International Workshop on "Future of CFD and Aerospace Sciences," April 23-25, 2012, Kobe, Japan

Current CFD Practices in Launch Vehicle Applications

by

Dochan Kwak and Cetin Kiris
NASA Ames Research Center

Summary

The quest for sustained space exploration will require the development of advanced launch vehicles, and efficient and reliable operating systems. Development of launch vehicles via test-fail-fix approach is very expensive and time consuming. For decision making, modeling and simulation (M&S) has played increasingly important roles in many aspects of launch vehicle development. It is therefore essential to develop and maintain most advanced M&S capability. More specifically computational fluid dynamics (CFD) has been providing critical data for developing launch vehicles complementing expensive testing. During the past three decades CFD capability has increased remarkably along with advances in computer hardware and computing technology. However, most of the fundamental CFD capability in launch vehicle applications is derived from the past advances. Specific gaps in the solution procedures are being filled primarily through "piggy backed" efforts on various projects while solving today's problems. Therefore, some of the advanced capabilities are not readily available for various new analysis tasks, and mission-support problems are often analyzed using ad hoc approaches. The current report is intended to present our view on state-of-the-art (SOA) in CFD and its shortcomings in support of space transport vehicle development. Best practices in solving current issues will be discussed using examples from ascending launch vehicle. Some of the pacing issues will be discussed in conjunction with these examples.

1. Background`

Historically, together with advances in computing hardware, CFD has been very successfully used in airplane aeronautics [1]. In space exploration area, however, applications of CFD have been behind aeronautics partially because space-related flow problems involve largely time-dependent flow phenomena and complex flow physics requiring advanced modeling [2,3]. Moreover, experimental and/or flight data for validation are limited and new data are difficult to obtain. Advanced CFD technology suitable for space transportation vehicle development requires maturation through realistic applications. When a new vehicle concept like the Space Shuttle launch vehicle was on the drawing board, CFD was not mature enough to make significant impacts. Therefore, until recently, most operational vehicles were designed heavily relying on empiricism. The CFD technology suitable for space applications like Space Shuttle has been developed in parallel with operational period. Subsequently, CFD became useful to investigate accidents, to support operational aspects, and to retrofit for improved components. Since post-Shuttle exploration requires new or replacement vehicles, conceptual design evaluation can rely on database generated by CFD, which is not thoroughly validated for the type of flow encountered in new vehicle concepts. Therefore, so-called "best practices" protocols have been developed in conjunction with recent tasks. Examples of these are given with requisite capabilities for extended applications in the future.

Regarding the development of requisite capabilities, steady investment has been made in the past, starting from some 40 years ago, to develop CFD tools of varying fidelity usable in aerospace problem solving. As a result, CFD technology has become an indispensable part of the design and operation of aerospace vehicles. There are several fundamental elements of CFD tools that include algorithm, geometry definition and grid generation, boundary condition procedure, physical modeling, pre- and post-processing tools. In practical problem solving environment, computer architecture, data management tools and networking play very important role in producing results in a timely manner. Following the early successes in developing solution algorithms and flow solvers, primary acceleration of solution time has been accomplished more through computer hardware speed up than advanced acceleration algorithms. Thus the parallel computing methods and associated data management schemes have played important role in utilizing compute resources. As the problem sizes grew bigger, development of advanced methods as well as compute hardware advances remain to be of major importance.

However, strategic investment on CFD has been constantly reduced to the point that existing tools are solely applied to solve today's problems. It is to be noted that successful application of these tools requires the synergy of computing facility, software, simulation tools, data analysis tools, and networks, coupled with the combined knowledge of engineering, flow physics, and computer science. Contrary to the common impression that CFD or more generally modeling and simulation technology is mature enough to the point it is usable by non-experts, available tools still lack prediction capability in many critical areas and requires experts in order to conduct successful simulation of surprisingly many problems, especially in space transportation vehicle applications. Thus it is necessary to advance state-of-the-art as well as produce CFD experts who posses critical skills both in numerical methods and physics.

In the 1980s and 1990s, increasing the fidelity of formulation and inclusion of more complete geometry in simulation were the focus largely in single discipline. The high-speed scientific computing environment has grown to the point that vehicles and components are to some degree amenable to computer simulation. Despite these advances, unsolved problems still exist in several areas critical to exploration mission successes such as high-fidelity simulation of unsteady flow for prediction of vibration loads on launch vehicles and prediction of massively separated flow are among the remaining challenges. CFD-related issues can vary depending on primary flow features of interest. Resolving all scales/features of complex problems is not necessarily needed in all flow analysis or in all tasks. In this report we will focus on issues in supporting space exploration, especially, related to launch vehicle performance and operations. Some features are cross-cutting in nature for both exploration and aeronautics. However, in tis report, hypersonics-related CFD simulation is not discussed. Our discussion starts with a look at what CFD can or cannot do now, and then what advances are needed most to support future tasks. These are based on primarily our experience at NASA for over the past three decades largely related to the Shuttle program and Constellation program, both terminated recently and replaced by Space Launch System development for deep space exploration.

2. Current CFD Capabilities

The state-of-the-art (SOA) in CFD can be viewed from various different angles. One possibility is to measure the SOA by the flow simulation capability required for space transportation vehicle development and operation. For the current paper, we are focusing on primarily flow analysis related to launch vehicle in ascent. This assessment can then be used to determine the areas that need intense future investment such that one can produce credible and predictive results to support comparative analysis of a wide range of potential configurations. The risk involved in space exploration is very high, especially, when loss of human and/or mission is involved. Since testing is very limited and expensive, CFD can play a critical role in making decisions. In general, CFD still lacks prediction capability. Then the primary question is how one can use CFD in credible ways for decision making. This involves simulation of complex flow phenomena with some acceptable level of confidence for a completely "NEW" configuration under on- or off-design conditions. This question will be partially answered in conjunction with current practice examples later. Current capabilities of CFD in resolving fundamental flow problems will be briefly reviewed first.

Key areas describing fundamental CFD capabilities, especially critical in launch vehicle applications are listed below without proof. These represent our opinion and partially supported by the examples illustrated in the following section.

The State-of-the-Art in Fundamental CFD

- Can produce reliable solutions for
  - Steady flow: Attached flows about complex geometries
  - Unsteady flow: Primarily attached flows about semi-complex geometries
- Cannot produce reliable and accurate solutions for
  - Massively separated flow
  - Unsteady/transient flow about complex geometries and complex flow physics
  - Noise
  - Vortex-dominated flows
- Dependence on grid quality continues to be an important factor to get consistent, accurate CFD predictions
- Post-processing of massive datasets for integrated quantities is routinely done, but feature extraction, especially for unsteady flow, still remains to be challenging

Physical modeling capability

Physical modeling is of major importance in obtaining accurate and reliable solutions, and several specific issues will be discussed later in conjunction with examples.

- Turbulence model
  General comments on the engineering model of turbulence will be presented in the next section. A short summary is listed below first:

- Engineering-level turbulence models exist, but the modeling approach has not been improved much since early days of CFD applications.
- Engineering-level turbulence models used in conjunction with complex problem have been developed primarily by those performing CFD applications tasks
- Separated flow needs ad hoc tuning or can be more accurately computed by utilizing LES features via hybrid RANS-LES approach
- Internal flow applications are not very well evaluated – may need different modeling approach
- DES /hybrid models are gaining popularity, however, accuracy and grid requirements issues in the wall region are the limitation
- To be economically viable, 1- or 2-equation models are used most frequently in conjunction with space flight vehicles
- Current models are not capable of predicting (or interpolating) massively separated and unsteady flow situations

* Transition model is practically non-existing for engineering
* Multi-phase flow computing is even farther behind in producing results for engineering analysis

Computer speed increase and its impact on simulation

* Resolution and turn around time improved a lot, but there are problems that require advanced algorithm and enhanced methods such as boundary condition procedures
* Future computer may result in several orders of magnitude speed-up possibly with substantially different architecture requiring new ways of implementing parallel computing

Examples of successful CFD applications in aerospace engineering:

* Retrofitting
* Accident Investigation
* Analysis and Design
  -Concept evaluation
  -Preliminary design
  -Critical design review
* Mission planning and risk assessment
* Operation support

The need for supporting high-priority current projects has led to the development of suit of CFD procedures for solving practical problems. These problems involve complex configurations and operating conditions not typically addressed by academia, and usually require in-depth understanding of the engineering problems to be resolved. The pervasive use of experiments and flight tests for validation increase the credibility of solutions, however, this does not guarantee the same level of solution quality when extended to other configurations. Very often ***best practice guidelines*** are used for generating database for a wide range of operating conditions for one-type of problems. Some specific issues as well as current capabilities will be discussed using examples.

3. On the Engineering Model of Turbulence

Turbulence is of major realistic concern for CFD applications to engineering problems. In the current practices of CFD to support aerospace engineering, there are a few models compatible with differential equation solvers. Current models mostly tuned to boundary layer flow have been successfully used in many vehicle calculations where flow is largely attached and steady-state solutions are main quantities of interest. In this section, some thoughts on the performance of current turbulence modeling practices and issues are given. These are authors opinion accumulated while working on application problems associated with aerospace vehicles and are presented without substantiating our comments by computed examples.

3.1 General Comments

Criteria for engineering model of turbulence:
To make timely impacts on engineering tasks from CFD approaches, it is important to have computing efficiency as well as to generate results consistent with turbulent flow characteristics. Users of codes may not have in-depth knowledge of turbulence modeling, therefore individual tuning of the model to the problem on hand should not be expected.

Several issues need to be considered on turbulence models required for mission support:

- Sensitivity: impact of the model on the overall computed results need to be assessed relative to effects of numerical accuracy such as algorithm and grid dependency
- Consistency and robustness: it should be usable by non-experts
- Range of applicability: most engineering-level models are tuned for limited cases, and therefore the applicable range needs to be defined

Engineering-level approach:
For attached boundary layer flow, turbulence scale is small and usual RANS-based model works well. Major bottleneck for developing advanced flow simulation tools is related to turbulence/transition model. Especially for massively separated flows and unsteady shear layer interaction problems such as jet-plume interaction problem, adequacy of particular model in use need to be examined. When RANS models, such as Spalart-Allmaras (S-A) [4], Baldwin-Barth (B-B) [5], SST model [6], have difficulties in predicting these flows, LES or LES-RANS hybrid model might offer an avenue to handle these flows, however, they are much more expensive than RANS approach in addition to their own limitations in range of applicability, especially for wall-bounded flow.

3.2 Current Modeling Practices

Fluid dynamic problems in and around space exploration vehicles are very complicated requiring high spatial and temporal resolution. The flow features involve flow phenomena different from aerodynamic vehicles, such as commercial airplanes with mostly attached flow.

Simulation of a launch vehicle at ascent typically requires CFD capability for

- Separated flow, typically along the long missile-type body
- Shear layer interaction during stage separation

- Proximity aero involving launch-abort simulation
- Plume induced flow separation
- Plume with multiple nozzles (engines)
- Jet impingement into cavity (flame trench) during launch
- Transient flow at launch

Simulation of rocket propulsion system involves
- Complex internal flow
- Turbopump flow with high-speed rotation effects with tip vortex, leakage and cavitation
- Combustion instability for chemical propulsion system

Simulation of these flows requires turbulence modeling of separated flow, unsteady flow, highly vortical flow and flow with curvature and rotational effects. Current turbulence models in production CFD codes need to be evaluated how models based on equilibrium turbulence can be utilized for mostly non-equilibrium turbulent phenomena encountered in the exploration vehicles.

Some specific comments follow on the two common approaches available in current practices.

3.2.1 RANS models

Reynolds-averaged Navier-Stokes (RANS) equations are derived from Reynolds decomposition (1894) assuming that the flow field can be decomposed into ensemble averaged quantity and deviation from that due to turbulent fluctuation. Both steady and unsteady (URANS) equations can be derived in this fashion. The Reynolds stresses created from this process need to be modeled to close the governing set of equations.

Commonly used method of representing the Reynolds stresses by average flow variables is via eddy viscosity model following Boussinesq hypothesis. Algebraic and one-equation models such as S-A and B-B are popular examples. Another approach is to model the Reynolds stresses directly (Reynolds stress model-RSM or second moment closure) that requires to solve additional 6 equations and introduces additional unknown coefficients. A wide variety of RANS modeling has been tried in the hopes of increasing generality.

Basic ideas for turbulence closures in conjunction with RANS equations were available in the late 1960s and 1970s (for example, see 1968 Stanford Conference on turbulent boundary layer prediction method calibration conference TBLPC-1968 [7], and the follow on conference in 1981 [8]). As the computer hardware speed increases, models following these ideas have been developed with finer mesh and with more flexibility for tuning parameters. Nevertheless, none of these is universal in prediction and modeling ideas remain basically at the same level presented during this period. However, these models in use encompass good features from previous experiences and offer workable tools in practice, especially when tuned to the type of flows being analyzed.

Despite all the resources invested in turbulence research, useful models for aerospace engineering have been developed by a few CFD researchers engaged in mission computing. Models such as B-B, S-A, SST and k-e models are such examples and have been successfully applied to many engineering calculations. These models were developed to alleviate the difficulty of specifying turbulence length scale explicitly required for algebraic models. These one- or two-equation models use turbulence transport equation to define turbulent eddy viscosity (or via turbulent kinetic energy) basically balancing turbulence production, dissipation and diffusion terms. However, in these models, empiricism played a critical role. Since the empirical data for validation are still limited and also due to the heuristic nature of the model equations, "prediction capability" for many complex turbulent flow phenomena is still limited.

In general, flows involving attached boundary layer such as aerodynamics of airplane in cruise conditions can be reasonably predicted using one- or two-equation models. For massively separated flows, turbulence modeling is still a challenge. It is essential to understand the applicability and limitations of the models in the flow simulation process in solving particular class of problem being solved.

3.2.2 Hybrid RANS-LES models

LES offers a possibility to be more predictive than RANS model since only small scale eddies need to be modeled. However, the computing requirement is still very large even with current computing hardware. Especially for wall-bounded flow, the definition of large eddies becomes not straightforward. To alleviate this, practical ideas are implemented such as DES and hybrid model. However, this approach will bring in limitations of RANS model in the wall region. The question is whether this approach is a temporary solution or whether it is worth developing further to a wider range of applications.

In space exploration cases, the flow encounters mixture of many different types requiring careful evaluation of impacts of turbulence modeling. In light of the computing economy, one- or two-equation model are still in use for production calculations. Current practices will determine whether the application of a RANS-LES hybrid model will be beneficial even with added computing cost. The decision will be problem dependent at this point.

3.2.3 Numerical scheme and resolution requirements

The turbulent flow analysis involving space transportation vehicle often requires resolution of turbulent eddies normally not captured by RANS model. LES approach offers a possibility for obtaining more physical and consistent solutions. However, LES requires special attention to the numerical methods used such as kinetic energy conserving property. Mesh requirement varies depending on the flow type, such as boundary layer type or detached/free shear flow type, and the mesh-related dissipation is often ignored or lumped into the model. Defining domain required for high resolution is also difficult since the local flow is not known a priori. In addition complex geometry may generate mixture of different types of turbulent flow regimes.

In a more generic sense, grid-based computation requires to resolve both small dissipative scale and large energy containing eddies. A quick order of magnitude estimate of mesh requirements can be as follows:

Dissipation length scale is on the order of Kolmogorov microscale, $\eta = \left(\nu^3/\varepsilon\right)^{1/4}$.
Here ν is the kinematic viscosity and ε is the energy dissipation. The energy dissipation can be expressed as $\varepsilon \approx q^3/L$, where q is a velocity measure and L is the representative length scale of large eddies. Then the mesh points required for three-dimensional computations for resolving both small and large scale motion can be estimated to be

$$N \propto \left(\frac{L}{\eta}\right)^3 \approx \left(\frac{qL}{\nu}\right)^{9/4} = R_T^{\ 9/4}$$

This estimate is based on the assumption that mesh required for resolving small-scale turbulence such as encountered in thin boundary layer will be populated throughout the entire computational domain. A similar estimate was derived for boundary layer type flow by Reynolds [9]:

$$N_{xyz} \propto R^{2.2}$$

This is obviously too restrictive in overall mesh requirement. Chapman [10] derived somewhat less restrictive requirements considering three different directional requirements for resolving boundary layer flows, resulting in

$$N_{xyz} \propto R^{1.8}$$

It is not our intention to thoroughly review mesh requirements in this article. Extensive analysis and in depth review can be found in the literature (for example by Chapman [10]).

It is thus natural to devise a method to capture some of the advantages of LES and yet to keep computational requirement within manageable level for engineering solutions. Even though RANS-LES hybrid models have been developed, better wall-layer models are yet to be devised, for example, to remove so-called “grey area” between RANS and LES. More manageable mesh requirements may result in better wall-layer models.

3.2.4 Near-term need for turbulence models

To accommodate phenomena particularly relevant to exploration vehicles such as separated flow, strong curvature effect (physical space limitation may require this), shear-layer interaction (plume-separated BL, multiple jets and wakes), it will be useful to assess current production versions of flow solvers and turbulence models. Sensitivity to grids, time-step sizes need to be assessed comparing the best solutions to results from other codes and experimental data. It is very critical to get consistent results and to define the range of applicability. Eventually specific modeling requirements can be defined to support vehicle development and operation for exploration mission.

4. Current Procedures and Issues

Excerpts from several simulations associated with an ascending launch vehicle are presented next to illustrate current practices of CFD. The CFD tasks shown in Figure 1 represent major areas where computational support is needed. Current challenges facing CFD applications are discussed using these examples with emphasis on following key issues:

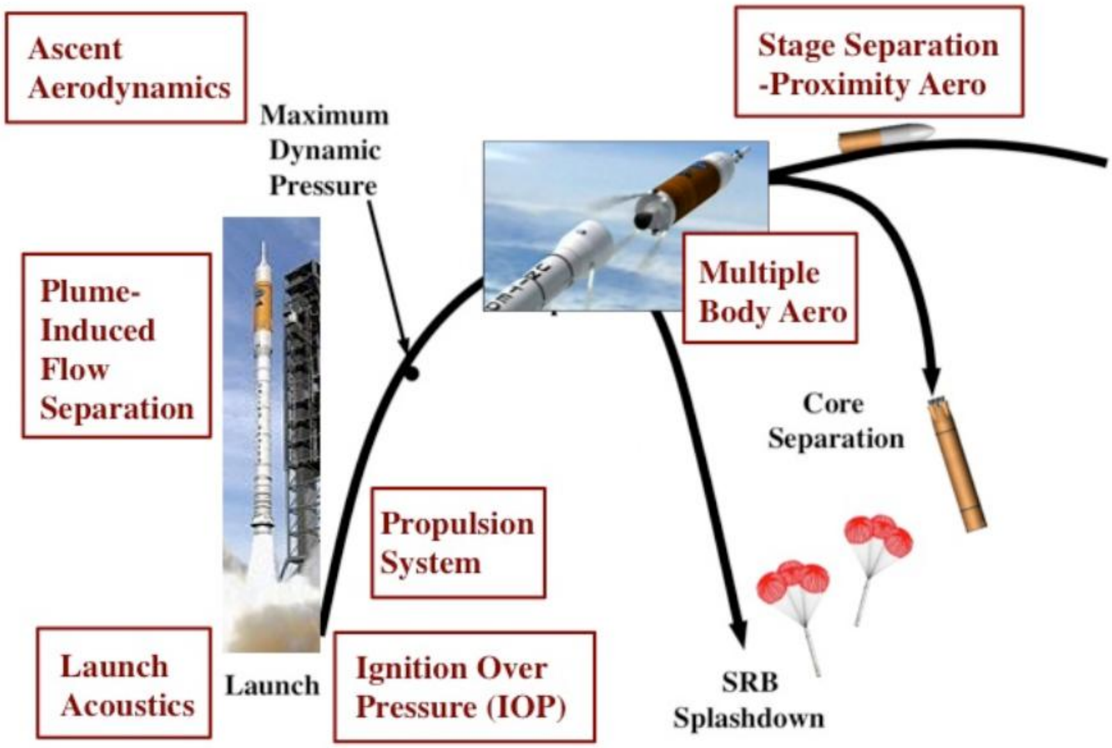


Figure 1. Schematic of CFD tasks associated with an ascending vehicle.

<u>*Ignition Overpressure and launch acoustics:*</u>
Prediction of transient pressure wave generated during ignition requires time accuracy. Computationally, resolving this severe flow during an extremely short time period requires to establish CFD procedures involving time integration schemes, sensitivity to time-step sizes and grid resolution requirements [2].

<u>*Aerodynamics of ascending vehicle:*</u>
Aerodynamic forces and moments data along the trajectory are in need to support vehicle development and operation. Uncertainties associated with the CFD-generated aero database need to be quantified. This requires detailed study on grid convergence, code-to-code variations, and the effects of turbulence models [3].

Turbulence modeling for separated flow induced by plume has non-negligible effects on the size of separated region. Turbulence modeling issues associated with boundary layer and plume interaction, nozzle exit boundary conditions and plume simulation need to be examined [11].

In addition, stage separation and proximity aerodynamics pose challenges in simulating separated flow and shear layer interactions. Engineering-level turbulence modeling for this flow is especially difficult. These and other issues are not discussed in this article and can be found in the reference cited [12].

Internal flow in rocket propulsion system:
Internal flow encountered in liquid-propellant rocket engines poses modeling challenges different from external aerodynamics. CFD applications can be classified into three major categories, namely, complex internal flow, flow through turbupumps (turbine and pump), and flow in combustion devices. Simulations of these flows have been performed individually in the past. Since early 1980s, CFD simulations for propulsion systems have been performed for designing, retrofitting and analysis of propulsion system, which eventually led to the formation of Propulsion CFD Consortium in the 1990s at the NASA/Marshall Space Flight Center (MSFC) [13, 14]. The goal was to develop advanced CFD capability for applications in propulsion technology. Through this effort, even though it did not continue to be fully mature useful approaches and issues relevant to propulsion CFD were discussed among experts in the field. Selected issues related to flow involving complex internal flow geometry and vibrational issue related to high-speed turbopump are discussed in this paper to illustrate real-world CFD practices in rocket propulsion area [15].

These issues will be discussed next using computed examples.

4.1 Time-dependent computations of Ignition Overpressure (IOP) wave propagation

Computation of time-dependent flow poses special challenges since there is no well-established guideline on time-step sizes and convergence criteria for obtaining time-accurate solutions. Computing time requirement is at least an order of magnitude higher than solving steady-state cases. In addition, since existing turbulence models for Reynolds Averaged Navier-Stokes (RANS) equations are mostly tuned using steady benchmark problems, applying these models to unsteady flow computations adds uncertainties to the solution procedures. Models such as LES representing non-equilibrium nature of the turbulence as encountered in separated and unsteady flows require huge computing resources except for limited number of cases. In this section, some computational issues will be illustrated using computed results on the Space Shuttle configuration by Kiris et al. [2].

4.1.1 Problem description

During the initial buildup of thrust when a rocket engine is ignited, large magnitude ignition overpressure (IOP) waves are generated. The IOP waves from the nozzle exit create piston-like action beneath the mobile launch pad (MLP). The strong compression waves, along with their reflections in the flame trench, travel back to the launch vehicle and through the exhaust holes of the rocket engine potentially having damaging effects on the vehicle and the surrounding structures (see Figure 2 for an illustration of launch pad arrangement.)

Accurately quantifying the magnitudes and frequencies of these waves will be required to assess potential impacts of the IOP waves to the flame trench, surrounding structures and the launch vehicle. Two major issues of the computational predictions are how to determine physical time step sizes for time accuracy while maintaining reasonable computing efficiency, and convergence criteria to determine sub-iteration convergence when a dual time stepping procedure is employed. In lieu of rigorous theoretical basis to determine these, Kiris et al. [2] established computational procedures to support launch operations for Ares IX experimental flight and to generate database for repairing flame trench wall damage incurred by IOP waves during STS-124 (Space Transportation System-124) launch.

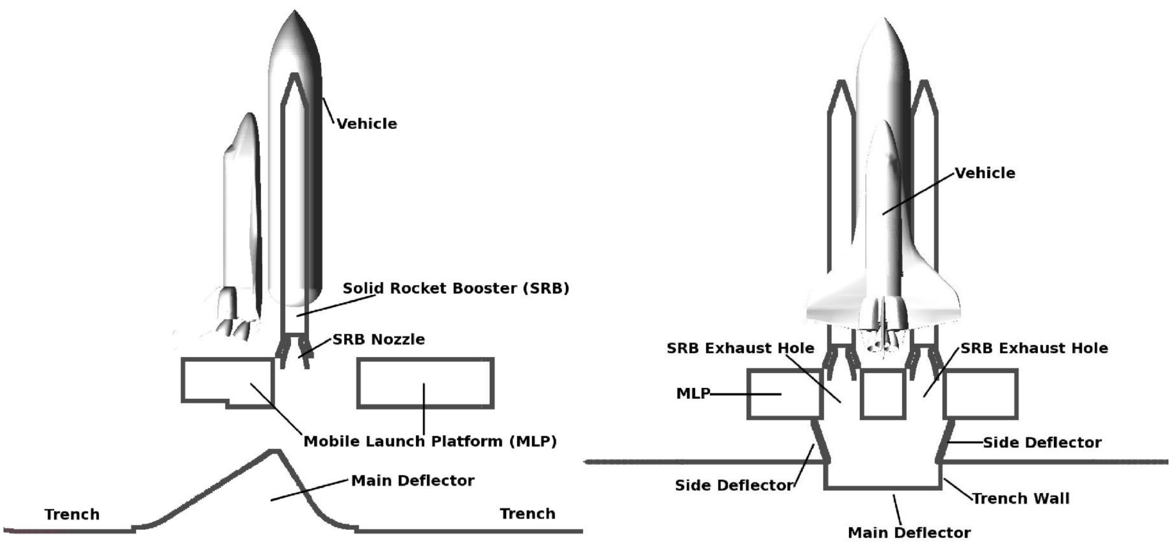


Figure 2. Longitudinal (left) and perpendicular (right) view of the launch vehicle and flame trench.

4.1.2 CFD procedure

The RANS equations are discretized with second order backward differencing in time, conservative upwind biased differencing of inviscid fluxes using roe scheme, and second order central differencing for the viscous terms.

First, one-dimensional convection equation is used to develop convergence criteria for dual time stepping procedure using varying grid and time resolution levels. Minimum error level was then established not to produce spurious solutions.

Next the unsteady procedure was developed to establish guidelines for time-step and sub-iteration parameters using 2-D cut of the flame trench. This enables an extensive study on physical and pseudo-time step sizes relative to sub-iteration numbers within reasonable computing resources.

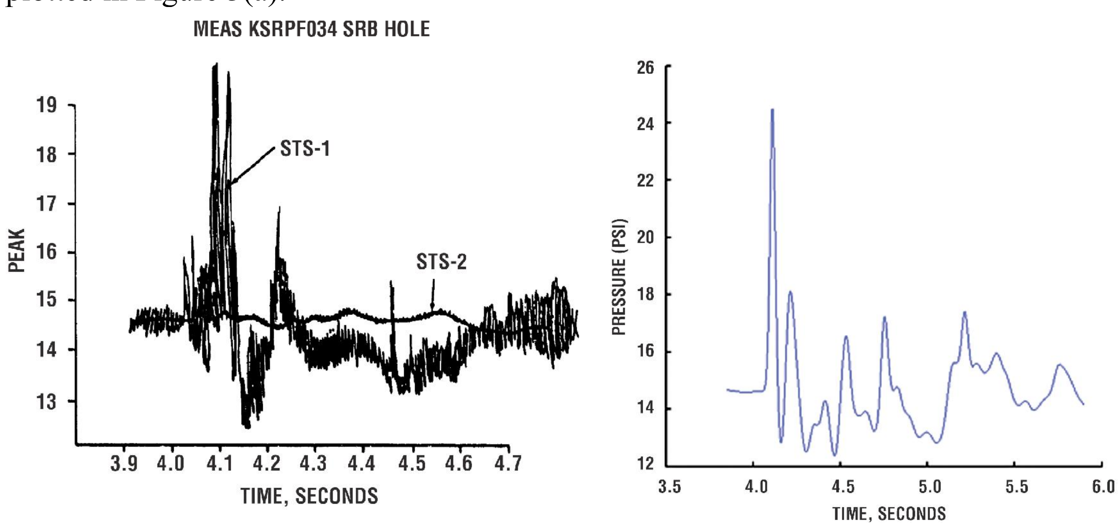
To isolate the sensitivity of the unsteady solutions to the time-step and sub-iteration parameters, grid system consisting of 25 overset grids with a total of 1.1 million grid points is fixed. Then various combinations have been experimented to investigate the sensitivity. The results of this study provide sensitivity for a wide range of parameter combinations. Through this process a range of the time-step size and sub-iteration numbers are recommended for 3-D analysis.

This best-practices guideline for 3-D applications is based on the expectation that a similar process can be justified when an equivalent grid system is used under the same unsteady plenum conditions.

4.1.3 Applications to 3-D IOP propagation

Once computational procedure is established, 3-D IOP computations were performed following the guidelines established using 2-D analysis. During the Shuttle program some IOP data were measured and thus these are used as a partial validation. However, only limited data are available for validation for 3-D real configurations.

Comparison with flight data:
Since the major IOP waves are originated from the two solid rocket boosters (SRB) in the shuttle configuration, the computational model includes the external tank, two SRBs, mobile launch pad (MLP) and the flame trench. Fluctuating pressure at a point on the launch vehicle is recorded (Ryan et al., 1981 [16]) during the first flight of the Shuttle, STS-1. This recorded STS-1 flight data are plotted in Figure 3(a) and the computed results are plotted in Figure 3(b). Considering that the complexity involved in taking the real-time flight data and in modeling this real geometry and computing with complex grid system and boundary conditions, the two compare fairly well. The peak IOP and the following pressure fluctuations can be regarded as quantitatively comparable. Since the STS-1 exhibits high IOP, water suppression system is implemented for the subsequent flights. The IOP for STS-2 with water suppression system is plotted in Figure 3(a).



(a) (b)

Figure 3. Comparison of IOP between STS-1 flight data (a), and computed results (b).

Design data generation:
During STS-124 launched in May 2008, a large section of trench wall was damaged. To assist repair of the flame trench wall, CFD was applied to generate unsteady wall pressure during launch. To further validate the computational procedure for this application, STS-4 case was simulated and compared with the flight data. Pressure tap locations in the flame trench are shown in Figure 4.

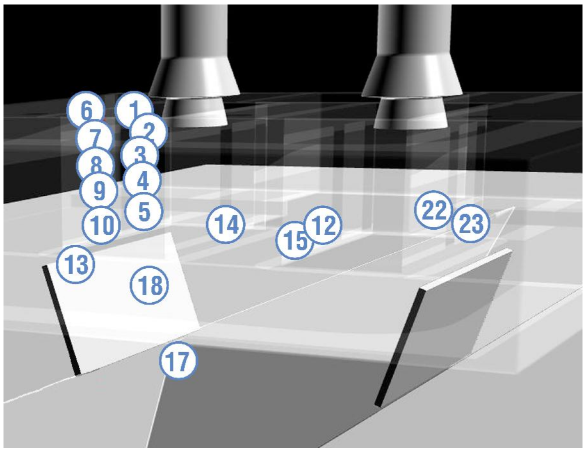


Figure 4. Pressure tab locations in the flame trench for STS-4.

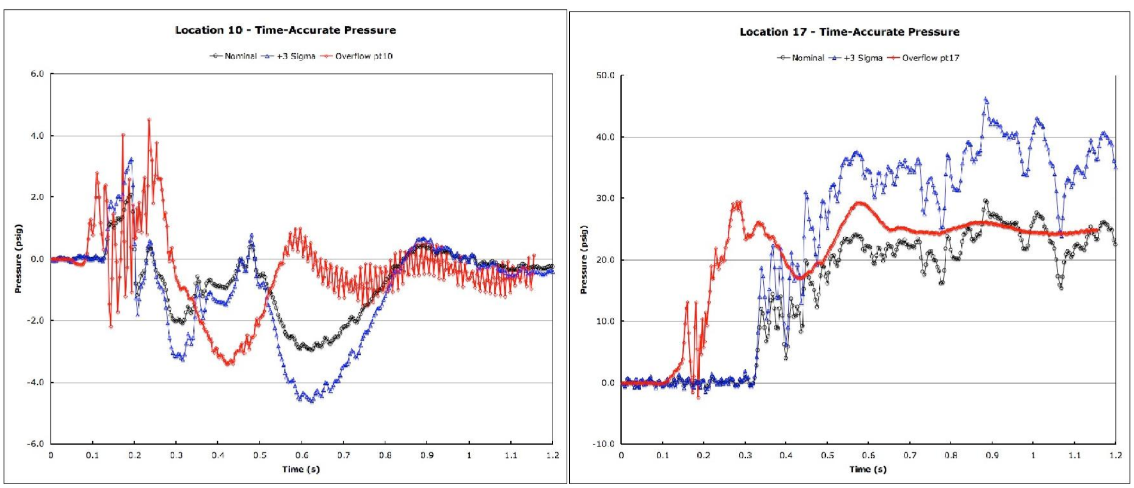


Figure 5. Comparison of time history of pressure between STS-4 data and computed results at pressure tap location 10 (left) and 17 (right).

Results at two pressure tap locations, location 10 and 17, are shown in Figure 5. In the figure, flight data (Nominal) and an error bound (+3 Sigma in blue color) and computed result using OVERFLOW code (red color) are compared. Both pressure taps 10 and 17 are below water suppression system. Water suppression system is to both suppress and delay IOP propagation. In the figure, computed results generated without a model of water suppression system show phase shifts and differences in the magnitudes of the peak pressure. Since the computations do not include two-phase flow model, this is expected. Even with these differences, the computed results provide useful information on the pressure distribution and fluctuations for the entire trench wall. The computed results are then used in repairing trench wall damage due to STS-124.

<u>*On the current practices:*</u>
Although there is no rigorous guidelines for unsteady computations of this nature, current practices as discussed in this section follows logical steps to produce physically valid results for the engineering task in support of space launch operation mission. For more accurate quantitative results and for more general applications to flame trench configuration with different types of water suppression system, multi-phase capability will be necessary.

4.2 Aerodynamics of ascending vehicle

To develop launch vehicles in support of future space exploration missions, NASA attempted to design Ares I and Ares V. To enable aerodynamic database generation in support of the launch vehicle design, three CFD solvers in use at NASA were applied. Since empirical data for validation are not fully available, best CFD practices were first established to support the future vehicle development [3]. Three available CFD codes are chosen to compare in the study. Those codes are
OVERFLOW-2: RANS solver based on structured overset grids. Production-level code heavily used for obtaining steady state solutions of complex geometry problems. Capable of solving flow problems of multiple bodies with relative motion.
USM3D: RANS solver based on unstructured tetrahedral grids.
CART3D: Inviscid flow solver (Euler) using Cartesian hexahedral cells with automated mesh generation capability.

4.2.1 Current practices for aero-database generation for ascending vehicle

The best practices for generating aero-database for Ares V in ascent were established by Kiris et al. [3] in order to support vehicle design. Current CFD codes performed reasonably well for characterizing fore body aerodynamics without the plume effects. This procedure shows an example of CFD capability for generating series of vehicle aerodynamics data when the flow is primarily attached and the plume effect is not included in the model.

The database generation is based on series of steady-state conditions along the trajectory. Since the flow does not involve massive separation, the current practices and lessons learned from the sensitivity evaluation can be valuable for vehicle development. Here the aerodynamic data at different points along the trajectory are represented by steady-state solutions using the external flow conditions at that location. Whether these solutions truly represent snapshots of the unsteady flow while the vehicle is in motion needs to be validated in the future.

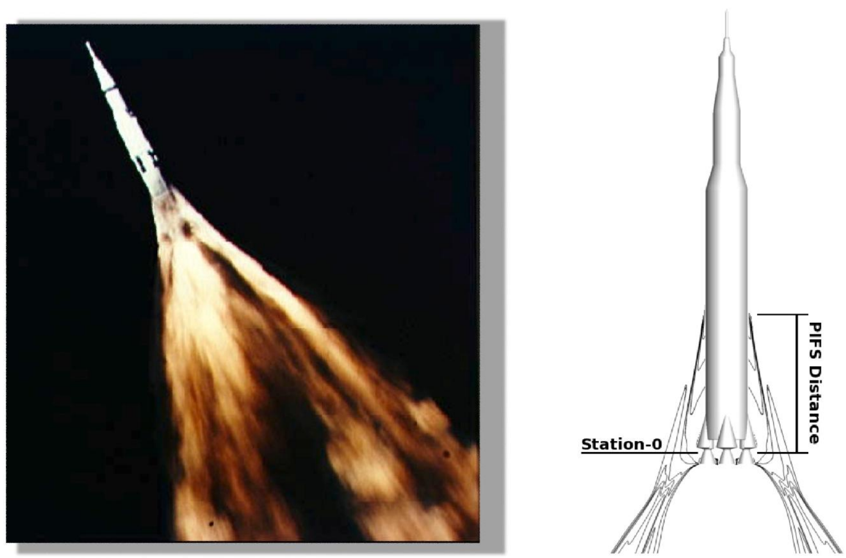
Next example shows similar practices involving plume-induced separation.

4.2.2 CFD procedures for ascending vehicle with plume

Plume impacts the overall aerodynamics of the launch vehicle. To simulate the flow for entire launch vehicle in ascent, plume effect has to be included in the computational model. In this case, there are additional issues to the CFD procedure discussed above in conjunction with an ascending vehicle aerodynamics without massive separation.

As the vehicle ascends exhaust plume expands with increasing altitude. This expanded plume creates adverse pressure gradient in the aft section of the rocket. This creates a phenomenon known as Plume-Induced Flow Separation (PIFS) for rockets at high altitude. Thus the rocket plume affects forces and moment coefficients and load distributions along the vehicle as well as base heating.

Gusman et al. [11] studied CFD procedures for aerodynamics of ascending vehicle with plume effects and the computed results are compared with existing flight data taken from Saturn V launch vehicle [17]. The quality of CFD results is assessed by measured and computed PIFS distance as defined in Figure 6 and Table 1. The PIFS distance was measured at 4 different Mach numbers representing 4 instances during ascent phase. Since the measurements were made from video footage of the launch, the uncertainty of data is considered at approximately 10% level.



(a) (b)

Figure 6. Definition of plume induced flow separation (PIFS) distance: (a) frame from chase plane footage of Apollo 6 flight with PIFS visible by extent of exhaust gas, (b) sketch of PIFS measurement

Correctly resolving PIFS distance is important since it affects the aerodynamic forces and the design of thermal protection system. Current CFD simulation procedures are established by defining

- Mesh resolution requirements
- Boundary conditions
- Physical properties of the plume

– Turbulence model for separated and unsteady flow
The plume inflow boundary conditions are defined at the nozzle exit taken from nozzle computations.

Table 1: Free-stream conditions at four different points in Saturn V ascent trajectory

	M_∞	P_∞ (Pa)	T_∞ (°K)	Re_D
5 F-1 engines firing	1.5	12111.0	217	6.1522×10^7
	2.7	2250.0	221	2.2623×10^7
	4.4	151.0	264	1.6970×10^6
After Center Engine Cut-Off (CECO), only 4 engines firing	6.5	22.0	247	4.0600×10^5

One of the issues to be considered which affect the solution quality is related to turbulence model in the massively separated region due to PIFS. Models used for production calculations such as SA and SST models were originally developed for external aerodynamic computations. Therefore, it is of interest to investigate solution sensitivity to turbulence models which are commonly used in production codes for aerodynamic computations. To study the variations due to existing turbulence models in use, both results from SA and SST models are compared by computing four different cases listed in Table 1.

As the altitude increases, the PIFS increases. The PIFS distances are computed at the four different Mach numbers representing different altitudes. These are compared in Figure 7. The results show high sensitivity to turbulence modeling especially at high altitude where the flow is massively separated.

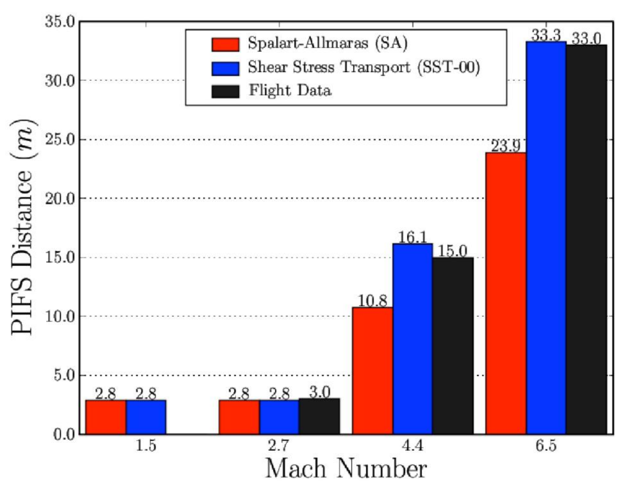


Figure 7. Comparison of computed PIFS distances using SA and SST model and flight data.

Depending on the model, different level of turbulent eddy viscosity is produced distributed through the flow field. This in turn affects the flow development especially in the aft region of the vehicle. To give some qualitative picture of this, a snapshot of turbulent eddy viscosity between the two turbulence models is shown in Figure 8.

In Figure 8, a distribution of normalized turbulent eddy viscosity μ_T/μ_{ref} is shown. The two models produce different levels of turbulent eddy viscosity that affects the plume expansion. This in turn results in different PIFS distances. As illustrated, capturing the correct plume expansion angle is critical for prediction of the PIFS distance. The SST model has a number of correction terms as options. Gusman et al. [11] compared computed results using these correction terms. However, the results show further variations in the PIFS distances and did not show improvements over the original model without any additional corrections.

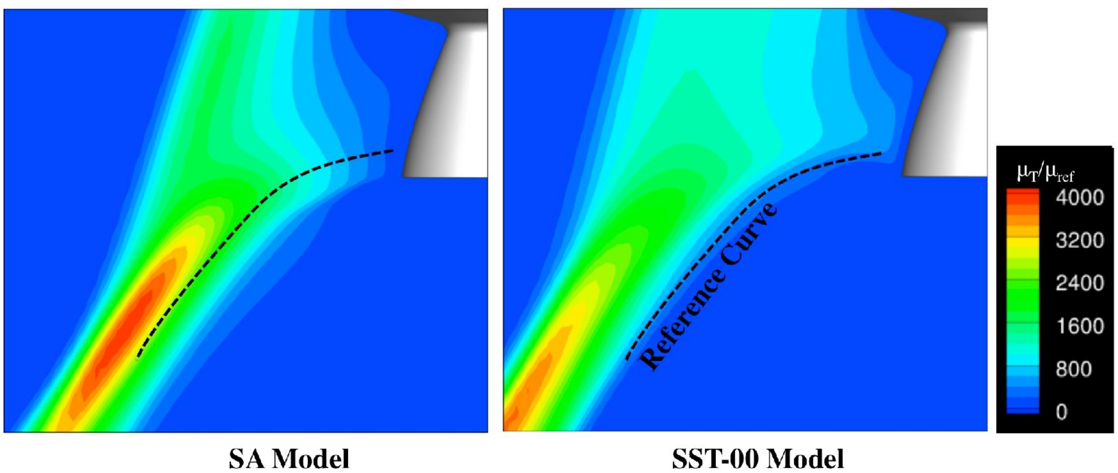


Figure 8. Contours of turbulent eddy viscosity on a slice through an outer F-1 nozzle.

Stage separation adds complexity to plume-vehicle flow computation. Especially jet on jet, jet-PIF interaction and unsteadiness involved makes the flow simulation extremely challenging. Computational results on this case can be found in Klopfer et al. [12].

On the current practices:
A number of factors affecting PIFS are included in establishing best practices procedure [11]. In the current discussion, only the turbulence modeling effect is reviewed. Plume requires various levels of modeling such as nozzle exit boundary conditions, multi-species model, and possibly chemistry involving multiple-engine base flow. All these require quantification of grid resolution, chemistry and turbulence modeling. In the current computational examples, SST model without any additional correction terms produces most reasonable PIFS distance. This suggests that, for general applications, specific guidelines on the range of applicability and accuracy for existing turbulence models will be practically useful. For example, guidelines will be valuable on which models produce consistently better results (or not reliable) for wakes, jets or separated flow regions. Without consistency defining error bound will be difficult thus the numerical results cannot provide reliable information for analysis and decision-making. This example shows yet another case where advanced turbulence modeling approach is in need to better represent non-equilibrium turbulence often encountered in real-world cases.

4.3 Propulsion Subsystems

Generally, there are three categories of fluid dynamics related sub-elements in liquid-propellant engine systems: 1) combustion devices, 2) turbopumps, and 3) complex internal flow subsystems. Analyses of these components offer three corresponding categories of simulation challenges: mixing and chemistry; flow with cavitation; and complex internal flow, respectively. Because of the complexity of the geometry, an experimental approach is extremely difficult, time consuming, and expensive. Therefore, computational simulations have offered an economical approach to complement experimental work in analyzing the original configuration, and to suggest new, improved design possibilities.

Each category has different flow features quite different from each other. The flow analysis can usually be performed in decoupled fashion. Even though quantitative predictions involving these flows are difficult, CFD has been utilized very successfully for engineering support to meet mission requiremenets. Some details on complex internal flow and turbopump analysis can be found in the book by Kwak and Kiris [15]. In this section, some fundamental issues which remain as unsolved problems will be illustrated. Future mission support will be greatly benefited from advanced capability on these issues.

4.3.1 Complex internal flow

A wide spectrum of fluid dynamics issues is associated with a liquid-propellant rocket engine where CFD has been very valuable for analysis and design. Even with early phase of flow simulation technology and computing power, CFD could be used to analyze relative changes in flow characteristics during design modifications, thus making significant contributions in retrofitting the shuttle engine. One of the issues is related to quantifying impact of turbulence in internal flow analysis with large curvatures. Even though the relative importance of strong curvature effect on the overall flow solutions [15], the issue is not fully addressed for computing complex internal flows in general. This is illustrated below:

Computational and experimental study of turbulent flow under strong curvature:

Turbulence models for RANS calculations of aerospace vehicles have been developed mostly for external flows such as on airfoils or wings. The primary focus for external flow has been predicting the boundary layer development and separation. Since these models lack universality, turbulence models for RANS calculations are adjusted to match characteristics of different types of external flow such as boundary layer or free shear flow. For the internal flow with large curvature an *ad hoc* modeling approach was adopted as explained below.

The simplest turbulence model for a RANS computation is an algebraic model. The basic assumption of algebraic models (or of any eddy viscosity models) is that the turbulence is in local equilibrium; that is, the production and dissipation of turbulence are balanced. For this modeling approach, turbulence length scale distribution needs to be prescribed based on empirical input. The empiricism has been mostly based on boundary-layer type flow. Even though this modeling approach cannot handle transport and history effects, simplicity of the expression has been the major advantage, and many external flow problems have been successfully computed using this approach. However, for flows with streamline curvature, separation and/or rotational effects, as well as for unsteady flows, it becomes very difficult to define a generally applicable length scale.

For fully developed internal flows in a duct with mild streamwise curvature, the location of the maximum velocity may be used to determine the eddy sizes. However, in the case of the complex curved duct, the location of the maximum velocity is not a good measure of the boundary layers associated with the two opposite walls. In this case, the flow consists of a pair opposite vorticities. Therefore, there must be a location where both of them vanish, which essentially divides the two boundary layers from opposite walls. In practice, however, it is difficult to locate the position where vorticity vanishes. One alternative is to use vorticity thickness as the distance from the wall to the position where vorticity becomes minimum. With these considerations, the following simple algebraic expression was used for the internal flow being analyzed:

$$\nu_t = l^2 \left|\omega\right| \tag{1}$$

where

ν_t = eddy viscosity

$$\frac{1}{l} = \sum_{i=1}^{n} \frac{1}{k y_i \left[1 - \exp\left(-y_i^+ / A^+\right)\right]} \tag{2}$$

k = *Karman* constant

The experimental configuration conducted at NASA Ames High Reynolds Channel I (HRC I) is depicted in Figure 9. HRC I is a blow down facility using unheated dry air at ambient temperature. After an entrance nozzle, a straight rectangular upstream duct is located with a dimension of 3.8-cm high and 38-cm wide (that is, an aspect ratio of 10), and 83-cm long. It is followed by a 180^o-bend with constant channel gap spacing equal to the centerline radius of 3.8 cm (that is, a radius to gap ratio of 1). Following the bend, another 54-cm long straight downstream section is placed. A throttle plate at the exit of a bottom-settling chamber controls the flow rate. This was adjusted so that the Mach number of the flow was 0.1, which is the same condition as for the turn around duct in the shuttle main engine powerhead.

Tests were conducted with total pressure, $p_{t,}$ at 1.2 and at 12 atmospheric pressure to achieve Reynolds numbers of 10^5 and 10^6, respectively. Large Plexiglas side windows allowed optical access to the entire bend, and to 12H up- or downstream regions. Inner windows incorporated suction slots spaced H apart upstream of the bend to remove the side-wall boundary layers. The suction (combined with the high AR) was intended to keep the flow as two-dimensional as possible. The full detail of this experiment is documented by Monson and Seegniller [18].

A numerical study performed by Monson et al. [19] assessed the ability of a simple algebraic modeling approach for computing this type of internal flow with strong curvature. An algebraic model given by equation (1) was used for this exercise. The length scale is calculated following Burke [20], who used the ad hoc formula given by equation (2) to account for n walls or surfaces.

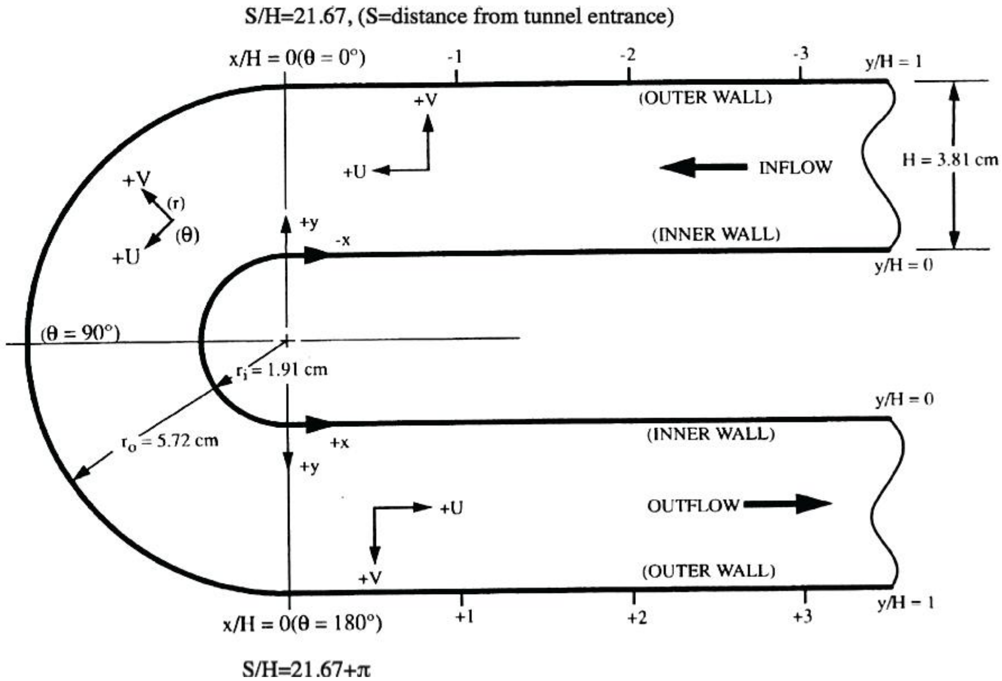


Figure 9. Geometry of 2-D turnaround duct experimental model at NASA Ames High Reynolds Number Channel I (HRC I).

Velocity profiles measured at 4H upstream of the bend are shown in Figure 10. The overlapping data in the figure are taken from separate runs from each wall. The curve fits, shown as a solid and a dotted line in the figure, are used as upstream boundary conditions for computation. The profiles show that about the middle third of the channel contains an inviscid core indicating that the flow is turbulent but not fully developed at this location. The intensity of this core is measured by the LDV at about 1 percent, which is the resolution of the instrument. The boundary layers on the inner wall are somewhat thicker due to an asymmetric test section of the inlet nozzle. The flow angle was measured as zero across the channel. Additional velocity and flow angles taken at several transverse positions in the duct show excellent two-dimensionality of the flow at this inflow location before entering into the bend region.

In Figure 11, computed and experimental velocity profiles are compared at the start of the bend ($\theta = 0^{o}$). The measured profile shows that the flow has begun to accelerate near the inner wall due to a favorable pressure gradient, and the peak velocity is much closer to the inner wall. On the other hand, the flow near the outer wall is decelerated. Generally, the computed and experimental results agree well at this point. This is probably because the flow is mainly driven by the pressure field, and turbulence structure has very little effect, so the algebraic model works reasonably well.

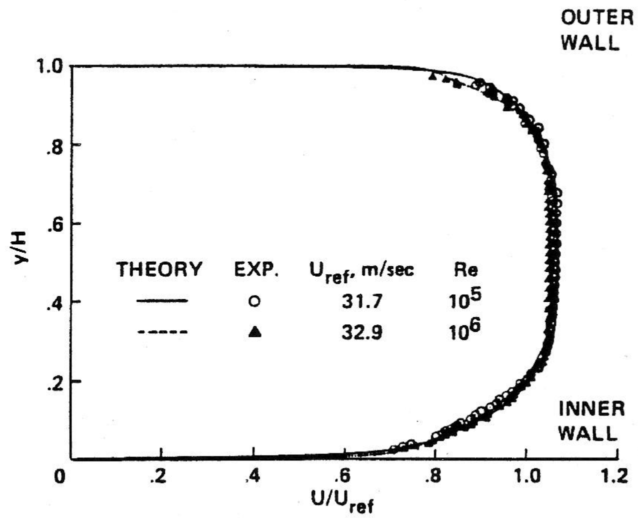
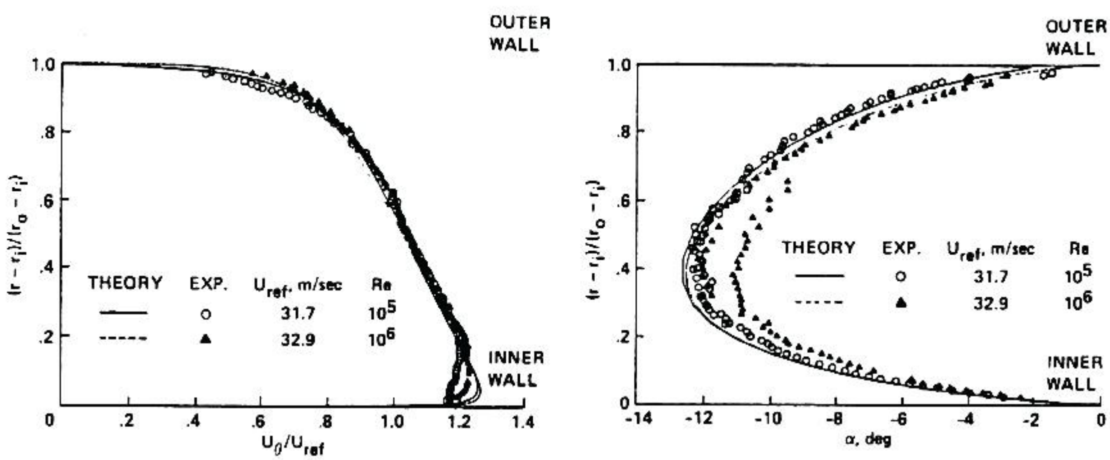


Figure 10. Longitudinal velocity in 2-D U-duct at x/H=-4.0, M=0.1.



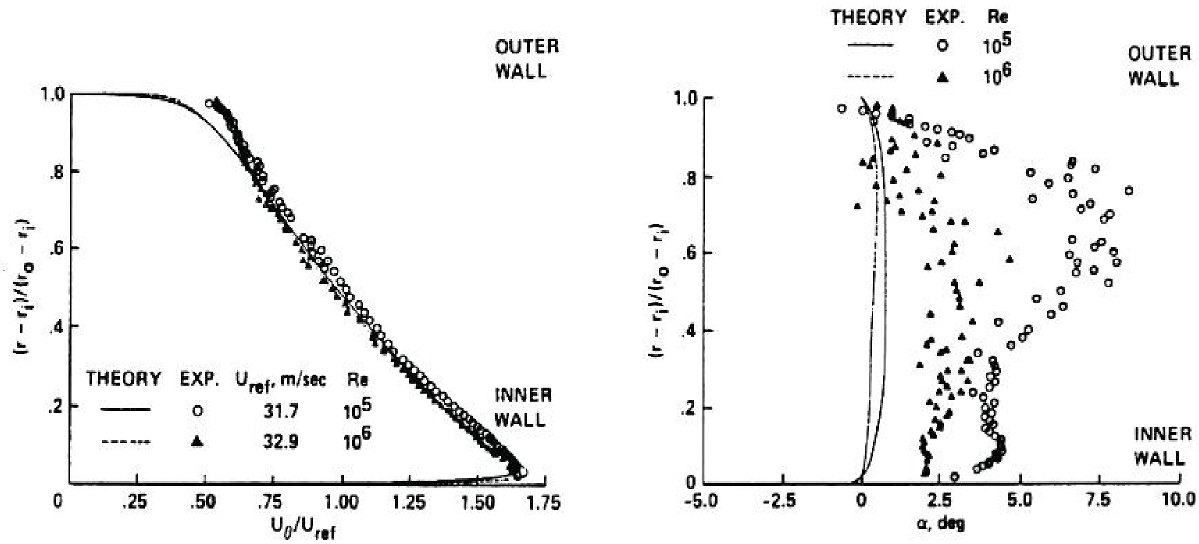
(a) Longitudinal velocity (b) Flow angle

Figure 11. Longitudinal velocity profile in 2-D turnaround duct at M=0.1 and $\theta = 0^o$.

In Figure 12, the velocity profile is shown halfway through the bend (θ=90°). The flow continues acceleration near the inner wall and deceleration near the outer wall. Very few differences are observed between the flow at the two Reynolds numbers.

The boundary layer on the inner wall is extremely thin and may actually be relaminarized. The computed results overpredict the boundary layer thickness on the outer wall. A similar trend is observed in an axisymmetric case that uses an algebraic model by Chang and Kwak [21]. Possible factors contributing to this discrepancy come from the high level of turbulence and unsteadiness in this region, observed during the LDV tests. Sandborn [22] presented evidence for the existence of highly time-dependent instabilities in this region of a turnaround duct. The computed mixing length along the outer wall is nearly constant, indicating that an algebraic modeling approach is likely to overpredict boundary layer thickness in the outer wall region.

The accuracy of an algebraic model suffers in this region. Using other models will result in different boundary layer thickness. For example, Chen and Sandborn [23] reported thinner boundary layer thickness using a $k-\varepsilon$ model. The difficulty may have started from the basic assumption of equilibrium turbulence in using an eddy viscosity model. This type of flow may require higher-level modeling. However, considering that the flow is highly unsteady at this point and will separate after this point, the simple algebraic recipe for eddy viscosity produces remarkably good results nonetherless.



(a) Longitudinal velocity

(b) Flow angle

Figure 12. Longitudinal velocity profile in 2-D turnaround duct at M=0.1 and $\theta = 90^{o}$.

In Figure 12(b), flow angles are plotted. The measured data indicate that flow turns with a much larger positive turning angle. The scatter in the data is due to the unsteadiness of measurements in this region. The computed results deviate widely from the measured flow angle, and this may be due to the extent of separation along the inner wall downstream of the turn.

In Figure 13, the velocity profile is shown at the end of the bend (θ=180^{o}). The velocity profiles indicate that separation bubbles exist on the inner wall just downstream of 150^{o} in the bend—a small one for Re=10^5 (height of 3.1% of H) and a much larger one for Re=10^6 (height of 8.5% of H). The reattachment occurs at x/H between 1 and 2 for Re=10^5, and at x/H=2 for Re=10^6. The separation bubbles observed from the real-time Doppler signals on an oscilloscope appear to be very unsteady and unstable at both Reynolds numbers. At times, the bubbles are completely swept away—then instantly reestablish at aperiodic frequency. The instability of separation bubbles could be a major reason for highly scattered measurement in this flow. Outside the separated region, the flow is accelerated around the bubbles. Sandborn [22] observed a similar phenomenon in his water flow test at Re=10^5.

On the current practices:
Many attempts have been made to develop a higher accuracy, more predictable turbulence models for internal flows since this study was performed. From a CFD applications viewpoint,

relative importance of model accuracy versus consistency need to be assessed. For example curvature effect of turbulence may be small on mean flow features which are dominated by inertial force due to geometric variations. For future engineering applications involving complex internal flows, however, curvature effects need to be incorporated into turbulence models as well as general internal flow models that can produce consistent results under varying degree of bends and turns which are not avoidable in design environment.

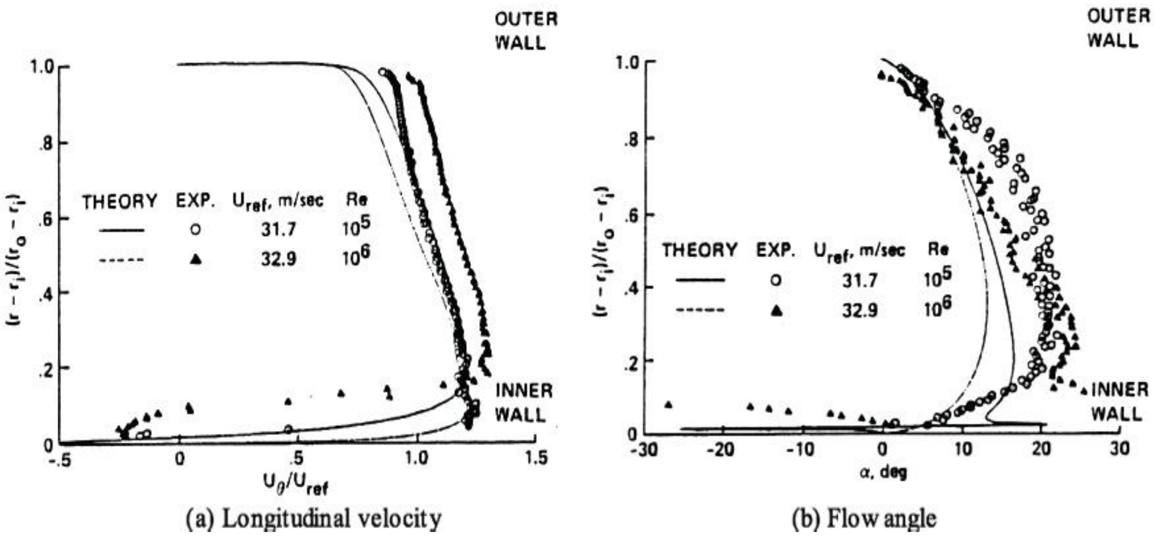


Figure 13. Longitudinal velocity profile in 2-D turnaround duct at M=0.1 and $\theta = 180^{o}$.

4.3.2 Flows involving turbopump

Highly sophisticated pumps driven by high-speed turbines have emerged as liquid-propellant rocket engine technology has advanced, particularly in conjunction with rocket engines for space exploration. Operating conditions of a turbopump vary depending on the specific impulse and the associated engine design approach. For example, the inlet and exit pressures and flow rate for a gas-generator cycle engine are different from that of a staged-combustion cycle engine. In general, liquid-propellant rocket engine turbopumps operate under very severe conditions and are a challenge for numerical simulation.

The turbopump subsystem is the most crucial and expensive element in a liquid-propellant rocket engine. However, turbopumps have been developed semi-empirically for many decades and the unsteady three-dimensional viscous flow phenomena have not been fully accounted for in the design process. Even though CFD applications for turbomachinery have been reported in the literature, realistic applications to design processes have been very limited. This may be due to the difficulties associated with quantifying the unsteady three-dimensional flow in turbopumps that include inducers, impellers, and diffusers (stationary). In addition, it takes many years from design to flight to develop a new or improved turbopump system. To predict pump performance, which is directly tied to the engine performance, salient features of 3-D viscous flow phenomena must be resolved, including wakes, boundary layers in the hub, Flow between shroud and blades, blade-hub juncture flows, and tip vortex flows.

Another important feature related to 3-D unsteady flow in turbopumps that affects the safety and reliability of a rocket engine, is the fatigue on the structure due to flow-induced vibration. Quantifying damaging frequencies and amplitude of this flow-induced vibration is an important challenge to CFD simulation of turbopumps. In addition, cavitation is almost always an issue in high-speed turbopumps in rocket engine, which can be a source of high-frequency vibration. All these phenomena offers many computational challenges, some of which are yet to be resolved.

Although all these factors vary among designs, CFD simulation challenges are partially illustrated here using a generic example from the shuttle flowliner. Full detail can be found in Kiris et al. [24]

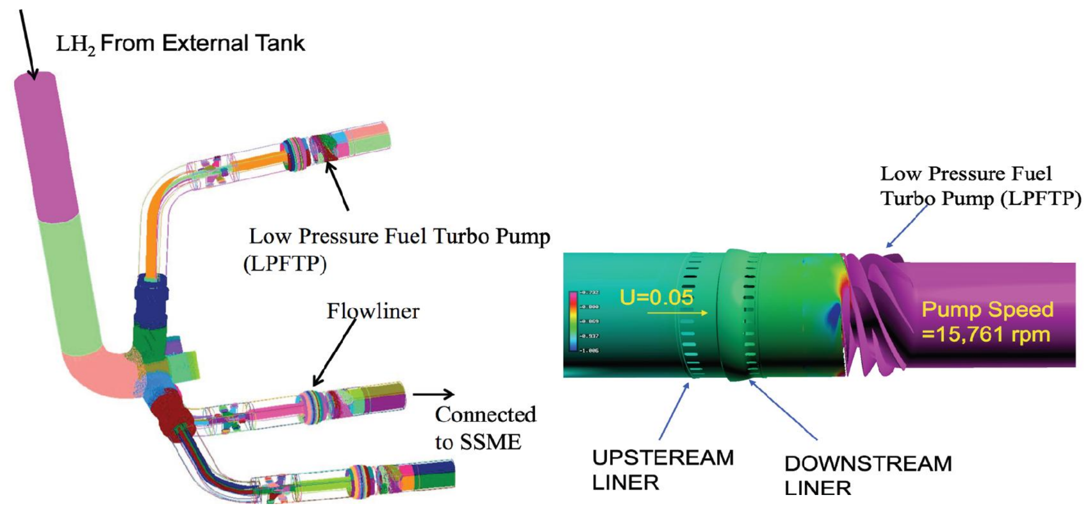
Brief description of the CFD task:
In the shuttle Main Propulsion System (MPS), there are two 17-inch diameter feedline manifolds, one for liquid oxygen (LOX) and another for liquid hydrogen (LH2). These feedlines contain three outlets each, which are connected to each SSME by 12-inch diameter feedlines providing LOX and LH2 to the engine (see Figure 14). Fuel enters the orbiter through gaseous and liquid fuel branches. In each liquid hydrogen branch, LH2 enters the low-pressure fuel turbopump (LPFTP) when the pre-valve is open.

The LPFTP creates transient flow features such as reverse flows, tip clearance effects, secondary flows, vortex shedding, junction flows, and cavitation effects. Flow unsteadiness originating from the orbiter LPFTP inducer is one of the major contributors to the high-frequency cyclic loading that results in high cycle fatigue damage to the gimbal flowliners. The reverse flow generated at the tips of the inducer blades travels upstream and interacts with the bellows cavity. The dynamic environment with limited means of validation makes it very difficult to simulate the flow. In order to characterize various aspects of the flow field near the flowliner and determine vibration frequencies generated from unsteady flow, full-scale tests were carried out using the entire configuration which has three branches, as well as a single flowliner model. The computational model of the test article is shown in Figure 14(a) with an enlarged view of the flowliner region as shown in Figure 14(b). The data obtained using the test articles with one LPFTP were used for validating the simulation procedure. In this section, we discuss only the computational issues associated with predicting flow induced vibration analysis.

The incompressible Navier-Stokes flow solver based on the artificial compressibility method was used to compute the flow of liquid hydrogen in this case. Computations included tip leakage effects with a radial tip clearance of 0.006 inches; a pump operating condition of 104.5 percent rated power level (RPL); a mass flow rate of 154.7 lbm/sec; and a rotational speed of 15,761 rpm. The problem was non-dimensionalized with a reference length of one inch and reference velocity equal to the inducer tip speed. The Reynolds number for these calculations is 3.6×10^7 per inch. Liquid hydrogen is treated as an incompressible single-phase fluid.

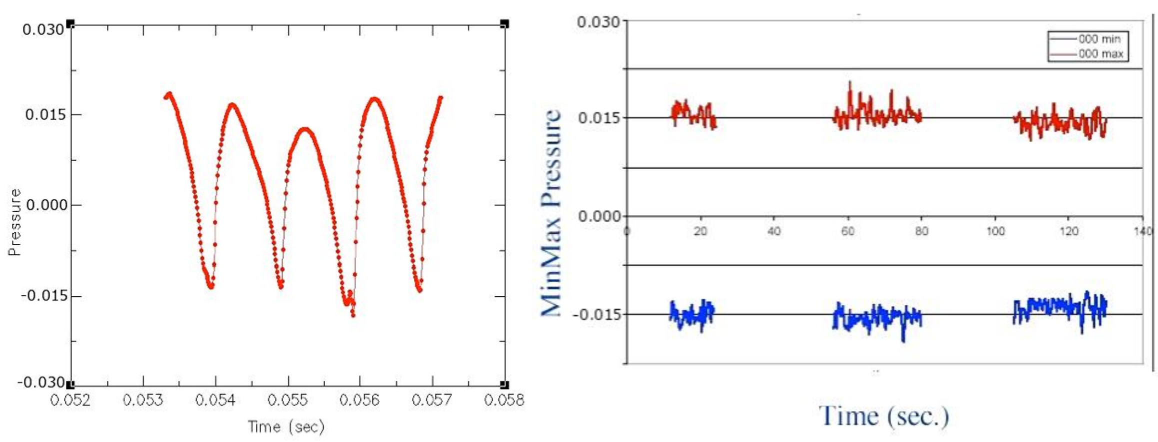
Initially, the flow is at rest. Then, the inducer is rotated at full speed. Mass flow is specified at the inflow, and characteristic boundary conditions are used at the outflow.

The time history of non-dimensional pressure difference from calculations at a location where experimental measurements are taken is plotted in Figure 15(a). Even though computed results have not fully converged to periodic solution in time and still show evidence of start-up transients, the dominant 4N (number of blade times the rotational speed) unsteadiness at a fixed location is seen in the figure. In Figure 15(b), maximum and minimum pressure values are recorded from the experimental data. Comparisons between CFD results and hot fire test data measured at a location near the downstream liner (see Figure 14) also show good correlation in the non-dimensional pressure amplitudes.



(a) (b)

Figure 14. Computational model and flow conditions of the shuttle flowliner.



(a) (b)

Figure 15. Time history of non-dimensional pressure during one inducer rotation, and Min/Max values of non-dimensional pressure from hot fire measurements near downstream liner.

On the current practices:
Quantification of flow features in this unsteady environment is very difficult. Identification of sources and magnitudes of vibration is extremely important to the safety and reliability of the vehicle components involving the pump. The amplitude and the frequency of the unsteadiness due to backflow could be calculated as shown in Figure 15. However, vibration at higher frequencies (two dominant frequencies were observed and the second one is roughly three times higher), possibly due to acoustics and cavitation, could not be resolved with the current computation and still remains to be resolved. In spite of the progress made in multi-phase simulations methods in recent years, cavitation remains an extremely complex physical phenomenon and quantitative prediction is still a major challenge.

5. Future CFD Development in Launch Vehicle Applications

Development of a launch vehicle takes years if not decades. Any successful launch vehicle once developed is used for a long time. For example, Space Shuttle launch vehicle has been used for 30 years. Even though upgrading and modification of some components were done, the original configuration remained more or less intact. For developing a new vehicle, CFD can play a significant role during the conceptual design and trade study phase. This requires consistency in computed results and quick turn-around to evaluate different ideas. Current CFD practices will be able to assess new designs if the configurations do not deviate those developed to date. However, significant benefits can be realized if CFD enables out-of-the-box thinking. We will consider desired capabilities and impact areas achievable under present project environment.

Current tools

To date usable tools have been developed independently from one another to meet their own project requirements. Naturally duplication of the similar capabilities exists. After some code consolidations, the number of codes in many organizations (e.g. different NASA Centers) is now at a more manageable level. Since one universal code may not be suitable for every situation, limited number of specialized codes has served the project needs in many instances.

In applying current simulation tools to determine fluid dynamic loads on a launch vehicle, users are often expected to have CFD and engineering expertise to produce useful results. Physics are often approximated by models of a curve-fitting nature that tends to break down when the range of application is beyond the operating boundaries of validation data. In other words, existing models are primarily designed for interpolation but are not as reliable for extrapolation. Predictive capability can be achieved by the use of modeling which assures the proper non-linear physical phenomena. This will alleviate the need for extensive ground- or flight-testing. The research areas discussed below are the ones needed for developing the next-generation launch vehicles and their operations. Therefore, it is important to identify the major areas of CFD reseaarch which will be needed in the near future to make significant impacts on conceptual design phase of the next vehicles. This will minimize the expensive “test-fail-fix” cycle of the past vehicle development practices. Therefore, it will be necessary to enhance credibility of CFD solutions while solving today’s problem.

Requisite capabilities

Recapitulating the capability required by space exploration mission support (launch and ground operations), we need to quantify the following:

- Forces and moment of launch vehicle in ascent
- 6-DOF- vehicle trajectory
- Stage separation
- Vehicle-propulsion interaction
- Vehicle / systems vibration load, especially, at launch (IOP and launch acoustics)_

Some of the CFD-related issues facing these applications are:

- Are series of steady-state solutions good for generating database for time-dependent flow, such as forces and moments coefficients for launch vehicle during ascent?
- Sensitivity or relative importance of girds need to be determined relative to the effects of turbulence modeling?
- For design applications such as for determining structural loading and for guidance and control, computed results using clean launch vehicles without plume and protuberances seem adequate. However, with plume, the prediction capability for separated region can drastically be reduced.
- Aeroacoustics computation capability is still one-step behind CFD capability
- Combustion instability is not predictable yet.

Target research areas

Specific candidate technologies not actively supported at the moment (within NASA), but can make significant contributions to the exploration mission are listed here. These are the areas where fundamental methodologies can be advanced such that resulting methods can be incorporated into usable tools.

Among various sources, the following areas can be considered major areas of research which can make impacts in launch vehicle applications:

- Unsteady and separated flow research
  To improve current practices for establishing CFD application procedures, advances in algorithm will be desired, such as enhanced time integration schemes/procedures combined with high-accuracy and grid adaption schemes.

- Defining range of applicability for existing or modified turbulence / transition models
  Even though intensive research on turbulence physics has been performed several decades to date, models useful for vehicle development and operations have been at engineering level. To meet the near-term need, it will be necessary to enhance CFD tools to simulate turbulent flow more consistently, for example for flows discussed in

this article. Therefore, establishing a usable recipe for turbulence modeling will be very valuable possibly using existing models and/or enhanced versions.

- Turbulent eddy simulation
  Eddy simulation may reduce uncertainties by modeling small eddies only. However, computing requirement is still an issue. In 1979 Chapman [10] projected that a full aircraft can be solved by LES in 1990s- we have seen 10+ years delayed and still with wall model. Now computers are at petaflop level, but only if the entire system can be allocated to one user. In reality it is reasonable to assume that about 10% of the system will be available to solve one problem. Therefore, it will make economic sense to develop an efficient new methods while computer hardware is being advanced further.

- Engineering-level multi-phase/cavitation model
  No usable guidelines are available yet. However, cavitation phenomena need to be included in CFD model in certain number of important applications.

- Combustion instability
  This requires a longer-term research combined with validation experiments.

Outlook for organized research

Research on many of the areas discussed above are already in progress but lacks coordination. Strategic investments are rare because of the erroneous assumption that CFD reached its maximum potential and further development will results in incremental improvement at best. However, CFD is not accurate enough for many major design decisions except in limited regions of operational conditions in fluid engineering and largely not reliable when extrapolated to un-experimented flight regimes or operational boundaries. It is to be noted that CFD has made a profound impact on airplane design, especially, commercial transport airplane. Similar impact can be expected in space vehicle development when "prediction" capability is improved and when CFD can produce consistent results. To close the current gap in launch vehicle applications, strategic investment at fundamental technology level would be desirable. However, the current environment is not very amenable to this approach. Limited focus areas have been supported based on medium term projection. However, long-term investment decision is complex and depends on schedule and resources available for any particular vehicle development program. Any major redirection to this trend may take a while. In the meantime, collaboration among research groups, especially on fundamental research, may accelerate the progress on some bottleneck areas.

6. Closing Remarks

The list presented above is based on authors experience related to launch vehicle analysis and operations, and may represent only a small portion of CFD challenges we face today. Historically, the Shuttle was designed without much help from CFD and has flown for 30 years since its maiden flight. Yet, subsequent advances in CFD and compute hardware has made significant impacts on many aspects of retrofitting, operational support and accident investigation ever since. Those requisite capabilities listed above if made available in the near term can make significant impacts on supporting the next generation exploration vehicle development and operations. We also believe that longer-term strategic research and development, especially on the development of more universally applicable turbulence and possibly transition models, will have far-reaching impacts on aerospace engineering in general as well as computational fluid engineering for space flight vehicles.

Acknowledgements

We present this work as a token of our high esteem of Professors Fujii, Nakahshi and Yang, and cherish our long association with them during the period when CFD became an indispensable element in aerospace sciences and engineering.

References

1. Johnson, F.T., Tinoco, E.N., and Yu, N.J., "Thirty Years of Development and Application of CFD at Boeing Commercial Airplane, Seattle," ***Computers and Fluids*** **34,** pp1115-1151, 2005.
2. Kiris, C., Housman, J.A., and Kwak, D., "Space/Time Convergence Analysis of a Ignition Overpressure in the flame Trench," CFD Review 2010, World Scientific, 2010.
3. Kiris, C., Housman, J., Gusman, M., Schauerhamer, D., Deere, K., Elmiligui, A., Abdol-Hamid, K., Parlette, E., Andrews, M., and Belvins, J., "Best Practices for Aero-Database CFD Simulations of Ares V ascent," 49th AIAA Aerospace Sciences Meeting, January 4-7, 2011, Orlando, FL.
4. Spalart, P.R., Allmaras, S.R., "A one-equation turbulence model for aerodynamic flows," AIAA 30th Aerospace Sciences Meeting, January 1992, Reno, NV, AIAA Paper 92-0439.
5. Baldwin, B.S., Barth, T.J.: A one-equation turbulence transport model for high Reynolds number wall-bounded flows. AIAA Paper No. 91-0610 (1991).
6. Menter, F., "Zonal Two Equation *k*-w Turbulence Models for Aerodynamic Flows," AIAA 23rd Fluid Dynamics, Plasma-dynamics and Lasers Conference, Orlando, FL, July 1993, AIAA Paper 93-2906.
7. Kline, S.J., Morkovin, M.V., Sovran, G., and Cockrell, D.J., "Computation of Turbulent Boundary Layers-1968," AFOSR-IFP-Stanford Conference Proceedings, Volume 1-Methods, Predictions, Evaluation and Flow Structure.
8. Kline, S. J., Cantwell, B. J., Lilley, G. M., "Conference on Complex Turbulent Flows: Comparison of Computation and Experiment," Stanford University, Stanford, CA, September 3-6, 1980, Proceedings. Volume 1 - Objectives, evaluation of data, specifications of test cases, discussion and position papers. Conference sponsored by the U.S. Air Force and Stanford University. Stanford, CA, Stanford University, 1981, 676 p.
9. Reynolds, W. C., "The Potential and Limitations of Direct and Large Eddy Simulations," Whither Turbulence? Turbulence at the Crossroads: Proceedings of a Workshop Held at Cornell University, Ithaca, NY, March 22–24, 1989. Editor: J. L. Lumley, Lecture Notes in Physics, vol. 357.
10. Chapman, D.R., "Computational Aerodynamics Develoment and Outlook," AIAA J. vol. 17, no. 12, December 1979.

11. Gusman, M., Housman, J., and Kiris, C., "Best Practices for CFD Simulations of Launch Vehicle Ascent with Plumes- OVERFLOW Perspective," 49th AIAA Aerospace Sciences Meeting, January 4-7, 2011, Orlando, FL.
12. Klopfer, G., Kless, J., Lee, H.C., Onufer, J.T., Pandya, S, and Chan, W., "Validation of OVERFLOW for Computing Plume Efffects during Ares I Stage Separation Process," 49th AIAA Aerospace Sciences Meeting, January 4-7, 2011, Orlando, FL.
13. McConnaughey, P.K. and Schutzenhofer, L.A., "Overview of the NASA/Marshall Space flight Center (MSFC) CFD Consortium for Applications in Propulsion Technology," AIAA, SAE, ASME, and ASEE Joint Propulsion conference and Exhibit, 28th, Nashville, TN, July 6-8, 1992.
14. Garcia, R., Jackson, E.D., Schutzenhofer, L.A.: A summary of the activities of the NASA/MSFC pump stage technology team. Proc. Fourth Intl. Symp. Transport Phenomena and Dynamics of Rotating Machinery, Honolulu, Hawaii, April 5-8, 1992.
15. Kwak, D., and Kiris, C., *Computation of Viscous Incompressible Flows*, Springer, November 2010.
16. Ryan, R.S., Jones, J.H., Guest, S.H., Struck, H.G., Rheinfurth, M.H., and Vederaime, V.S., "Propulsion System Ignition Overpressure for the Space Shuttle." NASA Technical Memorandum 82458, 1981.
17. Group, S.F.E.W., "Saturn V Launch Vehicle Flight Evaluation Report-AS-506 Apollo 11 Mission," NASA Technical Memorandum TMX-62558, NASA/MSFC, 1969.
18. Monson, D.J. and Seegmiller, H.L., "An Experimental Investigation od Subsonic Flow in a Two-Dimensional U-Duct," NASA TM 103931, July 1992.
19. Monson, D.J., Seegmiller, H.L., McConnaughey, P.K., "Comparison of LDV Measurements and Navier-Stokes Solutions in a Two-Dimensional 180-degree Turn-Around Duct," AIAA Paper 89-0275, 1989.
20. Burke, R.W., "Computation of Turbulent Incompressible Wing-Body Junction Flow," AIA Paper 89-0279, 27th Aerospace Sciences Meeting, Reno, NV, January 9-12, 1989.
21. Chang, J. L. C., and Kwak, D. "Numerical study of turbulent internal shear layer flow in an axi-symmetric U-duct," AIAA Paper 88-0596, 1988.
22. Sandborn, V.A. "Measurement of turbulent flow quantities in a rectangular duct with 180-degree bend," NASA CP 3012, Advanced Earth-to-Orbit Propulsion Technology, **II**, 292-304, Proc. Conference at NASA Marshall Space Flight Center, May 1988.
23. Chen, Y. S., Sandborn, V.A., "Computational and experimental study of turbulent flows in 180-degree bends," AIAA Paper 86-1516, 1986.
24. Kiris, C., Kwak, D., Chan, W., and Housman, J. A., "High-Fidelity Simulations of Unsteady Flow Through Turbopumps and Flowliners," ***Computers and Fluids*** **37,** pp536-546, 2008.