CFD2030 Grand Challenge: CFD-in-the-Loop Monte Carlo Simulation for Space Vehicle Design

David M. Schuster[1]
NASA Engineering and Safety Center, Hampton, Virginia 23188 USA

Space vehicle design and certification differs widely from aircraft design relying more on probabilistic approaches than deterministic. Monte Carlo simulation plays an important role in the probabilistic design of space vehicles to ensure robust and reliable operation. Today, Monte Carlo flight simulation requires 1000's of trajectory simulations that use databases to provide aerosciences models. These databases can be extremely expensive and time consuming to develop. Replacing these databases with unsteady computational fluid dynamics directly in the simulation loop has potential to significantly reduce the time required to analyze space vehicle concepts, improve simulation accuracy, and reduce the cost of space vehicle development. The CFD Vision 2030 Study outlined gaps and roadblocks to meeting the vision described in the study. The geometric, physical, and computational challenges associated with CFD-in-the-loop Monte Carlo simulation for space vehicle design are substantial and serve as an excellent grand challenge to advance the CFD 2030 vision.

I. Nomenclature

CFD	=	Computational Fluid Dynamics
CM	=	Crew Module
CREATE	=	Computational Research and Engineering Acquisition Tools and Environments
CREATE-AV	=	CREATE Air Vehicle Component
DES	=	Detached Eddy Simulation
DoD	=	United States Department of Defense
DSMC	=	Direct Simulation Monte Carlo
EDL	=	Entry, Descent, and Landing
HPC	=	High Performance Computing
HPCMP	=	DoD HPC Modernization Program
HRLES	=	Hybrid RAN-LES
LES	=	Large Eddy Simulation
MOR	=	Model Order Reduction
MSL	=	Mars Science Laboratory
OTIS	=	Optimized Trajectories by Implicit Simulation
POST	=	Program to Optimize Simulated Trajectories
RANS	=	Reynolds Averaged Navier-Stokes
RCS	=	Reaction Control System
ROM	=	Reduced Order Modeling
SLS	=	Space Launch System
URANS	=	Unsteady Reynolds Averaged Navier-Stokes
6DOF	=	Six degrees-of-freedom

[1] NASA Technical Fellow for Aerosciences, Associate Fellow, AIAA.

II. Introduction

Flight qualification of space vehicles is markedly different from aircraft flight certification. The concept of an extensive flight test campaign for a space vehicle does not exist, and vehicle designers must look to alternative techniques for demonstrating robust and reliable performance of their vehicles prior to operational flight. A space vehicle may undergo only a handful of flight tests in its development cycle, with each test representing a drastically different flight phase or flight configuration. For instance, NASA's Space Launch System (SLS) launch vehicle [1] and Orion spacecraft [2] will only see a total of four flight demonstrations before flying a crew on its first operational mission, and each flight demonstrates a unique vehicle configuration and/or set of flight conditions. The SLS will be flown only one time before it begins flying crews (Artemis 1). The Orion spacecraft crew module (CM) will have been tested twice, once on a Delta IV launch vehicle (Exploration Flight Test 1) and once as a fully integrated system with the SLS launch vehicle (Artemis 1). The Orion Launch abort system will have been tested twice, once in a pad abort scenario (Pad Abort 1) and once in an inflight abort scenario (Ascent Abort 2) on a modified Peacekeeper booster. Both of these latter tests involve only a boiler plate CM, not a functional Orion spacecraft. Thus, unlike aircraft, there is very little opportunity for engineers to assess and evaluate their preflight predictions.

Instead, space vehicle designers rely on Monte Carlo flight simulations [3] with detailed dispersions of predicted nominal flight behavior to determine how robust their design is to errors and uncertainties in system operation and the flight conditions their vehicle may encounter. These Monte Carlo analyses can entail thousands of trajectory simulations to demonstrate that the vehicle meets design requirements to a specified level of reliability and confidence. From an aerodynamics and aerothermodynamics perspective, these trajectory simulations are fueled by an extensive aerodynamic database that covers the complete range of flight conditions, vehicle configurations, and flight attitudes expected in a given mission [4][5]. Today, these databases amount to a table of engineering parameters that can be quickly interrogated by the trajectory simulator. The aerodynamic and aerothermodynamic databases, herein collectively referred to as aerosciences databases, are assembled via a series of ground tests, empirical and analytical analysis, physics-based computational analysis, applicable past flight performance data, and in some cases, engineering judgment. These databases generally take years to assemble for a new space vehicle system and in the case of SLS/Orion, over a decade of test and analysis have been expended to develop the extensive databases required to cover the myriad of configurations and potential flight conditions required for the operational system.

The primary shortcoming of the database approach is the long time required to develop a high fidelity aerosciences model suitable for flight simulations. When developing atmospheric flight simulations for Monte Carlo analyses, a primary requirement is always the delivery of an aerosciences database. While vehicle developers can mitigate the time required to develop the full database by using preliminary databases, it can still take months to several years to acquire these models. Any opportunity to reduce the time required to develop the aerosciences database allows designers to simulate their vehicles earlier and thus accelerate the design process. Aerosciences databases can be extremely large, often with ten or more independent parameters, depending on the flight condition covered by the dataset, for which engineering flight data must be predicted in a multi-dimensional matrix. The cost to develop these databases can be high, involving multiple wind tunnel tests and tens of thousands of CFD simulations.

In nominal operation, the vehicle typically only flies through a narrow corridor of the database. However, to effectively evaluate whether a vehicle is robust to dispersions in flight conditions and modeling errors, a database must cover a relatively broad range of flight conditions away from the nominal performance since the vehicle may be unstable or experience a system failure within the dispersed flight envelope. This results in databases where only a small fraction of the database is exercised on a regular basis and the majority of the rest of the database only gets accessed in the most extreme flight situations. As an example of the application of Monte Carlo flight simulation, Figure 1 shows an envelope of approximately 1000 trajectory simulations for a notional abort vehicle [6] where Mach number, dynamic pressure, altitude, and total angle-of-attack are responding to dispersions of uncertainties in system models, flight conditions, etc. In general the trajectories are constrained to a narrow corridor with the exception of total angle-of-attack late in the simulation where the vehicle exhibits reduced pitch stability. To fully cover the possible flight regime over which the vehicle may operate, extensive resources can be expended to develop areas of a database that is only lightly exercised. Lastly, spaceflight from an aerodynamics/aerothermodynamics standpoint is largely a highly transient event and compared to most aircraft, the vehicle moves quickly from one flight state to another. Figure 1 further illustrates this point in the rapid variation of Mach number, dynamic pressure and total angle-of-attack during the trajectory simulations. The present database approach is, at best, quasi-steady, and the databases themselves are based on steady or averaged aerosciences data. Thus the transient character of spaceflight may not always be accurately captured by the present database approach.

Recently, it has been proposed that Computational Fluid Dynamic (CFD) and computing capability may be reaching a point where it is foreseeable that CFD could be integrated directly into the production trajectory simulation tools used to design NASA's space vehicles. There are numerous advantages to pursuing this capability. First and foremost, maturing this capability would allow the vehicle designer to evaluate the performance of their designs much earlier in the process than presently available. The ability to simulate concept trades without having to develop an aerosciences database allows the designer to shave weeks, months, and even years off the time required to developed, and constrained only to the required flight parameters by simply outputting the data generated by the CFD as the simulation progresses. This approach has the added advantage that at specific points in the parameter space, differences in aerosciences data due to the transient nature of the flight can be included in the resulting database as uncertainties.

Since 2008, the Department of Defense High Performance Computing Modernization Program (DoD HPCMP) has been addressing the CFD-in the loop simulation problem for aircraft through their air vehicle component of the Computational Research and Engineering Acquisition Tools and Environments (CREATE) program known as CREATE-AV [7]. This effort has accomplished substantial progress toward development of an integrated CFD/flight simulation capability utilizing existing and emerging HPC technology. To demonstrate the capability for space vehicles, NASA has embarked on two demonstrations of this type, one where flight trajectory simulation equations are embedded in an existing CFD solver [8][9] and another where a production CFD solver is loosely coupled with a production trajectory simulation tool [10]. These efforts represent credible demonstrations of a future approach to flight trajectory simulation, but they remain far from the capability required to perform a full-up CFD-in-the-loop Monte Carlo trajectory simulation. At present, there are challenges in flow physics, geometry modeling and grid generation, automation, and computing that prevent this approach from being implemented in even a demonstration environment, let alone a production design environment. Therefore, this represents a viable grand challenge for computational methods addressing space vehicle design and development.

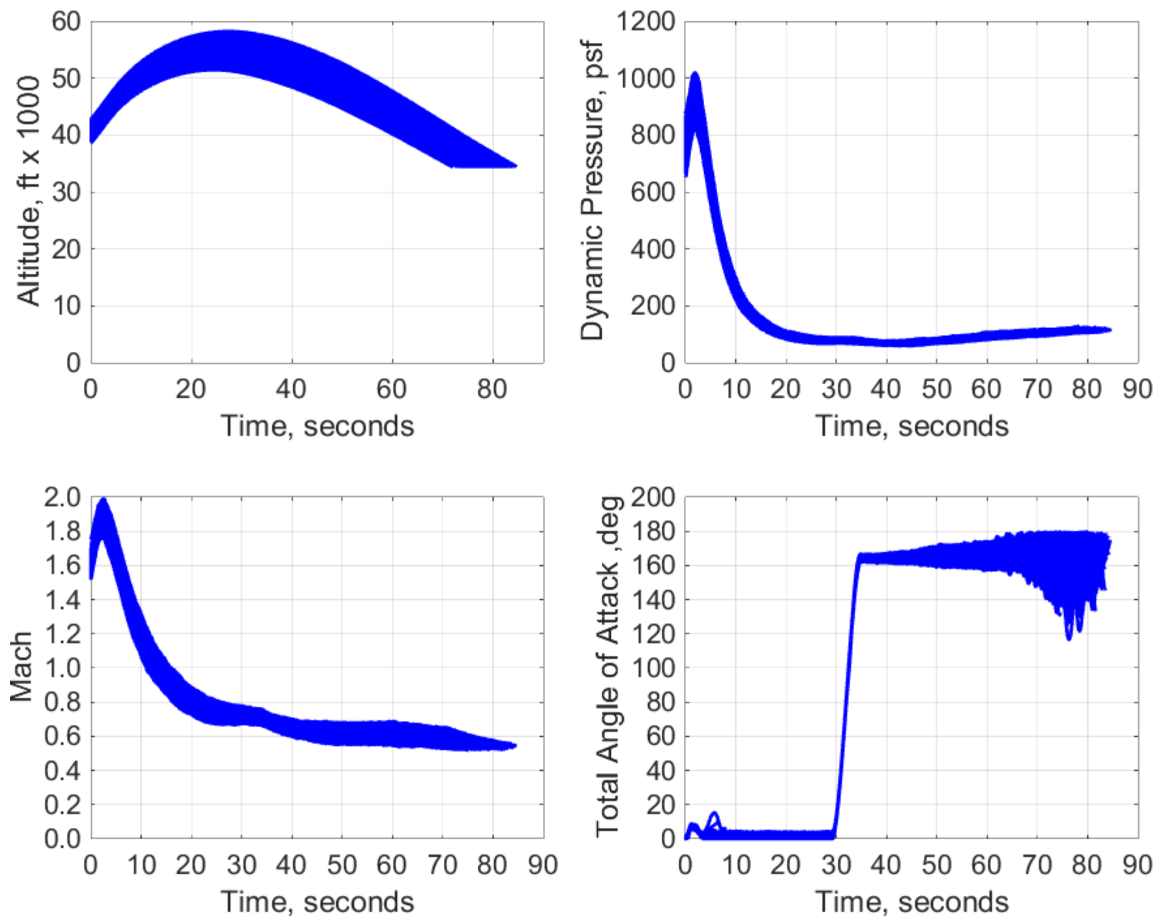

Fig. 1 Envelope of Monte Carlo trajectory simulations for a notional launch abort vehicle.

III. Space Applications of Interest

There are two primary areas of spaceflight that would benefit from large-scale use of CFD-in-the-loop Monte Carlo Simulation: ascent and Entry, Descent and Landing (EDL). The nominal ascent phase of flight usually begins with a vehicle sitting stationary on a launch pad and rapidly accelerating through an atmosphere to non-atmospheric flight in space. For vehicles carrying human crew, there is often a contingency phase of ascent, known as abort, that could be required in the event of a major vehicle malfunction during nominal ascent. Abort systems can often be considered separate ascent systems and their operation and flight conditions can be much different than the nominal ascent vehicle. For atmospheric entry systems, the EDL phase of flight involves transition from flight without an atmosphere at extremely high velocities, through hypersonic, supersonic, transonic, and subsonic conditions to landing on a hard, typically unprepared and often unknown, surface or splashdown in a large body of water. Numerous control and decelerator systems can be employed during EDL with their own complex physics considerations. Landing systems can be highly varied and also present challenges to effective simulation.

Ascent and EDL have one basic feature in common that separates them from many aircraft simulation problems. They must both fly through the complete speed regime from low subsonic to hypersonic flight and they generally have little ability to tailor their flight trajectory to avoid flight regimes that can produce undesirable or difficult to predict flow physics. Thus space vehicles routinely encounter massive unsteady smooth body separated flow, shock-induced dynamics due to moving shocks, and shock-induced separated flow. Aerodynamic heating is always a consideration, particularly for the entry phase of flight, and prediction of propulsive and control plume impingement effects is also a challenge. Space vehicles are typically not optimized for aerodynamic considerations, so vehicle geometries can be aerodynamically cumbersome and a challenge to model, both from a geometry standpoint and a flow physics standpoint.

A. Ascent Systems

Ascent is usually defined as the phase of flight beginning with transition from a stationary position on the launch pad to high speed flight, ultimately propelling the vehicle to orbit or a trajectory apogee. For reference, an example of an ascent trajectory for NASA's Space Launch System is presented in Fig. 2. Prior to engine ignition, the launch vehicle is subject to ground winds, which place the vehicle in a high angle-of-attack, low subsonic flow broadside to the vehicle. For so-called single-stick systems, like the SpaceX Falcon 9 or NASA's Saturn V, this amounts to essentially low speed flow past a circular cylinder and, depending on the flow conditions, can be quite dynamic and unsteady in nature. For multi-stick configurations like NASA's SLS or the Space Shuttle, the wind direction with respect to the vehicle can make a large difference to the geometry presented to the flow. While most launch pads have predominant wind speeds and directions based on the time of year, in general all contingencies for ground winds must be accounted for to ensure a vehicle is robust for launch. This phase of flight is one area where wind placards can be applied and flight managers can partially control the conditions into which the vehicle is being launched. Winds aloft are also taken into account on launch day, as are general meteorological conditions, and the vehicle can be placarded to ensure it is not launched into unfavorable conditions.

As the vehicle accelerates away from the pad, the relative wind angle quickly transitions from a broadside, 90 degree angle-of-attack, condition to a near zero angle-of-attack condition with wind almost directly on the launch vehicle (LV) nose. Immediately after clearing the launch pad the vehicle often conducts a flyaway maneuver, which can be a combination of pitch and roll to maneuver away from the launch pad and align itself with the proper flight trajectory to attain orbit and facilitate other flight operations. This maneuver is usually conducted via vectoring of the main engines and/or a separate roll control system. This introduces the possibility of plume interactions and impingements that can influence aerodynamic performance and heating and can be quite difficult to accurately predict. In addition, as the vehicle accelerates through the transonic and low supersonic speed regimes, unsteady flow due to shock movement and oscillations, as well as separated flow around vehicle protuberances and shock boundary layer interactions generate buffet and aeroacoustic loads that can be harmful to the vehicle crew, structure, and flight systems. Again, unlike aircraft, the vehicle cannot substantially adjust flight conditions to avoid or reduce these phenomena, and the vehicle must be designed to be robust to them.

Many launch vehicles are multi-stage and involve some form of staging along the ascent flightpath. In this flight phase, engines are usually shut down due to exhausted fuel supply and the vehicle fuel tanks and engines are discarded to save weight and allow the vehicle to attain its design performance objectives. Once the inoperable components are discarded, engines on the next vehicle stage are ignited to carry the vehicle further along its trajectory. Staging can be a complex maneuver and proper prediction of aerodynamic performance during the staging transition is crucial to ensuring the vehicle will meet design objectives. Transient aerosciences data due to the close proximity of the main vehicle and the expended stage as it moves away from the vehicle are important to developing a robust staging process.

During this process, not only are the flight conditions of the problem changing rapidly, but also the geometry of the problem as the stages separate and the close proximity of the two bodies significantly influence the aerosciences data. In the case of solid booster separation, as required for SLS, the boosters are pushed away from the vehicle using laterally firing rocket motors whose plumes can impinge on the main vehicle, further complicating the prediction of the separation. Ignition of the next stage's engines and potential interaction with the separated stage must also be accounted for in this stage of the flight.

For orbital vehicles, as the launch vehicle continues to ascend through the atmosphere, the flow density ultimately reduces to a point where continuum flow assumptions are no longer valid and rarefied gas assumptions must be utilized to predict the final aerosciences data prior to the vehicle entering orbit. Thus the flow physics change and the associated predictive tools change as the vehicle nears orbit.

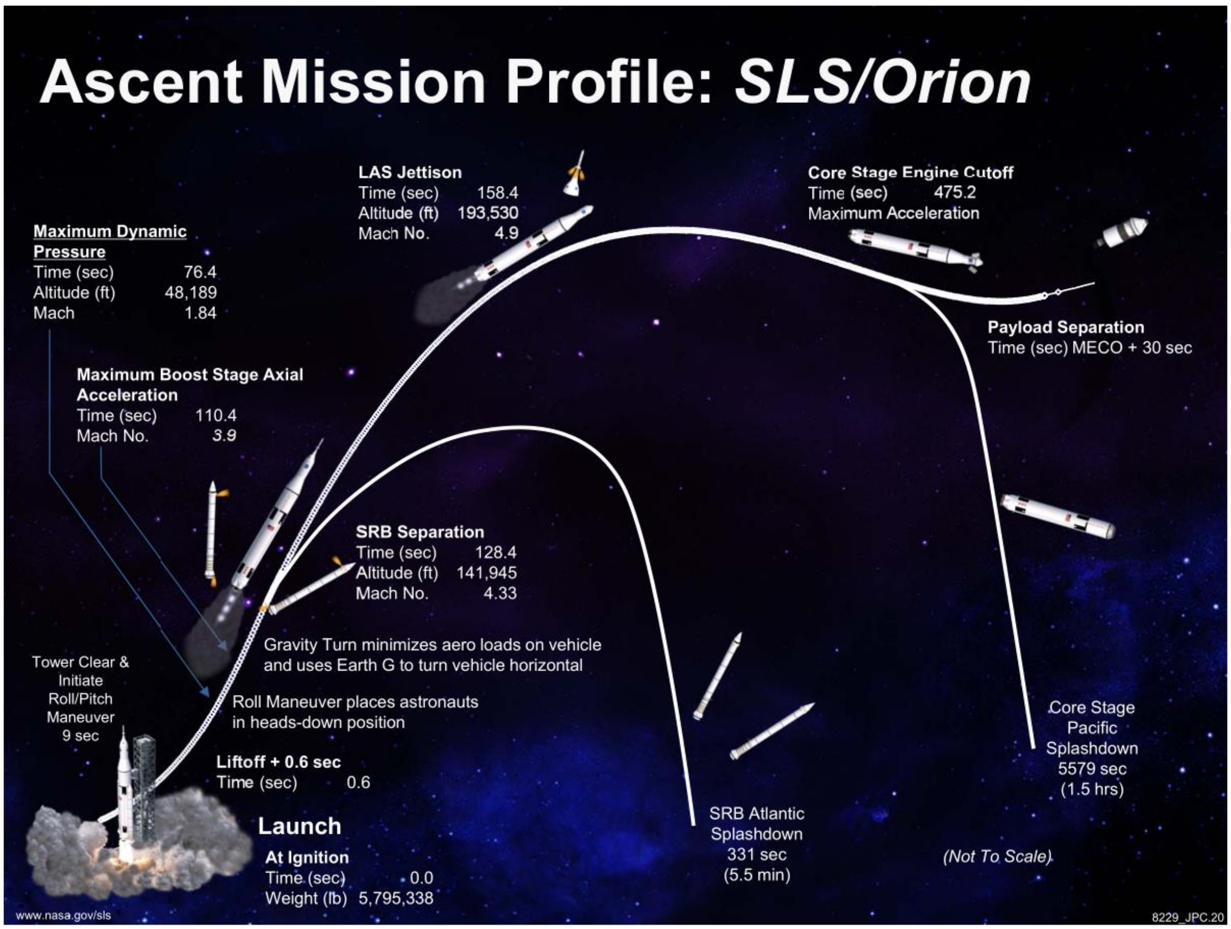

Fig. 2 Ascent trajectory and events for the NASA Space Launch System.

Human rated vehicles usually include an abort system as a contingency for crew escape should the launch vehicle experience a failure. These systems are designed to provide crew rescue at any point in the flight from the vehicle stationary on the launch pad, to anywhere along the trajectory, all the way to orbit. Early abort vehicle designs used a tractor system where rocket motors are attached to an escape tower above the crewed spacecraft, and NASA's Orion spacecraft continues to use this type of system today. More recently, commercial vehicles have moved to a pusher system, which places the abort motors on the side of, or behind, the spacecraft to minimize the complex engine plume interactions directly on the spacecraft. A concept of operations for a pusher type launch abort system concept [11] is shown in Fig. 3. With either system, the potential for strong interactions of the spacecraft with the abort motor plumes is an important and difficult problem to analyze. Abort systems must also be relatively maneuverable as compared to their launch vehicle counterparts. Abort systems separate from the launch vehicle by propelling themselves away from the failing vehicle. Their geometry and mass properties sometimes lead to an unstable configuration requiring additional active or passive control afforded by propulsive reaction controls or mechanical surfaces. Abort vehicles start by flying nose forward, but at some point in their trajectory usually have to reorient to a nose aft position to facilitate separation of the spacecraft so that it can activate its landing systems. Thus aerosciences analysis must be able to predict full 180 degree angle-of-attack range of performance for the abort system over a wide range of flight Mach numbers from low subsonic to high supersonic. Like the launch vehicle, staging events and component separations are routine, so large transient changes in geometry and configuration must be accommodated, and propulsive interactions can be extreme.

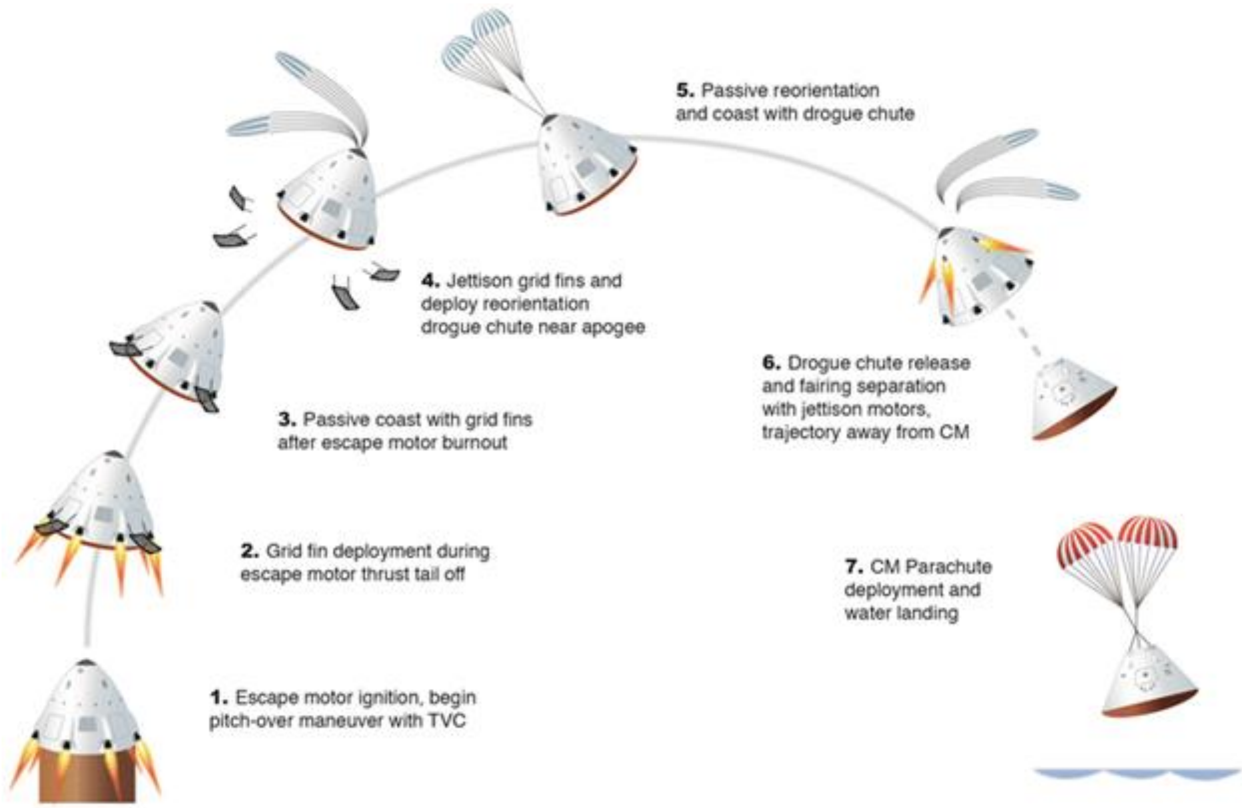

Fig. 3 Concept of operations for a pusher launch abort vehicle configuration.

B. Entry Systems

Vehicles entering the Earth's or other planetary atmospheres encounter their own unique set of aerosciences problems. Figure 4 shows the EDL sequence of events for the Mars Science Laboratory (MSL) [12]. In the early phases of EDL aerodynamic heating at hypersonic speeds is of primary concern. At entry interface, the atmosphere is usually too thin to effectively employ continuum methods, so rarefied gas simulation tools, like Direct Simulation Monte Carlo (DSMC) [13], are required at the upper reaches of the atmosphere before switching to more conventional continuum flow methods. Entry temperatures are often high enough where chemically reacting effects are important to the analysis.

Vehicle geometry is often relatively simple, as in capsule configurations, though lifting bodies like the space shuttle have been employed resulting in a proportionately more complex geometry. Protuberances and other surface discontinuities are minimized to avoid augmented heating caused by these features. Deployable decelerators such as inflatable and mechanical heatshields, ballutes, or parachutes can make the geometry modeling problem for entry more complex and difficult. Fluid structure interaction issues can be of critical importance in the aerosciences analysis of most deployable decelerators.

Vehicle stability during entry is also a primary concern as most capsule configurations are only marginally stable, requiring some form of reaction control system (RCS) to both maintain optimal vehicle orientation during entry, as well as to perform maneuvers to manage energy as the vehicle decelerates through the atmosphere. Plumes generated by RCS firings can interact with each other and with the vehicle external aerodynamics in complex modes and affect vehicle stability and control response. Thus accurate models of the RCS plume interaction with vehicle aerodynamics can be very important to predicting the vehicle performance and targeting for precision landing. Entry vehicles often change their orientation during entry to take optimal advantage of their flight conditions. This is sometimes accomplished by changing the mass distribution during entry, thus moving the center of gravity, as employed on MSL, or it can also be afforded by the use of aerodynamic flaps and other control surfaces.

Terminal descent and landing systems can be highly varied and complicated based on the approach chosen or the mission objectives and the landing precision requirements. Most systems use some type of propulsive landing system or parachutes. For propulsive systems, plume surface interactions can become extremely important for guidance and precision landing targeting, as well as the potential to damage the vehicle or its systems with surface debris churned up by the propulsion system. Parachutes present an extremely complex fluid/structure interaction problem and their performance can be especially hard to predict.

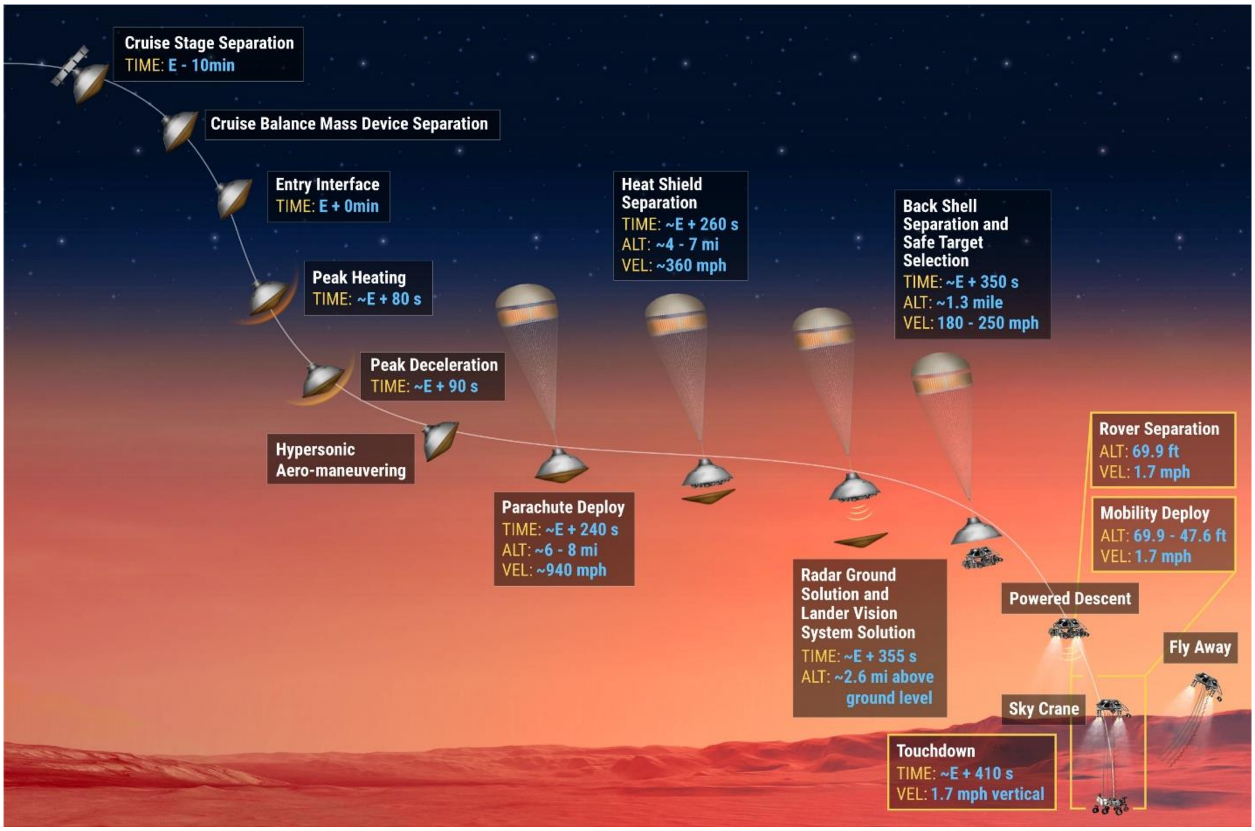

Fig. 4 EDL Sequence of events for the Mars Science Laboratory.

IV. CFD Vision 2030 Study Technology Gaps and Impediments

The CFD Vision 2030 Study [14] identified five principle gaps and impediments to meeting CFD tool development goals by 2030:

1. Effective Utilization of High Performance Computing (HPC)
2. Unsteady Turbulent Flow Simulation Including Transition and Separation
3. Autonomous and Reliable CFD Simulation
4. Knowledge Extraction and Visualization
5. Multidisciplinary/Multiphysics Simulations and Frameworks

The study also calls for efforts to address a series of grand challenges. The proposed challenge of CFD-in-the-loop Monte Carlo flight simulation directly addresses Grand Challenge Problem 4 in the study: Probabilistic Analysis of a Powered Space Access Configuration. While the proposed grand challenge must overcome elements of all five identified gaps and impediments, gaps 1, 2, 3, and 5 above present the greatest roadblocks to successfully completing this challenge.

V. Challenging Flow Physics for Space Vehicles

Space vehicle aerosciences encompasses the majority of critical flow phenomena addressed in the CFD Vision 2030 study and a few that were not explicitly called out, such as accelerating/transient flight, RCS/rocket plume aero interactions, and plume surface interactions. Among the most challenging flow physics problems for both the ascent and EDL phase of spaceflight are unsteady flows generated by smooth body separation and shock induced separation. For ascent, these flows manifest themselves while the launch vehicle is on the pad experiencing and responding to ground winds. Later in the ascent phase of flight, transonic and low supersonic flow past geometric transitions on the launch vehicle and protuberances can cause separated flow leading to aerodynamic buffet at frequencies less than a few hundred Hz and aeroacoustic response at much higher frequencies. Abort vehicles experience similar challenges. These phenomena can impact the vehicle's crew, structure, and critical flight systems. Smooth body separation is a difficult problem for EDL as well. Bluff bodies are used to mitigate aerodynamic heating and present heatshields to the oncoming flow. These bluff geometries generate smooth body separation where the vehicle transitions from the forebody heatshield onto afterbody, often leaving the entire afterbody in a massively separated wake flow. These separated wakes have a first-order impact on the vehicle's aerodynamic performance, stability, and heating.

Historically, environments generated by smooth body separated flow have been predicted using ground test and empirical flight data. More recently advanced CFD methods like Detached Eddy Simulation (DES) have shown significant utility in predicting launch vehicle buffet [15] and aeroacoustic problems and in some isolated cases to the separated wake EDL problems as well. DES and other Hybrid Reynolds Averaged Navier-Stokes - Large Eddy Simulation (HRLES) application is becoming more commonplace, but the grid resolution and computational resources to perform these simulation are restricting their broad usage for database construction and general vehicle design and development.

Aerodynamic/plume interaction is also a leading flow physics challenge for space vehicles in both the ascent and EDL phases of flight. Vehicle control is often maintained via thrust vectoring and/or propulsive reaction control systems. Aero/plume interaction has implications for launch vehicle afterbody heating as main engine plumes expand and are vectored for control. Also, at high altitudes where plumes become under-expanded, Plume-Induced Flow Separation (PLIFS) can occur, significantly complicating the aerosciences analysis of the vehicle [16]. Some launch vehicles and most entry vehicles employ some form of propulsive RCS to maintain and control vehicle orientation. These jets are usually operated for brief bursts of time in what is known as a bang-bang control system, so the control events are very transient in nature. Aerodynamic interactions have been shown to influence control authority [17], and in extreme cases, produce control reversal from the expected control input. These systems can also exhaust into separated flows with very unpredictable results. This forces designers to employ high levels of control authority uncertainty in their flight simulations. For abort vehicles, particularly tractor designs as used on NASA's Orion Launch Abort Vehicle (LAV), plume/plume, plume/vehicle, and plume/aerodynamic interactions produce a myriad of physics challenges in predicting LAV performance [18]. Adding to the issues already discussed, plume chemistry and chemically reacting flows in the plume can be important to vehicle performance and control prediction. Plume impingement on other plumes and the vehicle surface can produce vehicle loads that are difficult to predict and sometimes counter intuitive. In all these simulations, plume resolution can become an important factor to an accurate analysis, again pushing the community toward HRLES simulation to tackle the most difficult and critical interaction issues.

Time accurate simulation is a must for properly simulating all but the most mildly separated flows, and time accurate simulations can also be important to properly simulating many aero/plume interaction problems. The majority of CFD conducted for space vehicle performance prediction utilize steady RANS analysis. Many cases computed using this technique exhibit moderate unsteadiness in the resulting aerodynamic coefficients, even though the simulation attempts to converge to a steady state. Thus, some form of averaging technique is required to obtain reliable and repeatable aerodynamic performance coefficients and the unsteadiness encountered in the steady analysis must be factored into the prediction uncertainty at these conditions. In many cases, time-accurate unsteady RANS (URANS) is required to more accurately derive coefficients when steady simulations exhibit high levels of unsteadiness.

For the majority of space vehicle aerosciences problems of interest, vehicles accelerate rapidly through their flight regimes. Transient effects are not accurately captured by most implementations of RANS and URANS methods that hold parameters like Mach number, Reynolds number, freestream pressure, etc. constant during the analysis. Space vehicle maneuvers involving rigid body motion, stage separation, accelerating flight, and transient propulsion and RCS implementation further emphasize the need for time accurate CFD simulation. Looking forward, future routine CFD analysis should move away from steady flow implementations and focus on the development and application of time accurate simulations for complex flows, interactions, and dynamic vehicle performance.

Lastly, aerodynamic decelerators present a particularly difficult set of challenges to the CFD community. Space vehicles have long used parachutes operating in both the subsonic and supersonic speed regimes to augment deceleration during EDL. As we envision sending larger robotic payloads and human missions to Mars and other destinations with atmospheres, more capable aerodynamic decelerators are required as parachute scaling breaks down for very large payloads. Concepts like deployable heatshields using inflatable [19] or mechanical [20] systems allow spacecraft to greatly increase their surface area to improve vehicle deceleration as it enters the atmosphere, and at the same time, mitigate aerodynamic heating. Retro-propulsion systems operating as supersonic conditions are also being considered to augment vehicle deceleration during EDL [21]. The fluid dynamics considerations for these systems can be quite complex, involving massively separated wakes, and complex fluid structure interaction for parachutes and deployable heatshields, and chemically reacting aero/plume/vehicle interaction for retro-propulsion systems. Reference [22] provides a recent DES analysis of a human Mars lander using retro-propulsion deceleration and outlines the technical challenges and resource requirements to address this problem.

These challenges will continue to push CFD development toward higher physical fidelity from the current RANS state-of-the-art toward maturation of DES and HRLES technologies for mainstream application. In the near-term, computational technology and capacity will also drive research in adaptive physics to optimize the use of the most computationally intensive methods to the problems that are most dependent on, and sensitive to, highly detailed flow prediction.

VI. Geometry Modeling, Grid Generation and Automation

The importance of geometry modeling, grid generation, and automation cannot be overstated for the CFD-in-the-loop Monte Carlo simulation vision. This capability will require hundreds to thousands of end-to-end unsteady CFD simulations to perform a single analysis. During these simulations, the vehicle will be maneuvering arbitrarily and unpredictably as it responds to dispersed uncertainties in flight conditions and vehicle system operation. Vehicle orientation can and will change drastically during these simulations and the ability to reliably simulate the flowfield for any orientation will be paramount to the simulation. Presently, CFD methods can require significant user intervention to ensure geometry models and computational grids are constructed to capture the most relevant flow physics. In Monte Carlo simulations consisting of thousands of flow simulations, the user must be removed from the geometry model and grid development process as much as possible. Ultimately, this portion of the process must be automated to the point where the user provides a basic geometry outer mold line and the simulator and CFD method takes over the orientation, grid development, and subsequent grid optimization of the model to the flow situation under consideration.

These automated schemes must be capable of adapting to rigid body changes in orientation of the vehicle as it flies a given trajectory. They also must be able to adapt to moving and changing flow physics throughout the trajectory without compromising accuracy of the CFD analysis. Thus automated grid generation, moving mesh, and physics-adapted grid technologies will require significant maturation. The added complexity of stage separation and deployable systems will likely challenge developers for the next decade or more. Developers have begun to address these issues with a sketch-to-solution capability that has shown early promise for removing the user from the grid generation and adaptation process, even for relatively complex configurations [23]. The methodology has been shown to be capable of generating initial grids and adapting them to evolving flow phenomena without intervention of the user. Specifying and controlling CFD accuracy throughout this process continues to challenge developers, but this capability certainly represents a large first step in addressing the geometry modeling, grid generation, and automation capabilities required for CFD-in-the-loop Monte Carlo simulation.

VII. Computational Considerations

Today, Monte Carlo simulations utilizing database aerosciences models can be performed on workstation class computing clusters in a matter of hours. A typical space vehicle Monte Carlo analysis routinely requires 3000 or more independent trajectory simulations where up to a hundred design parameters are dispersed over an uncertainty space to fuel each trajectory calculation. While some RANS CFD simulations can be conducted on workstation class computers in a matter of hours, performing 1000's of these simulations for a Monte Carlo analysis will require considerably larger computing resources. As outlined above, flow phenomena on space vehicles will be complex in certain regimes even under nominal flight operations. Under these conditions, each trajectory will likely demand model sizes that require operation on supercomputers capable of parallel computing on 100's to 1000's of processors for dozens to days of wall-clock hours just for a single trajectory. If higher fidelity flow simulation involving DES or HRLES are warranted, grid and processor requirements could be an order of magnitude, or more, larger than for a RANS simulation.

The NASA Advanced Supercomputer (NAS) Pleiades presently has over 240,000 CPU cores. To put the scale of this problem in perspective, if the CFD simulation could be performed at the cost of 100 cores per trajectory, a 2400 trajectory Monte Carlo simulation would require 100% of Pleiades to be dedicated to the problem for the dozens of wall-clock hours required to perform each CFD trajectory computation. Most relevant RANS simulations for space vehicle applications require on the order of 1000 cores, ten times this example, so the present NASA computational capability doesn't meet minimal requirements for relevant space vehicle design problems.

CFD-in-the-loop Monte Carlo analysis could probably be demonstrated today if inviscid Cartesian grid Euler or coarse-grid RANS simulations were used as the aerodynamic engine for the trajectory simulation. With the right grid automation and adaptable flow physics, simulations using a combination of Euler and grid-resolved RANS may be possible on leadership class hardware like the Department of Energy's SUMMIT supercomputer [24]. Simulations based exclusively on grid-resolved RANS or higher fidelity CFD formulations will require envisioned computing advances over the next decade to become plausible. Continued research into the scaling of CFD methods on emerging supercomputer systems will be required to meet the vision of a CFD-in-the-loop Monte Carlo simulation. If the wall-clock time for a single CFD trajectory can be reduced or maintained at 8 hours or less and processor requirements can be constrained so that up to 3000 trajectory simulations can be performed simultaneously, then CFD-in-the-loop Monte Carlo analysis begins to become competitive with the current database capability. Thus continued pressure to improve the computational performance of our CFD methods and constrain the number of required processors for each trajectory computation must be maintained to realize this vision. Tuning the efficiency of a single trajectory calculation is the pacing item for this capability since once the cost of this computation can be controlled, the application of 1000's of these simulations becomes a relatively simple exercise in embarrassingly parallel computation, with each trajectory simulation able to proceed completely independently of all the others.

In the near-term, machine learning techniques [25] may be required to streamline the overall process and make the problem more tractable, especially if high fidelity CFD capability is required to address some components of the problem. Model Order Reduction (MOR), or Reduced Order Modeling (ROM), has been an active topic in fluid dynamics for well over a decade [26]. Development of these models as a natural course of performing limited CFD-in-the-loop flight simulation for dispersed uncertainties could significantly reduce the computational resources required to perform future simulations on a given configuration and reduce the cost of performing very large Monte Carlo analyses.

VIII. Multidisciplinary/Multiphysics Simulation Challenges

Numerous multidisciplinary/multiphysics challenges associated with space vehicle analysis have already been discussed in the forms of aero/plume interaction, chemically reacting flows, fluid/structure interaction, etc. For many of these multidisciplinary/multiphysics issues, it might be reasonable to build the multidisciplinary capability directly into the CFD solver, or develop a specialized solver to specifically address a given multiphysics problem. One challenge that has not been discussed is the integration of CFD methods with six degree-of-freedom (6DOF) simulation tools. There are many highly developed flight simulation tools available to the aerospace community, such as the Program to Optimize Simulated Trajectories 2 (POST2) [27], Optimal Trajectories by Implicit Simulation 4 (OTIS4) [28], the Trick® Simulation Development Toolkit [29], and other publicly available and proprietary software packages. It is unlikely that it would be optimal to build comprehensive 6DOF simulation and trajectory optimization capability directly into CFD codes. Given the complexity and capability of the available 6DOF simulators, it is more likely that coupling CFD tools and 6DOF simulators in a framework would be the more efficient and flexible solution to the problem. It was previously mentioned that most simulation tools presently run on desktop workstations or small computing clusters, while CFD analyses often must move to supercomputer platforms capable of parallel processing on 100's to 1000's of processors. Thus, moving the 6DOF simulation capability to the same computing platform as the CFD is an obvious way to approach the efficiently solution of the problem. This may or may not be a trivial exercise, both for the users of the 6DOF simulation tools or for the framework developers, as they address issues associated with coupling two methodologies that typically operate on drastically different computing platforms and timelines. This manuscript deals primarily with the CFD challenges to the problem, but the framework development challenges should not be neglected or deferred in efforts to demonstrate the overall grand challenge.

IX. Conclusion

A grand challenge to demonstrate CFD-in-the-loop Monte Carlo simulation for space vehicle analysis and design has been outlined. Two flight phases pertinent to atmospheric operation of space vehicles, ascent and EDL, were detailed, as were the maneuvers vehicles would perform in these regimes and the pertinent flow conditions they would encounter. Of the two possibilities presented, the EDL case may be a better choice for initially demonstrating this capability, mainly due to simpler vehicle geometries and some initial efforts already underway for this flight regime. Both phases of flight have attributes that may simply be too difficult for present or near-term capability to effectively address. For EDL, certain decelerators, like parachutes, are just now beginning to be analyzed using CFD, and expecting this capability to mature to a point where it could be automated and integrated into a CFD-in-the-loop simulation may be unreasonable in the next decade. Likewise, staging in the ascent problem could present similar hurdles. Therefore, it may be advisable to initially limit the scope of the Monte Carlo simulation to just a segment of the flight, like hypersonic entry for EDL or liftoff transition for ascent. A CFD-in-the-loop Monte Carlo simulation of even these more focused segments would significantly advance the state-of-the-art, provide a useful capability for designers, and serve as a basis for extension of the capability into more difficult problems, ultimately moving toward an end-to-end flight simulation capability based on integrated CFD aerosciences.

References

[1] Honeycutt, J., Cianciola, C., and Blevins, J., "NASA's Space Launch System: Progress Toward Launch," AIAA ASCEND 2020, November 16-18, 2020, Virtual Event, American Institute of Aeronautics and Astronautics, Reston, VA.

[2] Norris, S. D., et al., "Orion: Lessons Learned from EFT-1and, EM-1, AA-2, and EM-2 status," AIAA 2016-5416, AIAA SPACE Forum, Long Beach, CA, 13-16 September, 2061, American Institute of Aeronautics and Astronautics, Reston, VA

[3] Hanson, J. M. and Beard, B. B., "Applying Monte Carlo Simulation to Launch Vehicle Design and Requirements Analysis," NASA/TP-2010-216447, September, 2010.

[4] Bibb, K. L., Walker, E. L., Brauckmann, G. J., and Robinson, P. E., "Development of the Orion Crew Module Static Aerodynamic Database, Part I: Hypersonic," 29th AIAA Applied Aerodynamics Conference, Honolulu, HI, 27-30 June 2011, American Institute of Aeronautics and Astronautics, Reston, VA.

[5] Bibb, K. L., Walker, E. L., Brauckmann, G. J., and Robinson, P. E., "Development of the Orion Crew Module Static Aerodynamic Database, Part II: Supersonic/Subsonic," 29th AIAA Applied Aerodynamics Conference, Honolulu, HI, 27-30 June 2011, American Institute of Aeronautics and Astronautics, Reston, VA.

[6] Tartabini, P. V., Gilbert, M. G., and Starr, B. R., "A Proposed Ascent Abort Flight Test for the Max Launch Abort System,' AIAA 2016-0783, AIAA Atmospheric Flight Mechanics Conference, San Diego, CA, 4-8 January, 2016, San Diego, CA, American Institute of Aeronautics and Astronautics, Reston, VA.

[7] Morton, S. A., and Meakin, R. L., "HPCMP CREATE™-AV Kestrel Architecture, Capabilities, and Long Term Plan for Fixed-Wing Aircraft Simulations," AIAA 2016-0565, 54th Aerospace Sciences Meeting, 4-8 January, 2016, San Diego, CA, American Institute of Aeronautics and Astronautics, Reston, VA

[8] Stern, E. C., Gidzak, V. M., and Candler, G. V., "Estimation of Dynamic Stability Coefficients for Aerodynamic Decelerators Using CFD," AIAA 2012-3225, 30th AIAA Applied Aerodynamics Conference, 25-28 June, 2012, New Orleans, LA, American Institute of Aeronautics and Astronautics, Reston, VA.

[9] Brock, J. M., Stern, E. C., and Wilder, M. C., "CFD Simulation of the Supersonic Inflatable Aerodynamic Decelerator (SIAD) Ballistic Range Tests," AIAA-2017-1437, 55th AIAA Aerospace Sciences Meeting, 9-13 January, 2017, Grapevine, TX, American Institute of Aeronautics and Astronautics, Reston, VA.

[10] Ernst, Z. J., et al., "Coupling Computational Fluid Dynamics with 6DOF Rigid Body Dynamics for Unsteady Accelerated Flow Simulations," AIAA-2018-0291, 2018 AIAA Atmospheric Flight Mechanics Conference, 8-12 January, 2018, Kissimmee, FL, American Institute of Aeronautics and Astronautics, Reston, VA.

[11] Gilbert, M. G., "The MAX Launch Abort System – Concept, Flight Test, and Evolution," 7th International Association for the Advancement of Space Safety Conference, 20-22 October, 2014, Friedrichshafen, Germany.

[12] Steltzner, A. D., et al., "Mars Science Laboratory Entry, Descent, and Landing System Development Challenges," *Journal of Spacecraft and Rockets*, Vol. 51, No. 4, July-August 2014, pp. 994-1003.

[13] Bird, G. A., *The DSMC Method*, Version 1.2, G. A. Bird, Sydney, Australia, 2013.

[14] Slotnick, J., et al., "CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences," NASA/CR-2014-218178, March, 2014.

[15] Alter, S. J., "Time-Accurate Unsteady Pressure Loads Simulated for the Space Launch System at Wind Tunnel Conditions," AIAA 2015-3149, 33rd AIAA Applied Aerodynamics Conference, 22-26 June, 2015, Dallas, TX, American Institute of Aeronautics and Astronautics, Reston, VA.

[16] Morris, C. I., "Space Launch System Ascent Aerothermal Environments Methodology," AIAA 2015-0561, 53rd AIAA Aerospace Sciences Meeting, 5-9 January, 2015, Kissimmee, FL, American Institute of Aeronautics and Astronautics, Reston, VA.

[17] Schoenenberger, M., et al., "Characterization of Aerodynamic Interactions with the Mars Science Laboratory Reaction Control System Using Computation and Experiment," AIAA 2013-0971, 51st AIAA Aerospace Sciences Meeting, 7-10 January, Grapevine, TX, American Institute of Aeronautics and Astronautics, Reston, VA.
[18] Vicker, D. J., et al., "Effects of the Orion Launch Abort Vehicle Plumes on Aerodynamics and Controllability," AIAA 2013-0970, 51st AIAA Aerospace Sciences Meeting, 7-10 January, 2013, Grapevine, TX, American Institute of Aeronautics and Astronautics, Reston, VA.
[19] Hughes, S. J., et al., "Hypersonic Inflatable Aerodynamic Decelerator (HIAD)Technology Development Overview," AIAA 2011-2524, 21st AIAA Aerodynamic Decelerator Systems Technology Conference and Seminar, 23-26May, 2011, Dublin, Ireland, American Institute of Aeronautics and Astronautics, Reston, VA.
[20] Venkatapathy, E., et al., "Adaptive Deployable Entry and Placement Technology (ADEPT): A Feasibility Study for Human Missions to Mars," AIAA 2011-2608, 21st AIAA Aerodynamic Decelerator Technology Conference and Seminar, 23-26May, 2011, Dublin, Ireland, American Institute of Aeronautics and Astronautics, Reston, VA.
[21] Korzun, A. M., Braun, R. D. ,and Cruz, J. R., "Survey of Supersonic Retropropulsion Technology for Mars Entry, Descent, and Landing, Journal of Spacecraft and Rockets, Vol. 46, No. 5, September-October 2009, pp. 929-937.
[22] Korzun, A. M., et al., "Computational Investigation of Retropropulsion Operating Environments with a GPU-Enabled Detached Eddy Simulation Approach," AIAA 2020-4228, ASCEND 2020, November 16-18, 2020, Virtual Event, American Institute of Aeronautics and Astronautics, Reston, VA.
[23] Kleb, B., et al., "Sketch-to-Solution: An Exploration of Viscous CFD with Automatic Grids," AIAA 2019-2948, AIAA AVIATION 2019 Forum, 17-21 June, 2019, Dallas TX, American Institute of Aeronautics and Astronautics, Reston, VA.
[24] https://www.olcf.ornl.gov/summit/
[25] Alpaydin, Ethem, *Introduction to Machine Learning*, Fourth Edition, The MIT Press, 2020.
[26] Lasilla, T., et al., "Model Order Reduction in Fluid Dynamics: Challenges and Perspectives," *Reduced Order Methods for Modeling and Computational Reduction*, Springer Nature, Switzerland AG, 2020, pp. 235-273.
[27] Lugo, R. A., et al., "Launch Vehicle Ascent Trajectory Simulation Using the Program to Optimize Simulated Trajectories II (POST2)," AAS 17-24, 27th AAS/AIAA Space Flight Mechanics Meeting, 5-9 February, 2017, San Antonio, TX., American Astronautical Society, San Diego, CA.
[28] Hargraves, C.R., and Paris, S.W., "Direct Trajectory Optimization Using Nonlinear Programming and Collocation, Journal of Guidance, Control, and Dynamics, Vol. 10, No. 4, July–August 1987.
[29] Paddock, E. J., et al., "TRICK®: A Simulation Development Toolkit," AIAA 2003-5809, AIAA Modeling and Simulation Technologies Conference and Exhibit, 11-14 August, 2003, Austin, TX, American Institute of Aeronautics and Astronautics, Reston, VA.