# The Role of CFD Simulation in Rocket Propulsion Support Activities

July 26,2011

Presented by: Jeff West

Fluid Dynamics Branch (ER42)
Propulsion Systems Department
NASA/Marshall Space Flight Center
MSFC, AL 35812

# Presentation Overview

- CFD at NASA/MSFC
  - Flight Projects are the Customer—No Science Experiments
  - Customer Support
  - Guiding Philosophy and Resource Allocation
  - Where is CFD at NASA/MSFC?
- Examples of the expanding Role of CFD at NASA/MSFC
  - Liquid Rocket Engine Applications
    - Evolution from Symmetric and Steady to 3D Unsteady
  - Launch Pad Debris Transport-> Launch Pad Induced Environments
    - STS and Launch Pad Geometry-steady
    - Moving Body Shuttle Launch Simulations
    - IOP and Acoustics Simulations
  - General Purpose CFD Applications
  - Turbomachinery Applications
- Parting Thoughts

# The CFD Customer at NASA/MSFC

NASA/MSFC is a Space Flight Center. This means that Science Experiments are not appropriate. However, Applied Science to achieve the Customer's objectives is desired.

At NASA/MSFC, the Customer is a Program, and most of the time an actual Flight Program. Examples include:

- Space Shuttle Program
  - Space Shuttle Main Engine Element
  - Solid Rocket Booster Element
  - Ground Operations Element
- Constellation Program
  - Ground Operations Element
  - Solid Rocket Booster Element
  - J-2X Engine Element
  - Upper Stage Element
- External Entities
  - Air Force
  - Commercial Space Flight

# Definition of Successful Customer Support

**<u>In most high visibility situations, CFD must show value to the customer within three weeks of ATP.</u>**

- Once value is demonstrated, the Customer has more patience

The implication of this fact is that the entire CFD simulation process must be optimized for short turn-around operation

- Geometry and mesh generation
  - Mesh generation used to require weeks, now the typical timeframe from CAD to CFD simulation start is 1 to 2 days.
    - Hybrid mix of Commercial and Research software
      - » ANSA and AFLR (MSU)
- Highly parallel CFD software
  - Loci/Chem (MSU) and Loci/STREAM (MSU-SNI)
    - High Speed and Low Speed flows, respectively
- Highly parallel post processing software augmented with commercial applications
  - Home-grown, batch-mode post processing (AnimatorP)
  - Serial commercial software (Fieldview, Ensight, TecPlot)

# CFD Tool Development Philosophy

Select the most challenging ***practical and likely*** future simulation and create a suite of tools that, when completely developed, can meet that Grand Challenge

- Grand Challenge: Rocket Engine/Motor Combustion Instability

If the Grand Challenge problem is selected properly, then one can achieve practical progress while also being able to solve the semi-challenging problems along that path.

- The Grand Challenge tool is then the General Purpose tool.
- Not terribly efficient in terms of lower levels metrics, but efficient at the most visible level; the time from ATP to first result

The Working Challenge then becomes whether a collection of researchers, CFD algorithm developers, mesh generation developers and CFD analysts can be gathered to work in a sustained manner to complete the Grand Challenge.

- This challenge began FY 2001.

# Resource Allocation Philosophy

Spend controllable monetary resources on growth, not software licenses.

- In FY99, Fluent was $50K/cpu/year…
  - For tight budgets, this model fails to allow execution of the CFD discipline

Growth means software capability and local compute resources

- Support software development with perpetual, unlimited use arrangement
  - Required with most government contracts
- Characteristics of efficient local compute resources
  - Construct a Linux cluster for one or two applications
  - Use open source software
  - Choose system administration personnel wisely!
  - Dominant cost should be acquisition, not administration
- Exploit remote compute capability as much as possible (NASA/AMES, DoD, etc)
  - User will have virtually no influence over remote compute hardware- but the taxpayer has already paid for the hardware

# CFD Modeling Branches at MSFC

**ER42: Fluid Dynamics Branch, Propulsion Systems Department**

- Two Teams performing CFD
  - Propellant Delivery Team (14 CFD analysts)
    - Propellant Tanks: Slosh and Drain
    - Lines Ducts and Valves (MPS and Engine)
    - Turbo-machinery (Turbine and Pumps)
    - Launch Pad Induced Environments (IOP and Acoustics)
  - Combustion CFD Team (8 CFD analysts)
    - Lines, Duct, Valves, Manifolds
    - Injectors and Thrust Chambers (Performance, Stability and Environments)
    - Nozzles (Performance and Environments)
    - Solid Rocket Motors (Small and Large Scale)

**EV33: Aerosciences Branch, Spacecraft and Vehicle Systems Department**

- Two Teams performing CFD
  - Aero-Thermodynamics (6 CFD analysts)
    - Aerodynamics Heating
    - Plume Impingement Heating
    - Base Flow Environments
  - Aerodynamics Team (4 CFD Analysts)
    - Aerodynamic Drag
    - Aerodynamics Acoustics

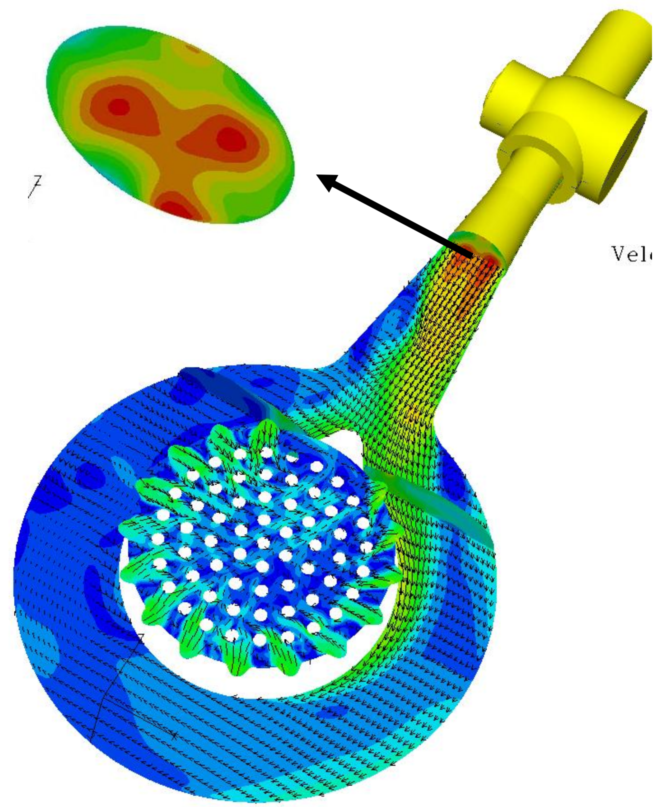
## **Examples of the Expanding Role of CFD at NASA/MSFC:**

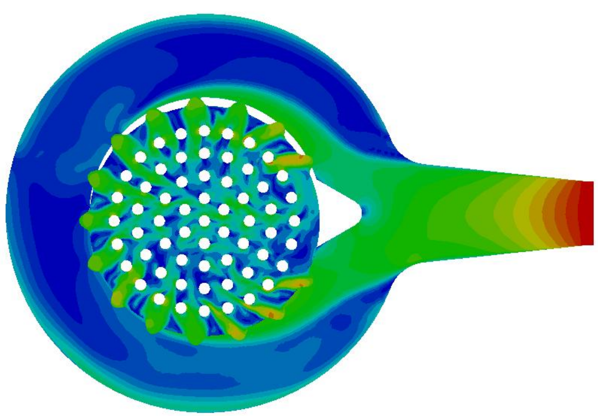
*Liquid Rocket Engine Applications*

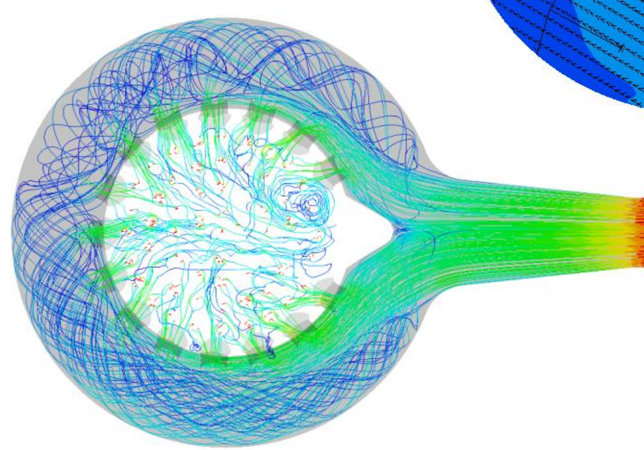
# Manifold Flow Uniformity-Steady

**<u>2004 to 2007 timeframe</u>**

- Steady state simulations
  - Primarily Loci/CHEM
- Unstructured mesh simulations
  - Gridgen surface geometry, AFLR Volume mesh
  - Integrate to the wall (y+~1)
  - 20 to 45 Million cells
- Objective was the assessment of flow uniformity



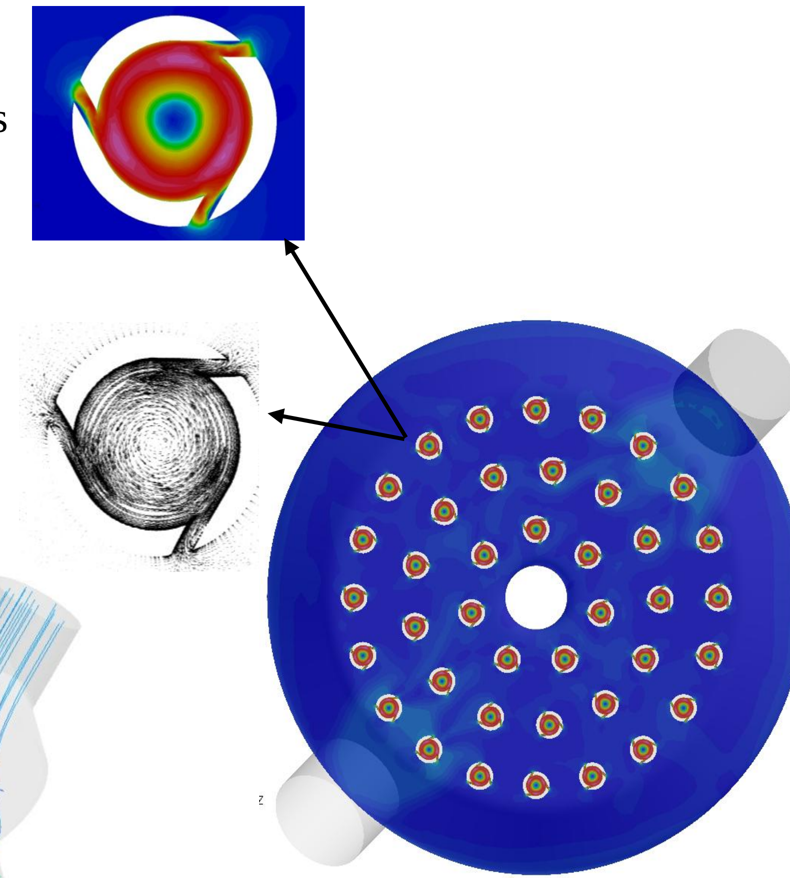


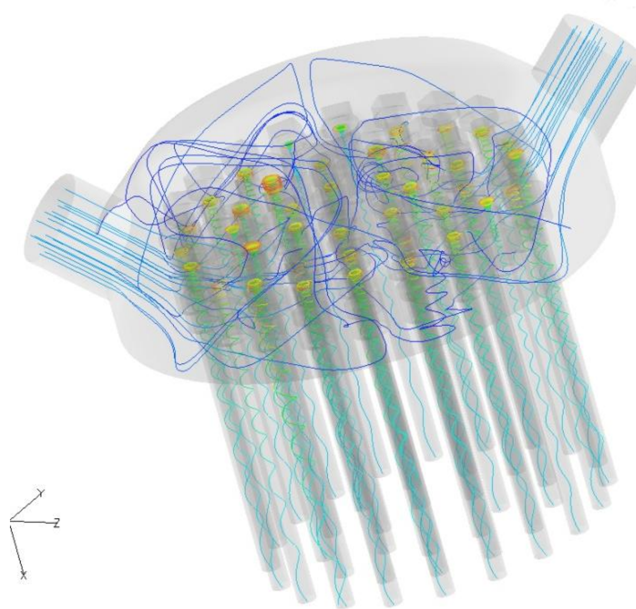


# High Fidelity Manifold Simulations

**<u>2007 onwards</u>**

- Oxidizer manifold (40 Elements)
  - Computational domain include injector inlets
  - Loci/Chem and Loci/STREAM
  - ANSA Surface Mesh and AFLR Volume mesh
  - Integrate to the wall y+~1
  - 30 to 90 Million Cells
- Objectives included Pressure drops in addition to Flow Uniformity

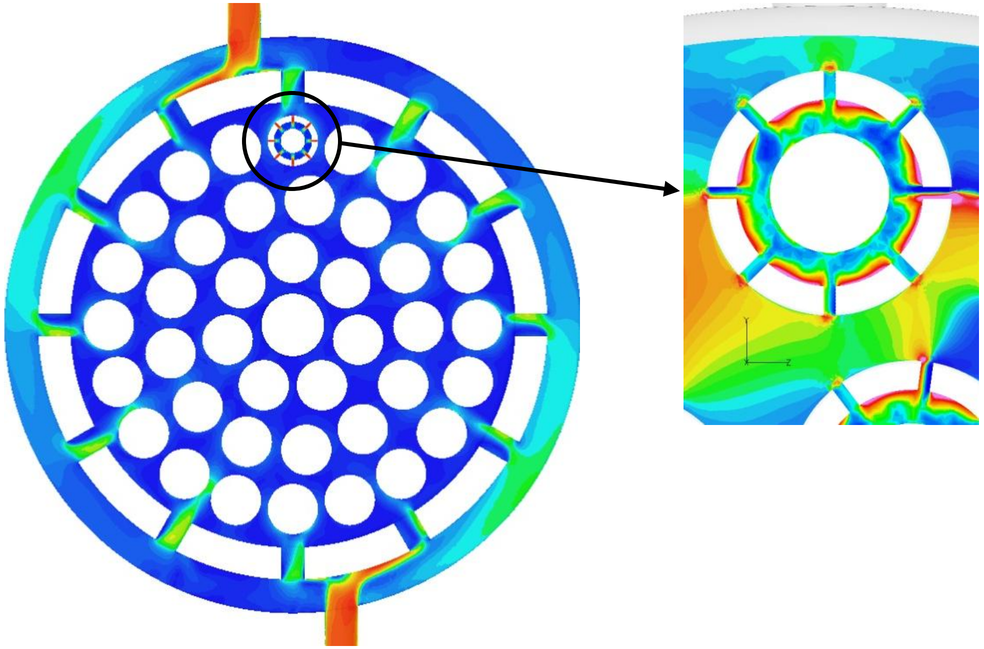


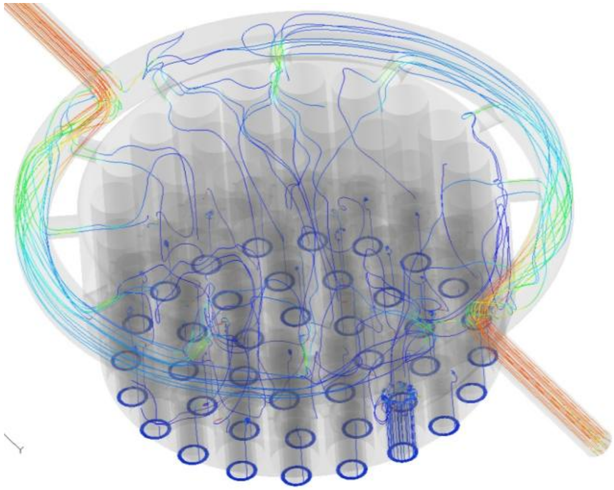


# Higher Fidelity Manifold Simulations

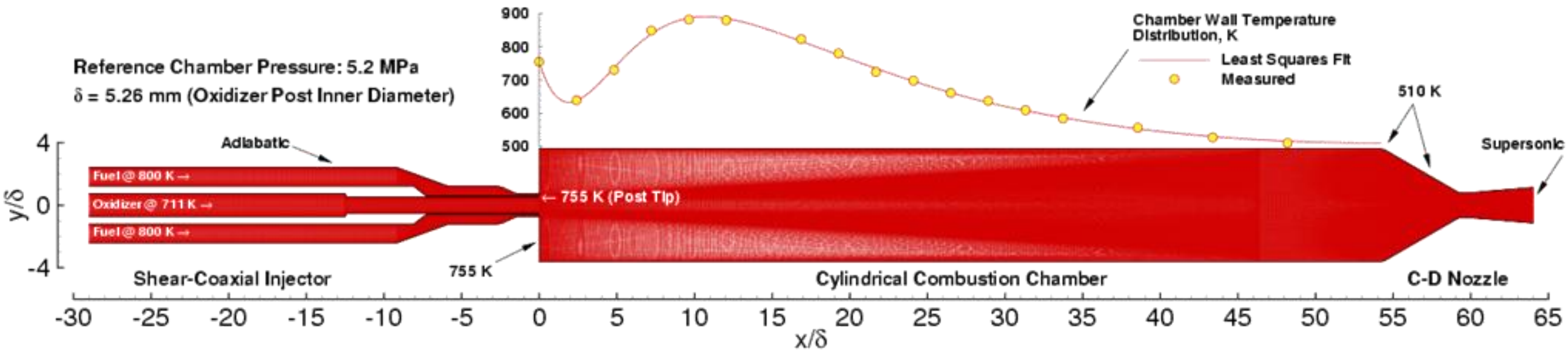
**2007 onwards**

- Oxidizer manifold (40 Elements)
  - Computational domain include injector inlets
  - Loci/Chem and Loci/Stream
  - ANSA Surface Mesh and AFLR Volume mesh
  - Integrate to the wall y+~1
  - 30 to 90 Million Cells
- Objectives included Pressure drops in addition to flow uniformity

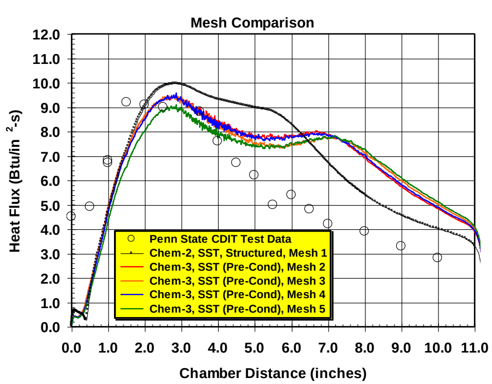




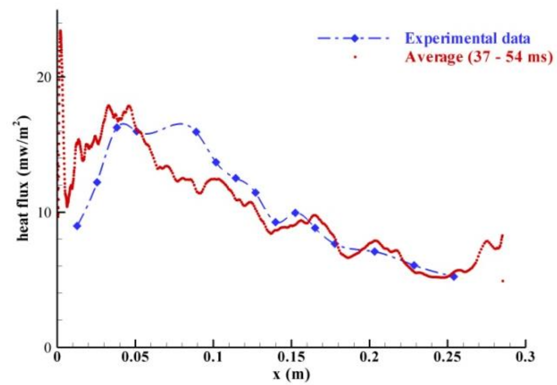
# Rocket Engine Injector Simulations



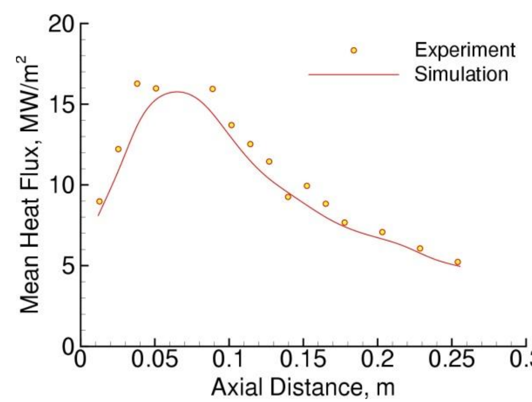

Penn. State University Validation Experiment Measured Chamber Wall Temperature and Heat Flux.
CFD Simulations *Specified* Chamber Wall Temperature and *Predicted* Chamber Wall Heat Flux.




Loci/Chem and Loci/Stream RANS
"Useful for Design"
2004-2005

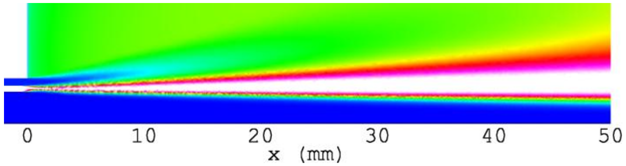



Purdue URANS (CUIP)
"Fully Capable for Design"
2006-2008

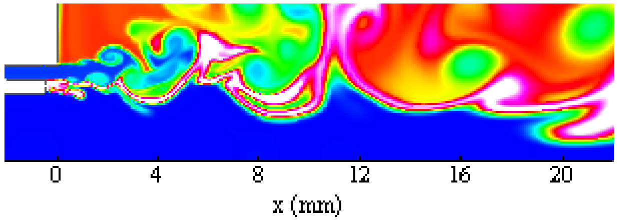



Sandia National Labs LES
"Computational Experiment"
2006-2008

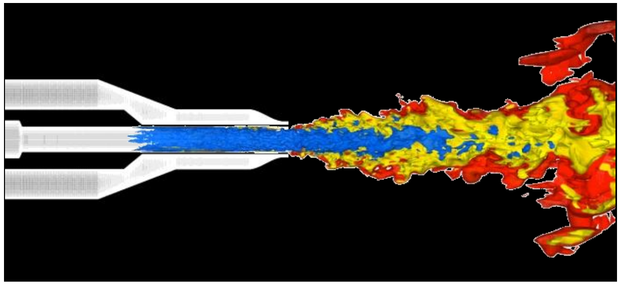
# Rocket Engine Injector Simulations

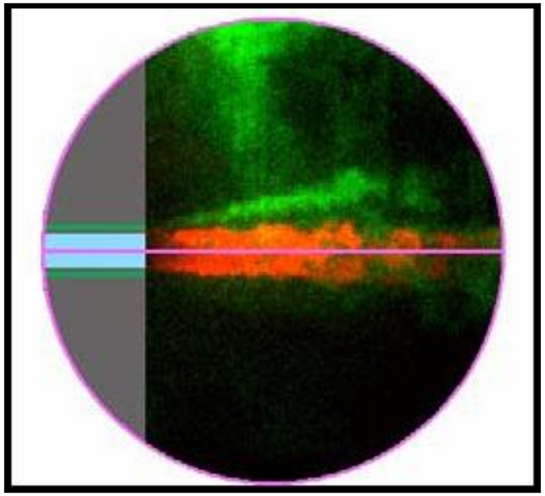


Loci/Chem Axi-symmetric RANS



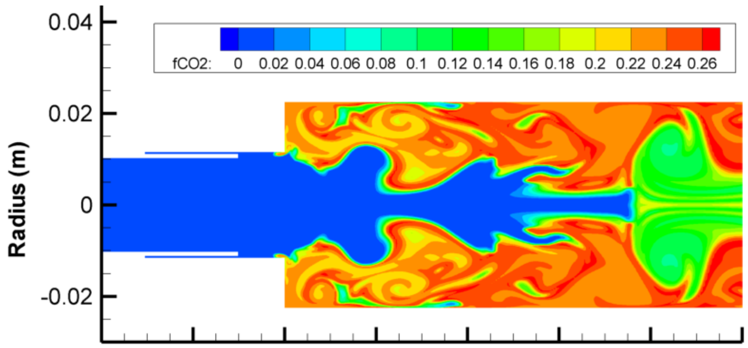
Purdue Axi-symmetric URANS (CUIP)


Sandia National Labs LES

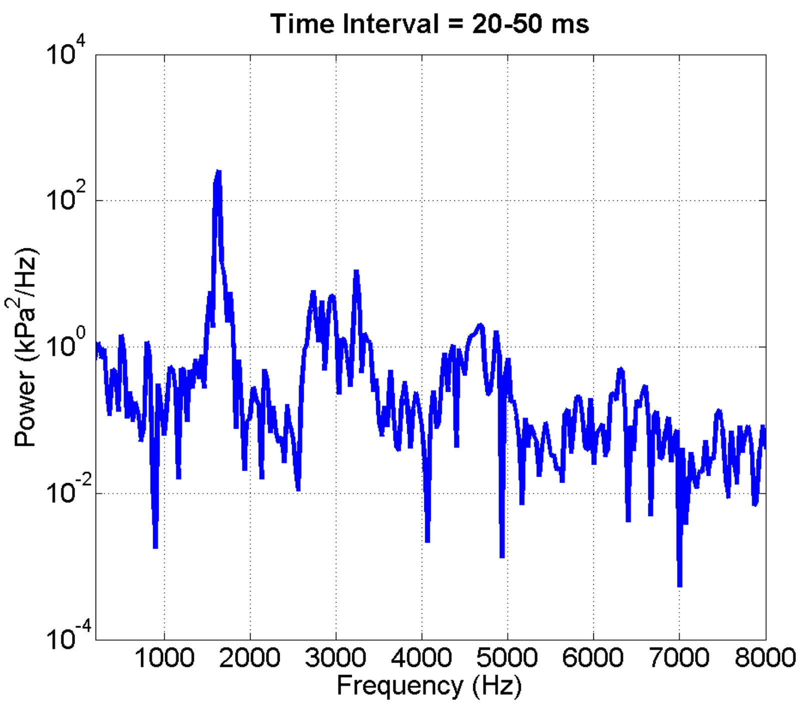

PSU Shear Coaxial Element
OH-PLIF Image

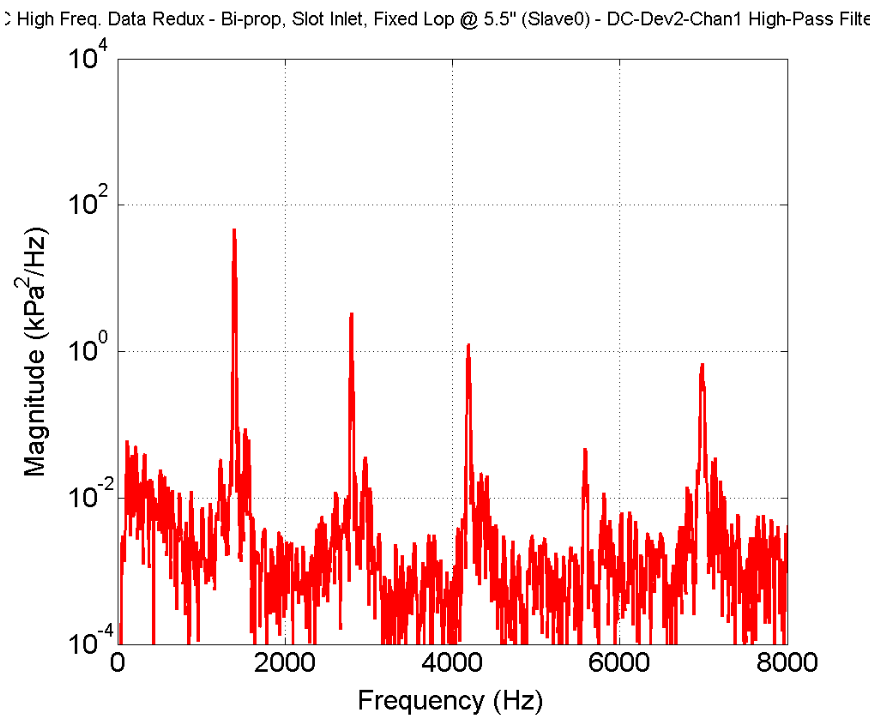
# URANS Axi-symmetric Modeling

## Loci/Chem

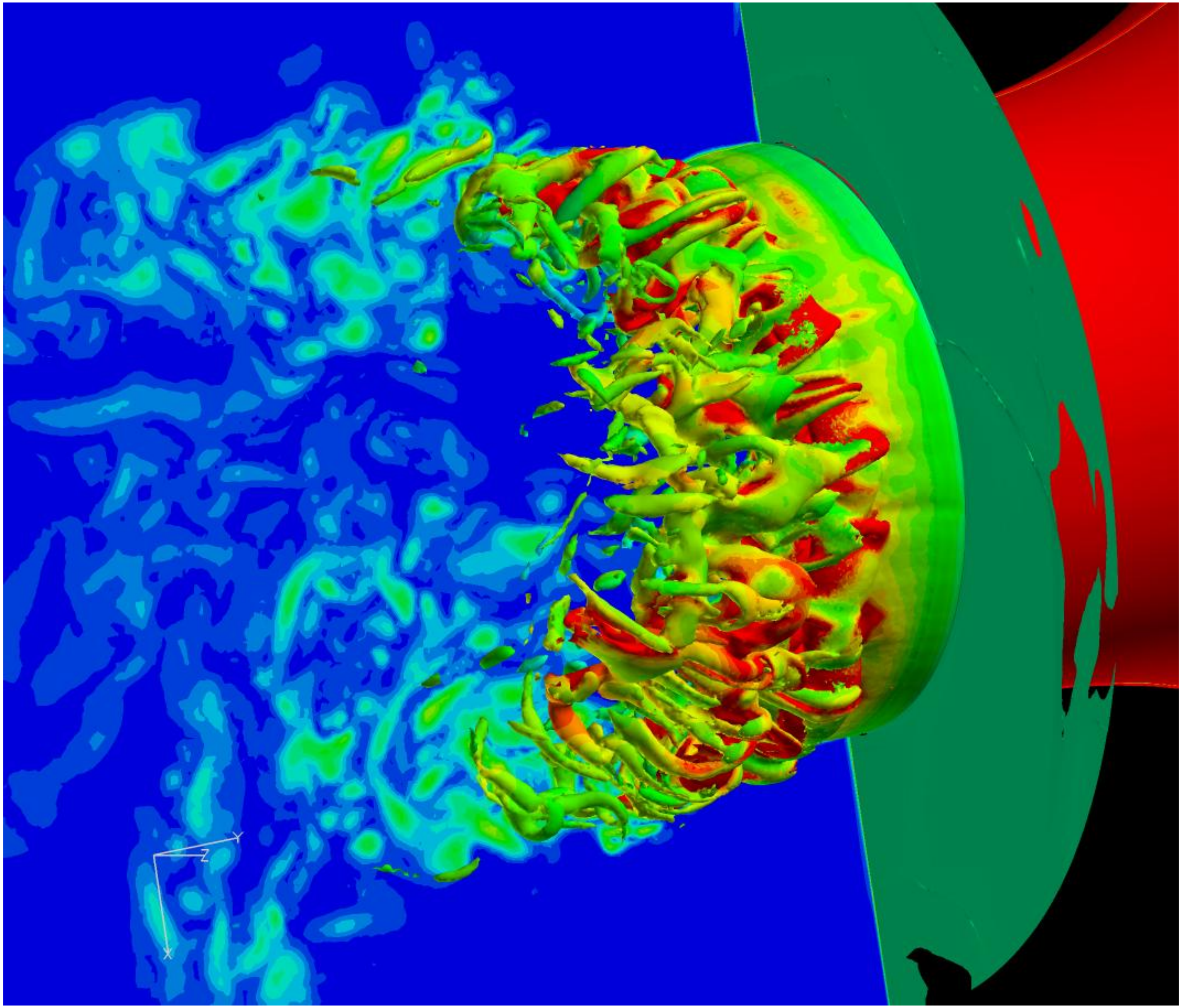



Loci/Chem URANS rocket engine injector simulations have been demonstrated to reproduce experimental unsteady characteristics.



# Unsteady 3D Modeling



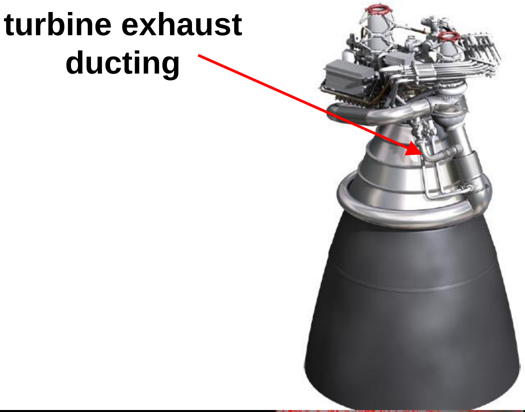
**<u>2009- 2010</u>**

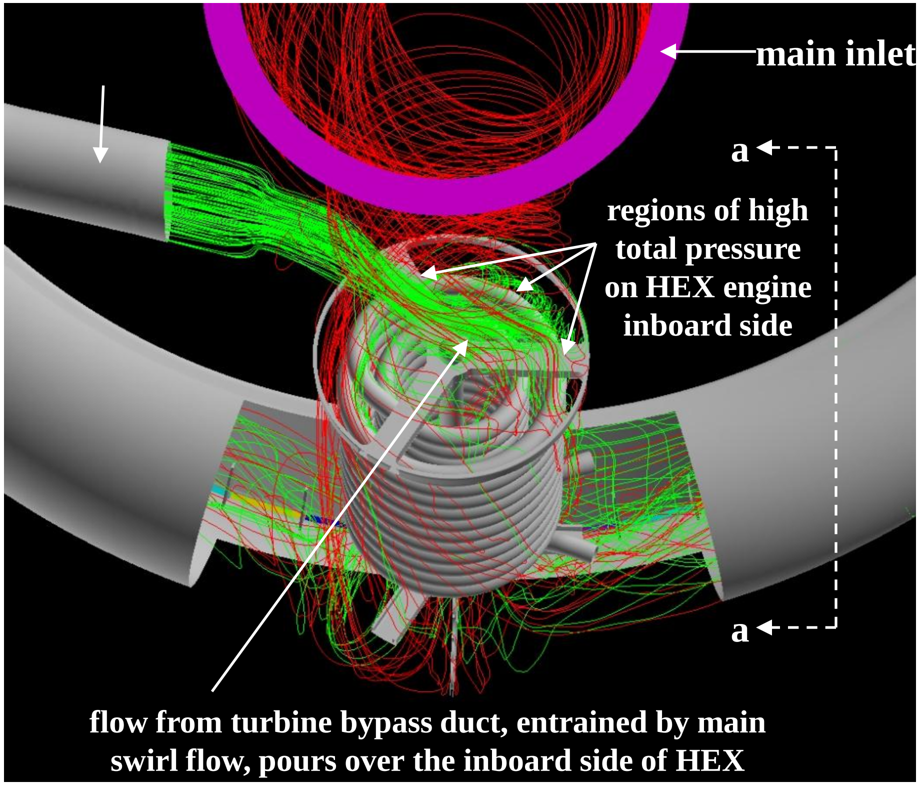
- Loci/Stream
- DES
- 40 Mcells

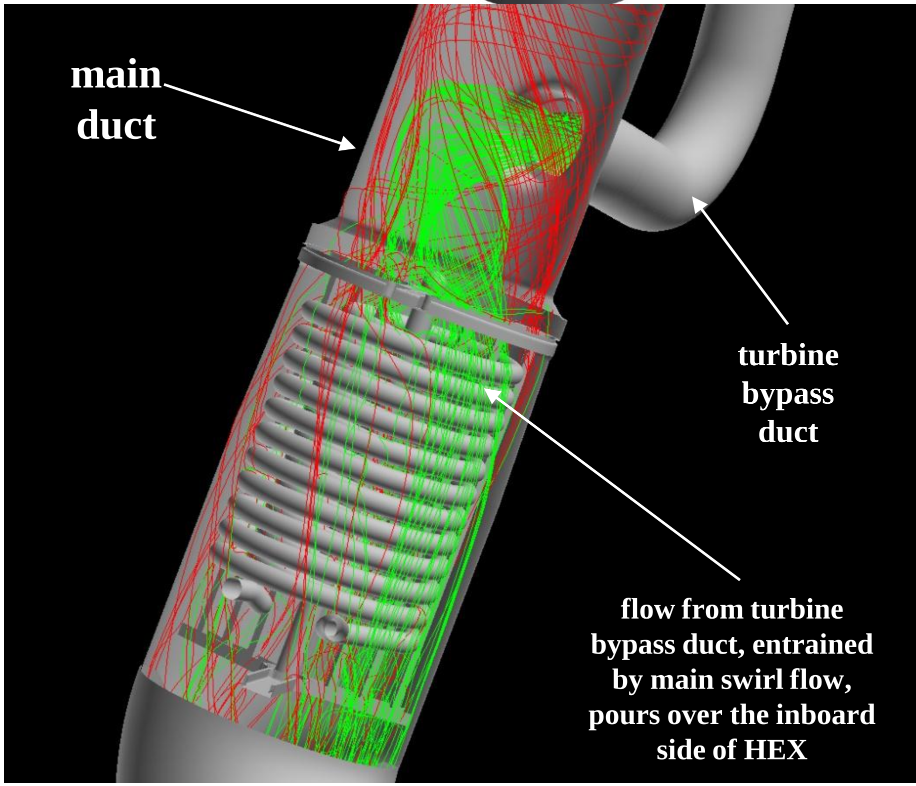
# Unsteady Duct Modeling

**2008-2010**

Loci/Chem hybrid RANS-LES simulations have been applied to improve designs of Liquid Rocket Engines. Approximately 30 simulations with 40 to 100 Million Cells.



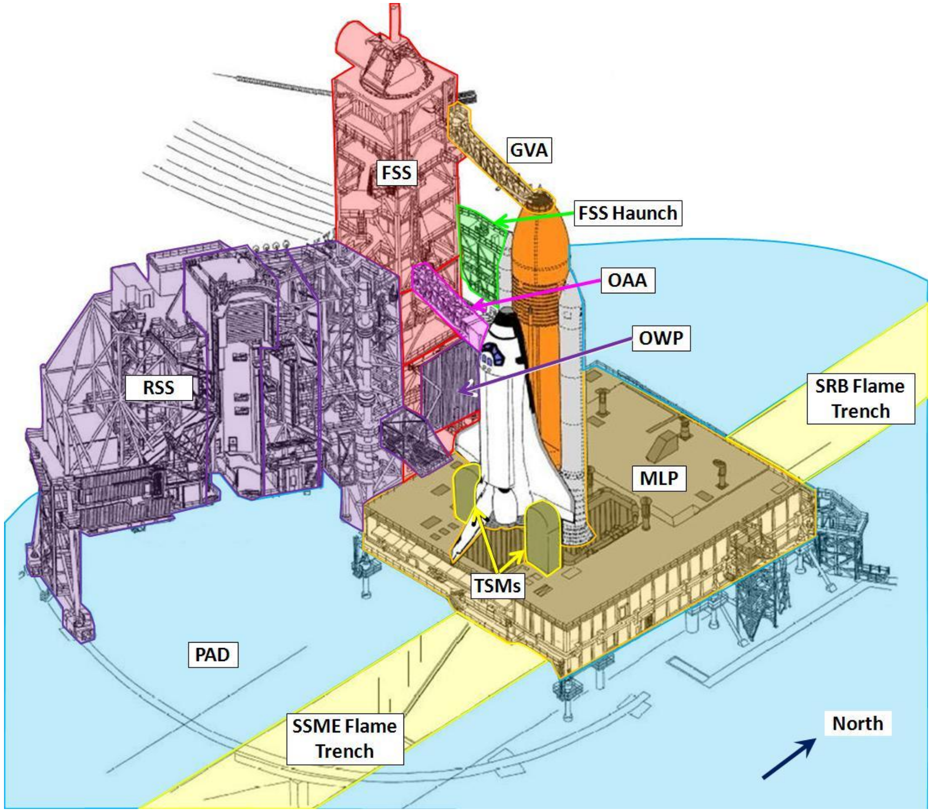
**Examples of the Expanding Role of CFD at NASA/MSFC:**

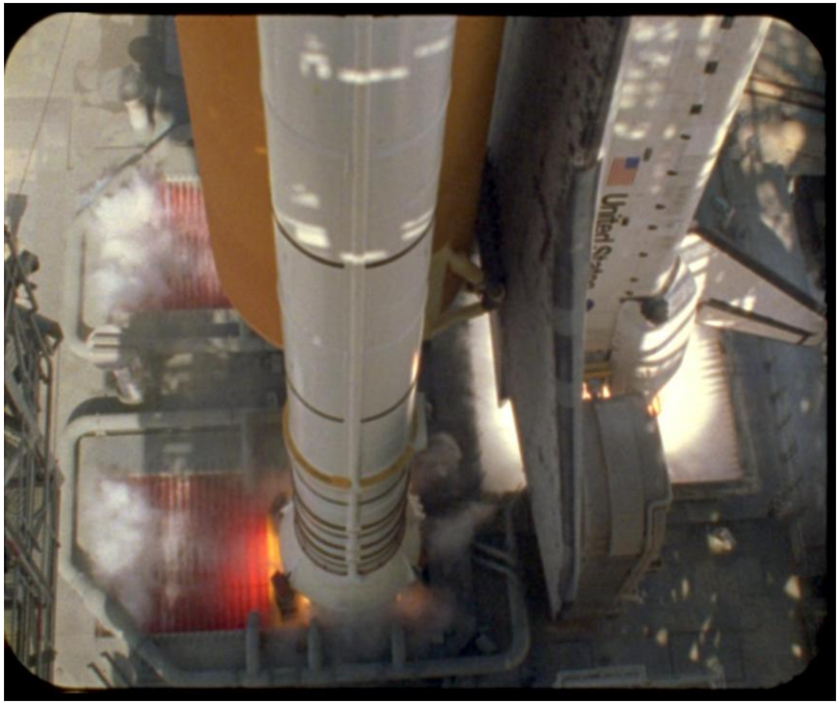
*CFD Simulations supporting Shuttle Liftoff Debris Mitigation*

# Space Shuttle Liftoff Debris

As part of the post-Columbia Return to Flight (2003 onwards), the risk of Debris during the Liftoff Phase was assessed.

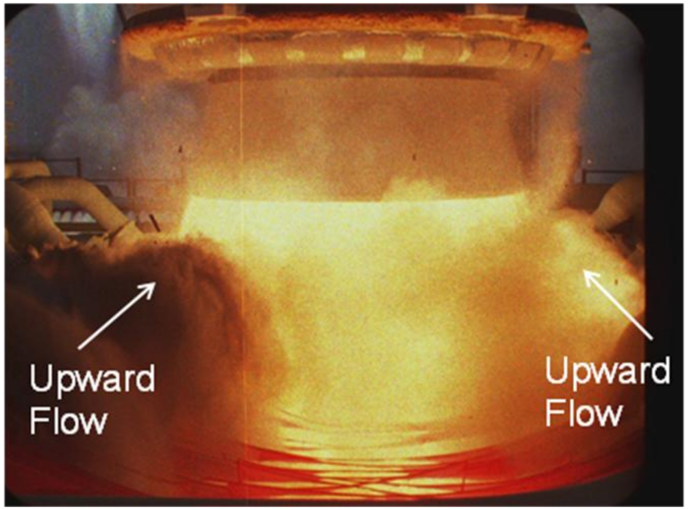
- FOD may fall from structure onto SSLV
- FOD/Debris can be propelled upward by plumes

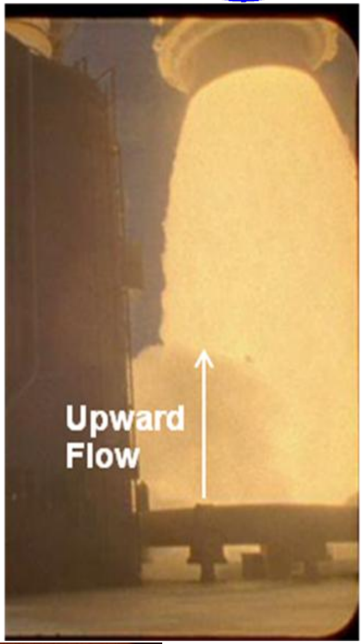


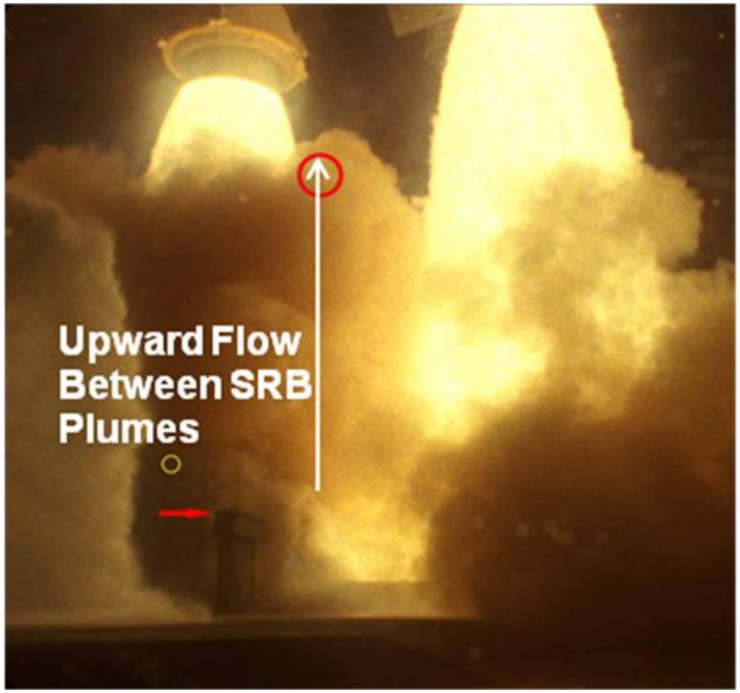



# Driving Force: Plume-Structure Interactions

To assess plume-driven debris, accurate predictions of Plume-Launch Pad Structure interactions had to be developed.



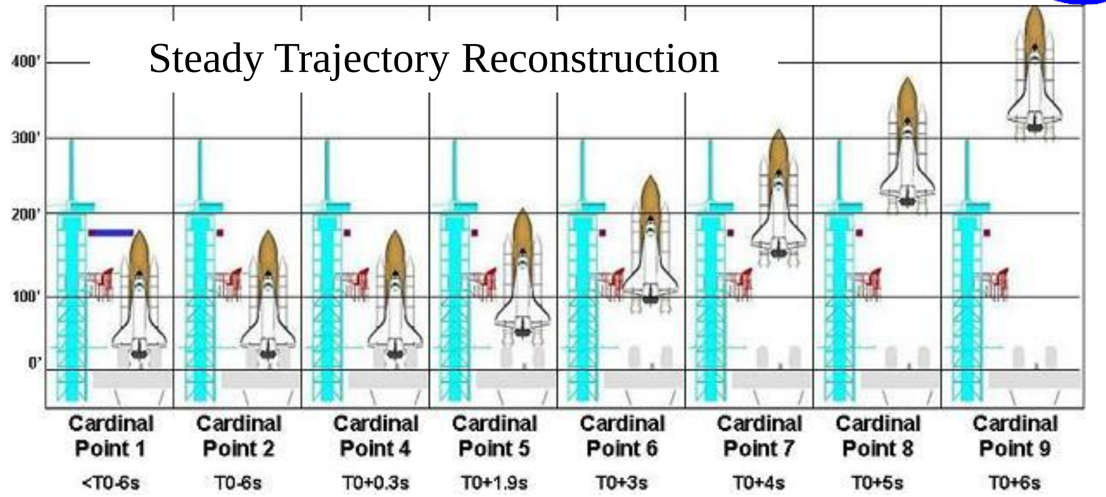



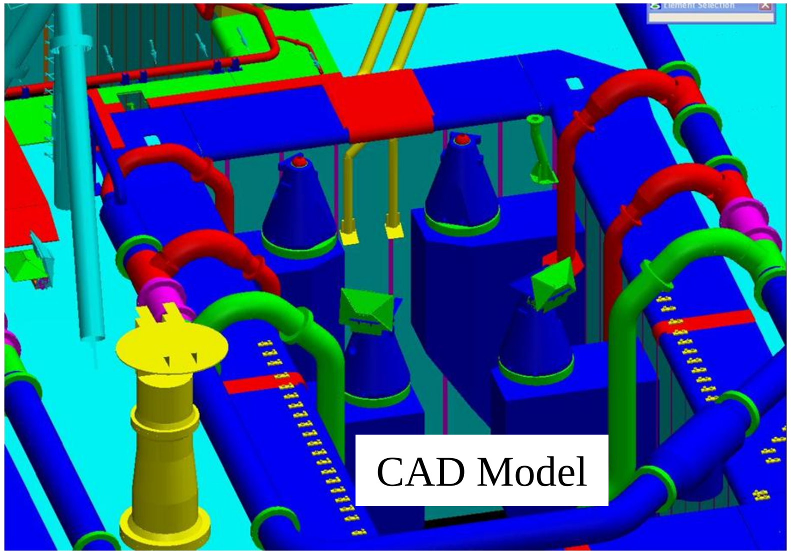



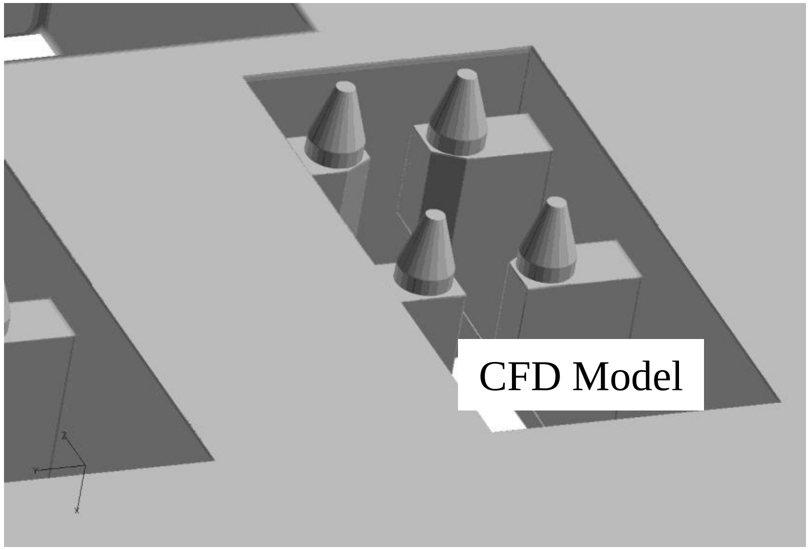

# Steady, Limited Geometry

**<u>2004-2005</u>**

Significant use and insight was obtained with steady state modeling using the Overflow CFD program. Plume-structure interactions remained a challenge.



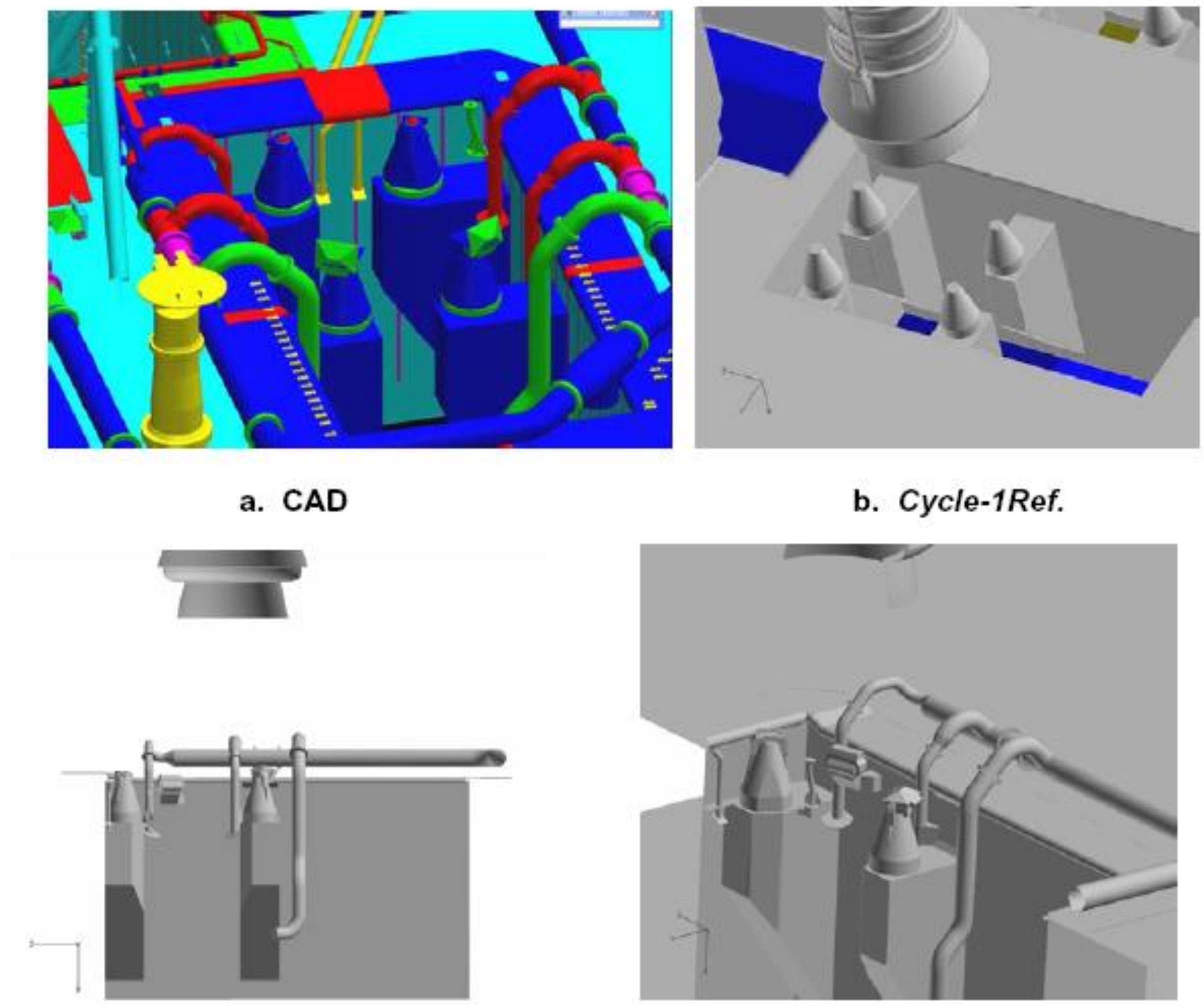



# High Fidelity Geometry- Steady Flow

**2005-2007**

Conversion to the use of AFLR unstructured meshes and Loci/Chem to address the prediction of upward flows resulting from Plume-Structure interactions.

- –Aggressive use of Geometric Symmetry
- –Highly detailed geometry
- –25 to 70 Mcells



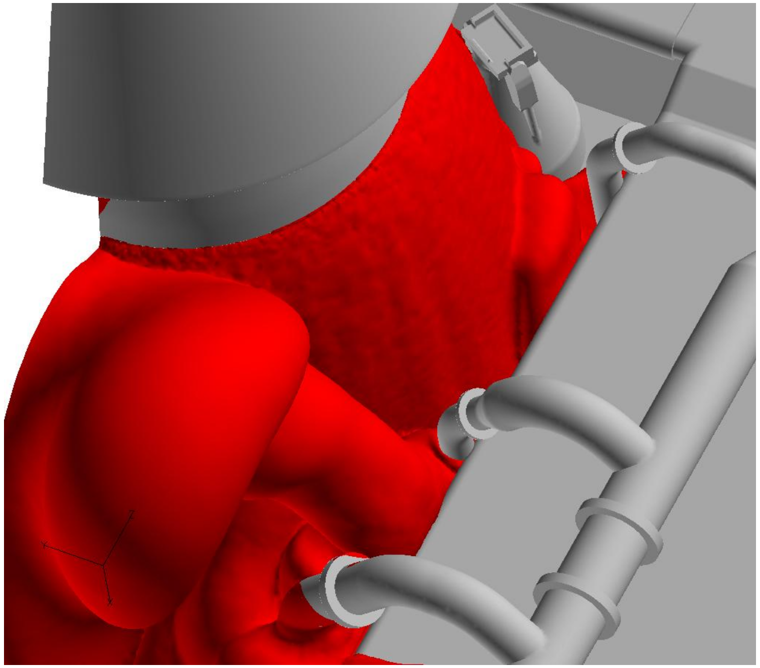

c. Cycle-2 (Detailed)

# High Fidelity Geometry-Steady Flow

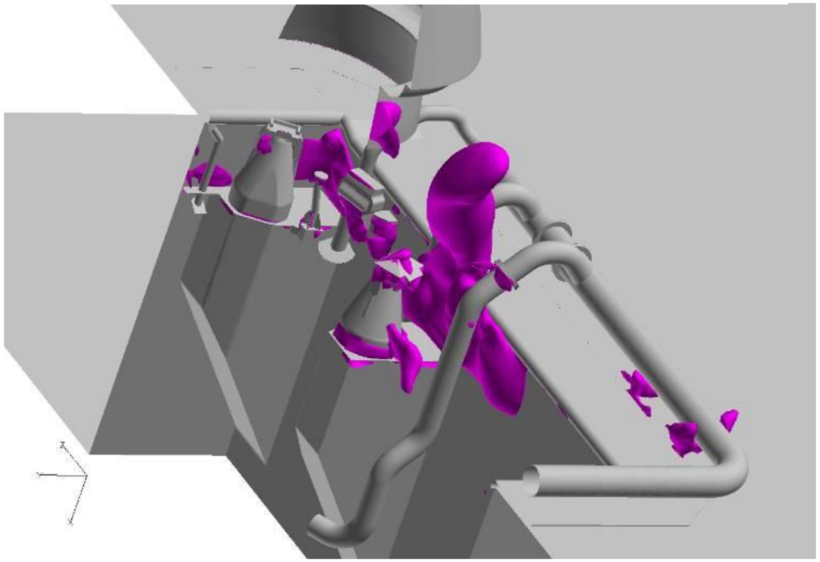
With detailed geometry, the correlation of predicted upward flows to the observed upward flows increased drastically. Significant insight into the risk posture was obtained with the analysis and use of the Loci/Chem flowfields. However, significant uncertainties remained.

- While the CFD accurately predicted the existence of upward flow, the strength and magnitude of the upward flow did not correlate to an acceptable degree

Plume Effluent isosurface



Upward flows contained inside purple isosurfaces

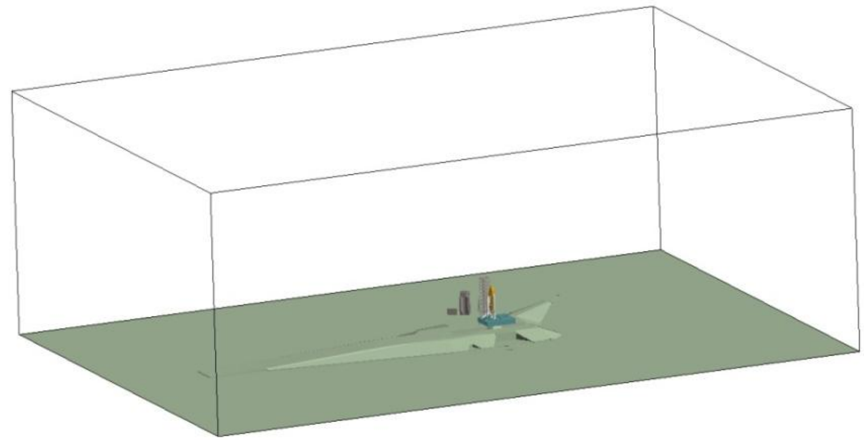


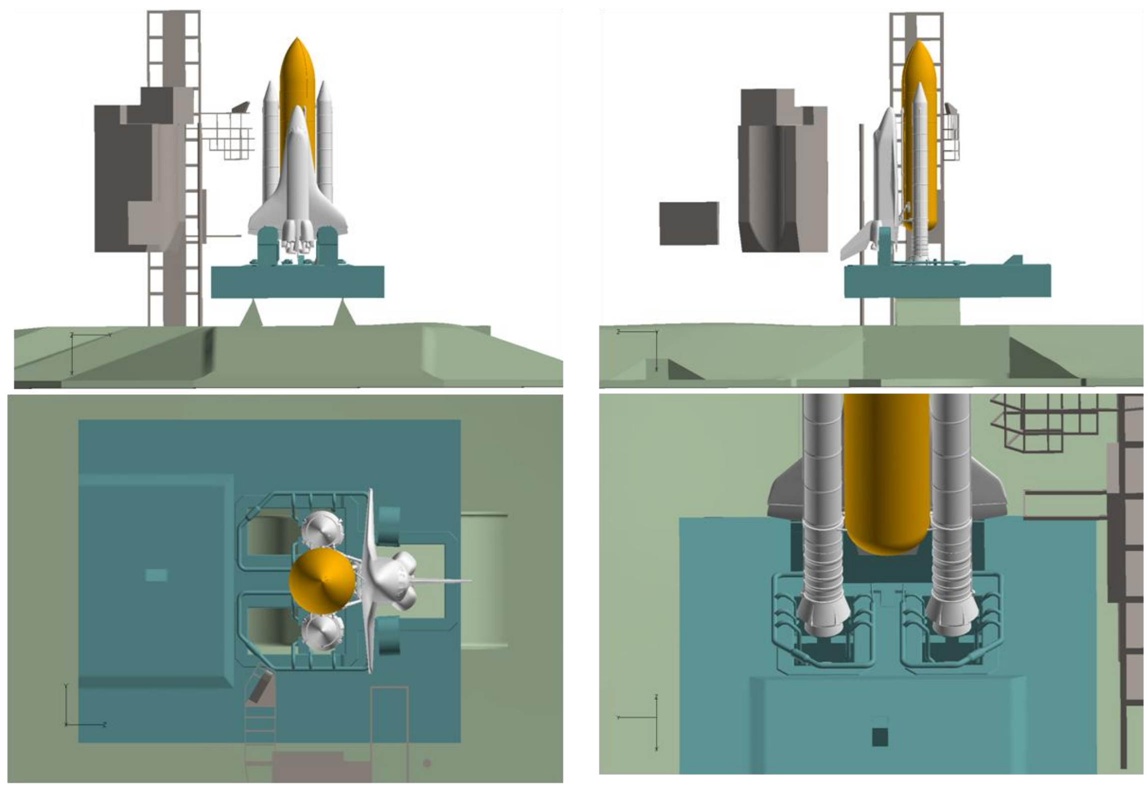
# Large Domain, High Fidelity Geometry

**<u>2007-2009</u>**

The high fidelity geometry approach was extended to the entire Launch Pad.

- 120 to 310 Million Cells
- Three day turn-around
- ~50 steady simulations

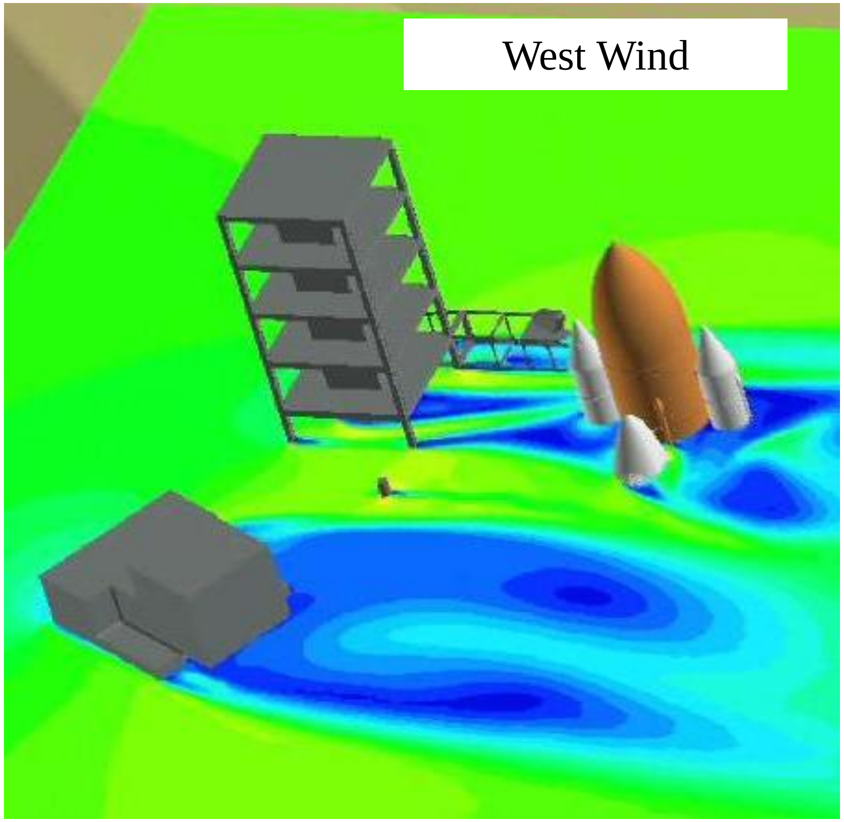


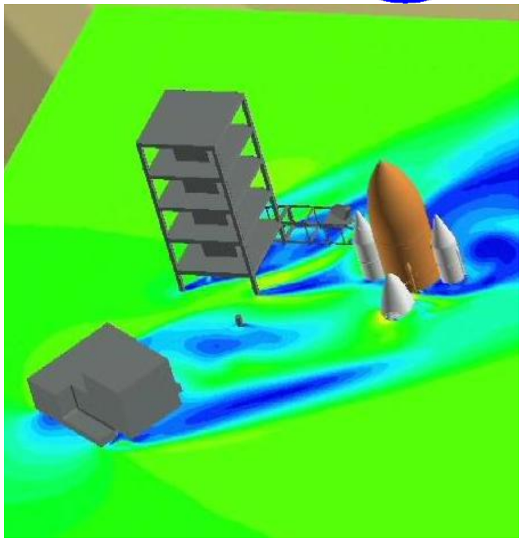


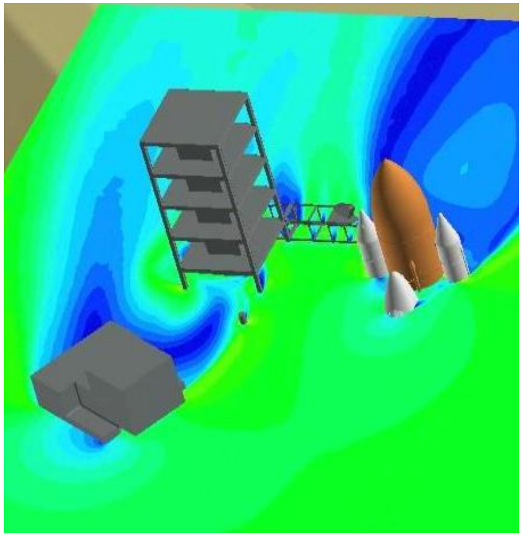
Continued AFLR development was instrumental in the construction of this model.

# Wind-Structure Interactions Resolved

The large-domain, high-fidelity geometry simulations allowed for the completion of the risk assessment due to gravity-and wind-driven debris.

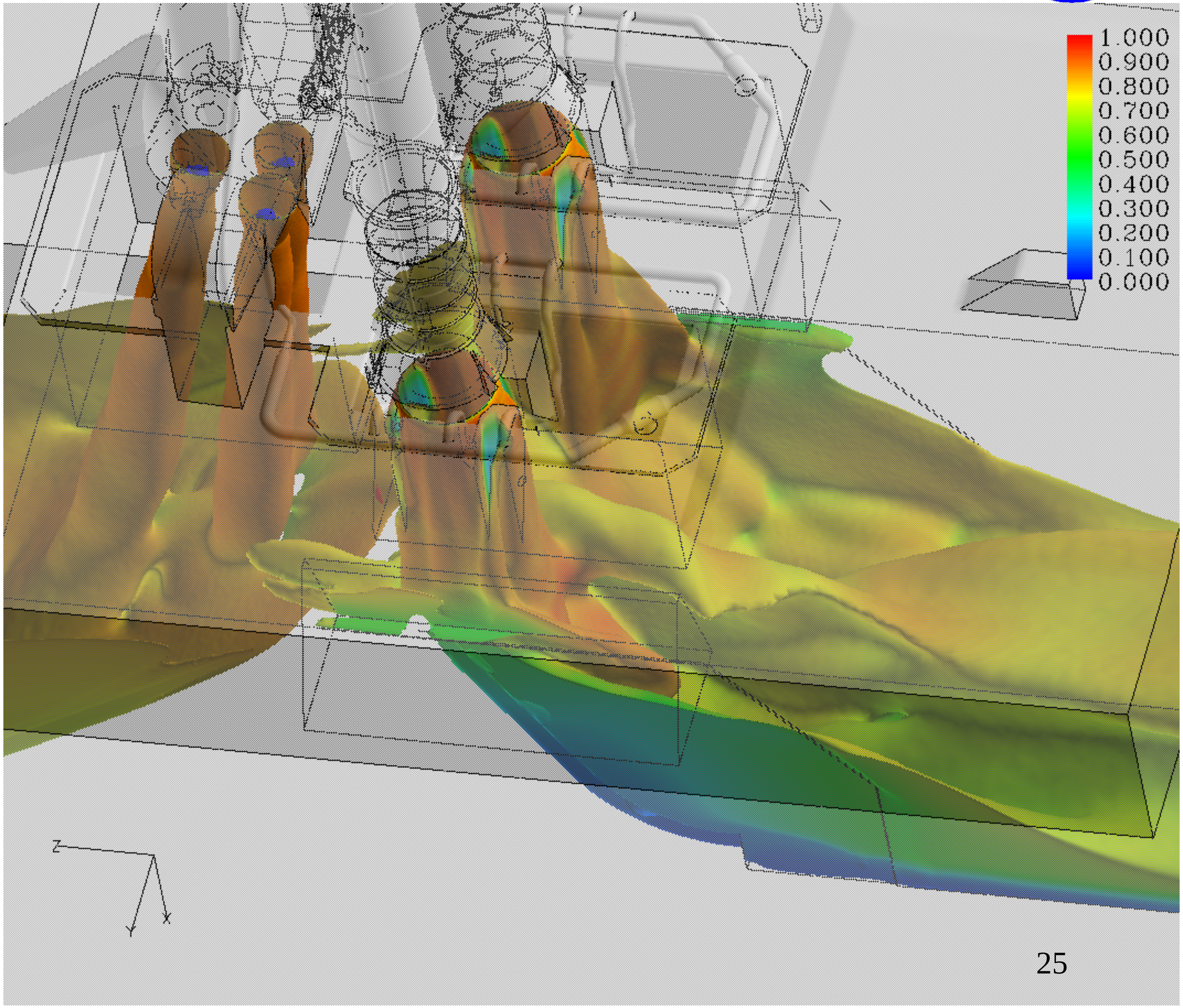




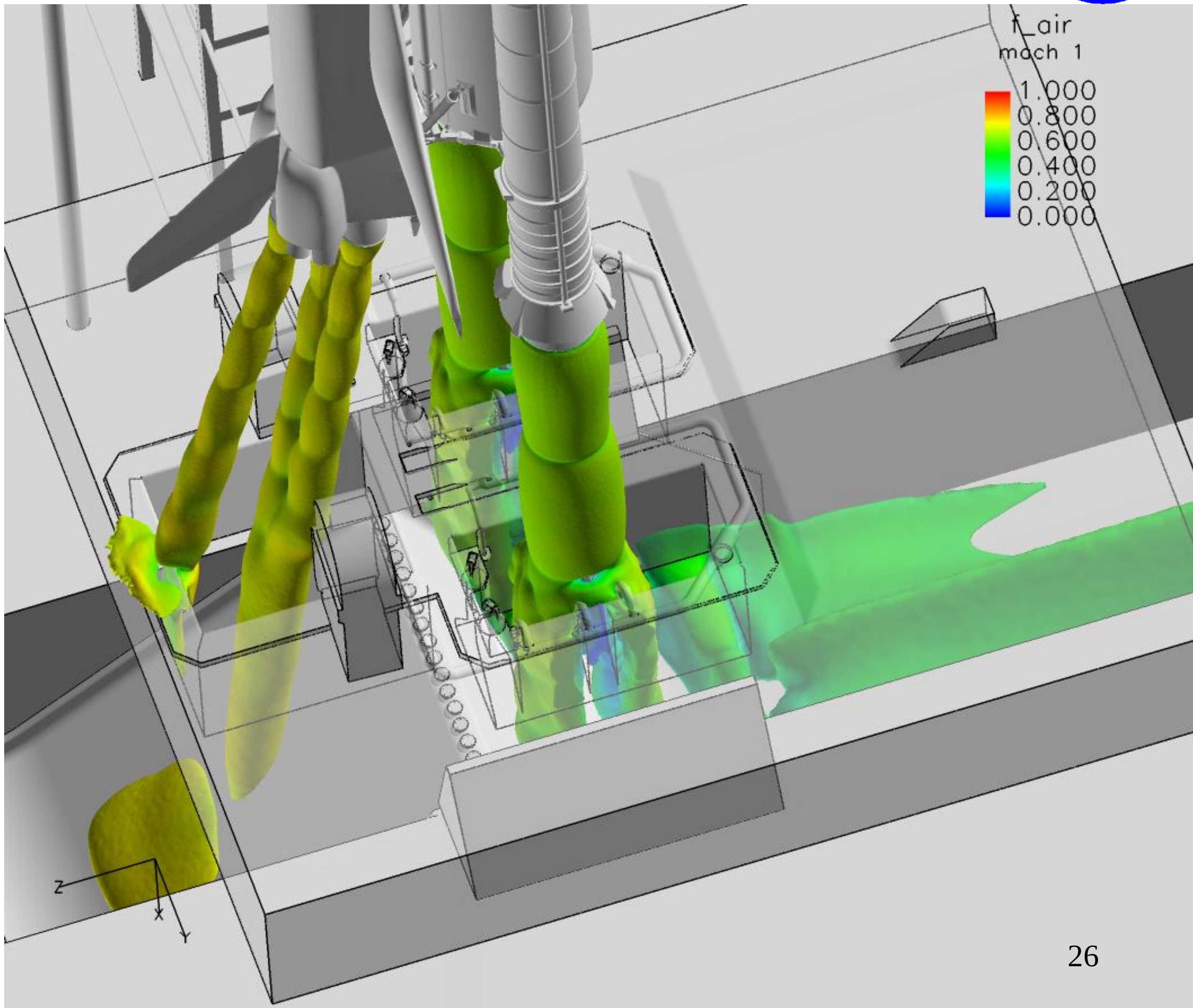
# High Fidelity Resolution of All Plumes

Significant Correlation of predicted plumes and plume-structure interactions was obtained. However, there was significant uncertainty in the risk due to the steady-state reconstruction of the vehicle trajectory.



# Simultaneous Plume-Structure Interactions

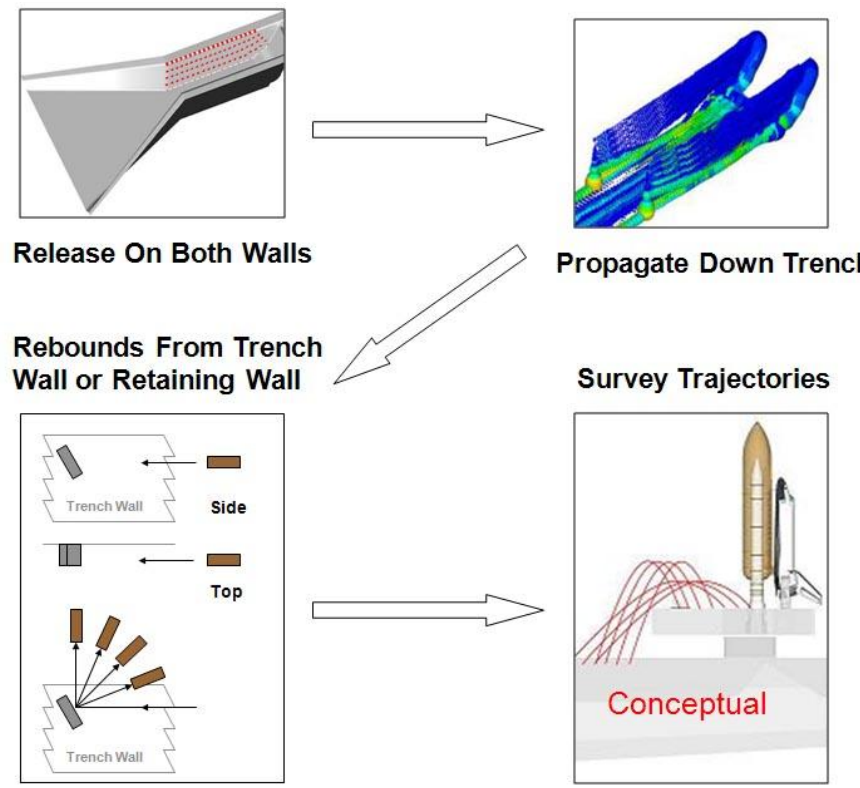
Remaining discrepancies between predicted and observed features attributed to lack of vehicle motion and the assumption of steady state fluid dynamics.



# 2008 STS-124 Brick Loss Event

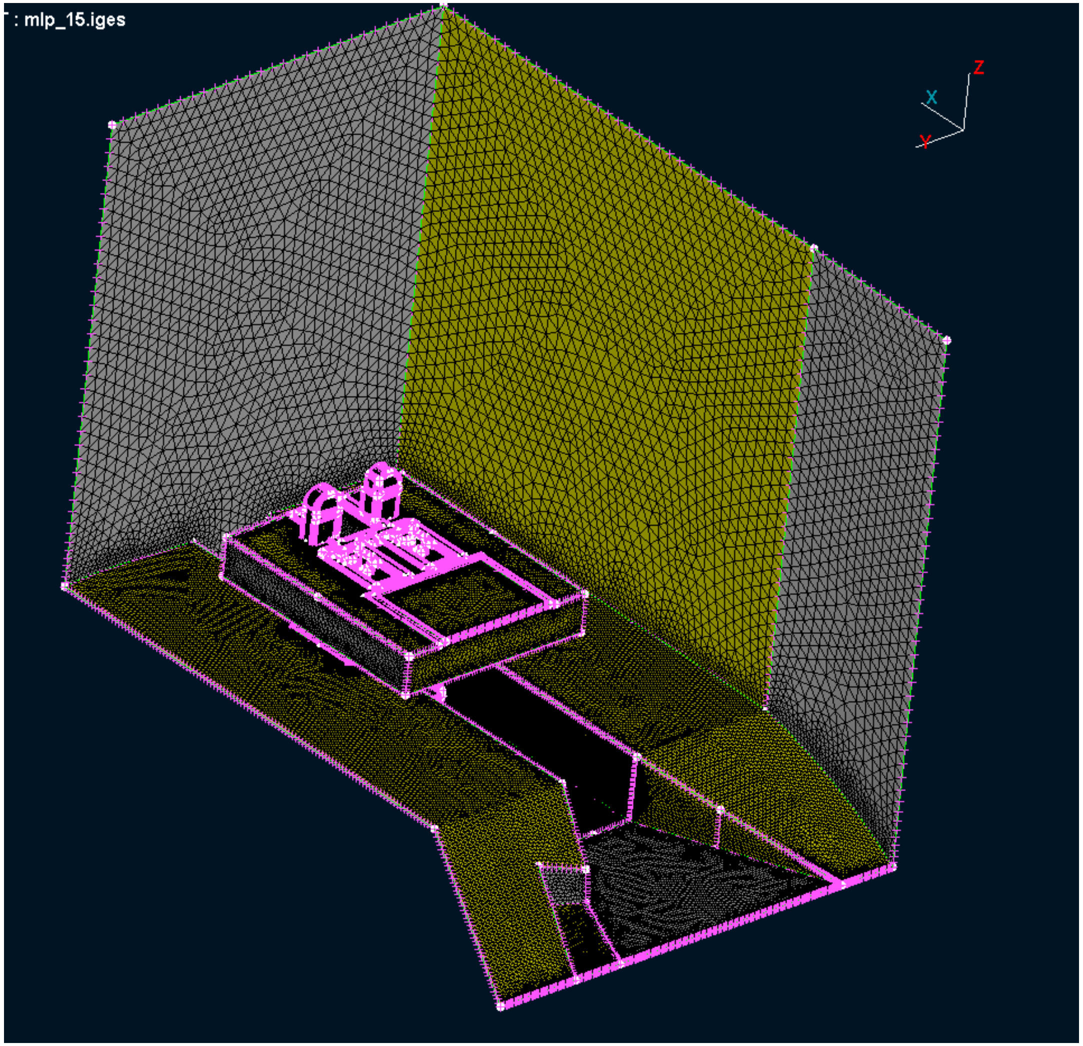
CFD Simulations were integral to the assessment of the risk to the Space Shuttle from Launch Pad Trench Debris if a Trench Failure re-occurred.

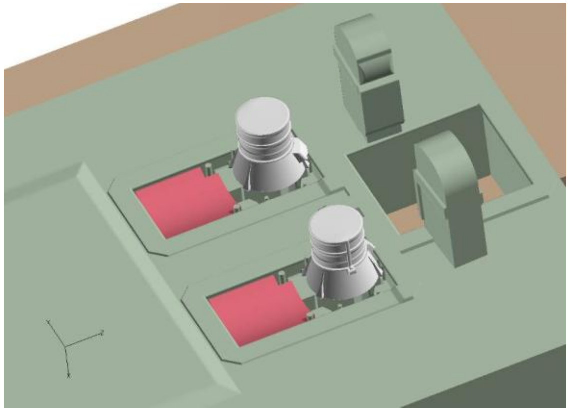
During the Launch of STS-124, thousands of bricks were released and ejected from the SRB-side Flame Trench. Original construction: 1965.

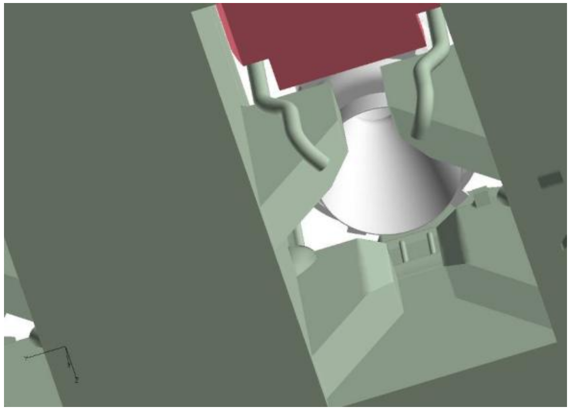


# Static Geometry, Transient Simulation

A small computational domain coupled with high-fidelity geometry(30 Mcells)

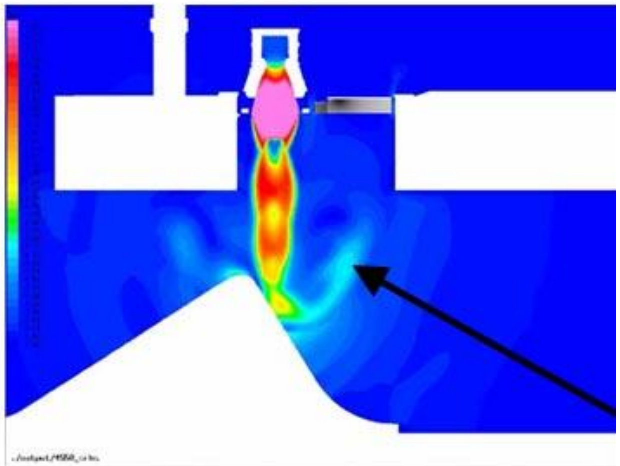


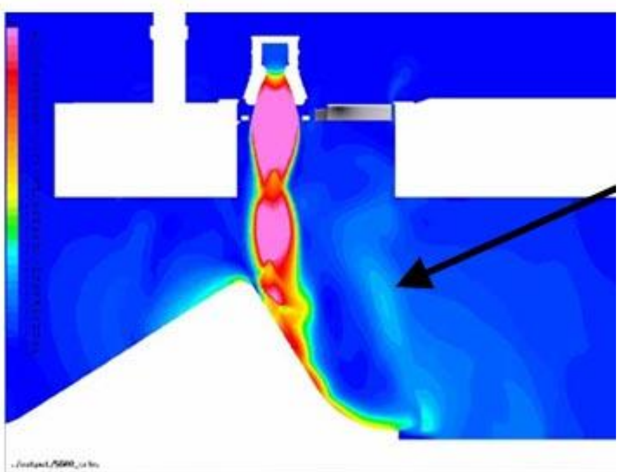



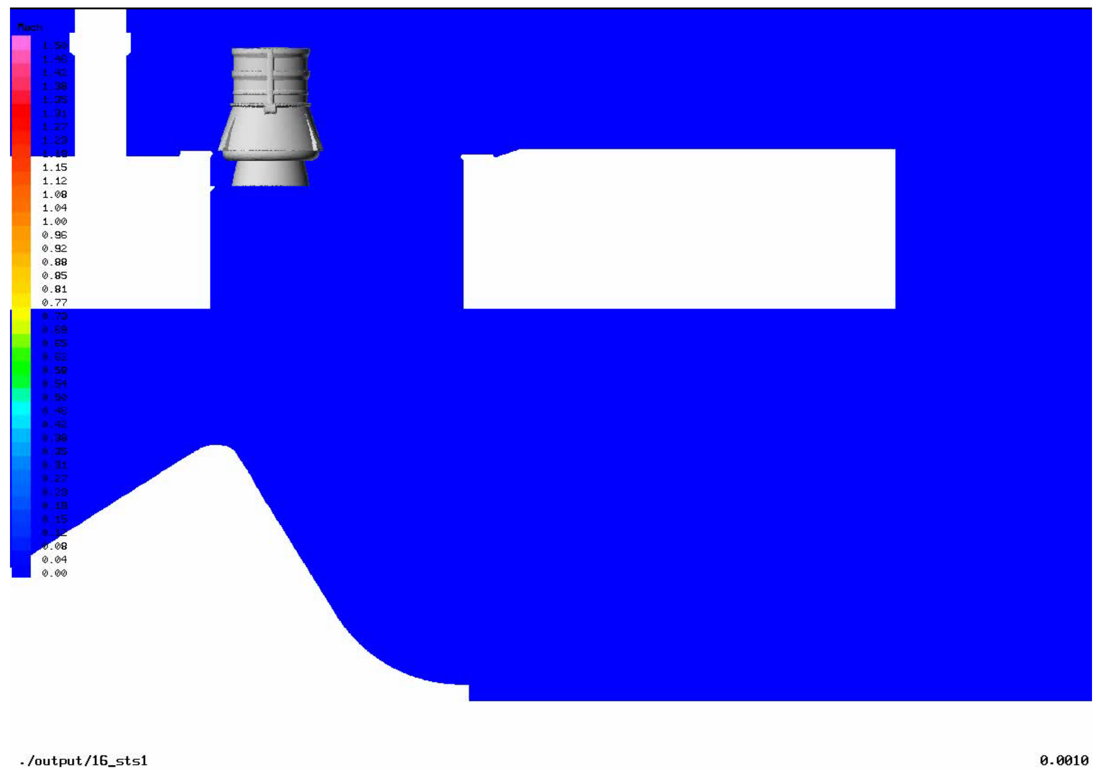


# Shuttle Plume Startup Simulation

The Shuttle SRB Startup Simulation resolved fluid dynamic features that are obscured by the plume during an actual launch



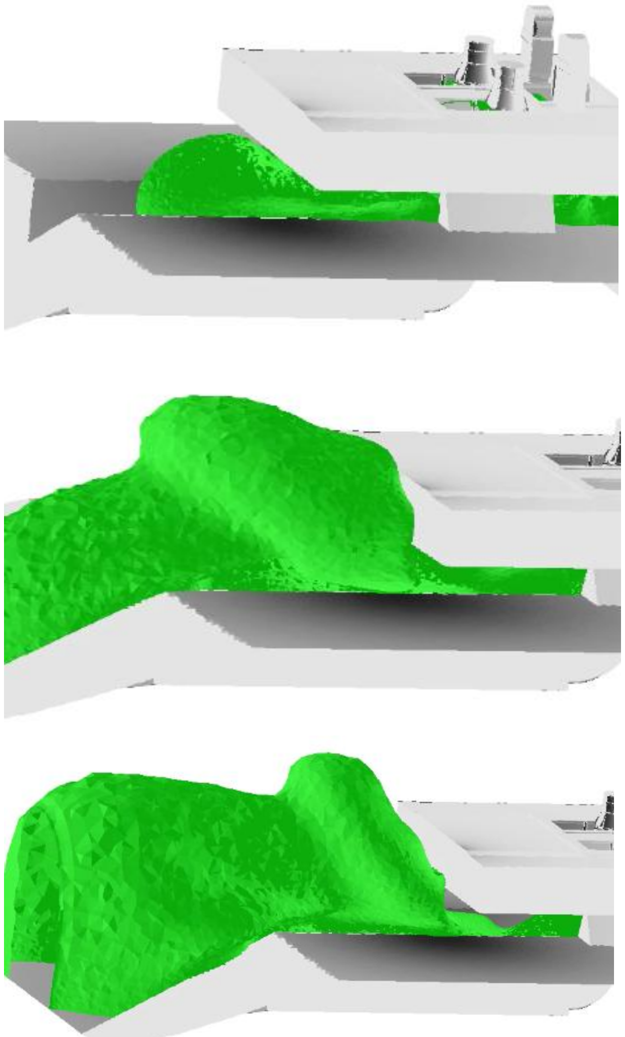


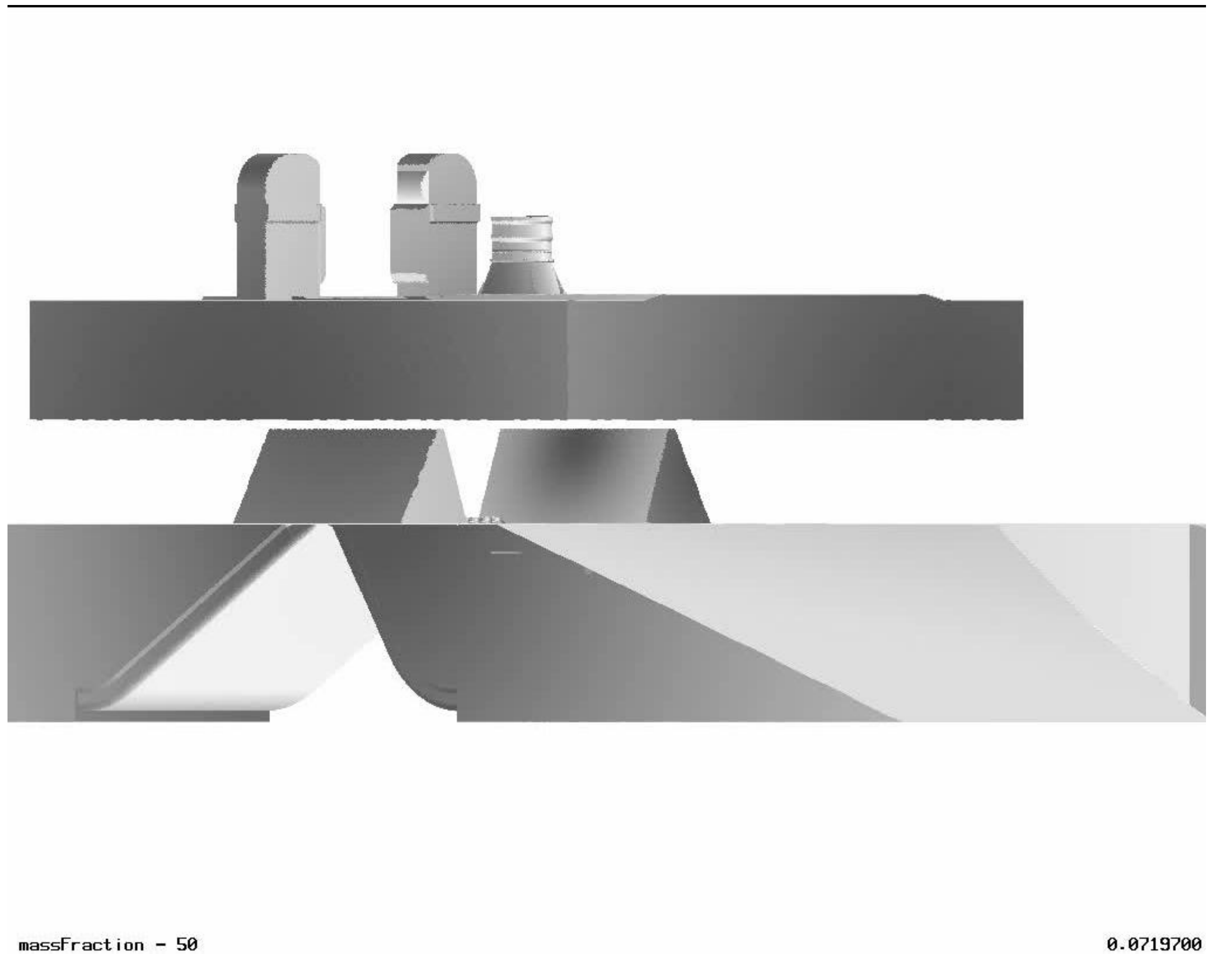



Shuttle-fountain-effect.mpg

# Shuttle Plume Startup Simulation

Observable features were replicated to a good degree.

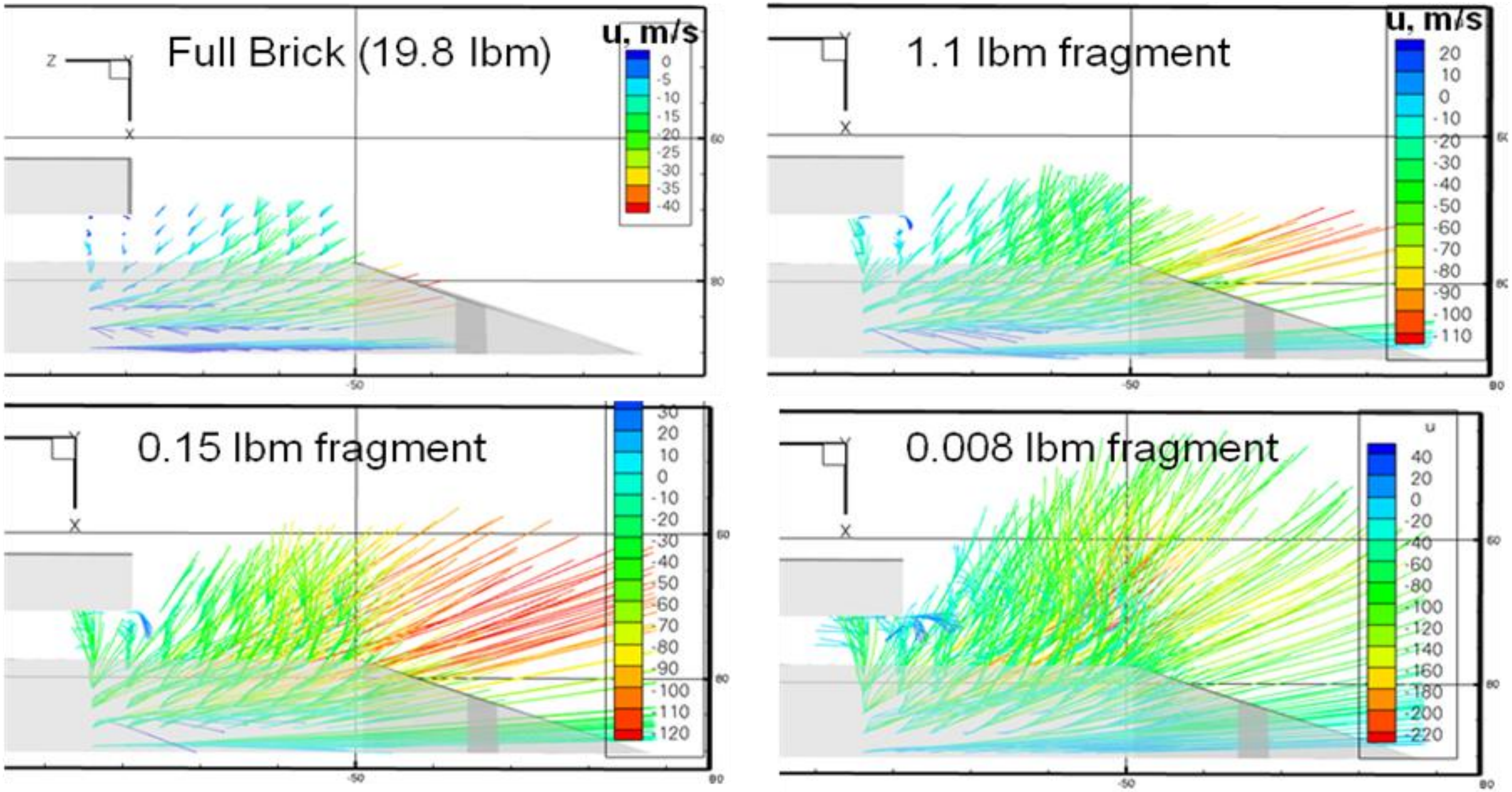





Shuttle-mushroom-effect.mpg

# Debris Transport Analysis using CFD

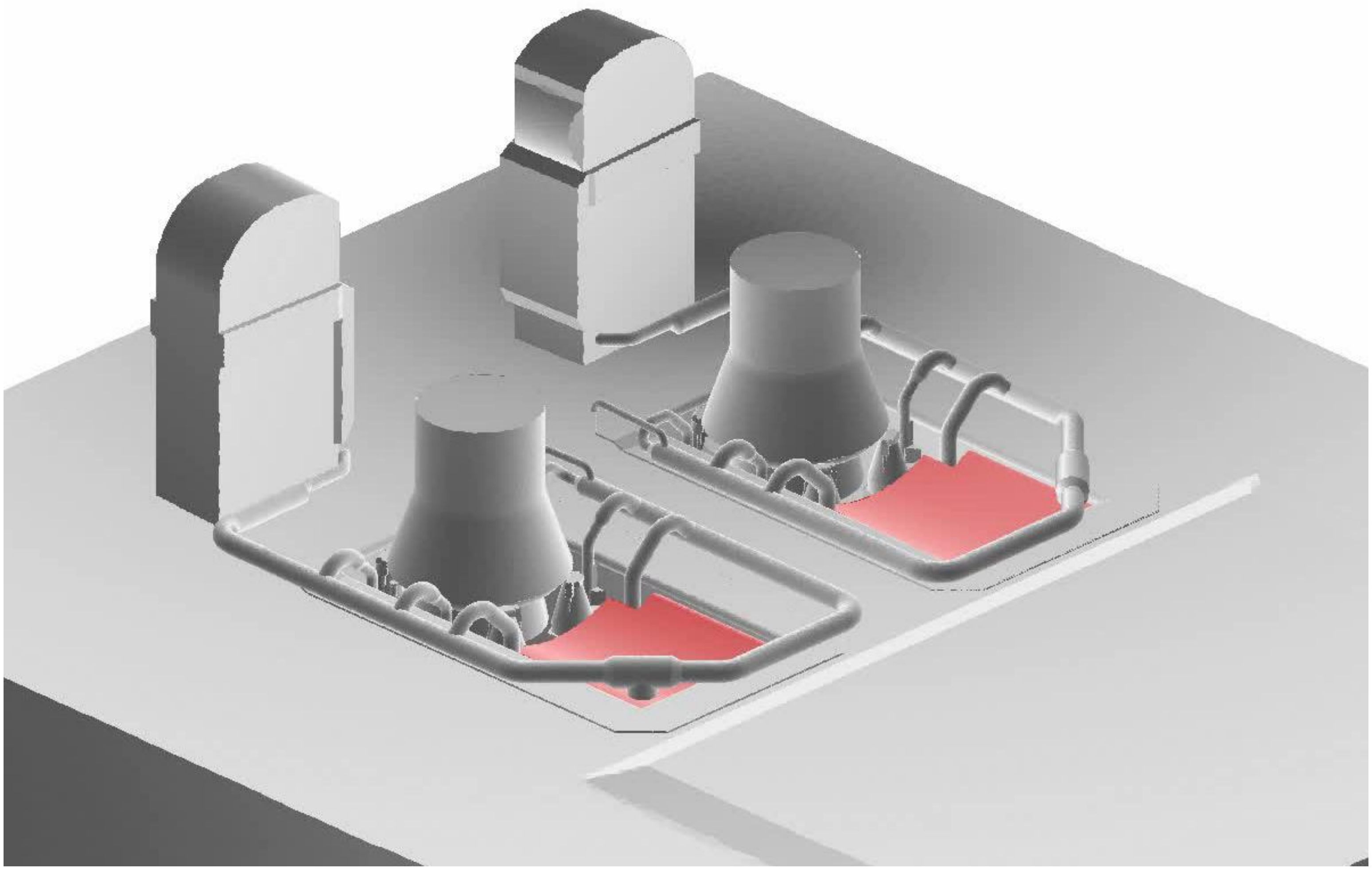
The CFD Simulation results allowed the assessment of the transport of Trench debris as a risk to the Space Shuttle. The program-focused analysis began on 1 June 2008 and was complete by 29 June 2008.



# Shuttle Moving Body Simulation

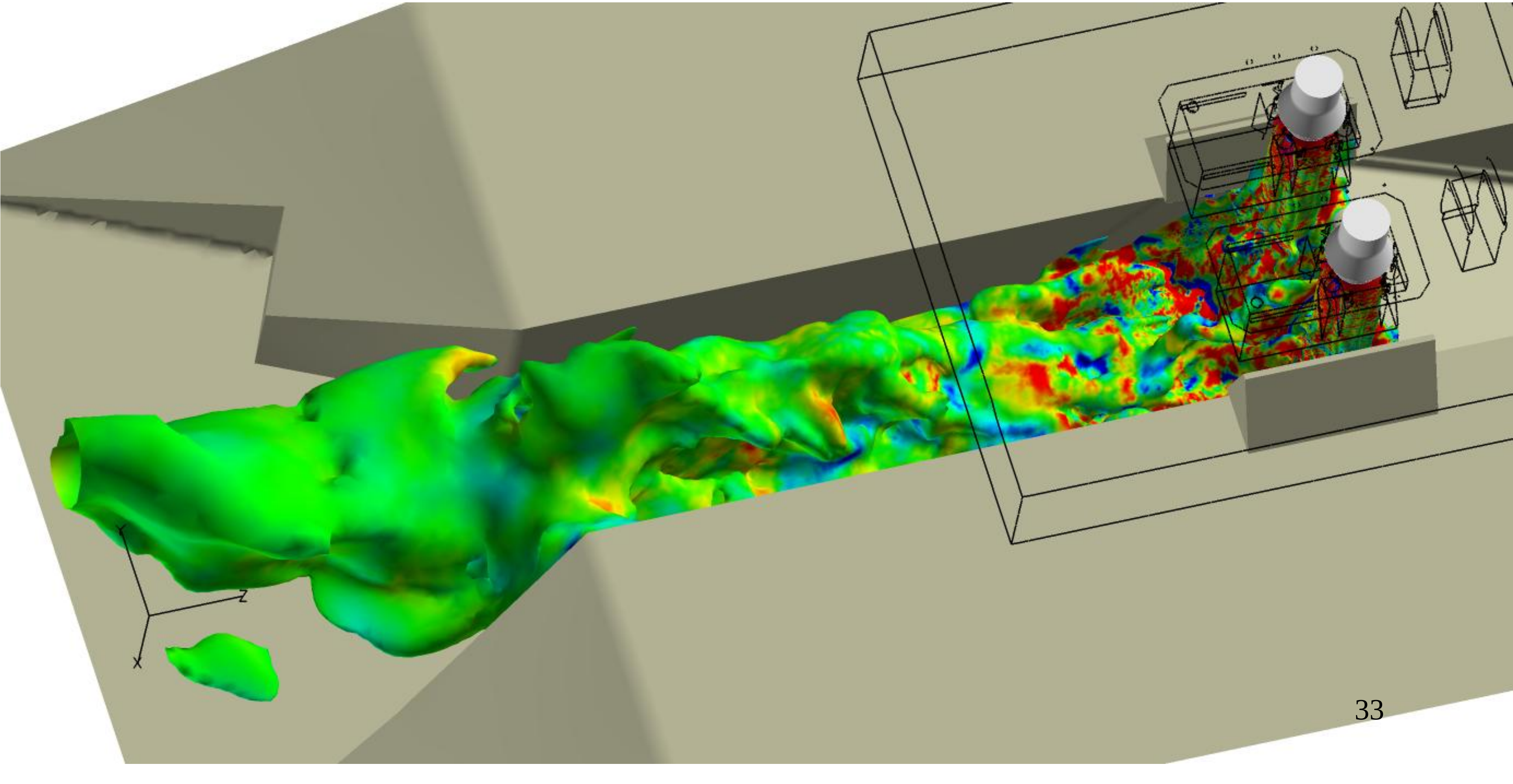
**<u>2008-2009:</u>** Overset Mesh treatment in Loci/Chem coupled with Hybrid RANS-LES allows accurate resolution of upward flows. 50 to 90 Million Cells.

Shuttle-smoke-movingbody.mpg

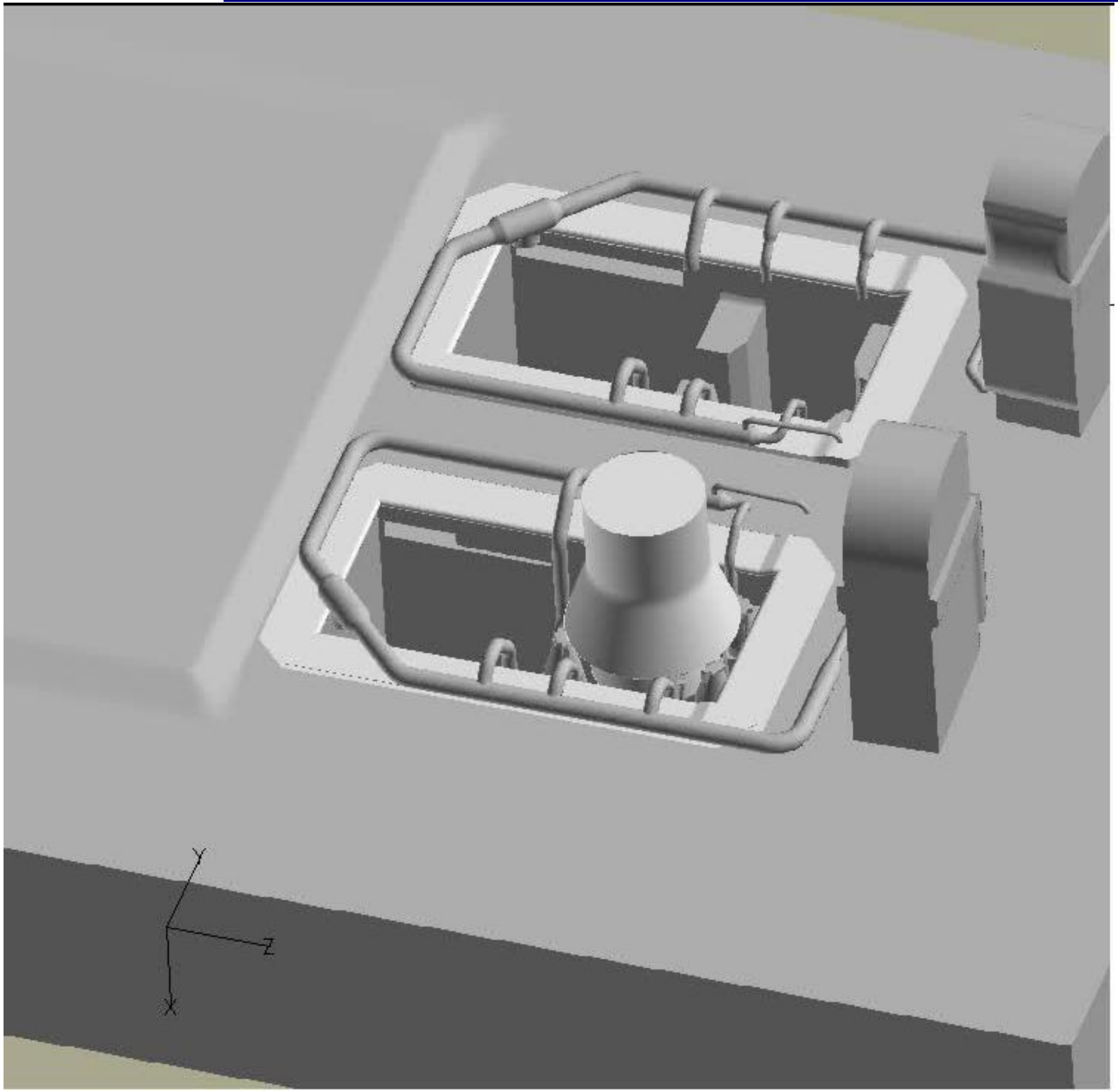


# Shuttle Moving Body Simulation

Shuttle SRB Moving Body with Hybrid RANS/LES treatment also results in more physically realistic trench flow dynamics.



# Ares I-X Moving Body Simulation




**2009-2010**

Overset Mesh treatment in Loci/Chem coupled with Hybrid RANS-LES treatment allows accurate resolution of plume Impingement flows and understanding of Launch Pad consequences. Two week turn-around time.

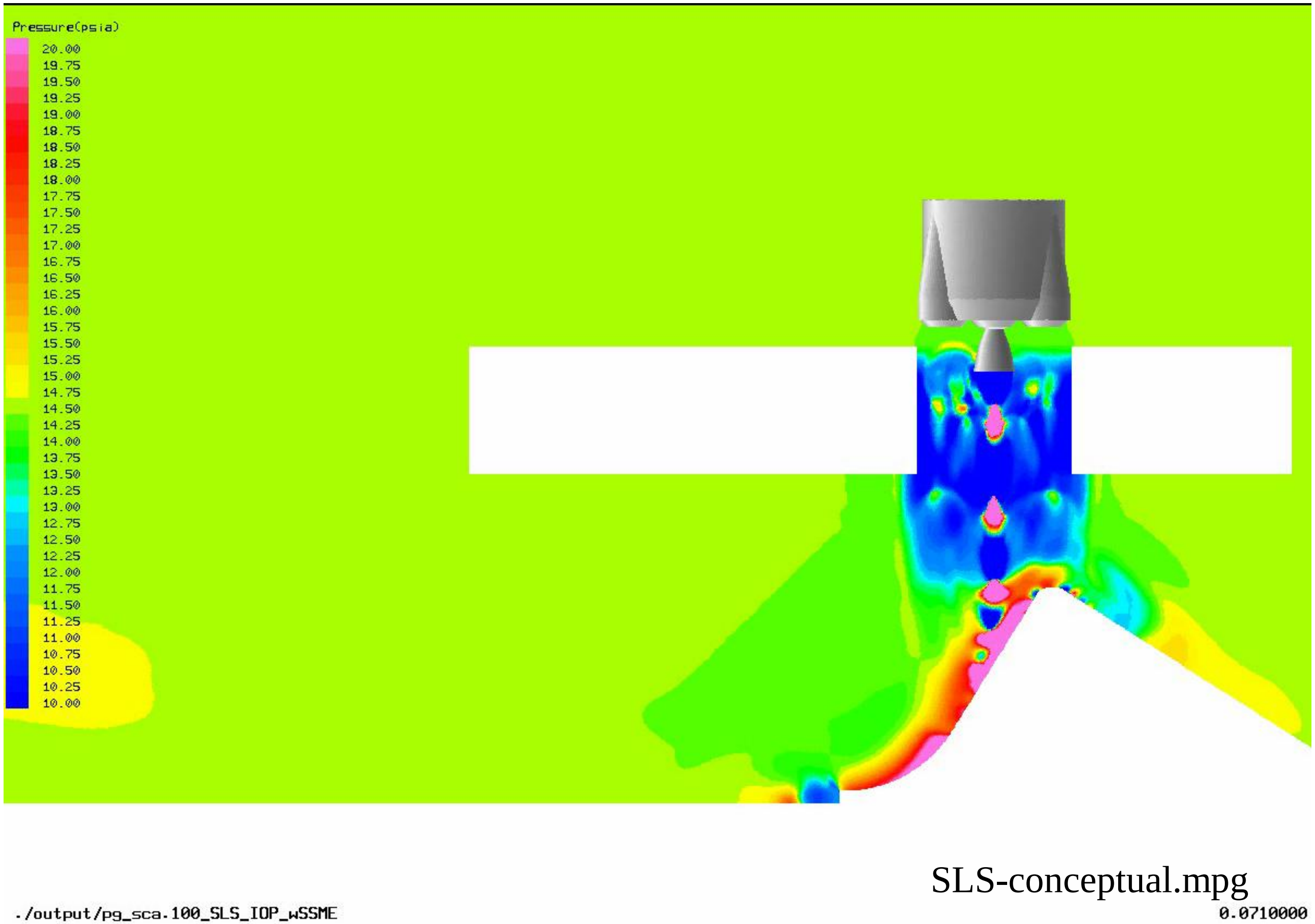
Ares-IX-movingbody-hot.mpg

# Ares I-X Launch Pad Effects

Very good correlation between CFD –predicted impingement location and strength with Launch Pad Evidence

# “What-if?” SLS Investigations

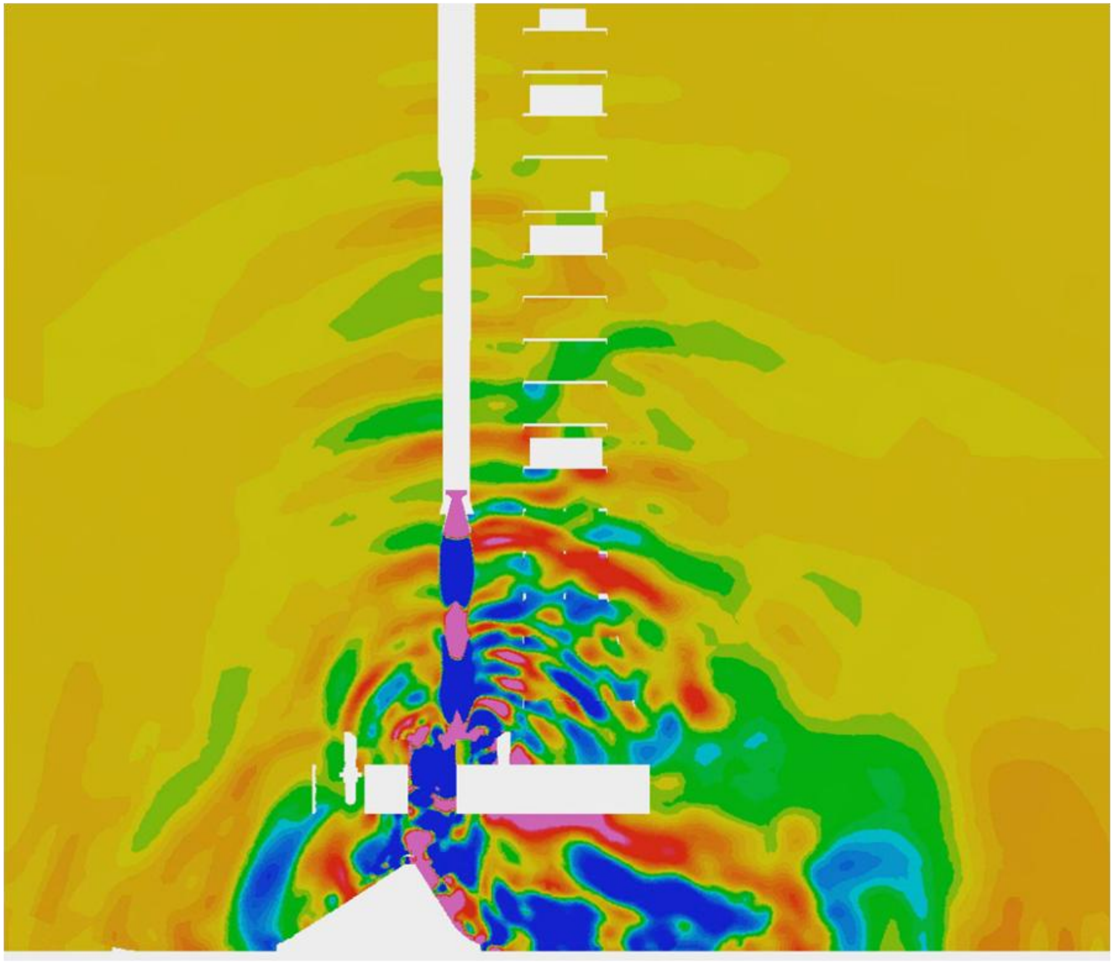
**Now:** The unsteady, moving body capability within Loci/Chem is being used to evaluate Launch Pad concept consequences for SLS architectures

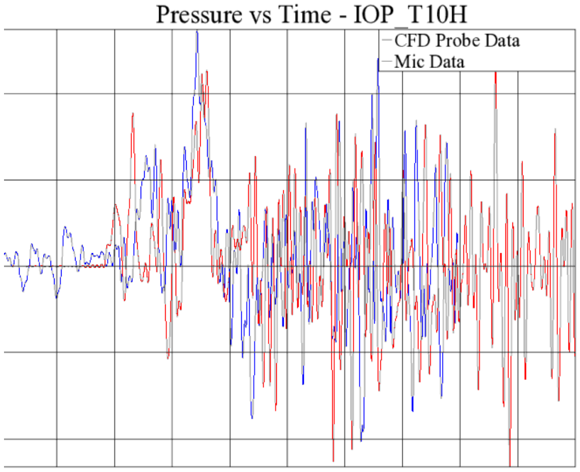


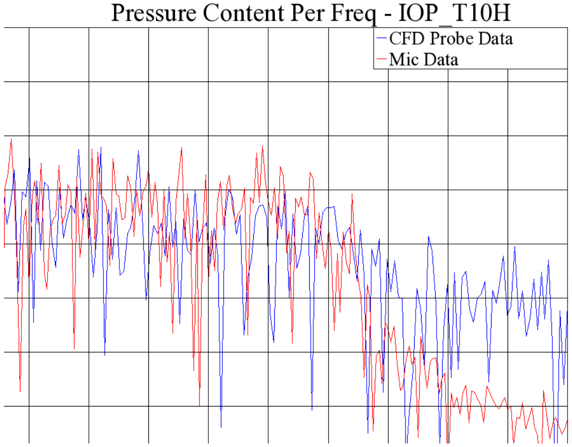

SLS-conceptual.mpg

# Prediction of Launch Pad Acoustics

**Now:** Initial Simulations indicate that Loci/CHEM can predict a useful fraction of Launch Pad Acoustics





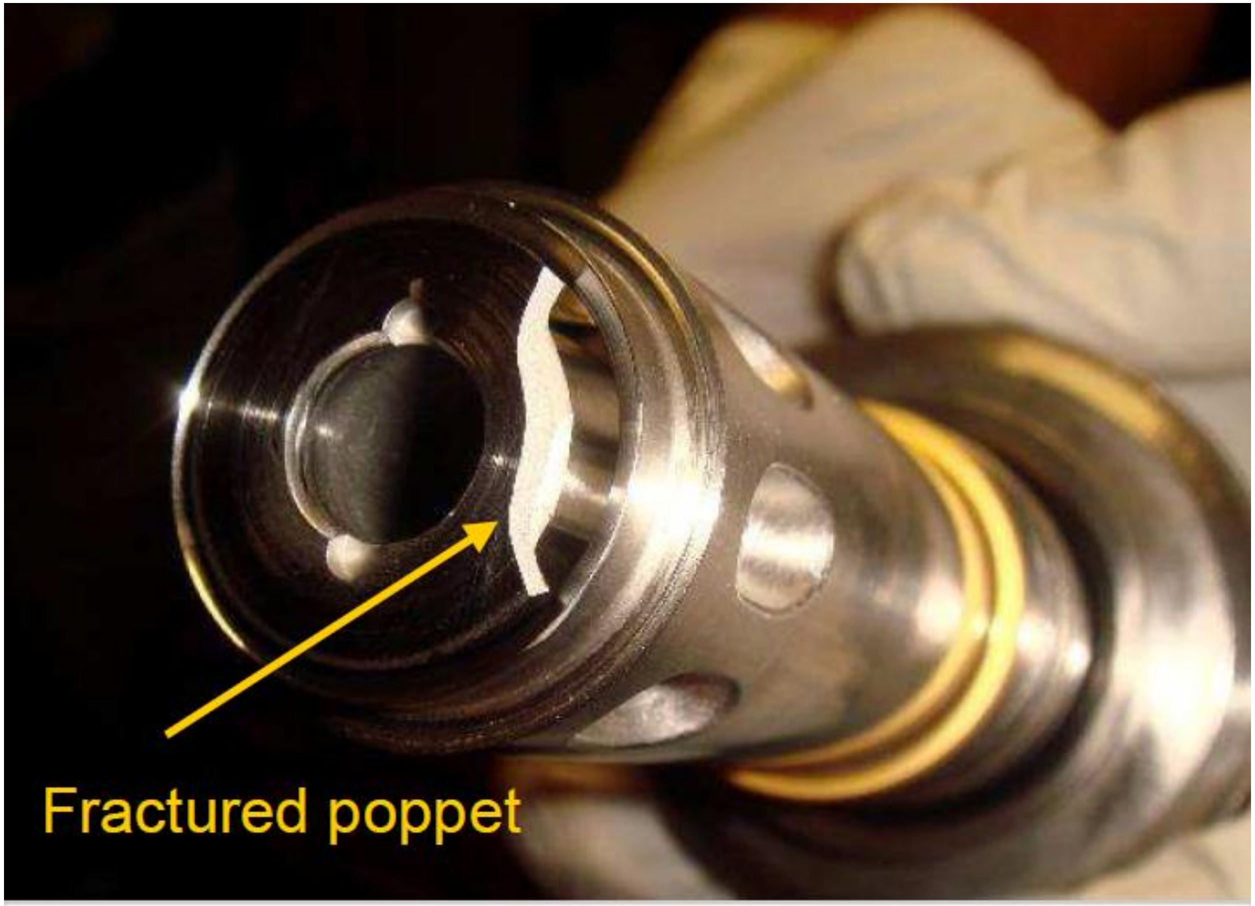

subscale-acoustics-cfd.mpg

## **Examples of the Expanding Role of CFD at NASA/MSFC:**

*CFD Simulations supporting Valve Problem Resolution and Valve Design*

# STS-126 In-flight Valve Failure

**<u>2009-2010:</u>** The Fuel Flow Control Valves (FCV) supply high-pressure gas from the SSME engines to pressurize the fuel tank during flight.  Three two-state valves are used to tailor the fuel tank pressure.

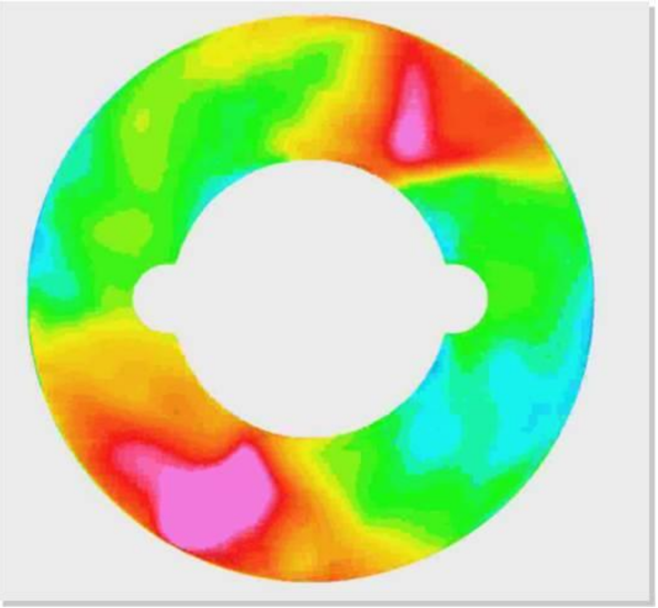


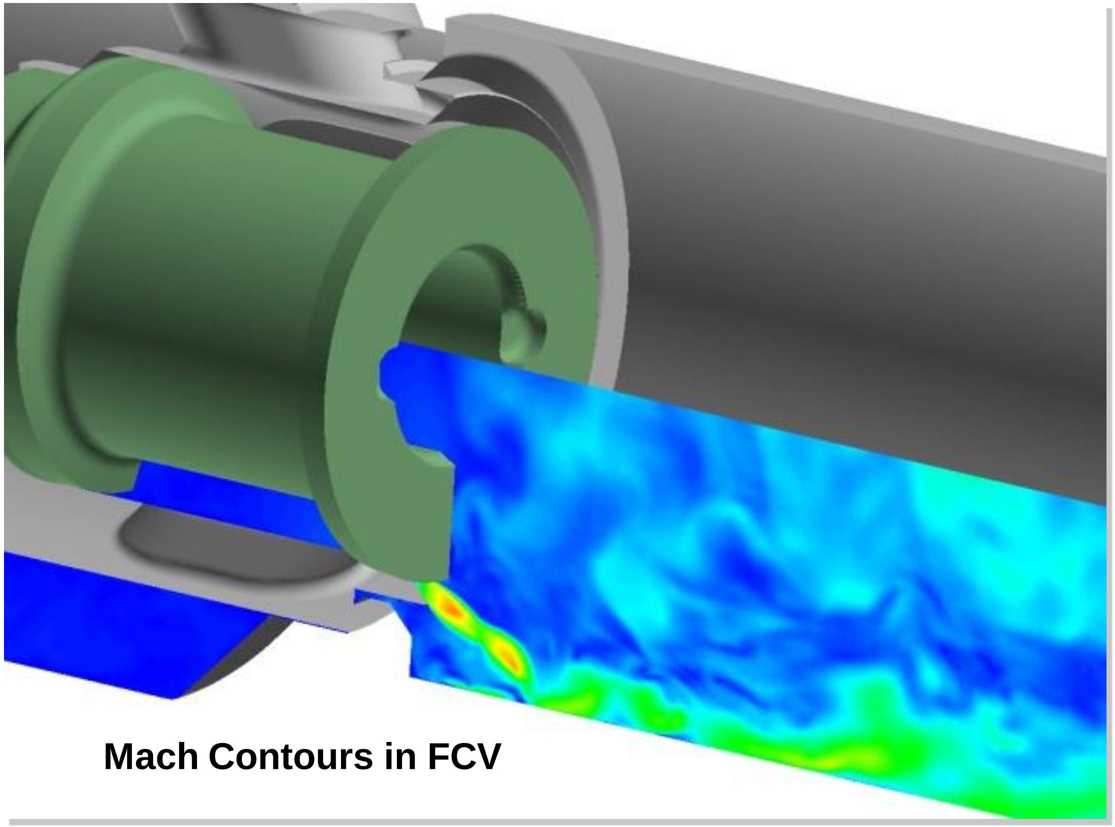

STS-126 Valve Poppet Following Removal

# Unsteady Environments in Static Valves

Pressure on Poppet Face

Loci/Chem was used to model the unsteady loads on the poppet face using Hybrid RANS-LES capability. The supersonic annular plume creates unsteady loads on the poppet face which were never envisoned during the design and redesign(s) of the FCV. (Root cause was evident to CFD analysts after first 24 hours on cpu)





**Mach Contours in FCV**

**CFD for SSME Flow Control Valve**

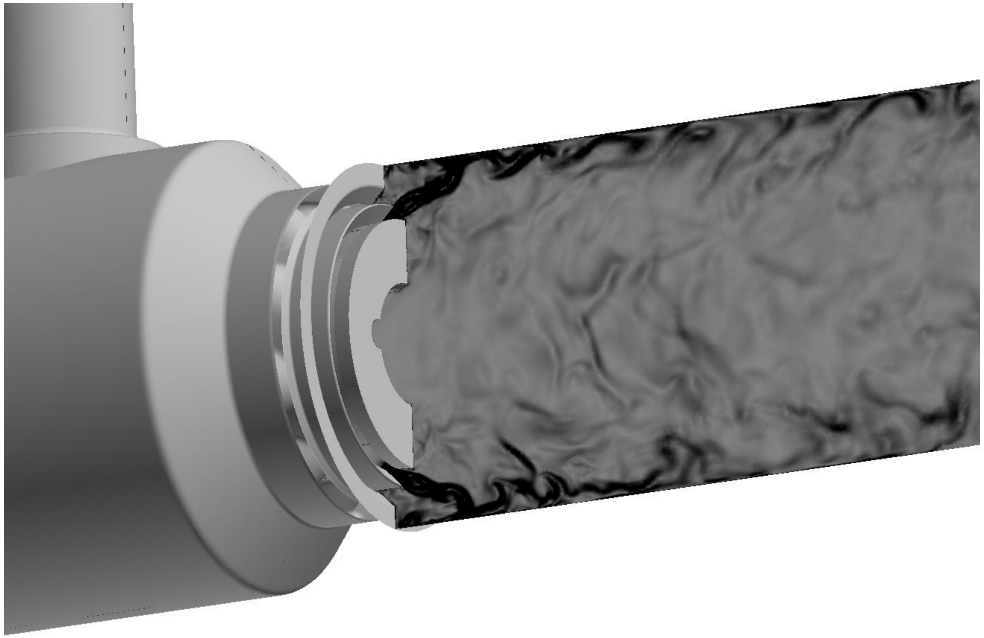
- Unsteady simulation of supersonic impingement on outflow pipe wall
- 15-plus unsteady 3D simulations with 60 Million cells
- 800 cpus using Pleiades at NASA/AMES, turn-around in 4 days

FCV-poppetface.mpg

# Unsteady Environments in Static Valves

**Unsteady CFD Simulation**

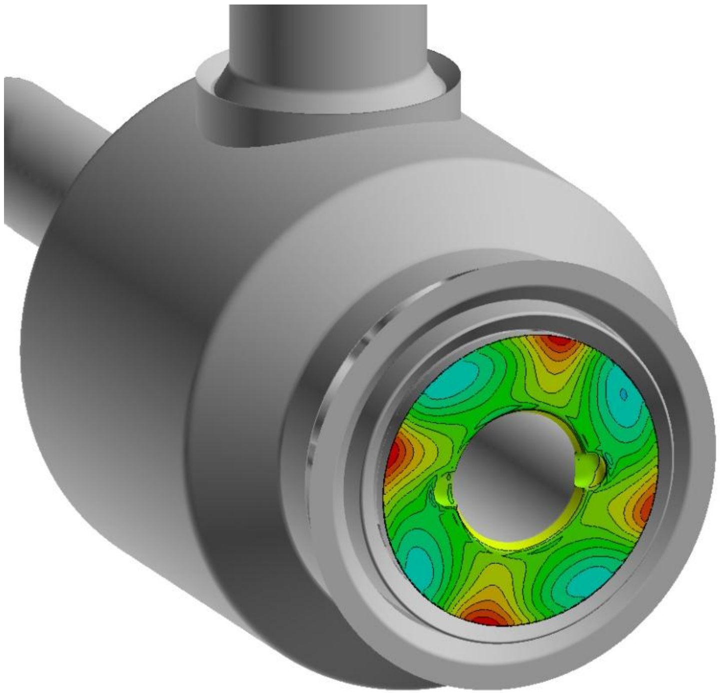
**Synthetic Schlierin View**



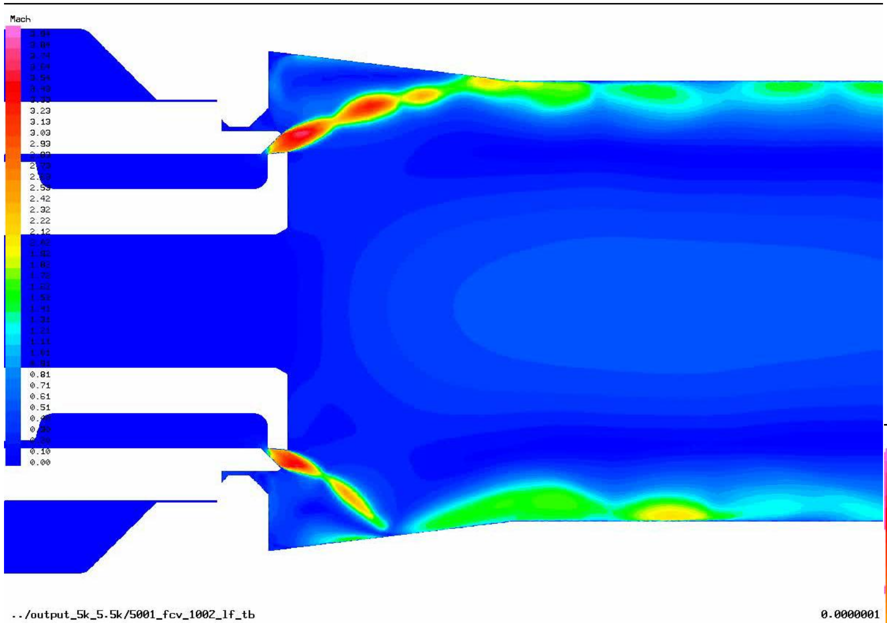
Supersonic gaseous H2 jet impinges on Outlet Pipe wall and pressure waves reverberate throughout downstream system

**Poppet Face Pressure**

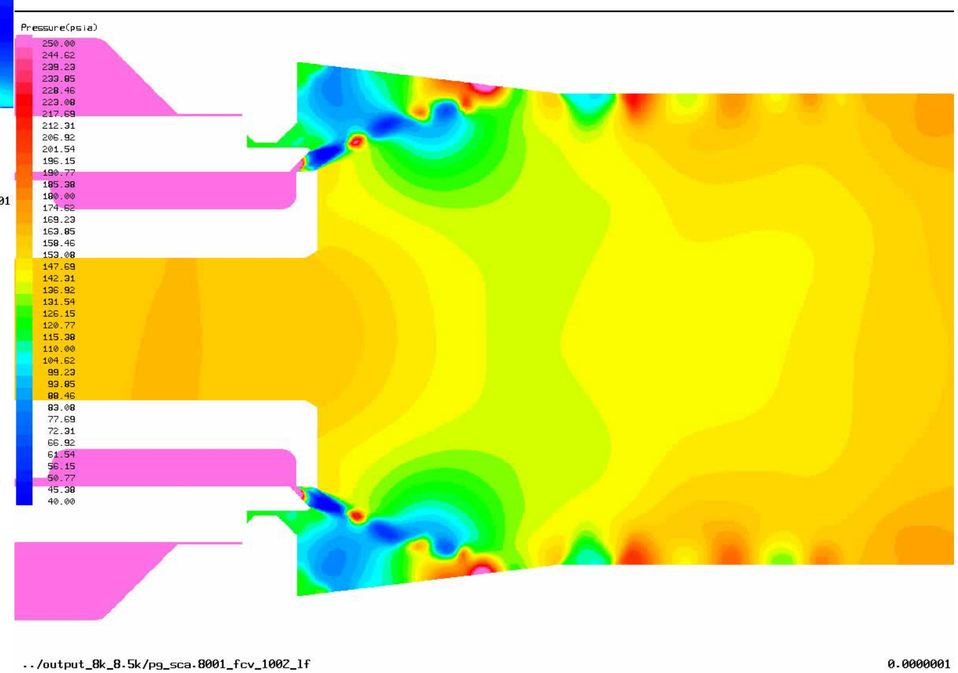
**4 Nodal Diameter Loading Pattern**



# Unsteady Environments in Static Valves



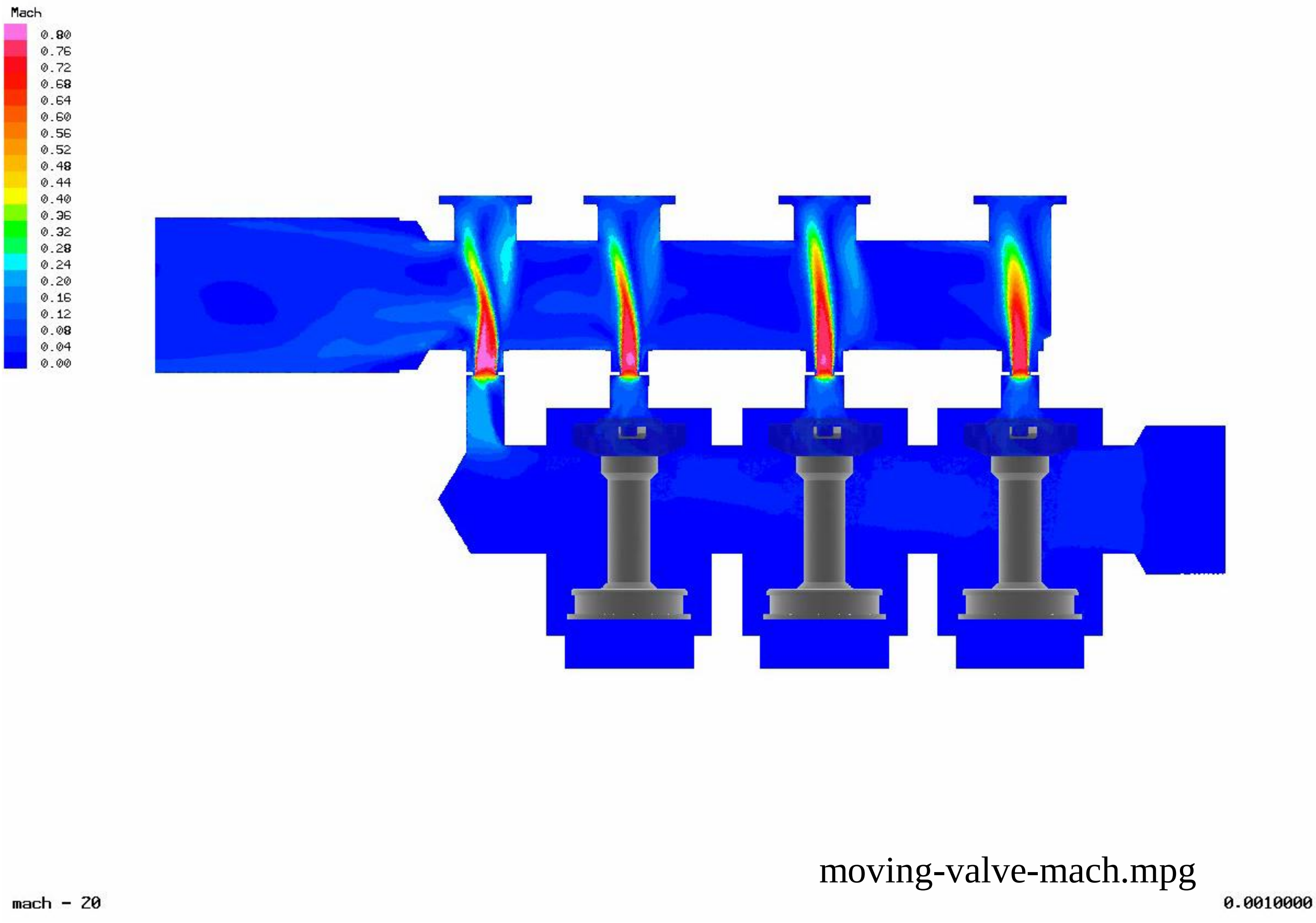

FCV-mach-plane.mpg




FCV-pressure-plane.mpg

# Unsteady Simulation of Valve Transients

**Now:** Applying the Overset capability of Loci/Chem to simulate valve transient operation. 120 Mcells, 5 days turn-around, 1600 cpus



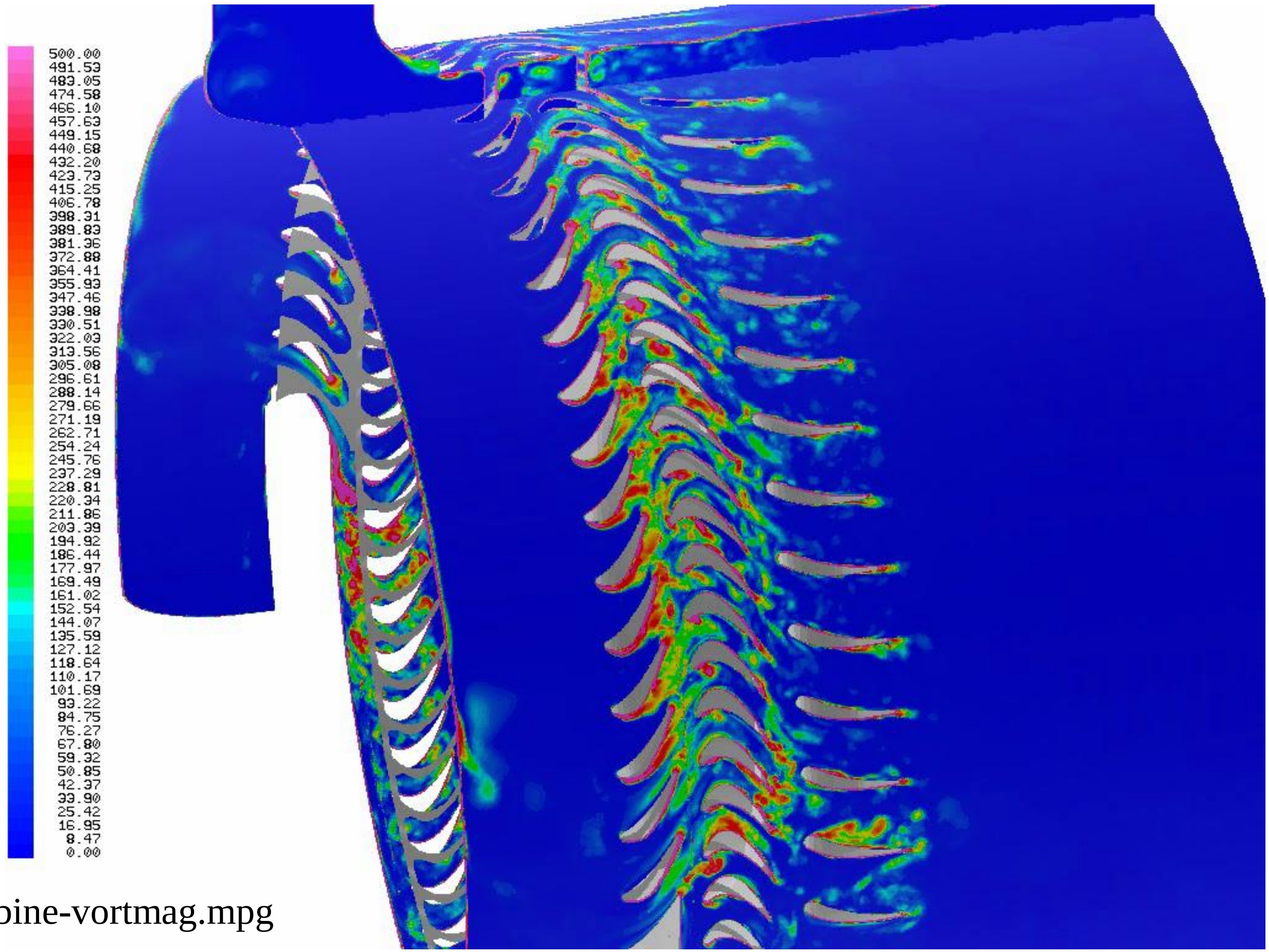

moving-valve-mach.mpg

## **Examples of the Expanding Role of CFD at NASA/MSFC:**

*CFD Simulations supporting Turbomachinery*

# Unsteady Simulations of Turbines

**<u>2010:</u>** Application of the Overset capability of Loci/Chem. One week turn-around.




turbine-vortmag.mpg

# Multi-Stage Pump Simulations

**<u>Now:</u>** After porting the Overset capability to Loci/Stream, the unsteady simulations of pumps is now possible on a large scale. 150-250 Million Cells, thousands of cpus.


Pump-multistage.mpg

# Parting Thoughts

NASA/MSFC has benefited from the long-term relationship with Mississippi State University. In addition, MSFC has benefited from the willingness of MSU personnel to collaborate with entities outside of the University. (Streamline Numerics, Tetra Research, CFD Research Corporation, ATK, Virginia Tech University, Purdue University, University of Florida, Sandia National Labs and many others.)

The development and application of new capabilities within both Loci CFD programs continues to be become more efficient.

- Benefits of the long term collaboration continue to accelerate year after year.

NASA/MSFC's CFD development path includes Mississippi State University for the continued advancement of:

- Loci
- Loci/Chem
- Loci/Stream
- AFLR

Backup