Application of Computational Fluid Dynamics in Missile Engineering

David A. Frostbutter, Brian E. McGrath, and Robert P. Rogér

The Mechanical and Aeronautical Engineering Group within the APL Air Defense Systems Department has developed a unique resource of personnel and specialized computer hardware and software to effectively practice the discipline of computational fluid dynamics (CFD) throughout the missile development process. A suite of CFD codes is regularly used to predict the performance of conceptual missiles, support and supplement wind tunnel tests, aid in the development of flight-critical hardware, and support forensic analyses of flight anomalies. Key to the successful application of CFD is understanding the limits that the discipline brings to the problem-solving process and diligence in validating both the computational codes and their results. Example problems and CFD solutions presented in the article illuminate the process and illustrate the added value CFD brings to missile engineering.

INTRODUCTION

Computational fluid dynamics (CFD) is increasingly applied throughout the development cycle of advanced tactical missiles. This increased reliance on CFD is due, in part, to the increased demands being made on the missiles for higher speed, greater maneuverability, multiple missions, and the maturity of the CFD discipline. Recent CFD developments in multizone structured, unstructured, and adaptive grids, along with multiprocessor algorithms, have drastically reduced the time required to obtain results.

Another factor in the increasing role of CFD is the engineering efficiency it brings to the missile system design and development process. Declining DoD research, development, test, and evaluation budgets require using the most efficient and effective techniques. Properly applied, the discipline of CFD has proven to be a capable alternative and sometimes adjunct to empirical and experimental approaches in solving complex flow and heat transfer problems. The versatility of the discipline provides a relatively low-cost means of evaluating design alternatives, and the computational output can be both prodigious and insightful.

The Air Defense Systems Department (ADSD) is committed to maintaining a superior CFD capability for its military defense customers. A multidisciplinary staff, comprehensive set of CFD tools, and relevant computer resources have been assembled to accomplish that goal. The mission is to apply developed CFD technology wisely, effectively, and efficiently.

Over the past decade, ADSD has become proficient in the application of a suite of generalized and specialized CFD codes. Grid generation codes, flow solvers, and flow visualization tools developed commercially, at government laboratories, or in academia are used. Fully viscous Reynolds-averaged Navier-Stokes (RANS) flow solvers are capable of handling complex flow phenomena such as laminar and turbulent boundary layer growth, separation, reattachment, eddies, viscous dissipation, and vortex formation and shedding. When the flow field has special properties (e.g., the flow is supersonic everywhere) or approximations about the flow are invoked (e.g., the boundary layer is thin and can be neglected), the Department employs lower-order codes that are fully capable of providing credible solutions. The decision of which code or method to use to solve a particular problem depends on the physics of the problem, previous experience, the needed fidelity, and scheduling and funding constraints.

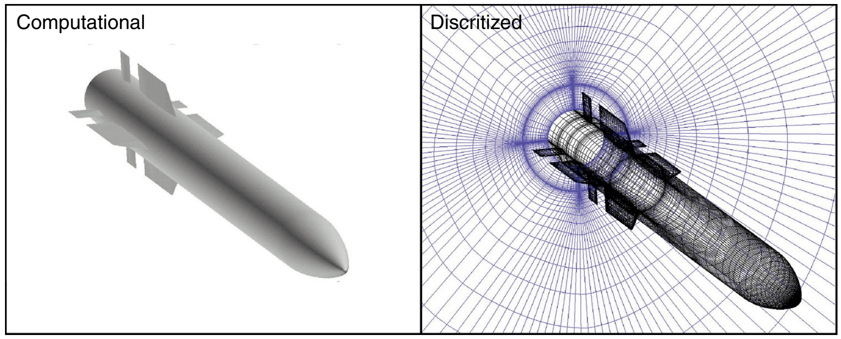


Figure 1. Continuous flow domain of interest.

The following sections describe CFD approaches to missile engineering problems, the resources at ADSD's disposal, specific applications of CFD, and the process used to validate the computations. Future challenges with CFD are also addressed in terms of the types of technical problems faced and the effort required for ADSD to maintain its relevance in this rapidly changing discipline.

The Discipline of CFD

Missile aerodynamics, from subsonic through hypersonic speed, is governed by the fundamental equations of fluid dynamics. These equations are mathematical statements for conservation of mass, momentum, and energy, together with the equations of state relating pressure, density, and temperature for the fluid. Direct solution of this equation set for a general flow problem is not possible because of its nonlinear nature. The discipline of CFD seeks numerical solutions to linearized versions of this equation set. The continuous flow domain of interest (Fig. 1) is discritized (i.e., divided into many small subvolumes constituting the computational grid), and the partial derivatives in the equation set are replaced with appropriate finite difference representations. The discritized equation set is then linearized, and the resulting system of linear equations, applied to each of the many subvolumes, is solved with standard numerical techniques.

CFD also attempts to correctly model additional physics related to what happens either in the fluid (e.g., turbulence, chemical reactions) or at the surface of solid boundaries (e.g., skin friction, heat transfer). The solution of the resulting large set of algebraic equations necessitates access to high-speed, multiprocessor digital computers. Over the years, the term "CFD" has become associated with large, fast computers.

The Place for CFD in Missile Design

There is a fairly well-defined process by which a missile goes from an initial concept to a fully operational system (Fig. 2). CFD has many roles in this process, covering the full gambit of development phases and ranging from concept evaluation, to wind tunnel testing, to hardware development and flight testing.

Characterizing missile airframe aerodynamic loads (forces and moments) and the aerothermal environment is a critical aspect in missile engineering. The aerodynamic information is used to assess stability and control, mission performance, and maneuverability. This information feeds into the design of the guidance and control system, actuators for control surfaces (fins), and development of high-fidelity six-degree-of-freedom (6-DOF) simulations. Aerothermal loads are required to determine protective insulation and control surface material requirements.

The aerodynamic data needed are generally extensive because air loads vary with Mach number, altitude, angle of attack (AoA), aerodynamic roll angle, control surface deflection, and thrust operation. Obtaining a large wind tunnel database by using scale models has traditionally been the first order of business after a preliminary design has been selected. The wind tunnel process produces a wealth of data and provides realistic results for many flow problems; however, the process is not trivial in terms of time and money.

CFD brings added intelligence to critical design decisions during the wind tunnel testing phase. For example, inviscid computations significantly assist in the scale-model design. Prediction of surface pressure and heat transfer distributions aids in instrumentation selection and placement. Proposed scaling laws and the effects of

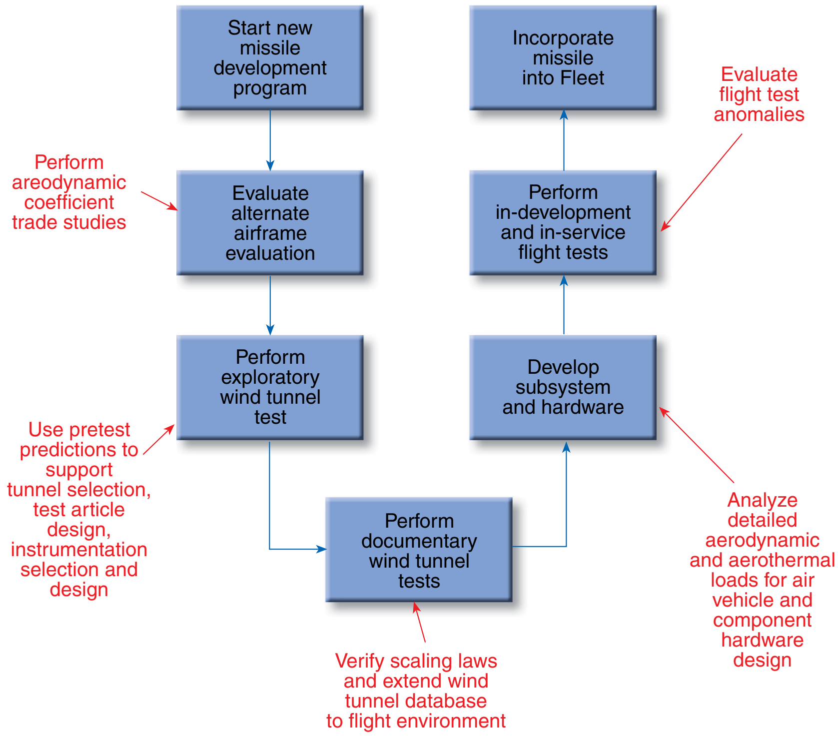


Figure 2. The role of CFD (red) in the overall missile airframe and component development process (blue).

test chamber walls and flow asymmetries can be evaluated before tests are conducted. The intelligent application of CFD can even aid in the design of the entire test matrix. In general, CFD is employed to supplement and extend test results. In addition, the flow visualization provided by CFD can be invaluable in all phases of wind tunnel testing. Surface streamlines, surface pressure distributions, wake line filaments, and other visualization aids provide opportunities to "observe" the flow and its interaction with various airframe components. These visual clues can offer valuable insight into the behavior found in test data and often lead to meaningful design changes.

In the foreseeable future, CFD will not replace a well-planned and well-executed wind tunnel test series; however, the infusion of CFD into the wind tunnel test process can save development costs and eliminate the unexpected costs of repeating poorly and hastily planned tests.

Another use of CFD can be found in hardware development. Testing alone, as a means of developing advanced technology missile component hardware, is very costly. This is particularly true if the survivability and operability of the hardware are critical to the mission and its margin of safety varies with the missile's operational envelope. Expensive testing is most effective when CFD analysis is applied early in the component design process. If the successful performance of the component involves direct flow of gases through the device or testing in a wind tunnel for flow over it, no wind tunnel test should be performed without preliminary CFD analysis. Failure to follow this approach has proved very costly, not only monetarily but also in significant schedule delays. For certain component designs, CFD analysis coupled with thermal and structural analyses is the accepted engineering paradigm, and testing is a verification process.

A final use of CFD is in the post-flight test forensics process. Excluding all direct mechanical, electrical, timing, or communication problems, flight test anomalies related in some way to aerodynamics can occur. Anomalies can range from an unpredicted perturbation during stage separation, to an asymmetry not found in the preflight prediction, to a catastrophic failure with few telemetry clues as to the cause. In these cases, the application of CFD has become a standard tool to investigate hypothesized aerodynamic causes of the anomaly. In a rapid fashion, CFD models of the suspected cause are developed and revised air loads are computed. These aerodynamic revisions are then incorporated into the 6-DOF aerodynamic model and simulation, which are then used to try to replicate the observed anomalous behavior. The ability to quickly model several hypothesized causes and test them has proven very effective in the postflight forensic process.

Time Required for Obtaining CFD Solutions

CFD usually solves the appropriate linearized flowfield equation set using a time-iterative process. Both time-accurate (unsteady) and steady-state solutions can be generated. Obtaining a time-accurate solution requires much longer (order of magnitude) run times than obtaining a steady-state solution. In the steady-state mode, a pseudo-time iteration scheme is usually employed in which time steps per iteration vary locally in the grid according to the size of the volume element. Solving a turbulent flow problem on a Silicon Graphics Inc. (SGI) Origin workstation with multiple 250-MHz processors requires times of the order of 40 μs per volume element to perform one iteration. A typical three-dimensional (3-D) computational grid contains 3×10^6 volume elements. It usually requires 2×10^4 iterations to produce a steady-state solution on a well-constructed computational grid. The availability of processing time on a suitable computer is also a factor. If a multiprocessor supercomputer is available and high priority can be obtained, it is possible to achieve 12 h of processing per day. Thus, by using only one processor on such an in-house resource, a typical 3-D problem can be completed in 8 to 10 weeks. If multiprocessing is possible, the same problem can be completed in 1 to 2 weeks using 8 to 10 processors simultaneously. Table 1 summarizes current time requirements for a typical 3-D model computation with 3×10^6 volume elements.

CFD Applications

The ADSD CFD Team uses its resources (Fig. 3) to apply well-established CFD techniques to solve a wide variety of fluid flow problems that occur throughout the missile development processes (Fig. 4). Subsonic and supersonic compressible external missile flows comprise a large part of this experience. These include low-to-moderate AoA aerodynamics, control surface aerodynamics, control thruster jet interactions, stage separation, actively cooled and uncooled infrared (IR) seeker windows, combustor inlet flows, and hypersonic chemically reacting flows. Low subsonic and incompressible flow applications complete the remainder of the types of problems that have been investigated at length. These include complete airframe missile aerodynamics and internal solid-gas generator missile control thruster flows. The next section discusses in detail a few of these topics and results produced.

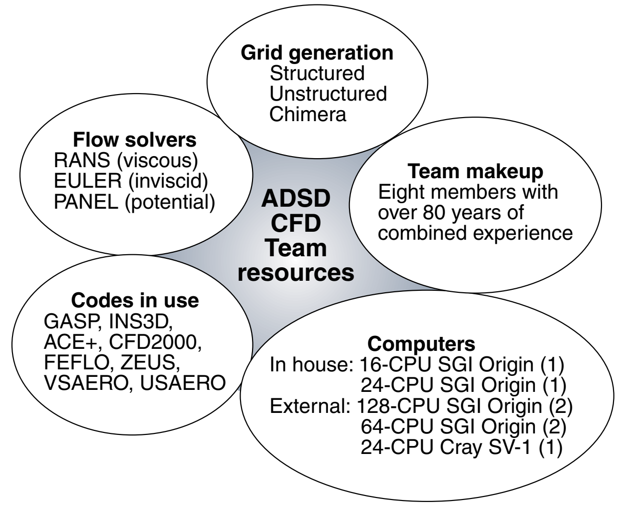


Figure 3. ADSD CFD Team resources.

SUPERSONIC AND SUBSONIC VISCOUS APPLICATIONS: SM-2 BLOCK IVA AND SM-3

The Navy is developing both an upper- and lower-tier variant of Standard Missile (SM) for tactical ballistic missile defense. Navy Area tactical ballistic missile defense will employ an SM-2 Block IVA interceptor operating within the atmosphere using a passive, side-mounted IR seeker for homing. The threat ballistic missile is destroyed using a conventional explosive warhead or by direct hit encounters. Navy Theater Wide (NTW) tactical ballistic missile defense will employ an SM-3 to defend significantly larger areas. SM-3 intercepts the threat outside the sensible atmosphere and uses a kinetic energy hit-to-kill vehicle, the kinetic warhead (KW), to destroy the threat. As Technical Direction Agent, ADSD is currently involved with the development and testing of all aspects of both systems, and CFD has been used to assist in the development of critical hardware.

Table 1. Current time requirements for CFD computations.

Computation	Time
Steady, turbulent, one central processing unit (CPU)	8 weeks
Steady, turbulent, eight CPUs	1 week
Steady, inviscid, eight CPUs	2 days
Steady, inviscid, incompressible, one CPU	1 day
Unsteady, inviscid, incompressible, one CPU	2 to 3 days

Note: Flow-field solution only; does not include grid model generation.

The SM-2 Block IVA IR hemispherical dome-shaped seeker window is actively cooled with argon gas jets. Window survivability and the ability of the seeker to acquire and home in on a target are highly dependent on coolant flow rate, trajectory, and flight duration. For the window and its cooling system, a highly detailed CFD model of the supersonic flow field over the missile forebody and IR seeker window was developed.

The SM-3 KW subsystem uses an IR seeker in combination with a system of control thrusters for steering. The thrusters employ a solid-fuel gas generation system. This Solid-rocket Divert and Attitude Control System (SDACS) is pushing the state of the art in solid propellant exhaust containment and control technology. Flow paths internal to the SDACS have been modeled with CFD to capture the complex flow field and heat transfer rates in support of its evolving design. The following sections summarize some of the CFD work supporting these two hardware development areas.

Actively Cooled Seeker Housing

Figure 5 shows the location of the side-mounted seeker dome on the missile and includes sample CFD predictions performed to supplement wind tunnel tests. The color-coded flow-field results correspond to a case

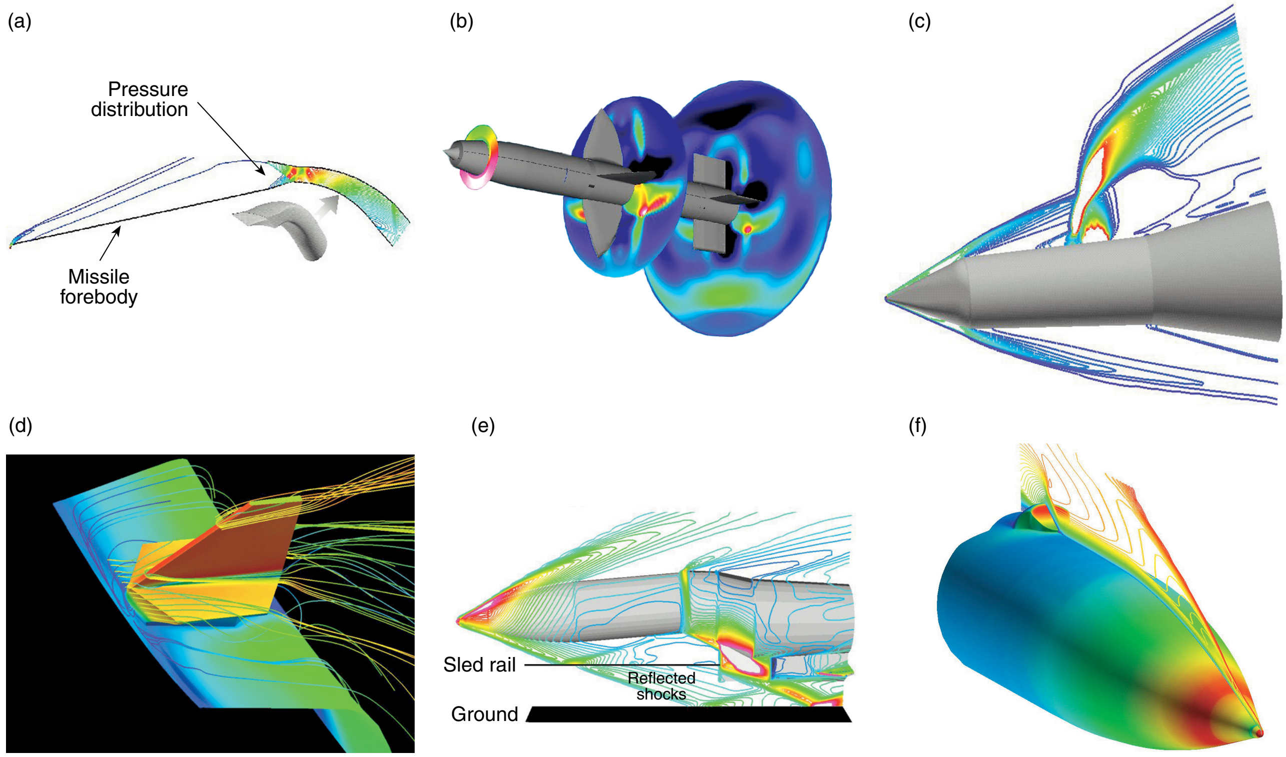


Figure 4. Examples of CFD in the missile development process: (a) supersonic inlet, (b) full airframe aerodynamics, (c) control jet interaction, (d) jet-vane thrust vector control, (e) forebody rocket sled test, and (f) IR seeker window environment.

in which the argon jets are activated. The distribution of pressure on the seeker window is an indication of the nonuniform heating to which the window is exposed. The supersonic character of the flow is evident from the missile bow shock as well as the shock system created by the coolant exiting the slots and by the dome.

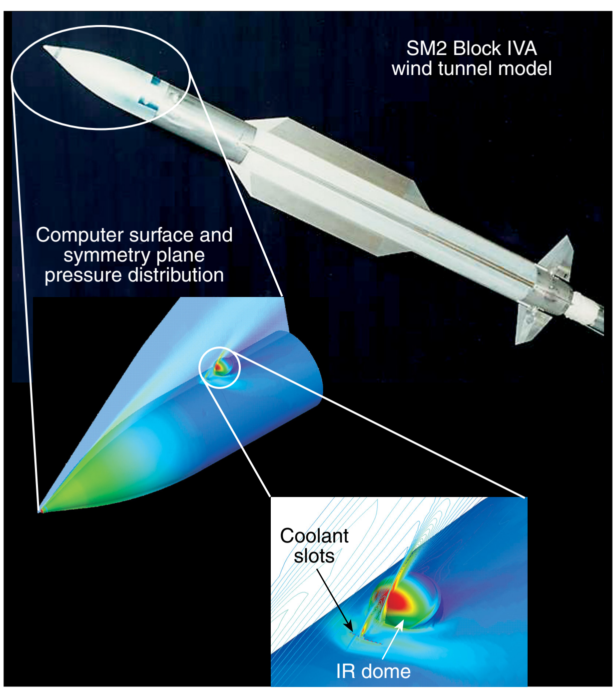


Figure 5. SM-2 Block IVA IR seeker.

The flow in the vicinity of the seeker window is very complex. The physics in this large, separated region involves the mixing of air and the argon coolant gas and the influence of the resultant combined transport properties of the mixture on window cooling effectiveness. Thus, a fully viscous multispecies solution is required. The computations for this system are performed with the General Aerodynamic Simulation Program (GASP) code, a well-verified compressible RANS flow solver.[1] GASP incorporates a finite-volume, characteristic-based algorithm employing a multizone structured grid to solve the thin-layer Navier-Stokes, parabolized Navier-Stokes, or Euler equations. Steady-state solutions are obtained by integrating the conservation equations globally in a pseudo–time-iterative fashion or in a space-marching mode. They are derived from a starting solution consisting of free-stream values everywhere. The flow is modeled as turbulent, using a low Reynolds number two-equation turbulence transport model.[2] Only part of the missile forebody, from nosetip to just downstream of the housing, is modeled. The computations on a grid containing 8.3 million points are performed on supercomputers provided by DoD.

The internal shape of the discrete Mach 2 argon jets themselves, as well as their distribution on the missile surface upstream of the dome, was almost completely designed using CFD. The position of the jets needed to provide adequate coolant as the missile undergoes pitch and yaw motions was determined through a lengthy parametric CFD study. The analysis also helped to determine which jet positions were fed argon gas as the missile maneuvers to engage the threat.

CFD was also used to supplement wind tunnel data in developing the coolant algorithm to be used onboard the missile during the test flight phase of the program. A comparison of CFD results with some wind tunnel data is shown in Fig. 6, in which the predicted shock structure for no coolant flow makes an excellent comparison to a wind tunnel test schlieren image.

CFD contributions to the design of the coolant jet configuration and the development of the coolant delivery algorithm involved more than 100 full 3-D computations, which were computed over a 5-year development period.

SDACS Internal Flow

The ADSD CFD Team members are active members of the Navy's Integrated Product Team for SM-3 SDACS development. One aspect of that role has been to observe and evaluate demonstration unit tests and support post-test analyses. An analysis was required to aid in understanding the flow field and heat transfer rates within the main thruster assembly (MTA) dome region following a test that resulted in a fractured dome. Figure 7a shows

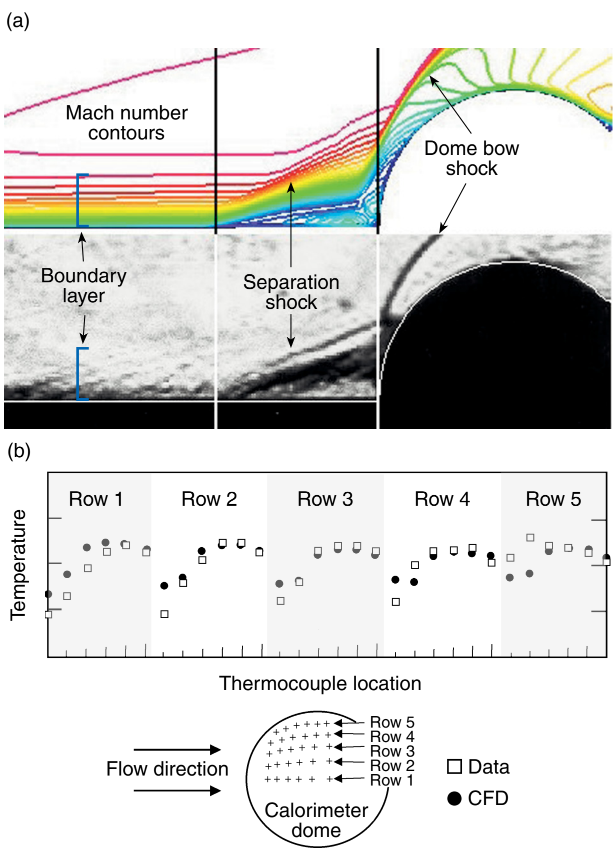


Figure 6. CFD modeling is used to supplement wind tunnel data, as in these examples: (a) comparison of CFD-predicted shock structure and an actual wind tunnel schlieren image, and (b) comparison of CFD-predicted temperature and dome calorimeter data.

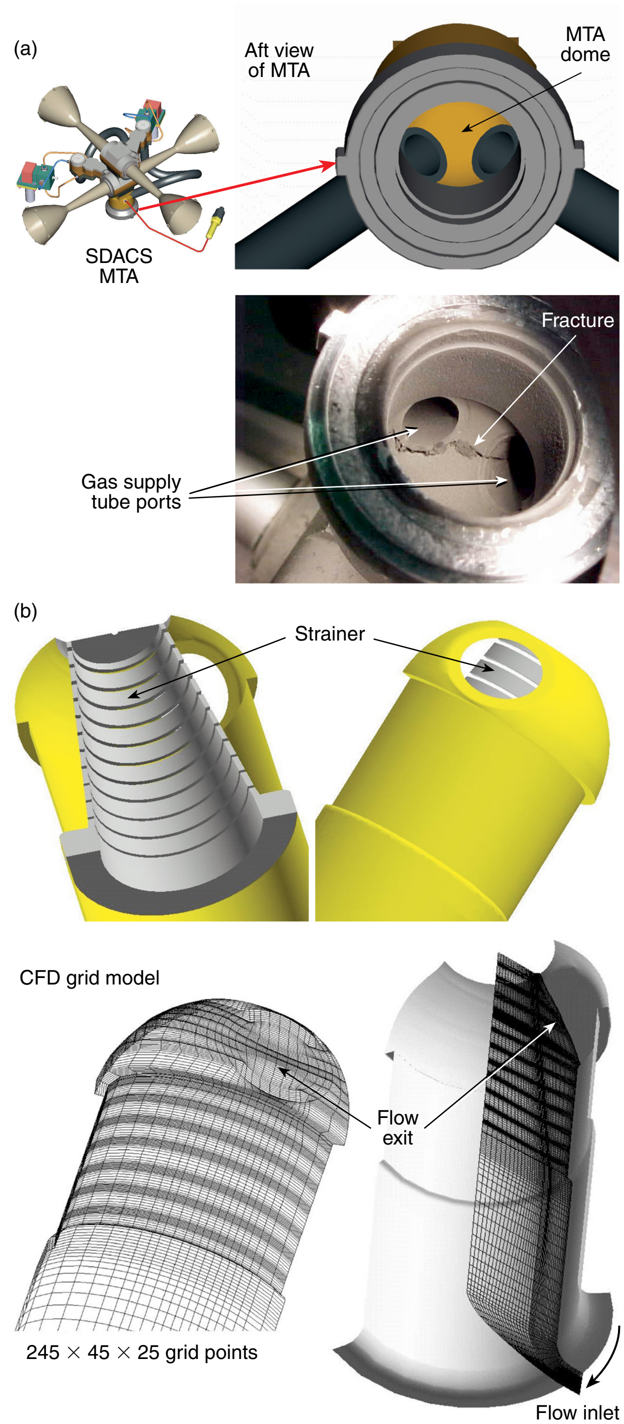


Figure 7. The SDACS main thruster assembly (MTA) was analyzed for the cause of a demonstration failure, which was due to a fracture in the MTA dome in (a). A CFD model of the dome area (b) was created, including a strainer that would be present in production units.

the SDACS MTA, the MTA dome, and a photo of the fracture in the dome. The hot gases flow from the burning solid propellant, go through a filter plate and strainer, enter the MTA dome to supply tubes, and flow to a switching device that finally directs the gas to the divert thruster nozzles. The MTA dome is where the gas flow splits for the main thrusters. The fracture occurred between the two gas supply tube locations in the dome. The demonstration unit test was conducted without an MTA strainer. The function of the strainer is to prevent debris (e.g., charred insulation) from entering the MTA valves, and its importance in the failure had to be determined.

Two separate 3-D CFD analyses, one with and one without the strainer, were needed to ascertain the differences in the flow patterns. The CFD model of the internal surface of the MTA dome and the strainer is shown in Fig. 7b. The pressure field and flow patterns within the MTA dome were performed in 3-D using a moderately tight computational grid. The steady-state boundary condition for the gas flow into the interface tube and dome was based on a separate APL CFD analysis[3] of the coarse strainer, filter plate, and central tube. The structure walls were set to be no-slip adiabatic surfaces. The exit plane was held at a pressure slightly less than the inlet. Computations were performed using GASP. The flow was modeled as turbulent using a standard high Reynolds number two-equation turbulence transport model. A sample of the computed MTA dome internal surface flow pattern is shown in Fig. 8. CFD predictions of the MTA dome flow showed that the pattern was relatively insensitive to the presence of the strainer, implying that heat transfer from the gases to the walls in that region is also essentially independent of the presence of the strainer. CFD visualization also showed a very complex gas swirling pattern as a result of the physical arrangement of the dome and thruster supply tube.

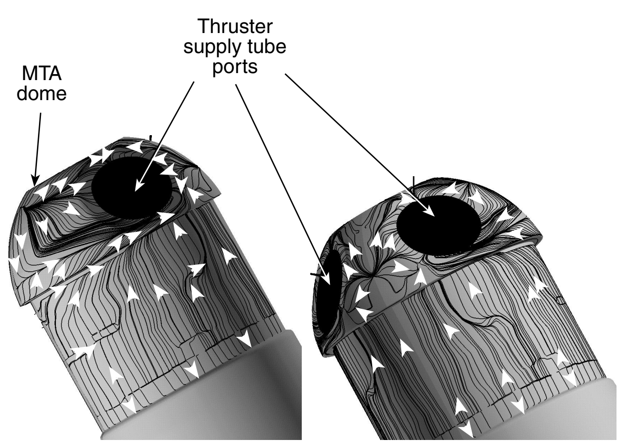


Figure 8. Computed flow patterns on the MTA dome inner surface with strainer.

Heat transfer coefficients in the dome and entrance region to the gas supply tubes were needed as well. Accurate heat transfer rates require a very tightly packed computational grid at the solid walls of the model. An ultrafine grid model dramatically increases the time for a converged solution; consequently, the analysis was conducted with a 2-D model, which was developed by sectioning a select region of the MTA dome. The 2-D flow and heat transfer computation was performed using CFD-ACE+.[4] This analysis showed that the maximum heat transfer rate occurs at the thruster supply tube dome intersection on the dome inner surface—exactly the area that fractured during the demonstration unit test.

The CFD-derived high heat transfer coefficient results were presented to the Integrated Product Team and the SDACS prime contractor, and the results eventually helped to contribute to a redesign of the MTA dome region. The value added to the SDACS program in this instance was after the fact. If the CFD analysis had been performed in the initial design phase, the failure of the demonstration unit, and much time and expense, might have been avoided.

SUPERSONIC INVISCID APPLICATIONS: SM TAIL FIN LOADS AND SM-2 BLOCK IVA AIRFRAME AERODYNAMICS

To augment supersonic wind tunnel aerodynamic data for a complete missile airframe, the principal CFD tool used by ADSD is the Zonal Euler Solver (ZEUS).[5] ZEUS is a fast-running code that solves the inviscid fluid dynamic equations and is specialized for supersonic tactical missile configurations. The inviscid equations are a good approximation when the flow is supersonic everywhere because viscous effects do not affect the pressure distribution significantly. Experience has shown that this assumption is valid in the Mach number range from 1.5 to 5 using ideal gas and Mach 5 or more using real gas properties. The steady Euler equations can be solved numerically using a spatially marching scheme that starts from an axial plane near the nosetip of the missile and marches down the axis of the missile to the aft end. The ZEUS code uses a second-order upwind numerical scheme that advances the flow information from one axial plane to the next.

Procedures for obtaining a starting solution at the nosetip for the marching algorithm accommodate both sharp and blunt nose shapes. The nose-to-tail computation of a missile aerodynamic flow field is usually not completed in one axial marching sweep, but involves a series of stops and restarts for redefining the grid in areas of geometry changes or changing the bow shock treatment when the bow shock wave impinges on a wing or a control surface. A grid sensitivity study is essential for each new missile configuration.

To obtain skin friction or heating information, a postprocessing analytic boundary layer code, ZEUSBL,[6] uses the inviscid ZEUS solution at the boundary layer edge. Special procedures account for boundary layer transition, shockwave/boundary layer interaction, and fin/wing leading edges.

ZEUS is often used to generate aerodynamic coefficients over a variety of flight conditions, including Mach number, AoA, aerodynamic roll angle, and various control surface deflections. Automated rezoning and regridding enable ZEUS to loop through the flight conditions. Data can be generated in roll sweeps at fixed AoAs or AoA sweeps at fixed roll angles. If tail control effectiveness is needed, the ZEUS code is restarted at a missile axial station location just before the tail panels and then looped over the tail section for each control deflection. Marching from the nose to the base of a typical missile configuration requires on the order of 15 min of CPU time. This procedure was used to generate ZEUS predictions to compare with wind tunnel measurements for code verification.[7] Figure 9 shows some comparisons with wind tunnel data for the normal force, pitching moment, and rolling moment coefficients as well as the center-of-pressure location. The ZEUS-computed center-of-pressure location is typically aft of that measured in the wind tunnel because of the absence of the viscous effects in the ZEUS solution.

Validation efforts at APL have shown that ZEUS accurately predicts the wave drag, normal and side force, and roll, pitch, and yaw moments up to 15° AoA. However, these investigations have also shown limitations of the code. ZEUS is bounded to a lower-limit Mach number of 1.5, low AoAs, and small fin deflections because flow conditions outside these bounds result in local regions of subsonic flow that cannot be handled by the ZEUS formulation.

ZEUS is used as a "computational wind tunnel" to generate full airframe aerodynamic coefficient data or as a tool to supplement wind tunnel data for a wide variety of problems.[8,9] It is also effective in determining air loads on individual components. One such application[10] generated SM tail fin loads to support flutter analysis. The approach was to run ZEUS at flight conditions and tail settings that were suspected of inducing tail fin flutter. The predicted fin surface pressures were used to compute the spanwise force distribution and center of pressure. Figure 10 shows the ZEUS-predicted spanwise distribution of the center of pressure, along with the fin overall center of pressure derived from both wind tunnel tests and ZEUS results.

Another component study computed the loads and influence of a side-mounted seeker on SM-2 Block IVA airframe aerodynamics. The seeker aerodynamic increments were determined by differencing predictions with and without the seeker over a wide range of flight conditions. Figure 11a depicts the surface pressure distribution on the airframe with the seeker. Flow details, as exemplified in Fig. 11b showing the shock wave generated

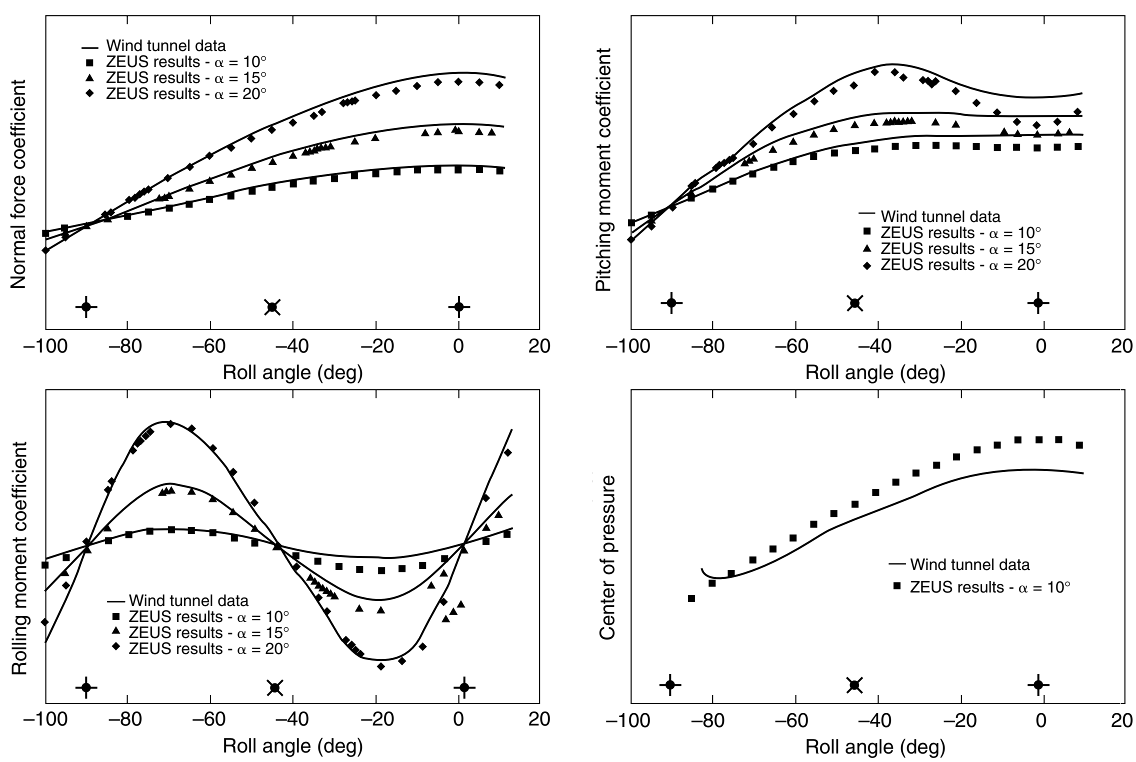


Figure 9. Comparison of ZEUS results with wind tunnel data (α = angle of attack; + and × represent missile orientation).

by the seeker, provide valuable insight on how the downstream flow affects other missile components.

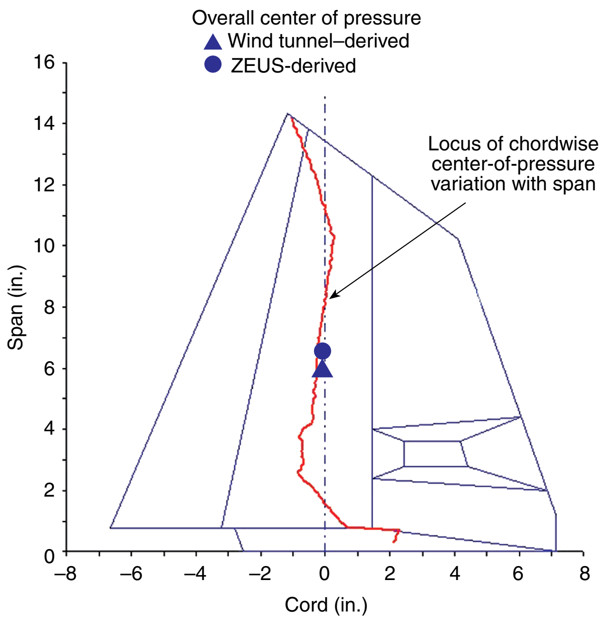


Figure 10. Tail control fin center-of-pressure location.

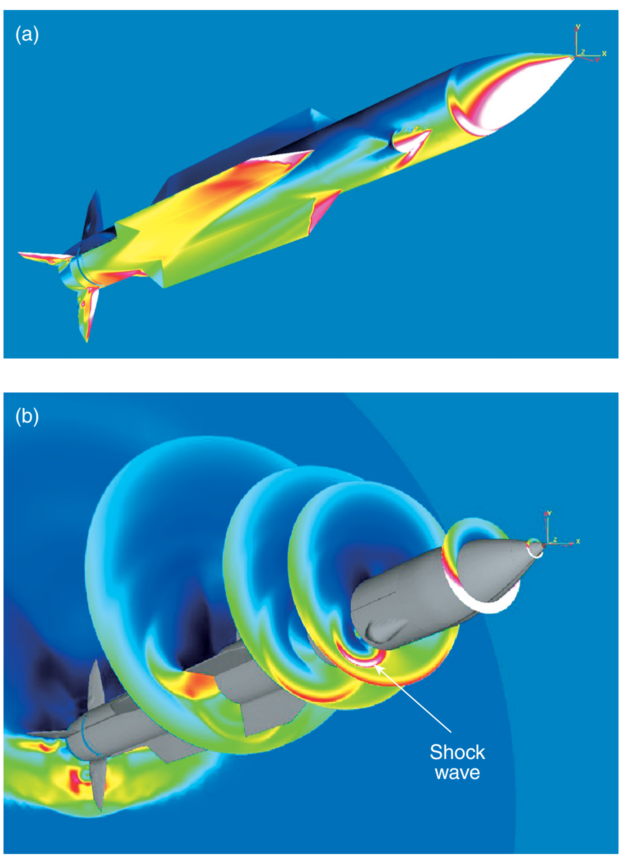


Figure 11. SM-2 Block IVA ZEUS CFD analysis can yield results such as the surface pressure distribution (a) and the flow field (b), both shown with a side-mounted seeker.

Over the last few years, reliance on ZEUS has grown, and it has been integrated with other techniques for generating full airframe aerodynamic coefficients for supersonic tactical missiles. Data conversion routines have been written to transfer results from ZEUS and other computational methods into a common analysis system. This facilitates a full hierarchy of capabilities to generate supersonic tactical missile aerodynamics, ranging from empirically based codes to ZEUS and finally to actual wind tunnel measurements in a seamless working environment. This coupling of techniques has enabled quick and thorough airframe analyses throughout all cycles of the missile development process.

SUBSONIC INVISCID APPLICATIONS: TACTICAL TOMAHAWK TAIL FIN DEPLOYMENT

Designed in the 1970s, Tomahawk is a highly versatile, deep-strike weapon system used to neutralize critical targets. To meet future missions, the Navy is developing an even more versatile missile. This next-generation Tactical Tomahawk (TACTOM) is being designed as a flexible battlefield weapon that is half the cost of a current Tomahawk. One aspect of the cost reduction is subassembly simplifications, including the number of tail control fins and the means to deploy the stowed fins once the missile is launched. In its role as Technical Advisor to the Navy, APL viewed the existing aerodynamic database as inadequate for designing the new tail fin deployment system. A more complete definition of the aerodynamic fin deployment loads (i.e., aerodynamic fin-folding moments) was required to ensure the robustness of the fin deployment system design. A fairly extensive database was needed; however, program schedule and funding levels initially ruled out the possibility of wind tunnel testing. CFD analysis was therefore recommended and a task accepted to develop results in a 3-month time frame.

Technical Approach

Actual deployment of the tail fins occurs over a short time, thereby making the flow computation unsteady in nature; the aerodynamic fin-folding moments are a function of time. Solution of the unsteady flow problem is a much more time-consuming and difficult computation than solution of the steady flow problem. Therefore, a widely excepted approximation to the computation of unsteady flow was applied, which is to treat the flow in a quasi-steady manner. This approach simply models the flow in the steady state at discrete points in time or, in the case of fin deployment, the steady-state flow is computed at specific fin deployment angles. The TACTOM with booster and fully deployed fins is depicted in Fig. 12a. The series of fin deployment angles

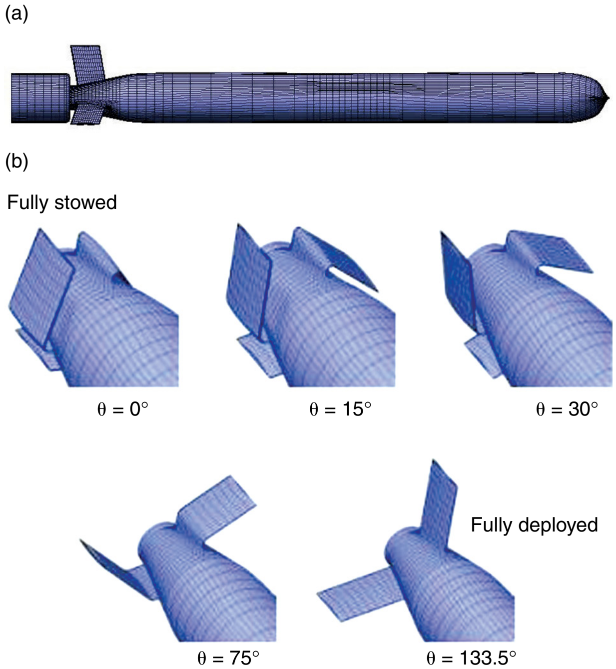


Figure 12. TACTOM airframe configuration: (a) side view with fully deployed tail fins, and (b) deployment of fins after launch (θ = fin deployment angle). (Booster is not displayed here but is included in the actual computations.)

(θ) for which computations were made are shown in Fig. 12b.

The aerodynamic fin-folding moments needed to be defined for a wide range of Mach numbers (M), AoAs (α), angles of sideslip (β), and fin deployment angles (θ). The flow conditions were in the low subsonic M regime and low-to-moderate α and β, with little or no regions of separated flow. The CFD methods that could be applied ranged from RANS to full potential panel codes. Taking into consideration the large number of required data points, coupled with the 3-month delivery time frame, a decision was made to use a lower-order full potential panel code. ADSD's experience and knowledge of the VSAERO[11,12] panel code made this the computational tool of choice to define the required aerodynamic database. The following section briefly discusses VSAERO and its underlying theoretical foundation.

Theory, Assumptions, and Validation

The VSAERO code computes the linearized potential flow external to a body surface. An integral boundary layer method can also be coupled with VSAERO to estimate the viscous boundary layer effects. For inviscid, incompressible, and irrotational flow, the potential function will satisfy the governing flow equation. The panel method solution is obtained by solving the governing equation as a set of equations and unknowns equal to the number of panels defining the body surface. Wake panels shedding off the trailing edges of the wings, tails, and body base model the vorticity shed into the flow. The wakes are allowed to relax or conform to a steady-state shape that satisfies the boundary condition across the wake panels. The integral boundary layer method can compute laminar and turbulent attached flow, boundary layer transition, separation points, and laminar boundary layer reattachment. Application of the integral boundary layer method adds a transpiration term in the boundary condition to account for the boundary layer displacement thickness that includes an estimate of the body surface skin friction. All solutions obtained for the TACTOM fin-folding moments include an estimation of the viscous boundary layer effect.

The validity of applying VSAERO to TACTOM fin-folding moment computations was not in question because the flow conditions to be examined fell mostly well within the applicable bounds of the VSAERO theory, and the code was successfully applied earlier to the TACTOM cruise configuration.[13] Figure 13 shows the VSAERO surface panel definition for this configuration.

VSAERO results were obtained for the cruise configuration for a comprehensive range of M, α, β, and wing incident angles. Figure 14 is one of many example comparisons made between VSAERO results and wind tunnel data. The results presented are the lift coefficient (C_L) and pitching moment coefficient (C_m) as a function of the lowest and highest M of interest. The computed force and moment coefficients obtained agreed well with wind tunnel data at most flow conditions.

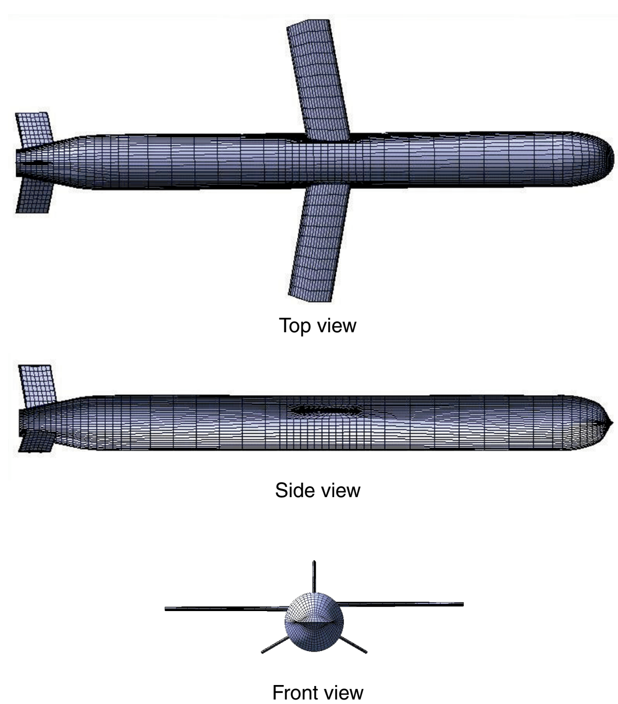


Figure 13. TACTOM cruise configuration geometry and paneling for VSAERO computations.

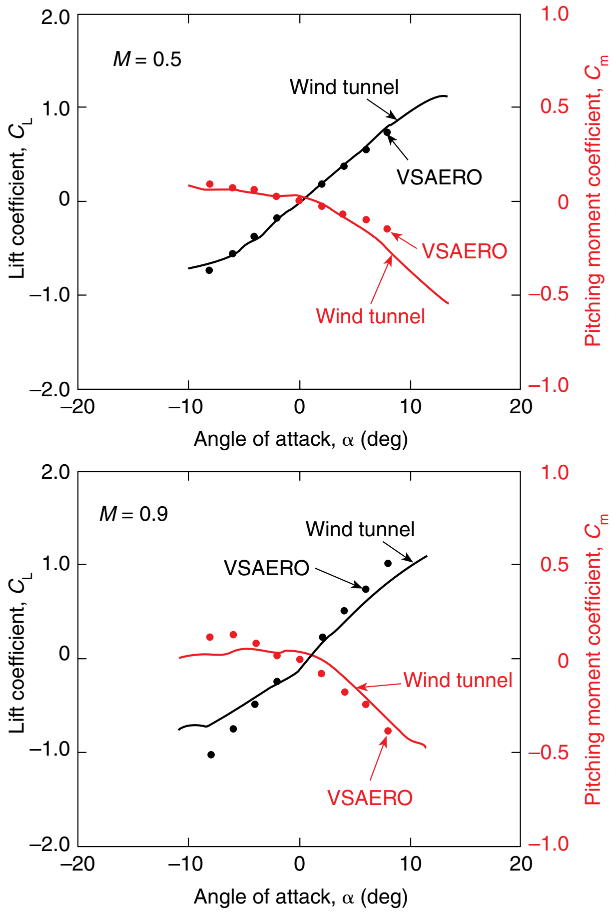


Figure 14. Sample comparison of TACTOM cruise configuration VSAERO results with wind tunnel data for a wing incident angle of −2° and sideslip angle of 5°.

For the low subsonic M, the agreement is excellent. However, as M increases, the agreement between VSAERO and the wind tunnel data degrades because VSAERO assumes incompressible flow. However, as M increases toward unity, fluid compressibility becomes an important physical aspect. The code attempts to account for compressibility with a theoretical correction, but the correction is also limited in its accuracy at the high subsonic M. The significant point of this comparison and validation is that VSAERO is accurate in the low-to-medium subsonic M regime but has reduced accuracy in the high subsonic M regime. Therefore, care is taken in applying this code within the applicable bounds of its theoretical foundation.

The cruise configuration work demonstrated the validity of applying VSAERO to the TACTOM and provided the basis for extending its application to the TACTOM tail fin deployment problem.

Tail Deployment CFD Results

Steady-state tail deployment computations were performed for each θ as a function of M, α, and β. There were 17 different flow conditions (i.e., M, α, and β) and 5 fin-folding angles for a total of 85 different cases. Results included the fin-folding moments as a function of θ, missile force and moment coefficients, and surface pressure coefficients (C_p). Because the fin-folding moments are of most interest, the discussion here concentrates on these results.

The computational resources required for VSAERO are significantly less than those required for a typical RANS computation. Computational run times varied from 1100 to 2000 s of CPU time, or approximately 2 to 3 h of real time for each computation. The multiple processor computers available at the Navy's Hydrodynamic/Hydroacoustic Technology Center, Carderock Division, in Bethesda, MD, enabled the computations to be performed over a 3-week period.

A sample of the VSAERO results is found in Fig. 15, which clearly shows the nonlinear behavior of the fin-folding moment as a function of θ. The behavior of the fin-folding moments for the individual tail fins as a function of θ is quite different for all three tail fins acting as an assisting or retarding moment, depending on the flow condition or the relative position of the free-stream velocity vector to each tail fin.

The computational fin-folding moments were significantly larger and with greater variation than the existing design aerodynamic database results. These unexpected results caused much consternation within

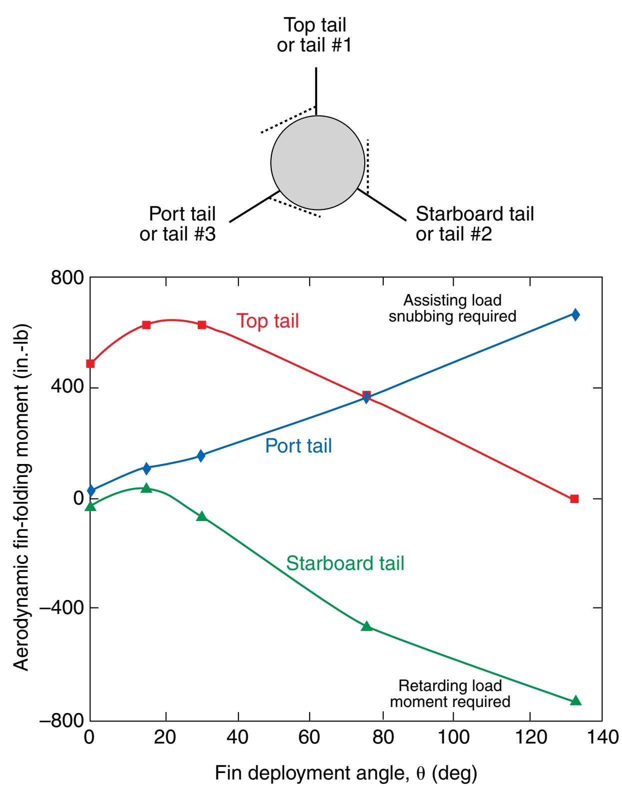


Figure 15. VSAERO results for aerodynamic fin-folding moments at $M = 0.30$, $\alpha = 15°$, and $\beta = 0°$.

the TACTOM Program. Simply put, dynamic and structural analysis using CFD results indicated that the fin deployment system might not completely deploy the tail fins in all instances and led to the decision to redesign the system. To support the redesign, a decision was made to conduct a wind tunnel test to obtain a comprehensive aerodynamic database. The wind tunnel test provided an opportunity to validate the CFD results.

The test was conducted in the fall of 2000 and was documented by C. L. Ratliff.[14] It covered a larger number of flow and geometric conditions than were included in the CFD results; several of the test conditions were very close to or exactly matched the flow conditions used in the computations. Figure 16 shows the comparison between wind tunnel data and computations for one of the matching conditions.

The comparison is generally good. The data do not match exactly point for point, nor are the magnitudes matched in all instances. However, all the trends as a function of θ for each separate tail fin-folding moment and the total fin-folding moment are captured quite well. It was known and expected that the results would not match exactly for at least two reasons: (1) some viscous effects are not fully captured by the computations, and (2) the theoretical basis of the computations would result in the underprediction of low pressure and overprediction of high pressure (i.e., the computations were expected to be conservative and were indeed shown as such). These comparisons validated the application of VSAERO to the TACTOM fin-folding moment computations. The computational results and wind tunnel data were used together to define the fin-folding moment loads in a redesign of the TACTOM fin deployment system.

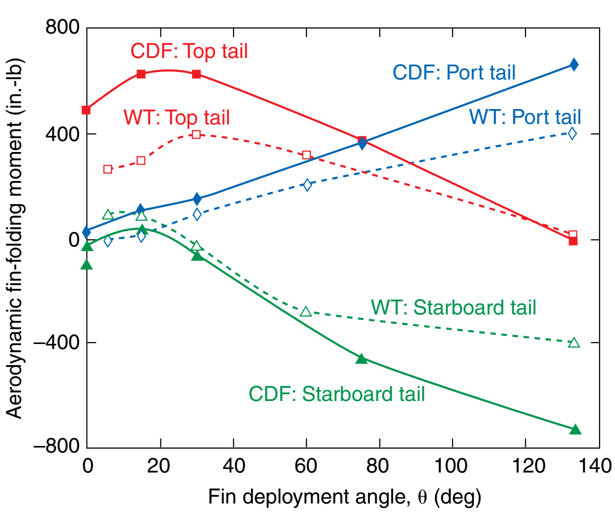


Figure 16. Comparison of VSAERO results with wind tunnel (WT) test data at $M = 0.30$, $\alpha = 15°$, and $\beta = 0°$.

CONCLUSION

This article provides excellent examples of how and where CFD has added value to the missile airframe and component development process. The SM Program has significantly benefited from CFD in the design of the IR dome cooling system, SDACS failure analysis, computation of full airframe aerodynamic databases, and tail load computations for flutter analysis. The TACTOM Program has also benefited from CFD through development of a fin-folding loads database and contributory analysis for fin deployment system redesign. These examples demonstrate contributions to the missile development process, from exploratory wind tunnel testing through in-development and in-service flight test phases. Although only a few examples are described here, CFD has impacted, contributed to, and added value to the missile development and risk identification process of several other programs. CFD has become and will continue to be an integral part of the missile design process.

The current trend in missile development is to evolve today's hardware and to design new hardware into multirole or multimission weapons systems. This trend calls for missiles to operate at even faster speeds, fly at higher altitudes, and be more maneuverable. These operational requirements demand analysis and evaluation over a much wider range and more severe aerodynamic and aerothermodynamic environments. Requirements such as these are significantly challenging the current state of the art of CFD while giving CFD analyses an even larger role in the design and engineering process.

CFD software must be continuously improved and updated. Several new CFD technologies are being considered to enhance capabilities, including large eddy simulations, adaptive and moving grid techniques, fluid-structural interaction methods, and high AoA flows.

Program requirements to rapidly perform CFD analyses continue. Not only is there a need for new and better CFD tools, but also for faster and more multiprocessor computers. The use of a combination of external and internal computer resources can fulfill this need. Externally, APL must continue to use DoD computer resources and promote expansion of those resources. Internally, the number of dedicated computers and processors will need to increase significantly to meet both program and computational requirements. By fulfilling these CFD needs, the Laboratory can continue to serve its Navy sponsors.

REFERENCES

[1]*The General Aerodynamic Simulation Program—GASP Version 3*, Aerosoft, Inc., Blacksburg, VA (May 1996).

[2]Chien, K. Y., "Predictions of Channel and Boundary Layer Flows with a Low Reynolds Number Turbulence Model," *AIAA J.* **20**, 33–38 (1982).

[3]Rogér, R. P., and Clark, M. E., *Computational Fluid Dynamics Analysis of the STANDARD Missile-3 SDACS Demonstration Unit 1*, A1C-00-098, JHU/APL, Laurel, MD (21 Jun 2000).

[4]*CFD-ACE+: Getting Started Guide*, (FI) Research Corp., Huntsville, AL (Oct 1998).

[5]Wardlaw, A. B. Jr., and Davis, S. F., *A Second Order Godunov Method for Supersonic Tactical Missiles*, NSWC TR 86-506, Naval Surface Weapons Center, Silver Spring, MD (Dec 1986).

[6]Wardlaw, A. B. Jr., and Baltakis, F. P., "An Integral Boundary Layer Procedure for Tactical Missiles," AIAA Paper 92-1026, in *Proc. 1992 Aerospace Design Conf.*, Irvine, CA (3–6 Feb 1992).

[7]Frostbutter, D. A., and Parthasarathy, K. N., *Improved Computational Fluid Dynamics Capabilities for Missile Flowfields*, AM-94-D0013, JHU/APL, Laurel, MD (Jun 1994).

[8]Yungkurth, C. B., and Frostbutter, D. A., *A ZEUS Aerodynamic Model of the Advanced Air-to-Air Missile (AAAM)*, AM-91-D0048, JHU/APL, Laurel, MD (6 Aug 1991).

[9]Heisler, R. R., and Frostbutter, D. A., *Three Dimensional Aerodynamic Model for the MM5*, A1D-1-98U-055, JHU/APL, Laurel, MD (Jun 1998).

[10]Frostbutter, D. A., *SM-2 Aerodynamics—Validation of the Zonal EUler Solver (ZEUS) Code, with Emphasis on the Calculation of Local Surface Pressure Distribution and Forces on the Tail Control Fins*, AM-91-D0045, JHU/APL, Laurel, MD (19 Jul 1991).

[11]Maskew, B., *Program VSAERO Theory Documentation*, NASA CR-4023, NASA Ames Research Center, Moffett Field, CA (Sep 1987).

[12]Nathman, J. K., *VSAERO—A Computer Program for Calculating the Nonlinear Aerodynamic Characteristics of Arbitrary Configurations*, User's Manual Version 6.1, Analytical Methods, Inc., Redmond, WA (1999).

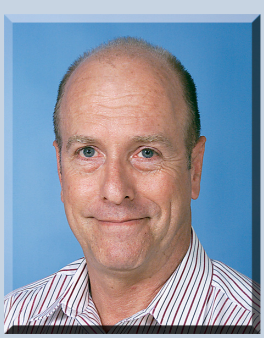
[13]McGrath, B. E., and Roberston, M. C., *VSAERO Computations for a Tactical Tomahawk Configuration with a Comparison to Wind Tunnel Data*, A1D-3-99U-033, JHU/APL, Laurel, MD (Jun 1999).

[14]Ratliff, C. L., *Posttest Summary of the 0.34356-Scale Tactical Tomahawk Rapid Prototype Tri-Tail Deployment Low Speed Wind Tunnel Test*, A1C-01-003, JHU/APL, Laurel, MD (Jan 2001).

THE AUTHORS

DAVID A. FROSTBUTTER is a member of APL's Principal Professional Staff and Supervisor of the Applied and Computational Fluid Dynamics Section in ADSD's Mechanical and Aeronautical Engineering Group. He received an A.A. in engineering from Prince George's Community College in 1969 and a B.S. in aerospace engineering from the University of Maryland in 1978. From 1970 to 1973 Mr. Frostbutter was on active duty with the Naval Reserve, then joined APL in 1973. He has been instrumental in leading the application of computational fluid dynamics to predict and evaluate the aerodynamic performance of tactical missiles. He is currently leading the effort to develop methods to integrate CFD computations with flight telemetry data, which will allow assessment of vehicle flight loads. He uses his piloting background to help understand problems in aerodynamics and in the FAA Safe Flight 21 new systems development tasks. He is a member of the AIAA and a certified flight instructor. His e-mail address is david.frostbutter@jhuapl.edu.

BRIAN E. McGRATH is a member of APL's Senior Professional Staff in ADSD's Mechanical and Aeronautical Engineering Group. He received B.S. and M.S. degrees in aerospace and ocean engineering in 1982 and 1985, respectively, both from Virginia Tech. Previously he worked for the Martin Marietta Corporation and NASA's Langley Research Center. He is the inventor and U.S. patent holder of the concept for serrated planform lifting surfaces. He joined APL in 1996 with an extensive background in experimental and computational aerodynamics within the subsonic and supersonic flight regimes. Mr. McGrath is primarily involved with the computational analysis and evaluation of aerodynamics and hydrodynamics of various missiles systems and underwater vehicles. He is also the lead aerodynamicist for APL's Aerospace Nuclear Safety Program. He is a senior member of the AIAA. His e-mail address is brian.mcgrath@jhuapl.edu.



ROBERT P. ROGÉR is a member of APL's Principal Professional Staff. He received the B.S.-Physics degree from Loyola University of the South in 1964 and a Ph.D. in physics from Louisiana State University (LSU) in 1969. For 15 years Dr. Rogér was a professor of physics and taught graduate and undergraduate physics at Northwestern State University of Louisiana and LSU. From 1984 to 1997 he worked for Lockheed Missiles and Space Company and Teledyne Brown Engineering. In 1997 he joined APL and currently works in ADSD's Mechanical and Aeronautical Engineering Group. His specialty is fluid mechanics and in particular computational fluid dynamics techniques for both compressible and incompressible laminar and turbulent fluid flows. Dr. Rogér has contributed to many refereed journal articles and unrefereed conference papers and has made numerous presentations at professional meetings. He is an active member of APS and AIAA. His e-mail address is robert.roger@jhuapl.edu.