

Computational Modeling of Mars Retropropulsion Concepts in the Langley Unitary Plan Wind Tunnel

Karl T. Edquist* Stephen J. Alter† Christopher E. Glass‡
Bil Kleb§ Ashley M. Korzun¶ William A. Wood||
NASA Langley Research Center, Hampton, Virginia 23681

Francisco Canabal**
NASA Marshall Space Flight Center, Huntsville, Alabama 35808

Robert E. Childs†† Logan D. Halstrom‡‡ Kristen V. Matsuno‡‡
NASA Ames Research Center, Moffett Field, California 94035

Future human Mars missions will require powered descent starting at supersonic conditions, something which has never been done before at Mars. Computational powered descent flowfield simulations have been completed at full-scale Mars conditions, but the available ground test data are not appropriate for calibrating computational uncertainties for aerodynamic interference on proposed Mars descent vehicles. A test will be conducted in the NASA Langley Unitary Plan Wind Tunnel to begin addressing powered descent aerodynamics risks for large-scale human Mars entry concepts and to identify gaps in computational predictive capabilities. This paper covers pre-test computational flowfield predictions of two different models derived from full-scale reference vehicles: a blunt low lift-to-drag vehicle and a more slender geometry. Calculations of the blunt model include variations in nozzle configuration: nozzle location, size, area ratio, and pointing direction. There exist some significant differences between solvers, but some general trends are observed from simulations of the blunt model. First, aerodynamic axial force from the heatshield decreases with increasing thrust due to expanding plume blockage. Second, nozzles that point along the model axis result in lower aerodynamic axial force compared to nozzles that have a radial thrust component. Finally, placing the nozzles further away from the model nose preserves more heatshield axial force with increasing thrust compared to nozzles that are closer to the nose. For the slender model, the axial force from the heatshield is similar to the non-blowing axial force regardless of thrust magnitude, due to the nozzle arrangement on the heatshield. Once the test is completed, direct comparisons between the computations and test data will be made to determine computational uncertainties in a wind tunnel environment, to identify gaps in predictive capabilities, and to inform planning for future ground and flight test programs for Mars powered descent vehicles.

*Aerospace Technologist, Atmospheric Flight and Entry Systems Branch, AIAA Associate Fellow, karl.t.edquist@nasa.gov.

†Aerospace Technologist, Aerothermodynamics Branch, AIAA Member.

‡Aerospace Technologist, Aerothermodynamics Branch, AIAA Senior Member.

§NASA Senior Researcher for Computational Aerothermodynamics, Aerothermodynamics Branch, AIAA Associate Fellow.

¶Aerospace Technologist, Atmospheric Flight and Entry Systems Branch, AIAA Senior Member.

||Aerospace Technologist, Aerothermodynamics Branch, AIAA Associate Fellow.

**Aerospace Engineer, Aerosciences Branch, AIAA Senior Member.

††Aerospace Engineer, Science and Technology Corporation, Systems Analysis Office.

‡‡Aerospace Engineer, Systems Analysis Office, Member.

Nomenclature

Symbols

A_e	nozzle exit area	in^2
A_{ref}	aerodynamic reference area = projected heatshield frontal area	in^2
A_t	nozzle throat area	in^2
C_F	aerodynamic force coefficient = $F/\bar{q}A_{ref}$	
C_p	pressure coefficient = $(p - p_\infty)/\bar{q}A_{ref}$	
C_T	vacuum thrust coefficient = $T/\bar{q}A_{ref}$	
F	aerodynamic force	lb_f
L/D	aerodynamic lift-to-drag ratio	
M_e	nozzle exit Mach number	
M_∞	freestream Mach number	
p_c	nozzle total pressure	$psia$
p_e	nozzle exit static pressure	$psia$
$\bar{q}$	dynamic pressure, $\frac{1}{2}\rho_\infty V_\infty^2$	lb_f/ft^2
R_n	nozzle exit radial location	in
R_b	heatshield radius	in
Re_∞	tunnel Reynolds number	$1/ft$
T	thrust	lb_f
T_c	nozzle total temperature	$^\circ F$
T_0	tunnel total temperature	$^\circ F$
V_∞	freestream velocity	m/s
α	angle of attack	deg
ρ_∞	freestream density	kg/m^3
θ_{cant}	angle between nozzle axis and model longitudinal axis	deg

Acronyms

AFLR3	Advancing Front with Local Reconnect
AMR	automatic mesh refinement
CFD	computational fluid dynamics
DDES	delayed detached eddy simulation
EDL	entry, descent, and landing
FUN3D	Fully Unstructured Navier-Stokes Three-Dimensional
HIAD	Hypersonic Inflatable Aerodynamic Decelerator
HLLC	Harten-Lax-van Leer-Contact
HLLE++	Harten-Lax-van Leer-Einfeldt
HPA	high pressure air
LSGS	line symmetric Gauss-Seidel
LUPWT	NASA Langley Unitary Plan Wind Tunnel
NASA	National Aeronautics and Space Administration
OVERFLOW	OVERset Grid FLOW Solver
QCR	Quadratic Constitutive Relation
SRP	supersonic retropropulsion
SST	shear stress transport
URANS	unsteady Reynolds-Averaged Navier-Stokes

I. Introduction

OVER the last several years, NASA studies of human-scale Mars entry, descent, and landing (EDL) have identified technologies to enhance or enable delivering payloads to Mars that are much larger than is currently possible.^{1–7} One of the enabling technologies is the use of retrorockets, starting at supersonic conditions, in place of a parachute. All studies show that supersonic retropropulsion (SRP) is an enabling Mars descent technology for payloads larger than approximately five metric tons. Supersonic parachutes have been used for all of NASA’s successful scientific robotic missions to Mars, but parachutes are not scalable for human exploration payloads (approximately 20 metric tons). Powered flight has been successfully executed at Mars subsonic conditions, but it has never been needed at supersonic speeds. The interactions between the retrorocket exhaust plumes and surrounding flowfield result in aerodynamic interference forces and moments that often are more difficult to predict than unpowered aerodynamics. The uncertainties in powered descent AI on entry vehicle stability, control, and aeroheating are not well understood within the aerosciences community. Thus, predictive CFD uncertainties must be investigated in much more detail than they have been to date, given how significantly they will factor into the overall EDL risk and future mission success for landing humans on Mars.

The current NASA Mars entry system concepts are shown in Figure 1. The reference vehicles are identified by the aerodynamic lift-to-drag ratio (L/D) that they can generate during the hypersonic unpowered entry phase of EDL. The first reference vehicle (Low- L/D) is based on a Hypersonic Inflatable Aerodynamic Decelerator (HIAD) aeroshell that can generate lift-to-drag ratios similar to what the rigid Mars robotic aeroshells can generate ($L/D=0.24$ for the Mars Science Laboratory aeroshell at an angle of attack of about 16 degrees). The second reference vehicle (Mid- L/D) is based on a more slender rigid aeroshell that has body flaps and can generate a higher lift-to-drag ratio ($L/D=0.54$ at a hypersonic trim angle of attack of 55 degrees). Both vehicles have been analyzed assuming a total of eight liquid oxygen and methane (LO_2/CH_4) gas generator cycle engines, each producing approximately 100 kN of thrust, to execute a powered descent phase that ends in landing within 50 meters of the target site.⁵

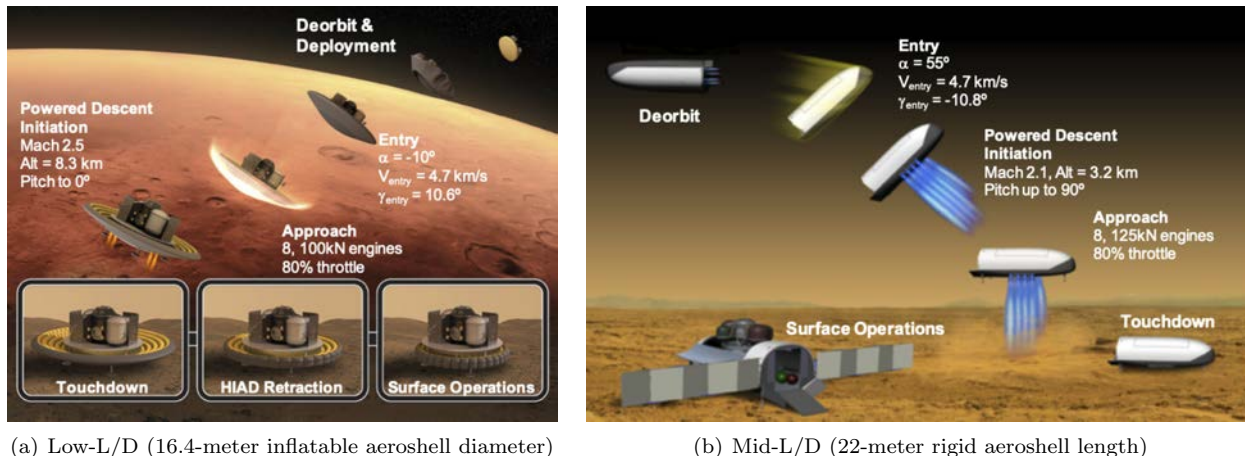


Figure 1. Concepts for human Mars entry systems.⁷

Wind tunnel testing, flight testing, and computational fluid dynamics (CFD) analysis all will play a critical role in advancing atmospheric powered descent aerosciences. A previous project completed two sub-scale SRP wind tunnel tests^{8,9} and companion flowfield analyses^{10–12} that showed promising results in the ability of CFD to capture the primary SRP flowfield features. A wind tunnel test will be conducted in the NASA Langley Unitary Plan Wind Tunnel (LUPWT) to address some of the limitations of past ground test data and to provide a richer data set against which to assess CFD models. First, the model geometries themselves are based on the current full-scale Mars concepts, both in terms of the heatshield outer mold line and the engine configuration. Second, the Low- L/D models were designed to explore engine parameters that are expected to influence the aero-propulsive interactions: nozzle size, area ratio, cant angle (angle of nozzle axis

relative to vehicle axis), and location on heatshield. A new flow-through six-component balance prototype was added to the test to provide the first known force and moment measurements in a SRP wind tunnel test. Finally, pressure-sensitive paint and discrete pressure measurements will be used to help explain measured forces and moments and to provide another data set to compare against CFD.

This paper provides a status of the CFD analysis based on the as-designed wind tunnel models and a subset of the planned wind tunnel test conditions. Section II summarizes the inputs to the CFD analysis: the wind tunnel conditions, model geometries, a description of the flowfield solvers, and a subset of the planned test matrix for analysis. Section III describes the results to date, focusing on the effects of the main test parameters: thrust, nozzle pointing direction, nozzle size, nozzle area ratio, nozzle location, and tunnel Mach number. Other aspects of the test campaign¹³ and CFD analysis include detailed computational sensitivities,^{14, 15} a new plume seeding method,¹⁶ and the flow-through six-component force and moment balance¹⁷ developed for the test. After the test has been completed, additional publications will follow that describe the detailed test results and uncertainty quantification of select CFD solvers executed at the as-tested conditions.

II. Approach

This section describes the inputs required to set up CFD solutions of the designed wind tunnel models in the LUPWT environment: tunnel conditions, models, CFD solver description and setup, and the pre-test run matrix.

A. Wind Tunnel Conditions

The NASA Langley Research Center LUPWT is a closed-circuit continuous-flow pressure tunnel with two test sections that each have a nominal 4-foot square cross section. The tunnel consists of a 100,000 horsepower compressor, a dry air supply and evacuation system, a cooling system, and paths that circulate air through either of the two test sections. The Mach number ranges are approximately 1.50 to 2.86 in test section 1 and 2.30 to 4.63 in Test Section 2. The SRP test will be conducted in Test Section 2 (see Figures 2 and 3), the same section that was used for a 2010 SRP test.⁸ The tunnel stagnation pressure can be varied up to a maximum of 100 *psia* in Test Section 2. The Mach number is controlled using an asymmetric sliding-block nozzle, which is used to select the nozzle-throat to test-section area ratio. New measurements of the tunnel recently were completed¹⁸ and will be used to select conditions for the upcoming SRP test. Non-uniform flow was measured in the test section, with varying Mach number and non-zero flow angularity in the 7-foot long section. This information will be used to define inflow conditions for CFD analysis of the SRP models with tunnel walls included.

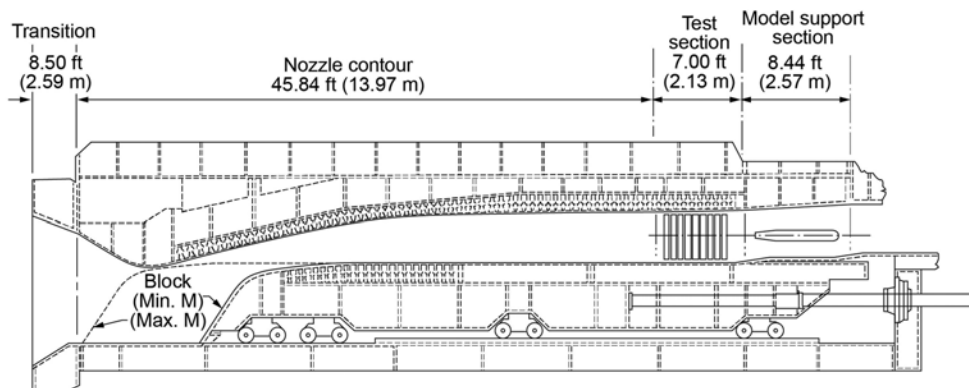


Figure 2. NASA Langley LUPWT Test Section 2.

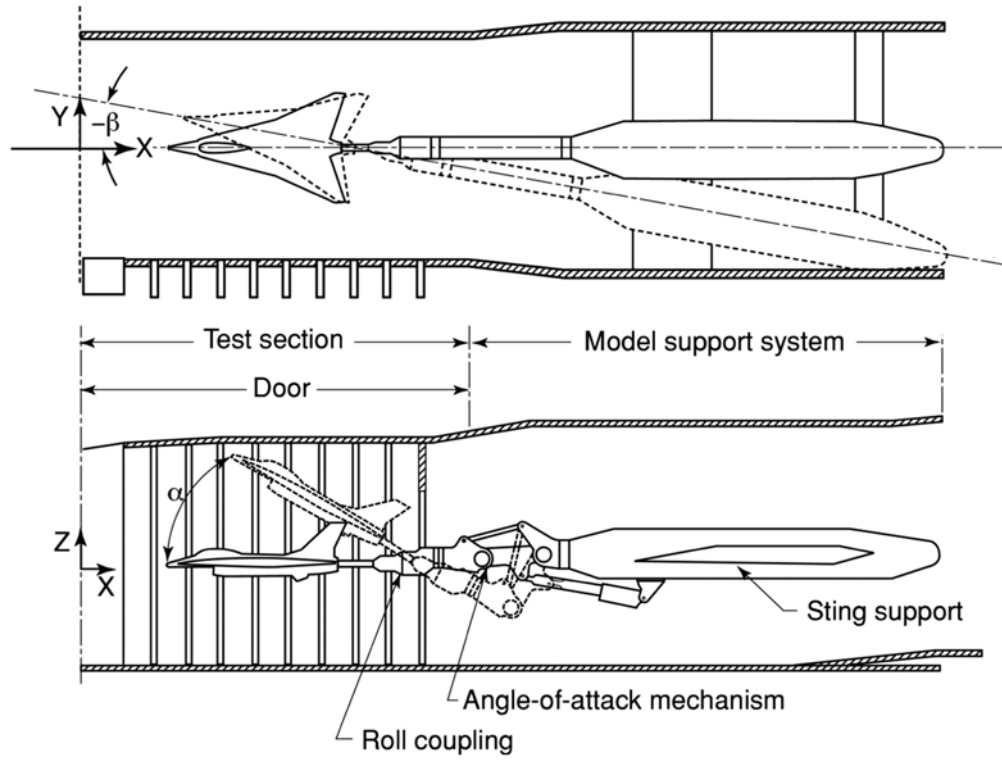


Figure 3. LUPWT model support.

Table 1 summarizes three LUPWT nominal conditions that are planned for the test campaign and have been used for CFD calculations to date. The test section re-characterization was accomplished using nineteen five-hole probes mounted to a rake that was swept through the test section envelope (Figure 4). The measurements confirmed non-uniform flow in the test section and the results will be used to support overall test uncertainties. CFD analysis of the tunnel test section also was completed¹⁹ and is used in the CFD analysis presented here.

Table 1. Planned LUPWT conditions.

Condition	M_∞	T_0 ($^{\circ}F$)	Re_∞/ft	$\bar{q}$ (lb_f/ft^2)
5	2.386	125	1	210.34
20	3.477	125	1	153.22
35	4.568	150	1.5	167.41

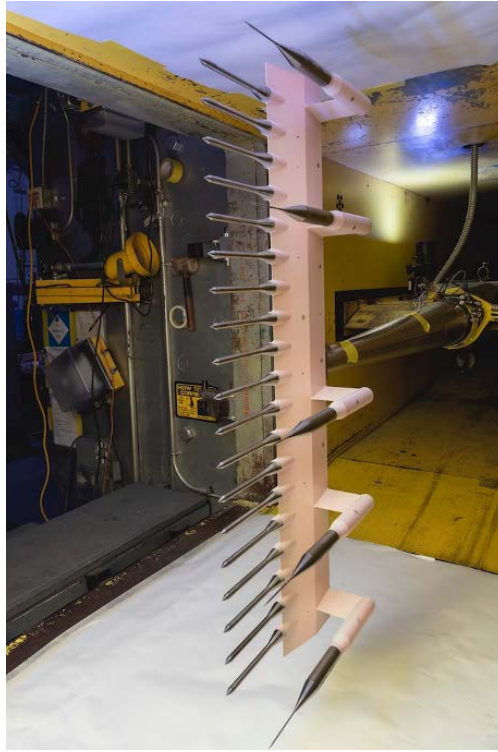


Figure 4. Flow survey rake used for LUPWT test section re-characterization.^{18–20}

B. Wind Tunnel Models

The primary goal of the wind tunnel model design effort was to make the models as relevant to current human-scale Mars flight reference vehicles as possible. The wind tunnel environment itself is different from powered flight conditions at Mars in number of ways, including the freestream gas (air in the tunnel versus CO₂ at Mars), the Mach and Reynolds numbers, and the plume gases: cold high pressure air (HPA) in the tunnel versus hot combustion products in flight. However, there are a few aspects of the model design that can be controlled to make the test closer to flight. The first is the model heatshield outer mold lines, which will be geometrically scaled from the flight geometries. Second, the simulated engine nozzle exit locations for some models will be placed on the heatshields at the same radial locations that previously were modeled at flight conditions.²¹ Third, some of the Low-L/D models, as well as the single Mid-L/D model, will have HPA nozzle exit areas that are the same, relative to the heatshield area, as the set of available flight reference vehicles. This approach for scaling the model heatshield geometries and nozzles makes the current test campaign more relevant for Mars powered flight compared to previous tests.

The Low-L/D and Mid-L/D models are shown in Figure 5 with the external geometry and the internal flow path for the HPA, which enters the sting at a 90-degree angle. The instrument housing and sting will be common between the two models and the remaining components are tailored to each model. The main part of the two model geometries that is scaled from flight is the front side of the heatshield. The external geometries elsewhere were designed to provide enough internal volume for the instrumentation and HPA flow path. The CFD analysis that has been completed to date at Mars flight conditions suggests that there are a few key nozzle parameters that will affect the plume interference characteristics and hence the magnitude of AI forces and moments measured in the wind tunnel. The Low-L/D model will be used to explore some of these key parameters by using interchangeable nozzle inserts. Nozzle location on the heatshield (R_n/R_b), nozzle cant angle (θ_{cant}), nozzle exit area relative to heatshield area ($\sum A_e/A_{ref}$), and nozzle area ratio (A_e/A_t) all will be varied in the Low-L/D model. Given limitations in the budget and test schedule, only two different values of each parameter will be tested. Only a single set of nozzles will be used in the Mid-L/D model.

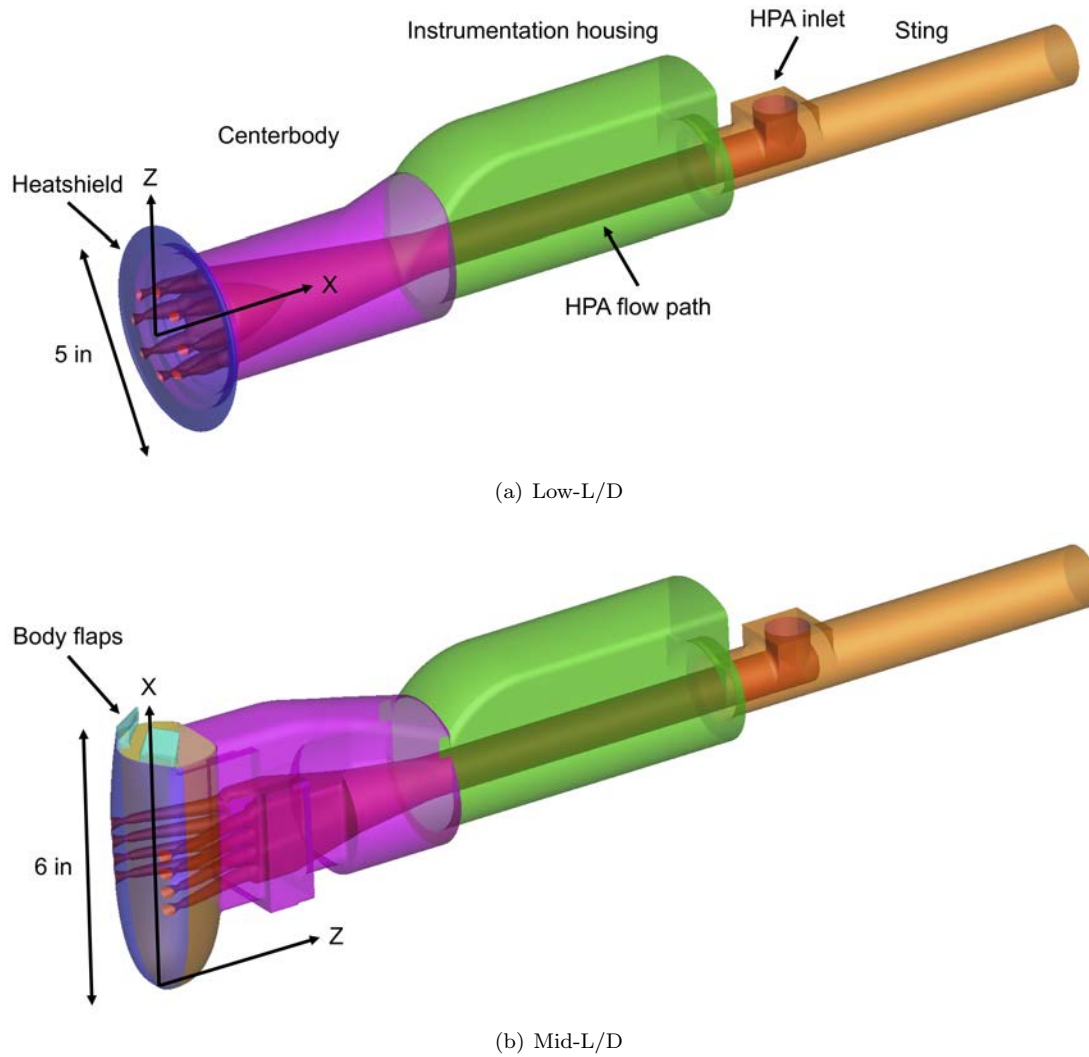


Figure 5. Low-L/D (a) and Mid-L/D (b) model external geometries and HPA flow paths.

Figure 6 shows front views of the two model variations: Low-L/D model with interchangeable nozzle inserts (1A through 1F) and Mid-L/D model with a single nozzle configuration (2A). The nozzle area ratios (A_e/A_t), cant angles (θ_{cant}), and radial location relative to the maximum heatshield radius (R_n/R_b) are shown. The Low-L/D nozzle profiles are also shown. The nozzles for model 2A are almost identical to those of model 1A. The red circles show the locations where static pressure will be measured. The pink and gray circles show locations for dynamic pressure measurements. As much of the heatshield and nozzle insert surfaces on the new models will be covered with pressure-sensitive paint (PSP) to give time-averaged pressure distributions. The steady-state PSP data also will be used to estimate the inviscid steady-state AI forces and moments from the painted surfaces. Originally, the steady-state PSP data were the highest priority data for this test because AI forces and moments are considered the highest aerosciences risk for human Mars EDL. Delays in the test due to COVID-19 and facility maintenance allowed the addition a new six-component flow-through force and moment balance design that was developed in a parallel effort at NASA Langley. The balance was designed to be retrofitted to the Low-L/D model to provide AI forces and moments from the model heatshield. The addition of the balance allowed the first known direct force and moment measurements of a retropropulsion wind tunnel model. Finally, high-speed schlieren video (at least 10,000 frames per second) will be used to visualize the nozzle plumes in order to give context for the other measurements and to provide a more complete picture of the flowfield interactions.

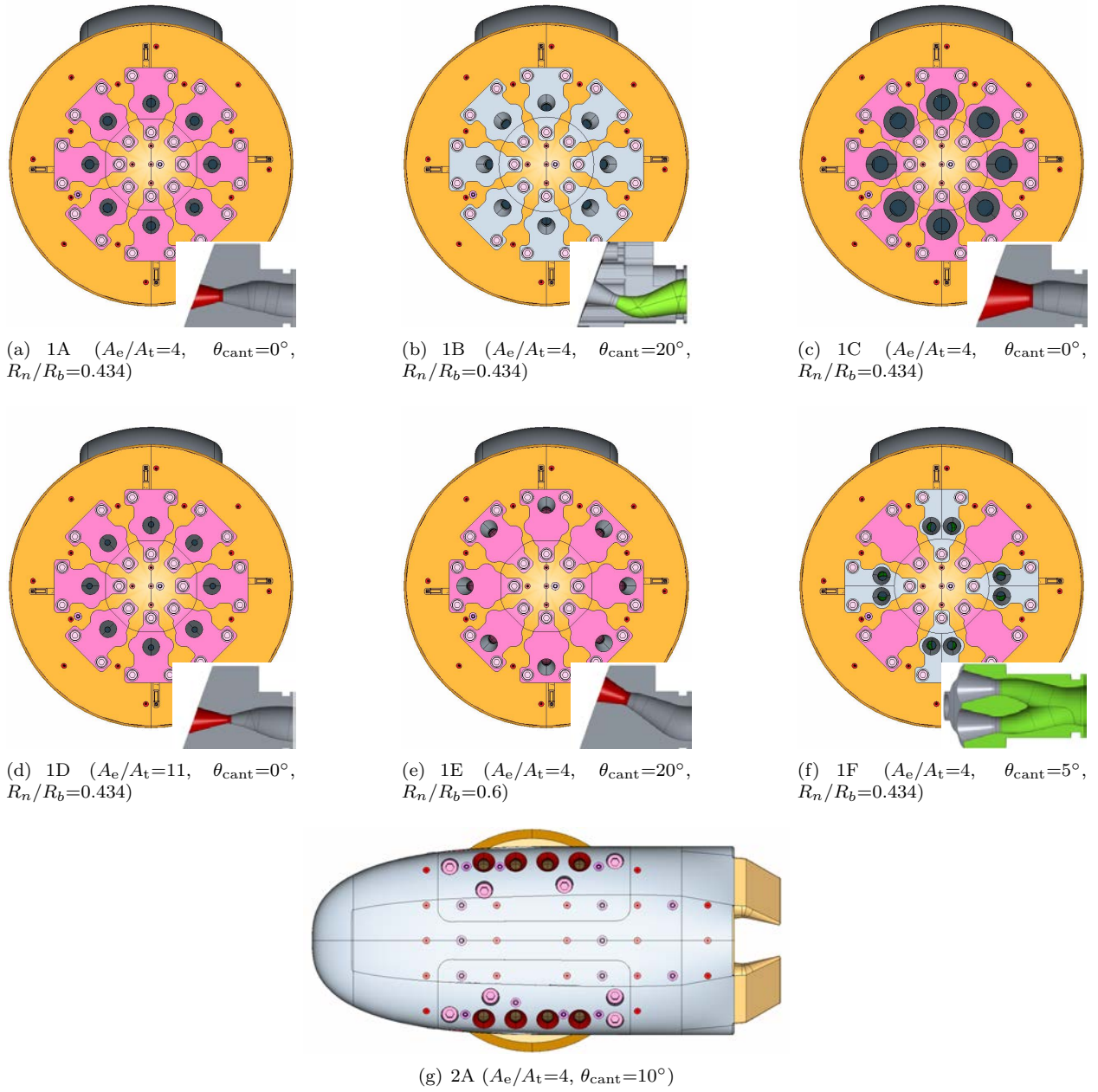


Figure 6. Wind tunnel models with a common heatshield and interchangeable nozzle inserts for the Low-L/D model (1A through 1F). Red circles are locations of steady-state pressure measurements. Pink and gray circles are locations of high-frequency pressure measurements.

The key nozzle parameters for both models are shown in Table 2: nozzle radial location on the heatshield (R_n/R_b), cant angle (θ_{cant}), nozzle total exit-to-heatshield area ratio ($\sum A_e/A_{\text{ref}}$), nozzle exit-to-throat area ratio (A_e/A_t), and nozzle exit Mach number from compressible flow theory (M_e).

Table 2. Model parameters compared to flight reference vehicles.

Geometry	R_n/R_b	θ_{cant} (deg)	$\sum A_e/A_{\text{ref}}$	A_e/A_t	M_e
Low-L/D (Flight)	0.434	5	0.062	177	5.6
1A	0.434	0	0.062	4	2.94
1B	0.434	20	0.062	4	2.94
1C	0.434	0	0.112	4	2.94
1D	0.434	0	0.062	11	4.03
1E	0.6	20	0.062	4	2.94
1F	0.434 (paired)	5	0.062	4	2.94
Mid-L/D (Flight)	N/A	10	0.051	177	5.6
2A	Same as flight	10	0.051	4	2.94

C. Flow Solvers

Three CFD solvers have been used to simulate the retropropulsion models in the LUPWT environment: OVERset grid FLOW solver (OVERFLOW), Fully Unstructured Navier-Stokes Three-Dimensional (FUN3D), and Loci/CHEM. All three solvers commonly are used for NASA missions, including for applications involving jet interactions. Both FUN3D and OVERFLOW were used in conjunction with SRP testing in the LUPWT in 2010 and the Ames 9x7 tunnel in 2011.^{10–12}

OVERFLOW²² uses overset grids that were developed using Chimera Grid Tools.^{23–25} The grid systems were designed to be used with automatic mesh refinement (AMR)²⁶ in the plume and bow shock region. AMR was allowed to progress to finer grid spacing until the heatshield axial force coefficient appeared to be within 0.01 of its grid asymptotic value. Approximately, the initial grid systems had 150 million points, and the refined grids had 250 million with all added grid points in the region of plumes and the bow shock. The effects of grid adaption for both models are detailed in Refs. 14–15. Solver parameters were selected to be appropriate for the present simulations. The Harten-Lax-van Leer-Contact (HLLC) spatial discretization algorithm was used throughout most of the computational domain, while the Harten-Lax-van Leer-Einfeldt (HLLC++) algorithm was used for the bow shock and plumes. All cases have been computed using time-accurate unsteady Reynolds-Averaged Navier-Stokes (URANS). The time step and number of sub-iterations were selected after a temporal accuracy study.¹⁵ The shear stress transport (SST) model²⁷ with zonal modeling options were used in all simulations. The Quadratic Constitutive Relation (QCR) model^{28,29} was used in the wall-adjacent tunnel grids to predict corner vortices. Throughout the domain, the rotation-curvature correction³⁰ is enabled and the compressibility correction³¹ is disabled. Steady state inflow conditions to the test section were obtained from prior time-average time-accurate solutions of the “full” empty tunnel performed with OVERFLOW.³² This is an approximation, as tunnel flow unsteadiness has been measured (and simulated) at several operating conditions. The resources needed to perform time-accurate simulations with the full tunnel and models are too large for production cases, due to the large discrepancies in time scales. Most of these URANS simulations gave steady time-asymptotic behavior and were well time-converged. However, some simulations exhibited unsteady time-asymptotic behavior, which is discussed in detail in Refs 14–15.

Loci/CHEM is a density-based, finite-volume solver built upon the Loci framework.^{33,34} Loci coordinates the numerical kernels and methods contained in Loci/CHEM. The Loci/CHEM retropropulsion calculations used unstructured grids generated with the Advancing Front with Local Reconnect (AFLR3) code. The grids consist of approximately 200 million cells with prism cells near solid surfaces and mostly tetrahedrons within the far-field. At solid surfaces the grid spacing normal to the surface is adjusted to obtain a y^+ value

of approximately 1. For each case the calculation is performed in two steps. In the first step the computation is executed using local time-stepping to generate the initial conditions to be used for the time-accurate calculation in the second step. The time-accurate URANS equations used Menter's SST two-equations turbulence model with Wilcox compressibility correction. The equations are integrated with second-order accuracy in space and time, and are solved fully implicit with the Line Symmetric Gauss-Seidel (LSGS) method. The inviscid fluxes are approximated using the Roe scheme with Venkatakrishnan limiters and HLLE at regions with strong shocks. To maintain a time-accurate integration, the calculation performs three Newton-Raphson sub-iterations per global iteration. A reduction of at least two orders of magnitude in the residuals has been observed on all the calculations after the completion of the sub-iterations. All calculations were run until reaching a stationary state with a time step of $0.1 \mu\text{s}$.

FUN3D is a node-based finite-volume solver developed at NASA Langley Research Center.³⁵ A second-order approach is used for spatial discretization on generic unstructured grids containing tetrahedra, pyramids, prisms, and hexahedra. The equations are integrated second order in time by using an implicit formulation with a physical time step of $40 \mu\text{s}$ and five sub-iterations per time step, which were determined through a time step sensitivity study while monitoring temporal error controller diagnostics. All solutions are generated using a modified delayed detached eddy simulation (DDES) approach³⁶ with the Spalart QCR2000 Reynolds stress model³⁷ with rotation/curvature correction³⁸ and Vatsa-Lockard modification to avoid spurious upstream turbulence generation.³⁹ A low-dissipation Roe scheme is used for inviscid flux construction with a Van Albada flux limiter.⁴⁰ The solutions began with mesh generation and solution development that use an unsteady sketch-to-solution workflow,⁴¹ where the inputs are the model geometry, domain dimension for the wind tunnel test section, the inflow boundary profile, and specifications for mesh growth and adaptation. In the second step, the meshes are adapted anisotropically to a multi-scale Mach-based Hessian error indicator averaged over approximately 5,000 time steps to minimize solution interpolation error. The inflow boundary profiles are obtained from prior FUN3D solutions of the LUPWT empty tunnel.¹⁹ The final meshes have approximately 50 million grid points with tetrahedral connectivity. Finally, the simulation time-varying statistics are taken over 50,000 time steps on the final mesh (generated during the second step) to eliminate any contribution of start-up transients in the final force and moment coefficient statistics.

Figure 7 shows the wind tunnel and model boundary conditions applied in the CFD solutions. The tunnel inflow plane is taken from companion CFD solutions of the tunnel settling chamber, nozzle, and test section. The tunnel walls are treated as viscous no-slip, adiabatic boundaries. The full model HPA flow path is not modeled; rather, the nozzles are truncated and a total pressure and temperature boundary condition is applied at the truncated faces. Based on CFD solutions of the internal HPA flow path, the total conditions at the nozzle inflow planes shown will be lower by a few percent than the conditions where the HPA is controlled. This difference will be corrected by adjusting the CFD total conditions to match the estimated total pressure and temperature loss using internal flow path measurements. Additional CFD solutions will be completed at the tested HPA conditions to estimate the total pressure and temperatures losses at the truncated nozzle faces.

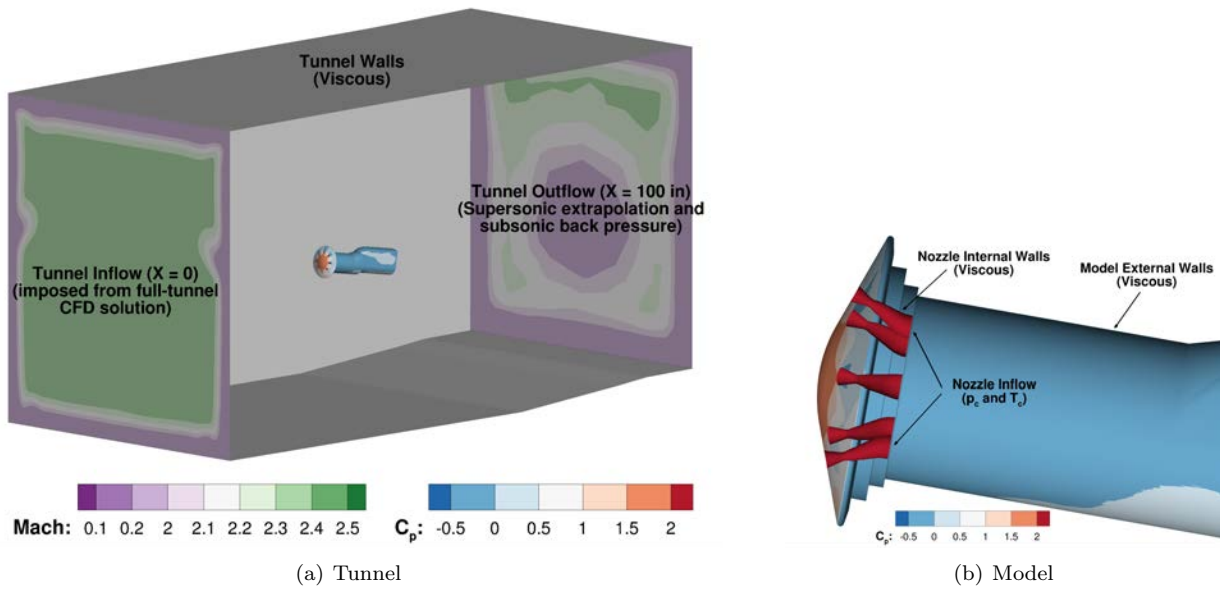


Figure 7. Tunnel (with one side wall not shown) and model boundary conditions.

Figure 8 shows the matrix of CFD cases planned prior to the start of the test: nine model configurations (including non-blowing), three angles of attack, four thrust coefficients (0, 0.5, 1, 2.5), and three tunnel Mach numbers (2.386, 3.477, 4.568). In total, over 350 cases have been completed to date. Once the test is underway, adjustments to the CFD boundary conditions or model location in the tunnel will be made before comparing to the data. Figure 9 shows an example of a FUN3D calculation of model 1A in the tunnel test section with tunnel walls included.

		Condition #5 (Mach = 2.386, Re = 1e6 / ft)								Condition #20 (Mach = 3.477, Re = 1e6 / ft)								Condition #35 (Mach = 4.568, Re = 1.5e6 / ft)							
Model	α	0		0.5		1		2.5		0		0.5		1		2.5		0		0.5		1		2.5	
		LC	F3D	OF	LC	F3D	OF	LC	F3D	OF	LC	F3D	OF	LC	F3D	OF	LC	F3D	OF	LC	F3D	OF	LC	F3D	OF
1A	00																								
1A	05																								
1A	10																								
1B	00																								
1B	05																								
1B	10																								
1C	00																								
1C	05																								
1C	10																								
1D	00																								
1D	05																								
1D	10																								
1E	00																								
1E	05																								
1E	10																								
1F	00																								
1F	05																								
1F	10																								
2A	90																								
2A	85																								
2A	80																								

Figure 8. Pre-test CFD run matrix. Cell colors indicate first priority cases (gold), second priority cases (green), and bonus cases (gray) for Loci/CHEM (LC), FUN3D (F3D), and OVERFLOW (OF).

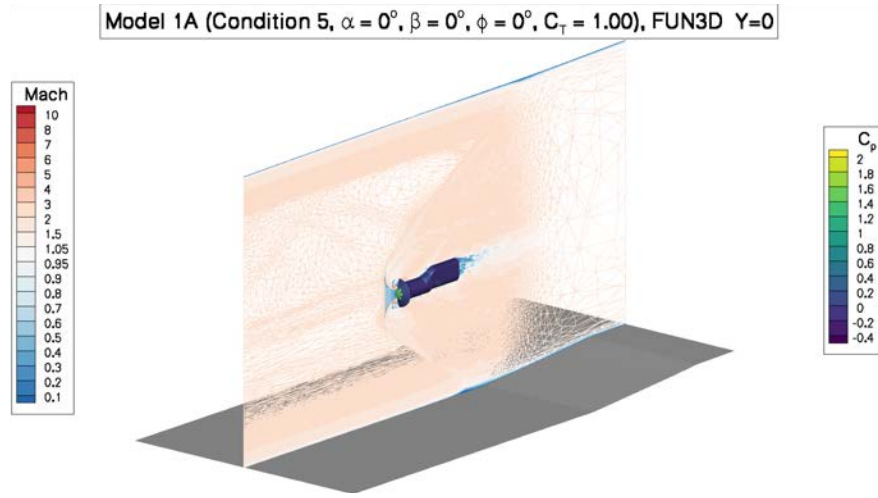


Figure 9. FUN3D case on adapted grid for model 1A at $C_T = 1$: time-averaged Mach number in the $Y = 0$ tunnel plane and time-averaged surface pressure coefficient at a tunnel Mach number of 2.386.

Table 3 shows the parameters that will be compared between the available CFD solutions and test data. Total mass flow through the nozzles will be measured as a function of HPA total pressure and compared to the CFD mass flows. The primary quantitative metrics for judging predictive accuracy will be aerodynamic axial force coefficient and discrete surface pressures on the heatshield.

Table 3. Parameters to be compared between CFD and test data.

Data type	Test measurement method
Total mass flow rate through blowing nozzles	Coriolis meter
Total thrust from blowing nozzles	Flow-through balance with tunnel off
Six-component forces & moments from heatshield	Flow-through balance with tunnel on
Discrete time-averaged pressure on heatshield	Static pressure taps
Discrete high-frequency unsteady pressure on heatshield	Kulite pressure transducers
Global time-averaged pressure on heatshield	Pressure-sensitive paint

III. Results

This section covers a subset of the CFD results generated to date that use the as-designed model geometries and expected test conditions: tunnel Mach and Reynolds numbers, HPA total pressure, model angle of attack, and model position in the tunnel test section. The CFD analysis has required extensive time and resources due to the large grids, the choice to model the tunnel non-uniform inflow and walls, and the need to run time-accurate cases for what is expected to be an unsteady flowfield for all conditions tested. Additional calculations will be completed to compare the differences between having the non-uniform inflow and tunnel walls versus using average fixed inflow conditions and no walls. Calculations have been completed to date that cover a representative subset of the planned test matrix prior to the test using established best practices for grid generation, turbulence model settings, time-step requirements for unsteady solutions, and grid sensitivity studies. The OVERFLOW and Loci/CHEM results included here are URANS; further analysis with DDES or similar approaches will be investigated as has been done with FUN3D. Some of the differences between solvers in the following sections probably are due to the URANS and DDES approaches, and will be addressed in a future publication.

A. Non-Blowing Models

Both wind tunnel models will be tested with all nozzles plugged in order to obtain baseline aerodynamics and to compare those results to previous test results in the LUPWT on similar geometries. Figure 10 shows results on both plugged models at three angles of attack and tunnel conditions 5 and 20. The axial force coefficients due to the heatshield fronts are shown for the available CFD solutions, $C_{F,x}$ for the Low-L/D model and $C_{F,z}$ for the Mid-L/D. For the Low-L/D model results, the differences between the solvers at a given tunnel condition and model angle of attack are at most about 3%, which is considered typical for the geometry and Mach numbers. Similar levels of agreement are observed for the Mid-L/D model results at tunnel conditions 5 and 20.

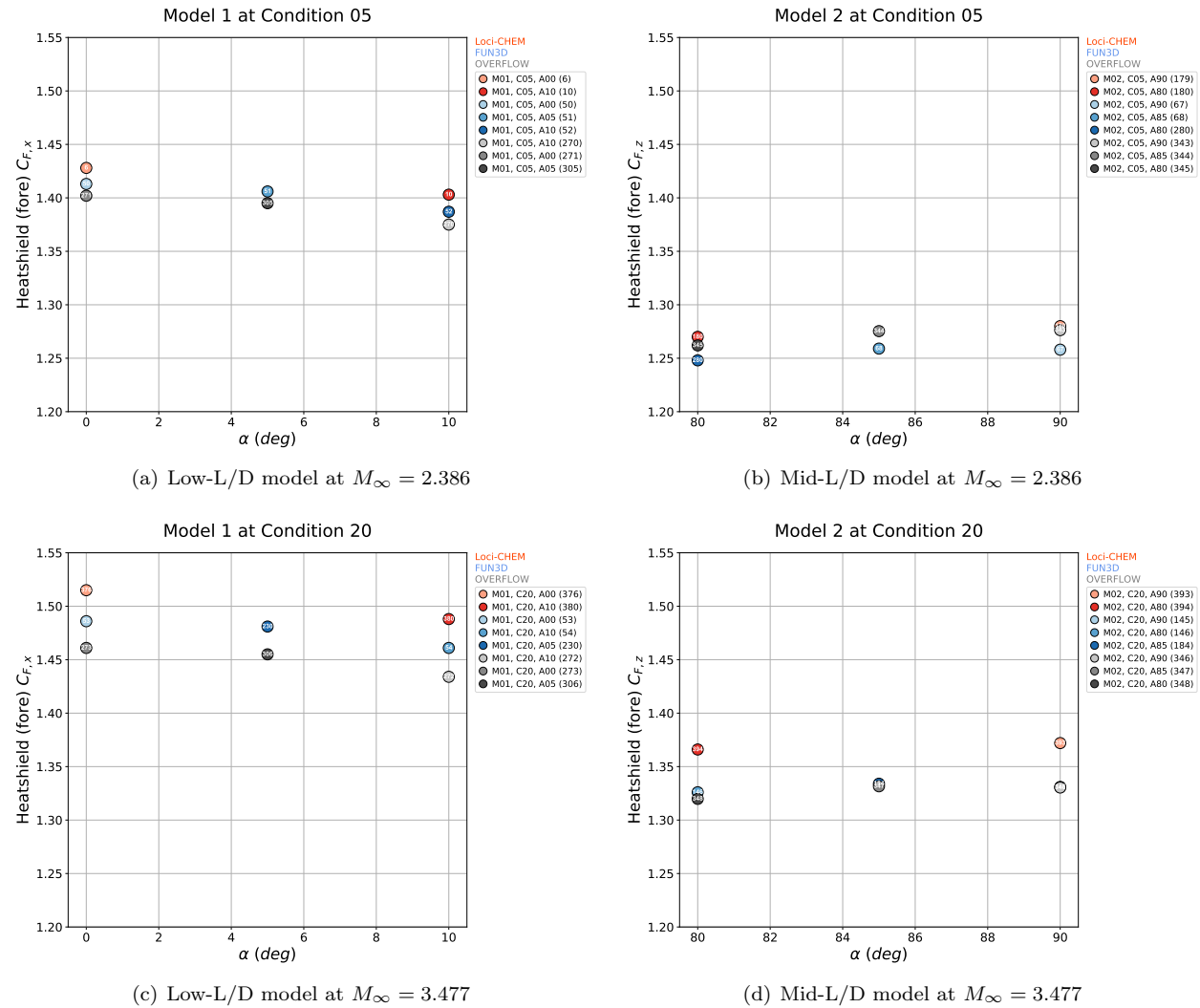


Figure 10. Predicted heatshield aerodynamic axial force coefficient for non-blowing models. Case identification in legend is model (M), tunnel condition (C), angle of attack (A), and case number in parentheses.

B. Predicted Mass Flow Rate and Thrust

During the wind tunnel test, the HPA total pressure (p_c) and total temperature (T_c) will dictate how much thrust is produced by the various model nozzle configurations. These HPA conditions are applied as the boundary condition to the nozzle plenum face, as shown in Figure 7, rather than calculating the entire HPA flow path. One of the sources of CFD uncertainty is how the predicted blowing parameters, mass flow rate and thrust, compare to actual values. Direct measurements of thrust from the flow-through balance with the tunnel off will be used as part of the uncertainty analysis. Figure 11 shows the predicted total mass flow and axial thrust from the available CFD solutions for models 1A and 2A. The results from quasi one-dimensional isentropic flow relations are also shown. As expected, there exists scatter between CFD solvers in both mass flow and thrust for a given HPA total pressure, with the largest differences at the highest pressure.

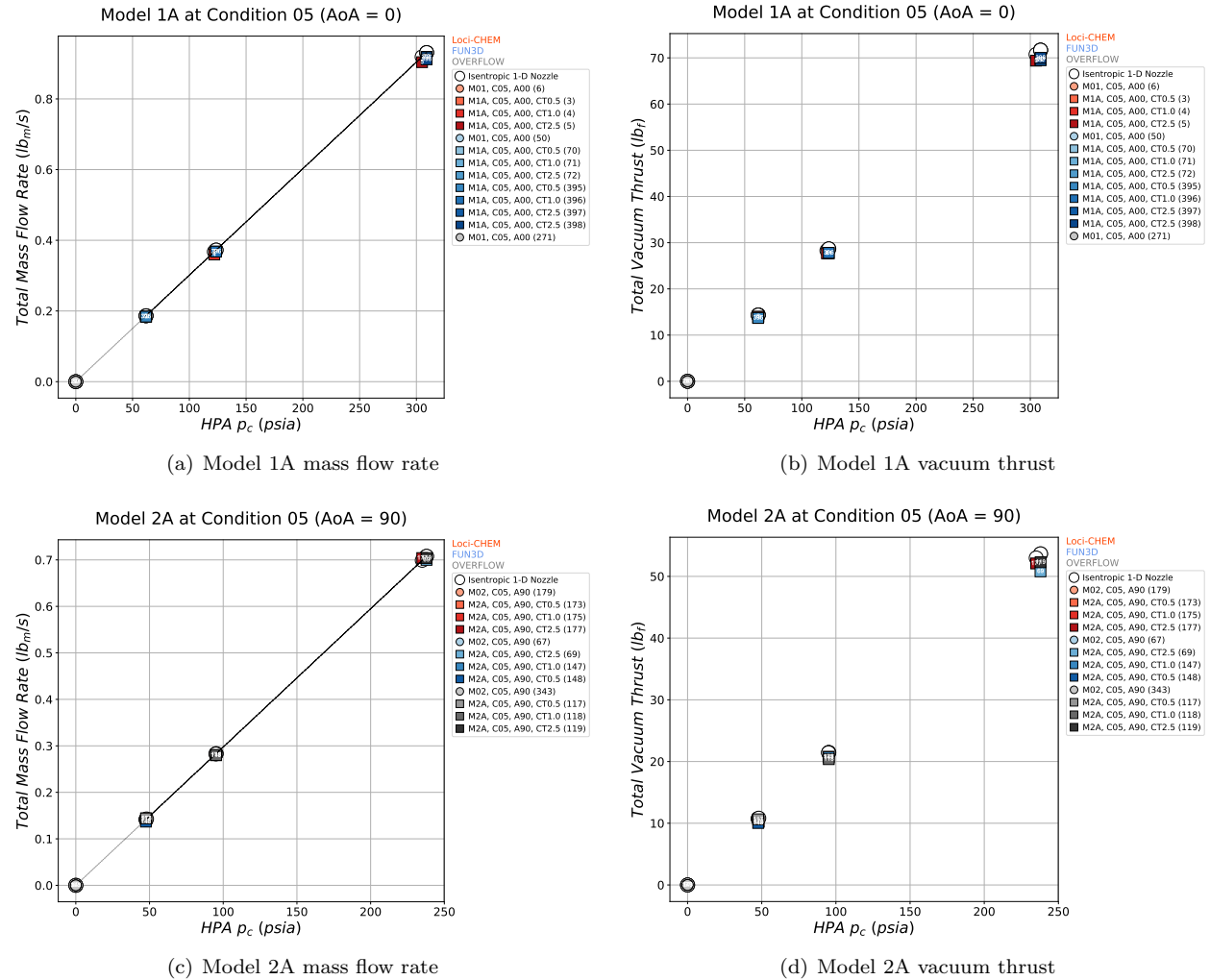


Figure 11. Predicted total mass flow rate and axial thrust for models 1A and 2A.

C. Predicted Effect of Blowing Nozzles

The predicted effect of blowing nozzles on the flowfield and AI axial force is shown here for model 1A at tunnel condition 5. Figure 12 shows model 1A FUN3D calculations ($\alpha = 0^\circ$) at a tunnel Mach number of 2.386 for increasing thrust coefficients from 0 to 2.5. The figure shows the adapted computational mesh on the $Y = 0$ tunnel plane colored by Mach number contours and model surface pressure coefficient (C_p) contours. The general trend, which has been observed in previous wind tunnel tests, is for the plumes to gradually expand and push out the bow shock from the model as thrust increases. For model 1A, even at $C_T = 2.5$, the tunnel flow is able to form a shock inboard of the plumes, which pressurizes the heatshield within the radius of the nozzle exits. At radial locations beyond the nozzle exits, the pressure coefficient gradually decreases with increasing thrust due to blockage from the expanding nozzle plumes. During the test campaign, the maximum thrust coefficient will be determined by whether the nozzle configuration and tunnel conditions result in tunnel wall interference or unstart; this will be determined by high-speed video and tunnel wall pressure measurements.

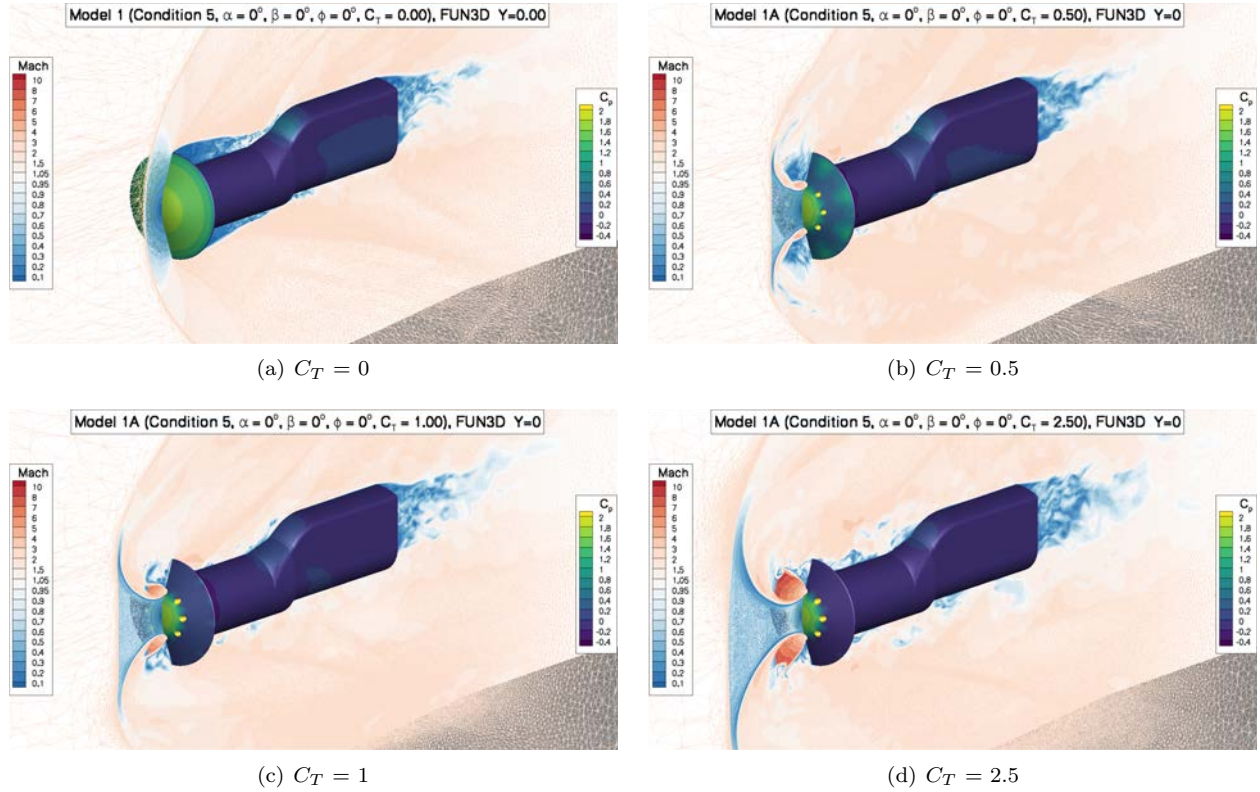


Figure 12. FUN3D time-averaged flowfield Mach number and model surface pressure coefficient for Low-L/D model 1A at $M_\infty = 2.386$ and $\alpha = 0^\circ$.

Figure 13 shows predicted aerodynamic axial force coefficient due to the front side of the heatshield for all Low-L/D models at tunnel condition 5 and $\alpha = 0^\circ$. The error bars represent $\pm$ one standard deviation in the unsteady aerodynamic coefficient from the time-accurate solution. The predicted trend is a gradual decrease in aerodynamic axial force with increasing thrust, to the point that the force essentially vanishes at a thrust coefficient of 2.5, due to the reduction in pressure coefficient outboard of the nozzle exits countering the higher pressure inboard. The FUN3D DDES results exhibit large error bars, especially at the two lower thrust coefficients, whereas the Loci/CHEM URANS solutions have much smaller standard deviations. There are differences between the CFD solvers that still need to be investigated, for example at the $C_T = 2.5$ condition; the FUN3D DDES solutions (repeated by different analysts) and Loci/CHEM result may indicate a deficiency in the Loci/CHEM URANS approach.

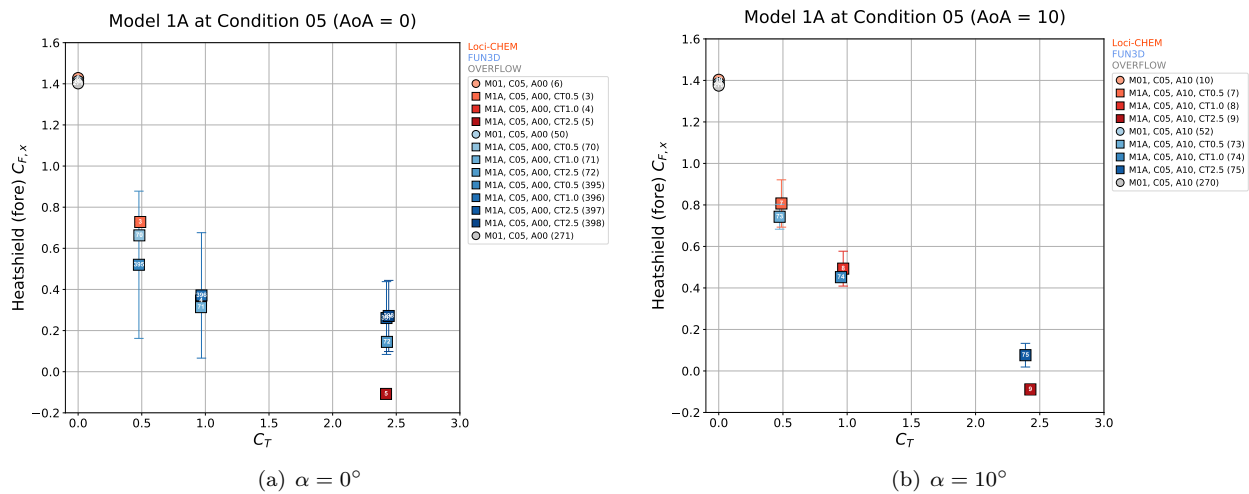


Figure 13. Predicted effect of thrust on heatshield aerodynamic axial force coefficient for Low-L/D model 1A at $M_\infty = 2.386$. Case identification in legend is model (M), tunnel condition (C), angle of attack (A), thrust coefficient (CT), and case number in parentheses.

The predicted effect of thrust variation on the Mid-L/D model is shown in Figures 14 and 15. The Mach number contours are shown in a constant Y plane that passes through one row of blowing nozzle exits. Again, the bow shock distance from the model heatshield grows with increasing thrust coefficient and the pressure coefficient in between the nozzle rows also increases as the nozzle plumes constrict the oncoming flow. In contrast to the model 1A results, the predictions for model 2A show that the axial force does not decrease much with increasing thrust. The largest discrepancies between solvers is at the highest thrust coefficient, where two of the solvers predict that the axial force is similar to the non-blowing value. The reasons for the differing trends between models 1A and 2A are due to the interplay between the model geometries and nozzle configurations, how the plumes form in front of the model, and whether the plumes block the external flow from reaching the heatshield. For model 2A, the area of the heatshield that is shielded by the nozzle plumes with increasing thrust is relatively small compared to the area where higher pressure coefficients are maintained. If the two rows of nozzle had been placed closer to one another, a reduction in axial force coefficient would be expected.

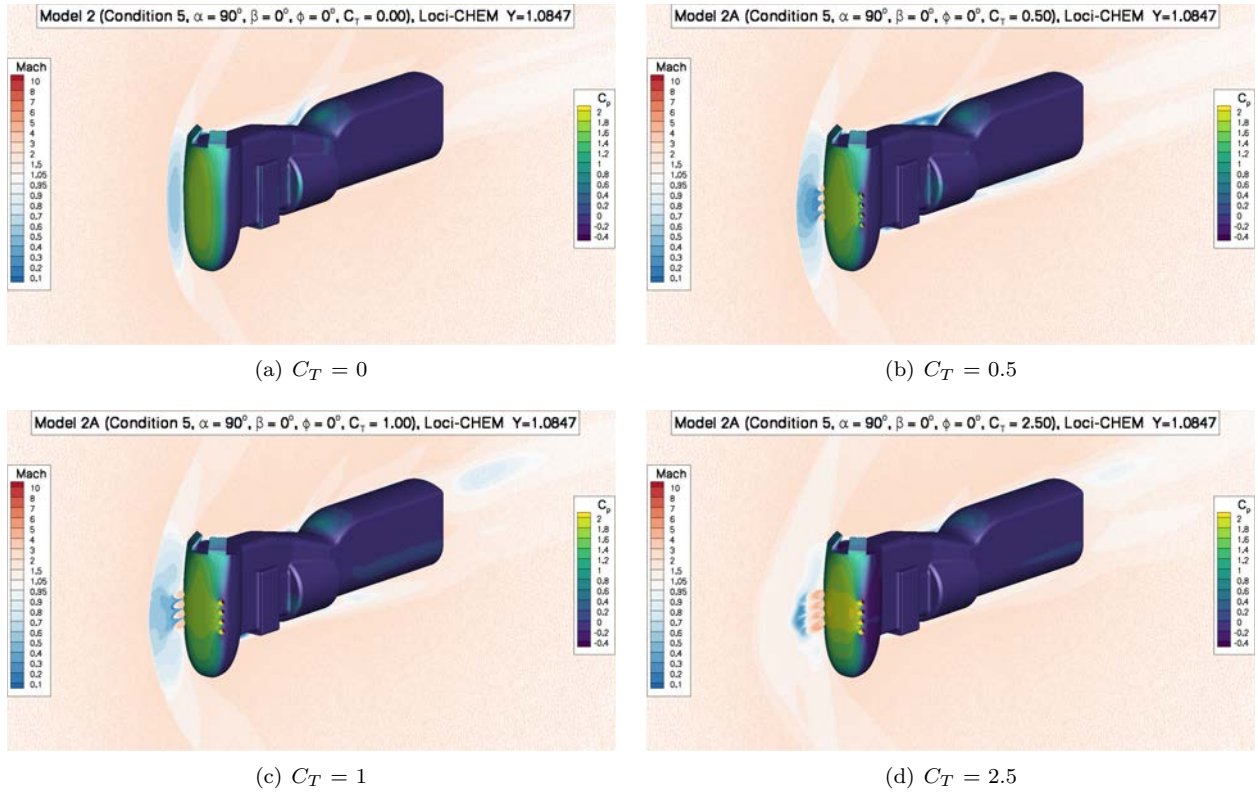


Figure 14. Loci/CHEM predicted time-averaged flowfield Mach number and model surface pressure coefficient for Mid-L/D model 2A at $M_\infty = 2.386$ and $\alpha = 90^\circ$.

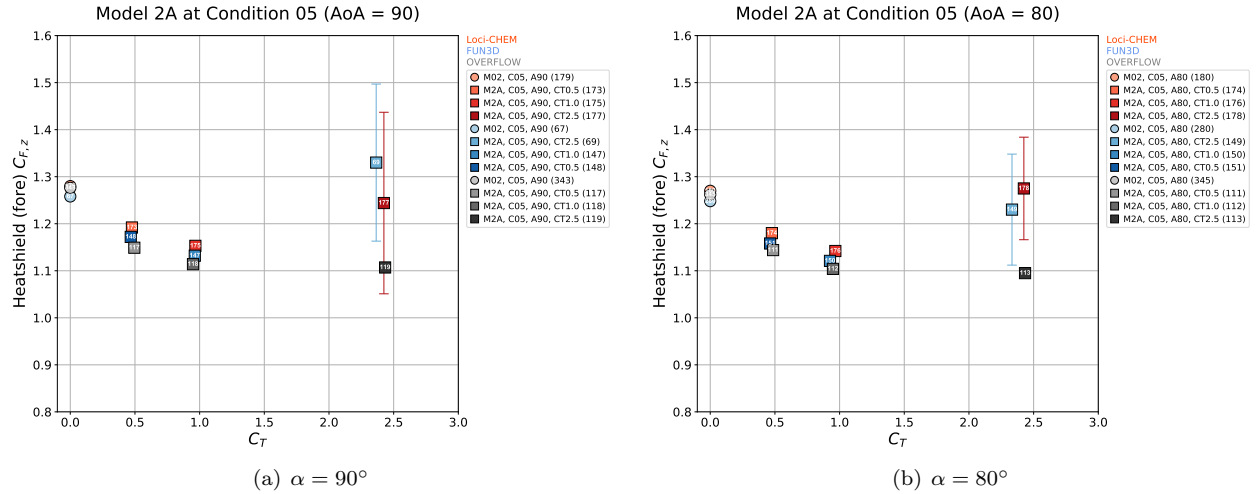


Figure 15. Predicted effect of thrust on heatshield aerodynamic axial force coefficient for Mid-L/D model 2A at $M_\infty = 2.386$.

D. Predicted Effect of Nozzle Cant Angle

The predicted effect of nozzle cant angle on the Low-L/D model is illustrated by comparing models 1A ($\theta_{\text{cant}} = 0^\circ$) and 1B ($\theta_{\text{cant}} = 20^\circ$). Figure 16 shows the time-averaged flowfield for $C_T = 1$ and a tunnel Mach number of 2.386 for those two models. The outward pointing direction of the nozzles for model 1B permits more of the external flow to reach the heatshield surface, and thus increases the overall area of higher surface pressure by a small amount. Figure 17 shows predicted aerodynamic axial force coefficient due to the front side of the heatshield. The general predicted trend is that a nozzle cant angle of 20 degrees results in a small increase in the force coefficient compared to a nozzle cant angle of 0 degrees.

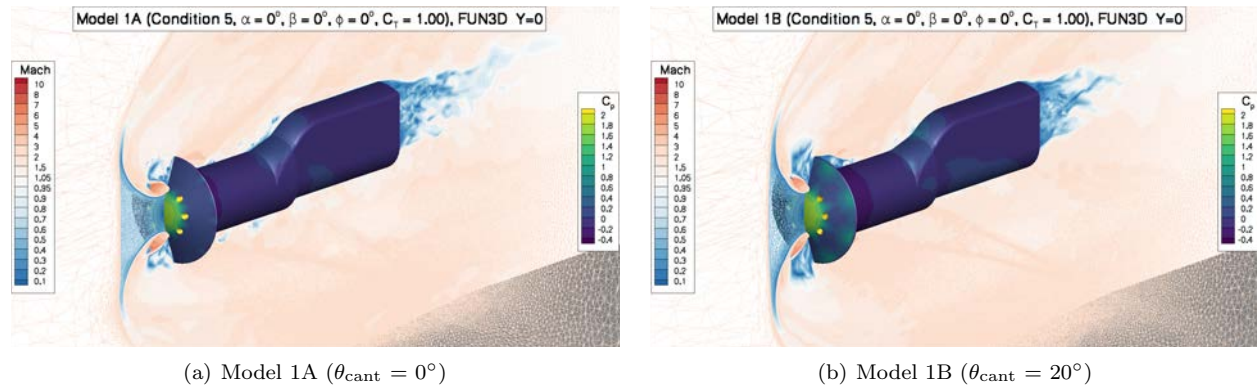


Figure 16. FUN3D predicted time-averaged flowfield Mach number and model surface pressure coefficient for Low-L/D models 1A and 1B at $M_\infty = 2.386$, $C_T = 1$, and $\alpha = 0^\circ$.

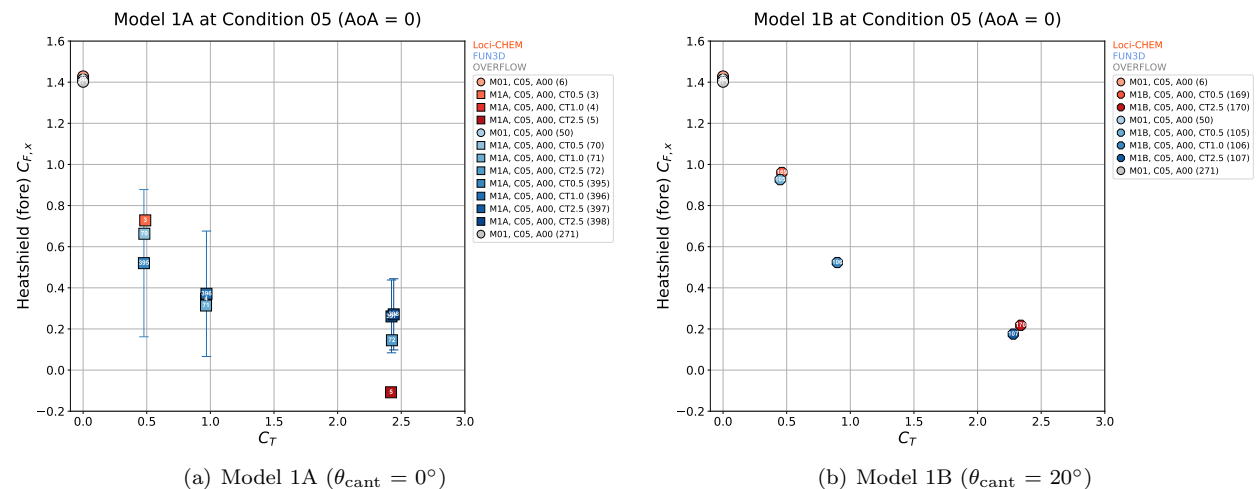


Figure 17. Predicted effect of nozzle cant angle on heatshield aerodynamic axial force coefficient at $M_\infty = 2.386$ and $\alpha = 0^\circ$.

E. Predicted Effect of Nozzle Size

The predicted effect of nozzle size on the blunt model is accomplished by comparing models 1A and 1C. The Mach number contours in Figure 18 show how the larger nozzles create larger plumes in front of the model heatshield. Figure 19 shows predicted aerodynamic axial force coefficient due to the front side of the heatshield. The predicted trend for axial force coefficient is that a larger nozzle exit results in a small decrease in the force coefficient compared to the small nozzle configuration. This result partly is due to the larger nozzles taking up more surface area on the heatshield, and thus leaving less area for aerodynamic contributions. The larger nozzles also create larger plumes that block more of the tunnel flow from reaching the heatshield surface.

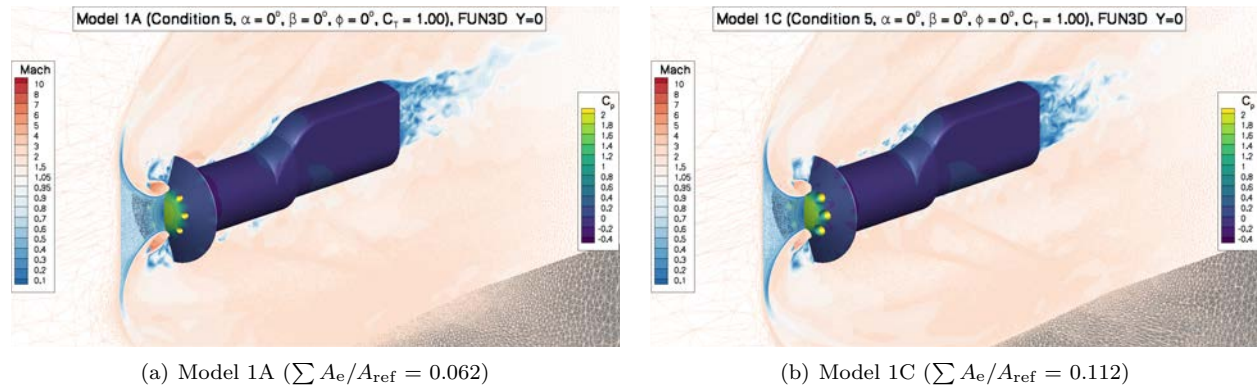


Figure 18. FUN3D predicted time-averaged flowfield Mach number and model surface pressure coefficient for Low-L/D models 1A and 1C at $M_\infty = 2.386$, $C_T = 1$, and $\alpha = 0^\circ$.

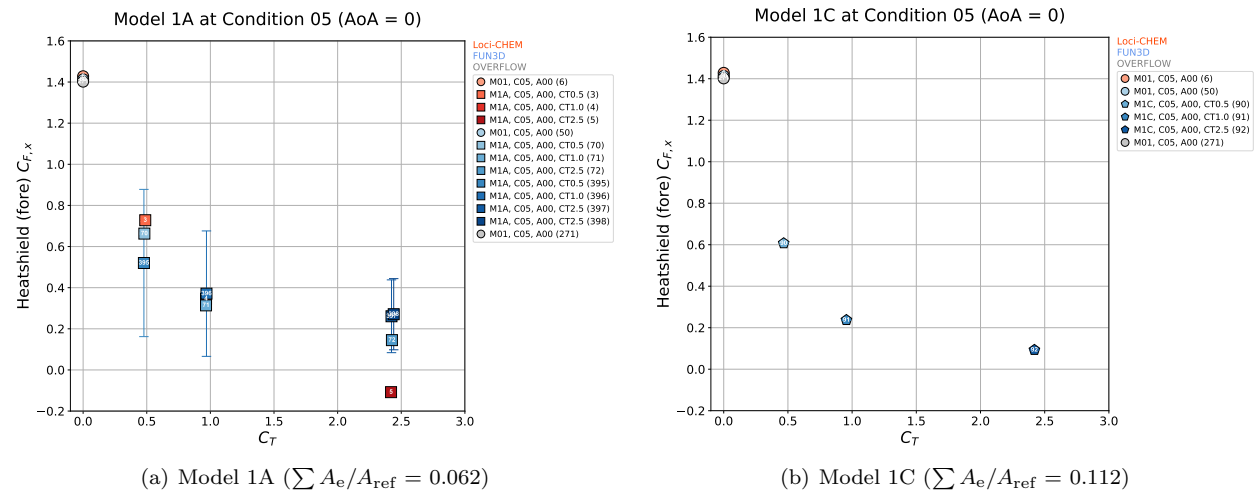


Figure 19. Predicted effect of nozzle size on heatshield aerodynamic axial force coefficient at $M_\infty = 2.386$ and $\alpha = 0^\circ$.

F. Predicted Effect of Nozzle Area Ratio

The predicted effect of nozzle area ratio on the blunt model is accomplished by comparing models 1A (4:1 nozzles) and 1D (11:1 nozzles). The higher area ratio is accomplished by reducing the nozzle throat diameter: the nozzle exit areas are the same. In addition to having a higher exit Mach number, the nozzle with the 11:1 area ratio requires a higher total pressure to achieve the same thrust coefficient. Figures 20 and 21 show predicted flowfields and aerodynamic axial force coefficient from the front side of the heatshield. The predicted trend for axial force coefficient is that a nozzle with a higher area ratio has little effect on the force coefficient compared to the baseline nozzle.

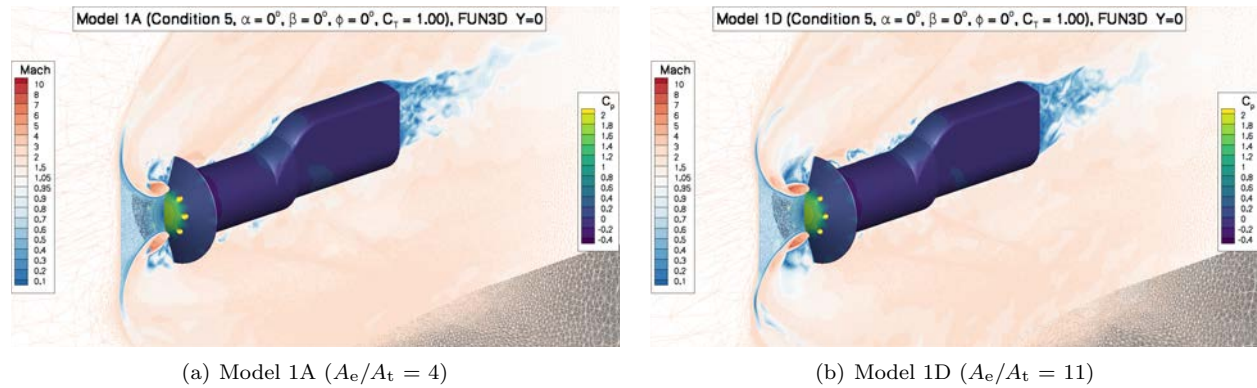


Figure 20. FUN3D predicted time-averaged flowfield Mach number and model surface pressure coefficient for Low-L/D models 1A and 1D at $M_\infty = 2.386$, $C_T = 1$, and $\alpha = 0^\circ$.

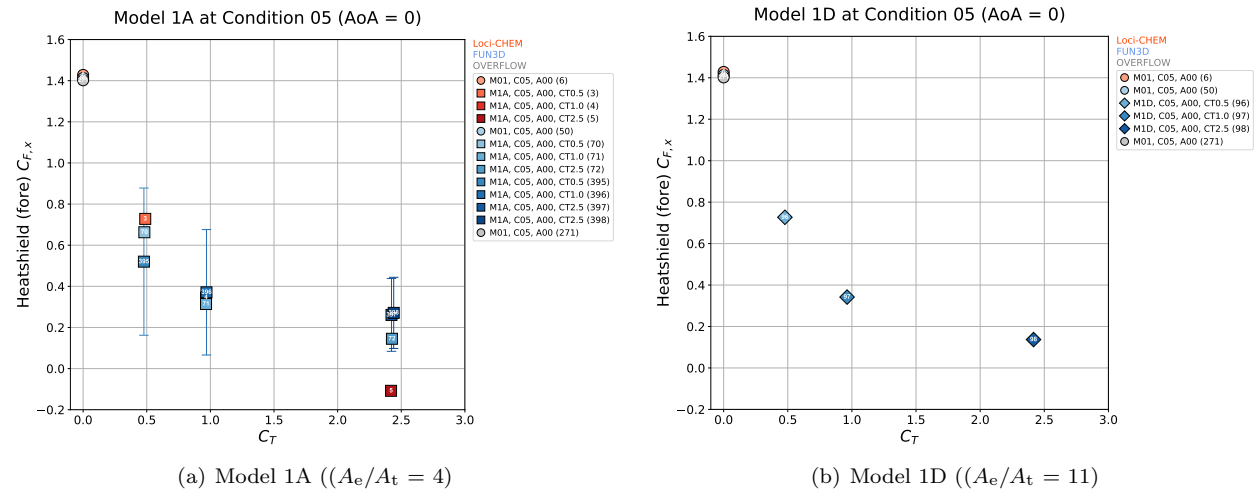


Figure 21. Predicted effect of nozzle area ratio on heatshield aerodynamic axial force coefficient at $M_\infty = 2.386$ and $\alpha = 0^\circ$.

G. Predicted Effect of Nozzle Radial Location

The predicted effect of nozzle location on the blunt model is accomplished by comparing models 1B (nozzles closer to the heatshield nose) and 1E (nozzles further outboard). Both nozzle configurations have a cant angle of 20 degrees. Figures 22 and 23 show predicted flowfields and aerodynamic axial force coefficient from the front side of the heatshield. The predicted trend for axial force coefficient is that nozzles that are further from the heatshield nose results in a higher aerodynamic axial force for thrust coefficients higher than 0.5. The main reason for this result is that the more outboard nozzles permits a larger area of the heatshield to be pressurized by the external flow when the nozzles are blowing.

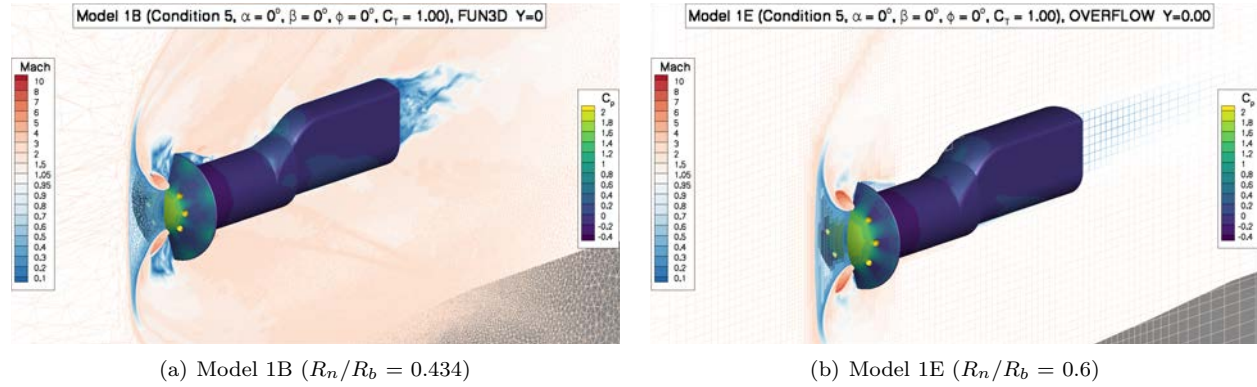


Figure 22. FUN3D (left) and OVERFLOW (right) predicted time-averaged flowfield Mach number and model surface pressure coefficient for Low-L/D models 1B and 1E at $M_\infty = 2.386$, $C_T = 1$, and $\alpha = 0^\circ$.

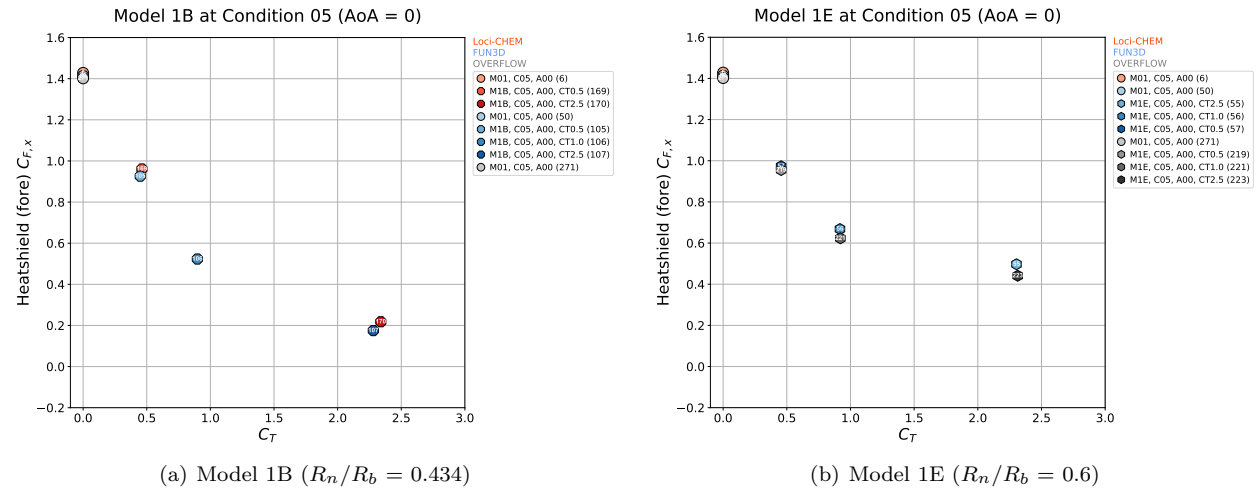


Figure 23. Predicted effect of nozzle location on heatshield aerodynamic axial force coefficient at $M_\infty = 2.386$ and $\alpha = 0^\circ$.

H. Predicted Effect of Nozzle Clustering

The predicted effect of how the Low-L/D nozzle are arranged is accomplished by comparing models 1A (nozzles spaced evenly) and 1F (nozzles clustered in pairs). In Figure 24 for a $C_T = 1$, the effect of the paired nozzles is to block more of the tunnel flow from pressurizing the area inboard of the nozzle exits, compared to the evenly spaced nozzles on model 1A. There is no clear trend from the available CFD solutions of the effect on heatshield aerodynamic axial force coefficient for a given thrust coefficient, as shown in Figure 25.

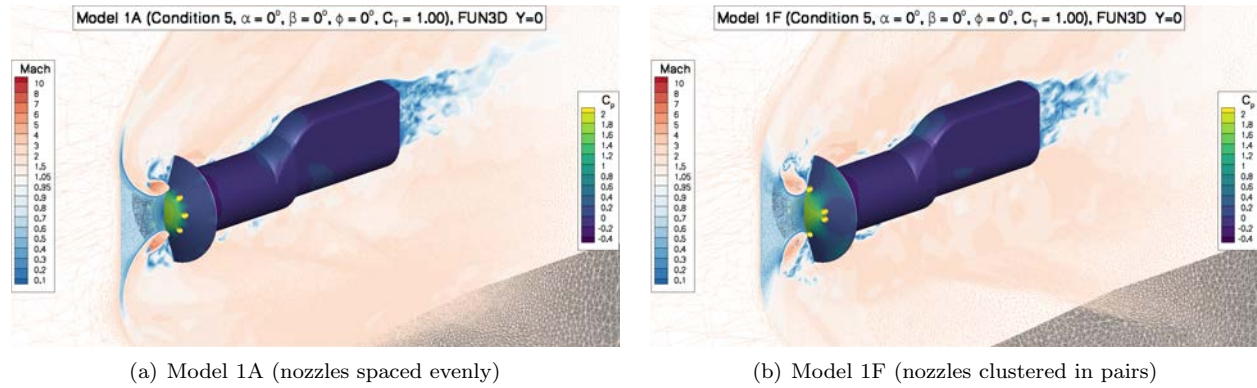


Figure 24. Predicted time-averaged flowfield Mach number and model surface pressure coefficient for Low-L/D models 1A and 1F at $M_\infty = 2.386$, $C_T = 1$, and $\alpha = 0^\circ$.

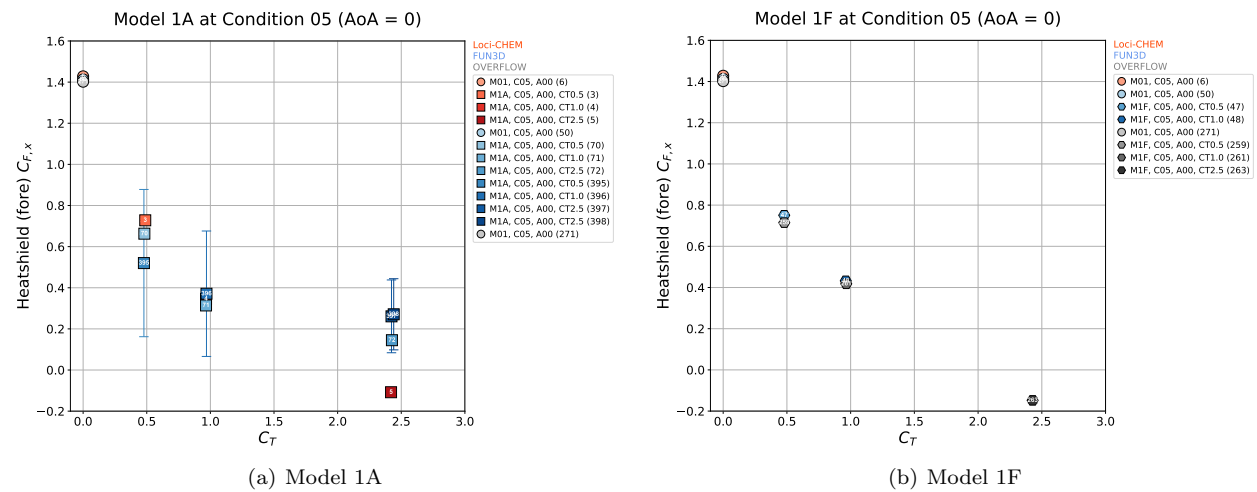


Figure 25. FUN3D predicted effect of nozzle location on heatshield aerodynamic axial force coefficient at $M_\infty = 2.386$ and $\alpha = 0^\circ$.

IV. Summary and Conclusions

A sub-scale test will be run in the NASA Langley Research Center Unitary Plan Wind Tunnel Test Section 2 (Mach number 2.30 to 4.63) to investigate aerodynamic interference effects due to simulated retro-rocket nozzle plumes at supersonic freestream conditions. For Mars human-scale entry system concepts, powered flight is enabling and must begin at supersonic conditions, rather than using a parachute like previous robotic Mars missions. The accurate prediction of aerodynamic interference effects on powered entry vehicle stability and control will be critical for mission success. However, the knowledge base for powered descent aerodynamic interference forces and moments at supersonic conditions is limited to a handful of wind tunnel tests and unvalidated computational flowfield models. Thus, the test is designed to provide retropropulsion test data to assess the accuracy of computational flowfield solvers for retropropulsion applications. The two wind tunnel geometries are scale versions of full-scale concepts and one of the models can accommodate different nozzle characteristics that may impact aerodynamic interference behavior: exit area relative to heatshield area, pointing direction, radial location on the heatshield, and exit-to-throat area ratio. Three flowfield solvers were used to complete a matrix of planned test conditions for both models, to establish best practices and prepare for production runs once the test data available. The main metrics for assessing predictive capabilities will be comparisons to measured forces and moments by using a flow-through balance, discrete steady and unsteady pressures on the heatshield surface, pressure-sensitive paint on the heatshield, and high-speed schlieren video.

To date, over 350 solutions have been completed between three solvers (OVERFLOW, FUN3D, and Loci/CHEM) on computational grids that incorporate non-uniform tunnel test section inflow and the tunnel walls. There are differences in numerical approaches between the solvers that will be addressed after the test is completed. Comparisons between the three solvers for the two non-blowing models generally show the expected level of agreement for heatshield axial force coefficient. For blowing cases with eight nozzles, the level of agreement between solvers varies with nozzle configuration and thrust. The largest differences observed between codes are for the highest thrust coefficient considered. Even with these differences, some consistent trends evident for the aerodynamic interference across the blunt model nozzle configurations. All codes predict that the blunt model heatshield will produce monotonically-decreasing aerodynamic axial force with increasing thrust, regardless of nozzle configuration. If the nozzles are pointed with a radially-outward component rather than directly along the model axis, the heatshield is predicted to generate more axial force. The same trend is observed if the nozzles are placed closer to the heatshield shoulder rather than closer to the nose. If the nozzles are clustered in pairs rather than evenly spaced, there is not a discernible difference in heatshield axial force. Neither nozzle-to-throat area ratio nor nozzle exit size significantly affect the predicted results. Direct comparisons between the test data and predicted results will be used to quantify uncertainties and to guide future investments in retropropulsion testing and analysis.

Acknowledgments

The wind tunnel model design, fabrication, and facility time are funded by NASA's Aerosciences Evaluation and Test Capability program office under the Aeronautics Research Mission Directorate. Most of the CFD calculations were run on NASA Advanced Supercomputing Division computers. CFD analysis to date has been funded by the NASA Space Technology Mission Directorate's Game Changing Development Program.

Nicholas Marino and Donald Morr of Millennium Engineering and Integration Services completed the model designs for fabrication. Veronica Hawke of Science and Technology Corporation prepared the geometries for grid generation software. Thank you to Michael Park of NASA Langley Research Center for geometry and mesh adaptation support of the FUN3D solutions, and Kyle Thompson of NASA Langley for implementing the FUN3D boundary condition necessary for using specified tunnel inflow conditions.

References

- ¹Braun, R. D., and Manning, R. M., “Mars Exploration Entry, Descent, and Landing Challenges,” *Journal of Spacecraft and Rockets*, Vol. 44, No. 2, 2007, pp. 310–323.
- ²Dwyer-Ciancolo, A. M., et al, “Entry, Descent and Landing Systems Analysis Study: Phase 1 Report,” NASA TM 216720, National Aeronautics and Space Administration, July 2010.
- ³Dwyer-Ciancolo, A. M., et al, “Entry, Descent and Landing Systems Analysis Study: Phase 2 Report on Exploration Feed-Forward Systems,” NASA TM 217055, National Aeronautics and Space Administration, February 2011.
- ⁴Edquist, K. T., Korzun, A. M., Dyakonov, A. A., Studak, J. W., Kipp, D. M., and Dupzyk, I. C., “Development of Supersonic Retropropulsion for Future Mars Entry, Descent, and Landing Systems,” *Journal of Spacecraft and Rockets*, Vol. 51, No. 3, 2014, pp. 650–663.
- ⁵Cianciolo, A. D., and Polsgrove, T. T., “Human Mars Entry, Descent, and Landing Architecture Study Overview,” AIAA Paper 2016-5494, September 2016.
- ⁶Cianciolo, A. D., and Polsgrove, T. T., “Human Mars Entry, Descent and Landing Architecture Study: Phase 2 Summary,” AIAA Paper 2018-5191, September 2018.
- ⁷Cianciolo, A. D., Korzun, A., Edquist, K., Samareh, J., Sostaric, R., Calderon, D., and Garcia, J. A., “Human Mars Entry, Descent and Landing Architecture Study: Phase 3 Summary,” AIAA Paper 2020-1509, January 2020.
- ⁸Berry, S. A., Rhode, M. R., and Edquist, K. T., “Supersonic Retropropulsion Validation Experiment in the NASA Langley Unitary Plan Wind Tunnel,” *Journal of Spacecraft and Rockets*, Vol. 51, No. 3, 2014, pp. 664–679.
- ⁹Berry, S. A., Rhode, M. R., and Edquist, K. T., “Supersonic Retropropulsion Experimental Results from NASA Ames 9×7 Foot Supersonic Wind Tunnel,” *Journal of Spacecraft and Rockets*, Vol. 51, No. 3, 2014, pp. 724–734.
- ¹⁰Zarchi, K. A., Schauerhamer, D. G., Kleb, W. L., Carlson, J.-R., and Edquist, K. T., “Analysis of Navier-Stokes Codes Applied to Supersonic Retropropulsion Wind-Tunnel Test,” *Journal of Spacecraft and Rockets*, Vol. 51, No. 3, 2014, pp. 680–692.
- ¹¹Schauerhamer, D. G. Zarchi, K. A., Kleb, W. L. Carlson, J.-R., and Edquist, K. T., “Supersonic Retropropulsion Computational Fluid Dynamics Validation with Langley 4x4 Foot Test Data,” *Journal of Spacecraft and Rockets*, Vol. 51, No. 3, 2014, pp. 693–714.
- ¹²Schauerhamer, D. G. Zarchi, K. A., Kleb, W. L., and Edquist, K. T., “Supersonic Retropropulsion Computational-Fluid-Dynamics Validation with Ames 9x7 Foot Test Data,” *Journal of Spacecraft and Rockets*, Vol. 51, No. 3, 2014, pp. 735–749.
- ¹³Edquist, K. T., Falman, B. E., Watkins, A. Neal, Burns, D. E., and Morr, D. E., “Testing of Mars Retropropulsion Concepts in the Langley Unitary Plan Wind Tunnel,” AIAA Paper 2022-XXXX, January 2022.
- ¹⁴Halstrom, L. D., Pulliam, T. H., Childs, R. E., and Stremel, P. M., “OVERFLOW Analysis of Supersonic Retropropulsion Testing on a Blunt Mars Entry Vehicle Concept,” AIAA Paper 2022-XXXX, January 2022.
- ¹⁵Matsuno, K. V., Childs, R. E., Stremel, P. M., Garcia, J. A., and Pulliam, T. H., “OVERFLOW Analysis of Supersonic Retropropulsion Testing on the CobraMRV Mars Entry Vehicle Concept,” AIAA Paper 2022-XXXX, January 2022.
- ¹⁶Acharya, A. S., Lowe, K. Todd, Danehy, P. M., Edquist, K. T., Burns, R. A., and Pham, H. T., “Seeding Method for Velocimetry and Visualization of Supersonic Retropropulsion Nozzle Plumes,” AIAA Paper 2022-XXXX, January 2022.
- ¹⁷Burns, D. E., Parker, P. A., Cagle, C. M., Hawke, V. M., Tor, K. G., and Kleb, B., “A New Flow-Through Wind Tunnel Balance for Retropropulsion Testing,” AIAA Paper 2022-XXXX, January 2022.
- ¹⁸Rhode, M. N., et al, “Flow Characterization of the NASA Langley Unitary Plan Wind Tunnel, Test Section 2: Experimental Results,” Aiaa presentation, August 2021.
- ¹⁹Childs, R. E., et al, “Flow Characterization of the NASA Langley Unitary Plan Wind Tunnel, Test Section 2: Computational Results,” AIAA Paper 2021-2963, August 2021.
- ²⁰Hubbard, E. P., and Houlden, H. P., “Evaluation of CFD as a Surrogate for Wind-Tunnel Testing: Experimental Uncertainty Quantification for the UPWT Flow Survey Test,” AIAA Paper 2021-2962, August 2021.
- ²¹Korzun, A., Canabal, F., Tang, C., Childs, R., Van Norman, J., Tynis, J., and Bibb, K., “Powered Descent Aerodynamics for Low and Mid Lift-to-Drag Human Mars Entry, Descent and Landing Vehicles,” AIAA Paper 2020-1510, January 2020.
- ²²Nichols, R. H. and Buning, P. G., “User’s Manual for OVERFLOW 2.3,” February 2021.
- ²³Rogers, S. E., Roth, K., Nash, S. M., Baker, M. D., Slotnick, J. P., Whitlock, M., and Cao, H. V., “Advances in Overset CFD Processes Applied to Subsonic High-Lift Aircraft,” AIAA Paper 2000-4216, 2000.
- ²⁴Chan, W. M., “The Overgrid Interface for Computational Simulations on Overset Grids,” AIAA Paper 2002-3188, 2002.
- ²⁵Chan, W. M., “Developments in Strategies and Software Tools for Overset Structured Grid Generation and Connectivity,” AIAA Paper 2011-3051, 2011.
- ²⁶Buning, P. G., and Pulliam, T. H., “Near-Body Grid Adaption for Overset Grids,” AIAA Paper 2016-3326, 2016.
- ²⁷Menter, F. R., “Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications,” *AIAA Journal*, Vol. 32, No. 8, 1994, pp. 1598–1605.
- ²⁸Spalart, P. R., “Strategies for turbulence modelling and simulations,” *International Journal of Heat and Fluid Flow*, Vol. 21, No. 3, 2000, pp. 252–263.
- ²⁹Mani, M., Babcock, D. A., Winkler, C. M., and Spalart, P. R., “Predictions of a Supersonic Turbulent Flow in a Square Duct,” AIAA Paper 2013-860, January 2013.
- ³⁰Nichols, R. H., “Algorithm and Turbulence Model Requirements for Simulating Vortical Flows,” AIAA Paper 2008-337, 2008.
- ³¹Suzen, Y., and Hoffmann, K., “Investigation of Supersonic Jet Exhaust Flow by One- and Two-Equation Turbulence Models,” AIAA Paper 1998-322, January 1998.
- ³²Childs, R., et al, “Flow Characterization of the NASA Langley Unitary Plan Wind Tunnel, Test Section 2: Computational Results,” AIAA Paper 2021-2963, 2021.

- ³³Luke, E., and George, T., “Loci: A Rule-Based Framework for Parallel Multidisciplinary Simulation Synthesis,” *Journal of Functional Programming*, Vol. 15, No. 3, May 2005, pp. 477–502.
- ³⁴Luke, E., *A Rule-Based Specification System for Computational Fluid Dynamics*, Master’s thesis, Mississippi State University, December 1999.
- ³⁵Biedron, R. T., et al, “FUN3D Manual: 13.7,” NASA TM 2020-5010139, Nov. 2020.
- ³⁶Spalart, P. R., Deck, S., Shur, M. L., Squires, K. D., Strelets, M. Kh., and Travin, A., “A New Version of Detached-Eddy Simulation, Resistant to Ambiguous Grid Densities,” *Theoretical and Computational Fluid Dynamics*, Vol. 20, No. 3, May 2006.
- ³⁷Spalart, P. R., “Strategies for Turbulence Modeling and Simulations,” *International Journal of Heat and Fluid Flow*, Vol. 21, No. 3, June 2000, pp. 252–263.
- ³⁸Shur, M. L., Strelets, M. K., Travin, A. K., Spalart, P. R., “Turbulence Modeling in Rotating and Curved Channels: Assessing the Spalart-Shur Correction,” *AIAA Journal*, Vol. 38, No. 5, 2000, pp. 784–792.
- ³⁹Vatsa, V. N., and Lockard, D. P., “Assessment of Hybrid RANS/LES Turbulence Models for Aeroacoustics Applications,” AIAA Paper 2010-4001, June 2010.
- ⁴⁰Van Leer, B., “Towards the Ultimate Conservative Difference Scheme. V. A Second-Order Sequel to Godunov’s Method,” *Journal of Computational Physics*, Vol. 32, No. 1, 1979, pp. 101–136.
- ⁴¹Kleb, W. L., Schoenenberger, M., Korzun, A. M., and Park, M. A., “Sketch-to-Solution: A Case Study in RCS Aerodynamic Interaction,” AIAA Paper 2020-0673, January 2020.