

TAURUS II STAGE TEST SIMULATIONS: USING LARGE-SCALE CFD SIMULATIONS TO PROVIDE CRITICAL INSIGHT INTO PLUME INDUCED ENVIRONMENTS DURING DESIGN

L.L. Strutzenberg and J.S. West
NASA/MSFC/ER42
Huntsville, Alabama

ABSTRACT

This paper describes the use of targeted Loci/CHEM CFD simulations to evaluate the effects of a dual-engine first-stage hot-fire test on an evolving integrated launch pad/test article design. This effort was undertaken as a part of the NESC Independent Assessment of the Taurus II Stage Test Series. The underlying conceptual model included development of a series of computational models and simulations to analyze the plume induced environments on the pad, facility structures and test article.

A pathfinder simulation was first developed, capable of providing quick-turn around evaluation of plume impingement pressures on the flame deflector. Results from this simulation were available in time to provide data for an ongoing structural assessment of the deflector. The resulting recommendation was available in a timely manner and was incorporated into construction schedule for the new launch stand under construction at Wallops Flight Facility.

A series of Reynolds-Averaged Navier-Stokes (RANS) quasi-steady simulations representative of various key elements of the test profile was performed to identify potential concerns with the test configuration and test profile. As required, unsteady Hybrid-RANS/LES simulations were performed, to provide additional insight into critical aspects of the test sequence. Modifications to the test-specific hardware and facility structures thermal protection as well as modifications to the planned hot-fire test profile were implemented based on these simulation results.

INTRODUCTION

The Commercial Crew and Cargo Project Office (CCCPO) at NASA/JSC has a Space Act Agreement (SAA) with Orbital Sciences Corporation (OSC) to develop a new rocket (Taurus II), spacecraft (Cygnus), and a new Pad 0A launch stand at Wallops Flight Facility (WFF) to fly a demonstration mission to the International Space Station (ISS). OSC is designing the launch facility to accommodate a series of tests using the flight first stage on Pad 0A prior to the launch of Taurus II. Because problems during the test could impact the launch facilities or the first flight, CCCPO requested that the NESC independently assess this plan and identify improvements in the test planning (or design and operations) to reduce overall risk of the testing.

The NESC assessment included evaluation of the environments that the flight hardware, test hardware, and launch pad would be subjected to during the test campaign. The hot-fire test sequence, referred to as the "7K test", consists of an approximately 30-second static firing of the two first stage AJ26-62 engines on the Pad 0A launch stand at the NASA WFF. This paper includes an overview of the modeling approach taken to support the assessment and highlights of significant results obtained.

7K Test Configuration and Model Geometry

The 7K test consists of an approximately 30-second static firing of the two first-stage AJ26-62 engines on the Pad 0A launch stand at the NASA Wallops Test Facility. Figure 1 shows an artist depiction of the launch vehicle at Pad 0A on the left, and a schematic of the 7K test configuration on the right.



Figure 1. Taurus II Launch and 7K Test Configurations at WFF Pad 0A

The baseline Taurus II 7K test hardware geometry description was provided in the 5K/7K CDR data package provided by OSC [1], and is shown as included in the CFD CAD model in Figure 2. The Launch Mount (LM), Acoustic Suppression Ring (ASR), “doghouses” and water supply pipes depicted are consistent with the flight hardware geometry, but an Extension Cylinder (EC) and Extension Ring (ER) were added to provide structural support for the 7K test. The ER is connected to a flight-like aft bay casing. Details of this geometry have changed after the initially provided data as described in later sections, but the basic concept remains the same.

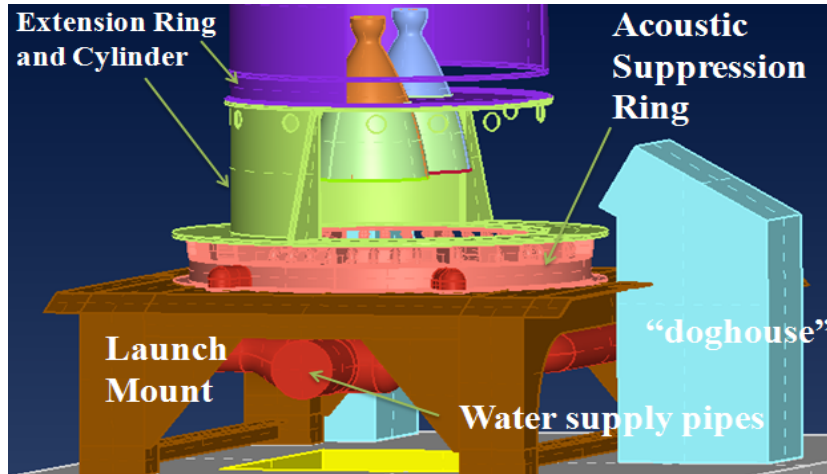


Figure 2. Taurus II 7K Test Configuration Nomenclature

The CAD model geometry included sufficient geometric fidelity to identify potential areas of concern for the 7K test configuration including the flame deflector and duct as the Pad0A launch structure was still under construction during the timeframe of the assessment. Estimates of distances to the water retention pond wall and seawall from the duct exit were scaled from provided drawings, the pertinent excerpt of which is included as Figure 3. The three distance measurements to the seawall used contained a fair amount of uncertainty due to the resolution of the drawing from which they were measured. Approximations of ground elevation were taken from drawings as well as imagery from the site visit. The seawall model extrapolated from these points has an obvious exaggeration of the angle of the seawall, but as the intent of this model was to determine order of magnitude impingement velocity and pressure on these structures, this exaggerated angle was included to allow assessment of the sensitivity of these measurements to the uncertainty inherent in the model development.

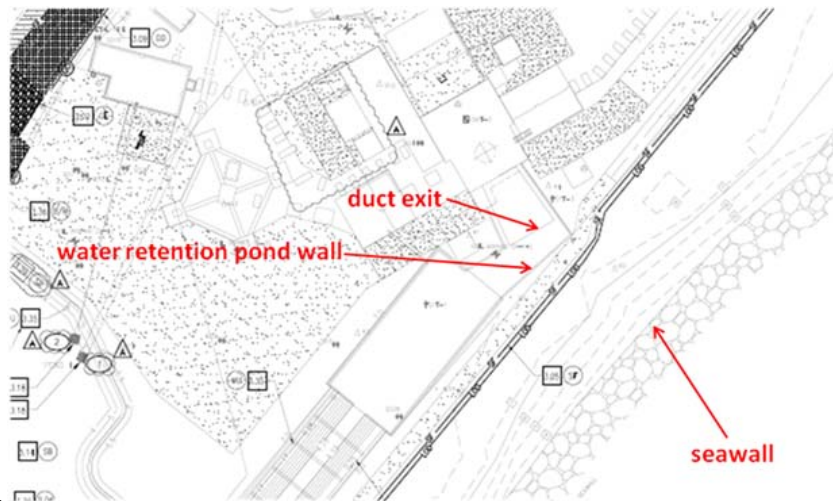


Figure 3. Plume Downstream Structure Included in CFD CAD

As shown in Figure 4, the external ground surface was meshed to capture the concrete “water retention pond wall” just downstream of the duct exit with a 6 inch scale and the seawall with a 12 inch scale. The farfield was meshed at a 12 foot scale.

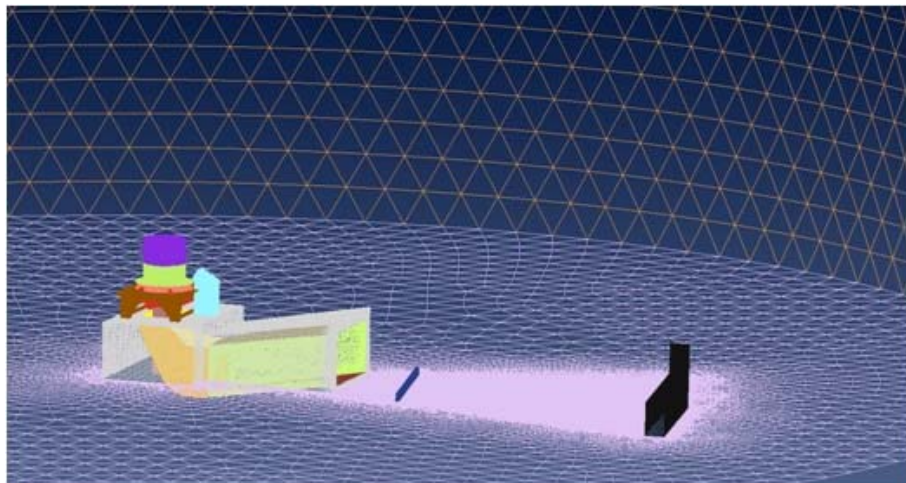


Figure 4. External Ground Surface Features and Domain Mesh

Near the plume, significant geometric fidelity imported directly from the provided CAD model was retained, although defeaturing required for mesh generation was performed. Fidelity was preserved in areas affecting plume entrainment (particularly significant blockages) and in areas with potential plume impingement during gimbals maneuvers. These model features are intended to provide approximate impingement pressures and velocities. Significant additional geometric fidelity as well as inclusion of deluge water would be required to provide truly predictive environments. Plume source cylinders were added to retain mesh clustering in the regions directly downstream of the nozzles as shown in Figure 5.

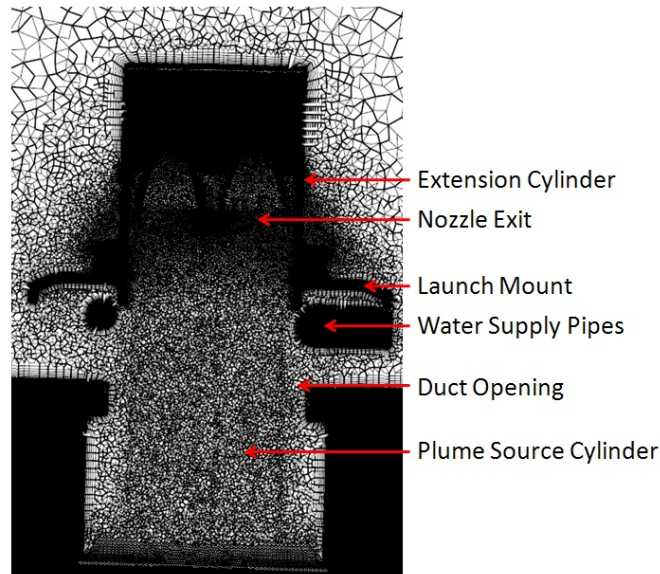


Figure 5. XZ Plane Cut Indicating Mesh Refinement in Proximity to Plumes

A plume source surface was added to retain mesh clustering in both the upper and lower shear layers of the plume downstream of the duct as shown in Figure 6.

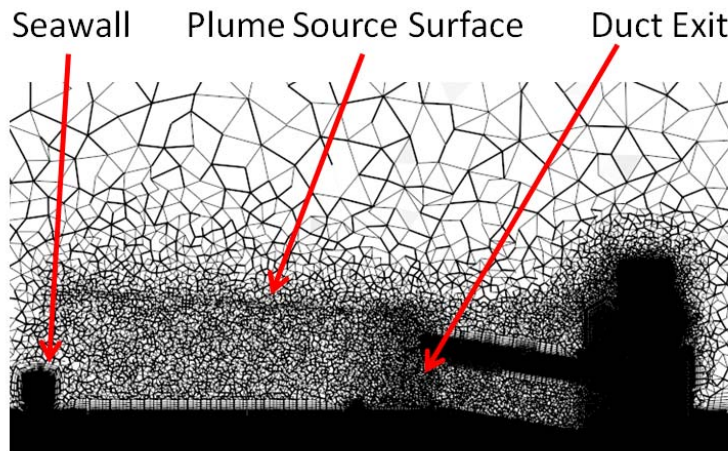


Figure 6. XY Plane Cut Showing Effect of Plume Source Surface on Mesh Refinement

7K Test Description and Simulation Objectives

The 7K test is to be a 30-second static firing including a time sequenced thrust and gimbal profile. The gimbal sequence includes 360° tandem gimbal slew sequences and a dual engine yaw-out maneuver at the end of the profile. For the 7K test, the baseline gimbal profile begins with the engines at 104.13% Pc as both engines executed a +2° pitch from null position ending at 7.5 s. In this position, both engines were to be angled “up” the deflector as shown in Figure 7. The engines would then follow the prescribed gimbal sweep arriving at the 2° yaw position at 8 s, the -2° pitch position (“down deflector”) at 8.5 s, the -2° yaw position at 9 s and returning to the +2° pitch position by 9.5 s. The engines would then return to 0° pitch while transitioning to a +0.25° yaw position. Additional gimbaling was to be performed at the 0.25° gimbal angle. Finally, at the end of the 7K test profile, the engines were to be commanded to 58.2% Pc while a “yaw-out” gimbal was executed.

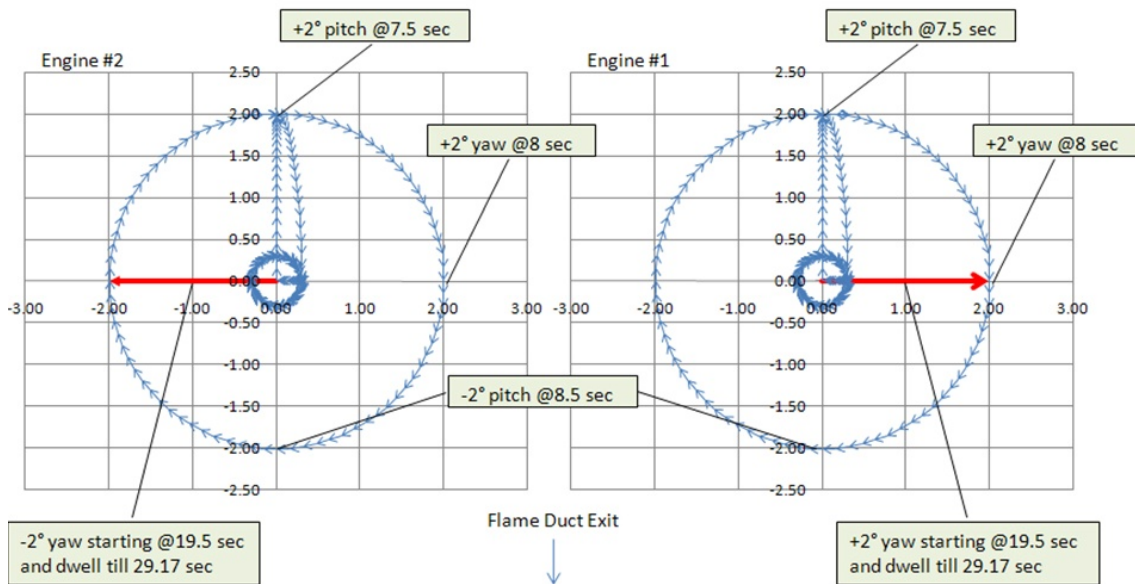


Figure 7. Baseline Engine Gimbal Profile [2]

The 7K Test commanded thrust profile provided along with the associated baseline gimbal sequence was used to define a series of quasi-steady RANS simulations characterizing the interaction of the plume flow field with the mounting hardware and launch pad structures. The objectives of these simulations were: 1) to characterize the environment on and around the attach structure including the LM, Spacer, ASR and EC and formulate recommendations to alter operations or design to mitigate adverse effects, 2) to characterize plume impingement pressures on the flame deflector to support recommendations regarding flame deflector design, and 3) to identify key profile segments requiring additional analysis, including unsteady simulations. The series of simulation cases defined for the baseline profile is summarized in Table 1.

Case Name	Gimbal Angle	%Pc	Case Description
g0v4_108	0	108	Zero gimbal, 108%Pc
g0v4_582	0	58.2	Zero gimbal, 58.2% Pc
g0v4_10413	0	104.13	Zero gimbal, 104.13%Pc
g0v4_100	0	100	Zero gimbal, 100%Pc
2gyaw_10413	Yaw +2	104.13	2° gimbal towards duct side
2gup_10413	Pitch +2	104.13	2° gimbal towards deflector top
2gdn_10413	Pitch -2	104.13	2° gimbal towards deflector base
141gup_10413	Pitch +1.41 Yaw +1.41	104.13	Up deflector towards corner
141gdn_10413	Pitch +1.41 Yaw +1.41	104.13	Down deflector towards corner
2gyout_582	Pitch +1.41 Yaw +1.41	58.2	Both engines “yaw-out” towards duct walls
2gyout_100	Pitch +1.41 Yaw +1.41	100	Both engines “yaw-out” towards duct walls

Table 1. Initial Definition of Quasi-steady RANS Simulations

For each RPL, mass flow rate and O/F ratio given in the thrust profile definition, the chemical composition, state and mass flow rate inputs used to describe the nozzle inlet boundary condition were derived from the Thermo-Chemical Equilibrium Program (TEP) [3] output. The nozzle CFD simulation was set up using a fixed composition, total temperature and mass flow rate. A nine-species “wet C-O” thermodynamic model (wetco.mdl) and finite-rate chemistry mechanism was constructed to model the evolving chemistry as the flow proceeds down and out of the nozzle based on an altered version of the ten-species kerosene model formulated [4]. A slug flow velocity profile was used at the nozzle inlet with assumed (as opposed to estimated) turbulent intensity values.

In parallel with development of the full model geometry, quasi-steady single-plume and dual-plume pathfinder cases were performed. The primary objective of these cases was to determine whether sufficient information was available to perform CFD simulations of the AJ26 plumes interacting with the 7K test hardware and launch pad structures. The pathfinder and subsequent simulations were performed using the general purpose CFD program Loci/CHEM [5] (version 3.2-pre1). The geometry for the dual-plume pathfinder case included a simplified representation of the duct, deflector and tunnel. The mesh resolution was shown to be sufficient to reveal shock structure down to the deflector.

The nozzle curvature was resolved with maximum of 3 degrees change in triangle surface normal from geometry curvature and the nozzle lip was resolved with 0.2 inch spacing. The flame deflector was resolved with 2 inch isotropic spacing. A smooth transition from the nozzle lip to the deflector was achieved by inclusion of a cylindrical source as shown in Figure 8.

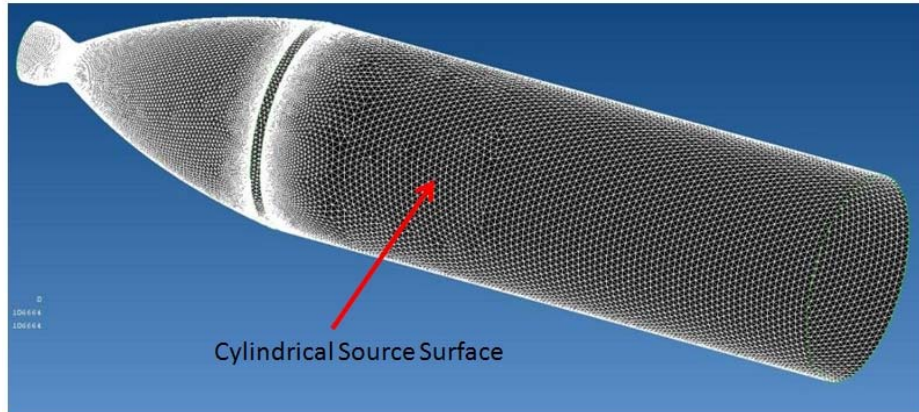


Figure 8. Nozzle Mesh Characteristics and Cylindrical Source Inclusion

RESULTS AND DISCUSSION

Pathfinder Case Results

The general purpose CFD program Loci/CHEM was chosen to perform this simulation with the following characteristics:

- Fixed mass inflow BC for nozzle chamber inlet
- Farfield BC with Mach number =0.05 and sea-level ambient conditions on the outer surfaces of the computational domain
- No-slip adiabatic boundary condition for solid surfaces
- Initial conditions: quiescent at sea-level ambient state
- Menter's SST turbulence model (integrate to the wall)
- Second order spatial resolution
- Local time-stepping mode with the following parameter settings: dtmax = 1 s, urelax =5%, cflmax=1e6
- Symmetric Gauss-Seidel matrix solver with 10 iterations per timestep

After 500 timesteps which took 10.5 hours of wall-clock time, the simulation was reasonably converged. Primary residuals dropped by as much as 2.5 orders of magnitude (typical for local timestepping approach and a real-world simulation). The computational domain mass fluxes were no longer changing significantly with increasing timesteps. The total mass flow balance was 1.8% of the nozzle flow rate and 1.3% of the combined nozzle inlet flow and air, which is typical of an acceptable solution.

An initial assessment of the provided data including the results from the pathfinder simulations was presented at the first Face-to-Face meeting of the NESC team with OSC August 25 – 27, 2010. Also at this meeting, the results of a preliminary structural assessment of the flame deflector refractory anchoring mechanism design were presented. At the time of the meeting, in order to affect the construction of the flame deflector without requiring rework, a recommendation was needed from the team within two weeks of the first meeting.

The plume structure and impingement characteristics of the dual-plume pathfinder simulation indicate that the mesh resolution is sufficient to reveal the shock structure down to the deflector. High pressures occur in two streamwise locations (initial and secondary turning

locations) as shown in Figure 9. These static pressures (about 100 psia) were consistent with PLIMP results for the primary impingement region [6]. PLIMP models the plume as an axisymmetric jet and the impinged structures are based on simplified shape such as cones, cylinders and flat plates. For the PLIMP model, the Taurus II plume was first computed using a combination of Reacting and Multiphase (RAMP) and Standardized Plume Flowfield (SPF/3) computer codes [7, 8]. The plume “splash” effects, not predictable using PLIMP, were shown to be significant, resulting in a secondary high pressure impingement region for each plume.

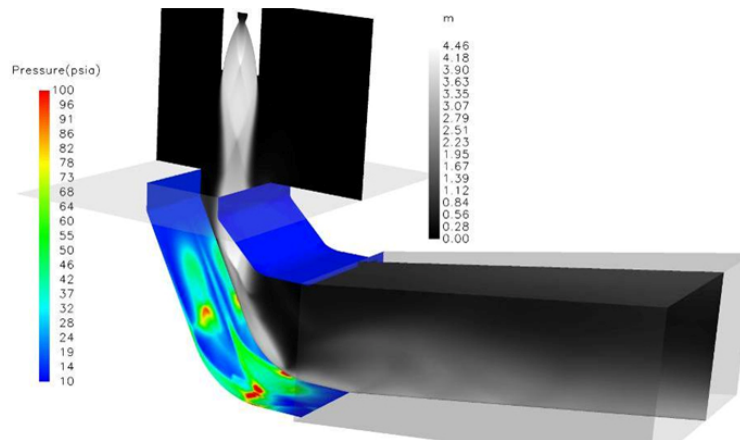


Figure 9. Plume Impingement Pressures on Flame Deflector Surface

In addition to the inclusion of the secondary impingement location, engine gimbals are expected to increase the areas of the deflector that are subjected to high pressures. Furthermore, plume unsteadiness is likely to increase the high pressure impingement region due to the dithering of the plume over a region. Given this variability, the dual-plume pathfinder case was used to define four regions of the deflector to be considered for high pressure impact environments. This recommendation was provided within weeks of the initial problem definition and the refractory anchoring system was modified with minimal impact to cost and schedule.

Zero-Gimbal Quasi-Steady Simulation Case Results

As described in Table 1, quasi-steady simulations were performed for four RPL conditions: 108% Pc, 58.2% Pc, 100% Pc and 104.13%. These zero-gimbal cases represent segments of the 7K test profile, and are also useful for comparison with various gimbal cases. A comparison of the pressures on the deflector resulting from the zero gimbal simulations performed, as shown in Figure 10, indicates that the environments derived from the pathfinder case were reasonable and conservative.

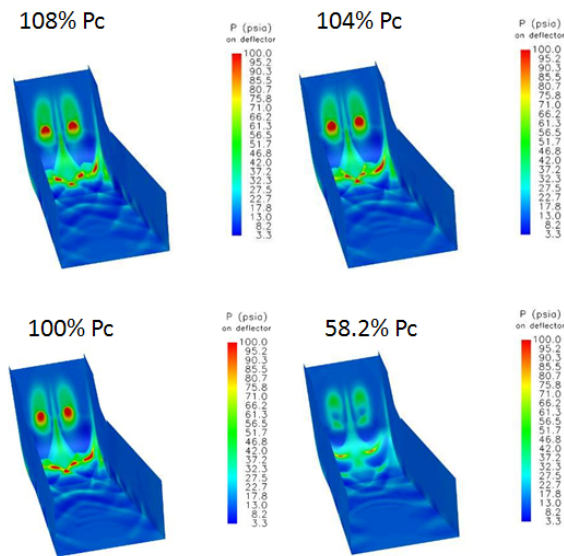


Figure 10. Plume Impingement Pressures on Flame Deflector Surface

For the 108% Pc, zero gimbal case a mesh sensitivity case was performed. Due to the very limited timeframe for the assessment, a thorough sensitivity study was not feasible. However, key areas with close proximity to the plume were refined significantly and compared with the baseline mesh results. The resulting mesh distribution in the primary region of interest is shown in Figure 11.

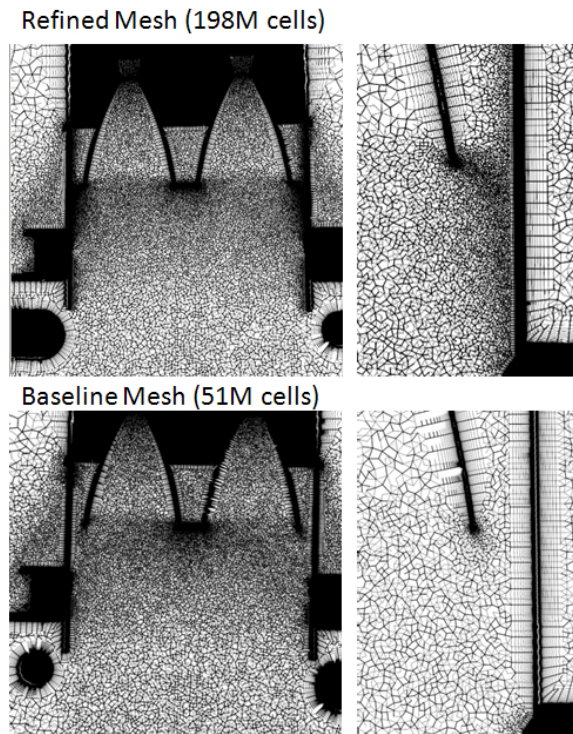


Figure 11. Comparison of Mesh Refinement for Sensitivity Study

As shown in the top row of Figure 12, the pressures within the External Cylinder are similar for the baseline and refined mesh and indicate plume aspiration driven convective pressure differentials. Similarly, the magnitudes of the maximum corresponding static temperature on the EC are in magnitude of the maximum temperature, although the locations of the higher heating areas are different for the two versions as is apparent in the middle set of images. However, this is within the expected variation for convection driven heating. The final row of Figure 12 shows a plane cut at the spacer for the two simulations indicating asymmetric convective flow in each case. The variation in the flow pattern results in differences in heating of the EC. Both cases are “quasi-steady” and include artifacts of this assumption. It was expected that the steady assumption would result in “worst case” environments compared with an unsteady solution. The validity of this expectation is addressed later in this paper.

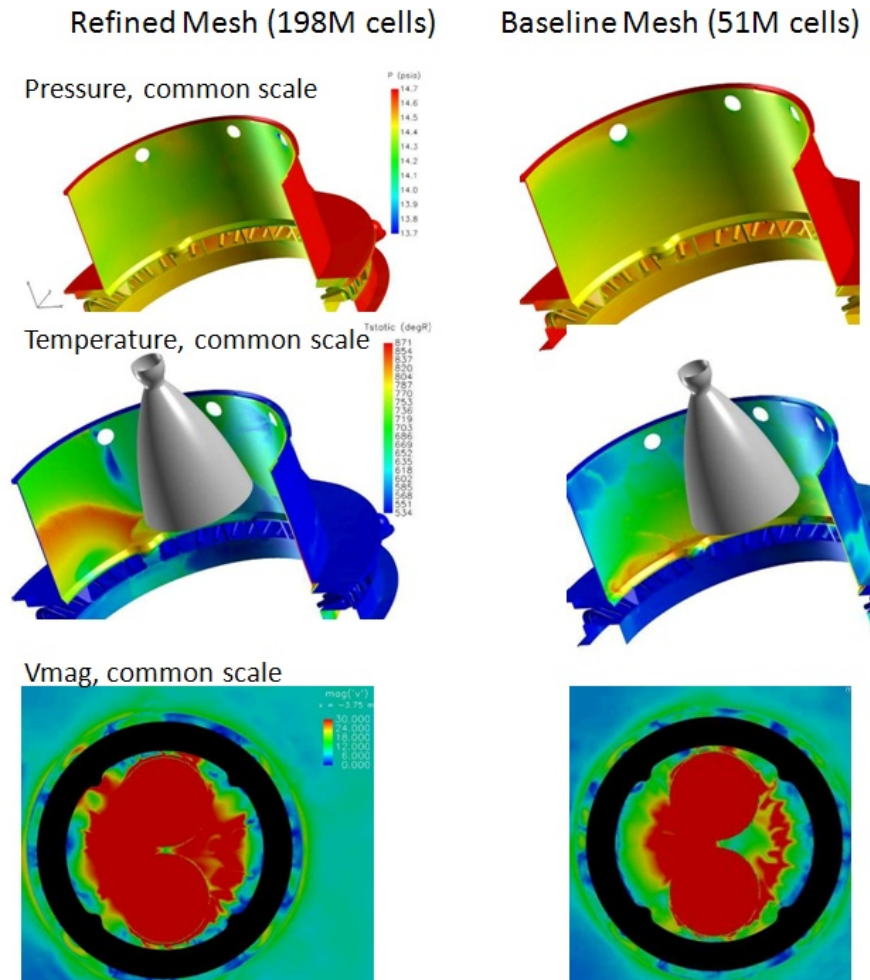


Figure 12. Comparison of Results, Baseline vs. Refined Mesh

This limited assessment indicates that the sensitivity of the provided results to this level of mesh refinement is within the expected error induced by quasi-steady simulation limitations, environmental uncertainty and geometric and configuration approximations. Given the short timeframe available during the study, additional mesh refinement studies were not performed. Each gimbal-specific case required a new mesh to be generated. Mesh sizes varied for each case, but were maintained within the range covered by this sensitivity assessment.

Two Degree Yaw-Out Gimbal Simulation Results

In the baseline 7K test profile, following the 0.25 degree gimbal slew, the chamber pressure is reduced from 104.13% Pc to 100%Pc. At 19.5 s, a two-degree yaw-out gimbal is commanded. The chamber pressure is further reduced to 58.2% Pc at 23.67 s. There is some potential overlap between the 100%Pc and the completion of the yaw-out maneuver. Pending clarification of timeline events and uncertainties, two CFD simulations were performed with the two-degree yaw-out gimbal: 100%Pc and 58.2%Pc.

As shown in Figure 13, the 100 % Pc simulation indicates shear layer impingement on the ASR, the Spacer, and the EC.

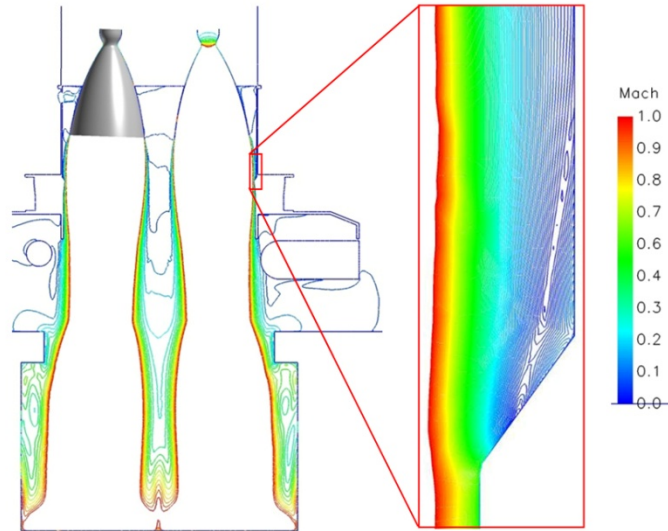


Figure 13. Shear Layer Impingement (2° yaw-out, 100%Pc)

The static temperature on the EC, shown in Figure 14, also indicates localized heating resulting from the proximity of the plume to the EC and spacer surfaces. The 58.2% Pc simulation is considerably more benign from a plume impingement standpoint.

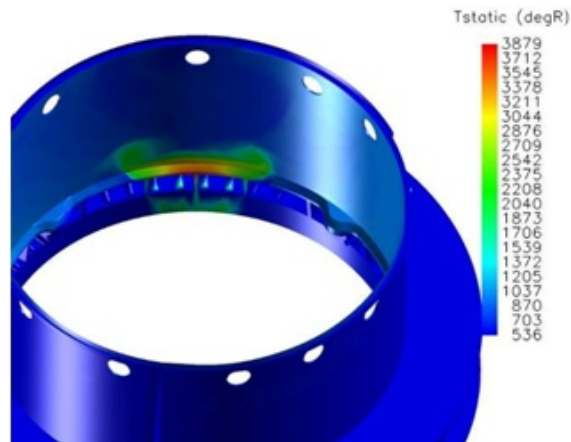


Figure 14. Static Temperature on EC (2° yaw-out, 100%Pc)

To perform thermal analysis using the quasi-steady CFD as an input, OSC requested that the CFD data be mapped to 70", 73", 74" and 75" cylindrical structured grids. AnimatorP [9, reference manual] was utilized to perform this post-processing. Requested data were provided in three datasets: Q1 (r, U, V, W, t); Q2 (fO2, fH2O, fCO, fCO2, fN2) and Q3 (cp, tmuu, tkconduct, kconduct and pg). Requested datasets were also mapped to 2D meshes along z=0 and y=0 cutting planes with boundary layer grid refinement as required.

As an example, Figure 15 depicts turbulent eddy viscosity extracted onto the 70" cylindrical structure grid intersecting a z=0 cutting plane in the unstructured solution file. This type of visualization was performed to qualitatively evaluate data extraction module performance prior to delivery of the extracted data to OSC.

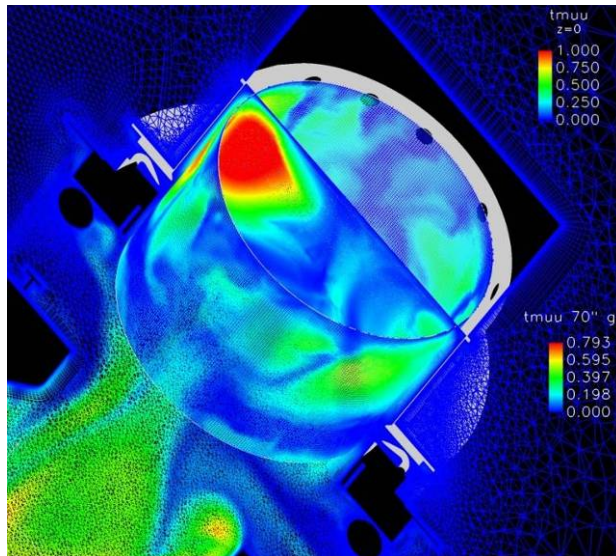


Figure 15. Example of CFD Solution Data Extracted for Delivery to OSC

Two Degree Gimbal Sweep Simulation Results

The "Two Degree Gimbal Sweep" segment of the baseline test profile begins at 7 s with the engines at a power level of 104.13% Pc as both engines execute a +2 degree pitch maneuver from null position ending at 7.5 s. For the +2 degree pitch maneuver, both engines are angled "up" the deflector as shown in Figure 7. The engines then follow the prescribed gimbal sweep arriving at the 2° yaw position at 8 s, the -2° pitch position ("down deflector") at 8.5 s, the -2° yaw position at 9 s and returning to the +2° pitch position by 9.5 s. The engines then return to 0° pitch while transitioning to a +0.25° yaw position.

Quasi-steady simulations performed to provide insight into the key flowfield features and potential areas of concern during this maneuver included +2° pitch (up deflector), -1.41° pitch/+1.41° yaw (45° "up deflector"), 2° yaw, -1.41° pitch/-1.41° yaw (45 degrees "down deflector") and -2° pitch ("down deflector"). The 2° yaw case simulated is on the "down" half of the sweep while the two 45° cases are actually on the "up" half of the sweep. Performing the 45° cases from the "opposite" side of the sweep was prioritized to evaluate the relative sensitivity to the geometric asymmetry prior to performing the full sweep of cases if necessary. Since the persistent shear layer impingement on the spacer was identified as the driving concern, additional cases using this gimbal sweep profile were not performed.

Key results of the quasi-steady gimbal sweep simulations include indication of plume shear layer impingement and significant flow blockage for each of the three cases with absolute value of 1.41° and greater gimbal in the yaw direction. This indicates that high heating persists in

proximity to the $y=0$ plane on the $-z$ side of the EC for at least 0.5 s during the gimballed sweep as the plume moves around the circular gimballed path and down the deflector. A similar high heating profile is expected to occur on the $+z$ side of the EC although the unsteady flow characteristics will differ, primarily due to the direction of the plume motion (up the deflector). For each of these “high heating” cases, a set of CFD outputs was provided to OSC for detailed thermal analysis.

Plume flow characteristics during the gimballed sweep including main flame deflector pressures and visualization of the Mach 1 isosurface are depicted in Figure 16.

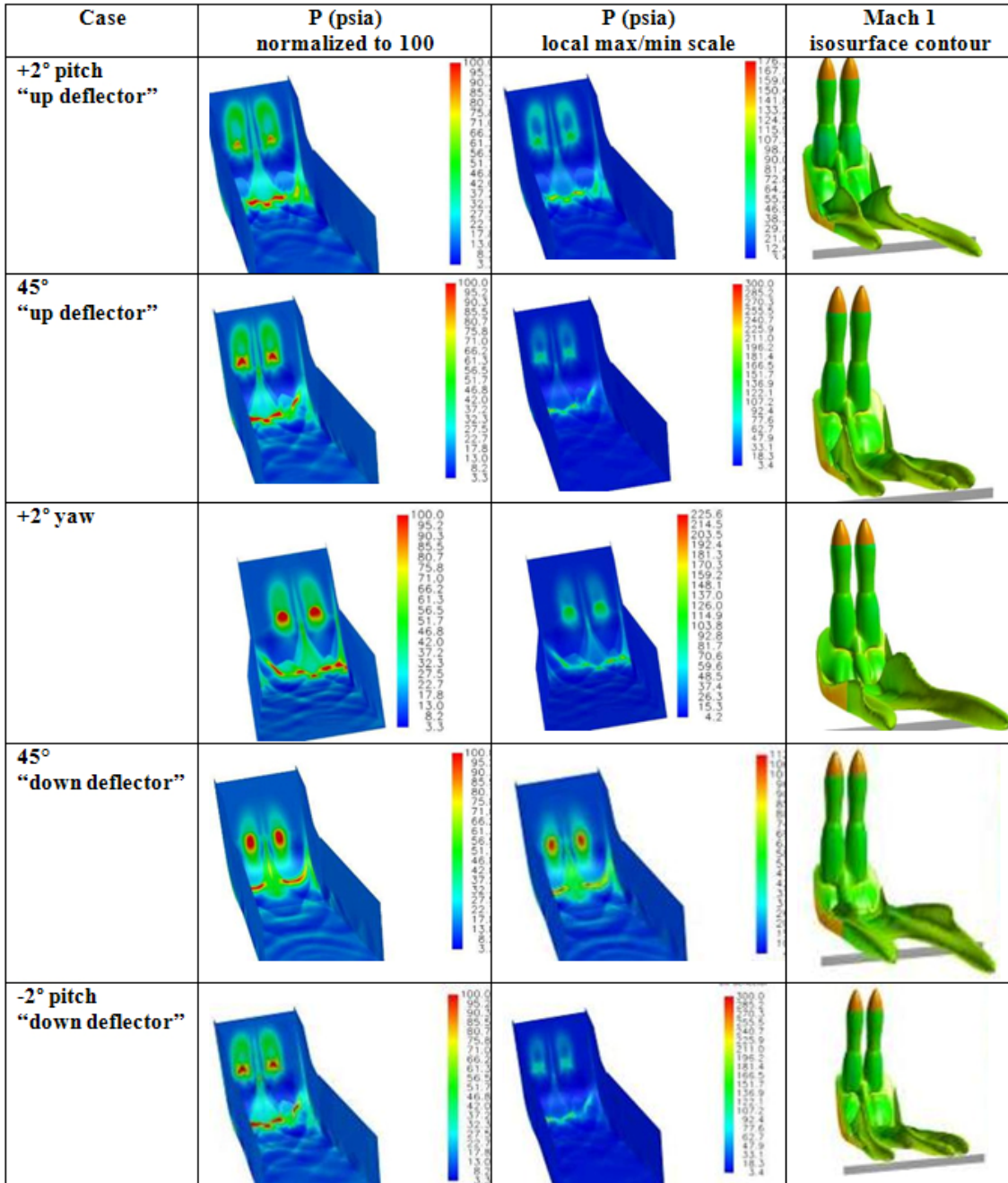


Figure 16. Gimbal Sweep, Plume Impingement and Flow Visualization

Additional Quasi-Steady Gimbal Case Results

Following the delivery of 2° yaw case results, OSC indicated a potential modification to the 7K Gimbal Profile may be possible while still meeting the intent of the requirements of the test. Supporting analysis for this decision included performing simulations of 1.0° and 1.5° yaw cases for comparison to the 2.0° yaw case previously delivered. As shown in Figure 17, there is still some interaction of the shear layer with the spacer for the 1.0° and 1.5° cases at 104.13% Pc, although the interaction in the 1.0° case is extremely minimal.

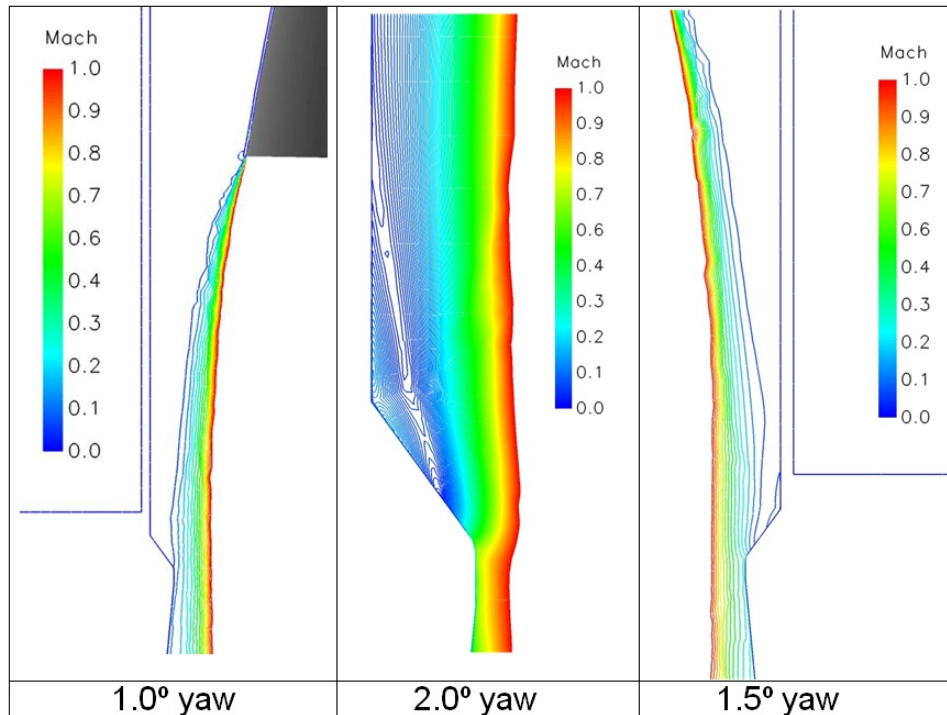


Figure 17. Shear Layer Impingement, 104.13% Pc Yaw Sensitivity Case Results

The interaction of the shear layer with the Spacer is not sufficient to result in elevated pressures on either the EC or the Spacer for the 1.0° yaw case, as shown in Figure 18. A slight elevated pressure is evident on the Spacer for the 1.5° gimbal case. For each of these sensitivity cases, data extracted from the quasi-steady CFD simulation were delivered to OSC for evaluation of the resulting thermal environment.

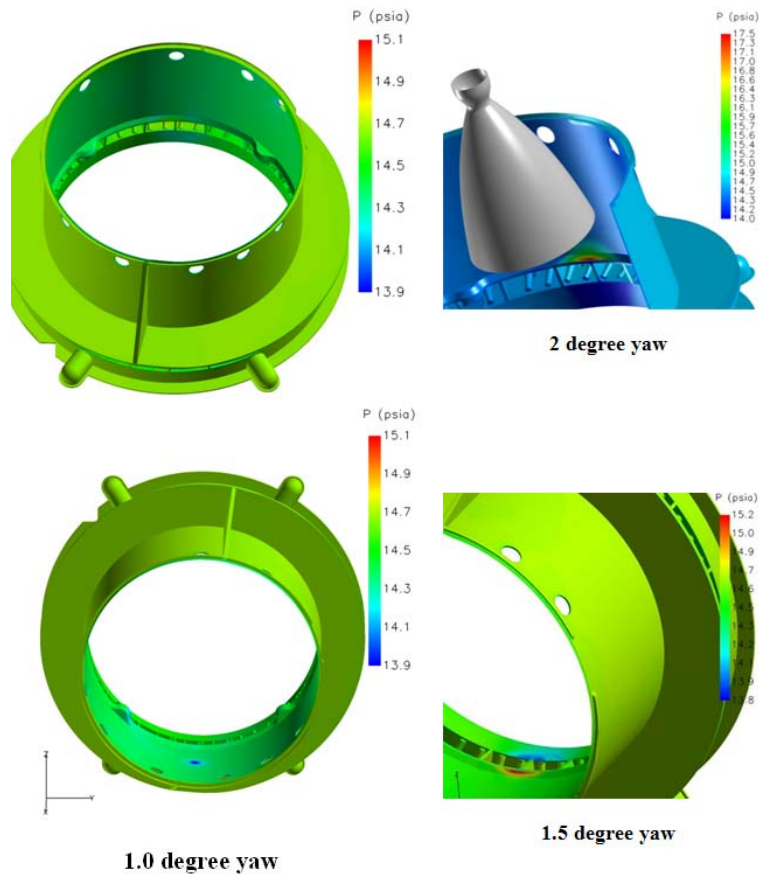


Figure 18. 104.13% Pc Yaw Sensitivity Case Results, Pressure

Subsequently, modification of the 7k Test Gimbal Profile to include an elliptical gimbal path with 2° pitch and a maximum of 1° yaw was proposed by OSC based on the results of the thermal analysis performed by OSC using the quasi-steady CFD simulation results provided to date as inputs. A set of follow-on quasi-steady CFD simulations were performed and delivered to support iterative analysis of this modified profile. These follow-on cases included a 1.25° yaw cases (including the stated mechanical uncertainty) and a 0.5° gimbal case.

To facilitate thermal analysis of the proposed elliptical gimbal path, a case with the plume directed towards the Spacer tab to the maximum extent occurring during this gimbal profile was also performed. The condition, defined and requested by OSC, was a gimbal angle of 0.55° yaw and 1.68° pitch. A steady-state simulation of this gimbal angle was performed and the extracted data provided to OSC. A summary of the major findings follows.

As shown in Figure 19, static temperatures indicate increased heating in the direction of the yaw gimbal that persists past the 45° case analyzed for the circular gimbal path. Although this case represents the closest approach to the spacer tab, the location of the increased heating persists near the $y=0$ plane. As the figure shows, it is shifted slightly towards the +y or “up deflector” side of this plane, but for purposes of estimated the elevated heating dwell time, the “towards tab” case still indicates some increased heating.

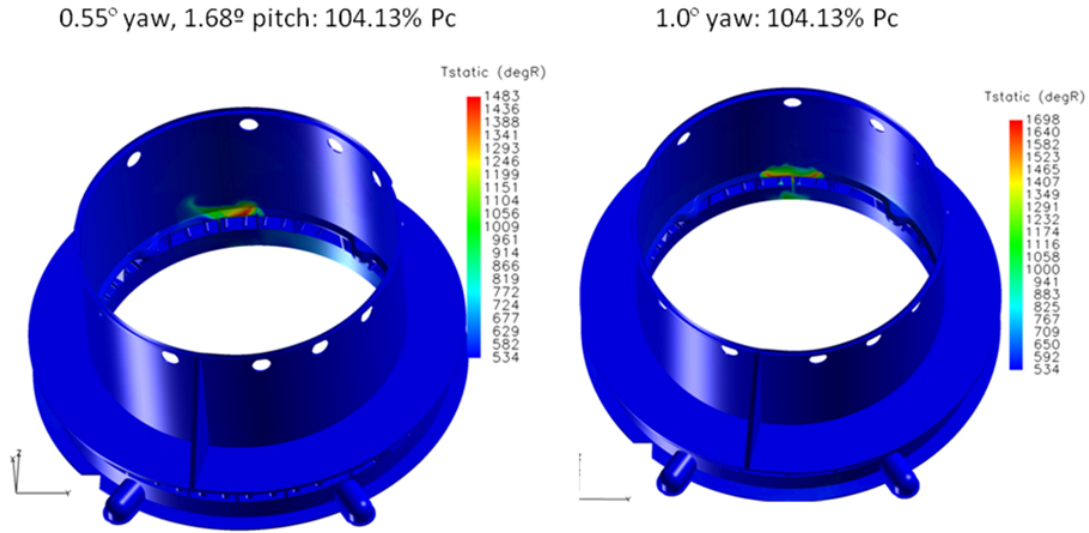


Figure 19. Elliptical Gimbal Profile Static Temperature Comparison

Although the defined angle (0.55 yaw, 1.68 pitch) is the closest approach to the tab, the plume is still significantly closer to the +z side of the spacer as illustrated by the top view of the Mach 0.25 isosurface in Figure 20.

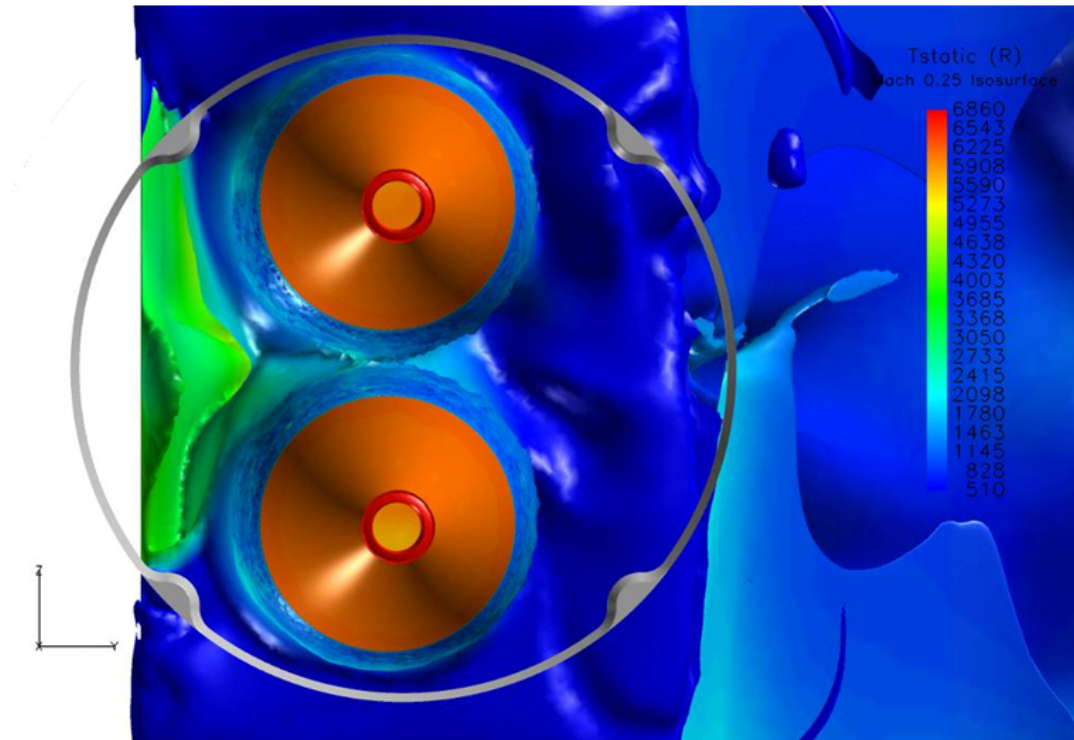


Figure 20. Mach 0.25 Isosurface for "Up Deflector, Towards Tab" Case

Unsteady Simulations

As noted previously, for all of the quasi-steady state simulations it was noted that a stable, but still unsteady flowfield was indicated. To gain additional insight into plume dynamics occurring during various key events of the 7K test profile, a series of unsteady simulations were executed. These simulations, which used the corresponding quasi-steady solution as an initial condition, were all based on static geometry for simplicity, modeling of relative motion is within the simulation capabilities, but was not executed due to the limited time available to perform the assessment. The unsteady simulations were executed with the following characteristics:

- Hybrid RANS/LES turbulence model
- Second order spatial resolution
- Second order temporal resolution

A stable, quasi-steady solution was obtained and used as the initial condition for the Hybrid RANS/LES time-accurate simulation. For this HRLES rerun, dtmax of 1e-4 was used. Recall that the quasi-steady solution for the zero gimbal cases was postulated to be driven by the convective flow with air entrained through the vent holes and aspirated from the EC through the ASR by the plumes. Water flow from the ASR is not included in either the quasi-steady or the time-accurate modeling. Figure 21 shows the starting condition static temperature on the EC (in K) from the first timestep of the unsteady simulation. An animation was created showing the dissipation of the high-heating areas on the EC as the simulation progresses and the flow conditions within the cylinder vary over time.

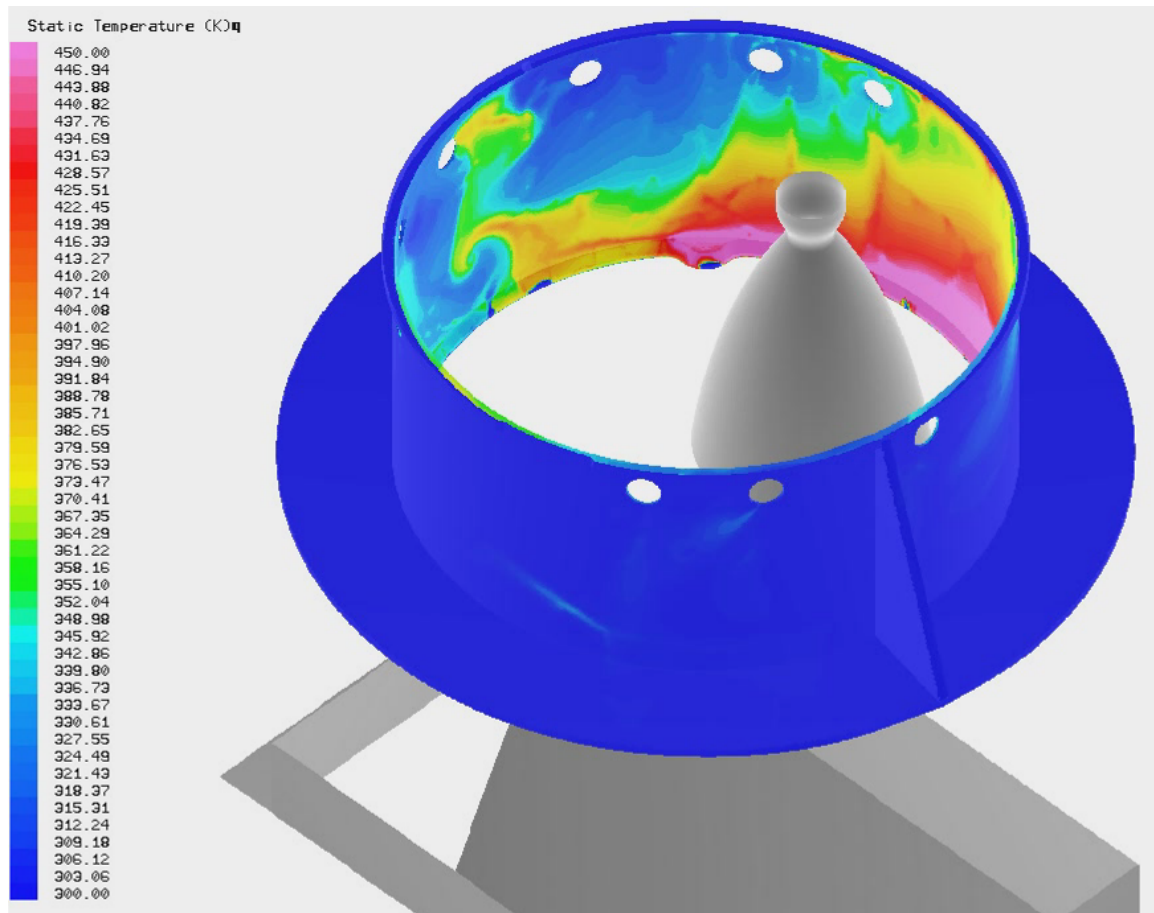


Figure 21. 108%Pc, 0 gimbal, Starting Condition for HRLES 0.001s (Static Temp, K)

As described previously, the higher heating condition on the $-y$ side of the EC is driven by asymmetric convective flow and the quasi-steady approximation. It was expected that this would result in “worst case” environments compared with an unsteady solution. In the unsteady simulation, the variability in the flowfield, assisted by the roiling of the eddies in the simulation quickly dissipates the region of elevated heating on the EC. By 0.125 s into the simulation, the temperatures are significantly reduced as seen in Figure 22, with continued dissipation through 0.300 s.

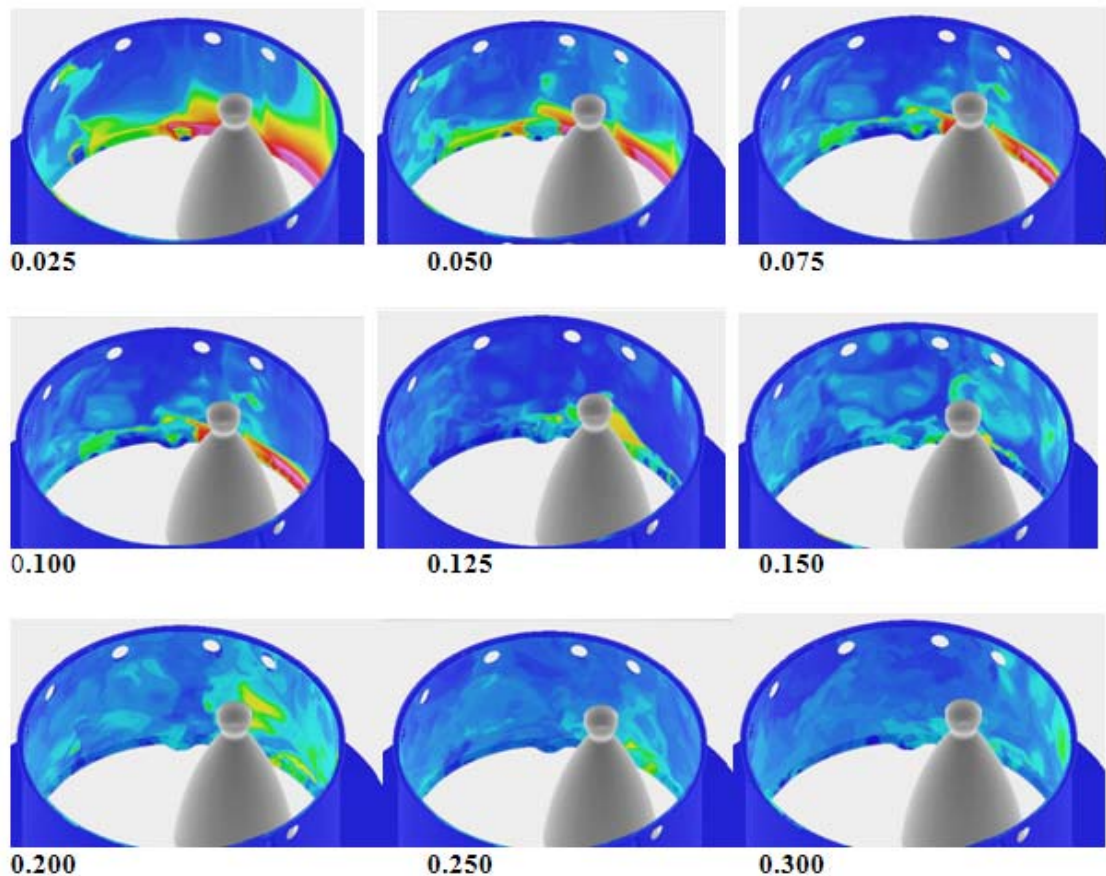


Figure 22. 108%Pc, 0 gimbal, Static Temperature Dissipation Sequence (time in s)

An unsteady HRLES simulation was also performed for various gimbal cases including the 104.13%Pc, 2° Yaw Gimbal case. In each case a stable, quasi-steady solution was obtained and used as the initial condition for the unsteady simulation. Animations were created showing the evolution of the high-heating areas on the EC at both of these scales as the simulation progresses and the flow conditions within the cylinder vary with time. Although the high heating areas away from the direction of the shear-layer impingement dissipate and become less organized early in the simulation, temperatures in the highest heating regions persist during the course of the simulations. The location and size of the “hot spot” varies cyclically with time as the flow through the venting holes varies and flow direction through the venting holes reverses, but it does not dissipate during the course of the simulation.

SUMMARY AND CONCLUSIONS

A series of 7K test models were developed including flight and test-specific mounting hardware, launch facility structures and a 9-species "wet C-O" thermodynamic model with conditions indicated by the selected segment of the planned test profile. Quasi-steady simulations using these models were performed to identify potential issues with the planned test profile and integrated configuration. Impingement pressures on the flame deflector indicated by the analysis, along with KSC structures input indicated that the reinforcement of the refractory on the flame deflector was inadequate to support the loads imparted by the plumes during the planned testing. Modifications to the anchor configuration on the flame deflector were made during the construction of the WFF Pad 0A facility as a result of the NESC recommendation.

The planned test profile included a 2° gimbals sweep (performed at 104.13%Pc) as well as a 2° gimbals "yaw-out" maneuver at the end of the test (performed at 100%Pc). A series of simulations at selected gimbals angles were performed to evaluate these segments of the planned test profile. It was found that the 2° yaw segment for each case would result in impingement of plume shear layer on test specific hardware and launch hardware mounting structures. OSC thermal analysis using CFD inputs corroborated the assessment that these environments were not sustainable with sufficient margin for the times required. The final "yaw-out" maneuver was eliminated and the 2° gimbals sweep at 104.13%Pc was reduced to an elliptical path with the yaw directions reduced to a smaller gimbals angle (1° nominal). Additional sensitivity analyses were performed to assist with the selection of these cases (1°, 1.25°, 0.5°). Thermal protection for the hardware is being sized based on these inputs.

In addition to the quasi-steady simulations, a series of representative simulations were performed using the quasi-steady solution as the initial condition for a Hybrid RANS/LES unsteady simulation. These cases provided additional insight regarding the extent to which artifacts of steady-state approximations for these simulations, particularly in areas which indicated stable but unsteady flow, were affecting the results used as inputs to the thermal analysis. As expected, for cases where the areas of high-heating were driven by asymmetric convective flow and the quasi-steady approximation, the variability in the flowfield assisted by the roiling of eddies inside the EC quickly dissipates the regions of elevated heating on the test hardware. However, for cases where the high-heating is driven by shear layer impingement, the unsteady simulation reveals that this is a persistent condition.

REFERENCES

1. Taurus II 5000/7000 Series Stage Test, Critical Design Review, 23-24 June 2010.
2. Song, Hee Jong, ER43(11-009), "Taurus II 7K Stage Test Thermal Assessment of the Extension Cylinder", April 24, 2011.
3. "Thermo-Chemical Equilibrium Program, SEA Software Inc., v. 1.6, 2003.
4. Wang, T.S., "Ten Species Kerosene-Oxygen Combustion Model", J. Thermophysics and Heat Transfer, Vol 15, No 2, April-June 2009 , p. 140-147.
5. Luke, E.A., et al., "CHEM 3: A Finite-Rate Viscous Chemistry Solver – The User Guide."
6. Vu, B.T and Smith, S.D., "Taurus II Pad 0A Flame Deflector Erosion Due to Plume Impingement", 25 October, 2010.
7. Smith, S.D., "Update to the RAMP2 Computer Program," SECA-FR-93-19, SECA, Inc., Huntsville, AL, December 1994.
8. Smith, S.D., "Unified Test Stand Design and Environmental Impact Model", PT-FR-03-01, Plumetech, Huntsville, AL, 16 July 2003.
9. Dorney, S., ER42(11-057), "CFD Analysis Activities", June 3 2011.



TAURUS II STAGE TEST SIMULATIONS: USING LARGE-SCALE CFD SIMULATIONS TO PROVIDE CRITICAL INSIGHT INTO PLUME INDUCED ENVIRONMENTS DURING DESIGN

L.L. Strutzenberg and J.S. West
NASA/MSFC/ER42
Huntsville, Alabama

**8th Modeling and Simulation Subcommittee (MSS) / 6th Liquid Propulsion Subcommittee
(LPS) / 5th Spacecraft Propulsion Subcommittee (SPS) Joint Meeting
December 5-9, 2011**

Approved for public release; distribution is unlimited



Background



- **Taurus II Vehicle**

The Commercial Crew and Cargo Project Office (CCCPO) at NASA/JSC has a Space Act Agreement (SAA) with Orbital Sciences Corporation (OSC) to develop a new rocket (Taurus II), spacecraft (Cygnus), and a new Pad 0A launch stand at Wallops Flight Facility (WFF) to fly a demonstration mission to the International Space Station (ISS).

- **7K (Hot-Fire) Test Sequence**

OSC is designing the launch facility to accommodate a series of tests using the flight first stage on Pad 0A prior to the launch of Taurus II. The hot-fire test sequence, referred to as the “7K test”, consists of an approximately 30-second static firing of the two first stage AJ26-62 engines.

- **NESC Assessment**

- CCCPO requested that the NESC independently assess this plan and identify improvements in the test planning, design or operations to reduce overall risk of the testing.
- The NESC assessment included evaluation of the environments that the flight hardware, test hardware, and launch pad would be subjected to during the test campaign.
- This presentation includes an overview of the CFD modeling approach taken to support the assessment and highlights of significant results obtained.



Purpose of Work



- **Taurus II Stage Test CFD Simulations**

- Targeted Loci/CHEM CFD simulations were performed to evaluate the effects of a hot-fire test on an evolving integrated launch pad/test article design.
- This presentation provides an overview of the CFD modeling approach taken to support the assessment in a timely manner and highlights of significant results obtained.

- **Pathfinder Simulation**

- Model was intended to assess adequacy of data provided for execution of simulations.
- This simple model was sufficient to provide quick-turn around evaluation of plume impingement pressures on the flame deflector.
- Results from this simulation were available in time to provide data for an ongoing structural assessment of the deflector.
- Recommendation was available in a timely manner and incorporated during construction of the new launch stand at Wallops Flight Facility Pad 0A.

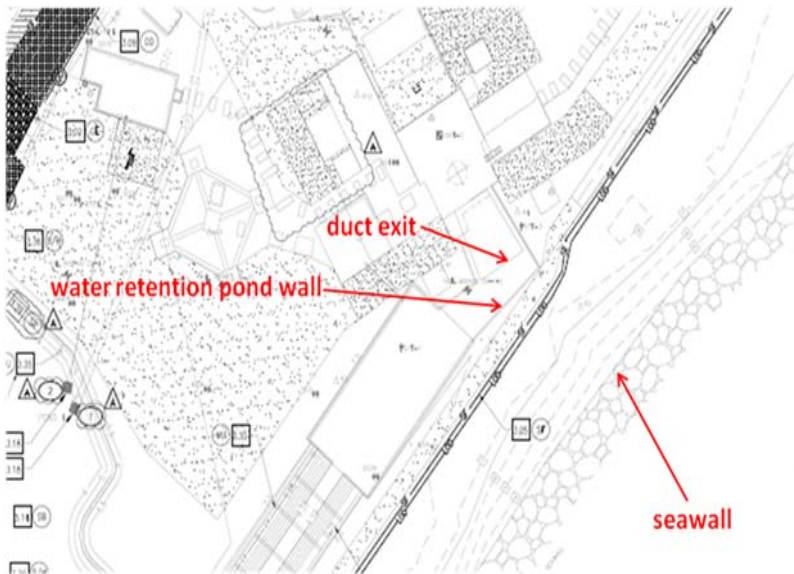
- **7K Test Series Simulations**

- A series of Reynolds-Averaged Navier-Stokes (RANS) quasi-steady simulations representative of various key elements of the test profile was performed to identify potential concerns with the test configuration and test profile.
- Unsteady Hybrid-RANS/LES simulations were performed as required to provide additional insight into critical aspects of the test sequence.
- Modifications to the test-specific hardware and facility structures thermal protection as well as modifications to the planned hot-fire test profile were implemented based on these simulation results.

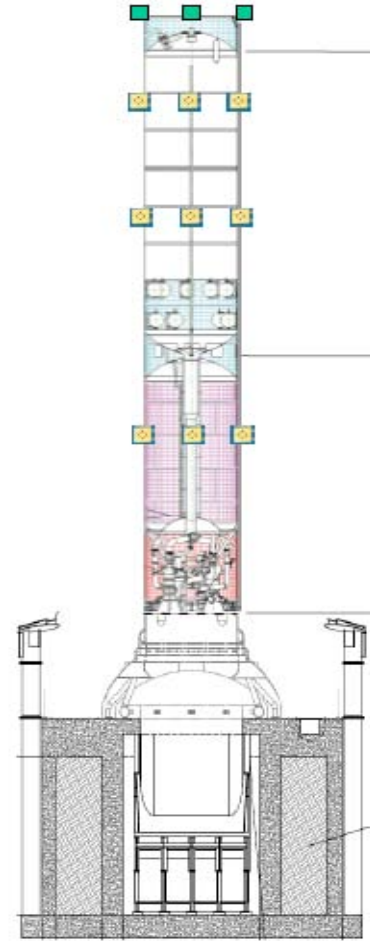
Taurus II Configuration at WFF Pad 0A



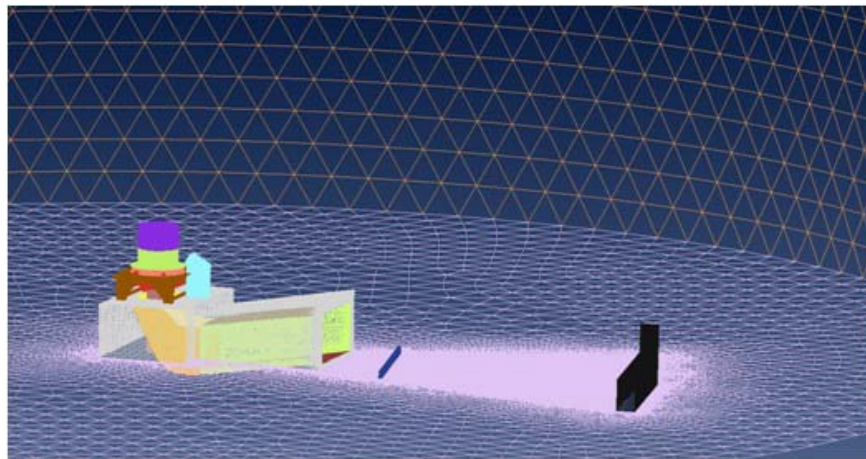
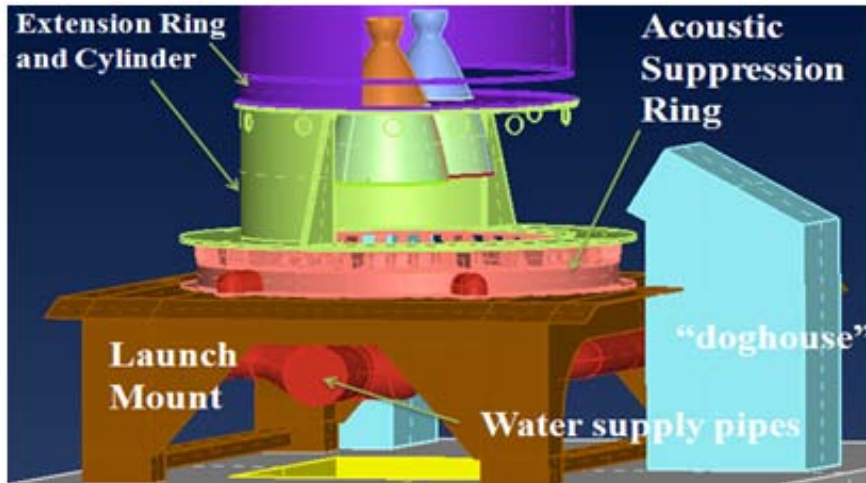
Artist Depiction of Launch Configuration



Plume Downstream Structures

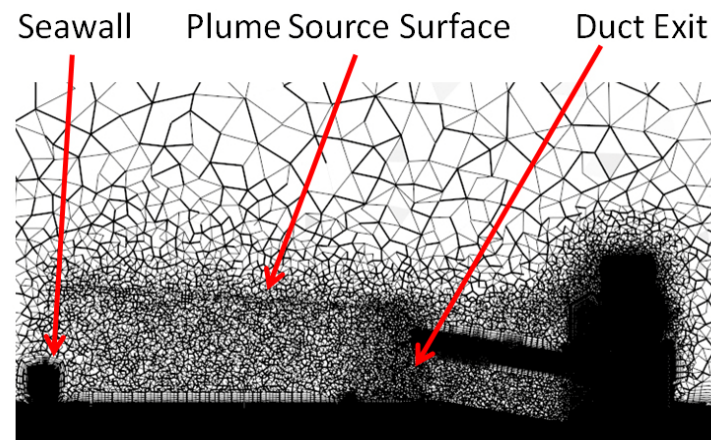
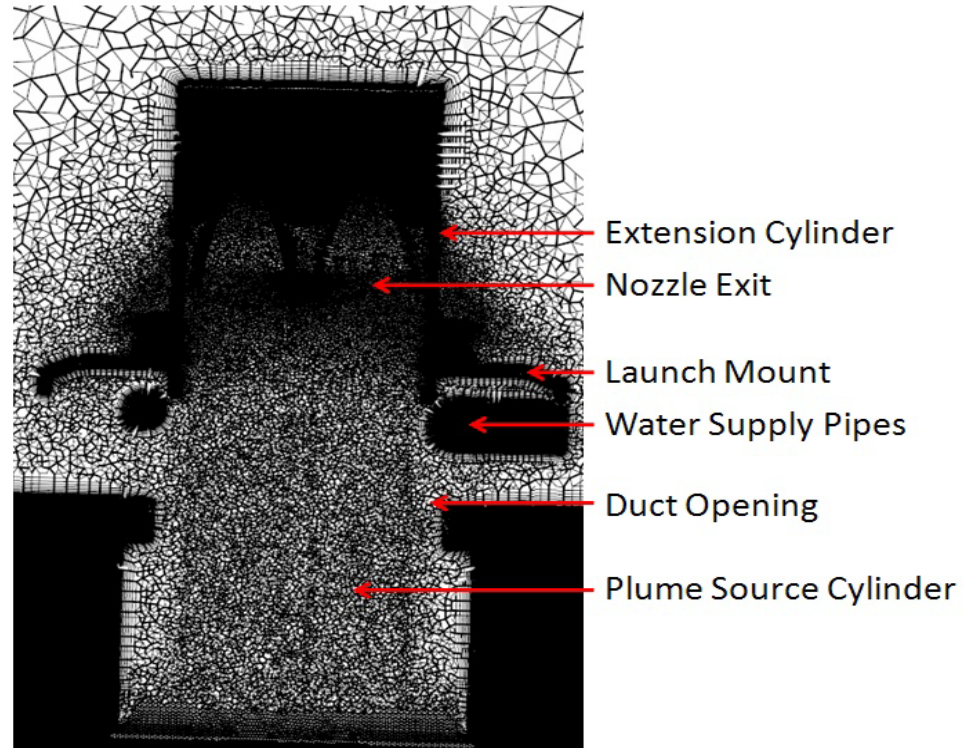


7K Test Configuration

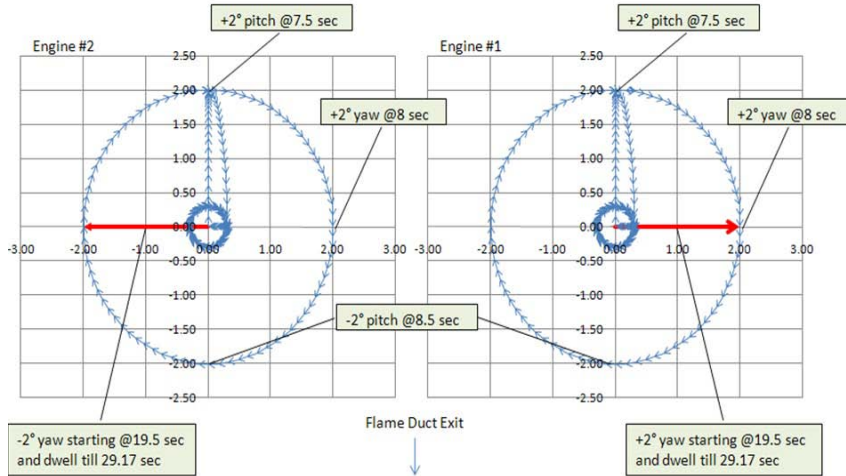


- Launch Mount (LM), Acoustic Suppression Ring (ASR), “doghouses” and water supply pipes depicted consistent with the flight hardware geometry
- Extension Cylinder (EC) and Extension Ring (ER) added to provide structural support for the 7K test
- ER connected to a flight-like aft bay casing
- Distances to the water retention pond wall and seawall from the duct exit scaled from provided drawings
- Exaggerated seawall angle was retained to allow assessment of the sensitivity of these measurements to the uncertainty inherent in the model development
- External ground surface meshed to capture the “water retention pond wall” with a 6” scale and the seawall with a 12” scale
- Farfield meshed at 12’ scale

- Significant geometric fidelity imported directly from the provided CAD model retained in proximity to plume
- Fidelity was preserved in areas affecting plume entrainment (particularly significant blockages) and in areas with potential plume impingement during gimbals maneuvers.
- Model intended to provide approximate impingement pressures and velocities
- Significant additional geometric fidelity as well as inclusion of deluge water would be required to provide truly predictive environments.
- Plume source cylinders added to retain mesh clustering in the regions directly downstream of the nozzles
- Plume source surface added to retain mesh clustering in both the upper and lower shear layers of the plume downstream of the duct



30-second static firing including a time sequenced thrust and gimbaling profile

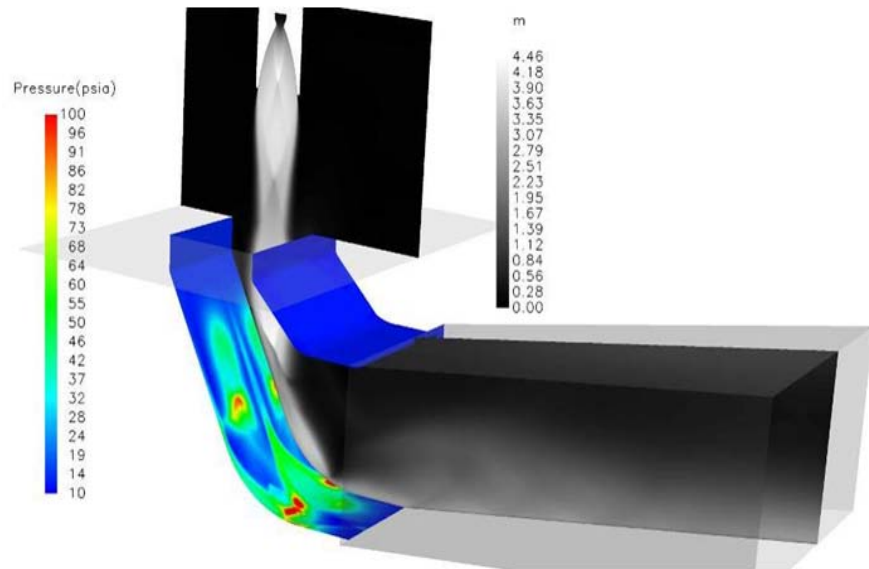
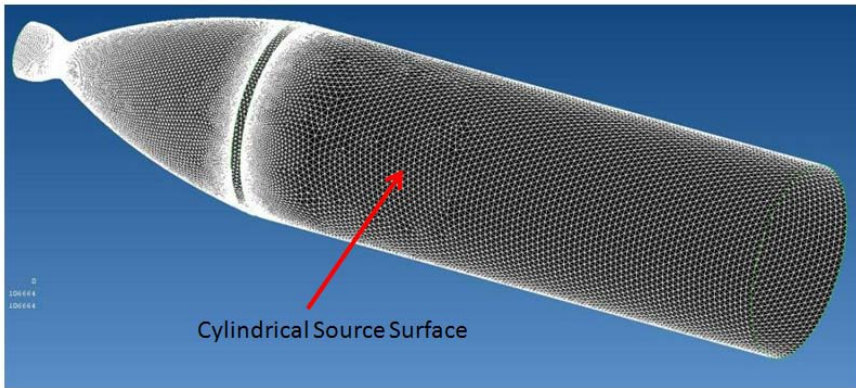


- 7K Test commanded thrust profile provided along with the associated baseline gimbaling sequence was used to define a series of quasi-steady RANS simulations characterizing the interaction of the plume flow field with the mounting hardware and launch pad structures.
- The objectives of these simulations were:
 - 1) to characterize the environment on and around the attach structure including the LM, Spacer, ASR and EC and formulate recommendations to alter operations or design to mitigate adverse effects,
 - 2) to characterize plume impingement pressures on the flame deflector to support recommendations regarding flame deflector design, and
 - 3) to identify key profile segments requiring additional analysis, including unsteady simulations.
- Nozzle inlet boundary condition derived from the Thermo-Chemical Equilibrium Program (TEP) output
- Nine-species “wet C-O” thermodynamic and finite rate chemistry model used to model evolving chemistry
- A slug flow velocity profile was used at the nozzle inlet with assumed (as opposed to estimated) turbulent intensity values

Definition of Initial Full-Model Quasi-Steady RANS Simulations

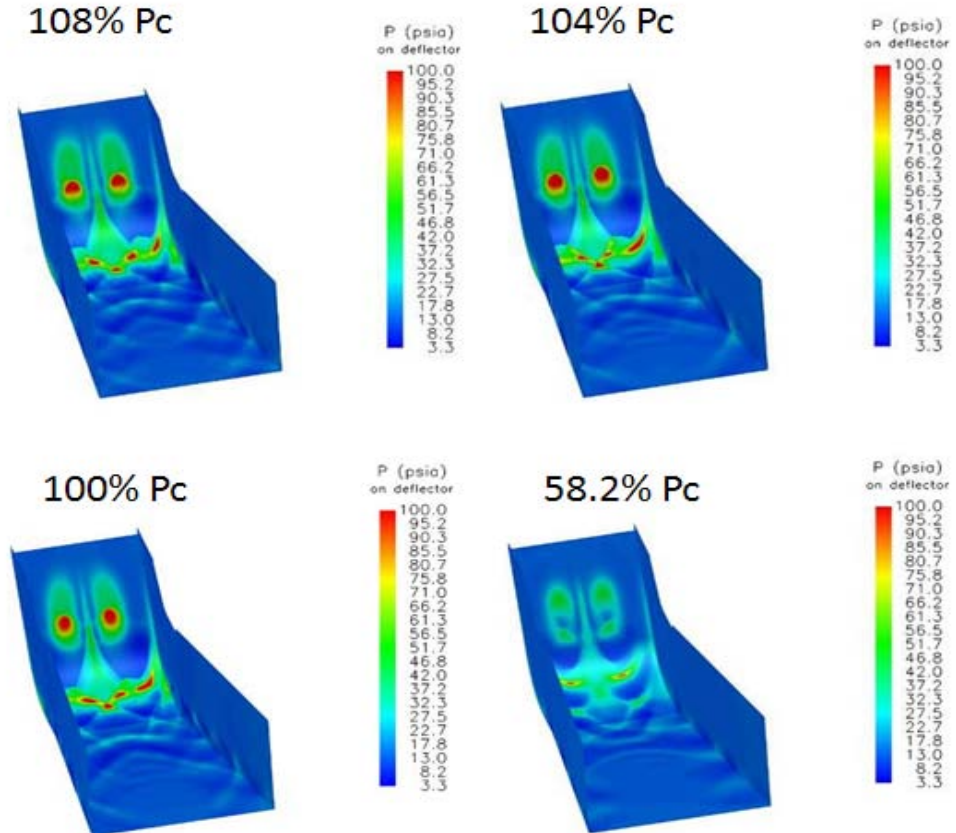
Case Name	Gimbal Angle	%Pc	Case Description
g0v4_108	0	108	Zero gimbal, 108%Pc
g0v4_582	0	58.2	Zero gimbal, 58.2% Pc
g0v4_10413	0	104.13	Zero gimbal, 104.13%Pc
g0v4_100	0	100	Zero gimbal, 100%Pc
2gyaw_10413	Yaw +2	104.13	2° gimbal towards duct side
2gup_10413	Pitch +2	104.13	2° gimbal towards deflector top
2gdn_10413	Pitch -2	104.13	2° gimbal towards deflector base
141gup_10413	Pitch +1.41 Yaw +1.41	104.13	Up deflector towards corner
141gdn_10413	Pitch +1.41 Yaw +1.41	104.13	Down deflector towards corner
2gyout_582	Pitch +1.41 Yaw +1.41	58.2	Both engines “yaw-out” towards duct walls
2gyout_100	Pitch +1.41 Yaw +1.41	100	Both engines “yaw-out” towards duct walls

In parallel with the development of the full model geometry, quasi-steady single-plume and dual-plume pathfinder cases were performed using the general purpose CFD program Loci/CHEM (version 3.2-pre1)



- Primary objective was to determine whether sufficient information was available to perform CFD simulations of the AJ26 plumes interacting with the 7K test hardware and launch pad structures.
 - Nozzle curvature resolved with max 3° change in triangle surface normal from geometry curvature
 - Nozzle lip resolved with 0.2" spacing
 - Flame deflector was resolved with 2" isotropic spacing
 - Smooth transition from the nozzle lip to the deflector achieved by inclusion of a cylindrical source surface
- The mesh resolution was shown to be sufficient to reveal shock structure down to the deflector.
- High pressures in two streamwise locations (initial and secondary turning locations)
- Static pressures (~100 psia) consistent with PLIMP results for the primary impingement region
- Plume “splash” effects result in a secondary high pressure impingement region for each plume
- Gimbal effects and dithering due to plume unsteadiness also considered in definition of four regions of the deflector to be considered for high pressure impact environments

- **Recommendation provided within a few weeks of problem definition**
- **Refractory anchoring system modified with minimal impact to cost and schedule**

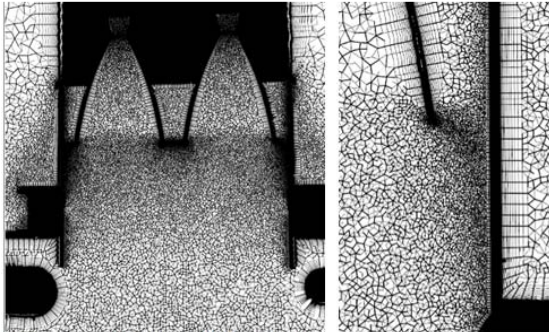


- These four zero-gimbal cases represent segments of the 7K test profile, and are also useful for comparison with various gimbal cases.
- A comparison of the pressures on the deflector resulting from the zero gimbal simulations indicates that the environments derived from the pathfinder case were reasonable and conservative.

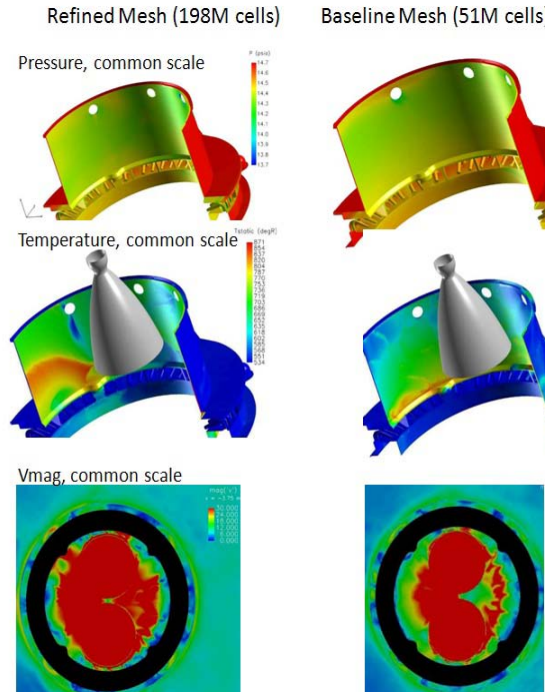
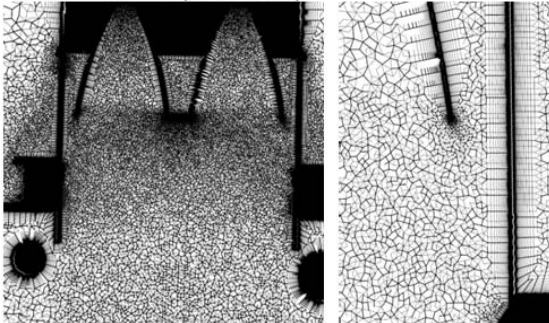
Quasi-steady simulations were performed for four RPL conditions:
108% Pc, 58.2% Pc, 100% Pc and 104.13% Pc

A mesh sensitivity assessment was performed using the 108% Pc zero gimbal case. Due to the very limited timeframe for the assessment, a thorough sensitivity study was not feasible. However, key areas with close proximity to the plume were refined significantly and compared with the baseline mesh results.

Refined Mesh (198M cells)



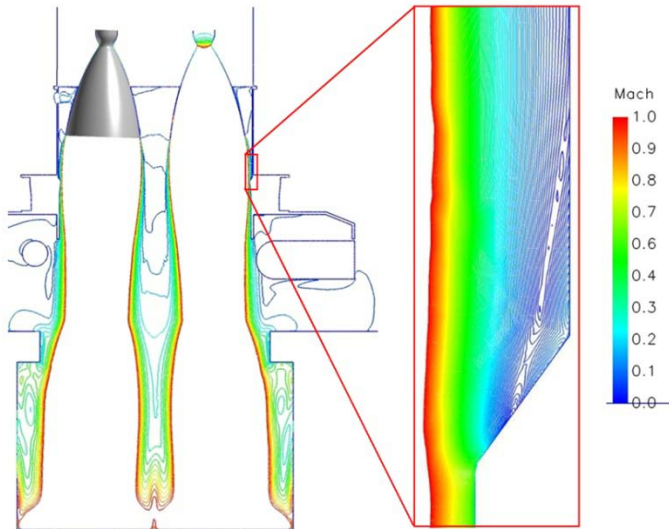
Baseline Mesh (51M cells)



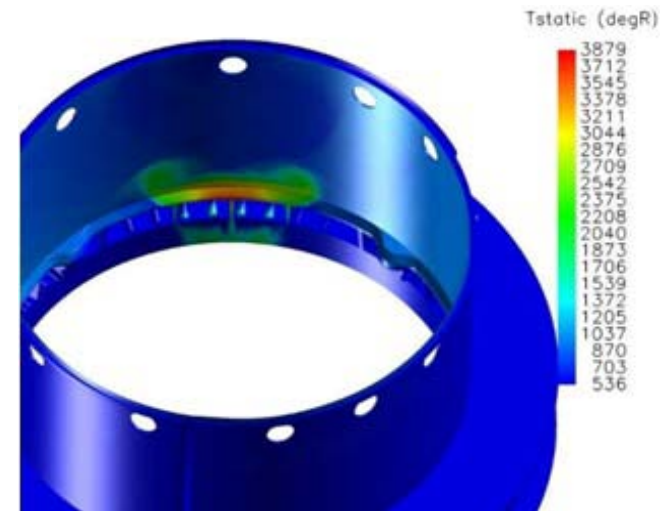
- EC pressures similar indicating plume aspiration driven convective pressure differentials.
- EC static temperatures on the EC similar in magnitude and within the expected variation for convection driven heating
- Variation between asymmetric convective flow patterns results in differences in EC heating
- Both cases are “quasi-steady” and include artifacts of this assumption.

- Sensitivity of the provided results to this level of mesh refinement within the expected error induced by quasi-steady limitations, environmental uncertainty and geometric and configuration approximations.
- Each gimbal-specific case required a new mesh to be generated. Mesh sizes varied for each case, but were maintained within the range covered by this sensitivity assessment.

A quasi-steady simulation was performed to provide input to a thermal assessment of the yaw-out maneuver at the end of the test. The baseline timeline included potential overlap between the 100% RPL chamber pressure and the completion of the yaw-out maneuver.



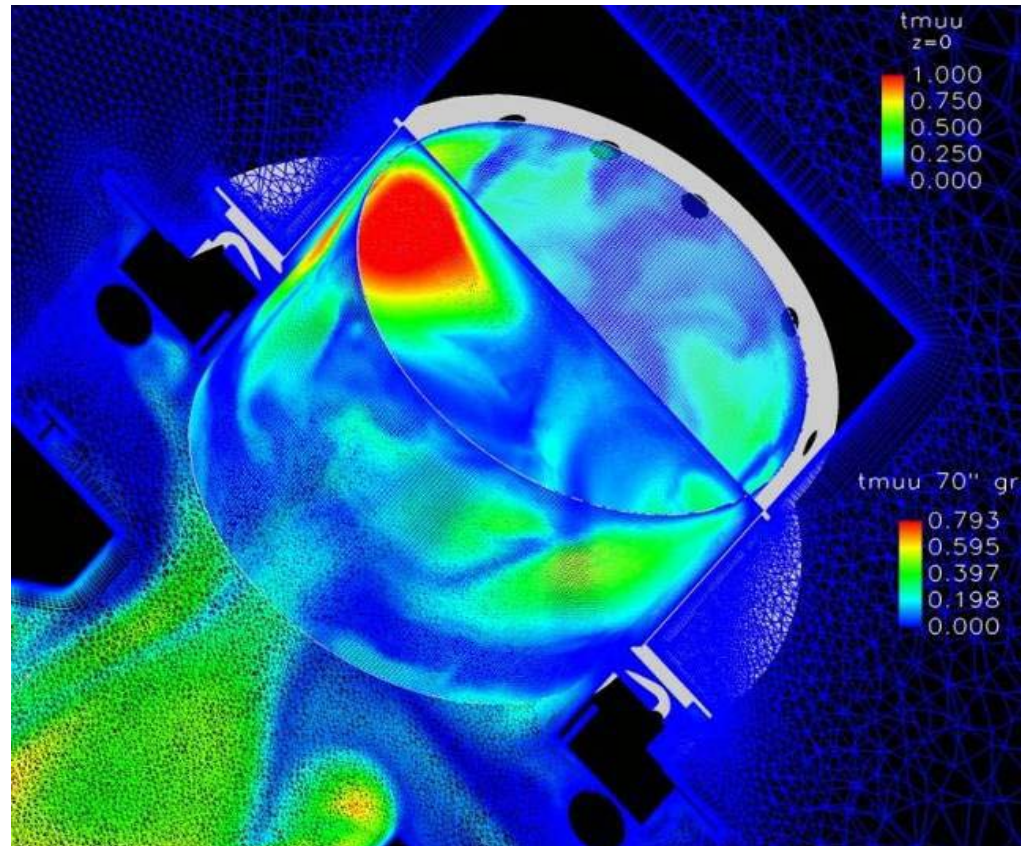
Shear Layer Impingement



Static Temperatures

- The 100 % Pc “2° yaw-out” simulation indicates shear layer impingement on the ASR, the Spacer, and the EC.
- The static temperature on the EC also indicates high localized heating resulting from the proximity of the plume to the EC and spacer surfaces.

- To perform thermal analysis using the quasi-steady CFD as an input, OSC requested that the CFD data be mapped to 70", 73", 74" and 75" cylindrical structured grids.
- Requested data were provided in three datasets:
 - Q1 (r, U, V, W, t)
 - Q2 (fO2, fH2O, fCO, fCO2, fN2)
 - Q3 (cp, tmuu, tkconduct, kconduct, pg)
- Requested datasets were also mapped to 2D meshes along $z=0$ and $y=0$ cutting planes with boundary layer grid refinement as required.



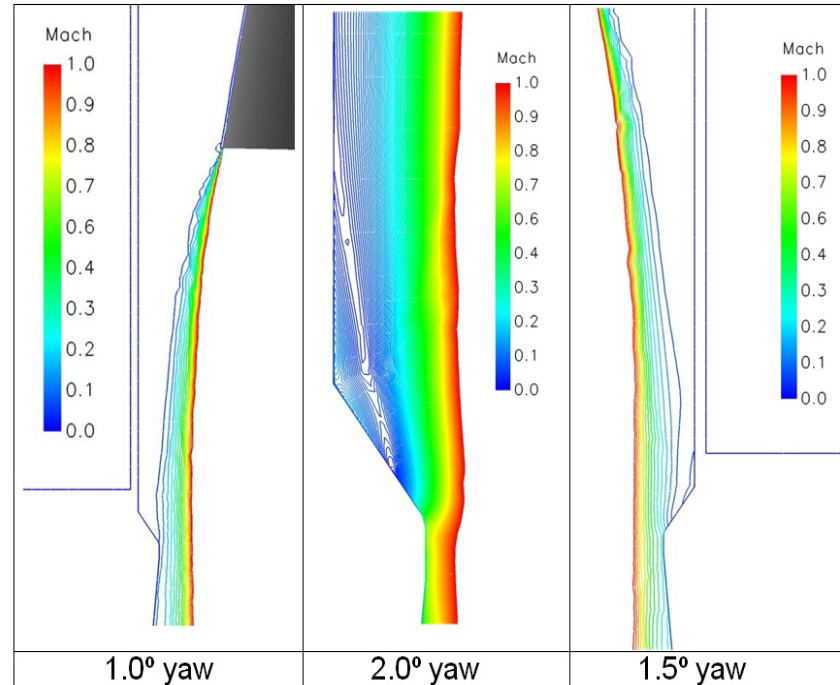
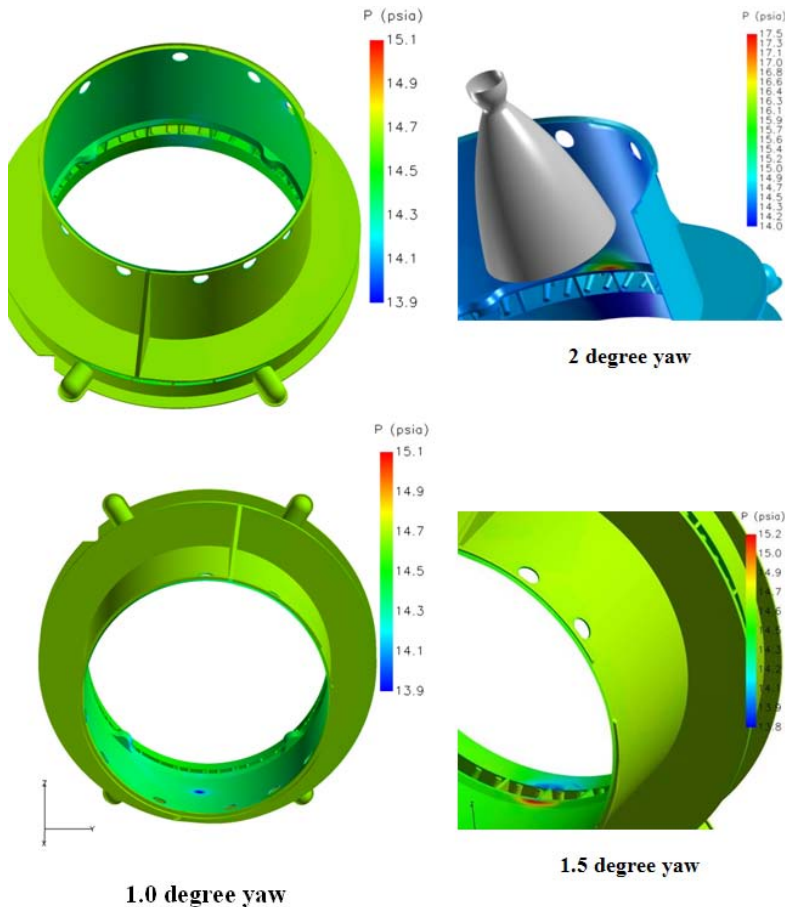
- Example of extracted data visualization: turbulent eddy viscosity extracted onto the 70" cylindrical structure grid intersecting a $z=0$ cutting plane in the unstructured solution file.
- This type of visualization was performed to qualitatively evaluate data extraction module performance prior to delivery of the extracted data to OSC.

Case	P (psia) normalized to 100	P (psia) local max/min scale	Mach 1 isosurface contour
+2° pitch “up deflector”			
45° “up deflector”			
+2° yaw			
45° “down deflector”			
-2° pitch “down deflector”			

Key results of the quasi-steady gimbal sweep simulations include:

- Indication of plume shear layer impingement and significant flow blockage for each of the three cases with absolute value of 1.41° and greater gimbal in the yaw direction
- High heating persists in proximity to the $y=0$ plane on the $-z$ side of the EC for at least 0.5 s during the gimbal sweep as the plume moves around the circular gimbal path and down the deflector
- Similarly high heating profile expected to occur on the $+z$ side of the EC although the unsteady flow characteristics will differ, primarily due to the direction of the plume motion (up the deflector).
- For each of these “high heating” cases, a set of CFD outputs was provided to OSC for detailed thermal analysis

Following the delivery of 2° yaw case results, OSC indicated a potential modification to the 7K Gimbal Profile may be possible while still meeting the intent of the requirements of the test. Supporting analysis for this decision included performing simulations of 1.0° and 1.5° yaw cases for comparison to the 2.0° yaw case previously delivered.

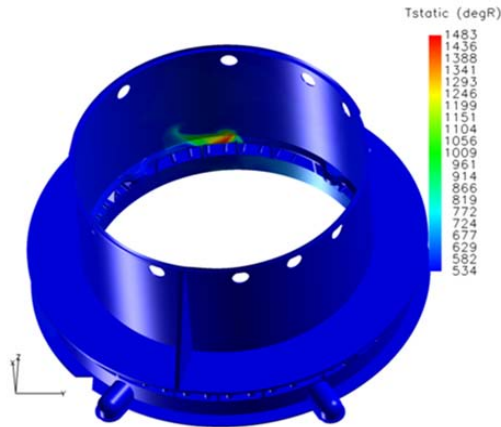


- Plume interaction not sufficient to result in elevated pressures on either the EC or the Spacer for the 1.0° yaw case.
- A slight elevated pressure is evident on the Spacer for the 1.5° gimbal case.
- For each of these sensitivity cases, data extracted from the quasi-steady CFD simulation were delivered to OSC for evaluation of the resulting thermal environment.

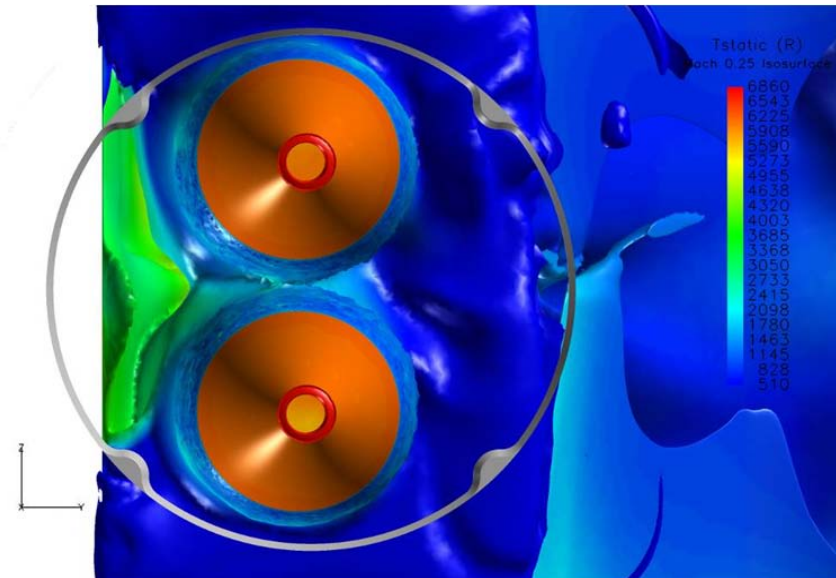
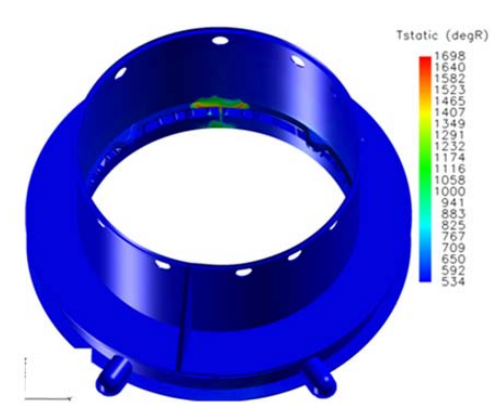
Modification of the 7k Test Gimbal Profile to include an elliptical gimbal path (2° pitch, 1° yaw) was proposed by OSC based on the thermal analysis results.

- Additional simulations were performed to assess this modified baseline profile:
 - 1.25° yaw cases (including the stated mechanical uncertainty)
 - 0.5° gimbal case
 - “towards Spacer tab” cases
- Static temperatures indicate increased heating in the direction of the yaw gimbal that persists past the 45° case analyzed for the circular gimbal path.
- The location of the increased heating persists near the $y=0$ plane through the “towards tab” case.
- Although the “towards tab” defined angle (0.55 yaw, 1.68 pitch) is the closest approach to the tab, the plume is still significantly closer to the $+z$ side of the spacer as illustrated by the top view of the Mach 0.25 isosurface.

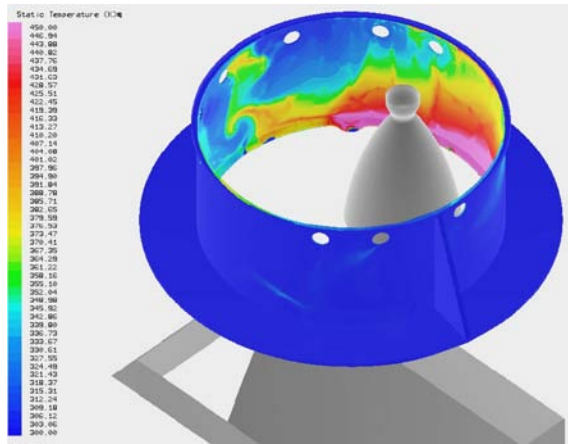
0.55° yaw, 1.68° pitch: 104.13% Pc



1.0° yaw: 104.13% Pc

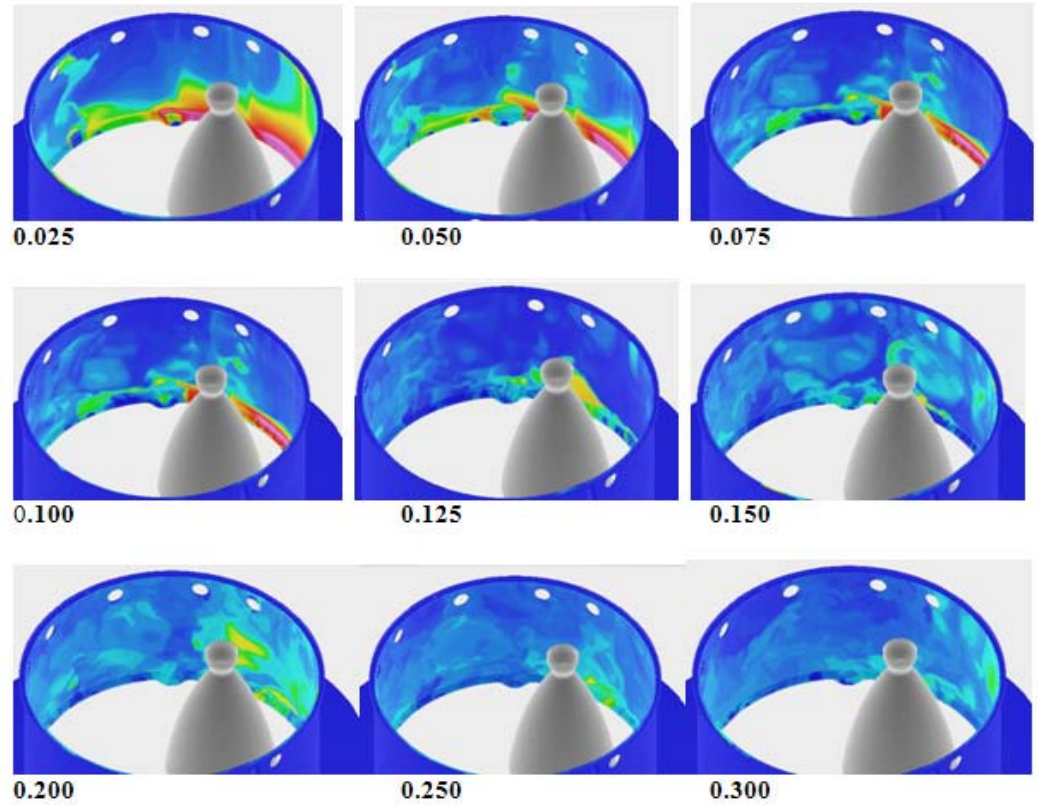


Mach 0.25 Isosurface for “Up Deflector, Towards Tab”



Recall that the quasi-steady solution for the zero gimballed cases was postulated to be driven by the convective flow with air entrained through the vent holes and aspirated from the EC through the ASR by the plumes. Water flow from the ASR is not included in either the quasi-steady or the time-accurate modeling.

This sequence shows the starting condition static temperature on the EC (in K) from the first timestep of the unsteady simulation. An animation was created showing the dissipation of the high-heating areas on the EC as the simulation progresses and the flow conditions within the cylinder vary over time.



The prior higher heating condition on the $-y$ side of the EC was driven by asymmetric convective flow and the quasi-steady approximation. It was expected that this would result in “worst case” environments compared with an unsteady solution.

In the unsteady simulation, the variability in the flowfield, assisted by the roiling of the eddies in the simulation quickly dissipates the region of elevated heating on the EC.

This result is in contrast to “impingement” cases in which the location and size of the “hot spot” varies cyclically with time as the flow through the venting holes varies and direction reverses, but it does not dissipate during the course of the simulation.



Summary and Conclusions



- **Quick turn-around Pathfinder model results were provided quickly and influenced the construction of the WFF Pad 0A facility as a result of the NESC recommendation.**
- **Modifications to the test-specific hardware and facility structures thermal protection as well as modifications to the planned hot-fire test profile were implemented based on quasi-steady simulation results.**
 - 2° yaw segment for each case would result in impingement of plume shear layer on test specific hardware and launch hardware mounting structures.
 - The final “yaw-out” maneuver was eliminated and the 2° gimbal sweep at 104.13%Pc was reduced to an elliptical path with the yaw directions reduced to a smaller gimbal angle.
 - Additional sensitivity analyses were performed to assist with the selection of these cases (1°, 1.25°, 0.5°). Thermal protection for the hardware is being sized based on these inputs.
- **Unsteady Hybrid-RANS/LES simulations were performed as required to provide additional insight into critical aspects of the test sequence.**
 - For cases where the areas of high-heating were driven by asymmetric convective flow and the quasi-steady approximation, the variability in the flowfield assisted by the roiling of eddies inside the EC quickly dissipates the regions of elevated heating on the test hardware.
 - For cases where the high-heating is driven by shear layer impingement, the unsteady simulation reveals that this is a persistent condition.