



High-Order Methods in NASA's Next Generation of Computational Fluid Dynamics Tools

**David M. Schuster, PhD.
NASA Engineering and Safety Center
NASA Langley Research Center
Hampton, Virginia
USA**

**European Workshop on High Order Nonlinear Numerical
Methods for Evolutionary PDEs: Theory and Applications
April 11-15, 2011
University of Trento, Italy**



Outline

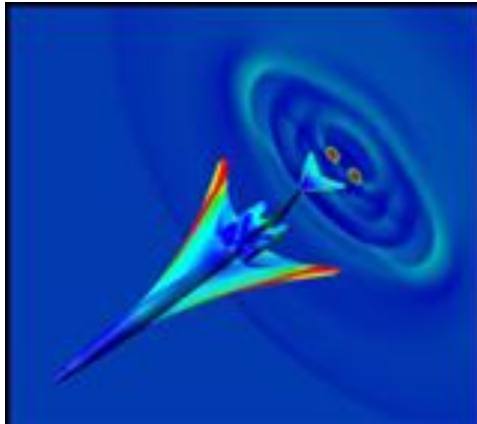
- NASA Missions and their requirements for Computational Fluid Dynamics (CFD) methodology.
- CFD state-of-the-art employed in NASA applications.
- Examples of applications challenging NASA's CFD capability.
- Desired characteristics of future CFD solvers and the role of high-order methods in their development.
- A vision for future production CFD methods.



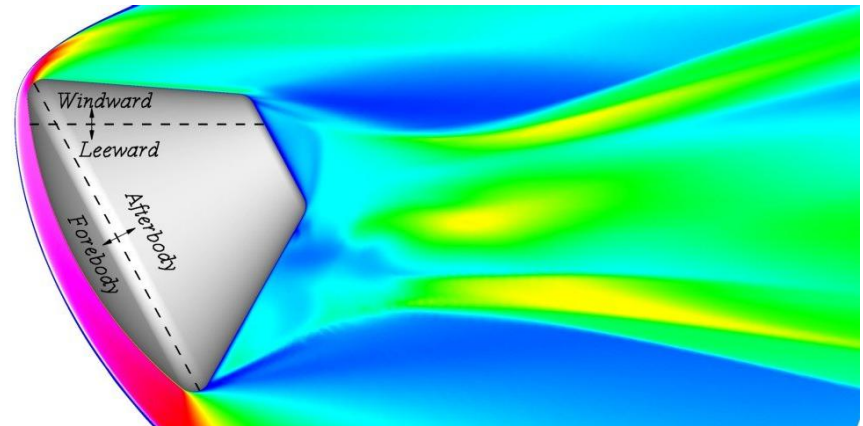
NASA Missions

NASA is presently organized into four primary Mission Directorates

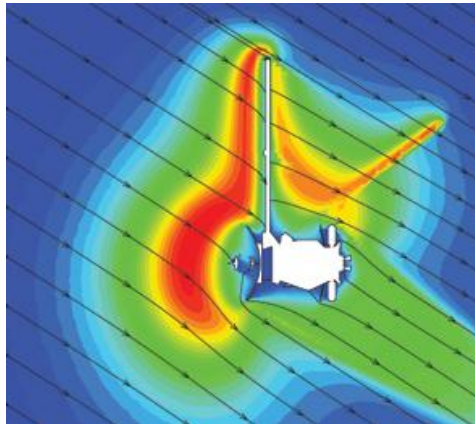
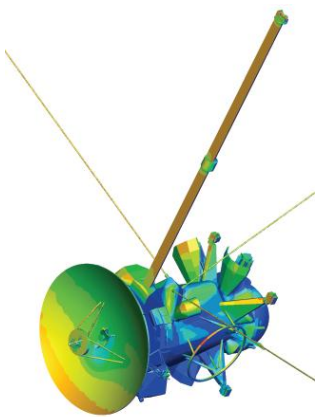
Aeronautics Research



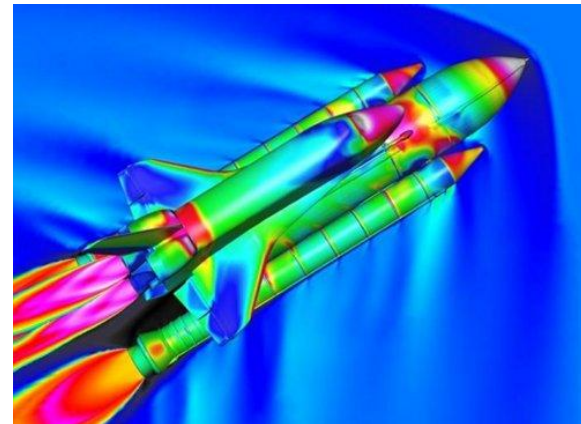
Exploration Systems



Science



Space Operations





NASA Engineering and Safety Center

- NESC established in 2003 in response to the Columbia accident.
 - Composed of the Agency's top discipline experts.
- Provide independent assessment, review, and engineering support for NASA programs and projects.
 - Engaged in some of the Agency's most difficult technical challenges.
- For the Aerosciences Discipline, this support involves the liberal use of CFD.





NASA's Aerosciences Discipline

Aerosciences at NASA is tasked with addressing the full spectrum of atmospheric flight and fluid flow.



Subsonic Low Reynolds Number





Aero-Structural Interactions





Hypersonic Aerodynamically Optimized





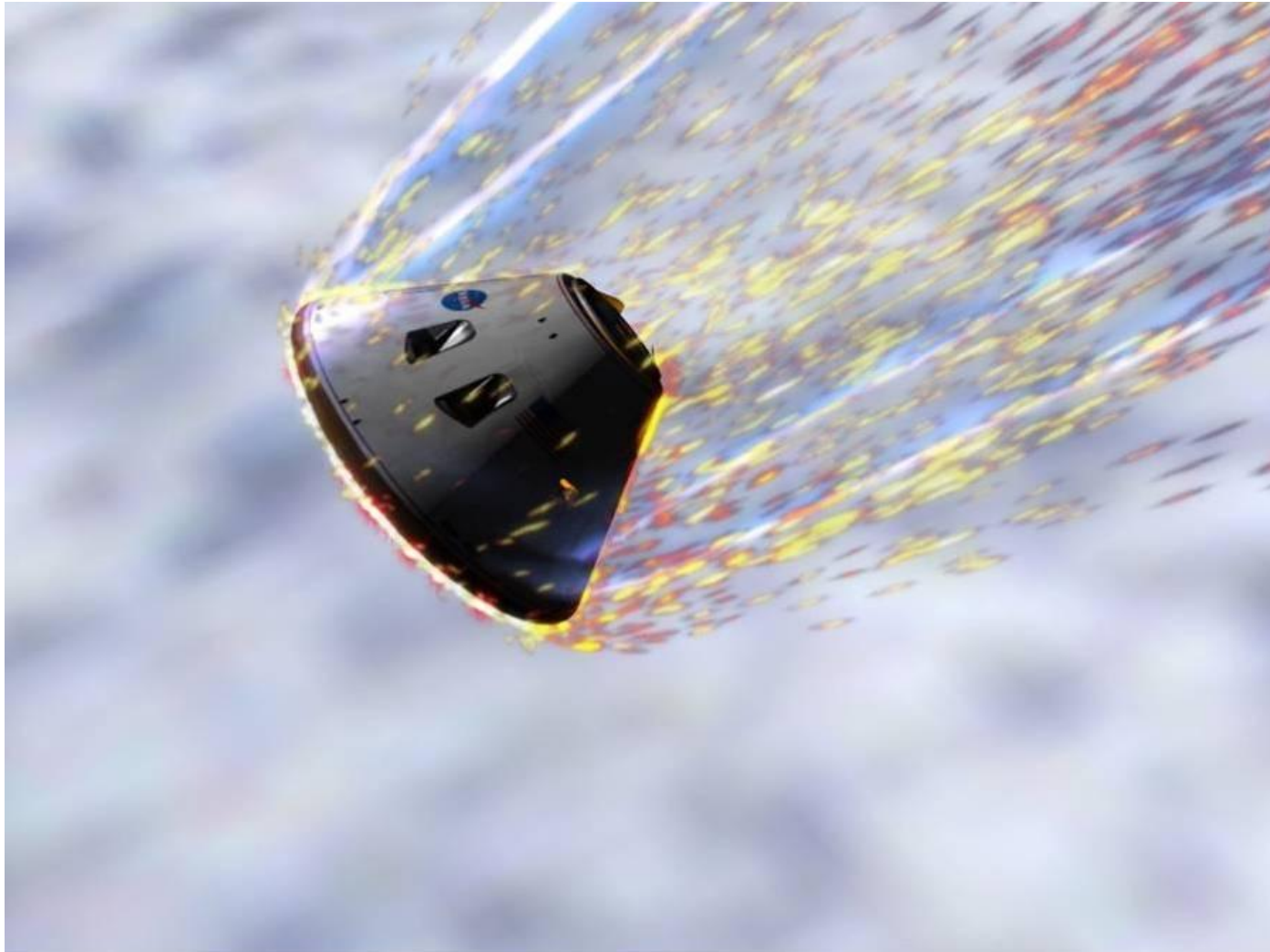
Complex Geometries and Mechanisms





Aeroheating

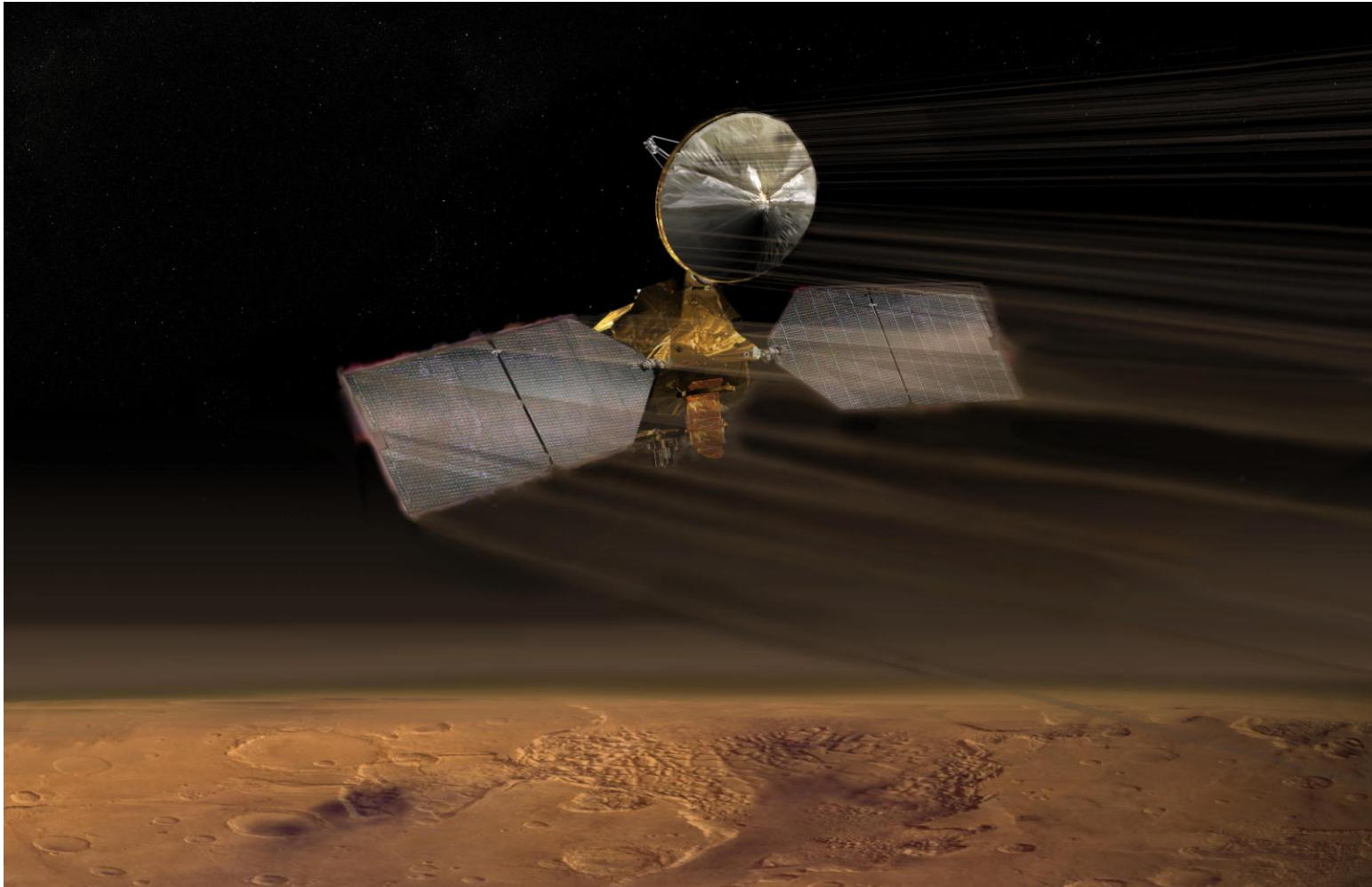
Orion Capsule Entry





Aerodynamically Non-Optimal Geometries

Mars Reconnaissance Orbiter Aerobraking for Mars Orbit Insertion



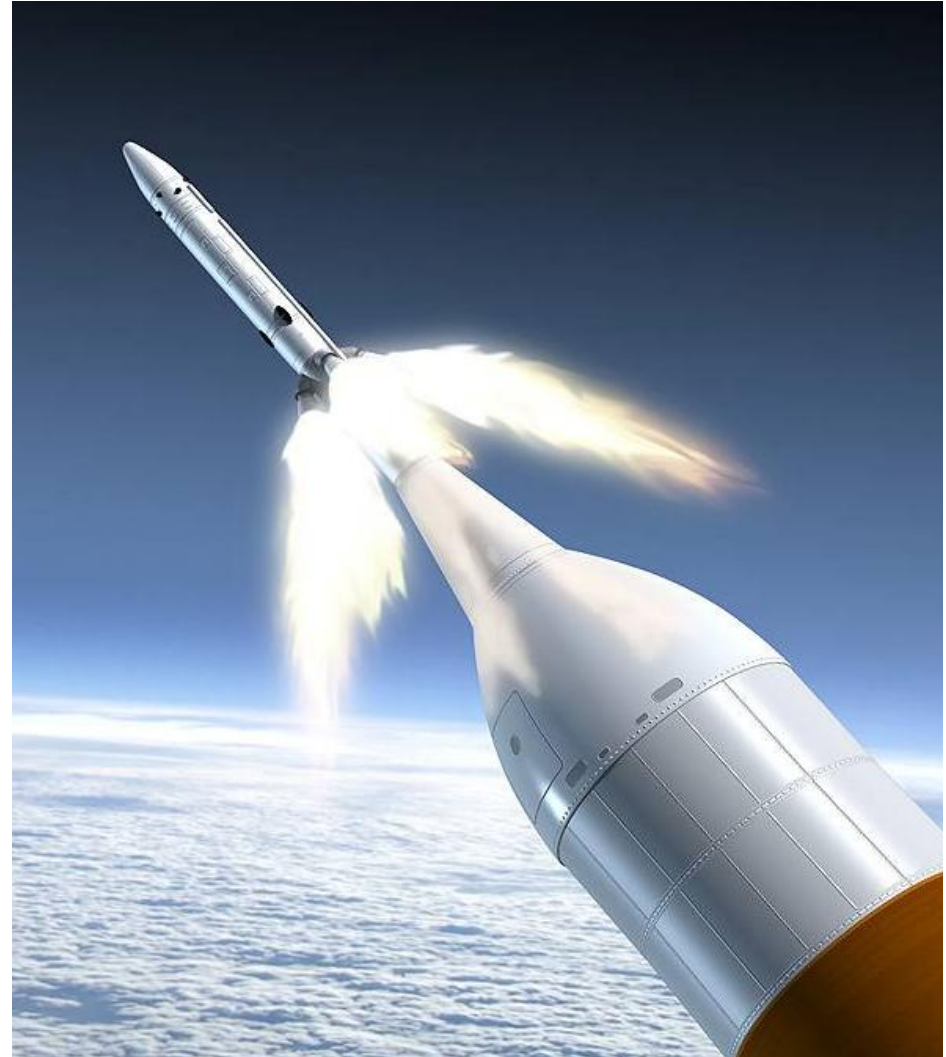


Aero-Propulsion Interaction

Space Shuttle



Orion Launch Abort System





NASA's Top Aeroscience Challenges

Three aeroscience areas are currently hindering the prediction of environments and performance for flight vehicles. (In order of criticality)

1. Aero-plume Interaction.

Plume interactions.

Aero/Reaction Control System (RCS) interactions.

2. Unsteady Separated Flows.

Aeroacoustic and buffet environments.

CFD Algorithms and turbulence modeling.

Aerodynamic performance prediction of bluff bodies and faired bodies with protuberances.

3. Aerothermodynamic Predictions.

Ablator performance.

Boundary layer transition.

Unstructured grid aerothermodynamic analysis.

Radiation heating prediction.

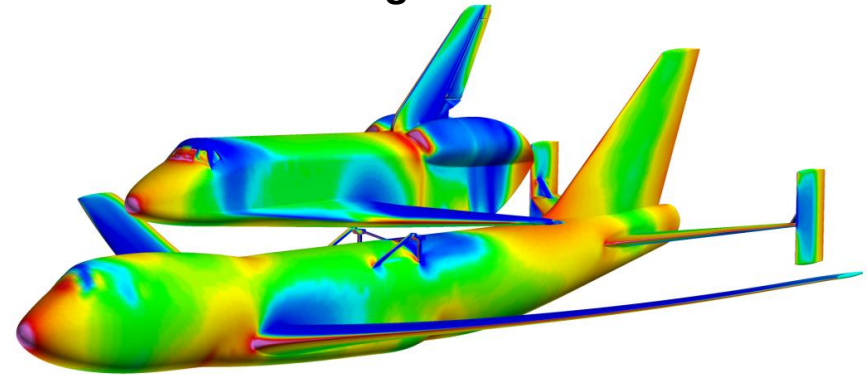


State-of-the-Art for CFD Application at NASA

Aerodynamic Performance

- Steady Reynolds Averaged Navier-Stokes (RANS) simulations on overset and unstructured grid topologies.
 - Point-matched structured grids are becoming less widely used due to geometry modeling and grid generation lead time.
- Some unsteady RANS for cases exhibiting convergence difficulties or where unsteady flow is a necessary component of the analysis.

RANS analysis of Shuttle Carrier Aircraft
Using OVERFLOW



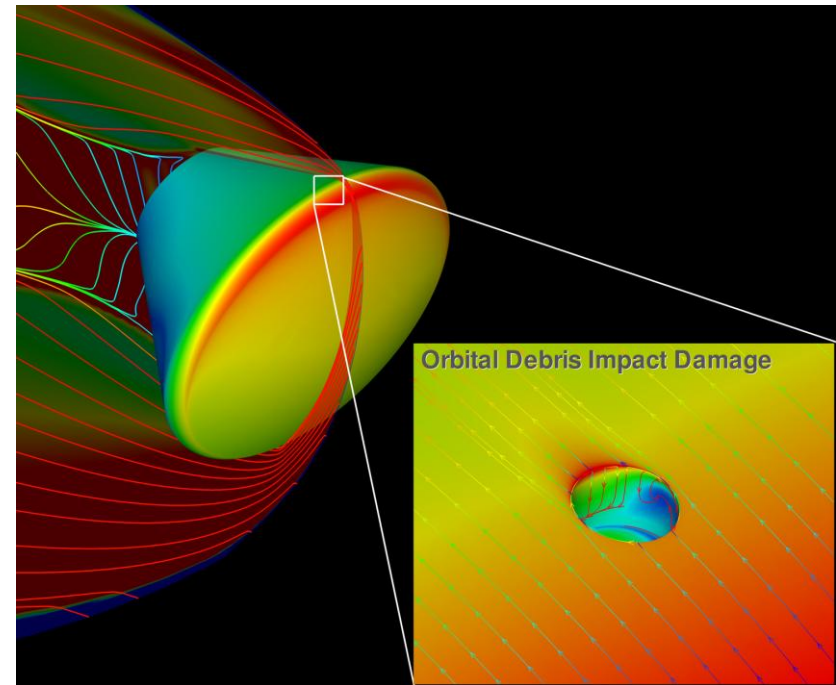


State-of-the-Art for CFD Application at NASA

Aerothermodynamic Performance

- Navier-Stokes simulations on structured grid topologies.
 - Thermal and chemical nonequilibrium.
- Unstructured grid methods continue to struggle with the prediction of surface heating.
- Rarefied gas simulation performed using Direct Simulation Monte Carlo.

Orion Debris Damage Heating
Analysis Using DPLR

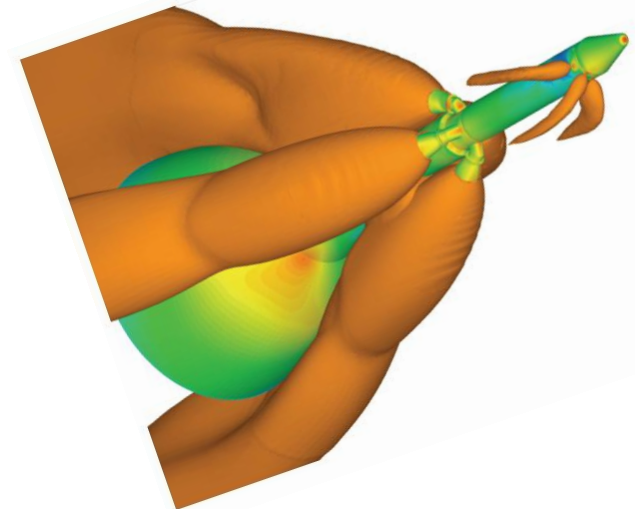
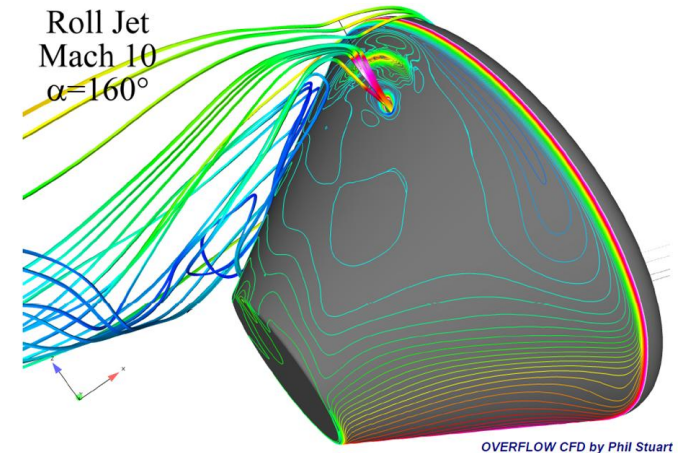




State-of-the-Art for CFD Application at NASA

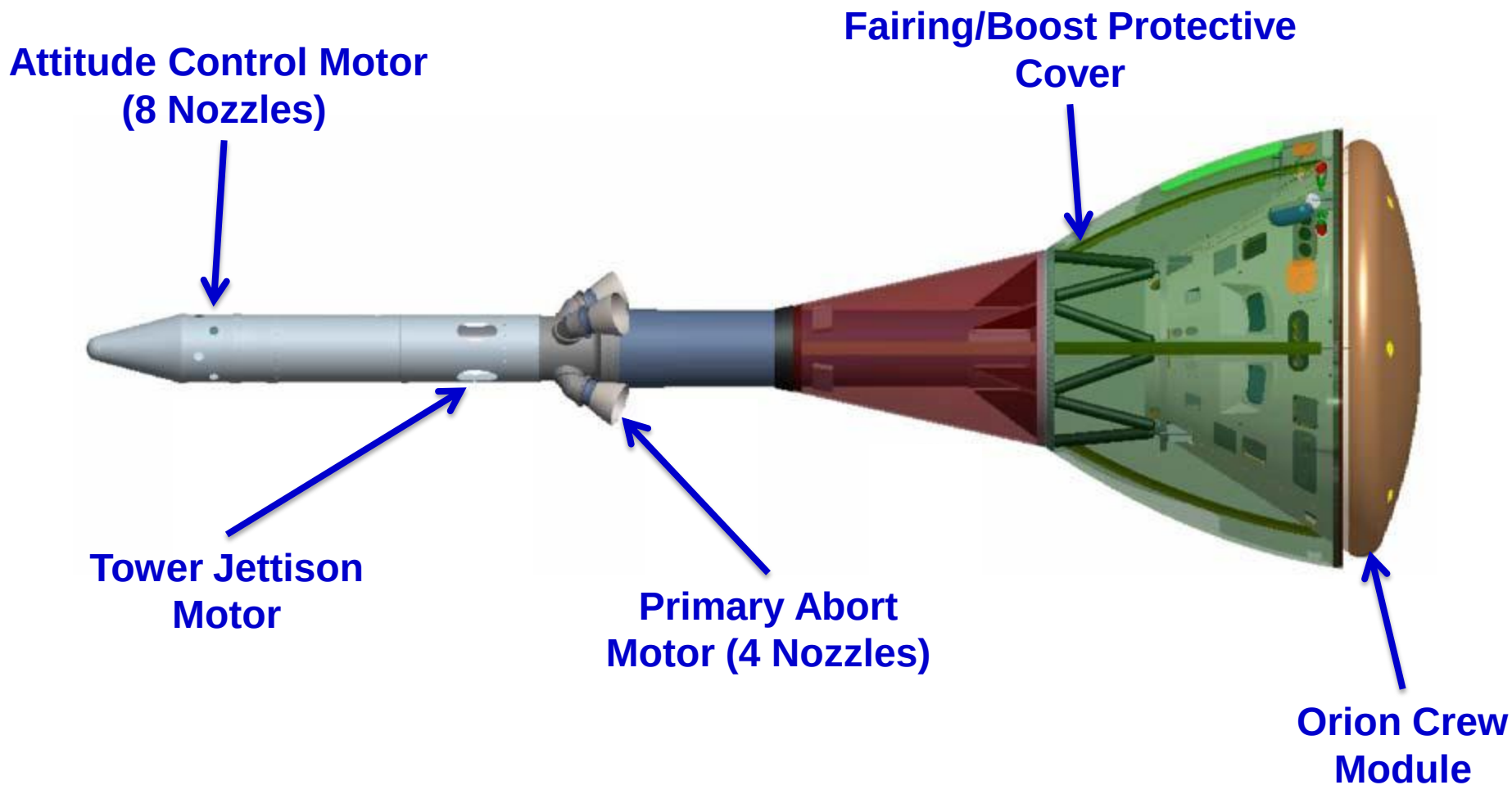
Aero-Propulsion Interaction

- Multiple-gas Navier-Stokes Simulations on structured and unstructured Grids.
 - Typically steady with frozen chemistry.
 - Some methods only capable of simulating a single gas.
- Chemical reactions can be important for rocket plume interaction problems.





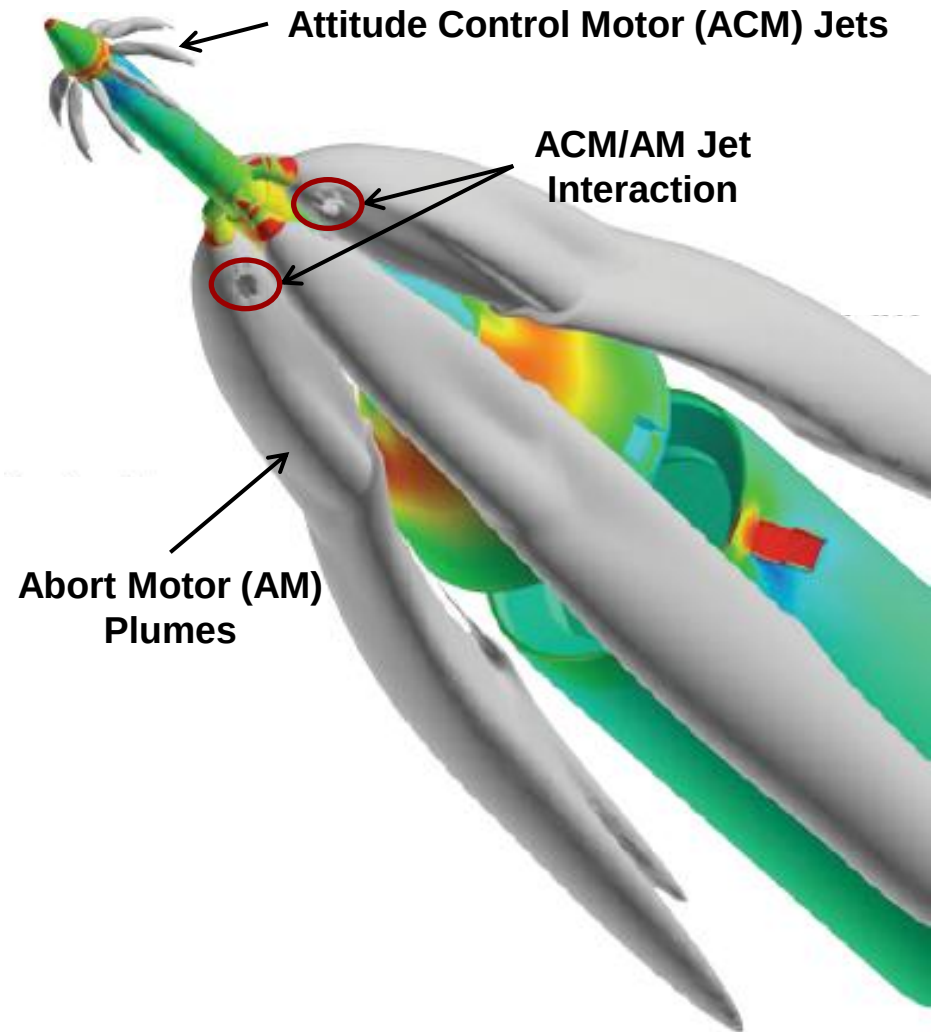
Orion Launch Abort Vehicle





Orion Launch Abort Vehicle CFD

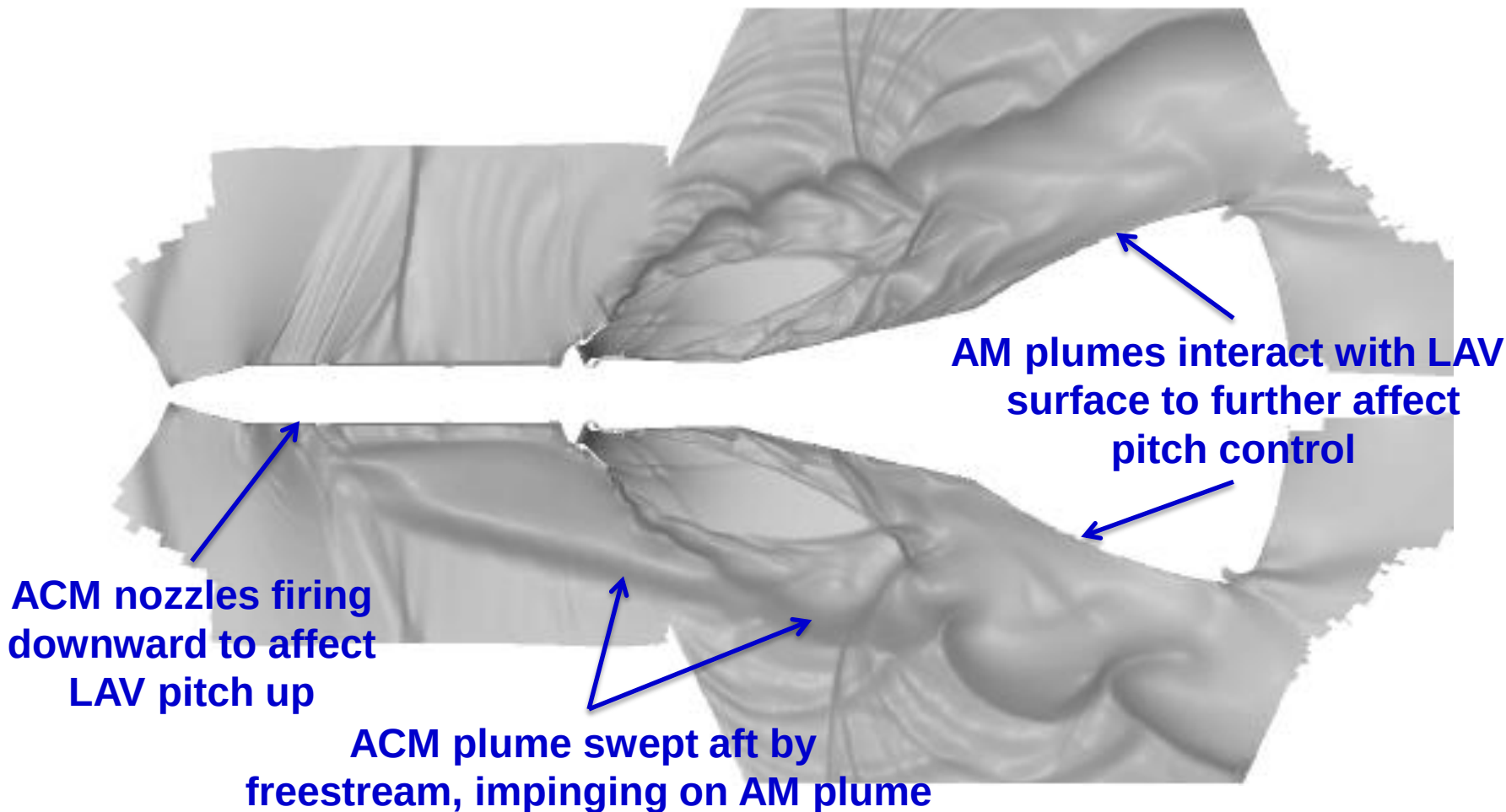
- CFD analysis required to compute massively separated flows over the bluff heatshield of the Orion capsule.
- As many as twelve interacting solid rocket motor plumes.
 - Eight Attitude Control Motor (ACM) nozzles can be metered individually leading to many parametric control cases.
- Vehicle proximity effects also predicted by CFD.
- Conditions range from low subsonic freestream to supersonic conditions.
 - Plume interactions at transonic conditions highly problematic.





Orion LAV Plume Interaction

CFD “Schlieren” of LAV Plumes at Transonic Conditions Show a Complex Interaction

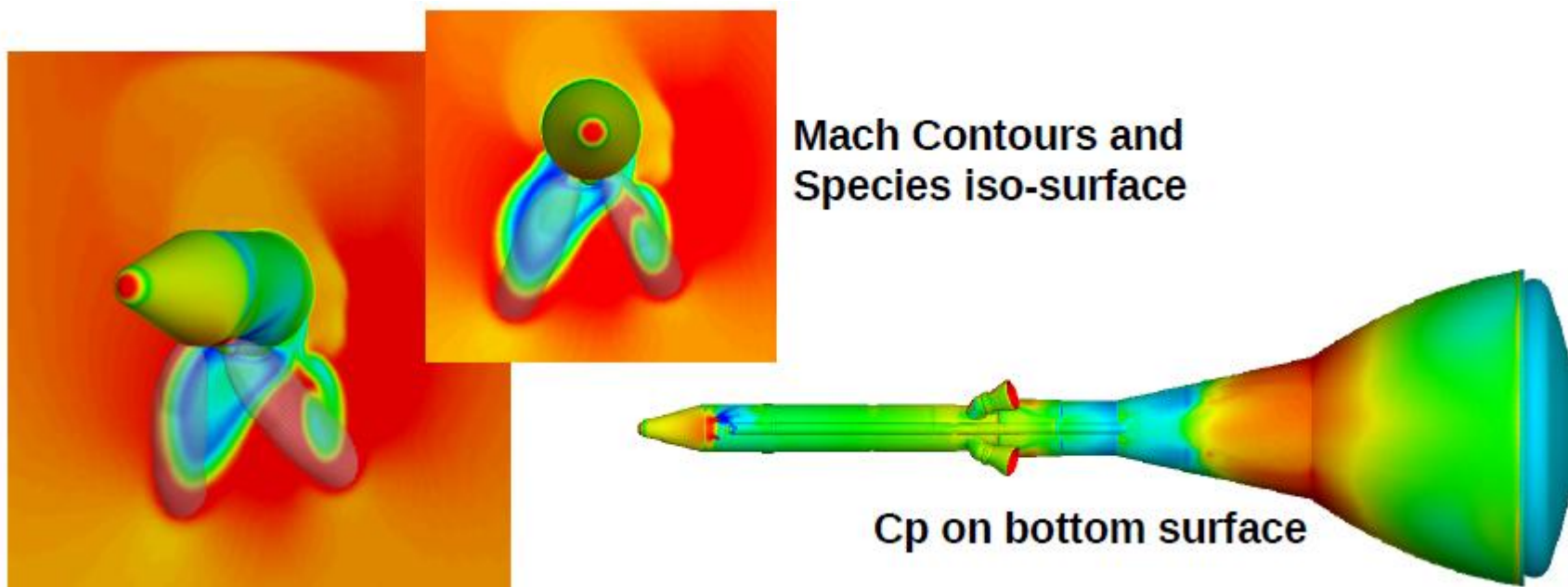




Orion LAV ACM Plume

Asymmetry

- Geometry and Flow should be mathematically symmetric left/right.
- Steady CFD shows ACM plume asymmetry at some transonic flight conditions.
 - Adversely impacts ACM pitch effectiveness and adds a yaw component.
- Phenomenon has been demonstrated for other ACM nozzle configurations (e.g. yaw).
- No wind tunnel data presently available to verify the phenomenon.

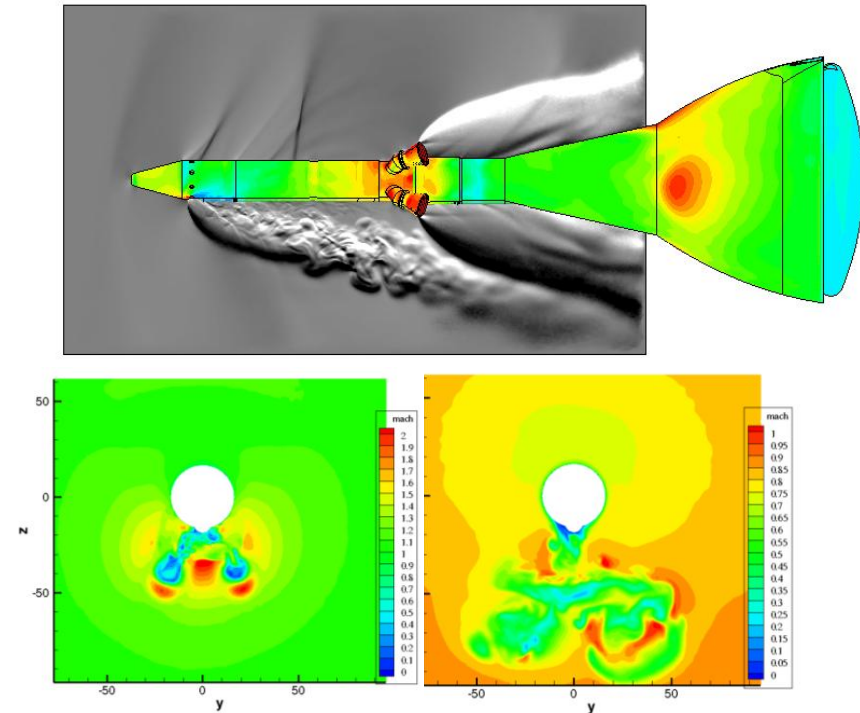




Orion AM/ACM Jet Interaction

- USM3D and OVERFLOW both predict Attitude Control Motor asymmetry at some transonic conditions.
 - Different code formulations, grid topologies, plume modeling verify issue as not numerical.
 - Both steady analyses.
- FUN3D has produced early hybrid RANS-LES simulations.
- Recent analyses of a jet-in-a-crossflow unit problem has demonstrated similar asymmetry and is proving to be a valuable test case on a much simpler configuration.

FUN3D Hybrid RANS-LES

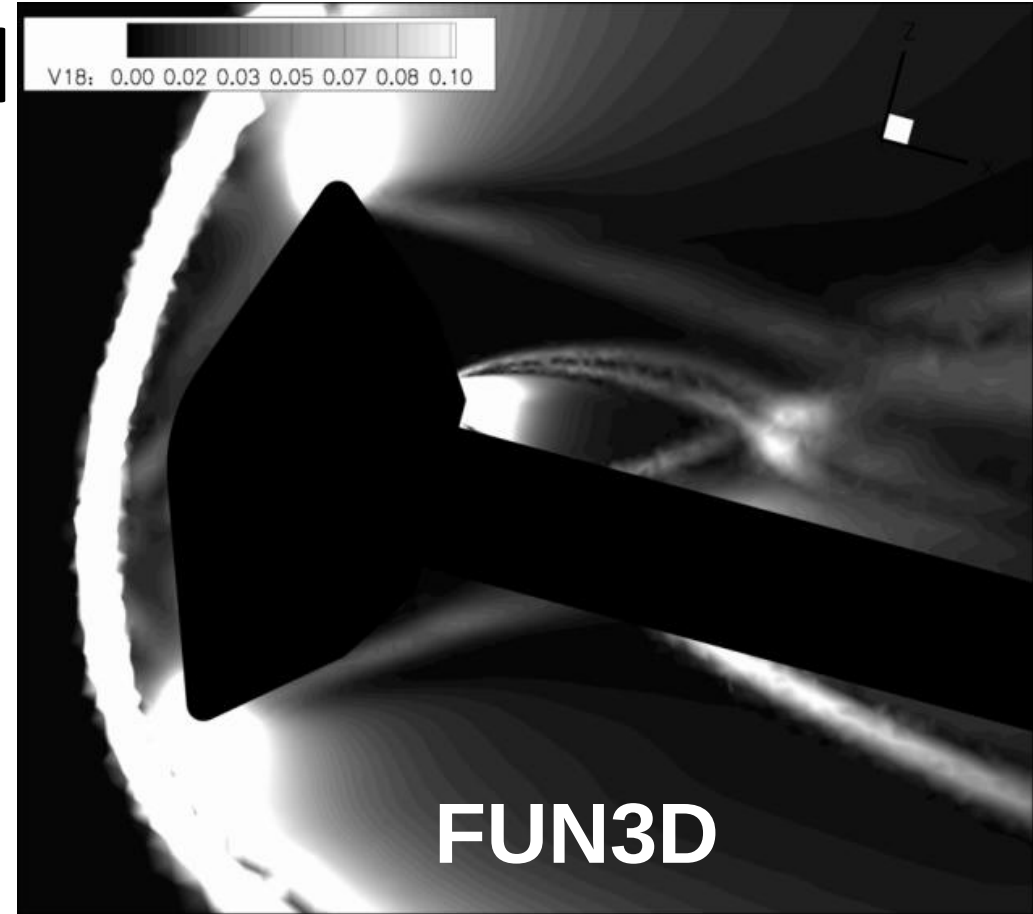




Mars Science Laboratory

Aero/Reaction Control System Jet

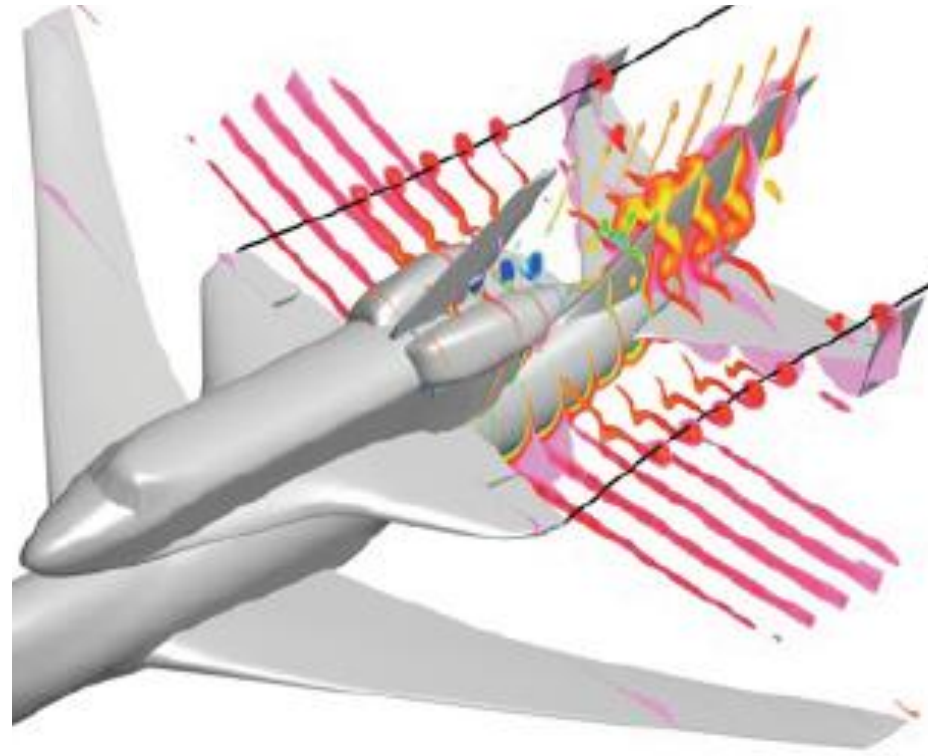
- Simulation of MSL capsule entering the Martian Atmosphere.
- Reaction Control System Jets firing into a separated flowfield in the spacecraft wake.
- Pitch, yaw, and roll control a strong function of the interaction of the RCS jet with the surrounding flowfield.





Shuttle Carrier Aircraft Modification

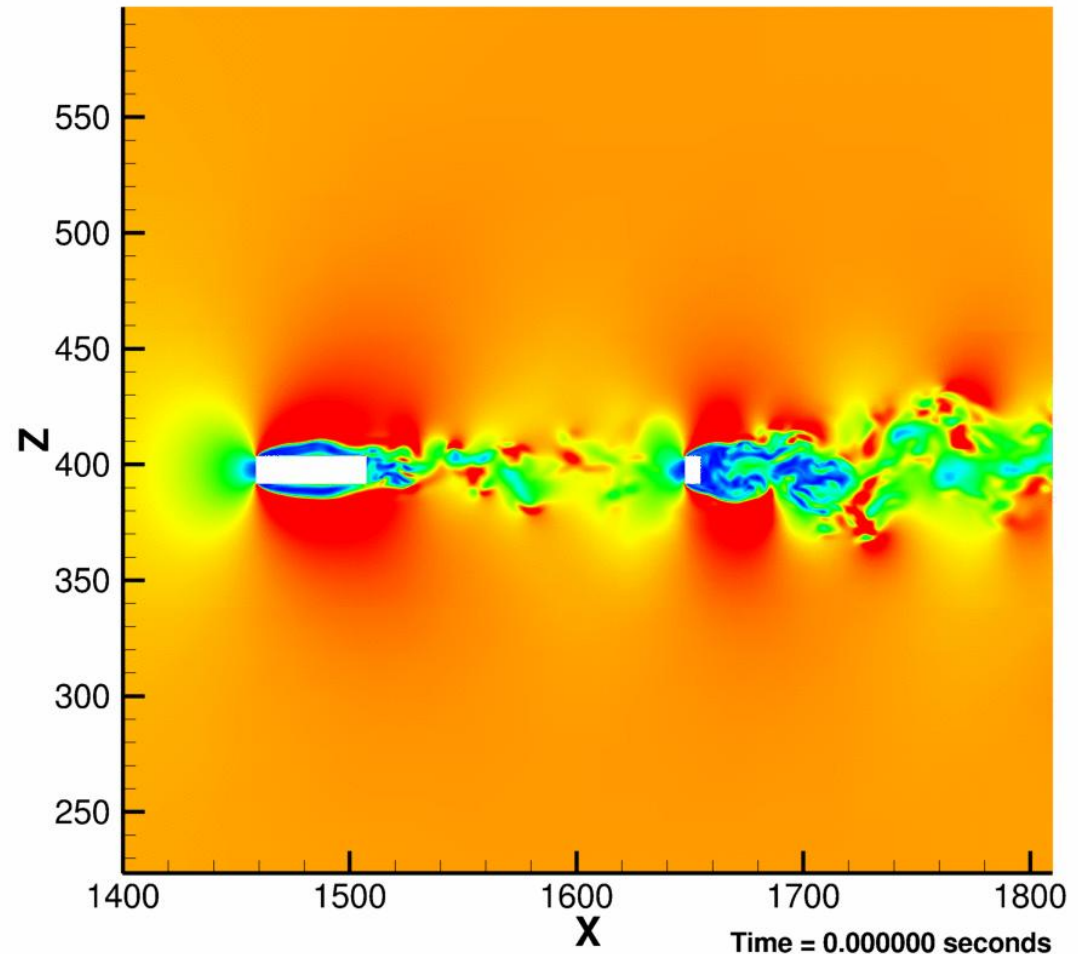
- Needed to understand aero/structural interactions at the Boeing 747 tail.
 - Fluctuating pressure amplitudes and frequency characteristics.
 - Required capability to accurately predict unsteady flows that have been convected many characteristic lengths downstream.
- Steady RANS used to predict the overall features of the flow at the tail.
 - Too dissipative to accurately predict unsteady flow downstream.
 - Higher order methods needed to solve these types of problems.





DES Application to Shuttle Carrier Aircraft Structural Elements

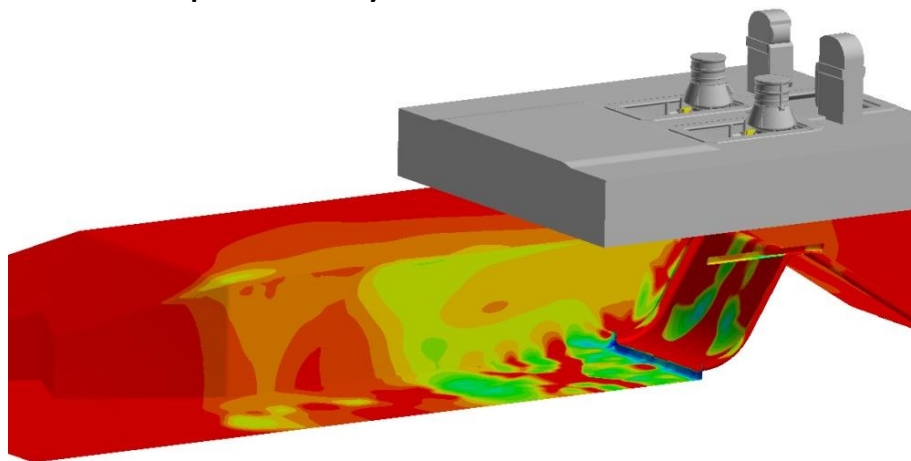
- Detached Eddy Simulation was also applied, but only on a much simpler structural component level.
- Time and resource constraints did not allow the simulation of a more complete vehicle.
- Tail environments ultimately acquired through flight test.



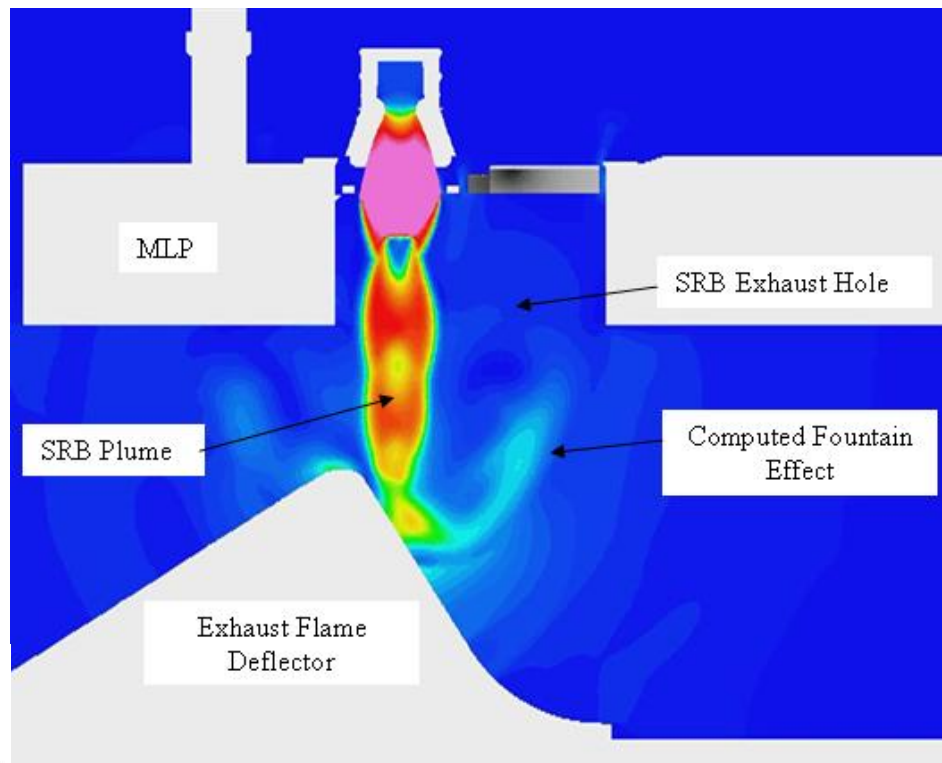


Launch Pad Environments

- NESC Aerosciences asked to review prediction of flame trench environments for investigation of STS-124 damage of Pad 39A
- High-energy, chemically reacting rocket plumes.
- Complex geometries.
- Multi-phase flows (water deluge for acoustic suppression).
- Pad surface loads for structural design.
- Flowfield characteristics for debris transport analysis.



T+0.440 seconds

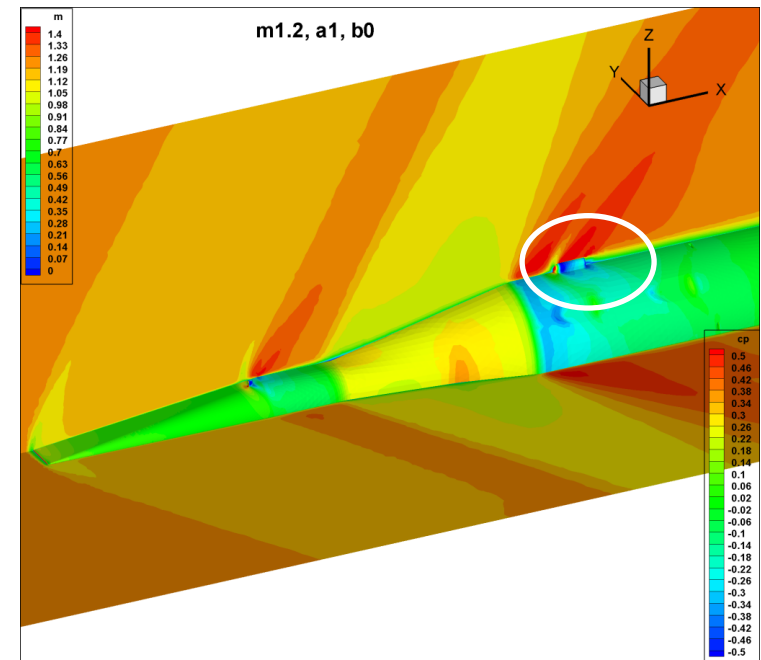
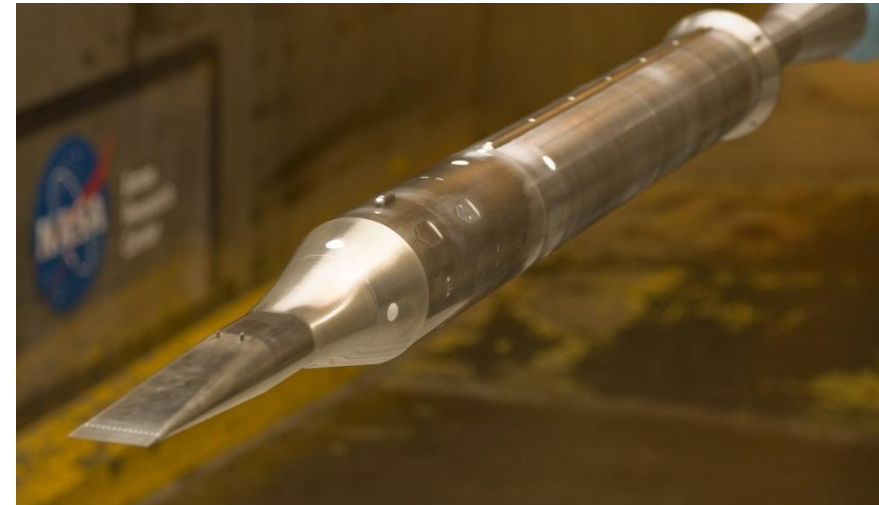




Hypersonic Boundary Layer Transition (HyBoLT)

Aeroacoustics Assessment

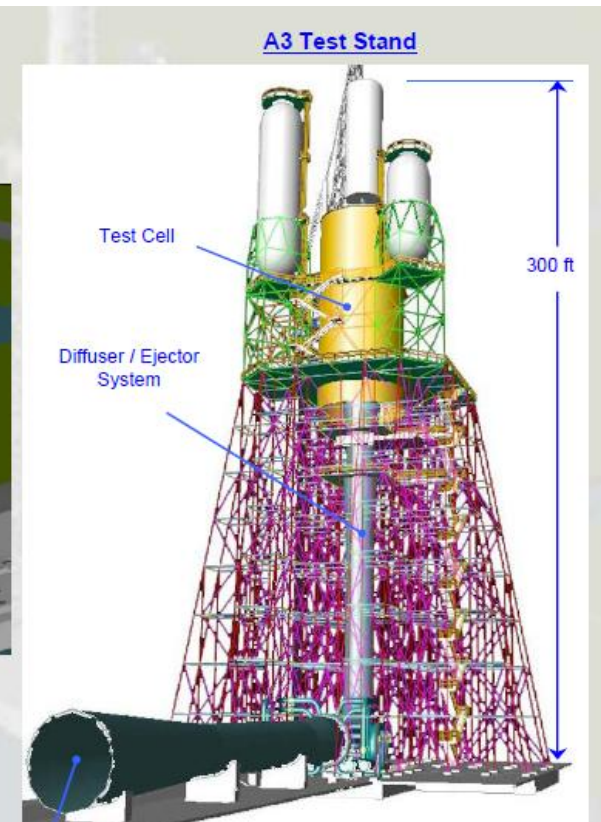
- Aeroacoustic assessment of the HyBoLT launch environment hampered by a very limited set of wind tunnel aeroacoustic data.
- State-of-the-art beyond wind tunnel is empirical prediction based on launch vehicle flight test and very limited operational data.
 - Applicability is challenged by non-similar geometries and flight conditions.
 - Steady CFD has been introduced to the process with some success to predict local flowfield characteristics such as local Mach, q , and boundary layer state.
 - Spectral content cannot be effectively addressed.
- Similar issues on Ares and Orion.





A3 Test Stand Loads Support

- One concern is Strouhal vortex shedding from tanks during hurricanes.
- How can we (efficiently) predict flows around a geometry this complex?
 - Can we even expect accurate results from our wind tunnels?
- Also requested to assess internal flow environments in support of operational dynamic loads.
 - Environments are being predicted by CFD.
 - Rocket plumes in ducts.





Common Simulation Themes

- Many common simulation themes identified among the top Aerosciences challenges:
 1. Flows involving large ranges of time and length scales.
 2. Extreme flow conditions and complex geometries that lead to highly nonlinear flow behavior.
 3. Unsteady flow.
 4. Modeling/grid adaptation required to properly capture relevant flow physics.



Emerging CFD Methodology

- High Order Algorithms:
 - Potential to deal with problems involving broad range of time and length scales.
- Unsteady simulations become the norm with steady solutions a natural subset.
 - Reverse of present situation where we run almost everything as a steady simulation.
 - Significant opportunity for fundamental algorithm development.
 - We've seen 30 years of concentrated CFD development optimizing for steady state convergence, its time to invest in algorithms developed specifically for unsteady flows.
- Higher order turbulence closure.
 - LES and Hybrid RANS/LES schemes, others?
- Adaptive algorithms.
 - Certainly adaptive grid, but maybe adaptive algorithm as well.



Application Requirements that Cannot Be Ignored

- Application engineers are not numerical algorithm experts:
 - Must simplify geometry modeling and grid generation requirements.
 - Minimize sensitivity to grid generation.
 - Effective adaptation.
 - Must have robust boundary conditions.
 - Key to order accuracy at the boundaries and simulation stability.
 - Critical for any type of zonal, patched, or overset grid approach.
 - Consistent, predictable performance and accuracy are key.
- Unsteady simulations pose a new set of post-processing challenges.
 - Time averaging techniques for integrated forces and moments.
 - unsteady data processing and possibility of short physical integration times.
 - Validation.



30 Years in the Future?

- Mesh Free Simulation:
 - Radial basis functions, wavelets, ...
 - Allows conserved variables to be optimized individually.
 - At present we solve all the conserved variables on the same grid, likely not optimal for any single conserved quantity.
 - Shouldn't we expect that there will be different optimal grid distributions for each of the conserved variables?
 - Problem size defined by a specified level of accuracy.
 - Distribution of dependent variable modeling is developed as a natural process of solving the problem ... built-in adaptation.
 - Approach is readily visualized spatially, but temporal implementation could be more challenging.



QUESTIONS?

COMMENTS?