EFFECTIVENESS OF CFD FOR AIRCRAFT DESIGN: *STATUS AND PROSPECTS*

Pradeep Raj
Professor, Kevin T. Crofton Department of Aerospace and Ocean Engineering
Virginia Polytechnic Institute and State University (Virginia Tech), Blacksburg, VA, USA

Abstract

Developing technologically superior aircraft at an affordable cost has been, and will continue to be, an overarching challenge for the aircraft industry. To tackle this challenge, the aircraft design community explored numerous approaches. Simulation based design (SBD), a highly promising approach, is aimed at producing virtual rather than physical prototypes which can faithfully represent the functional and operational characteristics of the airplane to be built. SBD is enabled by the ever increasing power of digital computers combined with increasingly sophisticated modeling and simulation technologies. Computational fluid dynamics (CFD) is the preferred simulation technology to produce aerodynamic data, such as forces and moments needed to assess airplane flight performance and handling qualities. SBD can successfully achieve its aim only if computational data are credible and available in a timely and cost-effective manner. In spite of phenomenal progress in CFD capabilities over the past six decades, today's workhorse methods remain deficient in producing credible data, especially for complex flows dominated by flow separation or free vortices. Examples are presented in this paper to substantiate this assertion. Prospects for CFD that is fully effective in meeting aircraft design needs are discussed.

Keywords: computational fluid dynamics; effectiveness of computational simulation; aircraft design

1. Introduction

The first manned, powered, controlled flight of the Wright brothers on December 17, 1903, lasted only 12 seconds but forever altered the course of human history. Airplanes have since *shrank the globe* by offering a mode of world-wide transportation that routinely moves thousands of passengers over thousands of miles every day. In addition, airplanes have emerged as an integral component of any credible national defense strategy.

Throughout the 20th century, both military and commercial customers demanded "higher, faster, farther" aircraft and the airplane industry responded by developing increasingly sophisticated aircraft which provided impressive improvements in flight performance However, airplane costs also escalated dramatically as illustrated in Figure 1 for US combat aircraft [1]. The trends for the commercial aircraft are quite similar. For example, the unit price of Boeing 707 in 1955 was 4.3 million US dollars [2] whereas the average unit price of Boeing 777-9 in 2022 was nearly 442 million US dollars [3]. As a consequence, aircraft developers continue to face a daunting challenge: *deliver improved performance at an affordable cost*.

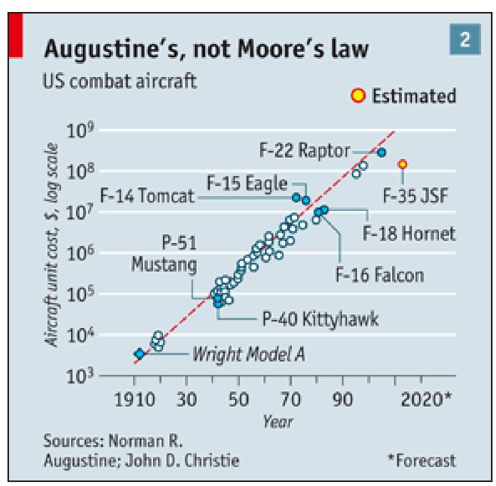

Figure 1 - US combat aircraft unit cost has grown 4X every 10 years [1]

It has been widely recognized that reducing the total life cycle cost (LCC)—conception to disposal—holds the key for successfully tackling the affordability challenge. The curve labeled '*Cost*' in Figure 2 indicates that design & test activities contribute about 20% to the LCC, production (or manufacturing) about 30%, and operations & support (O&S) functions about 50%. It stands to reason that LCC can be best reduced by aggressively devising and implementing means of reducing O&S and production costs. However, the curve labeled '*Impact On Cost*' reveals that decisions made by the design team in the early stages of design have a disproportionately large impact on LCC because up to 70% of the LCC is committed even before the preliminary design phase begins! Therefore, one could argue that the most effective strategy for successfully tackling the affordability challenge is to provide the design team with means of making *quality* decisions in a timely and cost-effective manner during the early stages of design. The design team can then carefully consider ways of reducing production, operations, and maintenance costs in the early stages of design. The simulation based design (SBD) paradigm offers one of the most promising approaches to execute the strategy for tackling the affordability challenge [4, 5, 6]. For timeliness and cost-effectiveness, SBD relies heavily on computational simulations as the primary means of data and information. For example, computational fluid dynamics (CFD) methods are preferred over wind-tunnel testing to generate the required aerodynamic data. Conducting wind-tunnel testing that minimizes adverse scale effects and wall interference effects significantly increases the time and cost of generating data. It cannot be overemphasized that data must be credible, i.e., *data must faithfully replicate reality*.

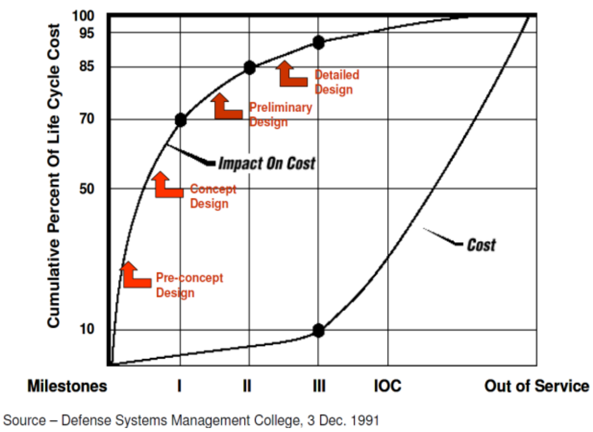

Figure 2 - Up to 70% of LCC is committed in the conceptual design stage.

The remainder of this paper is organized in five sections. Section 2 provides a brief overview of the traditional aircraft design process and the need to transition to SBD. This is followed by a brief overview of CFD and its capabilities in Section 3. The current status of CFD effectiveness is the topic of Section 4. We then highlight the prospects for a fully effective CFD (*credible data, on time, on budget*) in Section 5. A few concluding remarks in Section 6 complete the paper.

2. Aircraft Design Process: *Traditional to Modern*

The very early years of aircraft design were largely dominated by a *fly-fix-fly* approach. Its trial and error nature was inherently risky and inefficient for achieving performance targets. In the 1920s and 1930s, the fly-fix-fly approach quickly evolved into a *design-test-build* approach as a direct consequence of rapid advances in aeronautical sciences. The attendant increase in aeronautical engineering knowledge led to the development of analysis and ground test methods that could be used to systematically guide aircraft design with a much better probability of achieving the performance targets. The process that evolved to implement this approach is termed "traditional design process" in this paper. Key features of this process and its major deficiencies are outlined in Section 2.1. For rectifying these deficiencies, innovative paradigms evolved as discussed in Section 2.2. Implementation of SBD through multidisciplinary design optimization (MDO) frameworks is also discussed in Section 2.2.

2.1 Traditional Design Process

The traditional design process essentially evolved to keep pace with the rapidly changing world of aircraft development. With each passing decade of the 20th century, aircraft became ever more complex in response to customer demand of ever improved performance (higher, faster, farther). The resulting design process is schematically shown in Figure 3. The process is typically carried out in three stages (or phases): conceptual, preliminary and production or detail.

In each stage, the myriad of activities broadly falls into two categories: synthesis and analysis. Synthesis covers defining and altering concepts to meet customer requirements; analysis encompasses methods, tools and expertise to produce data that is used for evaluating candidate concepts. Synthesis is the responsibility of the design group, and analysis is performed by specialists in various disciplines such as aerodynamics, structures, controls, avionics, etc.

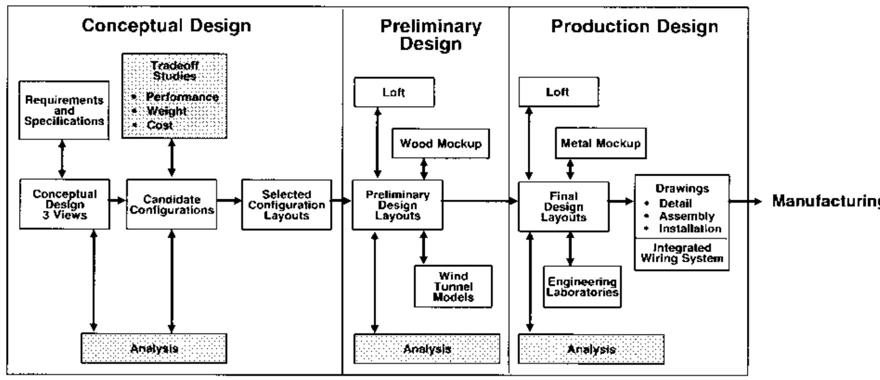

Figure 3 - Schematic of the traditional aircraft design process (courtesy of Sam Smyth, experienced aircraft designer at Lockheed, ca 1988)

As aircraft became more complex, it took a larger number of engineers in all disciplinary specialties to support the design groups. Over time, the disciplinary specialists got organized into separate departments, each with its own "lingo" and "standards." With ever expanding size and increasing specialization of the departments, effective communication and coordination became increasingly more difficult. "Throw it over the wall" became the norm. Interactions between departments and the design group as well as interactions among the disciplinary specialists within the department were severely hampered. To further compound the problem, design groups got segregated from manufacturing, marketing, and support organizations. This created a "stovepipe" environment in which design groups became more focused on their perceived priority of designing the "best" aircraft to meet customer requirements of high levels of performance and less on production or O&S costs. As a result, LCC was a *fall out* of the design effort—not something consciously targeted by the design group as a customer requirement.

In a traditional design process, resources (human, material and financial) allocated to the conceptual stage of design are typically quite small, of the order of 1 to 2%, as shown in Figure 2. Also, the design group is expected to deliver a competitive "winning" design in a matter of few weeks to a few months. After all, time is money [7]. This situation forces engineers to use tools that are fast, easy to use, and do not require a large amount of resources. Most of the decisions are, therefore, based on data from relatively crude and simplistic analyses and tests. Also, manufacturing, operation and support functions are incorporated into decision making in a rather superficial way, if at all. This particular traditional conceptual design approach loses sight of the crucial fact that *decisions made in the early stages essentially define the fundamental architecture of the airplane* and largely dictate how it will have to be manufactured, operated and supported—and hence determine the LCC.

It is worth noting that the traditional design process is not conducive to developing optimal configurations although the customer demand of aircraft with ever improving performance has provided a strong impetus to optimizing configurations. In the stovepipe environment of the traditional design process, each discipline separately analyses candidate concepts and provides its assessment, along with any recommended changes, directly to the design group. Many times, data from different disciplines place conflicting demands on the direction in which a design should be altered. Additional time and resources are then required to reconcile the conflicting demands. Even though knowledge about designs increases with time as more data from tests and analyses flow in, designers are constantly faced with a rather poor choice: either modify the design to achieve superior performance at the expense of more time and cost or meet cost and schedule targets and accept lower levels of performance.

Due to the *sequential* nature of the traditional design process, which is a direct result of the stovepipe environment, completing one design cycle can take a long time—many weeks and months, not a few days—for modern airplanes with high degree of complexity. As a consequence, the number of cycles that can be conducted within the schedule and cost constraints of a typical aircraft development project is too small to explore a wide range of alternative candidate concepts. Clearly, chances are small that a design group can deliver an “optimal” configuration.

2.2 Modern Design Paradigms: *CE, IPPD, and SBD*

Once the major deficiencies of the traditional design process were recognized, namely, its sequential nature resulting from the stovepipe environment and meager resources allocated in early stages, the Concurrent Engineering (CE) paradigm was introduced in the 1980s [8] as a means of rectifying these deficiencies. CE focuses on replacing the traditional design process with one characterized by simultaneous or concurrent consideration of *all* aspects of product development. It relies on considering all requirements and constraints *from the start* rather than altering a design in its later stages to facilitate manufacturing or accommodate product support needs.

2.2.1 IPPD

The US Department of Defense soon followed with an integrated product and process development (IPPD) concept to reap the potential benefits of CE, namely, reduced development time and cost. IPPD simultaneously integrates all essential acquisition activities through the use of multidisciplinary teams to optimize design, manufacturing, and supportability processes. Proper trade-offs can therefore be made in *early stages of design* and the need for design changes later on considerably reduced. Design practices based on IPPD are contrasted with the traditional ones in Figure 4.

IPPD is characterized by the following key attributes:

- Closer relationship with the customer and constant communication to better understand customer requirements.
- Integrated Product Teams (IPTs) for a more complete understanding of requirements; a broader and more balanced discussion of issues and alternatives; and simultaneous design of product and processes.
- Design for X (DFX) methodologies, where X includes manufacturability, producibility, testability, reliability, maintainability, safety, quality, cost, etc., to develop proper guidelines to improve product design.
- Integrated design and analysis tools using a digital product model to capture and refine product and process design data.
- Integrated design automation tools to streamline the design process and assure understanding of design intent.

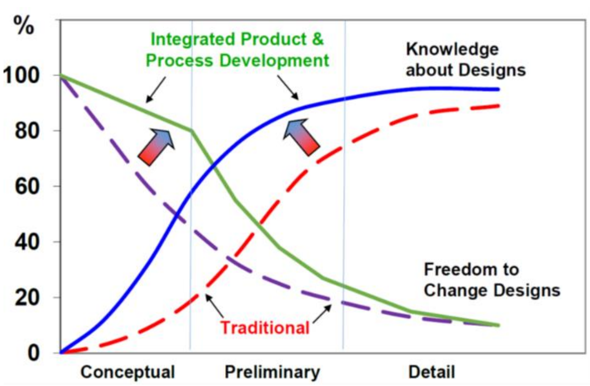

Figure 4 - Impact of IPPD on traditional aircraft design practices

- Extensive use of physics-based analysis methods and simulation tools for improved product performance with fewer design/build/test iterations.

At the most basic level, IPPD mimics the earlier days of airplane development characterized by close interaction and effective communication between (and within) small groups involved in design, analysis, test, and manufacturing functions, all working together in the same room. By harvesting the tremendous advances in information technology and computer-aided engineering, IPPD aims to foster closer collaboration and more effective communication among a much larger number of personnel supporting today's design projects.

2.2.2 SBD

The IPPD concept is implemented for vehicle design using the simulation based design (SBD) process. SBD derives time and cost savings by using simulations as the primary means of supporting design decisions. It employs integrated multi-disciplinary models, computational simulations and simulators to guide the development of a virtual prototype (VP) with a degree of functional realism comparable to a physical prototype. A traditional design effort, in contrast, produces physical prototypes to validate the functional characteristics. VPs are much more cost-effective for assessing and validating aircraft performance compared to physical prototypes. SBD enables development of VPs that concurrently incorporate inputs from the engineering teams about the aircraft configuration aspects, the manufacturing team about production aspects, the maintainers about logistical aspects, and the operators about training aspects.

Once a virtual prototype is finalized, it can serve many needs. For example, a VP can be used in a synthetic environment (that mimics the actual environment) to assess system-level capabilities. When the aircraft system is fielded, a VP can be used for training and indoctrination which can offer significant savings in time and resources by reducing or eliminating physical experimentation while overcoming safety, environmental and security constraints associated with live testing.

A Multidisciplinary Design Optimization (MDO) methodology is best suited for executing the SBD process by integrating computational simulation modules of disciplinary specialties including engineering, manufacturing, logistics, cost, etc., along with computational optimization modules. MDO is a formal methodology which facilitates concurrent accounting of the mutual interactions of multiple disciplines, such as aerodynamics, structures, controls, propulsion, and DFX (design for X) methods. A representative N^3 diagram in Fig. 5 illustrates one complete MDO cycle. Multiple cycles are needed to achieve an optimum configuration. Note that the disciplines are arranged in a sequence that maximizes 'pass-forward' data relationships and minimizes 'pass-back' ones. This greatly enhances the efficiency of process execution. In addition, each discipline can select a level of computational modeling that is most appropriate for the problem at hand. MDO can expeditiously provide design teams with data needed to make more informed decisions *early* and thereby alleviate the most serious shortcomings of the traditional design process. Design teams can then devote more time and effort to considering a

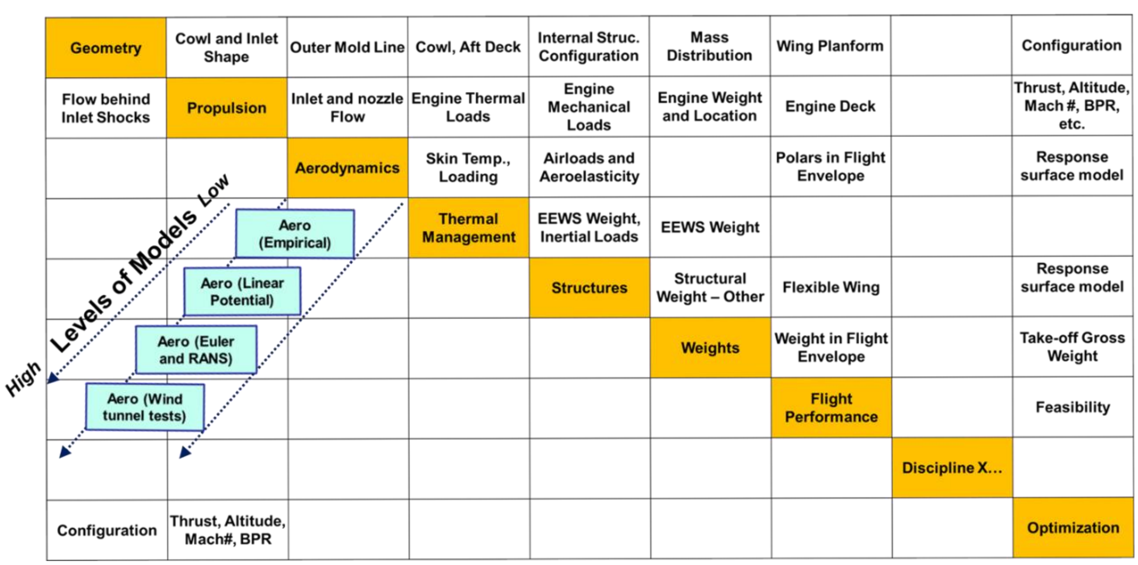

Figure 5 - A representative N^3 diagram illustrating the organization of multiple disciplines for one complete cycle of the MDO process

broader set of options with attendant improvements in quality and productivity [9]. It goes without saying that CFD is the primary means of aerodynamic simulations to support MDO. Capabilities and limitations of today's CFD are discussed in Section 3. However, before closing this section, we emphasize that the integrated multidisciplinary design process does not in any fundamental way differ from the traditional design process in terms of the activities that need to be carried out, the *difference is how*. Also, beware that such an integrated process must not be construed as an "autonomous" or "automatic" design process; it cannot substitute for human creativity and unique synthesis ability.

3. CFD Today

CFD is a sub-discipline of the Fluid Dynamics discipline which is a branch of applied science that is concerned with the movement of fluids. The purpose of CFD is to simulate the flow field around a body moving through fluid. This purpose is accomplished by numerically solving the governing fluid dynamic equations in a specified domain surrounding a virtual model of the body using digital computers. CFD is a relatively recent addition to the two other longstanding sub-disciplines of fluid dynamics, namely, analytical fluid dynamics (AFD) and experimental fluid dynamics (EFD). Synergistic use of all three is now recognized as essential for a comprehensive understanding of fluid dynamic phenomena. We highlight the four key components of CFD in Section 3.1 and we discuss capabilities of four levels of today's CFD in Section 3.2.

3.1 Four Key Components of CFD

Figure 6 shows the four key components of CFD. The governing equations are mathematical formulations of fluid flow in a domain through which the fluid moves. These are typically a set of partial differential equations (PDEs). One chooses a numerical model from a variety of available models, each with its own advantages and drawbacks, to convert the set of PDEs into a set of difference equations in a discretized computational domain. For solving the difference equations, one develops an algorithm and produces a computer program (also commonly referred to as software or code) that can be run on a digital computing platform. Many different types of algorithms have been developed to solve the difference equations and different algorithms produce different computer programs. The output of the computer program is a large dataset that contains values of a set of flow variables, such as density, pressure, velocity components, etc., at all of the discrete points in the computational domain. Utility software is used to process the dataset to generate quantities of engineering interest including forces, moments, pressures, and temperatures on the body. Flow simulations may be steady or unsteady depending on the required data.

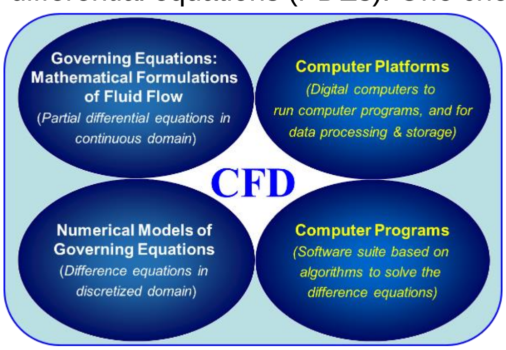

Figure 6 - Key components of CFD

3.2 Four Levels of CFD Capabilities

Today, one can choose from a wide variety of CFD methods with varying ranges of applicability (in terms of speed and attitude angle) to support aircraft design needs. The methods can be broadly categorized into four levels shown in Figure 7. This categorization is principally based on (i) the underlying governing equations each representing different levels of approximations of the Navier-Stokes (N-S) equations; and (ii) the timeframe (shown in parentheses) when the methods of that level began to make inroads into design. Note that Figure 7 also shows a few additional methods beyond the Level IV, namely, Unsteady RANS (URANS), Detached Eddy Simulation (DES), Large-Eddy Simulation (LES) and Direct Numerical Simulation (DNS). They all offer more sophisticated representations of flow physics than RANS, but they are not yet in widespread use. Figure 8 depicts the relationship of the governing equations for each of the four levels to the N-S equations.

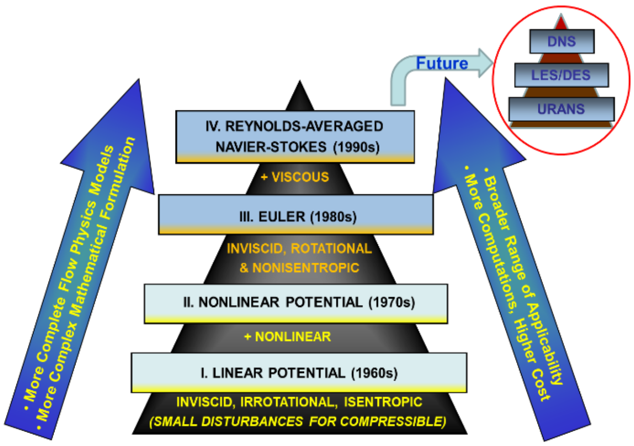

Figure 7 - Four levels of CFD

It has long been known that methods based on the Navier-Stokes equations, can simulate nearly all flow phenomena of interest for which the continuum assumption is valid. (The Boltzmann equations based on the kinetic theory of gases are needed to model molecular flows; the related numerical methods are not covered here.) However, adequate computer power and efficient numerical algorithms were not available until the 1990s for routine application of the Level IV RANS methods to aircraft design. Prior to that, the lower level methods based on simplified models of physics were developed and used. We highlight the basic features of each of the four level of CFD methods and their capabilities in the following subsections.

3.2.1 Linear Potential Methods (Level I)

The linear potential methods represent the lowest level, and the simplest, inviscid approximation to the N-S equations. The governing Prandtl-Glauert or Laplace equations are linear second-order partial differential equations (PDEs). These equations, along with the boundary conditions, are cast in a surface-integral form using Green's theorem. The solution is constructed by discretizing the geometry into small elements and assigning a singularity (sources, doublets, or vortex filaments) to each element. The singularity strengths are determined by satisfying the no-normal-flow condition at a control point on each element.

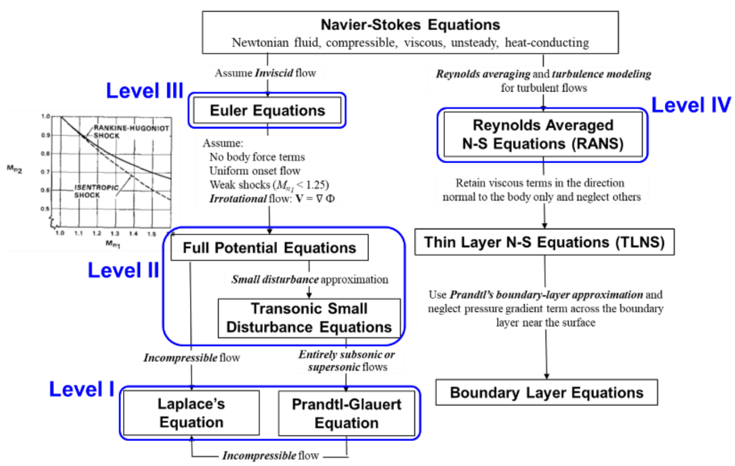

Figure 8 - N-S Approximations for the four levels of CFD

Depending upon the approximations used in surface discretization (mean surface or actual surface) and the type and functional form of singularities (constant source, constant doublet, linear doublet, etc.), codes with different characteristics [10] can be developed. To meet the aerodynamic data needs of the aeroelastic and flutter disciplines, versions of the doublet-lattice method are the codes of choice. Their computational times range from a few seconds to a few minutes on modern workstations. However, user expertise and experience are crucial to ensuring proper interpretation of the solutions.

The simplest codes, widely known as vortex-lattice methods (VLMs), employ mean-surface representation of geometry and vortex filaments as singularities. The VORLAX code [11] is a representative example. When the actual surface geometry is discretized, the methods are commonly called surface panel methods, or simply panel methods. Low-order singularity distributions, constant on each element, have been employed in QUADPAN [12] and higher-order ones, linear or quadratic, in PANAIR [13].

The simplified physical and mathematical formulation of the linear potential methods inherently restricts their validity to purely subsonic and supersonic attached flows. In spite of these restrictions, the codes are quite extensively used in design efforts due to their ease of use, computational efficiency, and relatively high level of confidence built upon years of use. An experienced user can set up a computational model in a matter of hours even for relatively complex configurations like a complete aircraft. Figure 9 shows a QUADPAN simulation of the C_L-C_m characteristics of the P-3 Orion aircraft with and without a rotodome for airborne early warning and control [14].

The VLMs and panel methods generally provide good estimates of forces, moments and distributed airloads for steady flight conditions as long as the flow is attached. The data form the basis for performance and weight estimations in the early stages of design. Some of the codes also offer a design option that can be used to determine geometric characteristics (like twist and camber of a wing) for a prescribed set of aerodynamic parameters.

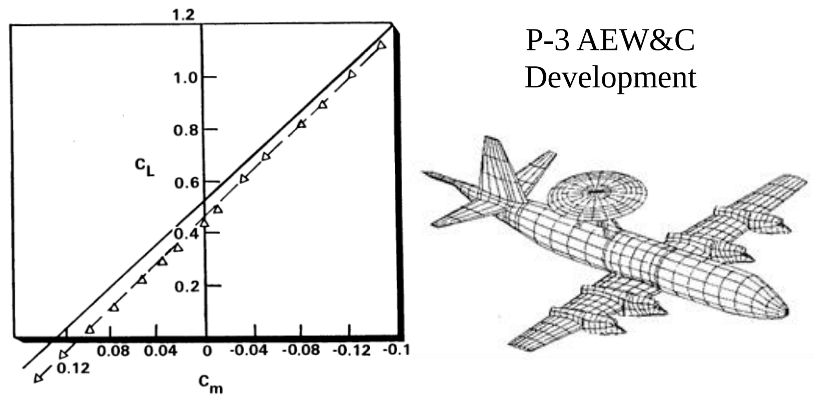

Figure 9 - QUADPAN Model (right) and comparison of computed C_L-C_m results of P-3 with rotodome (dashed line with symbols) and without (solid line)

The linear potential codes were first introduced into the aircraft design environment in the 1960s and the entire class of codes reached a high level of maturity in the early ‘80s. Relatively small amount of effort is presently going into research and development of this level of CFD codes with the possible exception of the oscillatory aerodynamic codes [15, 16].

3.2.2 Nonlinear Potential Methods (Level II)

The nonlinear potential methods are based on either transonic small disturbance (TSD), also known as transonic small perturbation (TSP), equations or full-potential equations (FPE). Both are nonlinear second order PDEs. Their ability to model transonic flows with shocks is the most significant benefit over the Level I linear potential methods. However, this benefit comes at the expense of added complexity stemming from the need to resort to a field approach to solve the nonlinear PDEs. The field approach requires that a region surrounding a given configuration be divided into small elementary volumes; it is no longer sufficient to just divide the surface.

Considerable progress was made in the 1970s towards the development of practical transonic-flow analysis methods [17]. The progress had its genesis in the landmark paper of Murman and Cole [18]. The TSP code of Boppe [19] and the FLO-series of FPE codes of Jameson and Caughey [20] made major inroads into aircraft design environment in the 1970s. In practice, TSP codes are easier to use than FPE codes, especially for complex geometries. The TSP approach permits a simplified treatment of the boundary condition based on the application of the no-normal-flow condition at a mean surface. In contrast, the FPE approach requires application at the actual surface. Consequently, Cartesian field-grid systems suffice for the TSP codes whereas the FPE codes need boundary-conforming grids. Cartesian grids are considerably easier to set up compared to the boundary-conforming grids. Of course, the TSP codes suffer from limitations on the class of geometries and flow conditions that they can model accurately—a direct result of their simplified boundary-condition treatment.

The promise and excitement of the newly-found ability of computing transonic flows were so strong in the ‘70s that even wing design procedures [21, 22] were developed while the analysis methods were still evolving. Since transonic flows are particularly susceptible to viscous effects associated with shock/boundary-layer interaction, considerable research was done in coupling inviscid TSP and FPE codes with boundary-layer codes. Aeroelastic analysis capabilities based on the TSP formulation [23] were also developed.

Although the Level II codes provided the much needed capability of simulating transonic flows, their usefulness was severely limited due to a variety of factors. For example, the level of grid generation technology was not mature enough to support routine application of the FPE methods to anything more complicated than a wing or a wing-body. In addition, applications of the codes confirmed, as one might have suspected, that the codes did not accurately simulate flows containing strong shock waves or large regions of vorticity (e.g., leading-edge vortices). Therefore, they made little inroads into fighter aircraft design since the performance of a typical modern fighter aircraft is greatly influenced by the leading-edge vortices and strong shock waves.

In the author’s opinion, nonlinear potential codes were basically taken over by the rapid pace of advances in Euler codes in the early 1980s. The TRANAIR code [24] was an exception to this trend. TRANAIR adopts an unconventional hybrid approach combining the flexibility of panel methods to handle complex geometries with the ability of FPE formulations implemented on Cartesian grids to handle nonlinearities of transonic flows.

3.2.3 Euler Methods (Level III)

The Euler equations, a system of nonlinear first-order PDEs, form the basis of the Level III codes. They represent the highest-level inviscid approximation of the N-S equations. The Euler codes are applicable for simulations in all flight regimes from subsonic to hypersonic. This, combined with their demonstrated ability to automatically capture rotational flow regions (such as wakes shed behind wings and vortices emanating from sharp leading edges of fighter-type wings), requiring no explicit a priori definition of such regions, renders them significantly more useful than the Level I or II codes. However, the enhanced capability comes at the expense of additional computational cost of solving at least four and generally five coupled first-order PDEs instead of one second-order PDE.

Two factors at the dawn of the eighties convinced most researchers to shift their focus to developing Euler methods: (i) projected growth in computer power and (ii) development of more efficient numerical algorithms to solve the Euler equations [25, 26]. In addition, the accelerated pace of boundary-conforming grid generation technology, along with the introduction of the finite-volume concept to decouple flow solvers from grid mappings, held considerable promise for converting the dream of routinely analyzing complete aircraft geometries into reality. A synopsis of the impressive progress is presented here; interested readers can find many details in References 27 through 30.

Three distinct development paths can be identified for Euler codes. During the early part of the eighties, most researchers focused their energies on structured-grid methods with emphasis on patched multiblock [31] or overset grids [32] to analyze complex geometries. One example is presented here to illustrate the capabilities of a typical patched multiblock Euler code, TEAM, which was extensively used to support F-22 design needs. (More details about the code itself can be found in Reference 28, Chapter 18, and Reference 29.) This example involves full aircraft airloads prediction [33]. Nearly 370 analyses were done covering six Mach numbers, several angles of attack and a few yaw angles. Effects of leading- and trailing-edge flap deflections, horizontal tail deflections and rudder deflections were cost-effectively simulated using the surface transpiration concept [34]. Figure 10 shows a comparison of computed and measured surface pressures.

The development path shifted towards unstructured-grid methods [35] from the mid-eighties onwards. The shift was prompted by the realization that unstructured grids afforded greater flexibility in handling complex geometries and the grid-generation process could be automated. Cartesian-grid methods [36, 37] evolved in the early nineties and offered yet another attractive alternative because they essentially dispensed with the difficulties of grid generation leading to considerable reductions in time and manual labor. As far as flow solvers are concerned, structured- and unstructured-grid

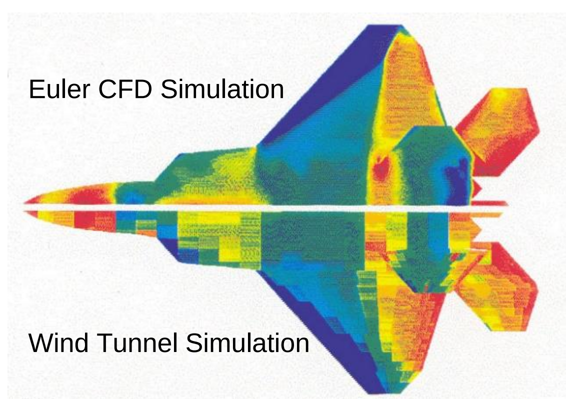

Figure 10 - Comparison of measured and computed surface pressure distributions on F-22 at 0.9 Mach number and 8° angle of attack

methods as well as Cartesian-grid methods can provide steady-state solutions on full aircraft configurations in a matter of hours using today's high-performance computers (HPCs).

Two other aspects of Euler codes development deserve mention here. First, shock-capturing rather than shock-fitting has become the preferred approach; both upwind and central-difference with adaptive-dissipation schemes have enjoyed a great deal of success. Second, most codes solve the time-dependent form of Euler equations even for modeling steady flows. Convergence acceleration techniques, such as local time stepping and multigrid, are employed to obtain time-asymptotic steady-state solutions in a computationally efficient manner. Both explicit and implicit time-marching schemes have been effectively utilized. Due to the use of time-dependent equations, extending the codes to model unsteady flows is relatively straightforward. An example presented here involves prediction of time history of hammershock overpressure in the F-22 inlet duct at a supersonic flight condition [38]. Figure 11 shows a snapshot of duct pressures and the Mach field around the forebody. The ability to compute unsteady flows has been exploited to develop dynamic aeroelastic analysis methods [39]. Inverse design [40] and aerodynamic design optimization [41] methodologies based on Euler codes are also noteworthy.

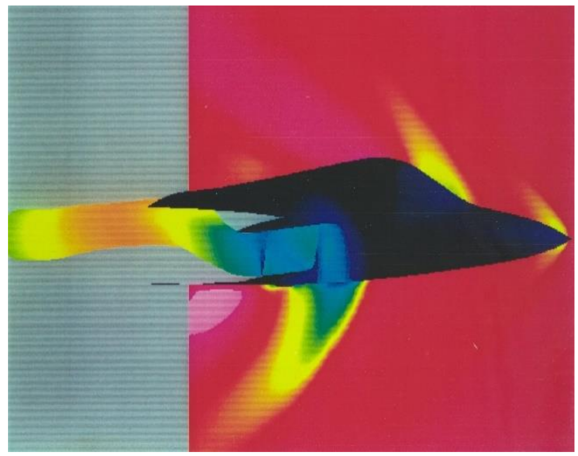

Figure 11 - F-22 inlet hammershock overpressure prediction using Euler CFD [38]

Whereas Euler codes are superior to nonlinear potential (Level II) codes in modeling strong shocks, the solutions do not necessarily replicate reality because shock/boundary-layer interaction effects are missing due to the inviscid formulation. Some researchers have combined Euler codes with boundary-layer codes to correct this deficiency. The Euler codes do have an edge over potential-flow methods in capturing leading-edge vortices. But the location and strength of the primary vortices may not be accurate in cases where the secondary and/or tertiary vortices exert considerable influence [27, 42]. Also, the codes cannot provide an estimate of total drag (including skin-friction) or model flow separation from smooth surfaces. It is, therefore, not surprising that the development of N-S codes has been aggressively pursued in parallel.

3.2.4 RANS Methods (Level IV)

Since Navier-Stokes equations have a great deal in common with Euler equations, a single code usually serves the need of solving both Euler and N-S equations. Elimination of diffusion terms readily converts the N-S equations to the Euler equations as they both share a common set of convective terms. However, the practical implications of this seemingly minor difference are enormous. In order to accurately resolve the diffusion terms, highly clustered grids are required close to solid surfaces (as well as in other regions where viscous stresses are large). With appropriate grid clustering, one can use the N-S equations to simulate laminar flows in a relatively straightforward fashion. But using DNS for even simple turbulent flows stretches the supercomputers to their limits [43].

At present, the RANS equations are used almost exclusively to simulate turbulent flows on aircraft configurations. But Reynolds averaging employed in their derivation leads to the addition of Reynolds stress terms which need to be modeled as a function of the mean flow through a turbulence model to close the equation set. A variety of turbulence models are now available ranging from relatively simple algebraic models to more sophisticated Reynolds-stress transport models.

The RANS codes have been applied to produce impressive results. Two representative examples are included here.

1. P-3C: Goble and Hooker [44] computed lift and drag forces as well as surface pressures on the P-3C aircraft using the NASA TetrUSS unstructured-grid code. Spalart-Allmaras turbulence model coupled with a wall-function model to reduce grid clustering requirements was used for this effort which was undertaken in support of the P-3C Service Life Extension Program. A sample solution is shown in Figure 12. Although CFD data correlated well in general with wind-tunnel and flight data, discrepancies were attributed to turbulence modeling.

2. JSF: Wooden and Azevedo [45] used a structured-grid CFD method, Falcon, for refining JSF outer mold lines. Figure 13 shows a typical surface pressure coefficient (C_p) distribution used for identifying drag reduction targets. Such solutions were generated using a multi-block structured-grid code with 177 blocks having a total of 27 million grid points. A two-equation k-kl turbulence model was used in their calculations.

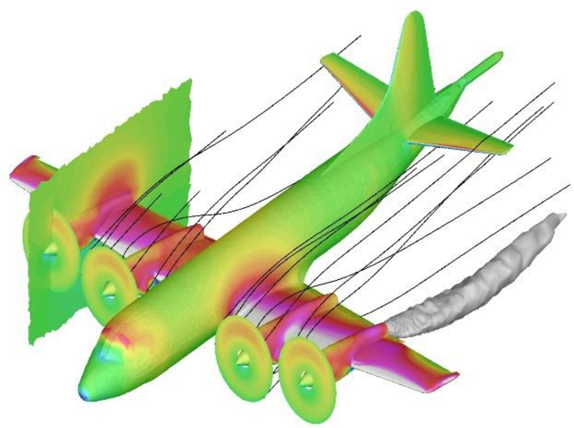

Figure 12 - Simulation of propeller effects on P-3C using NASA TetrUSS RANS software [44]

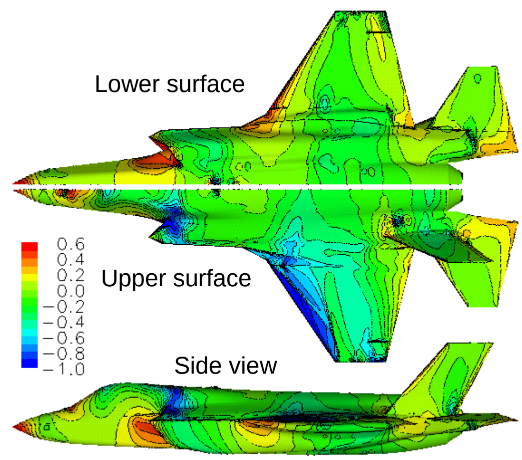

Figure 13 - JSF surface C_p distribution [45]

3.2.5 Summary Assessment of Capabilities

The preceding sections highlight the tremendous advancement in CFD capabilities Of course, many more examples can be cited to demonstrate application of today's CFD codes to simulate flow about complex aircraft configurations. The multitude of applications has helped us learn to exploit the complementary strengths of CFD and wind tunnels.

Based on CFD applications to the F-22 design during the early 1990s, Bangert et al [33] point out that the design team relied heavily on wind-tunnel data due to the limitations of CFD codes in accurately simulating viscous effects, especially when applied to full aircraft geometries and the full speed, altitude, and maneuver flight envelope. This is not too surprising due to the then level of maturity of the RANS methods. More than 40,000 hours of wind-tunnel testing was done to support the F-22 development. Nearly a decade later, the development team of the Joint Strike Fighter (JSF) aircraft [45] also relied on extensive wind-tunnel testing for generating aerodynamic performance data to meet Key Performance Parameters and Performance Specifications related to Mission Performance. Since 2002 more than 48,000 wind tunnel test hours on the Lightening II have been completed at 18 facilities around the world.

The continued heavy reliance on wind tunnels is clearly incompatible with the SBD vision of relying on computational methods as the *primary* means of generating all design data (see Section 2.2.2). Meeting the demands of SBD for future aircraft design will require dramatic improvements in CFD effectiveness as discussed in the next section.

4. CFD Effectiveness*: Current Status*

The true measure of merit of a CFD method for aircraft design applications is its effectiveness in meeting design team's desires and expectations. One of the most crucial expectations is to have credible aerodynamic data on time and on budget. The data is considered *credible* if it *faithfully replicates reality*. The design team can then confidently use the data to make good design decisions.

Assessing the effectiveness is greatly facilitated by expressing it as a product of two factors as proposed by Miranda [10]:

***E**ffectiveness = **Q**uality x **A**cceptance*

Here the "Quality" factor refers to the level of credibility of computational data for the target application, and "Acceptance" represents the level of acceptability of the computational method by users and customers. In Reference 10, effectiveness is treated as a qualitative concept with no numerical value to be assigned to it. The author has developed a semi-quantitative approach for effectiveness assessment that is analogous to risk assessment. This approach determines an *effectiveness index*, *E*, as a product of a *quality index*, *Q*, and an *acceptance index*, *A*, i.e., $E = Q \times A$. The effectiveness index, *E,* may be depicted as shown in Fig. 14. A procedure for estimating *E* is described below. Computational methods that rapidly and inexpensively produce credible data naturally have a higher effectiveness index.

The *quality index, Q,* represents the level of data credibility using two criteria: (i) *accuracy,* and (ii) *realism*. Accuracy is a measure of the correctness of the numerical simulation itself and is analogous to verification of the code. Realism indicates the degree to which computational results represent reality. The *acceptance index, A,* represents the level of acceptability by users and customers using four criteria: (a) *applicability,* (b) *usability,* (c) *cost,* and (d) *responsiveness*. Applicability is a measure of the degree to which a method is suitable for the problem at hand; usability is the ease of use by 'non-expert' users; affordability is higher when the cost (labor + computer) is lower; and responsiveness is higher when the turnaround time (elapsed time from go-ahead to delivery of data) is lower. We can estimate a numerical value for *Q* and *A* using the following expressions:

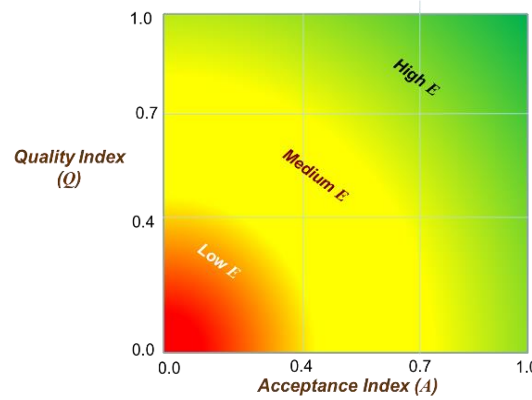

Figure 14 - Effectiveness index, $E = Q \times A$

$$Q\ or\ A = \sum_{i=1}^{N} W_i S_i \quad \text{where} \quad 0 \le W_i \le 1 \ , \ \sum_{i=1}^{N} W_i = 1 \quad \text{and} \quad 0 \le S_i \le 1$$

We can select the weight factor, W_i, for each criterion based on our assessment of their relative importance. A possible approach for selecting a score, S_i, is to assign a value between 0 and 0.4 if the level of the criterion is assessed 'low', between 0.4 and 0.7 for 'medium,' and 0.7 to 1.0 for 'high.' Note that *N = 2* for *Q* and *N = 4* for *A* in the present formulation.

4.1 RANS CFD Effectiveness: *Less than Satisfactory*

In this paper, we focus on the RANS CFD methods since they are now in widespread use for aircraft design. Also, it is reasonable to expect that these Level IV CFD methods with more complete flow physics models should provide credible aerodynamic data to meet aircraft design needs compared to the lower-level methods with simplified models of physics. Over the past three decades, effectiveness of RANS CFD for simulating flow about full aircraft configurations has considerably increased. This is largely due to considerable decrease in turnaround time and labor plus computing cost of practical applications as a result of significant advances in computing hardware and software. But the predicted computed results aren't always credible, especially for complex flows that are dominated by flow separation and/or free vortices, i.e., the quality of the results for complex flows has not significantly increased.

Figure 15 shows one example of CFD application of the NASA TetrUSS software suite to simulate flow about a simple tailless unscrewed combat vehicle scale model [46]. This software solves the RANS

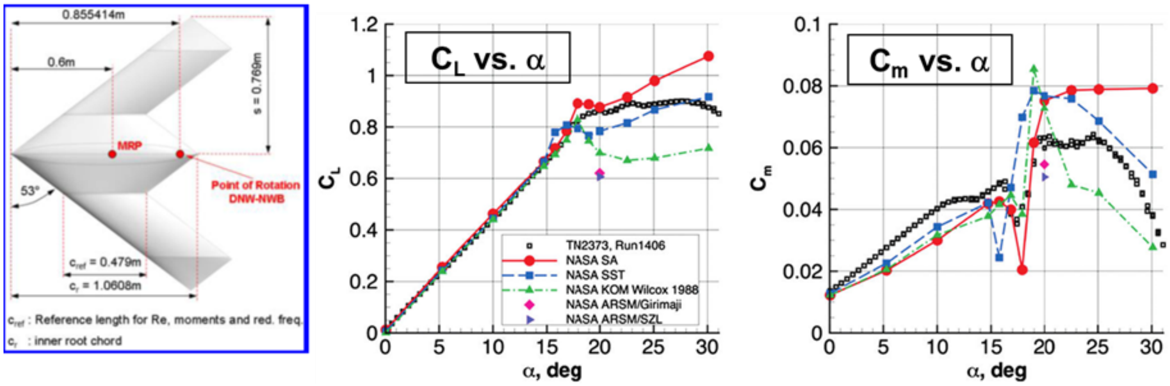

Figure 15 - NATO RTO assessment of RANS-based CFD for the SACCON model; M = 0.419, Re = 1.6×10^6 [46]

equations on unstructured tetrahedral grids. The challenge of producing credible data is clearly evident from the wide variation in C_L and C_m values for $\alpha > 15^o$ for different turbulence models in the same code. Note that all predicted lift coefficient values disagree with each other as well as with the wind-tunnel data in the separated flow region. The same disagreements are seen for moment coefficient across the entire angle-of-attack range. It is assumed that the wind tunnel data is 'faithfully' simulating reality; there is no reason to suspect otherwise.

Figure 16 shows another example of sensitivity of computed free vortex flow solutions to turbulence models. Computed vorticity magnitudes on JSF STOVL configuration [45] are shown for a two-equation k-kl model and two variants (ASM1 and ASM2) of an explicit algebraic stress model. These RANS solutions are generated using a patched multiblock structured-grid method, Falcon, using a 197-block grid with a total of 36 million grid points.

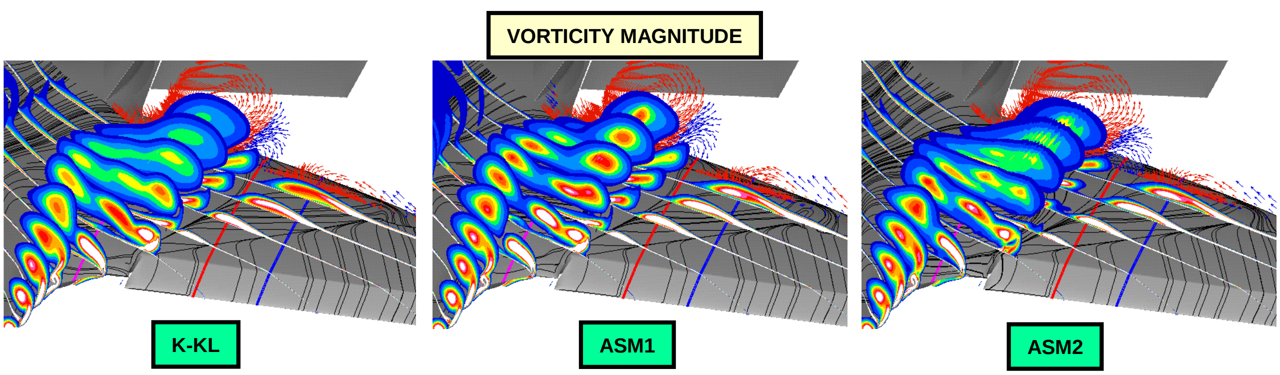

Figure 16 - Sensitivity of vorticity magnitudes of free vortex flows simulated using RANS CFD code, Falcon, for JSF STOVL configuration at takeoff conditions [45]

In addition to the effect of turbulence models on data quality, numerical models used to discretize the PDEs affect the quality of the computed results. A solution may be considered numerically accurate if it shows little or no sensitivity to changes in grids as well as numerical parameters related to the algorithm. (It is assumed that the code has been verified for *solving the problem right*.) Figure 17 [47] shows surface pressure distribution on the F-22 configuration along with regions of vortical flow at one instant in time from an unsteady flow simulation using an unstructured-grid CFD code, USM3D. The grey envelopes depict the 3-D flow reversal surfaces associated with vortex breakdown. Inside these surfaces, the velocity field is generally flowing forward relative to the airplane. The first set of CFD runs had a relatively coarse grid on the vertical fins for conservation of computational resources and faster turnaround times. However, the computed unsteady CFD pressure distributions exhibited discrepancies with test data, and computed root-mean-square (RMS) pressure levels were very low. When a finer grid was used for the vertical fin to more accurately represent a small rounded leading edge, computed RMS levels were very much in line with the measured wind tunnel data. Clearly, systematic parametric studies are essential to estimating the effects of grid resolution and numerical parameters. In the absence of wind-tunnel data, it is nearly impossible to assess the quality of the solutions. Schedule and cost constraints of a typical aircraft design effort do not permit extensive investigations to determine "optimal" grids and numerical parameters. *What we need is built-in means of quantifying the level of accuracy*. Estimation of errors due to numerical modeling is admittedly a difficult problem but an effective solution is urgently needed.

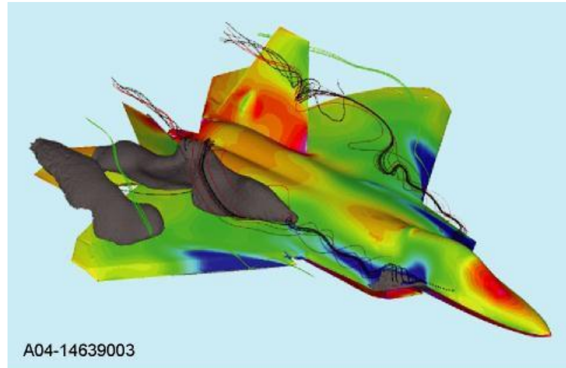

Figure 17 - F-22 flow-field features at high angle of attack computed using unsteady unstructured-grid CFD code [47]

Findings of the seven AIAA CFD Drag Prediction Workshops held to date since 2001 and the 8th scheduled for 2026 [48], four High Lift Prediction Workshops to date since 2009 with the 5th scheduled for 2024 [49] also highlight very similar challenges for producing credible data using RANS codes, especially for complex flows dominated by flow separation and/or free vortices. NASA's CFD Vision 2030 Study [50] was largely motivated by the need to address limitations of RANS methods in accurately simulating separated flows: "…*the well-known limitations of RANS methods for separated flows have confined reliable use of CFD to a small region of the flight envelope ..."* NASA's clarion call to the community through this study reflects the urgent need to develop revolutionary capabilities with maximum effectiveness in predicting aircraft aerodynamics under *all* flight conditions.

4.2 Overarching Challenge for *Fully Effective* RANS CFD

Maximizing "Effectiveness" by *simultaneously* increasing both "Quality" and "Acceptance" has been the 'North Star' of author's efforts throughout his long professional career. The author contributed to increasing CFD effectiveness by conducting and leading many research, development, and application campaigns during his 30+ years with Lockheed Martin. Based on extensive experiences to date, the author came to the conclusion that RANS CFD was unlikely to be fully effective anytime soon, if ever. With advances in HPC and numerical modeling, effectiveness of RANS CFD will steadily increase, but *the computed data will not be credible* due to turbulence [and transition] modeling deficiencies. This is certainly not due to a shortage of turbulence models. Models ranging from very simple algebraic to highly sophisticated Reynolds stress transport models are available. However, accurate modeling of complex, multiscale, nonlinear phenomena that characterize turbulence by using a few free parameters is an extremely long shot indeed.

The following excerpts from an article by Spalart [51], a well-known and highly respected expert in the field of turbulence, about RANS CFD offer credence to our assertions above:

> *"…the author deems it unlikely that a RANS model, even complex and costly, will provide the accuracy needed in the variety of separated and vortical flows we need to predict."*
>
> *"…it is more than plausible that Reynolds averaging suppresses too much information, and that the only recourse is to renounce it to some extent, which means calculating at least the largest eddies simply for their nonlinear interaction with the mean flow."*

If RANS CFD cannot provide credible solutions, what other options do we have to develop fully effective CFD? Fully effective CFD is essential to realizing the full potential benefits of SBD needed for the design and development of quality affordable aircraft. We discuss the options and challenges in the next section.

5. Future Prospects for Fully Effective CFD

Table 1 compares four options, namely, RANS, DES, LES, and DNS, for simulating turbulent flow about a typical commercial transport aircraft wing of aspect ratio 12 and Reynolds number 50 million based on the distance from the leading edge [52]. The estimated feasibility demonstration time frames in the table assume that Moore's Law will still hold.

Table 1: Status and prospects of turbulent flow simulation about typical commercial transport aircraft wing
AR = 12, Re_x = 50 million

	RANS (Reynolds-Averaged Navier-Stokes)	**DES** (Detached Eddy Simulation)	**LES** (Large Eddy Simulation)	**DNS** (Direct Numerical Simulation)
Level of Empiricism	**High**	**Medium**	**Low**	**None**
Unsteady Flows	**No**	**Yes**	**Yes**	**Yes**
# of Grid Points	**10^7**	**10^7 to 10^8**	**10^{11}**	**10^{20}**
Feasibility Demonstration	**1995**	**2010**	**2045**	**2080**

DNS *with no empiricism* is the only path to fully effective CFD but it is not expected to be feasible—even for a wing—until around 2080. To generate accurate solutions, extremely dense grids are needed in a small domain surrounding a configuration. The leads to a very large number of grid points which increases the number of time steps and computational time. The required HPC platforms are expensive as well.

LES with low empiricism appears to be a more promising mid-term option but we do need to develop a much more comprehensive understanding of the implications of its relatively 'low' level of empiricism. Its pace of progress is closely tied to advances in many areas including

- *Grids: truly* boundary conforming grid generation (not just a piecewise linear approximation of the surface) and grid adaption;
- *Models:* advanced sub-grid-scale models for wall-modeled LES (WM-LES); an assessment of the level of refined grids for wall-resolved LES (WR-LES); higher-order numerical models for discretization;
- *Software:* highly efficient software that exploits constantly evolving computer hardware architectures; effective approaches for verification and validation of complex software and for uncertainty quantification;
- *Data Management:* cost-effective ways of efficiently managing large amounts of data, and for processing of extremely large datasets to extract information of value for aircraft designers. visualization and management of enormous amounts of data associated with unsteady simulations.

To gain a better appreciation of the capabilities and challenges, we present one representative example of DNS and LES application to simulate flow about the SD7003 low Reynolds number airfoil at 0.1 Mach number, 4° angle of attack, and 60,000 Reynolds number [53]. The airfoil has 8.5% max thickness at 24.4% chord location, and 1.2% camber at 38.3%. At these flow conditions, there is a fairly stable laminar separation bubble on the upper surface and shear layer transition leads to turbulent flow. The authors of this study use the implicit LES (ILES) approach, also known as WR-LES, requiring no additional models unlike WM- LES.

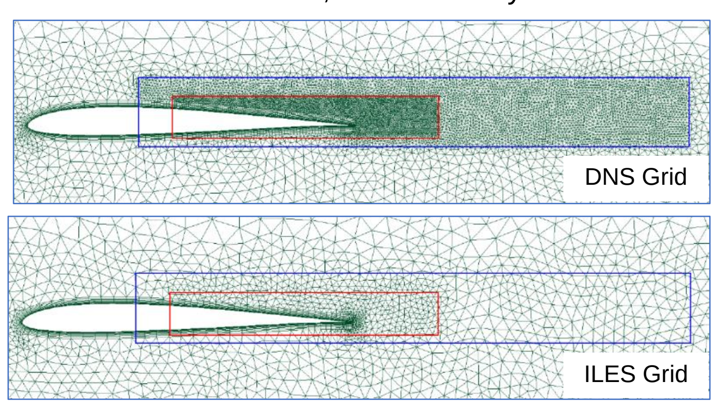

Figure 18 - Comparison of DNS and ILES grid densities [53]

Figure 18 compares small sections of the DNS grid with 84,700 hexahedra (646,100 wedges) and the ILES grid with 8,700 hexahedra (47,900 wedges). The DNS grid is much denser in the immediate vicinity of the airfoil as compared to the ILES grid. Figure 19 shows the temporal evolution of lift and drag coefficients for DNS and ILES simulations. Note that t_c is convective time to travel one chord length.

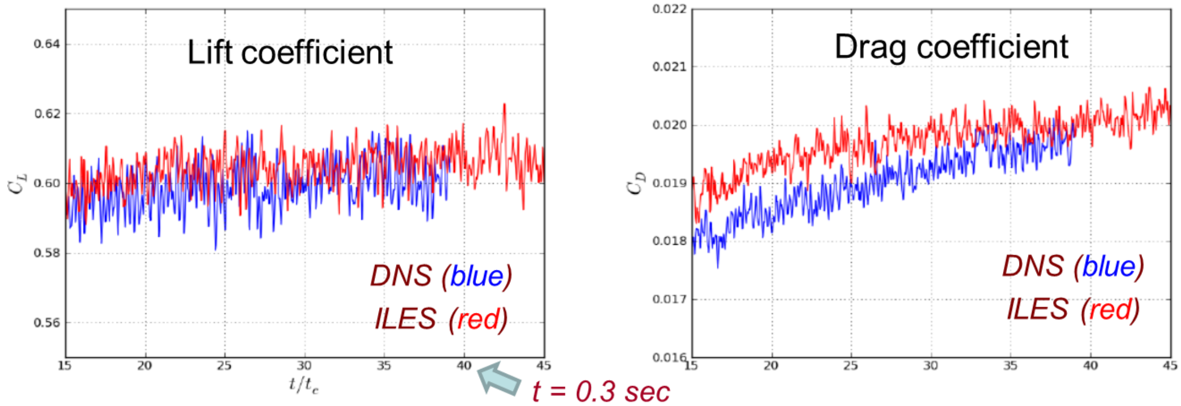

Figure 19 - Temporal evolution of lift and drag coefficients for DNS and ILES [53]

Even though the computations are not fully converged, it is instructive to compare the computational data with experimental measurements and with the results of a lower order method, XFOIL, from Reference 54. Table 2 summarizes these results. It is worth noting that the CPU-hours (on a 16,000 core "Jugene" supercomputer) for DNS are 25 times more than for ILES.

Table 2: Comparison of DNS and ILES results [53] with experimental data and XFOIL predictions [54] for SD7003 airfoil, M = 0.1, Re_c = 60,000, and angle of attack = 4°

	DNS	ILES	XFoil	Expt. (TU-BS)	Expt. (AFRL)
Freestream Turbulence	0	0	*(N_{crit} = 7.5)*	0.08%	~ 0.1%
C_L *(mean)*	0.602	0.607	0.583	-	
C_D *(mean)*	0.0196	0.020	0.0181	-	
Separation (x_{sep}/c)	0.209	0.207	0.26	0.30	0.18
Reattachment (x_r/c)	0.654	0.647	0.57	0.62	0.58
CPU-Hrs* for one t_c	**11,001**	**415**	-	-	

Figure 20 compares a snapshot of vorticity norm from the DNS and ILES simulations. It is clear from the instantaneous vorticity norms that the solutions exhibit general agreement in terms of locations of the separation point, the transition position and the turbulent region. However, the small turbulent structures that we see in the DNS results are filtered by the ILES.

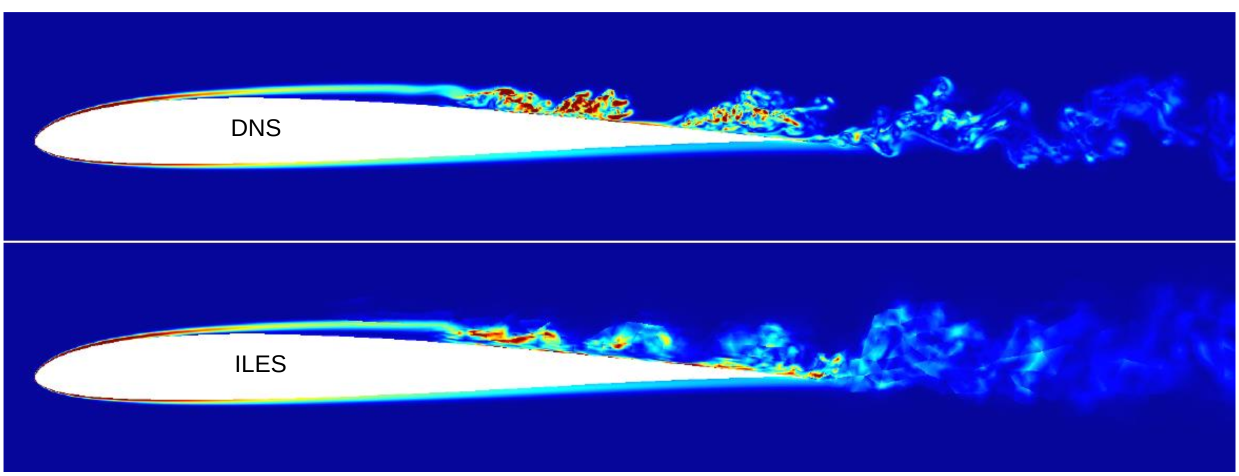

Figure 20: Snapshot of vorticity norm for DNS and ILES solutions [53]

This example clearly illustrates the challenge we face: *DNS can produce credible solutions but making it fully effective for meeting aircraft design needs will require incredible reductions in turnaround times and total costs.*

The ongoing revolutionary research in quantum computing could potentially move the DNS feasibility demonstration time frame much closer. For example, BosonQ Psi [55] has developed quantum-inspired, hybrid quantum-classical, and quantum algorithms to overcome the limitations of classical algorithms. Simulations using quantum algorithms on today's HPCs are an order of magnitude faster and cost effective than using classical algorithms. Quantum algorithms running on a simulation platform consisting of HPCs and quantum computers could reduce the time and cost by nearly three orders of magnitude! It's naturally a very exciting prospect for all CFD practitioners. The progress of the revolutionary quantum computing is being closely watched especially in light of what Prof. Seth Lloyd, Mechanical Engineering and Physics, MIT, says [56]: *"When's my date for when we actually have commercially viable quantum computers? I say 10 years, plus or minus never."*

For the near term, it is clear that *RANS is here to stay!* Therefore, one needs to take solace in Bradshaw's wise observations [57]:

> *"...we cannot calculate all flows of engineering interest to engineering accuracy. However, the best modern methods allow almost all flows to be calculated to higher accuracy than the best informed guess, which means that the methods are genuinely useful even if they cannot replace experiments."*

Despite relatively low effectiveness for simulating complex flows, RANS methods can, and routinely do, add value if and when used wisely.

6. Concluding Remarks

In today's marketplace for commercial and military aircraft, increased performance at an affordable cost is the challenge for aircraft developers. Aircraft of the future are likely to be as complex as the ones in service today, if not more so. Advances in technology are expected to continue at an accelerated pace. These advances will have to be incorporated into new designs, or used for upgrading the existing airplanes, because the customer demands technological superiority. But the cost must be affordable.

Simulation based design offers a promising approach to tackle the challenge. Success of SBD relies on quality data being delivered to design teams as early as possible so that more informed decisions can be made in the early stages of design. This SBD requirement can only be met if fast, accurate and cost-effective means are used to generate credible data in each contributing discipline as well as for modeling complex interdisciplinary relationships among design variables. The disciplines include the traditional ones such as aerodynamics, structures, signatures, etc., and the non-traditional ones such as producibility and supportability.

CFD plays a pivotal role in SBD. There are two primary reasons for it. First, every design project has to ensure that the design would be able to carry out the intended mission. Aerodynamic data from CFD provides critical inputs for assessing the airplane mission performance. Second, engineering teams of several disciplines need input from the aerodynamic team for their tasks. For example, airframe structural design team requires steady and unsteady airloads that CFD can provide. Flight control system (FCS) design team requires estimates of airplane response to control commands; CFD methods can be used to determine the incremental forces and moments produced by the movement of control surfaces per the FCS commands. In addition, CFD can provide on- and off-body data to the systems integration team for an improved understanding of the flow features surrounding an airplane which offers valuable guidance for modifying the design.

Fully effective CFD which delivers credible data, on time, on budget is essential to the success of SBD. The higher level RANS methods that are quite mature and in widespread use now have less than satisfactory level of effectiveness. They can deliver data on time and on budget but the data are not credible, especially for complex flows dominated by flow separation and vortical flows. Prospects for developing fully effective CFD capability are quite challenging, but it's not impossible to achieve such a capability. Until we get there, using RANS codes wisely can reduce reliance on extensive experimental tests to generate data for configuration design. This will facilitate the use of virtual prototypes instead of physical prototypes to verify functional and operational characteristics and help the aircraft developers meet customer expectations of high quality and affordable products.

7. Contact Author Email Address

mailto: praj@vt.edu

8. Copyright Statement

References

[1] "The cost of weapons: Defence spending in a time of austerity," The Economist, August 26th, 2010.
[2] https://simple.wikipedia.org/wiki/Boeing_707
[3] https://www.statista.com/statistics/273941/prices-of-boeing-aircraft-by-type/
[4] Raj, P., and Friedman, A., "On timely and cost-effective prediction of aerodynamic data to meet aircraft design needs," Paper H.1, RAeS Applied Aerodynamics Conference, Bristol, UK, July 22-24, 2014.
[5] Raj, P., "Computational Uncertainty: *Achilles' Heel* of Simulation Based Aircraft Design (Invited)," NATO/RTO Air Vehicle Technology (AVT) Symposium on Computational Uncertainty in Military Vehicle Design, Athens, Greece, December 3-6, 2007.
[6] Raj, P., "Aircraft Design in the 21st Century: Implications for Design Methods (Invited)," AIAA 98-2895, 29th AIAA Fluid Dynamics Conference, Albuquerque, NM, June 15-18, 1998.
[7] Nicolai, L.M. and Carty, A., "Role of the Aerodynamicist in a Concurrent Multidisciplinary Design Process," Symposium on Aerodynamic Design and Optimisation of Flight Vehicles in a Concurrent Multi-Disciplinary Environment, NATO RTO AVT MP-35, Ottawa, Canada, 18-21 October 1999.
[8] Wikipedia URL http://en.wikipedia.org/wiki/Concurrent_engineering
[9] Carty, A., "An Approach to Multidisciplinary Design, Analysis & Optimization for Rapid Conceptual Design," AIAA 2002-5438, 9th AIAA/ISSMO Symposium on Multidisciplinary Analysis and Optimization, Atlanta, Georgia, 4-6 September 2002.
[10] Miranda, L.R., "Application of Computational Aerodynamics to Airplane Design," *AIAA Journal of Aircraft*, Vol. 21, No. 6, pp. 355-370, 1984.
[11] Miranda, L.R., Elliott, R.D., and Baker, W.M., "A Generalized Vortex Lattice Method for Subsonic and Supersonic Flow Applications," NASA-CR 2865, December 1977.
[12] Youngren, H.H., Bouchard, E.E., Coopersmith, R.M., and Miranda, L.R., "Comparison of Panel Method Formulations and Its Influence on the Development of QUADPAN, an Advanced Low-Order Panel Method," *AIAA Paper 83-1827*, July 1983.
[13] Magnus, A.E., Ehlers, F.E., and Epton, M.A., "PANAIR - A Computer Program for Predicting Subsonic or Supersonic Linear Potential Flow About Arbitrary Configurations Using a Higher-Order Panel Method," NASA CR-3251, April 1980.
[14] Johnston, C.E., Youngren, H.H., and Sikora, J.S., "Engineering Applications of an Advanced Low-Order Panel Method," *SAE Paper 851793*, October 1985.

[15] Chen, P.C., Lee, H.W., and Liu, D.D., "Unsteady Subsonic Aerodynamics for Bodies and Wings with External Stores Including Wake Effects," *Journal of Aircraft 30*, May 1993, pp. 618-628.

[16] Chen, P.C. and Liu, D.D., "Unsteady Supersonic Computations of Arbitrary Wing-Body Configurations Including External Stores," *Journal of Aircraft 27*, Feb. 1990, pp. 108-116.

[17] "Transonic Aerodynamics," *Progress in Astronautics and Aeronautics*, Vol. 81, American Institute of Aeronautics and Astronautics, Washington DC, 1982, Nixon, D. (ed.)

[18] Murman, E.M. and Cole, J.D., “Calculation of Plane Steady Transonic Flows,” *AIAA Journal 9*, January 1971, pp. 114-121

[19] Boppe, C., "Transonic Flowfield Analysis for Wing Fuselage Configurations," NASA CR 3243, 1980.

[20] Caughey, D.A., and Jameson, A., "Progress in Finite-Volume Calculations for Wing-Fuselage Combinations," *AIAA Journal 18*, Nov. 1980, pp 1281-1288.

[21] Hicks, R.M. and Henne, P.A., "Wing Design by Numerical Optimization," *AIAA Paper 77-1247*, 1977.

[22] Raj, P., “A Multigrid Method for Transonic Wing Analysis and Design,” *AIAA Journal of Aircraft*, Vol. 21, No. 2, February 1984, pp 143-150.

[23] Borland, C.J. and Rizetta, D.P., "Transonic Unsteady Aerodynamics for Aeroelastic Applications, Vol. I, Technical Development Summary for XTRANS3S," AFWAL-TR-80-3107, 1982.

[24] Samant, S.S., Bussoletti, J.E., Johnson, F.T., Burkhart, R.H., Everson, B.L., Melvin, R.G., Young, D.P., Erickson, L.L., Madson, M.D., and Woo, A.C., "TRANAIR: A Computer Code for Transonic Analysis of Arbitrary Configuration," *AIAA Paper 87-0034*, January 1987.

[25] Jameson, A., Schmidt, W. and Turkel, E., "Numerical Solution of the Euler Equations by Finite-Volume Methods Using Runge-Kutta Time-Stepping Schemes," *AIAA Paper 81-1259*, June 1981.

[26] Rizzi, A., "Damped Euler Equation Method to Compute Transonic Flow About Wing-Body Combinations," *AIAA Journal 20*, Oct. 1982, pp. 1321-1328.

[27] Raj, P., “Recent Developments in the Computational Solutions of Euler Equations (Invited),” Third International Congress of Fluid Mechanics, Cairo, Egypt, January 2-4, 1990.

[28] “Applied Computational Aerodynamics,” Edited by P.A. Henne, Progress in Astronautics and Aeronautics, Vol. 125, American Institute of Aeronautics and Astronautics, Washington DC, 1990.

[29] Raj, P., “Recent Advances in Euler/Navier-Stokes Computational Methods (Invited),” International Symposium on Advances in Aerospace Sciences and Engineering, Bangalore, India, December 1992.

[30] "Computational Aerodynamics Based on the Euler Equations," AGARD-AG-325, September 1994, Sloof, J.W. and Schmidt, W. (eds.)

[31] Weatherill, N.P., and Forsey, C.R., “Grid Generation and Flow Calculations for Aircraft Geometries,” Journal of Aircraft, Vol. 22, No. 10, 1985, pp. 855-860.

[32] Benek, J.A., Buning, P.G., and Steger, J.L., “A 3-D Chimera Grid Embedding Technique,” AIAA Paper 85-1523, June 1985.

[33] Bangert, L.H., Johnston, C.E., and Schoop, M.J., "CFD Applications in F-22 Design," *AIAA Paper 93-3055*, July 1993.

[34] Raj, P. and Harris, B.W., “Using Surface Transpiration with an Euler Method for Cost-effective Aerodynamic Analysis,” *AIAA Paper 93-3506*, August 1993.

[35] Jameson, A., Baker, T.J., and Weatherill, N., “Calculation of Inviscid Transonic Flow over a Complete Aircraft,” AIAA Paper 86-0103, Jan. 1986.

[36] Tidd, D.M., Strash, D.J., Epstein, B., Luntz, A., Nachshon, A., and Rubin, T., "Application of an Efficient 3-D Multigrid Euler Method (MGAERO) to Complete Aircraft Configurations," *AIAA Paper 91-3236-CP*, 1991.

[37] Karman, S., "SPLITFLOW: A 3D Unstructured Cartesian/Prismatic Grid CFD Code for Complex Geometries," *AIAA Paper 95-0343,* January 1995.

[38] Goble, B.D., King, S., Terry, J. and Schoop, M., "Inlet Hammershock Analysis Using a 3-D Unsteady Euler/Navier-Stokes Code," *AIAA Paper 96-2547*, July 1996.
[39] Guruswamy, G.P., "Unsteady Aerodynamics and Aeroelastic Calculations for Wings Using Euler Equations," *AIAA Journal 28*, 1990, pp. 461-469.
[40] Campbell, R.L. and Smith, L.A., "A Hybrid Algorithm for Transonic Airfoil and Wing Design," *AIAA Paper 87-2552*, 1987.
[41] Reuther, J., Jameson, A., Alonso, J.J., Rimlinger, M.J., and Saunders, D., "Constrained Multipoint Aerodynamic Shape Optimization Using an Adjoint Formulation and Parallel Computers," AIAA 97-0103, January 1997.
[42] Raj, P., Kinard, T.A. and Vermeersch, S.A., "Vortical Flow Simulation Using An Unstructured-Grid Euler Method," *ICAS-96-1.4.5*, September 1996.
[43] Moin, P. and Mahesh, K., "Direct numerical simulation: a tool in turbulence research," CTR Manuscript 166, Center for Turbulence Research, Stanford University, Stanford, California, April 1997.
[44] Goble, B.D. and Hooker, J.R., "Validation of an Unstructured Grid Euler/ Navier-Stokes Code on a Full Aircraft with Propellers," *AIAA Paper 2001-1003, Aerospace Sciences Meeting, 2001*
[45] Wooden, P.A., and Azevedo, J.J., "Use of CFD in Developing JSF F-35 Outer Mold Lines," *AIAA-2006-3663, 24th Applied Aerodynamics Conference, 2006.*
[46] Frink, N., Tormalm, M., and Schmidt, S., "Three Unstructured Computational Fluid Dynamics Studies on Generic Uninhabited Combat Air Vehicle," *AIAA Journal of Aircraft*, Vol. 49, No. 6, pp 1619-1637, 2012.
[47] Cunningham, A.M., Jr., Anderson, W.D., Patel, S.R., and Black, C.L., "Lockheed Martin Aeronautics Perspective on Aircraft Buffet Prediction and Analysis," Symposium on Flow-Induced Unsteady Loads and Impact on Military Applications, Applied Vehicle Technology Panel (AVT), NATO, Budapest, Hungary, 25-28 April 2005.
[48] https://aiaa-dpw.larc.nasa.gov/
[49] https://hiliftpw.larc.nasa.gov/
[50] Slotnick, J., Khodadoust, A., Alonso, J., Darmofal, D., Gropp, W., Lurie, E., and Mavriplis, D., "CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences," NASA/CR-2014-218178, March 2014. https://permanent.access.gpo.gov/gpo53497/20140003093.pdf
[51] Spalart, P.R., "Strategies for turbulence modelling and simulations," International Journal of Heat and Fluid Flow, 21, pp 252-263, 2000. DOI: 10.1016/S0142-727X(00)00007-2
[52] Spalart, P., Reflections on RANS Modeling, June-August 2012. https://www.cespr.fsu.edu/people/myh/CFD-Conference/Session-4/Philippe-Spalart-Presentation.pdf
[53] Carton de Wiart, C., and Hillewaert, K., "DNS and ILES of transitional flows around a SD7003 using a high order Discontinuous Galerkin Method," ICCFD7-2012-3604, Seventh International Conference on Computational Fluid Dynamics, Big Island, Hawaii, 9-13 July 2012. https://www.iccfd.org/iccfd7/assets/pdf/papers/ICCFD7-3604_paper.pdf
[54] Uranga, A., Persson, P-O, Drela, M., and Peraire, J., "Implicit Large Eddy Simulation of Transitional Flows Over Airfoils and Wings," AIAA 2009-4131, 19th AIAA Computational Fluid Dynamics Conference, San Antonio, TX, 22-25 June 2009. https://doi.org/10.2514/6.2009-4131
[55] Personal communication, Abhishek Chopra, BosonQ Psi (https://www.bosonqpsi.com/)
[56] Kelvey, J., "4 Reasons to Root for Quantum Computing," AIAA Aerospace America, July/August 2024.
[57] Bradshaw, P., "Turbulent Secondary Flows," *Annual Review of Fluid Mechanics*, 19, 1987, pp. 53-74.