulent kinetic energy on deck, plume temperature, and ignition overpressure propagation.

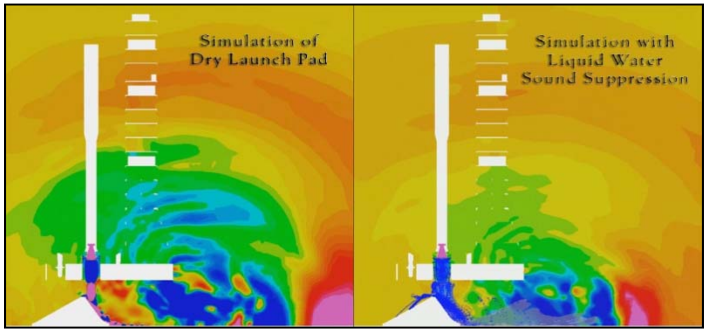




Reduction of Plume Temperature by Water Deluge



Reduction of Kinetic Energy at Deck Level



Reduction of Ignition Overpressure

SUMMARY

- The Fluid Dynamics Branch at MSFC has the mission is to support NASA and other customers with discipline expertise to enable successful accomplishment of program/project goals
- The branch is responsible for all aspects of the discipline of fluid dynamics, analysis and testing, applied to propulsion or propulsion-induced loads and environments, which includes the propellant delivery system, combustion devices, coupled systems, and launch and separation events
- ER42 supports projects from design through development, and into anomaly and failure investigations
- ER42 is committed to continually improving the state-of-its-practice to provide accurate, effective, and timely fluid dynamics assessments and in extending the state-of-the-art of the discipline