NASA/TP–2012-217602

Role of Computational Fluid Dynamics and Wind Tunnels in Aeronautics R&D

Edited by:
Mujeeb R. Malik and Dennis M. Bushnell
Langley Research Center, Hampton, Virginia

September 2012

NASA STI Program . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA scientific and technical information (STI) program plays a key part in helping NASA maintain this important role.

The NASA STI program operates under the auspices of the Agency Chief Information Officer. It collects, organizes, provides for archiving, and disseminates NASA's STI. The NASA STI program provides access to the NASA Aeronautics and Space Database and its public interface, the NASA Technical Report Server, thus providing one of the largest collections of aeronautical and space science STI in the world. Results are published in both non-NASA channels and by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA Programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers, but having less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services also include organizing and publishing research results, distributing specialized research announcements and feeds, providing information desk and personal search support, and enabling data exchange services.

For more information about the NASA STI program, see the following:

- Access the NASA STI program home page at *http://www.sti.nasa.gov*

- E-mail your question to help@sti.nasa.gov

- Fax your question to the NASA STI Information Desk at 443-757-5803

- Phone the NASA STI Information Desk at 443-757-5802

- Write to:
  STI Information Desk
  NASA Center for AeroSpace Information
  7115 Standard Drive
  Hanover, MD 21076-1320

NASA/TP–2012-217602

Role of Computational Fluid Dynamics and Wind Tunnels in Aeronautics R&D

Edited by:
Mujeeb R. Malik and Dennis M. Bushnell
Langley Research Center, Hampton, Virginia

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23681-2199

September 2012

Contributors

Ramesh K. Agarwal
W. Kyle Anderson
Douglas N. Ball
John A. Benek
Edward L. Burnett
Jay P. Boris
Raymond C. Cosner
Edward M. Kraft
Dimitri J. Mavriplis

Robert W. MacCormack
Mark R. Melanson
Douglas E. Post
Eldon S. Powell
Brian R. Smith
Philippe Spalart
Timothy T. Takahashi
Edward N. Tinoco

Acknowledgments

The Editors would like to thank Harold Atkins, Mark Carpenter, Robert Hall, Khaled Abdol-Hamid, Mehdi Khorrami, William Kilgore, Laurence Leavitt, Wei Liao, Joseph Morrison, Eric Nielsen, Christopher Rumsey, and James Thomas (all of NASA Langley Research Center), Dennis Jespersen (NASA Ames Research Center), and Josip Loncaric (Los Alamos National Laboratory) for their insight and contribution to this work.

Available from:

NASA Center for AeroSpace Information
7115 Standard Drive
Hanover, MD 21076-1320
443-757-5802

Foreword

This study was conducted in the fall of 2009 at the direction of Ms. Lesa Roe, Director of the National Aeronautics and Space Administration's (NASA) Langley Research Center (LaRC). The task was to assess modeling and simulation (MODSIM) capabilities and the role of MODSIM in replacing physical testing infrastructure over the next 10 to 20 years. The major focus of this study was to intuit how the current state of the art of MODSIM will change going forward in light of projected computer developments. During this same time period, computing speed is projected to increase 1000-fold to reach two orders of magnitude beyond the computing speed of a human brain, which will have a profound impact on MODSIM capabilities. The original scope of the study was to involve the full spectrum of MODSIM; however, for a number of reasons, the scope shifted to a comparison of computational fluid dynamics (CFD) capabilities versus wind-tunnel analysis. These reasons include the fact that CFD is several years ahead of some other aerospace computational disciplines (e.g., computational aeroacoustics); furthermore, LaRC facilities include a number of wind tunnels for which CFD is the applicable computational discipline. The study was designed as a virtual study, where experts from industry, academia, and government laboratories (other than NASA) were asked to provide their position electronically within a period of 3 to 4 weeks on the question of MODSIM versus physical testing. The invited experts included CFD developers, aircraft designers, and leaders in aerodynamics. The inputs that were provided by the contributors were disseminated electronically to all other invitees. The editors compiled and edited the input received, added material to fill the gaps, and returned the information to the contributors for comment. This report is the result of that collective effort. In this report, the roles that CFD and wind tunnels can play in meeting the research and development challenges that are facing aeronautics is explored. The process adopted in this virtual study is described in greater detail in appendix A.

The opinions expressed herein are those of the contributors and the editors but not necessarily of their respective organizations. (See appendix B for an alphabetical list of the contributors.)

While a preliminary report was prepared in January 2010 for the LaRC Center Director, final publication was delayed in part due to the review process and to the editors' involvement in other activities.

Table of Contents

List of Figures

List of Tables

Acronyms and Nomenclature

3D	Three-dimensional
ADIGMA	Adaptive, Higher Order Variational Methods for Aerospace Applications
AFOSR	Air Force Office of Scientific Research
AIAA	American Institute of Aeronautics and Astronautics
ARC	Ames Research Center
ARMD	Aeronautics Research Mission Directorate
CAWAPI	Cranked Arrow Wing Aerodynamics Project International
CFD	Computational fluid dynamics
CFX	Skin friction
C_L	Coefficient of lift
CPU	Central processing unit
CREATE	Computational research and engineering acquisition tools and environments
DARPA	Defense Advanced Research Projects Agency
DES	Detached-eddy simulation
DLR	Deutsches Zentrum für Luft- und Raumfahrt (German Aerospace Center)
DNS	Direct numerical simulation
DoD	Department of Defense
DoE	Department of Energy
DPW	Drag Prediction Workshop
ETA	Wing span position
GMRES	Generalized minimal residual method
GPU	Graphics processing unit
HPC	High-performance computing
IBM	International Business Machines
ICASE	Institute for Computer Applications in Science and Engineering
IDIHOM	Industrialization of High-Order Methods
LaRC	Langley Research Center
LES	Large-eddy simulation
LFC	Laminar-flow control
MDO	Multidisciplinary design optimization
MODSIM	Modeling and simulation
MW	Megawatts
NASA	National Aeronautics and Space Administration
NSF	National Science Foundation
NTF	National Transonic Facility
OOM	Orders of magnitude
ORNL	Oak Ridge National Laboratory
R&D	Research and development
RANS	Reynolds-averaged Navier-Stokes
RSM	Reynolds stress modeling
S&C	Stability and control

SST	Shear stress transport
U.S.	United States
UAV	Uninhabited aerial vehicles
URANS	Unsteady Reynolds-averaged Navier-Stokes
USC	University of Southern California
V_{EAS}	Equivalent air speed
V&V	Verification and validation

Executive Summary

Computational fluid dynamics (CFD) has had a profound impact on the aircraft design process in the past four decades and is partially responsible for the reduction in the amount of wind-tunnel testing that was conducted during the same period. Because computer speed is expected to continue to increase, CFD will continue to encroach upon the need for physical testing requirements and eventually will replace the wind tunnel. When that milestone will be reached is the fundamental question that is addressed in this study.

The panel agreed that for the cruise case with minimal separated flow, which is only a small part of the flight envelope, wind tunnels are now used only for final configuration design checks; this is one reason for the reduced utilization of the wind tunnels at the National Aeronautics and Space Administration (NASA) Langley Research Center . Final design testing is currently part of the certification process, and this process would have to change to enable the demise of this remnant of cruise-condition physical testing. The success of CFD has resulted both from the availability of faster computers and from the investments in algorithmic and turbulence model development.

For other flight conditions (i.e., for which the vehicle is designed for greater loading and which in general include complex flow separation), wind tunnels are and are expected to continue to be an important part of the design process. The essential reason for this is twofold: the lack of computational capacity to produce parametric designs within a reasonable time scale and the lack of adequate modeling for the exceedingly complex turbulence that is associated with such separated flows. Based on Moore's Law, conventional computer improvements will require more than 20 years to provide adequate computational capability to provide sufficiently accurate *ab initio* turbulence computations, although the problem of design turnaround will probably be satisfactorily addressed within that time frame.

The challenges that are faced by CFD (e.g., unsteady separation, boundary-layer transition) are such that they cannot be resolved by the mere availability of faster machines. Research is needed for the development of more accurate numerical schemes, advanced solver technology, grid adaptation, error estimation, physics modeling, and schemes for efficiently exploiting the capabilities of future massively parallel machines. The full potential of ever-increasing computer power cannot be realized without strategic investments in the computational infrastructure.

This study concludes that, based on current and projected machine shortfalls with respect to turbulence in complex separated flows, wind tunnels will still be required; in fact, additional wind-tunnel testing will be required both to adequately foster the development of appropriate turbulence models and to subsequently build confidence in these models. Therefore, additional wind-tunnel closures at LaRC over the next 10 to 20 years will probably result more from a combination of reduced utilization and overall cost/infrastructure drivers than from the complete obviation of the need for wind tunnels by modeling and simulation (MODSIM).

The study further concludes that, given research in various numerical issues along with relevant flow physics, as well as another 20 years of Moore's Law advancements, CFD could principally replace wind tunnels for off-design conditions if an accurate turbulence modeling approach is developed for complex flows. Therefore, the team recommends a significant increase in research investments to advance the state of the art of MODSIM.

1.0 Introduction

Since the time of the Wright Brothers, wind tunnels have played a critical role in the design and development of aircraft and other aerospace vehicles, and major wind-tunnel facilities were constructed both in Europe and the United States to help fuel the aeronautical revolution of the 20th century [1−6]. The National Aeronautics and Space Administration (NASA) continued to design and construct new wind tunnels with increasing capabilities and application ranges up through the early 1980s; few or no serious additions have been made to the inventory since that time. The last major NASA wind tunnel was the National Transonic Facility (NTF) [7, 8], which came online in 1981 at NASA Langley Research Center (LaRC). The European Transonic Wind Tunnel was completed in 1993 to provide test data at flight Reynolds numbers for transport aircraft [9, 10]. In 1994, a national study of aeronautics research and development (R&D) facilities was conducted in the United States; this study recommended the construction of large subsonic and transonic wind tunnels at a cost of about $3.2 billion to provide world-class facilities for both commercial and military aircraft development [11]. However, this attempt failed to yield any results. On the contrary, the overall trend has been to close wind tunnels in the last three decades; this has been the case both in the United States and abroad. Since 1980, NASA LaRC alone has closed 12 hypersonic tunnels, 7 transonic tunnels, and 3 subsonic tunnels. Seventeen of these have been closed, demolished, or abandoned since 1995. The list includes many major facilities, such as the 16-Foot Transonic Tunnel, and this list continues to grow.[1]

The reasons for this about-face in wind-tunnel construction and utilization are many, and particularly for NASA, include:

- **Increasing maintenance costs:** The maintenance costs have continued to increase for the existing, aging wind-tunnel inventory, some dating back to the 1930s, to include the costs associated with the huge electrical power consumption of the major facilities.
- **A shift in wind-tunnel operational financing:** Prior to the 1990s, wind-tunnel operating costs were covered in overhead accounts; their operation was essentially without cost to the R&D programs of both NASA and the Department of Defense (DoD) and was often without cost to commercial projects if cogent arguments could be made regarding value to NASA research. NASA now attempts cost recovery for wind-tunnel utilization; this practice has lead to a reduction in wind-tunnel utilization across the board.
- **Reductions in the number of aeronautical systems in development:** During the 1960s, ’70s, and ’80s, the nation was actively working several fighter, bomber, and missile programs and pursuing major research programs for advanced civilian transports. Today, few such programs exist, although NASA is attempting to resurrect a program in frontier transport development [12]. This change has been well documented on a national level [13].
- **Increased level of understanding:** The level of understanding of transonic aerodynamics has increased as compared with 30 to 40 years ago.
- **The capability of CFD to replace physical wind-tunnel testing for an ever-increasing number of tasks/functionalities:** This is documented in reference [14], which indicates

---

[1] Two more tunnels will be mothballed by the end of 2012.

> that decade by decade since the 1960s computations have subsumed ever-increasing roles in design, enabled by the massive improvements in computer capability. Computer speed has increased by some seven orders of magnitude (OOM) since the early 1960s, with another eight or more OOM foreseen over the next 25 to 30 years. Furthermore, many scaling issues exist with wind tunnels, including walls, stings, aeroelastics, Reynolds number, propulsion effects, and so on, which CFD either obviates or takes into account. Increasingly, wind tunnels have shifted from being a flight predictor to being a calibrator of computational approaches, which are subsequently utilized for flight predictions and design.

The purpose of the present report is to investigate the status and future projections for the question of supplantation of wind tunnels by computation in design and to intuit the potential impact of computation approaches on wind-tunnel utilization—all with an eye toward reducing the infrastructure cost at aeronautics R&D centers. Wind tunnels have been closing for the myriad reasons previously indicated, and such closings have reduced infrastructure costs. Further cost reductions are desired, and the work herein attempts to project which wind-tunnel capabilities can be replaced in the future and, if possible, the timing of such. If the possibility exists to project when a facility could be closed, then maintenance and other associated costs could be rescheduled accordingly (i.e., before the fact) to obtain an even greater infrastructure cost reduction.

In a recent study [15] sponsored by the National Science Foundation (NSF) and other federal agencies, including the Department of Energy (DoE) and NASA, a panel of experts has provided an assessment of the international R&D activities in the field of simulation-based engineering and sciences. The panel noted that "…computer simulation is more pervasive today—and having more impact—than any time in the human history." This is certainly true of the impact of CFD on aeronautics, where CFD methods have covered a broad spectrum, including panel methods, potential flow, and Euler and Reynolds-averaged Navier-Stokes (RANS) codes. However, going forward, the focus of this discussion will be on higher fidelity CFD methods that cover the spectrum from RANS to large-eddy simulation (LES). This is because, as will be discussed later, the major challenge that faces CFD is the computation of complex turbulent flows, including flow separation, which requires use of the methods from the higher end of the computational spectrum.

The major focus of the study is to intuit how the current situation will change going forward over the next 10 to 20 years in light of projected machine developments. The 20-petaflop (human brain speed) machine is slated to be delivered in 2012; machines are now at around 2 petaflops.[2] The exaflops machine is being designed, which would deliver 1,000 petaflops. Projections indicate that the trend of increasing computer speed will accelerate as quantum computing replaces silicon-based computing (see appendix C). Critical issues include whether any type of cusp has been reached; whether simple extrapolation of the progress of computations versus experiments over the last four decades forward can be achieved; and whether, given another three to six OOM in computing speed in next 10 to 20 years, additional whole classes of problems could be subsumed (i.e., problems that do indeed now require physical experimentation/testing). Seven OOM in computing speed on silicon have been reached since the

---

[2] This was the case at the original writing of this report in November 2009. The world's fastest computer is currently IBM's Sequoia at 16.3 petaflops, an eightfold increase in speed in less than 3 years.

early 1960s–to yield a metric for what an additional three to six OOM might provide. The question of whether advancements in computer hardware alone will be sufficient to resolve the turbulence scales that are needed to solve the noncruise aircraft design problem is addressed here.

Aeronautics applications cover a wide speed range from subsonic to hypersonic and include fixed-wing and rotary aircraft, as well as launch and entry vehicles. While most of the discussion here will focus on the application of CFD to aircraft design, a common theme, namely the inability to compute complex turbulent separated flows, connects all of the vehicle classes and flow regimes (e.g., internal flows, external flows, wakes, jets, vortices, plumes, and so on). Therefore, the discussion in this report is relevant to all vehicle development efforts within aeronautics for which CFD is becoming a major player in the design process.

For aerothermodynamics and propulsion applications, important issues (e.g., ablation modeling) that are associated with flow chemistry and combustion require further research to improve the predictive capability of the computational codes. These modeling aspects are not addressed in this report.

This report is organized as follows. The RAND Corporation has examined the outlook for NASA facilities in fulfilling the nation’s R&D needs in aeronautics. Their findings are summarized first.

Then, four stages of CFD penetration into the aerodynamic design process and the current design practice are discussed. Next, the issues that are facing CFD and those that are preventing it from replacing physical testing are discussed. The well-known issue of wind-tunnel scaling to flight is treated to demonstrate that wind tunnels are not capable of providing all of the design data that are necessary for aircraft development programs and to demonstrate the inherent role of CFD in filling these gaps, irrespective of the associated cost.

The possibility that CFD may replace wind tunnels hinges primarily on the availability of much more powerful computing hardware; thus, the question of what these hardware developments could actually deliver is addressed next. The argument is made that hardware alone will not solve all of the issues that are associated with CFD; therefore, the study team has advocated the need for increased research effort to advance computational technology, including algorithmic development, flow physics modeling, and required verification and validation (V&V). The issue of wind-tunnel closings is treated next in an attempt to answer the primary question posed to this study team. Finally, recommendations are provided for the way forward, both for wind tunnels and CFD.

2.0 The RAND Report

NASA and the Office of the Secretary of Defense sponsored a one-year study, which was carried out by the RAND Corporation, to examine the nation’s wind-tunnel and propulsion testing needs and the ability of NASA’s major wind-tunnel and propulsion test facilities to fulfill those needs. The scope of the study included subsonic, supersonic, and hypersonic facilities and evaluated the available capabilities with respect to Reynolds number, Mach number, flutter, aeroacoustics, rotorcraft, icing, propulsion simulation, and so on. To collect the data, the RAND panel interviewed personnel from NASA, DoD, several domestic and foreign test facilities, and the leading aerospace companies with commercial and military products. Their findings were published in a 2004 technical report [16].

Figure 1, which is reproduced here from the RAND report, shows that the number of new aerospace vehicles that have been put into production has decreased significantly over the past six decades. This decrease has been one of the major factors in the reduction in wind-tunnel test requirements and has resulted in the closure of some facilities.

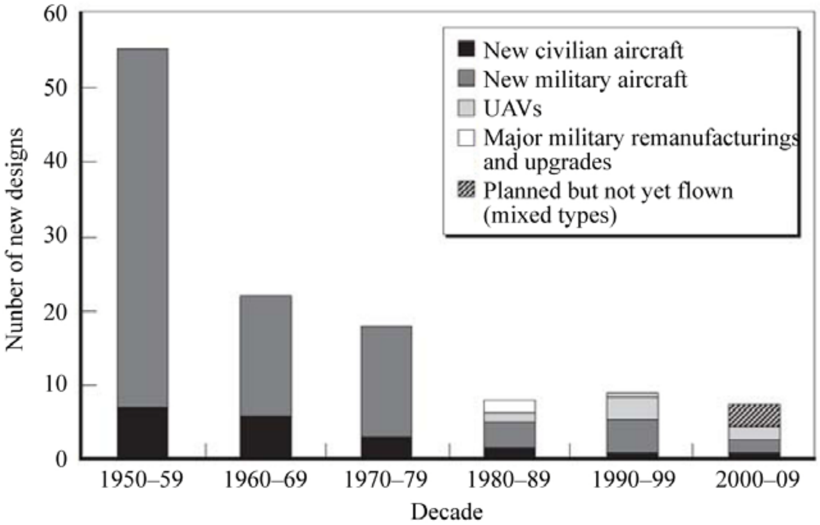


Figure 1. Estimated number of new aircraft designs reaching first flight: 1950–2009.

The RAND team also addressed the impact of CFD on wind-tunnel test requirements. The substantial role of CFD in configuration screening and refinement, particularly at cruise conditions for transport aircraft, was recognized as contributing to a reduction of about 50 percent in the number of required testing hours for such applications. This was based on the industry survey that was conducted by the study team. The study also noted that CFD is sometimes used to answer questions that arise during production and to address problems that are uncovered during flight testing. However, the study concluded that "…CFD is not yet considered reliable for predicting the characteristics of the complex separated flows that dominate many critical design points for an aircraft." The report added that, even when the codes are reliable, CFD methods are not able to generate the vast amounts of aerodynamic data that are needed for the design process in a timely manner. The study noted that computational technology will not eliminate the need for test facilities in the foreseeable future and that estimates of the time frame for CFD to be fully capable of replacing wind-tunnel testing are on the order of decades. The study emphasized that the envelope of applicability of CFD could be expanded only through steady investment in the technology and that the validation process will require carefully designed wind-tunnel experiments.

3.0 Stages of CFD Penetration into the Design Process

This section reviews the stages that the engineering community would follow in eliminating the use of wind tunnels in favor of CFD. This discussion must be carried forward in terms of individual categories of testing; most definitely, blanket statements cannot be made. The metrics (i.e., quality, schedule, cost, and risk) for the acceptance of CFD in the design process are described in appendix D. The four stages of CFD penetration into the design process are discussed below.

- **Stage 1:** CFD must produce equivalent data, with better quality, cost, schedule, and/or risk over the entire envelope, without prohibitive drawbacks in one of these areas. At this stage, CFD is often not part of the product development process, but the product development teams are paying close attention to the evolving capabilities of CFD. At this stage, CFD is used to explore a portion of the flight envelope at which problems are anticipated in stability and control (S&C) or loads, but this is not the same as building a full database for aerodynamic performance.

- **Stage 2:** CFD must begin to penetrate into the engineering process as a complement to various categories of testing. In this stage, CFD is used to add depth to information (e.g., the entire flow field versus just the polars). The wind tunnel is the primary data source, and CFD is a value-added complementary source of data. As the second independent source of data, CFD is viewed as vital in risk management. A very indistinct boundary exists between this stage and the next.

- **Stage 3:** Over time, and with consistently good CFD results, engineering practice evolves to CFD as the primary source, but the wind tunnel is retained as a risk-reducing independent source of data. At the end of this stage, wind-tunnel testing will drop to a few tests that are conducted fairly late in the development process. The second independent source of data (i.e., the wind tunnel) is seen as vital in risk management.

- **Stage 4:** The engineering program makes the huge leap to abandon the wind tunnel entirely, and uses only CFD. This step requires sufficient confidence in the validity of CFD results throughout the envelope (quality and risk), and it requires enough experience in prior stages so that the engineering team is confident that CFD work can be accomplished within the program parameters (cost and schedule).

The first three stages are evolutionary, with incremental advances. If the CFD results are disappointing or suspect, then the engineering program can increase its planned utilization of the wind tunnel. Advancing to the fourth stage requires a paradigm shift, which engineering managers will only make when sufficient confidence exists in CFD to perceive it as the best way to meet a data requirement in terms of all four metrics—quality, schedule, cost, and risk.

Program managers are focused on creating a product, and risk is their mortal enemy. Time and money are serious constraints. They live in fear of unknown risks. Today, a shift from wind-tunnel testing to CFD is often seen as adding risk. Risk is estimated by CFD experts and risk is perceived by the program manager. The latter is what counts. To gain acceptance from engineering managers, CFD must offer clear and meaningful advantages over alternative data sources in at least one of the four metrics, and offsetting weaknesses cannot exist in any of the other metrics.

The precise question posed to this study group was as follows: When will CFD be at stage 4? The consensus is that a great deal of time will be spent at stage 3 before moving to stage 4, unless the government (or a controlling authority) chooses to force transition to this stage by eliminating a wind tunnel before the utilization has dropped to low levels.

Establishing a framework for discussion and then identifying the current relationship between CFD and wind-tunnel testing (in meeting various data requirements) are the prerequisite steps to answer the above question. Next, we need to consider the advances that CFD must make to move to a higher stage of utilization as the primary data source (stage 3) and ultimately as the sole data source (stage 4).

In assessing CFD versus wind-tunnel testing, the stage of the product life cycle (i.e., conceptual design, preliminary design, detailed design, flight testing, and operational service) must also be considered. In early design, the engineering team often needs only a small amount of data that does not have to be highly accurate. Analysis methods ranging from simple planform and panel methods up to Euler and RANS methods have monopolized this stage for a long time. As the design approaches maturity, the data requirements rise exponentially. More data are required, with higher accuracy on shorter time scales, and large databases must be obtained for loads, S&C, flight simulation, and other purposes. One must also consider emergency scenarios in addition to the nominal operating conditions.

Once a product is in service, regular, planned improvement activities (e.g., putting a new sensor, antenna, or store on the airplane) are scheduled, and problem-resolution activities arise as operational problems are encountered and resolved. The vehicle application also must be considered (e.g., commercial programs have a much lower tolerance for risk than do military programs). This means that some military programs will likely move to stage 4 earlier than commercial programs. Within the military programs, a wide variation exists in risk tolerance. Low risk is the key on many programs, but the Defense Advanced Research Projects Agency (DARPA), for example, actively seeks higher levels of risk in search of radical advances. Examples of military vehicle development programs can be cited where the mold lines were frozen based on CFD alone.

4.0 Current Design Practice

Recent experience indicates that CFD is at stage 3 for aircraft development programs in which the design is developed and matured using CFD only, with confirming wind-tunnel tests performed near the end of the development program only on the leading configuration or, perhaps, on one or two leading variants. Today, this is essentially the norm for transonic aerodynamics performance work for conventional configurations. The wind tunnel is seen as a vital, risk-management-independent confirmation of the data. This situation reflects high but not complete confidence in CFD predictions. Extensive wind-tunnel testing was necessary for the development of the Boeing 777 and 787; however, the amount of wind-tunnel testing that was required was significantly less than for previous designs. This was due in part to the use of CFD, which enabled "smarter" testing, thereby reducing the size of the test matrix that was necessary to acquire the critical aerodynamic data and, in some instances, eliminating the need for testing. Stage 3 also applies in areas such as transonic inlet integration.

The large transonic database tasks fall into stage 2, in general, except that CFD may be at stage 3 for applications where the flight envelope is narrow and a small number of data points will cover the envelope (e.g., a missile).

Thus, CFD has had a significant impact in the development of high-speed (transonic cruise condition) lines for an aircraft; this is true both in the United States and Europe (see refs. [17–21]). The use of CFD, specifically in design/optimization, has reduced the use of wind tunnels to that of validation of the cruise drag characteristics of the configuration. However, low cruise drag alone does not guarantee a viable aircraft configuration. The aircraft must be certified to operate over a wide range of flight conditions and geometric configurations (e.g., flaps, controls), as illustrated in figure 2 [22].

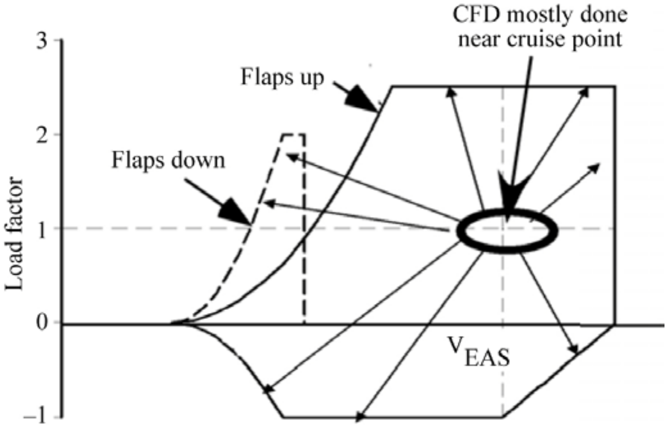


Figure 2. Flight *V*- *n* envelope.

The role of CFD in the aircraft development effort is illustrated in figure 3, which notionally shows that for a large transport aircraft the high-speed lines design (cruise design) makes up less than 20 percent of the total aerodynamics-related aircraft development effort (see ref. [22]). The exact division of the development effort is greatly influenced by the intended mission of the aircraft. The bulk, if not all, of the CFD is based on nonlinear methods including full potential with coupled boundary layer, Euler, and RANS. To be able to expand the use of CFD over the entire flight envelope, considerable progress in algorithm, physics modeling, and hardware technology is still necessary. Many flight conditions are characterized by large regions of separated flows. Transport aircraft, for example, encounter such conditions (1) at low speed with deployed high-lift devices, (2) at their structural design loads conditions, or (3) as they are subjected to in-flight upsets that expose them to speed and/or angle-of-attack conditions outside the envelope of normal flight conditions. The routine use of CFD based on the RANS approach to generate the large databases that are needed for loads and S&C will require OOM improvement in throughput (productivity) through improvements in geometry modeling and grid generation, improvements in turbulence modeling, and improvements in algorithm and hardware performance. These necessary improvements are discussed in later sections.

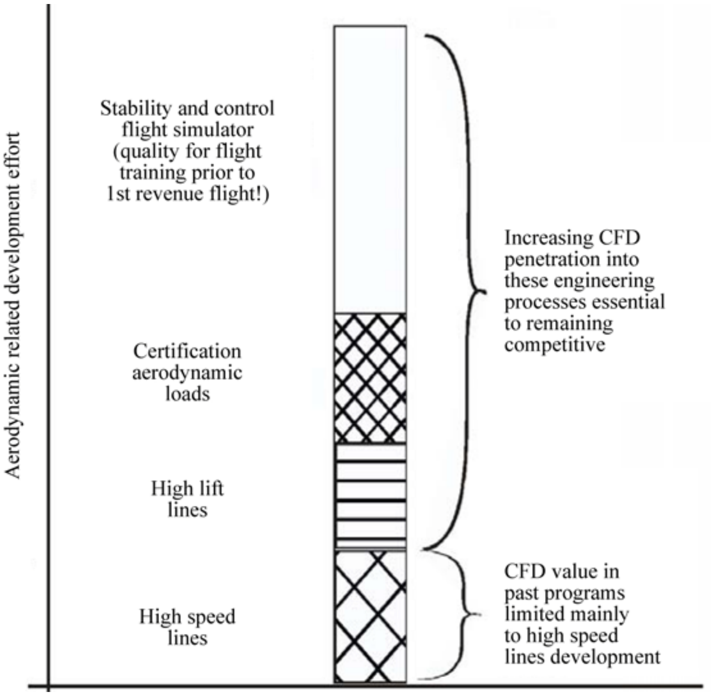


Figure 3. Role of CFD in aircraft development.

Currently, a typical aircraft development program goes through two or three design cycles, and each design cycle requires a great deal of CFD work and significant wind-tunnel testing (see ref. [17]). Each design cycle defines the aerodynamic and structural characteristics of the aircraft with sufficient detail to begin manufacture, if necessary. Additional design cycles further refine and optimize the configuration. Wind-tunnel testing generates 500,000 to 1,000,000 data points that are used to create the necessary aerodynamic databases for design! As previously stated, the database must cover the entire flight envelope and involves the testing of several different wind-tunnel models at several different tunnels. While CFD has reduced the amount of testing that is necessary to develop a low-drag high-speed cruise configuration, the advent of increased certification requirements, maneuver load alleviation, envelope protection, fly by wire, and so on, has increased the size of the loads, S&C, and simulator databases that are needed for aircraft development. The aircraft development process has reached a stage where, in general, prototypes for large transport aircraft are no longer built. The new configuration is expected to be certified and enter service within a year from the first flight. The flight characteristics and aerodynamic loads must be predicted with a very high degree of accuracy prior to the first flight in order to minimize the surprises and the required changes that are uncovered during flight testing. The common practice is to use CFD to make corrections to the wind-tunnel-derived databases to account for Reynolds-number scaling, wind-tunnel wall and mounting-system interference, and geometric differences between the flight article and the wind-tunnel model, and in some cases, to enhance (i.e., fill in) the databases where no wind-tunnel data were taken.

For aircraft manufacturers, the target is to reduce the aircraft development process to one design cycle. This will require significant improvements in both CFD and wind tunnels. For example, new design concepts are driving the need for higher Reynolds number testing, and the productivity of existing cryogenic, high-Reynolds-number wind tunnels needs to be improved by at least an OOM. The alternative is more risk that must be resolved in flight or the abandonment of potentially good concepts that entail too much unresolved risk. Current CFD technology cannot mitigate this risk. The required accuracy and throughput capabilities are simply not there, and the test data that are required to validate the CFD do not exist. Current wind-tunnel instrumentation cannot provide sufficiently detailed flow diagnostics at cryogenic conditions.

In vehicle development programs, situations arise in which wind tunnels may not be able to provide all of the necessary data, in which case CFD is used to augment the dataset to fulfill the requirements. This was the case for the Ares-I vehicle, for which wind-tunnel testing was the primary means for generating the large databases; however, for certain areas in the parameteric space CFD became indispensible [23] (e.g., line loads, large flight Reynolds numbers (up to approximately 1 billion), and plume effects). One advantage to using CFD is that it provides a complete description of surface pressure and the forces and moments on the vehicle, including all of the protuberances; current wind-tunnel instrumentation cannot provide this level of detail. For the Ares-I application, the CFD was first validated against the available wind-tunnel data, and three codes were run for selected cases to gain further confidence in CFD predictions. Finally, two CFD codes (one unstructured-grid and one overset grid) were selected to help generate the aero database for the vehicle [24, 25]. Depending upon the levels of complexities of the geometry details to be simulated, grids consisting of 70 to 200 million cells were used.

CFD is heavily used in non-aerospace applications (e.g., in the automotive, manufacturing, biomedical, food, materials, energy, and appliance industries). In these applications, the problems can sometimes be more complex in terms of geometry and grid generation; however, in many cases the accuracy requirements are not as stringent because the safety requirements are more relaxed as compared with aerospace applications, where failure in flight is virtually unallowable. Furthermore, some of the difficult issues in aerospace applications, such as transition, rarefied gas dynamics, unsteady vortical flows under high maneuver, and so on, are not considered significant in many nonaerospace applications. Nevertheless, CFD validation against experimental data is crucial, and this is essentially why, for example, automobile companies have continued to build new wind tunnels, particularly for aeroacoustic considerations. Note, however, that some racing-car companies have plans to develop cars using CFD without wind-tunnel testing.

5.0 Issues Facing CFD

Most of the required databases for aircraft design involve separated flow, for example, deployed leading- and trailing-edge flaps, spoiler deflections, large control deflections, high angles of attack, high angles of sideslip, high (transonic) Mach numbers, engine-out conditions, icing, and so on. These calculations are performed and used for initial estimates, but their accuracy is not sufficient for final design work. For example, figure 4 [26] shows a comparison of the computed wing pressure distributions at four span stations with wind-tunnel data for a multiengine configuration at a Mach number and lift coefficient that correspond to approximately a 2.5-g pull-up at the maximum design Mach number. The color plot on the right shows regions of attached and separated flow. Computed pressure coefficients compare fairly well with the experimental data at the inboard location where the flow is mostly attached, but the comparison deteriorates at the outboard locations where larger regions of separated flow are present. These results exhibit typical issues that are faced by RANS CFD at the edge of the flight envelope. What is the validity of CFD at these conditions with massive flow separation? What can be expected of the turbulence models at these conditions? Are the grid points appropriately distributed? Is a steady RANS approach sufficient, or is an unsteady RANS (URANS) or a higher fidelity computational approach needed? The validity of the wind-tunnel data at these conditions could also be questioned.

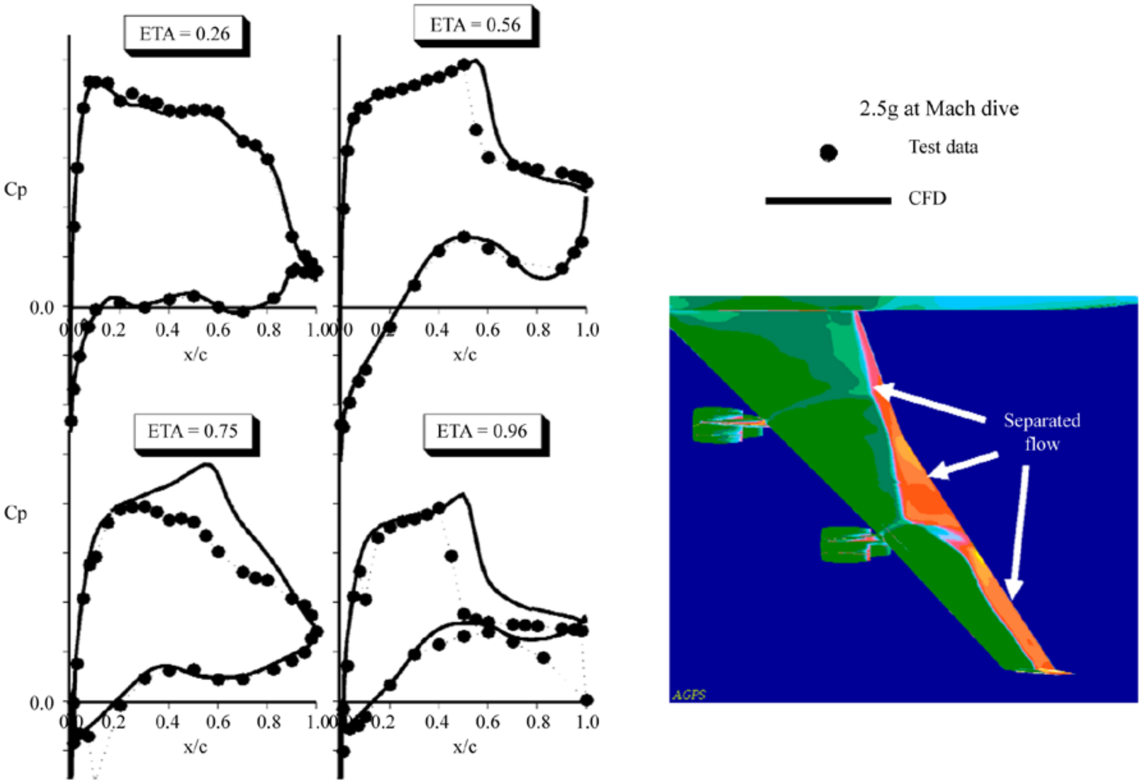


Figure 4. Computed and measured wing pressure distribution at four spanwise locations for four-engine transport aircraft. The color plot on the right shows regions of attached and separated flow.

In figure 5, surface skin friction is illustrated to identify areas of significant flow separation in a solution for an aircraft that is making an emergency descent with deployed speed brakes/spoilers. There is little confidence in the prediction of such flows and the associated loads with currently available CFD technology. The interaction of the region of separated flow with a vortex generator just inboard of the outboard nacelle makes solution convergence problematic. In the development of a large commercial transport, the determination of final aerodynamic flight loads demands a high degree of accuracy. Too much conservatism in the loads prediction can result in unacceptable excess structural weight. Underpredicting the flight loads may result in an aircraft that is not certifiable or certification to a lower takeoff gross weight with significant performance penalties until the structure is improved. CFD data can be used to determine initial structural sizing, to support wing trade studies, and to help scale the wind-tunnel database to flight conditions. However, CFD is not yet able to deliver the required degree of accuracy (low single-digit percentages over a wide range of conditions) and the throughput that is necessary to replace extensive wind-tunnel testing. Note that extrapolating low- to medium–Reynolds-number wind-tunnel data to flight also entails risk. Decades of experience with the conventional tube and wing transport configurations gives great confidence in Reynolds number extrapolation for that configuration. This would not necessarily be the case for a significantly different type of configuration. The challenge today is how best to combine the use of CFD with wind-tunnel testing to improve the prediction of aerodynamic flight loads while reducing the development cycle time.

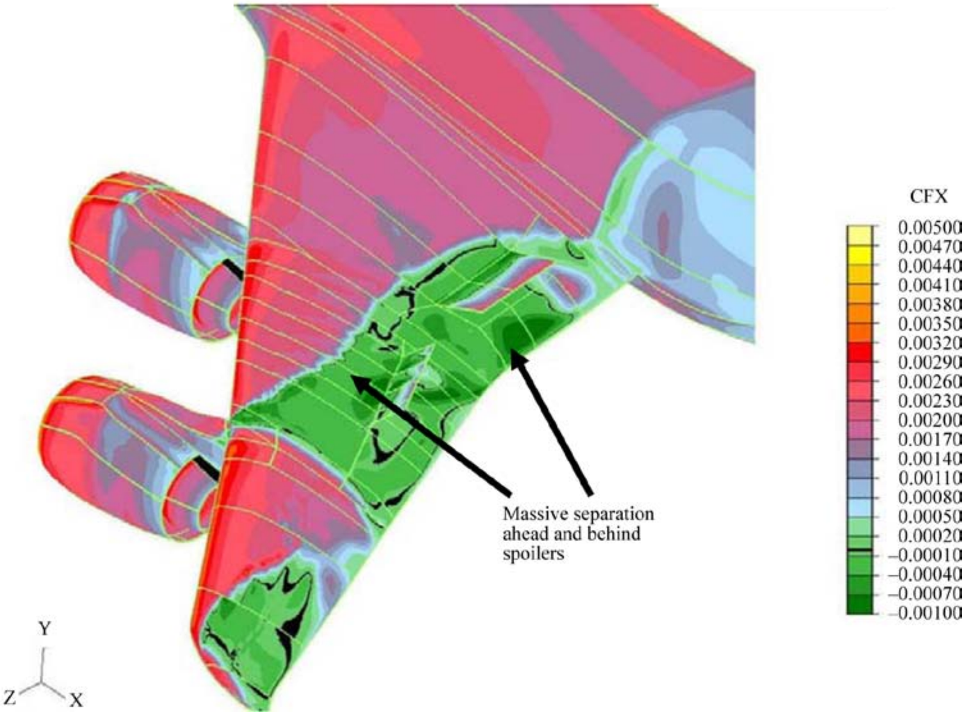


Figure 5. CFD solution (skin friction) with massive flow separation.

Today’s CFD allows the computation of highly complex flows over complex geometries. The solutions may show acceptable agreement with test data over a range of conditions but fall short over the entire flight range (e.g., see figure 6 [22]). CFD cannot be used to develop high-lift system details to maximize lift if the adequate prediction of maximum lift cannot be made! The geometries and flow physics are highly complex. Bigger, faster computers are finally providing the capability to represent these complex geometries, but computer speed alone cannot resolve the flow physics in the foreseeable future. Even with the grid in all the right places, the turbulence models are not adequate. Boundary-layer transition details are important, but these can be neither computed nor measured over the necessary range of flight conditions.

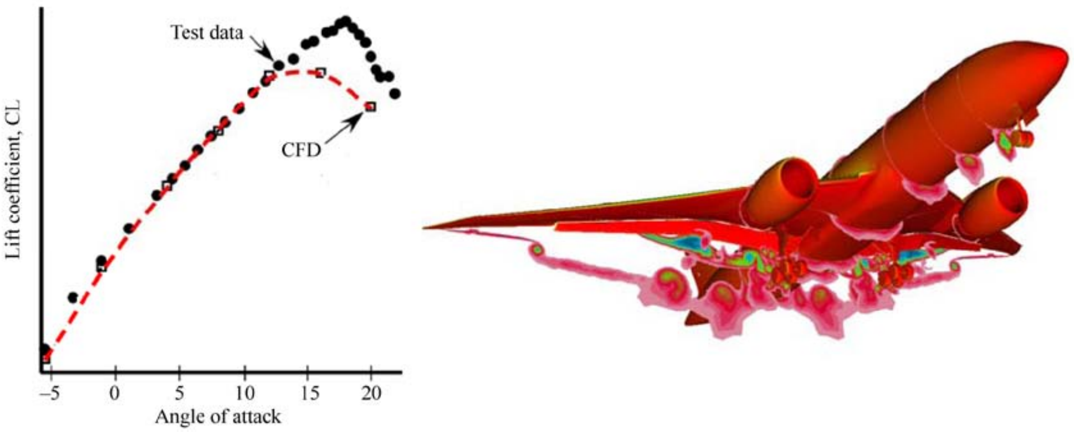


Figure 6. Computed and measured coefficient of lift (C_L) on a high-lift configuration. The right side shows a representative high-lift configuration.

Even in the early design phase, the capability to eliminate configurations that exhibit good cruise performance but that could have handling characteristics that would not be certifiable is an important one. These unacceptable handling characteristics typically occur in flight regimes that are dominated by significant flow separation. For example, Federal Aviation Administration regulations place limits on the nonlinearity of the pitching-moment characteristics of the aircraft. Modern control systems can compensate for many of these nonlinearities, provided that they can be properly identified and that control sensors and actuators are capable of providing the required compensation. Small changes in the separated-flow patterns determine whether or not these nonlinearities and the resulting handling characteristics are acceptable. Derivatives in the linear range are adequately predicted, but most critical conditions lie in the nonlinear range. The inability to capture these differences adequately is a problem for both CFD and typical low-Reynolds-number wind-tunnel testing [22]. This might also be a problem for high-Reynolds-number (flight) wind-tunnel testing, but facility limitations have prevented the exploration of this area in detail. The result is that today many of these handling quality issues must be identified and resolved in flight tests.

NASA and DoD sponsored a workshop on aerodynamic flight predictions in Williamsburg, Virginia, in November 2002, which included participants from large and small aerospace companies, NASA, the U.S. Navy, and the U.S. Air Force. The lack of robust, accurate prediction methods for aerodynamic S&C was cited as a major shortcoming in the available design methodology. One of the conclusions that was reached at this workshop was, “Prediction of the onset of separated flows across the speed range (with the attendant issues of transition prediction, turbulence modeling, unsteady flows, etc.), and the character and impact of separated flow on aircraft capabilities, is the single most critical fundamental issue to be addressed and should receive a very high priority in aerodynamic R&D programs.” Similar conclusions were reached in a follow-up symposium on computational methods for S&C, which was held in September 2003 [27]. Thus, the previous discussion on the shortcomings of CFD is fully consistent with the collective opinion of the greater aerodynamics community.

While low speed (e.g., takeoff/landing) is a problem for CFD, supersonic speed is even more difficult because of the strong shock–boundary-layer interactions, and hypersonic speed is probably the most difficult of all because of the critical roles of boundary-layer transition, heat transfer, and chemical reaction. Hypersonics is also an area where wind-tunnel testing has severe challenges (e.g., quality, cost, schedule, and data risk). These challenges result in a greater reliance on CFD, but the risk factors are quite high. As a result of these challenges in both hypersonic CFD and testing, the hypersonic community is strongly focused on flight testing as an R&D path, that is, high costs and long schedules, but greater data quality and data risk (i.e., risk in using the data to develop a viable vehicle).

Table 1, which is reprinted here from reference [16], provides a list of flow physics issues that are relevant to air vehicle design and the associated state of the art of CFD. CFD falls short in several of the flow physics situations that are encountered by vehicles in flight. After about 40 years of R&D in CFD, well-behaved attached flows on quite complex geometries can be computed; however, the details of the geometry treatment and all manner of numerical details are a work in progress even for these well-behaved flows, and the codes are not "idiot-proof" even for these cases (e.g., the drag prediction workshops discussed in section 9.0).

Table 1. Controlling Flow Physics and Issues for Air Vehicles [16]

Flow physics features and issues	Vehicle flight characteristics affected
Boundary-layer transition status and location[a]	
• Natural transition	▸ Drag
• Transition fixing (viscous simulation)	▸ Maximum lift
• Delaying transition (laminar flow control (LFC))	▸ S&C characteristics
• Laminar vs. turbulent boundary-layer separations	▸ Buffet onset
• Surface condition effects	▸ LFC effectiveness
• Wind-tunnel turbulence levels and noise	▸ Aerodynamic heating
• Relaminarization on high-lift systems in flight	▸ Heat transfer effects
Turbulent boundary-layer *attached* flows[b]	
• Reynolds-number effects:	▸ Drag (skin friction, form, and interference)
- Skin-friction levels	▸ Linear S&C characteristics
- Displacement thickness	
- Surface irregularities	
Viscous flow separation onset and progression[a]	
• Leading-edge separations	▸ Maximum lift capability
• Trailing-edge separations	▸ Drag
• Shock-wave–boundary-layer interactions	▸ Nonlinear S&C characteristics
• Juncture flow separations	▸ Buffet onset and progression characteristics
• Off-body flow reversals	▸ Flow control concept effectiveness
• Laminar vs. turbulent boundary-layer separations	▸ Spin "departure"
• Reynolds number effects	▸ Flutter
• Separation onset control/delay	
Fully separated flows[a]	
• Base flows	▸ Noise sources
• Cavities	▸ Post-stall pitching characteristics
• Wakes behind bluff bodies	▸ Post-buffet onset/progression pitch characteristics
• Post separation onset/progression	▸ Drag
	▸ Spin
Flow merging and mixing[a]	
• Multi-element high-lift system wakes and viscous layers	▸ Maximum lift capability
• Propulsive jet interactions	▸ Interference drag
	▸ S&C characteristics
Vortex phenomena[a]	
• Vortex/viscous interactions for flow control	▸ Maximum lift capability
• Wake vortex characteristics	▸ Buffet onset levels and ensuing pitch characteristics
• Interactions with downstream components	▸ Aircraft spacing at takeoff and landing
• Surface-edge effects	▸ Airframe noise levels
• Surface-sweep effects	▸ Nonlinear S&C characteristics
	▸ Undesirable unsteady flows (e.g., tail buffeting)
	▸ Rotorcraft
Shockwave characteristics[b]	
• Off-body characteristics	▸ Sonic boom for supersonic vehicles
• Bodies in proximity	▸ Aero heating
• Pressure rise and turbulence amplification through shocks	▸ Flow control concepts for reducing shock-wave drag
• Shock position/movement	
Ice accretion characteristics and effects[a]	
• Impingement limits vs. droplet size	▸ Maximum lift/stall margin
• Ice accretions (typically irregularly shaped and very rough)	▸ Drag
• Computed versus measured shapes	▸ S&C characteristics (including tail stall)
• Reynolds-number and heat-transfer effects	▸ Flight safety

[a] Not handled well by CFD.
[b] Handled well by CFD.

For complex flows that involve three-dimensional (3D) flow separation, relaxing turbulent shear flows, shock-wave/turbulence interactions, organized dynamic vorticity, transition, and so on, calculations can be generated that may be accurate enough to be useful, but this is not always the case. Major portions of this parameter space are terra incognita, and the responsibility falls to NASA to conduct research in these areas, to include the required algorithm development. The well-recognized challenges for CFD technology include:

- Aeroelastic distortion
- Boundary conditions (e.g., wind-tunnel walls, stings, and in-flight boundary conditions for higher fidelity simulations)
- Turbulence modeling (e.g., formulations, variable constants, gross shortfalls in capability)
- Predictions for drag (which are far less accurate than those for lift)
- “Untrustworthy” results near the outer portions of the flight envelope
- Boundary-layer transition location(s)/locus, subsequent “paths to turbulence”
- Discretization errors
- Mismatches in (macro and micro) geometry, computation-to-experiment and application

Another class of issues with CFD exists that is not even generally recognized, primarily because most practitioners have not delved far enough into the physics to realize that many of these are errors or problems. The issues in this class include:

- Embedded instabilities in turbulent shear flows (e.g., Görtler, Rayleigh-Taylor, Karman, Kelvin-Helmhotz, edge tones, “breathing” global instabilities in separated flows that occur at acoustic frequencies)
- Shock-capturing/shock-smearing effects upon dynamics (e.g., limiters)
- Shock dynamics and shock-turbulence interactions and their effects on turbulence amplification and spectra, including shock-dynamic waviness and oscillations

6.0 Problems with Wind-Tunnel Scaling

The issues that are associated with wind-tunnel-to-flight scaling are well-known and have been thoroughly reviewed by Bushnell [14]. Wind tunnels, to the extent practicable and where required, utilize flight-applicable values of the appropriate scaling parameters: Mach number, Reynolds number, Prandtl number, thermodynamic properties, geometric and structural dynamic similitude, Knudson number, Schmidt number, and wall-to-total-temperature ratio. Among these scaling parameters, Reynolds-number differences, which are caused by the model-scale factor as modified by various tunnel pressurization/cryogenic mitigation approaches, constitute an often major and long-standing flight-to-wind-tunnel scaling issue. Other major scaling issues include the presence of wind-tunnel walls, aeroelastic distortion differences (flight to ground), model-mounting effects, and influences of installed propulsion, among others. (See table 2.) These scaling issues can in turn be influenced by other flight-to-wind-tunnel differences, such as the effects of stream disturbances on boundary-layer transition and, hence, on Reynolds-number scaling.

Table 2. Wind-Tunnel-to-Flight Scaling Issues

1.	Wind tunnel walls: solid, porous/slotted, adaptive, open jet.
2.	Aeroelastic distortion differences: specific wind-tunnel/model conditions versus flight.
3.	Sufficient Reynolds number scaling: especially critical for transonic flows, longitudinal vortices, large transitional flow influences (separation, hypersonics).
4.	Stream disturbance fields: vorticity dynamics, acoustics, entropy spottiness, particulates, and, especially, influence(s) on transition.
5.	Model mounting influences: sting, strut, wire (e.g., rear, side).
6.	Stream gross unsteadiness, of special concern for buffet.
7.	Installed propulsion influences or lack thereof: various propulsion simulators/effects.
8.	Geometric fidelity: potential criticality of even minor differences in flight to ground, including curvatures and second derivatives, difficulties in scaling small features, boundary-layer tripping, and trip drag.
9.	Stream mean distortions/inhomogeneities.
10.	Leakage/spillage/efflux differences.
11.	Wall-to-total-temperature ratio, humidity.
12.	Differences flight to ground in instrumentation details (i.e., nature, locations, and accuracy), including variability of the various/multitudinous transition detection schemes and approaches, and data-reduction errors.
13.	High-energy, high-density effects for hypersonics.

Wind tunnels are utilized to determine an extensive variety of physical parameters for a tremendous spectrum of technological applications that involve various 3D fluid mechanics phenomena. (See table 3.) The influences of the various scaling issues (table 2) on these disparate applications and physical parameters are quite diverse. Various specialized wind tunnels have been developed to address specific applications, physical phenomena, and scaling issues. These include cryogenic nitrogen test gas tunnels for transonic, high-Reynolds-number conditions, low disturbance/quiet tunnels for transition research, and "heavy gas" tunnels for flutter.

Industry has developed rigorous testing techniques and complementary computational tools to account for many of the scaling effects, including Reynolds-number effects, wall-interference effects, support effects, and so on. However, in some cases, accounting for certain effects, such as the influence of free-stream disturbances through their effect on transition, is not possible. The ground-to-flight scaling details are often considered to be "sensitive information" by various industrial players; therefore, the available published details are often sparse or even absent. Exceptions include public projects, such as the space shuttle, the NASA high-angle-of-attack effort of the 1980s to early 1990s, and certain military fighters. These have perhaps the most complete, openly available scaling information sets. General information and reviews for ground-to-flight scaling are available in references [28–41].

In general, state-of-the-art scaling is reasonable to good for subsonic, attached, cruise flows on relatively small-Reynolds-number vehicles. Scaling accuracy degrades at high Reynolds numbers (e.g., transport aircraft), at transonic conditions, in the presence of flow separation (high-lift, stall-spin, high angle of attack), and various combinations thereof. Huge discrepancies also exist between wind-tunnel results and flight for the case of maneuvering fighters. For higher speed flows, where transition becomes a major issue for drag and heating, wind tunnels do not provide adequate information that is applicable to the flight environment, particularly for hypersonic vehicles. Thus, wind tunnels are not capable of providing all of the design data that are necessary for aircraft and aerospace vehicle development programs. CFD has an inherent role in filling these capability gaps, and this role will continue to expand as the issues discussed in section 5 above are resolved.

Table 3. Wind-Tunnel Utilization Landscape

1.	Wind-tunnel applications: Aircraft (i.e., low speed to hypersonic, small to large, short to long haul, vertical takeoff and landing/rotary wing to conventional takeoff and landing, low to high altitude); missiles/bombs/munitions; ground vehicles (i.e., cars, buses, trucks, trains); architectural aerodynamics/wind engineering (i.e., buildings, street canyons, street lights, trash cans, bridges, wind mills, shelter belts); interfacial phenomena, including wind waves, parachutes/deceleration devices, bioaerodynamics (i.e., birds/bats/insects/leaves); (deeply submerged) submarines; diffusion/environmental studies.
2.	Wind-tunnel measurements: Lift, drag, S&C; stall/spin; buffet/flutter; heat transfer; acoustics; sonic boom; vortex hazard; thrust/operability of propulsion devices; aero-optics; sensor performance; icing; store separation; ground effects; rain/dust; air or planetary gases; surface catalysis.
3.	3D fluid dynamics phenomena: Attached flows; cavity/separated flows; shock waves; plasmas; reacting flows; vortical flows; laminar/transitional/turbulent flows; multiphase flows; multicomponent flows; steady/dynamic flows; acoustics.

7.0 What Can the Increasing Computational Capability Deliver?

Computer technology is changing fast. The outlook for future developments in this area is given in appendix C, which also presents support for all statements regarding computer performance. What will the availability of ever-increasing computational power do to CFD? To answer this question, we must look back at projections that were made in the past. At least two such studies are available from Chapman [42] and Peterson et al. [43], the latter of which is cited here. Figure 7 shows the projected computer and memory requirements for turbulence physics research calculated with 1988 CFD technology and assuming a run time of 200 hr. For LES, where only the energy-carrying large-scale motions of turbulence are resolved and the small-scale motions are modeled, estimates projected that "…it is feasible, in principle, to compute the flow over a complete aircraft at a moderate Reynolds number with a machine of slightly above the teraflops class. With the current trend in the improvement of computer hardware it may well be within the reach of aircraft design engineers to use the LES approach in their design process." NASA now has a machine with a 0.5-petaflop capability, and if available numerical algorithms achieve 10 percent of the machine's peak performance, then 50-teraflop computational capability is at hand. However, no one is contemplating an LES of a wing at large Reynolds numbers, let alone the full aircraft, at least not within 200 hours of computational time, even though the CFD technology of today is far superior to that which was available in 1988. According to their estimate, an LES of a wing up to a chord Reynolds number of 30 million should be a routine computation by now, which was clearly too optimistic (as was Chapman's estimate [42] that an LES of a full aircraft would be possible during the 1990s).

Spalart et al. [44] also estimated the resources that would be required for an LES of a "clean" wing that was free of separation. For a chord Reynolds number of 10 million, a wing aspect ratio of approximately 8, and a typical leading-edge radius, Spalart et al. estimated that LES would require 10^{11} grid points under the most aggressive assumptions (Chapman [42] estimated 8×10^8

grid points for the same Reynolds number and wing aspect ratio). Spalart et al. [44] estimated that 5 million time steps would be needed for 50 chords of flow travel time, which is the time that is required for establishing a trailing vortex system to airline-industry accuracy. The total number of floating-point operations for this computation was expected to approach 10^{20}. Spalart et al. projected that this computation would be a grand challenge problem in 2045 (see ref. [45].) The difference in the above estimates (refs. [42–44]) for the required number of grid points is discussed later in section 8.2.

As also pointed out by Spalart et al. [44], their estimate of the required number of grid points to resolve turbulent eddies was too lenient, particularly for an unstructured-grid code. The estimated required computational effort for the clean wing case would be at least an OOM higher, and an accurate LES of the complete aircraft would be a much bigger challenge; perhaps, with the current CFD capability, beyond 2050.

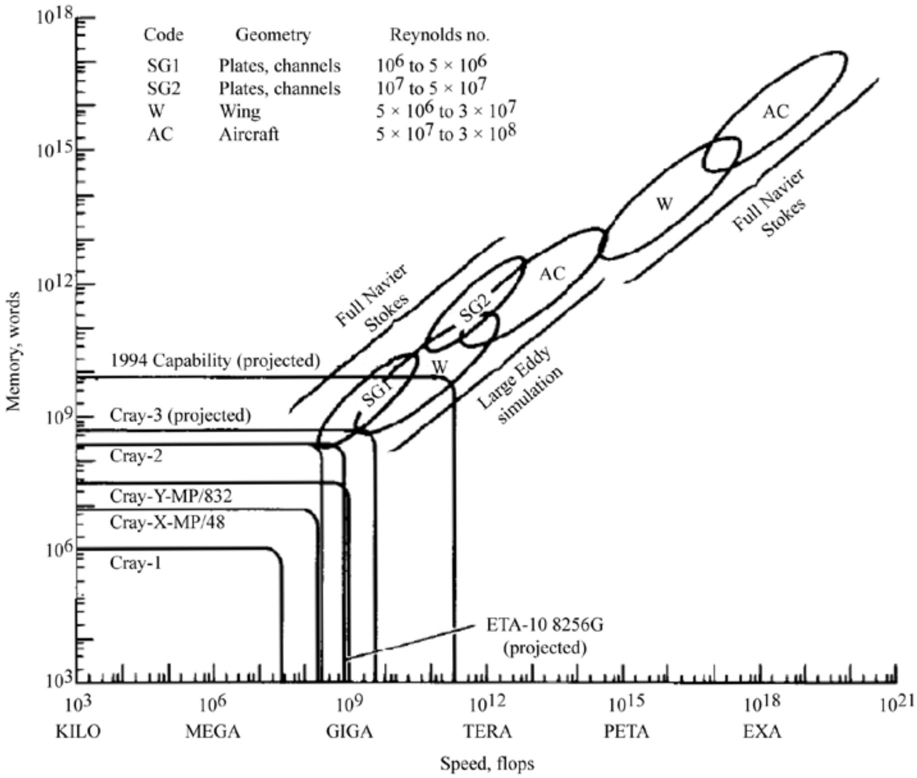


Figure 7. Computer speed and memory requirements for turbulence physics research compared with capabilities of various machines using 200-hr runs with 1988 algorithms [43].

It is important to note here that both the computer hardware and algorithmic developments have contributed to the advancements in computational science, and continued development in both areas is necessary to make progress toward the prediction of complex turbulent flows. Peterson et al. [43] noted that for the period from 1970 to 1983, reductions in solution times for the fluid dynamic equations due to improved algorithms were about the same as the reductions that resulted from faster machines. This point was also made in “Grand Challenges 1993: High-performance Computing and Communications,” published by the Federal Coordinating Council for Science and Engineering committee on physical, mathematical, and engineering sciences. Based on their report, figure 8 shows that "…the same factor of 16 × 10^6 in performance improvement that would be accumulated by Moore's Law in a 36-year span is achieved by improvements in numerical analysis for solving the electrostatic potential equation on a cube over a similar period, stretching from the demonstration that Gaussian elimination was feasible in the limited arithmetic of digital computers to the publication of the…full multigrid algorithm" [46].

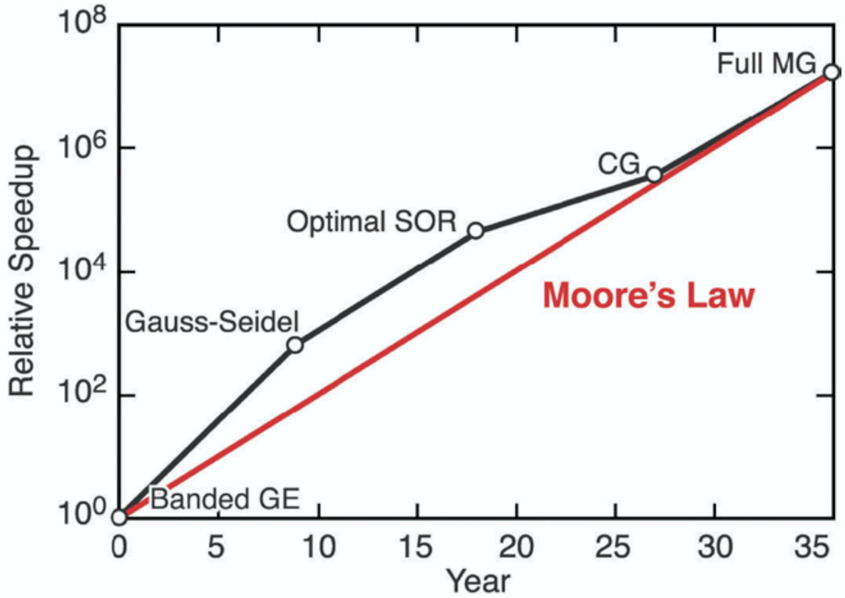


Figure 8. Relative gains of solution algorithms for solution of electrostatic potential (Poisson) equation and improvements in computer hardware (both contributed equally over a 36-year period [46]).

Computational modeling plays a critical role in all aspects of plasma physics research. Figure 9 (reprinted from ref. [47]) shows the relative speed gains that have been achieved from algorithmic improvements and faster hardware over a 30-year period for the magnetic fusion energy simulations. Clearly, algorithmic improvements played an equal, if not more important, role in large-scale simulations. This has been true for CFD simulations as well, and this trend must continue. The task of developing advanced computational algorithms is perhaps as difficult as that of developing low-energy exascale computers, but the reward is of the same OOM. The only difference is that while many technology drivers exist for developing faster computers, there is no systematic investment in the U.S. to advance the state of the art of computational science for application to fluid dynamics problems. Algorithmic developments of relevance to aeronautics applications will require substantial investments from NASA, which is critical because "Future technology advancements in aerodynamics will hinge on our ability to understand, model, and control complex, three-dimensional, unsteady viscous flow across the speed range" [48].

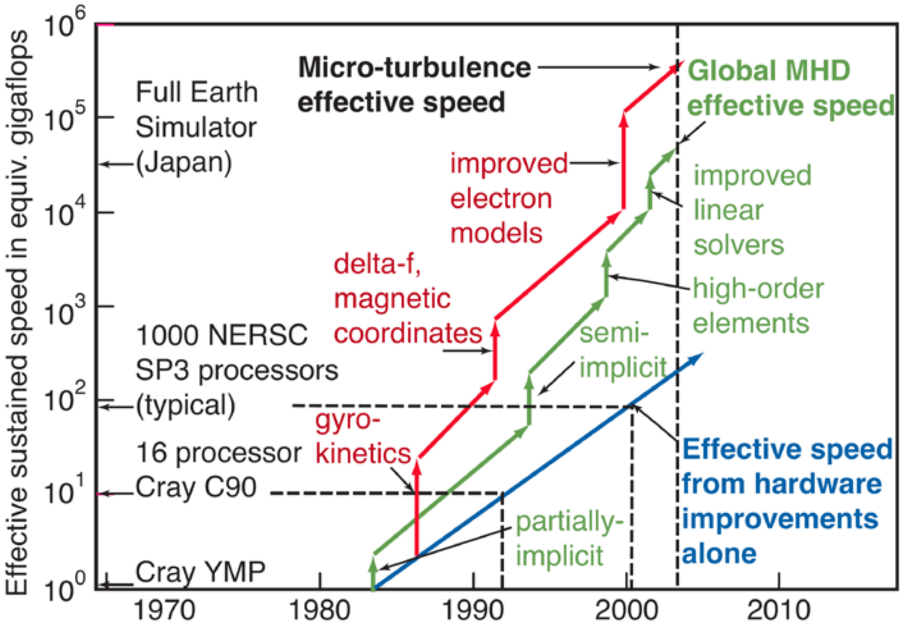


Figure 9. "Effective speed" increases in magnetic fusion energy simulations, which have resulted from both faster hardware and improved algorithms [47].

8.0 Requisite Research and Investments in CFD

Two main types of errors occur in computational fluid dynamics solutions, namely, numerical errors and physics modeling errors. Numerical errors are associated with the discretization of the governing equations and the subsequent solution of the discretized system. The physics modeling errors result due to the fact that an empirically truncated set of governing equations is solved in practice owing to the limited computational power available and incomplete knowledge of the turbulence physics. These errors depend on the level of approximation that is inherent in the physics model. Progress in the computation of aerodynamic flows of practical interest requires that research be carried out both in numerical methods development and in enhancement of the physical models. Here, we discuss both of these areas of research.

8.1 Advanced CFD Methods

A CFD simulation involves various steps, including geometry construction, grid generation, discretization (e.g., finite-difference, finite-volume, finite-element) of the Navier-Stokes equations, domain decomposition (partitioning of very large grids to achieve good parallel efficiency), solvers (efficient solution of the discretized equations), verification/validation (V&V), flow visualization, and interpretation of the computed data to obtain information of physical or engineering interest. Research is needed in many of these areas to obtain physically meaningful results from CFD simulations with an OOM reduction in the computational cost; however, only those areas with great potential, for which NASA investments are particularly needed, are discussed here. A separate section is devoted to the subject of V&V, owing to its significance in CFD.

Mark Carpenter [49] estimates that, at least theoretically, achieving algorithmic efficiencies of six OOM is possible, primarily based on advancements in three areas:

- Potential developments in the discretizations (i.e., high-order methods) could yield between 2.5 to 3 orders of improvement in capabilities of general-purpose CFD codes.
- Solver improvements could account for an additional 1.5 to 2 orders of improvement, largely a result of the careful integration of existing solver technology.
- The improvements that grid generation and adaptation could achieve depend on the scale separation of the problem that is being solved, but an additional 2 orders of improvement are possible.

While a six-OOM improvement is considered an upper limit, Carpenter believes that a 6000-fold improvement in computational capability can be had from algorithmic development (30× from discretizations, 10× from solver technology, and 20× from grid adaptation) with relatively modest investments. Note that these estimates are considered low by some researchers. David Keyes, a computational mathematician at Columbia University, together with Stephen Jardin of the Princeton Plasma Physics Laboratory, estimates a potential speedup of up to 12 OOM for fusion-related simulations, with 3 OOM resulting from hardware and the remaining 9 OOM resulting from algorithmic developments [50]. It is important to note that the three areas of potential improvement are inseparable, as the research to date indicates that high-order methods without adaptive grids and, in particular, without better solvers will be a step backward in efficiency rather than the several OOM improvements suggested above.

Because high-order methods require fewer unknowns to represent a flow field, the need for less memory and smaller, more easily partitioned grids are added benefits. The associated compact data structure requires less parallel communication, which is well suited to the projected computer architecture. On the other hand, high-order discretizations are generally stiffer and more difficult to solve than low-order discretizations and require curved grids to accurately capture curved geometries. Because the solution cost of high-order methods varies strongly with grid size, robust grid adaptation is needed to optimally distribute the grid and minimize the overall grid size. Therefore, the success of high-order-methods technology requires advancement in efficient solvers, as well as high-fidelity grid generation and adaptation. In addition, the treatment of shocks and discontinuities remains a challenge for any high-order method. Increased activity has been noted in high-order methods research in the recent past to overcome some of these difficulties (cf. [51]); of particular note are two recent European initiatives, the Adaptive Higher-Order Variational Methods for Aerodynamic Applications in Industry (ADIGMA) [52] and the Industrialization of High-Order Methods (IDIHOM) [53], for the application of high-order methods to practical aerodynamic simulations. One of the conclusions of the recently completed ADIGMA project was that grid adaptation and error estimates are key ingredients in improving the overall efficiency of high-order methods and that significant progress is required in the areas of solver efficiency, robustness, shock capturing, and computation of turbulent high-Reynolds-number flows.

Research in grid generation is generally under-valued, but it is important to note that grid generation is not a solved problem; in fact, it often turns out to be one of the main bottlenecks in CFD for accurate simulation of viscous flows over complex geometries. Several gridding approaches are available, including multiblock structured grids, Cartesian grids, unstructured polyhedral grids, and hybrid grids that contain a combination of the above. Overset grid technology offers yet another option that consists of overlapping sets of structured or unstructured grids with appropriate interpolates. A number of commercial and government-developed grid-generation software solutions are available and are commonly used in CFD simulations; however, the time that is needed to generate grids, particularly for a new geometry, generally constitutes a substantial part of the overall simulation. The adequacy of a given grid requires repeated solutions for multiples of grids to establish grid convergence, which adds to the overall cost of the solution (see ref. [54] for a discussion of the problems that are associated with grid convergence). However, an acceptable numerical solution could be obtained with a relatively small number of grid points provided that an optimal distribution of grid points is known and prescribed *a priori*. Although several good ideas have been formulated, experience has shown that new ideas can take a while to be brought to fruition. A good example is the adjoint approach for output-based grid adaptation and error estimation. The adjoint-error estimation technology was first developed in the 1990s by the finite-element community. One of the first publications in the aerospace community was that of Venditti and Darmofal [55]. FUN3D [56] and CART3D [57] have both demonstrated an inviscid grid-adaptation capability through the use of adjoints. Adjoint capability in the RANS code FUN3D was developed a number of years ago, but no production adjoint-based viscous adaptive mesh-refinement capability is available yet, mainly due to complications associated with the large aspect ratio cells that are required to resolve the boundary layer. This is a testament to the increasing software complexity that must be managed to bring many of these ideas to fruition. One may then ask: "When will a production LES capability with automatic error control be brought to a level to which it can be used in the production mode?" Some recent progress has been made in developing time-dependent adjoint methods (cf. [58]), but the need for robust, viscous adaptive grid methods cannot be overemphasized.

After the governing partial-differential equations have been discretized on a given grid, the next step is to solve these equations in a way that minimizes the time to solution. A solver is a critical part of the CFD code, and the key attributes of a desirable solver are that it must be fast (i.e., the computational cost grows no more than proportionally to the size of the problem) and robust (i.e., the computational effort does not depend strongly on the details of the inputs or the flowfield). Fast and robust solvers are needed in particular for the stiff discretizations that are generally associated with high-order methods. Areas of research include implicit and semi-implicit time-marching schemes, Krylov (e.g., restarted generalized minimal residual method (GMRES)) schemes, and *h-p* (grid-polynomial) multigrid schemes. In addition, research in new approaches to obtain the solution to flow physics problems may prove useful (e.g., refs. 59–62). The interplay of numerical solution schemes with massively parallel computers is a critical part of developing an overall solver strategy.

While high-performance computing (HPC) hardware is expected to continue to advance at a rapid rate, note that this advance in HPC hardware will not automatically translate into advances in predictive CFD, or more generally, into the multidisciplinary design optimization (MDO) capabilities that are required for the efficient design of aerospace systems. In some sense, CFD capabilities have plateaued in the last 10 to 15 years. From the 1970s to the 1990s, the transition was made from potential methods, to Euler methods, to RANS methods, and widespread adoption of unstructured grid technology has been observed. However, progress in the last decade has stalled with the use of RANS methods; LES and other, more advanced techniques (e.g., hybrid RANS/LES) have not made a sizable impact on the design process, which is primarily due to the computational cost and the lack of robustness of these advanced approaches.

NASA, to some extent, has continued to invest in leading-edge hardware, but these are used much more for increasing job throughput (e.g., dedicated computer usage for the Ares-I aerodynamic database generation and the shuttle return to flight) with no jobs taking advantage of the full hardware capabilities all at once, as in the past on vector machines. However, one recent example of a large HPC application in aeronautics can be cited. This involves use of the overset-grid RANS solver OVERFLOW [25] for the accurate computation of vortices that develop off rotorcraft blades and propagate through the flow domain [63]. This impressive three-billion grid-point simulation (the largest RANS CFD run to date) was performed for a three-blade and hub component only (one-fourth scale) of the V-22 Osprey tiltrotor aircraft on the Pleiades supercomputer at NASA Ames Research Center (ARC) using 16,384 cores (one-third of the Pleiades capacity at the time of the simulation). The results are shown in figure 10, where the computed variation in vortex core diameter is shown to tend toward the experimental data as the grid is successively refined from a baseline grid of about 14 million points up to the huge grid of a little over 3 billion points. For 40,000 integration time steps, the wall-clock time that was required for the huge grid was about 20 hr. Pleiades does not readily provide performance statistics, but OVERFLOW is estimated to have performed at no more than 10 to 15 percent of the peak performance of the machine, with performance decreasing with the number of cores due to communication bottleneck; this would be comparable to the performance of other workhorse CFD codes. If three billion points are required to simulate a three-blade and rotor flow field accurately by using unsteady RANS, one can only imagine the grid-point requirements for an LES of a full aircraft. However, further development of high-order grid-adaptive technology will dramatically reduce the grid-point requirements for such simulations.

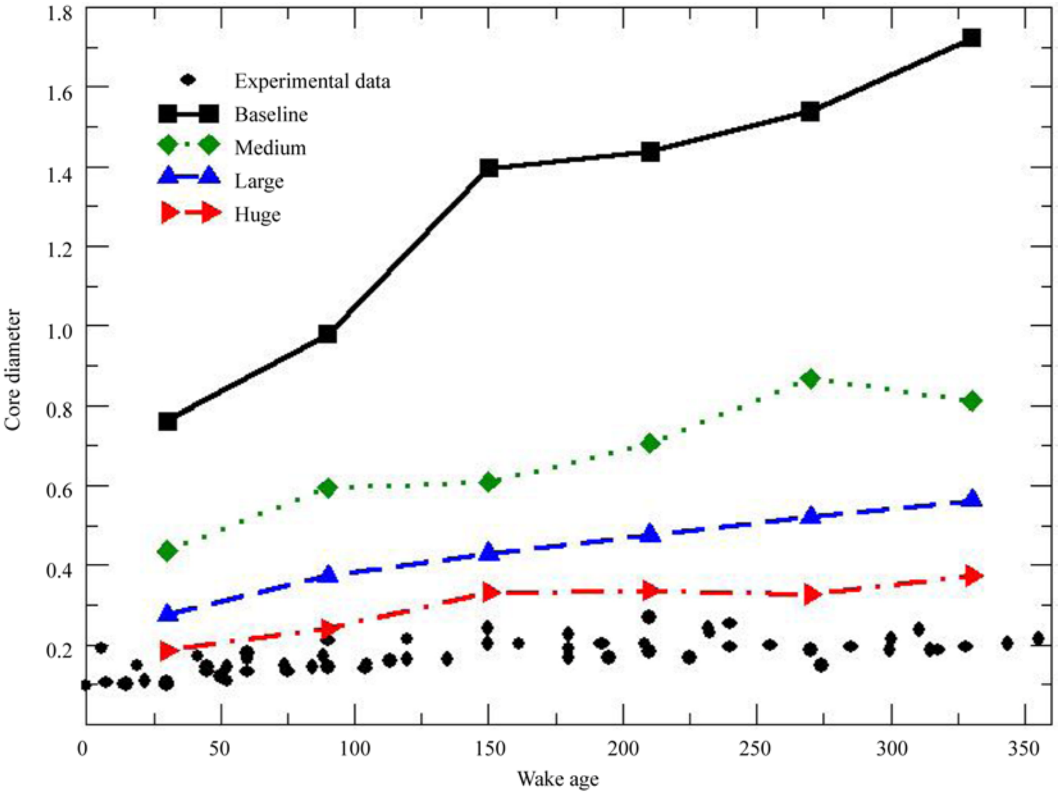


Figure 10. Vortex core diameter computed with OVERFLOW code. Computation for three-blade and rotor geometry of V-22 Osprey tiltrotor aircraft was performed on Pleiades with baseline grid of 14 million points and huge grid of three billion points [63].

Only limited exploratory studies of NASA CFD code scalability to large numbers of cores have been carried out, but the results so far have been encouraging. Figure 11 shows scaling studies for the Drag Prediction Workshop (DPW) (see section 9) and for Ares-I configurations that utilized the unstructured FUN3D code [56], up to approximately 8,500 cores [64]. The code performed almost linearly with the increasing number of cores; however, such performance gains cannot be simply extrapolated to the millions of cores that are envisioned in future hardware. New programming paradigms will need to be developed to harness the available computing power.

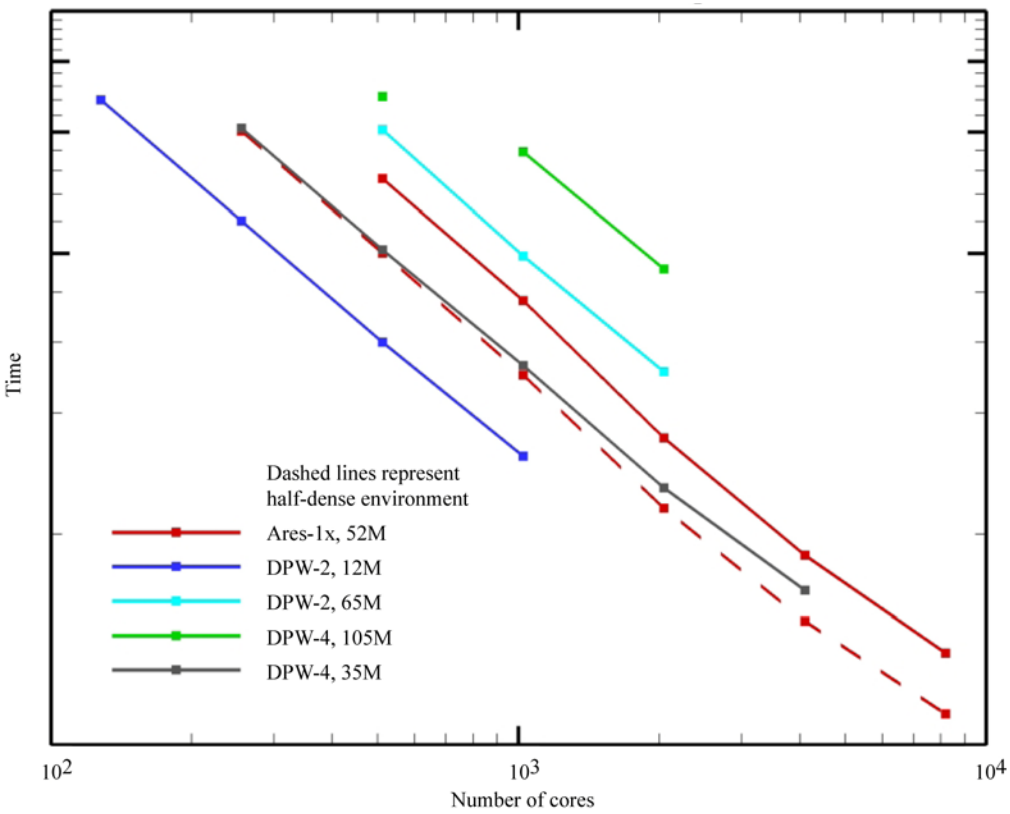


Figure 11. Parallel scalability of unstructured-grid Navier-Stokes code, FUN3D, for various applications [64].

As discussed in appendix C, an inflexion point in the advances of HPC hardware capabilities has been reached. As in the early 1990s during the transition from vector machines to highly parallel computer clusters, Moore's Law is no longer being extended by increasing the clock speed of the central processing unit (CPU) but by employing multicore architectures, with the number of cores on a chip expected to double every 18 months. This changeover is already apparent in the use of graphics processing units (GPU) for CFD computations, which contain hundreds of cores on a single GPU (ideally with many GPUs in parallel). How to take advantage of this paradigm shift in hardware architecture is not yet clear. Some suggest that this may require a complete rethinking of current programming tools and techniques. However, in the 1990s a federal government program was in place to address the transition from vector machines to parallel clusters (through which the Message Passing Interface was at least partly developed). Today in the aeronautics community, this issue is not being addressed. Thus, the risk exists that some CFD tools will remain confined to several thousand cores even when several million may be available. Algorithmic research in this area is needed now to exploit the potential computational capability that will become available in the future. To advance the state of the art of CFD and to exploit the promised potential of HPC, NASA must invest in solving computational grand challenge problems of interest to the aeronautics community [65]. The purpose of the current petaflop-type installations (e.g., the Oak Ridge National Laboratory (ORNL)) is not necessarily to increase throughput of current capabilities but to serve as a research test bed to develop future simulation tools that will run on future commodity hardware, which may mimic current large installations, albeit more cost effectively. In these areas, the aeronautics CFD industry is completely disengaged from the rest of the computational science community.

The availability of large-scale computing capability will certainly help solve the aero-database generation problem in the regime in which CFD codes are considered accurate (i.e., the attached-flow regime that is associated with cruise flight). With sufficient computational power and decreased processing time (which requires an increased emphasis on development in geometry modeling, grid-generation, and post-processing), CFD computations can be made fast enough to soon replace wind tunnels for aero-database generation for aircraft cruise design. While no flow physics issues will likely require the use of wind tunnels for this flow regime, flight certification procedures will have to be changed to eliminate the need for wind tunnels in the cruise design. A computational infrastructure, such as DoD's Computational Research and Engineering Acquisition Tools and Environments (CREATE) (see appendix E), will be needed to employ CFD technology as a digital wind tunnel for aerodynamic database generation with an equal or better efficiency than that of a physical wind tunnel.

Major research efforts are underway along many fronts to develop quantum computing, a rapidly emerging "wild card." The estimated performance of such capability is up to several tens of OOM better than the current day computers, a vast upturn in Moore's Law. Useful quantum computing is projected to be possible in a couple of decades. The availability of quantum computing would obviate the need for most physical testing, including the use of wind tunnels. Turbulence issues could be addressed via brute force in a design context for arbitrary configurations and flight regimes. While such projections are speculative at best, the quantum computing developments should be monitored carefully and plans altered accordingly because a paradigm shift in algorithmic developments will be required to exploit the potential of quantum computing in the CFD context.

8.2 Turbulence and Flow-Physics Modeling

As mentioned previously, two main types of errors occur in CFD solutions, namely, numerical errors and physics modeling errors. While the former may eventually be reduced to manageable levels in space and time with ever-increasing computer power and as high-order methods become practical, eliminating modeling errors will require the development of better physical models. Experiments indicate that the higher wave-number portion of the turbulence spectrum is universal; however, the lower wave-number region is highly problem-dependent, both in terms of detailed characteristics and level. The conventional, historical approach to turbulence in engineering flows involves RANS methodology and the construction of heuristic models of the Reynolds stresses, which appear in the equations for the mean flow. In all such modeling, an innate assumption is made that the spectrum is similar and only the level of turbulence intensity changes. The existence of discrete instabilities in turbulent flows, such as the Kelvin-Helmholtz and centrifugal instabilities, as well as the observed effects of altered boundary conditions, belies this assumption. For higher speeds that involve shock waves, a large number of direct shock-turbulence interaction amplification mechanisms significantly alter the turbulence energy spectrum and are not included in the usual RANS approaches. The assumption that similar turbulence energy spectra are embedded in turbulence models makes them incapable of capturing all of the richness of turbulence/shock interactions, including the production of dynamic vorticity from curved shocks, shock motions, and direct shock amplification. These additional discrete dynamic vorticity-production mechanisms distort and differentially amplify the spectra, and this is why the turbulence modeling constants change in complex flows.

Direct numerical simulation (DNS) of the Navier-Stokes equations captures all of the spatial and temporal scales in the flow and, thus, is independent of the modeling assumptions, but the computational cost scales as Re^3; therefore, its use will continue to be limited to simplistic geometries with an objective to obtain physical insight into transitional and turbulent flows. The proper role of DNS in turbulent-flow simulations must be that of a research tool and not a brute-force engineering tool for the characterization of aerodynamic flows, even if advancements in quantum computing routinely would allow such computations. No reason exists to compute all of the hairpin vortices and the ensuing flow structures for the innovative design of future aircraft or aerospace vehicles.

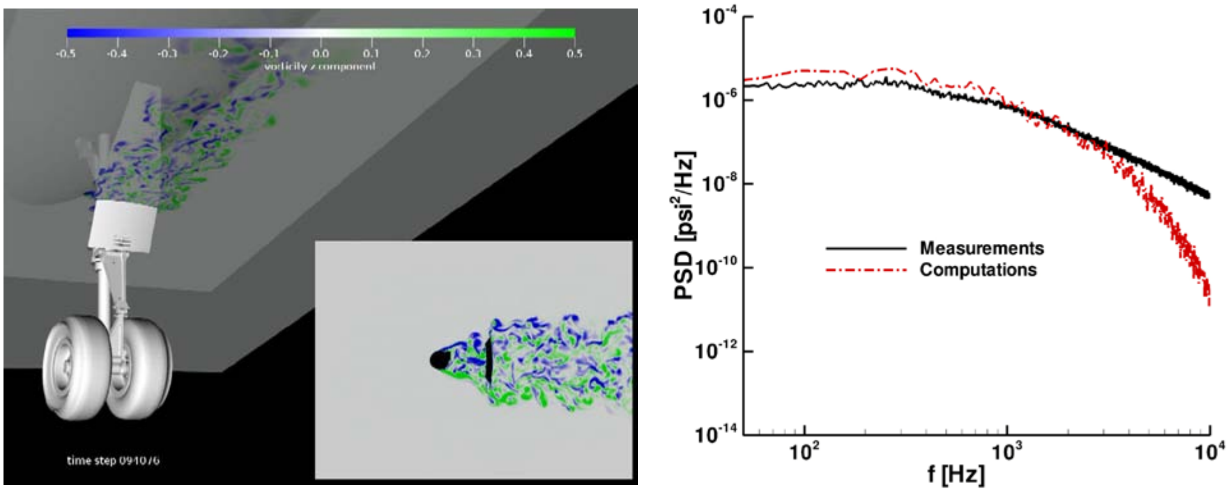
The next best thing to DNS is LES, where small scales are modeled and large scales are numerically resolved. This approach can overcome the previously discussed modeling problems by computing more of the spectrum and moving to ever-higher wave numbers as computers become faster and cheaper. This should solve the turbulence-modeling problem, as most of the flow-to-flow variability occurs at the lower wave-number portion of the spectrum and should solve the shock dynamics and the shock-turbulence interaction issues as well as the imbedded instabilities issues. However, the need exists for new algorithms and CFD codes that are tailored for dynamic problems with low dissipation and dispersion errors for practical configurations. Some fundamental questions concerning the conceptual foundations of LES and the methodologies used in its application have been addressed in reference [66].

For free-shear flows, turbulent transport processes are most affected by the large-scale (resolved) motions, and the energy cascading is from these large scales to the mostly universal small scales, which can be modeled. This is why LES is becoming the computational tool of choice for such flows. It is, for example, being used in jet-noise prediction [67, 68] with grids now approaching $O(10^9)$ points in some cases. The situation is quite different for turbulent boundary layers because of the energetic small eddies in the inner layer.

The outer layer of a turbulent boundary layer is akin to free-shear flow because of the significance of the large-scale energetic eddies in that region. The number of grid points that is required to resolve these large scales is estimated by Chapman [42] to vary as $Re^{0.4}$. However, in the near-wall region of the boundary layer, momentum transfer is affected by small eddies that scale with viscous wall units, and Chapman [42] estimated that the grid-point requirement is much more stringent ($\sim Re^{1.8}$) for a wall-resolved LES. Currently, no suitable approach is available for modeling the wall effects accurately (cf. [68]), which necessitates the use of wall-resolved LES and, hence, the stringent grid requirements that limit its applicability to small Reynolds numbers at this time. Wall modeling in the LES context is an active area of research and, if successful, will extend the applicability of LES to large Reynolds numbers. The optimistic estimates put forth in reference [43], which were reproduced in Figure 7, are based on the outer-layer scaling only. In a recent paper, Choi and Moin [69] showed that the number of grid points for high Reynolds number flows actually varies as Re for wall-modeled LES, which is much worse than the $Re^{0.4}$ variation suggested by Chapman [42] and used in reference [43]. We note that Spalart et al. [44] used the correct outer-layer scaling in their estimates and assumed that subgrid-scale-model improvements would obviate the direct resolution of near-wall small scales. They termed this approach as “true” LES and wall-resolved LES as quasi-direct numerical simulation, as the latter approach attempts to resolve the streaky sub-layer structures. The difference in the outer-layer scaling is primarily responsible for the differences in the estimated number of grid points required for LES computations given in references [42–44].

Although the rapid rate of increase in HPC capabilities is impressive, in some cases this must be tempered by the poor asymptotic requirements of many of the current methods. For example, doubling the resolution for a 3D explicit LES calculation requires a factor of 16 increase in computing power. This further emphasizes the need to develop efficient high-order methods to enable high-Reynolds-number LES computations. Because such simulations for aerodynamically relevant configurations will remain a challenge for some time to come, Spalart et al. [44] proposed an interim solution: a hybrid approach that blends the RANS approach with LES, which they termed detached-eddy simulation (DES). In this approach, RANS is used in the attached flow regions (which are generally well captured by the available turbulence models), and LES is used in the separated flow regions (where turbulence models behave poorly). Hybrid methods are extremely useful because they allow more accurate, LES-like simulations in flow regions of interest. However, critical issues exist with the interface between the RANS and LES regions and with the inherent inability of the RANS regions to feed turbulent structures into the LES regions. The manner in which the two regions are blended has been the subject of considerable research in the past decade, and various flavors of the hybrid approach have emerged (e.g., delayed DES [70], the partially-averaged Navier-Stokes approach [71], and the quasi-laminar zonal model [72]); however, none of these are fully satisfactory. This promising area of research is expected to provide a practical tool for the computational modeling of unsteady turbulent flows of industrial significance, provided a satisfactory approach to interfacing RANS with LES can be developed.

Figure 12 shows an application of the hybrid approach in which the unstructured-grid code FUN3D [56] has been used to simulate the flow field around a nose landing gear configuration [73] by employing a hybrid turbulence-modeling approach that is described in reference [74]. A total of 71 million nodes were used to obtain these results. Computations were performed on the Pleiades computer with 1200 processors. Two months were required to complete the simulation, and about 1.7 million CPU hours were required to generate the data. While the results appear to be quite good, greater resolution (perhaps three to four times greater) is needed to accurately capture the necessary details of the unsteady flow features. This higher resolution simulation would require 6 to 8 months to complete on 1200 processors. Because the FUN3D code scales linearly up to about 10,000 cores, this simulation should, in principle, require about one month to complete, but the current Pleiades scheduling policy does not permit the assignment of such a large number of cores to a single project. As we move toward exascale computing, we can reasonably expect that relatively large numbers of cores can be assigned to such projects and, therefore, hybrid RANS-LES simulations for complex configurations should become routine during the next decade.



a) Vorticity component in a plane cutting the oleo and the door.

b) Comparison of computed and measured power spectral density (PSD) at a location on the door facing oncoming flow.

Figure 12. Hybrid RANS-LES computations [73] using FUN3D code for a nose landing gear configuration.

As another alternative for the computation of complex turbulent flows, Reynolds-averaged modeling must be improved. For five decades, because computers have been capable of solving Reynolds-averaged mean-flow equations, the turbulence-modeling approaches of choice have been mainly of the simple eddy viscosity genre, which unfortunately are only suitable for turbulent flows near equilibrium (i.e., noncomplex flows). A much more complete (in terms of physics) set of closure/modeling approaches exists, generally termed as higher moment modeling with second-order closures or Reynolds-stress modeling (RSM) [75–77] as one example. These latter closures are derived via modeling at a higher level in an equation set that is calculated by taking moments of the mean-flow equations. These second-order closures are in their infancy partly due to the complexity of modeling all the required sub-models. In addition, the equation set is quite stiff and causes difficulties in the usual general-purpose RANS computational codes. In some studies, second-order closures have been applied to turbulent flows with the observation that the second-order closure results are superior [78], which indicates the efficacy of further research on both the development and application of such. As an example, the second-order closures should be able to predict the major impacts of flow curvature and rotation on the turbulence stress levels. Speziale [79] proposed an approach in which RSM provides the bridge between LES and the RANS limit for the computation of complex turbulent flows.

Currently, several key research issues exist with second-order closures. These include the refinement of the RANS solution computational machinery to handle the stiff equation issues so that second-order closures can be readily applied. In terms of the Reynolds stress modeling, major opportunities exist for improving the length-scale equations, pressure-strain modeling, wall-region treatments, and the inclusion of compressibility, including dynamic shock interaction effects. Some have suggested that the two-point correlation equations could be utilized to produce anisotropic length-scale equations. Such an approach could also conceivably aid the inclusion of elliptic/volumetric pressure-fluctuation effects in the pressure-strain modeling. Research on RSM closures has not been seriously pursued for more than a decade and requires a renaissance in effort to aid in the handling of complex turbulent flows using CFD for engineering problems in the relative near term. DNS and LES could prove useful resources for providing data to improve the weak links of the RSM. Research in other modeling approaches, such as structured-based modeling [80, 81], may also prove useful.

NASA, and the now defunct Institute for Computer Applications in Science and Engineering (ICASE), held a turbulence modeling workshop [82] in 2001; at that time, the participating turbulence model experts recommended that "… NASA should support long-term research on Algebraic Stress Models and Reynolds Stress Models. The emphasis should be placed on improving the length-scale equation, since it is the least understood and is a key component of two-equation and higher models. Second priority should be given to the development of improved near-wall models. DNS and LES would provide valuable guidance in developing and validating new RANS models." The recommendations in the present report are aligned with those of the ICASE workshop that was held more than a decade ago.

8.3 Recommended Investments

Just as wind tunnels have been closing for past several decades for the myriad of reasons mentioned above, the support for CFD research has been in steady decline for almost as long at the three NASA aeronautics research centers (Ames, Glenn, and Langley). In the 1980s and before, many branches at Langley alone were actively engaged in CFD algorithm research, but that effort has now been reduced to a small portion of a single branch. Langley established ICASE, which focused on numerical algorithms and turbulence modeling and brought in unique, world class talent, but ICASE was closed in 2003. An urgent need exists to reinvigorate research in computational science and engineering at NASA, as well as to provide access to the state-of-the art computer hardware to NASA scientists. The following broad recommendations from a DoE study [46] are also relevant to aeronautics MODSIM:

1. "Major new investments in computational science are needed in all of the mission areas. …. Such investments will extend the important scientific opportunities that have been obtained by a fusion of sustained advances in scientific models, mathematical algorithms, computer architecture, and scientific software engineering."

2. "Extensive investment in new computational facilities is strongly recommended, since simulation now cost-effectively complements experimentation in the pursuit of the answers to numerous scientific questions. New facilities should strike a balance between capability computing for those 'heroic simulations' that cannot be performed any other way, and capacity computing for 'production' simulations that contribute to the steady stream of progress."

3. "Additional investments in hardware facilities and software infrastructure should be accompanied by sustained collateral investments in algorithm research and theoretical development. Improvements in basic theory and algorithms have contributed as much to increases in computational simulation capability as improvements in hardware and software over the first six decades of scientific computing."

In order to advance the state of the art of computational technology for aeronautics applications, CFD research must progress along multiple paths, including:

- Higher moment (e.g., second-order-closure) turbulence modeling
- Accurate and robust wall modeling for LES
- Modeling of a continuous interface between RANS and LES
- High-order methods for improved solution accuracy (low-dissipation/low-dispersion schemes)
- Fast, robust solver technology
- Output-based viscous grid adaptation and error estimation
- Strategies for exploiting the potential of HPC hardware architecture

9.0 The Required Verification and Validation

The V&V of CFD simulation tools represents an important phase in the development and deployment of new computational tools. This phase has become increasingly critical as the complexity of the software and the simulated phenomena continually advances.

The task of verification constitutes "the process of determining that a computational model accurately represents the underlying mathematical model and its solution from the perspective of the intended uses of Modeling and Simulation" [83]. This task, in part, can be achieved by comparing simulation results with exact analytical results, if available (e.g., using the method of manufactured solutions [84–87]). The identification, tracking, and fixing of software glitches is also part of the verification process. Extensive effort has recently been devoted to the adoption of rigorous software engineering practices and regression testing in NASA-developed CFD software [88]. Industry now generally recognizes the need to incorporate rigorous verification techniques into CFD software development from the outset. The R&D of more sophisticated verification techniques, as well as the incorporation of these techniques as an integral part of the software development process, must be supported by NASA and other government agencies within the aerospace community. These techniques, which are typically used to verify the solution accuracy, can equally be applied to verify (and establish) the efficiency of the iterative solution. The techniques are applicable within very simple analytic settings, but their greatest potential lies with analytic solutions that are as close as possible to the actual variations of the intended applications [89, 90]. Toward this end, NASA, in partnership with academia and industry, should develop a standard set of analytic or manufactured solutions.

It is important to note that verification can only be conducted for idealized situations (linear partial differential equations, smooth solutions, uniform grids, and so on) which are never present in realistic simulations. Further, most theoretical properties are in terms of rates, such as the rate of mesh convergence or of iterative convergence, and there are few absolutes that can be "verified" in such a manner. For example: What is the effect of nonsmooth meshes? What are nonlinear effects? How do they respond to unresolved features? All of these issues impact not only accuracy but also robustness and solver convergence. In order to obtain robustness, it is essential to establish the usable range of numerical algorithms and the performance of numerical algorithms throughout that range. This cannot be done through verification but requires validation.

Validation constitutes "…the process of determining the degree to which a model or a simulation is an accurate representation of the real world from the perspective of the intended uses of the model or the simulation" [83]. The use of CFD tools by aerospace engineers will require the determination of how well these computational tools predict important design characteristics of the target aerospace vehicles. In other words, research efforts must establish to what degree these CFD tools can be trusted to replace wind-tunnel experiments for the purposes of predicting aerodynamic characteristics, or, more importantly, to what degree these tools can be trusted to predict the aerodynamic characteristics of aerospace vehicles in flight. Validation is necessary as a test of the utility of a CFD tool and should be undertaken only after verification has been established. Establishing the credibility [91] of MODSIM results is the ultimate test of the utility of any computational tool. Establishing credibility involves the following: "…verification, validation, input pedigree, results uncertainty, results robustness, use history, Modeling and Simulation management, and people qualifications" [92]. However, the present discussion is limited to V&V only.

In the aerospace industry, most companies have performed extensive validation between CFD, wind-tunnel, and flight testing and have developed a good engineering knowledge of how, where, and with what level of confidence their in-house simulation tools can be used. However, most of this information is of little use in identifying the root causes of inaccuracies in CFD codes and in contributing to the development of more capable simulation techniques and models. The R&D of more sophisticated and capable simulation techniques and models has traditionally been conducted by NASA, the DoD, other government agencies, and academia. Therefore, these agencies must take the lead in developing, performing, and disseminating CFD validation experiments that are designed to advance the state of the art in CFD simulation.

The most useful validation exercise takes a quantitative approach by seeking to establish ranges of confidence for the use of each specific CFD tool in an engineering design environment and seeking to identify discrepancies in specific simulated quantities or physical phenomena with the goal of elucidating the underlying causes of these discrepancies. As such, experiments that are designed specifically for CFD validation purposes differ substantially from traditional product-development-focused wind-tunnel or flight-test experiments. In addition to the traditional force and moment data, detailed flow physics measurements are required to provide data for CFD validation and for the identification of specific weaknesses and strengths in a simulation. This may include the use of detailed surface-pressure measurements; pressure-sensitive paint; skin-friction measurements; and off-body flow physics with the use of rakes, five-/seven-hole probes, and other advanced flow-field measurement techniques such as particle image velocimetry. CFD validation experiments may range from fundamental building-block flow-physics experiments (i.e., experiments that are suitable for validating or developing turbulence model characteristics) all the way to highly instrumented realistic aircraft configurations, such as wind-tunnel or flight-test experiments. NASA, the DoD, and the international community have supported a number of such experiments in the past, and the importance of carefully designed validation experiments will only increase in the future with the extension of simulation capabilities. Some current efforts in this area are outlined below.

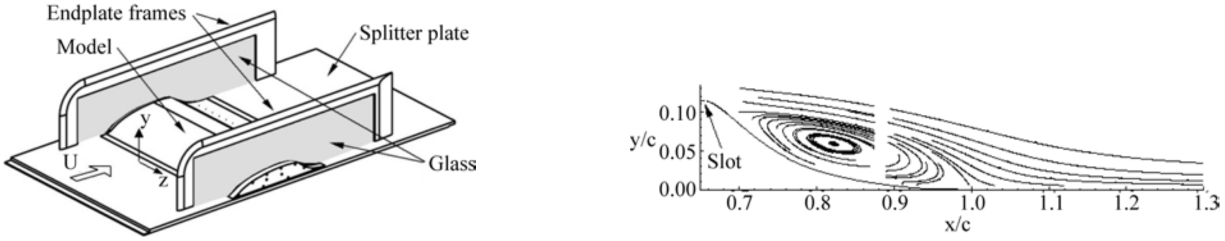
In the 1990s, NASA led an effort to understand high-lift flow physics better and to develop computational tools for predicting high lift. This effort included early flight-test experiments and detailed wind-tunnel experiments and can be credited with accelerating the development of RANS-based CFD tools for computing high-lift flows, which were essentially unavailable 20 years ago. In the early 1990s, flight tests that were conducted with the NASA Langley Boeing 737-100 gathered extensive surface-pressure measurements and skin-friction measurements on the triple-slotted flap system of the 737-100 aircraft; these measurements were used for CFD validation [93]. Quasi-two-dimensional experiments that were conducted in the NASA LaRC Low-Turbulence Pressurized Tunnel with a modern high-lift configuration (i.e., leading-edge slat with a single slotted flap), produced force, moment, and extensive off-body boundary-layer and wake-velocity profile data that were used as the basis for a two-dimensional CFD validation exercise [94]. This continues to serve as a public database for CFD high-lift validation. In 1998, a 3D high-lift configuration that was denoted as the trapezoidal wing (i.e., trap-wing) [95] was tested in both the 14- × 22-Foot Tunnel at NASA LaRC and in the 12-Foot Pressure Tunnel at NASA ARC. These experiments have been used over the years for validating NASA and industry in-house 3D high-lift CFD codes.

Since 2001, the AIAA has sponsored the DPW series, which has sought to establish the current state of the art in CFD drag-prediction capabilities for transport aircraft configurations using RANS CFD tools. To date, four workshops have been held (i.e., 2001, 2003, 2006, and 2009) with follow-on workshops currently in the planning stages. The workshop series was initiated as an unfunded grass-roots effort, with the first workshop based on published data from a Deutsches Zentrum für Luft- und Raumfahrt (DLR)-F4 generic transport aircraft configuration that was tested in several European wind tunnels [96]. DLR also provided support to build additional model elements for the F6 in DPW-III. More recently, NASA has provided increasing support with the allocation of several NTF wind-tunnel tests and, most recently, with the construction and testing of a new, Common Research Model, which served as the basis for the fourth workshop in 2009. The Common Research Model is representative of a modern wing-body-tail transport aircraft configuration and has been designed to be extensible (i.e., to include pylon-nacelle, control surfaces, and so on), and will serve as the basis for generating aerodynamic CFD validation data for years to come. The first workshop (2001), based on the DLR-F4 configuration, resulted in a large scatter of the predicted drag values that produced a standard deviation of 35 counts in predicted drag values for a core group of CFD codes. This was in sharp contrast to the four-count drag variation between the various wind-tunnel campaigns and the target one-count accuracy that is desired by the aircraft designer. The presence of a wing-root juncture region of separated flow, as well as a region of trailing-edge separation, was found to be partly responsible for the variation in predicted drag values. Subsequent workshops showed improved predictive capability, particularly for a core group of well-established aerodynamic CFD codes. The workshop series has firmly established discretization error as one of the principal sources of simulation error in aerodynamic drag prediction, has identified the accurate prediction of small regions of separation as crucial toward successful drag prediction, and has documented the steady improvement in CFD drag-prediction capabilities over the last decade. This was made possible by the use of continually increasing grid resolution (almost two OOM since the first workshop), increased computational power, improved numerical techniques, and more user experience. The drag-prediction workshop series has produced many follow-on studies [97–100] and has established a public database of experimental and computational results (including complete sets of computational meshes) which serves to standardize and simplify validation efforts and which is continually accessed by the community at large.

The Cranked Arrow Wing Aerodynamics Project International (CAWAPI) consisted of CFD validation exercise-based flight-test data that were acquired on a modified F-16 airframe (F-16-XL). This validation exercise was targeted at aerodynamic flows with dominant vortical structures, which are typical of high-performance military aircraft configurations. This work underscores the need to perform validation that is targeted for specific flow physics because the requirements for accurate simulation of transport cruise-condition aerodynamics can be vastly different from those that are required for military aircraft aerodynamics due to the different flow physics involved. The CAWAPI flight-test data consisted of an array of wing surface-pressure measurements, along with a sparser set of skin-friction measurements and some boundary-layer-rake measurements at various flight conditions. The project involved ten different participant groups using both steady and unsteady RANS and DES computational techniques and demonstrated the importance of capturing vortical flow structures for successful surface-pressure-distribution prediction. As in the DPW series, the importance of grid resolution was again shown to be a dominant source of error. One of the conclusions of the project was that both modeling errors and discretization errors contributed significantly to the uncertainty in the predictions, although the individual quantitative effects for either source of error were not determined [101, 102].

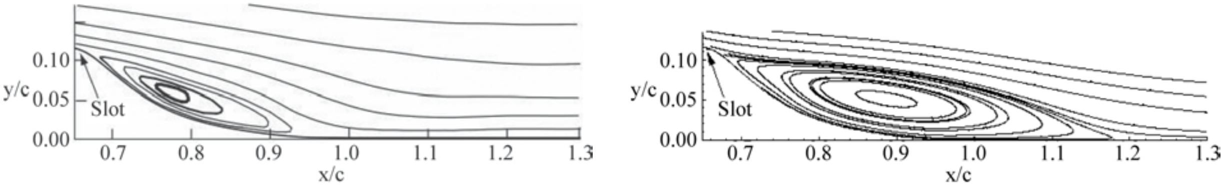
In order to assess the role of modeling errors and develop better physical models, a need exists for detailed unit experiments that involve simple configurations for which grid generation and the associated discretization errors are not the issue. An example of such an experiment is an examination of flow over a hump, which was conducted at NASA as part of the CFDVAL2004 workshop [103]. This problem also has been included as part of the online database that is maintained by the European Research Community on Flow, Turbulence, and Combustion [104]. As a result, this case has been computed by more than 15 different groups. The experimental setup consists of flow over a nominally two-dimensional wall-mounted hump (see Fig. 13(a)), with a chord Reynolds number of approximately one million both with (i.e., either steady or oscillatory suction) and without flow control. In the case with no control, flow separates on this model at about 65-percent chord and reattaches downstream of the hump in the flat-plate region. The length of the separation region was one of the metrics for comparison between the experimental and the computational results. Suction applied through a slot near the separation point was able to control the length of the separation region.

Figures 13(b) through 13(f) compare the experimental results for the case of steady suction against computations that were calculated with LES [105] and three different turbulence models [106, 107]. Experimental results show that the flow reattaches at $x/c = 0.94 \pm 0.005$. The three turbulence models that were used in the comparison included: the Spalart-Allmaras model, Menter's shear stress transport (SST) model, and the explicit algebraic stress model (based on the $k-\omega$ formulation). All three models predict the reattachment point to be at $x/c = 1.1$; therefore, in this case the computed separation length is about 50 percent longer than the experimental value. The longer separation length was attributed to significant underprediction of the eddy viscosity, which resulted in much less mixing and, hence, delayed reattachment. On the other hand, LES [105] predicted the reattachment at $x/c = 0.95$, in good agreement with the experimental results.



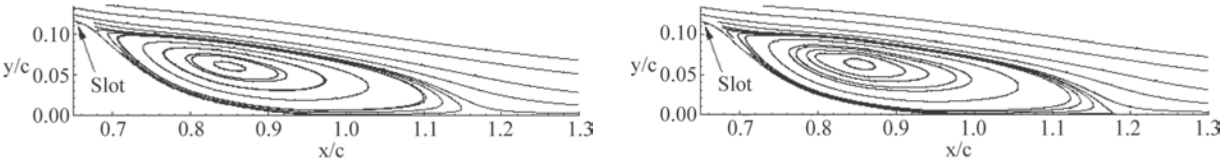
a) Experimental (hump) setup.

b) Experimental streamlines.



c) LES.

d) Spalart-Allmaras.



e) SST.

f) Algebraic stress model.

Figure 13. Comparison of experimental separation region with LES [105] and three different turbulence models [107].

More recently, data for validating airframe-noise prediction tools has been obtained at NASA for tandem cylinders [108] and a nose landing-gear configuration [109]. These two test cases are included in the Workshop on Benchmark Problems for Airframe Noise Computations, which was held in June 2010 in conjunction with the 16th AIAA Aeroacoustics Conference. Because the datasets include unsteady surface pressure distribution, these constitute a valuable resource for evaluating the unsteady flow prediction capability of CFD. In 2010, AIAA also sponsored a high-lift workshop [110] to provide an assessment of the CFD prediction capability for high-lift configurations. The focus of the workshop was the trapezoidal wing experiment [95] that is noted above.

Most needed are additional unit experiments that provide detailed data for assessing modeling errors in the available CFD technology and the development of new models for complex turbulent flows. These experiments need to be performed at sufficiently high Reynolds numbers to minimize the signature of laminar-turbulent transition and must be designed to capture the effects of curvature, pressure gradients, longitudinal vortices, shock–boundary-layer interaction, flow separation, three dimensionality, compressibility, wall temperature, and high enthalpy. In addition, turbulence-chemistry interactions and the effects of shock–mixing-layer interaction are relevant for propulsion applications. These experiments need to be designed by flow physicists (modelers), CFD practitioners, and experimentalists working in consort. Data must include mean flow and all of the Reynolds stresses, along with heat and mass-flux vectors, where applicable. These experimental data, along with available and/or future DNS, should be used to develop and evaluate second-order-closure models, as well as hybrid methods that are based on RANS and LES. Because turbulence models are not universal, the class or classes of effects that will be considered in a given experiment must be selected at the outset. Only with the aid of a collection of such experimental results would one be able to develop new physics-based turbulence models both in the Reynolds-averaged category and for the hybrid approach. This will be a long-term endeavor that will require sustained funding at an adequate level if the state of the art of CFD is to be advanced to a level to potentially replace wind tunnels as the primary tool for aircraft design.

A drastic increase in the knowledge of both the ambient and vehicle-generated disturbance fields and modifiers thereof is necessary to enable transition to be predicted as an initial-boundary-value problem. For transition, the issue is the dynamic specification of all initial and boundary conditions at least down to the level of 0.0001 percent of the mean flow. Transition processes amplify initial disturbance fields exponentially over many OOM. Such knowledge of these fields for flight, both vehicle-generated and atmospheric, is essentially absent and is also largely unknown for ground facilities. Computational tools for receptivity prediction are at a stage that, given disturbance field information, initial attempts at linking transition onset with wind-tunnel and flight disturbance environments could be made. This new capability would provide a missing link in scaling wind-tunnel transition data to flight. Therefore, detailed disturbance-field data must be collected during flight and wind-tunnel experiments.

10.0 Wind Tunnel Closing?

In the past two to three decades, a steady decline has occurred in the availability of and requirement for ground-test facilities and a steady increase has been noted in the reliance on CFD. The central question that was addressed by this virtual study team has been: “Will this trend continue, and will the role of wind-tunnel testing diminish within the next 20 years?”

Before the above question can be answered, we must note that during the stated time period (i.e., 20 years) only a few new commercial airplanes will be developed. In the absence of a major aeronautics initiative, these new airplanes will be expected to adhere to essentially the same basic design paradigm that presently exists. While some exciting possibilities are under consideration for the future, such as a blended wing-body, these designs are not expected to enter the commercial market during the given time period due to the risks that are associated with certification, public acceptance, and changes to existing infrastructure that would be required for accommodating and maintaining these aircraft.

The trend of using high-fidelity simulations throughout the design process will likely continue, which will largely be attributable to automating the simulation process through improved grid generation, better error estimates, and the incorporation of sensitivity analysis and design capabilities into the codes so that the process is less cumbersome for design engineers. Progress in both grid generation and in obtaining error estimates will be assisted by larger and faster computers because points can be more liberally distributed in well-known critical areas (i.e., leading and trailing edges) and spanwise stretching can be eliminated. This will also assist with problems that are associated with the adaptation of high-aspect-ratio cells, which is ultimately needed for the automatic control of discretization errors.

CFD-based methods are now increasingly used in the conceptual design and early analysis phases of aircraft development programs. The current state of the art of the technology is adequate for cruise conditions, but, as noted before, little confidence exists for predicting off-design conditions and for accurately predicting flows with large separation, increments in high-lift performance, and other areas of flow physics (e.g., active flow control) which are necessary to advance the current state of the art. For tactical military aircraft, enormous databases exist for S&C, store separation, and high-lift system performance; current CFD codes are unable to simulate these accurately. In a few minutes of wind-tunnel run time, a sweep of angle of attack or some other parameter can be made that currently would require on the order of tens of thousands to a million CPU hours to reproduce (with questionable accuracy) with a RANS model. RANS is almost certainly insufficient for this task over major portions of the flight envelope. As such, CFD can largely replace wind-tunnel testing for easy cases, but wind tunnels will be necessary for making measurable advances in aeronautics unless the shortfalls in the CFD capability that are discussed above are overcome. The aerospace industry continues to use wind-tunnel testing extensively, and the expectation is that this trend will continue for the foreseeable future as a result of the inadequacy of the physical models in the current CFD codes. Product complexity, flight-envelope expansion, risk aversion, and extensive flight-control systems are the additional drivers for the continued use of the wind tunnel. Another is the maturity of some parts of the aerospace vehicle design process. As technologies develop, performance improvement becomes increasingly difficult. This requires a larger volume of data and higher accuracy. The use of CFD for early concept development and phenomena investigation complements the use of experiments for gathering the large volume of data that are needed to develop any significant product. Therefore, the team does not foresee the need for wind-tunnel testing to be eliminated in the next 20 years, but does see a continued evolution of the relationship between computational modeling and experiments as both capabilities evolve. CFD also has had a significant impact on the aerodynamic design of automobiles. However, automobile companies have developed new wind tunnels primarily to improve passenger cabin noise, for which computational aeroacoustics methods are lagging significantly behind traditional CFD technology.

A case for the closure of any wind tunnel, at least in the next 20 years, cannot be made based on the premise that CFD capability will advance sufficiently to obviate the need for the wind tunnel. This is true for commercial as well as military systems. Some argue that increased computer speed is the least important parameter for determining if and when computational modeling will dramatically change the need for wind-tunnel and flight testing; furthermore, because modeling is a design tool, some prototype testing will always be necessary to uncover the flaws in the design. Prototyping is also a critical step in systems engineering. More important than the hardware and software, however, is creating the organizational processes and intellectual capital to enable use of CFD in a robust manner across all aerospace systems. This has been done to a great extent in the development of commercial systems but is lacking for military systems.

The costs of the traditional wind-tunnel approach will undoubtedly increase based on the increasing cost of keeping the testing infrastructure in place given the predicted decline in the number of programs. However, when a major aircraft program does enter a design phase, a significant jump in demand on a diverse set of wind-tunnel facilities across the nation will result. The demand is currently and will continue to be much more irregular than in the past. Thus, the cost of continuing to use the traditional design process will be significantly higher because of the cost of maintaining the testing infrastructure for times when it is needed. These cost considerations may well provide a justification for retiring certain facilities, but these must be separated from the technological considerations. Thus, the decision to close a given facility will have to be made by management based on the relevant business case.

Closure of aerospace testing facilities is not unique to the U.S. and has occurred in Europe for reasons that are similar to those faced here in the U.S.–primarily the cost of maintaining and keeping wind tunnels operational in spite of an ever-decreasing business base. Europeans also have noted the changing role of wind tunnels from the "polar machines," where aerodynamics performance data (polars) are generated, to tools for CFD validation. The old perspective of cut and try is slowly giving way to validation for the CFD experts. Facility customers are increasingly becoming more CFD oriented and, to provide CFD validation data, the new breed of wind-tunnel users are looking more into nonintrusive instrumentation and better models with more detailed geometry and finish requirements. The impact of CFD is that much less commercial transport business is available to the wind-tunnel facilities. The picture is considerably different from a military perspective, where configurations tend to be different and include many compromises to the aerodynamics for multidisciplinary reasons.

The question of closure to any specific NASA facility was not posed to the study team. However, some team members volunteered the names of the tunnels that they would definitely be interested in seeing operational in the foreseeable future. These include LaRC's NTF for its unique high-Reynolds-number capability; the Transonic Dynamic Tunnel for flutter research and testing; ARC's 11-foot Transonic Pressure Tunnel; and Glenn Research Center's Icing Tunnel. U.S. companies are expected to continue to use these facilities, as well as their own and European wind-tunnel facilities, for aircraft development in the near future.

The team agreed that the national discussion at this stage should not be about shutting down test facilities in the near future because of HPC but about how to use HPC to increase the effectiveness of the aeronautical development process by reducing the design/acquisition cycle time [111]. The question to ask is "What new computational tools and hardware should be developed alongside new testing facilities and techniques so that a complementary set of tools to best advance product development efforts and reduce risk in the future can be realized?" Other fields have been following this paradigm. For example, numerical weather prediction and climate modeling are some of the most prominent uses of HPC and are regularly the drivers for spending large sums of money on leading-edge hardware. However, at the same time, airborne measurement platforms have remained well funded and continue to provide experimental data, which, when combined with the numerical codes, result in better predictive outcomes.

Another question to ask is "What is the most effective use of computed and measured (digital and analog) aerodynamic data to reduce the cost and the risk associated with the development of new aerospace systems? In this respect, a comparison of the strengths and weakness of the two approaches for producing aerodynamic data is considered in table 4.

Table 4. Comparison of Wind Tunnel and CFD

<table>
<tr><th>Wind tunnel</th><th>CFD</th></tr>
<tr><td colspan="2">Geometry</td></tr>
<tr><td>Fidelity depends on model scale.</td><td>Full-scale fidelity as required (assuming that the computer-aided design is valid).</td></tr>
<tr><td colspan="2">Physics</td></tr>
<tr><td>True physics at wind-tunnel conditions, but cannot match flight Reynolds numbers for subsonic or supersonic flight. Cannot match flight conditions for hypersonic flight. Can introduce wind-tunnel-specific physics (e.g., effect on boundary-layer transition).</td><td>Can match flight Reynolds number and Mach.
Physics models are inadequate or not adequately validated for:
• Turbulence
• Aero-thermo-chemistry
• Boundary-layer transition
• High-altitude continuum/rarefied flow transition</td></tr>
<tr><td colspan="2">Data generation</td></tr>
<tr><td><i>De facto</i> standard for data:
Rapid data generation (after tunnel installation).

Preparation time (months/years):
• Model design and fabrication.
• Scheduling time in production wind tunnel.</td><td>Data generation time (hours, days, weeks), depending on models, resolution, and convergence.
Preparation time (hours, days, weeks).
Grid generation (depends on geometric complexity and required resolution).
Computer resource access.</td></tr>
<tr><td></td><td></td></tr>
<tr><td colspan="2">Time-dependent phenomena</td></tr>
<tr><td>Inherently captures unsteady effects, such as turbulence and separation.
Can capture low-amplitude, high-frequency, and slow model motion.</td><td>Turbulence models address average properties. For LES/DES models, large computing resource requirements preclude production use on large configurations.
Captures general unsteady motion for short periods of time (few seconds) because of large demand on computing resources.</td></tr>
<tr><td colspan="2">Simulation artifacts</td></tr>
<tr><td>Facility effects can affect data accuracy or require empirical corrections.
• Support interference and associated model geometry compromises.
• Wall interference.
• Flow quality.
• Measurement uncertainty.</td><td>Resolution (i.e., grid effects).
Numerical uncertainty (currently difficult to quantify).</td></tr>
<tr><td colspan="2">Cost</td></tr>
<tr><td colspan="2">The difference in cost to produce a data point between these two approaches is narrowing as CFD becomes more accurate and incorporates more fidelity into physics models. The large computing resources that will be required to make “first principle” simulations are expected to soon rival the cost of wind-tunnel facilities. Because one method is not demonstrably superior in all aspects, consideration of how to use the strengths of each is a better question to explore at this time.</td></tr>
</table>

The integration of wind-tunnel testing and CFD would require the systematic mutual V&V of analog/digital simulations. More effective use of wind-tunnel facilities would require the following modifications to current testing practices:

- Design of experiments to reduce test matrix size.
- Increase in high-density measurements, especially for non-intrusive methods such as pressure-sensitive paint, shear-sensitive paint, particle image velocimetry, and so on.
- Inclusion of more aeroelastic testing and capture of model geometry data under loads (e.g., photogrammetry) to enable comparison with CFD.
- Inclusion of hardware in the loop test methods to improve subsystem integration.

We must create a vision of what the needs will be in 2030 f or aeronautical systems development, while recognizing that demand will have diminished (but will still be nonzero). This may be the time to “cash in” the current aging infrastructure to build a test facility that is designed to fully implement advances in computational science and engineering and advanced diagnostic tools. Such a facility could be energy efficient and “green” as well. In fact, a vision for a 2025+ wind-tunnel testing facility was recently presented by Steinle, Mickle, and Mills [112]. On a longer term horizon, however, the role of CFD and wind tunnels will continue to shift in favor of the former (see appendix F).

11.0 Conclusions and Recommendations

As a result of the continued availability of faster and cheaper computing hardware, the role of CFD in the aircraft design process has increased significantly over the past few decades. This trend is expected to continue, and CFD will soon be able to replace wind-tunnel testing for attached-flow cruise conditions. However, the need for wind-tunnel experiments will continue for off-design conditions, which greatly influence the design of an aircraft, and for any truly new configurations. For CFD to eventually replace wind-tunnel testing, significant investment will be required in modeling efforts to allow complex physical phenomena to be predicted with confidence. To achieve that goal, a significant amount of laboratory testing will be required to validate the physical models. Thus, the most important question is how to best deploy resources to achieve the maximum potential increase in predictive capability with both computational and experimental facilities. This must occur in an environment in which the total industry need for wind-tunnel infrastructure will continue to decrease largely based on a lack of R&D anticipated in both the commercial and military arenas. Global competition and economics will continue to put pressure on national development activities and investments. A national strategy that focuses investment in both the computational and experimental arenas to optimize both capability and availability is essential.

The current debate should be on how to integrate CFD and wind-tunnel testing technologies comprehensively to bring more value to the product-development cycle. As CFD continues to become more reliable without the aid of wind-tunnel testing, the wind tunnels can be decommissioned.

The team has made important observations about the limitations of CFD simulations in addressing the as-yet unsolved problems of accurate prediction of laminar-turbulent transition and the computation of complex turbulent flows. The issues that are related to the inadequacy of physical models are well-known and have been thoroughly articulated in this report. The issues regarding the difficulty of generating high-accuracy grids and coupling them with flow solvers have been mentioned. Major progress in the accurate computation of turbulent flows will come from turbulence-resolving regions (i.e., direct simulation of large eddies). Because the required computing power increases with the fourth power of resolution that is required by LES, brute-force application of computing power will not be sufficient to accurately simulate turbulence for another 40 to 50 years. Research is needed for the advancement of CFD algorithms with respect to accuracy, speed, and robustness, as well as in the development of advanced turbulence models.

The findings of this study can be summarized as follows:

1) Except for flight-certification issues, CFD will soon be ready to replace wind tunnels for the attached-flow cruise conditions.

2) With another 10 to 20 years of Moore's Law advancements and assuming that research in relevant numerical and flow physics issues is successful in developing an accurate turbulence-modeling approach for complex flows, CFD could primarily replace wind tunnels for off-design conditions.

3) If an accurate turbulence-modeling construct is not developed, several more decades of Moore's Law advancements or a paradigm shift as claimed by quantum computing may be necessary to finally successfully compute turbulence. This is, of course, subject to the availability of the required initial and boundary conditions.

4) To shorten the aircraft and aerospace vehicle design cycle, enable efficient and economical study and development of advanced, nontraditional design concepts, and reduce the costs associated with physical testing/infrastructure, an aggressive research program in turbulence modeling and CFD algorithmic/numerical/hardware architecture issues, including experimental validation, is required. Specifically, research needs to be conducted in the following areas:
    - Higher moment (e.g., second-order-closure) turbulence modeling.
    - Accurate wall modeling for LES.
    - Modeling of a continuous interface between RANS and LES.
    - High-order methods for low dissipation/dispersion schemes.
    - Fast, robust solver technology.
    - Output-based 3D viscous grid adaptation and error estimation.
    - Strategies for exploiting the potential of future computer hardware.
    - Carefully designed experiments to aid in the development of physical models and CFD validation.

5) The prediction of boundary-layer transition requires considerable data acquisition regarding the initial and boundary disturbance fields, both ambient and vehicle produced. Therefore, such data must be collected in both flight and wind-tunnel experiments.

The body of this report generally has been concerned with "conventional" Moore's Law computing developments. However, the availability of quantum computing could obviate the need for most physical testing, including the application of wind tunnels. Turbulence issues could be addressed via brute force in a design context for arbitrary configurations and flight regimes. While such projections are speculative at best, developments in quantum computing should be carefully monitored and plans altered accordingly.

12.0 References

1. Balls, D. D. and Corliss, W. R.: "Wind Tunnels of NASA," NASA SP-440, 1981.
2. Lee, J. L.: "Into the Wind: A History of the American Wind Tunnel 1896–1941," PhD Thesis, Department of History, Auburn University, Auburn, AL, 2001.
3. Smelt, R.: *Review of Aeronautical Wind Tunnel Facilities*, National Academy Press, city, state, 1988.
4. Mack, M. D. and McMasters, J. H.: "High Reynolds Number Testing in Support of Transport Air-plane Development," AIAA Paper 92-3982, 1992.
5. Chambers, J. R.: "Concept to Reality: Contributions of the Langley Research Center to U.S. Civil Aircraft of the 1990s," NASA SP-2003-4529, June 2003.
6. Chambers, J. R.: "Partners in Freedom: Contributions of the NASA Langley Research Center to U.S. Military Aircraft of the 1990's," Monograph in Aerospace History Number 19, The NASA History Series, NASA SP-2000-4519, October 2000.
7. Campbell, J. F.: "The National Transonic Facility: A Research Prospective," AIAA Paper 84-2150, 1984.
8. Wahls, R. A.: "The National Transonic Facility: A Research Perspective," AIAA Paper 2001-0754, January 2001.
9. Hatrzuiker, J. P.: "European Transonic Wind Tunnel ETW: Design Concepts and Plans," AIAA Paper 86-0731, 1986.
10. European Transonic Windtunnel Web site, URL: http://www.etw.de/, accessed on January 29, 2012.
11. Beach, H. L., Jr. and Bolino, J. V.: "National Planning for Aeronautical Test Facilities," AIAA Paper 94-2474, 1994.
12. ARMD NRA: Advanced Concepts Studies Awardees, URL: http://www.aeronautics.nasa.gov/nra_awardees_10_06_08.htm, accessed on January 29, 2012.
13. King, D. R.: "The American Aircraft Industrial Base; On the Brink," *Air and Space Power Journal,* URL: www.airpower.au.af.mil/airchronicles/apj/apj06/spr06/king.html, March 2006, accessed on January 29, 2012.
14. Bushnell, D. M.: "Scaling: Wind Tunnel to Flight," *Annual Review of Fluid Mechanics*, Vol. 38, 2006, pp. 111–128.
15. Glotzer, S. C.; Kim, S.; Cumings, P. T.; Deshmukh, A.; Head-Gordon, M.; Karniadakis, G.; Pezold, L.; Sagui, C.; and Shinozuka, M.: "International Assessment of Research and Development In Simulation-Based Engineering and Science," World Technology Evaluation

Center, Inc., URL: http://www.wtec.org/sbes/SBES-GlobalFinalReport.pdf, 2009, accessed on January 29, 2012.

16. Anton, P. S.; Johnson, D. J.; Block, M.; Brown, M.; Drezner, J.; Dryden, J.; Gritton, E. C.; Hamilton, T.; Hogan, T.; Mesic, R.; Peetz, D.; Raman, R.; Steinberg, P.; Strong, J.; and Trimble, W.: "Wind Tunnel and Propulsion Test Facilities: Supporting Analyses to an Assessment of NASA's Capabilities to Serve National Needs," The RAND Corporation Technical Report TR-134, 2004.
17. Rubbert, P. E.: "AIAA Wright Brothers Lecture: CFD and the Changing World of Airplane Design," ICAS-94-0.2, September 1994.
18. Tinoco, E. N.: "The Changing Role of Computational Fluid Dynamics in Aircraft Development," AIAA-98-2512, June 1998.
19. Johnson, F. T.; Tinoco E. N.; and Yu, N. J.: "Thirty Years of Development and Application of CFD at Boeing Commercial Airplanes, Seattle," *Computers & Fluids*, Vol. 34, 2005, pp. 1115-1151.
20. Goldhammer, M. I.: "Boeing 787—Design for Optimal Airplane Performance," *CEAS/KATnet Conference on Key Aerodynamic Technologies*, Bremen, Germany, June 2005.
21. Gillette, W.: *Aerospace America*, AIAA, Reston, VA, 2004, p. 34.
22. Tinoco, E. N.; Bogue, D. R.; Kao, T.-J.; Yu, N. J.; Li, P.; and Ball, D. N.: "Progress Toward CFD for Full Flight Envelope," *The Aeronautical Journal*, Vol. 109, Oct. 2005, pp. 451-460.
23. Hall, R.: Private Communication, NASA Langley Research Center, 2009.
24. Abdol-Hamid, K. S.; Ghaffari, F.; and Parlette, E. B.: "Overview of Ares-I CFD Ascent Aerodynamic Data Development and Analysis based on USM3D," AIAA 2011-15, 2011.
25. Gusman, M.; Housman, J.; and Kiris, C.: "Best Practices for CFD Simulations of Launch Vehicle Ascent with Plumes: OVERFLOW Perspective," AIAA 2011-1054, 2011.
26. Ball, D.: "Recent Applications of CFD to the Design of Boeing Commercial Transports," *HPC User Forum*, Roanoke, VA, URL: http://www.hpcuserforum.com/presentations/April2009Roanoke/BOEINGApplicationsofCFD.pdf, April 2009, accessed on January 29, 2012.
27. Fremaux, C. M. and Hall, R. M.: *COMSAC: Computational Methods for Stability and Control*, NASA/CP-2004-213028, Parts 1 and 2, 2004.
28. AGARD Fluid Mechanics Panel: "Flight/Ground Testing Facility Correlation," AGARD-CP-339, 1983.
29. AGARD Fluid Mechanics Panel: "Ground/Flight Test Techniques and Correlation," AGARD-CP-339, 1983.
30. Bowers, G. M.: "Aircraft Lift and Drag Prediction and Measurement: Prediction Methods for Aircraft Aerodynamic Characteristics," AGARD-LS-67, Paper 4, 1974.
31. Williams, J.: "Technical Evaluation Report on the Fluid Mechanics Panel Symposium on Ground/Flight Test Techniques and Correlation," AGARD-AR-191, 1983.

32. Whitfield, J. D.; Griffith, B. J.; Bang, C.; and Butler, R. W.: "Overview of Flight and Ground Testing with Emphasis on the Wind Tunnnel," AIAA Paper 81-2474, 1981.

33. McDonald, H.; Ross, J.; Driver, D.; and Smith, S.: "Wind Tunnels and Flight," *Proc. 10th Int. Symp. Appl. Laser Tech. Fluid Mech.*, Lisbon, Portugal, July 10–13, 2000.

34. McKinney, L. W. and Baals, D. D. (Eds.): "Wind Tunnel/Flight Correlation – 1981" NASA CP 2225, 1982.

35. Haines, A. B.: "Prediction of Scale Effects at Transonic Speeds: Current Practice and a Gaze into the Future," *Aeronautical Journal*, September 2000, pp. 421–431.

36. Dresser, H. S.; Newberry, C. F.; Surber, T. E.; Szema, K. Y.; and Chakravarthy, S. R.: "A Comparison of CFD, Wind Tunnel and Flight Pressures on the Space Shuttle Orbiter Payload Doors," AIAA Paper 90-3213, 1990.

37. Hamilton, J. T.; Wallace, R. O.; and Dill, C. C.: "Launch Vehicle Aerodynamic Data Base Development Comparison with Flight DATA," NASA CP-2283, Pt. 1, 1983, pp. 19–36.

38. Haney, J. W.: "Orbiter Entry Heating Lessons Learned from Development Flight Test Program," NASA CP-2283, Pt. 2, 1983, pp. 719–751.

39. Banks, D. W.; Fisher, D. F.; Hall, R. M.; Erickson, G.E.; Murri, D. G.; Grafton, S. B.; and Sewall, W. G.: "The F/A-18 High Angle of Attack Ground-to-Flight Correlation: Lessons Learned," NASA TM-4783, 1997.

40. Cites, R.; Rueger, M.; Evans, D.; and Lehman, S.: "Comparison of Transonic Wind Tunnel and Flight Data for High Performance Aircraft," AIAA Paper 92-3983, 1992.

41. Fisher, D. F. and Dougherty, N. S., Jr.: "Flight and Wind-Tunnel Correlation of Boundary Layer Transition on the AEDC Transition Cone," NASA TM-84902, 1982.

42. Chapman, D. R.: "Computational Aerodynamics Development and Outlook," *AIAA Journal*, Vol. 17, No. 12, December 1979, pp. 1293–1313.

43. Peterson, V. L.; Kim, J.; Holst, T. L.; Deiwert, G. S.; Cooper, D. M.; Watson, A. B.; and Bailey, R. F..: "Supercomputer Requirements for Selected Disciplines Important to Aerospace," *Proceedings of the IEEE*, Vol. 77, No. 7, July 1989, pp. 1038–1055.

44. Spalart, P. R.; Jou, W-H.; Strelets, M.; and Allmaras, S. R.: "Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach," *First AFOSR International Conference on DNS/LES*, Ruston, Louisiana, August 1997.

45. Spalart, P. R.: "Strategies for Turbulence Modeling and Simulations," *International Journal of Heat and Fluid Flow*, Vol. 21, 2000, pp. 252–263.

46. Office of Science: "A Science-Based Case for Large-Scale Simulation," Vol. 1, U.S. Department of Energy, 2003.

47. Office of Science: "A Science-Based Case for Large-Scale Simulation," Vol. 2, U. S. Department of Energy, 2004.

48. Kumar, A. and Hefner, J. N.: "Future Challenges and Opportunities in Aerodynamics," ICAS 2000-0.2.13, 2000.

49. Carpenter, M.: Private Communication, NASA Langley Research Center, 2009.

50. Sipics, M.: "Taking on the ITER Challenge, Scientists Look to Innovative Algorithms, Petascale Computers," *SIAM News*, Vol. 39, No. 7, 2006, p. 1.

51. Barter, G. E. and Darmofal, D. L.: "Shock Capturing with High-Order, PDE-Based Artificial Viscosity," AIAA 2007-3823, 2007.

52. Kroll, N.; Bieler, H.; Deconinck, H.; Couallier, V.; van der Ven, H.; and Sorensen, K. (Eds.): "ADIGMA – A European Initiative on the Development of Adaptive High-Order Variational Methods for Aerospace Applications," Springer, 2010.

53. Institute of Aerodynamics and Flow Technology: URL: http://www.dlr.de/as/en/desktopdefault.aspx/tabid-7027/11654_read-27492/, accessed on January 29, 2012.

54. Salas, M. D. and Atkins, H.: "Some Observations on Grid Convergence," *Computers & Fluids*, Vol. 35, 2006, pp. 688–692.

55. Venditti, D. A. and Darmofal, D. L.: "Anisotropic Grid Adaptation for Functional Outputs: Application to Two-dimensional Viscous Flows," *Journal of Computational Physics*, Vol. 187, No. 1, 2003, pp. 22–46.

56. Anonymous: "FUN3D: Fully Unstructured Navier-Stokes," User Manual for FUN3D, Ver. 11.6, URL: http://fun3d.larc.nasa.gov/, National Aeronautics and Space Administration, accessed on January 29, 2012.

57. Anonymous: "Cart3D: Software and User Manual for Cart3D," Ver. 1.4, URL: http://people.nas.nasa.gov/~aftosmis/cart3d/, National Aeronautics and Space Administration, accessed on January 29, 2012.

58. Nielsen, E.J.; Diskin, B.; and Yamaleev, N.K.: "Discrete Adjoint-Based Design Optimization of Unsteady Turbulent Flows on Dynamic Unstructured Grids," *AIAA Journal*, Vol. 48, No. 6, 2010, pp. 1195–1206.

59. Fares, E. and Nolting, S.: "Unsteady Flow Simulation of a High-Lift configuration Using a Lattice Boltzmann Approach," AIAA 2011-869, 2011.

60. Nishikawa, H.: "New-Generation Hyperbolic Navier-Stokes Schemes: O(1/h) Speed-Up and Accurate Viscous/Heat Fluxes," AIAA 2011-3043, 2011.

61. Gnoffo, P. A.: "Multi-Dimensional, Inviscid Flux Reconstruction for Simulation of Hypersonic Heating on Tetrahederal Grids," AIAA 2009-599, 2009.

62. Chang, S.-C.: "A New Approach for Constructing Highly Stable High Order CESE Schemes," AIAA 2010-543, 2010.

63. Pulliam, T. H. and Jespersen, D. C.: "Large Scale Aerodynamic Calculation on Pleiades," Presented at the *21st International Conference on Parallel Computational Fluid Dynamics*, Moffett Field, California, May 18–22, 2009.

64. Nielson, E. and Hammond, D.: Private Communication, NASA Langley Research Center, 2009.

65. Mavriplis, D.; Keyes, D.; and Turner, M.: "Petaflops Opportunities for the NASA Fundamental Aeronautics Program," AIAA Paper No. 2007-4084, 2007.

66. Pope, S. B.: "Ten Questions Concerning the Large-Eddy Simulation of Turbulent Flows," *New Journal of Physics*, Vol. 6, No. 35, 2004, DOI: 10.1088/1367-2630/6/1/035.

67. Nichols, J. W.; Ham, F. E.; and Lele, S.K.: "High Fidelity Large-Eddy Simulation for Supersonic Rectangular Jet Noise Prediction," AIAA-2011-2919, 2011.

68. Uzun, A. and Hussaini, M. Y.: "Simulation of Noise Generation in Near-Nozzle Region of a Chevron Nozzle Jet," *AIAA Journal*, Vol. 47, No. 9, 2009, pp. 1793–1810.

69. Choi, H. and Moin, P.: "Grid-Point Requirements for Large Eddy Simulation: Chapman's Estimates Revisited," *Physics of Fluids*, Vol. 24, 2012, 011702-1.

70. Spalart, P. R.: "Detached-Eddy Simulation," *Annual Review of Fluid Mechanics*, Vol. 41, pp. 181–202, 2009.

71. Girimaji, S. S.; Jeong, E.; and Srinivasan, R.: "Partially Averaged Navier-Stokes Method for Turbulence: Fixed Point Analysis and Comparison With Unsteady Partially Averaged Navier-Stokes," *Journal of Applied Mechanics*, Vol. 73, No. 3, 2006, pp. 422–429.

72. Khorrami, M. R.; Lockard, D. L.; Choudhari, M. M.; Jenkins, L. N.; Neuhart, D. H.; and McGinley, C. B.: "Simulations of Bluff Body Flow Interaction for Noise Source Modeling," AIAA 2006-3203, 2006.

73. Vatsa, V. N.; Lockard, D. P.; and Khorrami, M. R.: "Application of FUN3D Solver for Aeroacoustics Simulation of a Nose Landing Gear Configuration," AIAA 2011-2820, 2011.

74. Lynch, C. and Smith, M. J.: "Hybrid RANS-LES Turbulence Models on Unstructured Grids," AIAA Paper 2008-3854, 2008.

75. Launder, B. E.; Reece, G. J.; and Rodi, W.: "Progress in the Development of A Reynolds-Stress Turbulence Closure," *Journal of Fluid Mechanics*, Vol. 68, 1975, pp. 537–566.

76. Speziale, C. G.; Sarkar, S.; and Gatski, T. B.: "Modeling the Pressure-Strain Correlation of Turbulence: An Invariant Dynamical Systems Approach," *Journal of Fluid Mechanics*, Vol. 227, 1991, pp. 245–272.

77. Hanjalic, K. and Launder, B: *Modelling Turbulence in Engineering and the Environment*, Cambridge University Press, Cambridge, UK, 2011.

78. Eisfeld, B.: "Reynolds Stress Modeling for Complex Aerodynamic Flows," Presented at the *European Conference on Computational Fluid Dynamics, ECCOMAS CFD 2010*, Lisbon, Portugal, June 14–17, 2010.

79. Speziale, C. G.: "Turbulence Modeling for Time-Dependent RANS and VLES: A Review." *AIAA Journal*, Vol. 36, No. 2, 1998, pp. 173–184.

80. Kassinos, S. C.; Langer, C. A.; Haire, S. L.; and Reynolds, W. C.: "Structure Based Turbulence Modeling for Turbulent Flows," *International Journal of Heat and Fluid Flows*, Vol. 21, 2000, pp. 599–621.

81. Aupoix, B.; Kassinos, S. C.; and Langer, C. A.: "ASBM-BSL: An Easy Access to the Structure Based Model Technology," Presented at the *Sixth International Symposium on Turbulence and Shear Flow Phenomena*, Seoul, Korea, June 22–24, 2009.

82. Rubinstein, R.; Rumsey, C. L.; Salas, M. D.; and Thomas, J. L.: *Turbulence Modeling Workshop*. ICASE Interim Report No. 37, 2001.

83. Blattnig, S. R.; Luckring, J. M.; Morrison, J. H.; Sylvester, A. J.; Tripathi, R. K.; and Zang, T. A.: "NASA Standard for Models and Simulations: Philosophy and Requirements Overview," AIAA-2009-1010, 2009.

84. Oberkampf, W. L. and Roy, C. J.: "*Verification and Validation in Scientific Computing*," Cambridge University Press, Cambridge, United Kingdome (ISBN-13: 9780521113601), 2010.

85. Eca, L.; Hoekstra, M.; Roache, P. J.; and Coleman, H. W.: "Code Verification, Solution Verification and Validation: an Overview of the 3rd Lisbon Workshop," AIAA Paper 2009-3647, 2009.

86. Eca, L. and Hoekstra, M.: "Evaluation of Numerical Error Estimation based on Grid Refinement Studies with the Method of Manufactured Solutions," *Computers & Fluids*, Vol. 38, 2009, pp. 1580–1591.

87. Rumsey, C. L.; Smith, B. R.; and Huang, G. P.: "Description of a Website Resource for Turbulence Modeling Verification and Validation," AIAA Paper 2010-4742, 2010.

88. Alexandrov, N. M.; Atkins, H. L.; Bibb, K. L.; Biedron, R. T.; Carpenter, M. H.; Gnoffo, P. A.; Hammond, D. P.; Jones, W. T.; Kleb, W. L.; Lee-Rausch, E. M.; Nielsen, E. J.; Park, M. A.; Raman, V. V.; Roberts, T. W.; Thomas, J. L.; Vatsa, V. N.; Viken, S. A.; White, J. A.; and Wood, W. A.: "Team Software Development for Aerothermodynamic and Aerodynamic Analysis and Design," NASA TM-2003-212421, November 2003.

89. Thomas, J. L.; Diskin, B.; and Rumsey, C. L.: "Toward Verification of Unstructured-Grid Solvers," *AIAA Journal*, Vol. 46, No. 12, 2008, pp. 3070–3079.

90. Diskin, B. and Thomas, J. L.: "Accuracy Analysis for Mixed-Element Finite Volume Discretization Schemes," National Inst. Of Aerospace TR 2007-8, Hampton, VA, August 2007.

91. Mehta, U.: "Simulation Credibility Level," Presented at the *5th Joint Army-Navy-NASA-Air Force (JANNAF) Modeling and Simulation Subcommittee Meeting*, Denver, Colorado, May 14–17, 2007.

92. Babula, M.; Bertch, W. J.; Green, L. L.; Hale, J. P.; Mosier, G. E.; Steele, M. J.; and Woods, J.: "NASA Standard for Models and Simulations: Credibility Assessment Scale," AIAA-2009-1011, 2009.

93. Yip, L. P.; Vijgen, P. M. H. W.; Hardin, J.; and van Dam, C. P.: "In-Flight Pressure Distributions and Skin Friction Measurements on a Subsonic Transport High-Lift Wing Section," *High-Lift System Aerodynamics*, AGARD-CP-15, 1993.

94. Chin, V. D.; Peters, D. W.; Spaid, F. W.; and McGhee, R. J.: "Flowfield Measurements about Multi-Element Airfoil at High Reynolds Numbers," AIAA-93-3137, July 1993.

95. Johnson, P. L.; Jones, K. M.; and Madson, M. D.: "Experimental Investigation of a Simplified 3D High Lift Configuration in Support of CFD Validation," AIAA-2000-4217, 2000.

96. Levy, D. W.; Wahls, R. A.; Zikuhr, T.; Vassberg, J.; Agrawal, S.; Pirzadeh, S.; and Hemsch, M. J.: "Summary of Data from the First AIAA CFD Drag Prediction Workshop," AIAA-2002-0841, 2002.

97. Hemsch, M. J. and Morrison, J. H.: "Statistical Analysis of CFD Solutions from the 2nd Drag Prediction Workshop," AIAA-2004-556, 2004.

98. Morrison, J. H. and Hemsch, M. J.: "Statistical Analysis of CFD Solutions from the 3rd Drag Prediction Workshop (Invited)," AIAA-2007-0254, 2007.

99. Vassberg, J.: "DPW IV Summary," AIAA-2010-4547, 2010.

100. Morrison, J. H.: "Statistical Analysis of CFD Solutions from the Fourth AIAA Drag Prediction Workshop," AIAA-2010-4673, 2010.

101. Lamar, J. E. and Obara, C. J.: "Review of Cranked-Arrow Wing Aerodynamics Project: Its International Aeronautical Community Role," AIAA-2007-0487, 2007.

102. Rizzi, A.; Boelens, O.; Jirasek, A.; and Badcock, K.: "What Was Learned From Numerical Simulations of F-16XL (CAWAPI) at Flight Conditions," AIAA-2007-0683, 2007.

103. Greenblatt, D.; Paschal, K. B.; Yao, C. S.; Harris, J.; Scheffler, N. W.; and Washburn, A. E.: "Experimental Investigation of Separation Control: Part 1: Baseline and Steady Suction," *AIAA Journal*, Vol. 44, No. 12, 2006, pp. 2820–2830.

104. European Research Community on Flow, Turbulence, and Combustion Web site: URL: http://cfd.mace.manchester.ac.uk/cgi-bin/cfddb/ezdb.cgi?ercdb+search+retrieve+&&&ConfFlow2b=Yes%%%%dm=Line, accessed on January 29, 2012.

105. You, D.; Wang, M.; and Moin, P.: "Large-Eddy Simulation of Flow over a Wall-Mounted Hump with Separation Control," *AIAA Journal*, Vol. 44, No. 11, 2006, pp. 2571–2577.

106. Rumsey, C. L.: "Successes and Challenges for Flow Control Simulations," AIAA-2008-4311, 2008.

107. Rumsey, C. L.: "Reynolds-Averaged Navier-Stokes Analysis of Zero Efflux Flow Control over a Hump Model," *Journal of Aircraft*, Vol. 44, No. 2, 2007, pp. 444-452.

108. Jenkins, L. N.; Neuhart, D. H.; McGinley, C. B.; Choudhari, M. M.; and Khorrami, M. R.: "Measurements of Unsteady Wake Interference Between Tandem Cylinders," AIAA-2006-3202, 2006.

109. Neuhart, D. H.; Khorrami, M. R.; and Choudhari, M. M.: “Aerodynamics of a Gulfstream G550 Nose Landing Gear Model,” AIAA-2009-3152, 2009.

110. Rumsey, C. L.; Long, M.; Stuever, R. A.; and Wayman, T. R.: “Summary of the First AIAA CFD High Lift Workshop,” AIAA 2011-939, 2011.

111. Kraft, E. M.: “Integrating Computational Science and Engineering with Testing to Re-engineer the Aeronautical Development Process,” AIAA-2010-0139, 2010.

112. Steinle, F.; Mickle, E.; and Mills, M.: “A 2025+ View of the Art of Wind Tunnel Testing,” *ITEA Journal*, Vol. 31, 2010, pp. 131–145.

113. Kogge, P. (Ed.): “ExaScale Computing Study: Technology Challenges in Achieving Exascale Systems,” Contractor report for AFRL Contract No. FA8650-07-C-7724, September 2008.

114. Wikipedia: “LINPACK,” URL: http://en.wikipedia.org/wiki/LINPACK, accessed on January 29, 2012.

115. Geim, A. K. and Kim, P.: “Carbon Wonderland,” *Scientific American*, Vol. 298, 2008, pp. 90–97.

116. Wikipedia: “Quantum computer,” URL: http://en.wikipedia.org/wiki/Quantum_computer, accessed on January 29, 2012.

117. Ladd, T. D.; Jelezko, F.; Laflamme, R.; Nakamura, Y.; Monroe, C.; and O’Brien, J. L.: “Quantum Computing,” *Nature*, Vol. 464, 4 March 2010, pp. 45–53.

118. Dicarlo, L.; Chow, J. M.; Gambetta, J. M.; Bishop, L. S.; Johnson, B. R.; Schuster, D. I.; Majer, J.; Blais, A.; Frunzio, L.; Girvin, S. M.; and Schoelkopf, R. J.: “Demonstration of Two-Qubit Algorithms with a Superconducting Quantum Processor," *Nature*, Vol. 460, 28 June 2009, pp. 240–244, URL: http://www.nature.com/nature/journal/vaop/ncurrent/pdf/nature08121.pdf.

119. Harrow, A. W.; Hassidim, A.; and Lloyd, S.: “Quantum Algorithm for Linear Systems of Equations,” *Physical Review Letters*, Vol. 103, October 9, 2009, 150502.

120. Meakin, R. L.; Atwood, C. A.; and Hariharan, N.: “Development, Deployment, and Support of Multi-Disciplinary, Physics-Based Simulation Software Products,” AIAA 2011-1104, 2011.

121. Hoffman, D. E.: “Supercomputers Offer Tools for Nuclear Testing and Solving Nuclear Mysteries,” *The Washington Post*, November 3, 2011.

Appendix A. Virtual Study Process

The typical practice that has been adopted for a fast-paced National Aeronautics and Space Administration (NASA) study in which expert opinion is sought on a particular subject is to hold a workshop and invite the experts. These experts present their positions on the selected subject, and, after the ensuing discussions, the organizers summarize the collective position of the expert panel. This requires advanced planning, availability of experts on the selected dates, and the ability to both travel and cover the travel-related costs. The "editors" of this study decided to do a "virtual" study because they were given only ten weeks within which they were required to prepare a preliminary report of the findings, and no funding was allocated to conduct the study.

The study was virtual in that all communications were conducted through email. The first step was to select a list of experts for the study. The editors set the requirement that both computational fluid dynamics (CFD) experts and experimentalists would be part of the study team, which would include representation from industry, academia, and government agencies other than NASA. After some brainstorming sessions, two dozen names were selected, and a message was sent to each with the details of the study and an invitation to participate. About two-thirds of the invitees agreed to participate. The question that was posed to them was:

> "We are organizing a virtual study of which physical testing capabilities could, going forward, be supplanted by computation (Modeling and Simulation) and therefore closed down, including the timing for such and the associated V&V aspects. Compared to several decades ago, at LaRC we no longer do a vast array of aero testing due to the improving accuracy and usefulness of MODSIM in the large. As a consequence, we have closed many facilities. We are curious as to whether this trend can be projected into the future for aerospace facilities, not just but including 'wind tunnels.' The contract for the 20-petaflop machine is on track for a 2012 delivery date; the exaflop machine is being designed. Beyond that is bio, optical, quantum, nano, molecular, and atomic computing. How much of that is required to computationally and practically address each of the myriad design issues we now utilize physical testing infrastructure to address? Please, at your earliest convenience, send us your understanding of the current and projected progress of aerospace MODSIM and potential impact upon physical testing infrastructure requirements."

All email communications were required to be "Reply All," so that others could see each panel member's point of view and could comment on specific material. Panel members presented varying opinions in different levels of participation and discussion. Finally, the editors compiled the input into a report and reconciled as many of the differences of opinion as possible. The draft report was sent to each participant, and each participant was asked to comment on the content. Additional discussion ensued. Finally, panel members were given the opportunity to opt out of authorship of the report if they disagreed with the conclusions. After some back and forth discussion, no contributor elected to opt out of the authorship.

Appendix B. Affiliation of Contributors

Professor Ramesh K. Agarwal
Department of Mechanical and Aerospace Engineering
Washington University
St. Louis, MO

Professor W. Kyle Anderson
National Center for Computational Engineering
University of Tennessee at Chattanooga
Chattanooga, TN

Dr. Douglas N. Ball
Chief Engineer, Aero Characteristics and Flight Performance
Boeing Commercial Airplanes
Seattle, WA

Dr. John A. Benek
Senior Scientist
Air Force Research Laboratory
Wright-Patterson AFB, OH

Mr. Edward L. Burnett
Lockheed Martin Senior Fellow, Modeling, Simulation, and Controls
Lockheed Martin Aeronautics
Palmdale, CA

Dr. Jay P. Boris
Chief Scientist and Director
Laboratory for Computational Physics and Fluid Dynamics
Naval Research Laboratory
Washington, DC

Dr. Raymond R. Cosner
Boeing Senior Technical Fellow
Boeing BDS Phantom Works
St. Louis, MO

Dr. Edward M. Kraft
Chief Technologist
AEDC/CZ
Arnold AFB, TN

Professor Dimitri J. Mavriplis
Department of Mechanical Engineering
University of Wyoming
Laramie, WY

Professor Robert W. MacCormack
Department of Aeronautics and Astronautics
Stanford University
Palo Alto, CA

Dr. Mark R. Melanson
Manager, Model Design and Test
Lockeed Martin Aeronautics
Fort Worth, TX

Dr. Douglas E. Post
Chief Scientist and CREATE Program Manager
DoD High performance Computing Modernization Program
Lorton, VA

Mr. Eldon S. Powell
Aerospace Testing Alliance
AEDC
Arnold AFB, TN

Dr. Brian R. Smith
Lockheed-Martin Fellow
Lockheed Martin Aeronautics
Fort Worth, TX

Dr. Philippe Spalart
Senior Technical Fellow
Boeing Commercial Airplanes
Seattle, WA

Dr. Tim T. Takahashi
MDO Manager – Advanced Flight Sciences
Northrop-Grumman Aerospace Systems
One Hornet Way
El Segundo, CA

Dr. Edward N. Tinoco
Technical Fellow (Retired)
Boeing Commercial Airplanes
Seattle, WA

Appendix C. Computer Development Outlook

Silicon-based computer technology has continued to progress in accordance with "Moore's Law" advancement, which implies a doubling of the number of semiconductor devices per unit area on a chip every 18 to 24 months. However, Moore's Law, if (incorrectly) interpreted as doubling of performance every 18 to 24 months, has hit a power wall, where clock rates have been essentially flat since the early 2000s [113]. Therefore, increasing explicit parallelism is the primary method for improving performance, and the supercomputer developers have adopted the "many cores" strategy for future advancements.

At the time of the writing of this section (November 2009), the Jaguar at Oak Ridge National Laboratory is the fastest computer in the world.[3] Built by Cray, Jaguar has 224,162 AMD Opteron™ processor cores, with a peak performance of more than 2.33 petaflops (10^{15} floating-point operations per second). The IBM® Roadrunner, at Los Alamos National Laboratory, is second at about one petaflops and is the world's first hybrid supercomputer that connects 12,240 Opteron cores and 12,240 enhanced cell chips in its compute nodes. In comparison, the Pleiades at NASA Ames Research Center is a Silicon Graphics International system that employs 56,320 Intel® Xeon® processor cores and runs at about 0.5 petaflops on the LINPACK benchmark [114], the industry standard for measuring a system's floating point computing power. Computing efficiency, which is defined as the ratio of the number of useful operations obtained from a computer system per second to the peak theoretical number of operations per second, has been found to be between 40 and 90 percent for the top ten supercomputers that are running LINPACK. For more complex computations, which are typical of a computational fluid dynamics (CFD) code, this efficiency is much less. The CFD workhorse code OVERFLOW, for example, runs at no more than 10 to 15 percent of the peak performance of Pleiades. Similar performance is expected from other CFD codes running on present-day massively parallel computers.

IBM has announced a new computer, Sequoia[3], to be released in 2012, which will achieve the human brain speed of 20 petaflops and will employ 1.6 million processing cores. Exaflops computers will become a reality by 2020 and will employ millions of cores. While Moore's Law may continue to be relevant during the next decade, the limiting factors are expected to be power and resiliency with additional challenges in memory, network, and storage.

The power consumption for Pleiades is 2.35 MW, and that of Jaguar is 7 MW. The Roadrunner is three times more power efficient than Jaguar but at the cost of restructuring algorithms for hybrid computing. A hybrid computer has more than one type of microprocessor. In the case of Roadrunner, the main structure is a standard cluster of AMD Opteron dual-core microprocessors, but each core is attached internally to another type of chip, which is the enhanced Cell (a technology that was originally developed for video games). The Cell, which acts as a turbocharger, can potentially boost performance by an order of magnitude (OOM) over that of an Opteron computer core alone. However, new algorithms must be developed to take advantage of this potential increase so that CFD codes can run on the hybrid machines.

---

[3] At the time of publishing of this report, the top spot is claimed by IBM's Sequoia computer at 16.3 petaflops with about 1.6 million cores; Jaguar is now number six, Pleiades is number eleven, and Roadrunner is number nineteen. This clearly demonstrates the rapid pace of computer hardware development.

The power requirement for exaflops (1,000 petaflops) machines is estimated to be in excess of 100 MW with current technologies. Work on energy-efficient solutions continues, and supercomputer companies are aiming at developing 20-MW exascale systems. A recent Defense Advanced Research Projects Agency (DARPA)-sponsored study [113] assessed the key challenges that impact the future of high-performance computing (HPC). The Exascale Working Group specifically considered the possible availability of exascale computing capability in the 2015 timeframe. Using various assumptions, the group projected a rise in computer speed both with and without power constraints (see figure D-1). The group noted that the exascale challenge goal could be achieved by 2015 with some aggressive assumptions (which were considered dubious at best) and unconstrained power; but noted that the goal could not be achieved with the power limited to 20 MW. With evolutionary technologies, exascale computing capability can be achieved by 2020, but power will continue to be an issue. The study concluded that significant research is needed to overcome four major challenges that are facing exascale systems:

- Power (developing efficient solutions for the energy requirements of future systems)
- Memory (developing technology to retain data at high capacities and access it at high rates)
- Concurrency (developing massive parallelism to increase system performance due to flattening of clock rates)
- Resiliency (developing technologies to enable continued system operation in the presence of either faults or performance fluctuations in a multimillion-core environment).

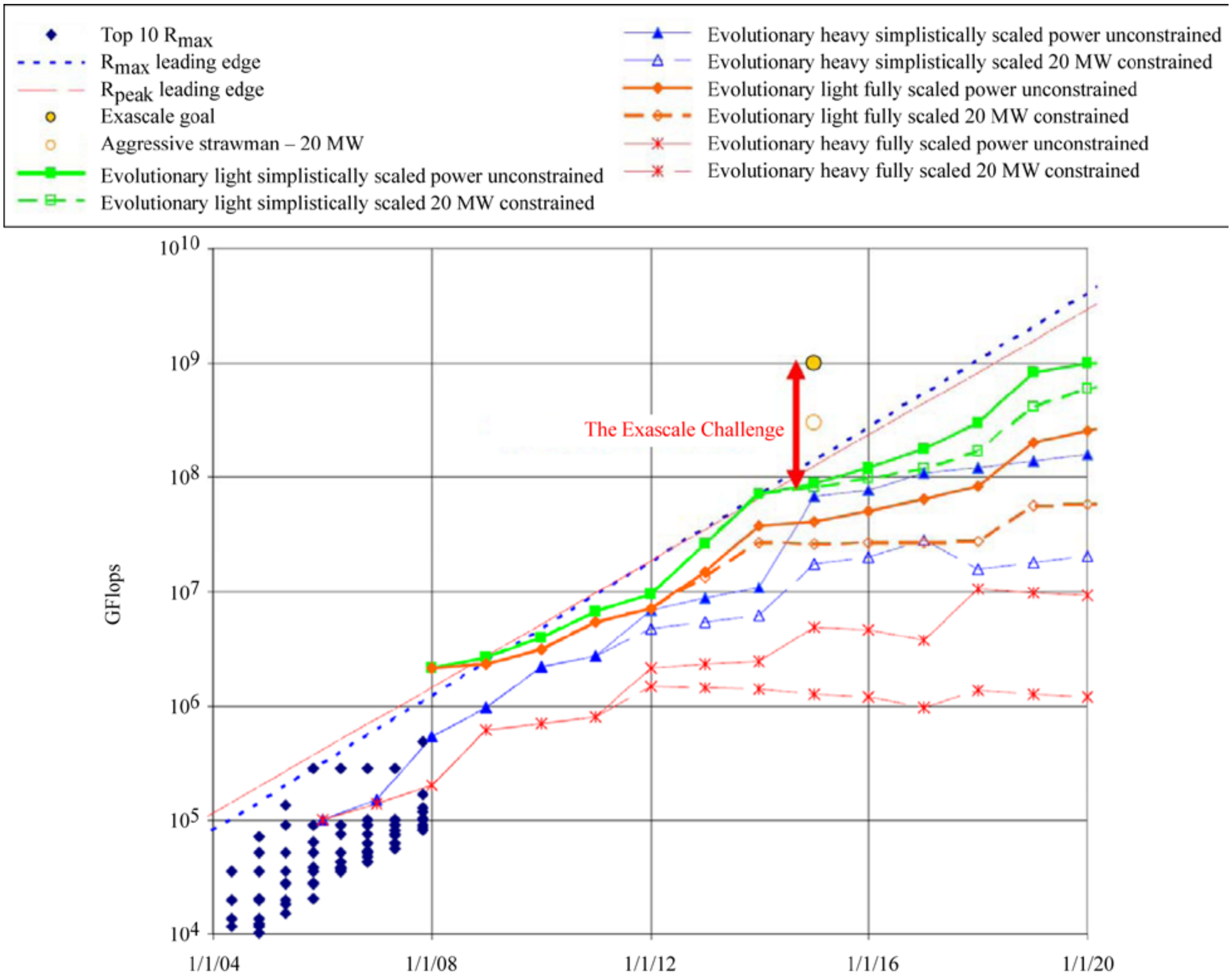


Figure D-1. Exascale goals and projected LINPACK performance under various assumptions with constrained (20 MW) and unconstrained power requirements (Reprinted from [113]).

The challenges are many:

- How does one find a high-performance algorithm that reduces the amount of energy that is expended to compute the solution to a relevant set of governing equations?
- How does one manage billion-way parallelism, where some component could possibly fail every few minutes?
- How does one develop new algorithms that scale to the level required to exploit the full potential of the machine?

However, the history of computer development is encouraging enough to inspire optimism. Depending on the pace of development in overcoming the above challenges, exascale computing can reasonably be expected to become a reality within the next 10 to 15 years. Cray, Inc. has just announced a new program that is aimed at delivering a supercomputer that is capable of performing one exaflop per second by the end of the next decade. This research initiative will explore new ideas and technologies for overcoming the challenges of delivering such a system, namely, power and cooling infrastructure, system and application resiliency, efficient processor and network architectures, and new programming models. To accomplish the goal, Cray will work with a team of researchers from the supercomputer centers at the University of Edinburgh, the Swiss National Supercomputing Center, and its European software partner, Allinea Software Ltd. Both Intel™ and IBM have similar plans to develop exascale technology in roughly the same time frame. Clearly, the future computing paradigm is multicore and massive parallelism, which is an unsolved research problem.

If current investment trends continue, the Department of Energy (DoE) and the National Science Foundation (NSF) will be the major technology drivers in the area of computational science and engineering. The 2009 Bell Prize winning team at Oak Ridge National Laboratory employed 223,000 cores of Jaguar for an application to analyze the effects of temperature on magnetic systems and achieved 1.84 petaflops, which represents nearly 80 percent of the peak performance of the machine. In another parallel data-visualization application, DoE researchers used up to four trillion grid points to gain insight into the potential bottlenecks and opportunities for machine-performance optimization for ultra-large datasets. NASA could benefit from collaboration with DoE research labs and HPC development elsewhere but will clearly lag behind technology leaders at current levels of investment. (For example, the FY11 budget request for DoE's advanced scientific computing research is $426 million as compared with NASA's budget request of $580 million for the entire Aeronautics Research Mission Directorate (ARMD), of which only a small fraction will go toward advanced CFD methods development.)

The technological change that is hitting the computer industry also poses substantial obstacles in harnessing the increasing capabilities of the new hardware to improve simulation outcomes. The drive toward massive parallelism means that many existing algorithms will be left behind and new algorithms, and possibly even new programming paradigms, will be required for simulation software to scale to hundreds of millions of cores. Hybrid computing, using hardware such as the Roadrunner at Los Alamos National Laboratory, which involves a mixture of traditional central processing units and cell processors, or GPU computing, will require a substantial rethinking of the algorithms and their software implementation. Additionally, as noted above, massive parallelism will place additional emphasis on fault-tolerant algorithms, which must be capable of recovering from an isolated hardware failure, because the probability of an isolated component failure in a massively parallel architecture increases linearly with the number of hardware components. Achieving revolutionary advances in aerodynamic simulation capability will require the development of novel algorithms that are both optimal from a theoretical or mathematical point of view in terms of accuracy and convergence and that can make effective use of the latest trends in HPC hardware development, which by all accounts will involve massive parallelism.

Computer technology developments are progressing on other fronts as well, which could add another dimension to Moore's Law. A breakthrough in chip-stacking technology has paved the way for three-dimensional (3D) chips, as compared with the traditional two-dimensional chip layouts. Chip stacking results in a compact sandwich of components that dramatically reduces the size of the overall chip package and shortens the distance that information on a chip needs to travel by three OOM. The 3D technology will fundamentally change the manner in which memory communicates with a microprocessor by significantly enhancing the data flow between the microprocessor and the memory. This capability would enable a new generation of supercomputers.

Beyond silicon is graphene [115], which is a form of carbon that consists of layers one atom thick that could potentially be used for the construction of transistors. The advantage of using graphene is that electrons can move through it with almost no resistance, which generates little heat, and graphene is in itself a good thermal conductor that allows heat to dissipate quickly. Given a much lower heat load, graphene-based transistors potentially could perform almost two OOM faster than silicon-based technology. Research work is continuing to make graphene-based technology practical for future-generation electronics.

Quantum computing [116] is yet another exciting possibility for the future that could dramatically improve computational power for particular tasks [117] while requiring less energy than silicon-based computing. Recent developments in both early quantum-computing hardware and software algorithms promise a major revolution in machine capability that could enable the *ab initio* treatment of turbulence and essentially provide the death knell for wind tunnels. The development of quantum computers will result in a paradigm shift in computational science; however, the technology is still at an early stage (see, e.g., references [118] and [119]), and it is too premature at this stage to make realistic projections. Furthermore, the question of initial and boundary conditions in the flight environment will need to be resolved for such *ab initio* simulations to be relevant to aircraft design. In any case, NASA should explore the potential of quantum computing for its applications so as to be at the forefront of technical innovations. The Air Force Office of Scientific Research (AFOSR) has launched an initiative in transformational computing to steer the research toward an understanding of how anticipated technology innovations in quantum computer architecture, software, and algorithms will enable new computational capabilities of relevance to the aerospace community. Similarly, the University of Southern California (USC) and Lockheed Martin have jointly acquired a 128-quantum-bits computer to explore the potential of quantum computing technology for faster, more secure optimization calculations.

Appendix D. Metrics for the Acceptance of CFD in the Design Process

Wind tunnels provide aerodynamic performance data, which is of primary interest in the design of aircraft and aerospace vehicles. Computational fluid dynamics (CFD) is an alternative source for generating aerodynamic data. Engineering program managers will willingly adopt a CFD-based design process, depending on how well CFD improves the process in four general areas:

- **Quality (accuracy):** Can CFD provide data with meaningfully equivalent or greater accuracy than alternative sources? "Meaningful accuracy" is what is required by the engineering process; this measurement depends on many things—the application (i.e., type of vehicle, flight envelope), product life cycle stage, and so on. If a design process requires only three-digit accuracy in a quantity, then providing an answer to four or five significant digits is more than what is required, and the cost associated with the extra accuracy may not be "meaningful" for that application.
- **Schedule (time):** Can the use of CFD shorten the time from the first identification of need to the final post-processing of data so that data are available to the engineering team to support engineering decisions in a timely manner? Remember that CFD cases commonly must be run a number of times before the data meet the needs of the engineering team. Engineering programs are rarely scheduled with as much time as is needed to do the job. However, note that CFD computations can be started at any instant (assuming the availability of competent CFD practitioners, computers, and so on), given the model geometry, but wind-tunnel tests must be scheduled based on the requirements from other competing projects.
- **Cost:** Can engineering data be obtained more cheaply with the use of CFD over alternative sources? Fully burdened cost includes computers, licenses, man-hours, facility costs (e.g., cost of floor space, power, and air conditioning), and so on. Wind-tunnel models and tests are generally quite expensive, but rough estimates for a large transonic database-generation task indicate that data can be obtained from wind-tunnel testing for as low as $50 per data point, with a new data point (flight condition) available every couple of seconds once the tunnel is running. At the current state of technology, CFD engineers may spend much more time just setting up a computer job. Large database requirements (hundreds of thousands of individual flight conditions) may be met cheaper via wind-tunnel testing for a long time into the future.
- **Risk:** Is the degree to which CFD data can be trusted understood? Wind-tunnel data are by no means perfect, but the engineering community has a century of experience in understanding how wind-tunnel data can be applied for conventional configurations and in establishing the degree of confidence that can be placed on individual data items for such applications. The trade-off between CFD and wind-tunnel test data is not yet clear for new, revolutionary configurations.

Appendix E. DoD's "CREATE" Program

The Computational Research and Engineering Acquisition Tools and Environments (CREATE) program is a new program that was established in FY08 by the Department of Defense (DoD) to improve the acquisition of major new military weapon systems. The goal of CREATE is to develop and deploy three sets of advanced, physics-based computational engineering design tools for DoD ships, air vehicles, and radio-frequency antennas (integrated with platforms).

The CREATE program will initiate replacement of the existing DoD design paradigm that relies on the extrapolation of historical data and experimental testing with a new paradigm that utilizes physics-based computational tools to develop optimized designs that are then validated by experimental testing. Extrapolation of historical designs has not been a good basis for new products that incorporate new concepts and materials. In addition, most current product-development paradigms are based on repeated "design–build–test" cycles that rely on repeated designing, building, and testing of physical products. Many industries and government agencies are beginning to replace this experimentally based methodology with a modern systems engineering approach that is based on the iterated use of computational engineering tools in "design–mesh–analyze" cycles. Often, physical testing is only necessary for the final optimized design. This results in more fully optimized designs, reduced time to market, lower project risk and costs, earlier system integration, fewer design defects, lower product development and testing costs, and increased ability to respond quickly to changing requirements and markets.

CREATE, which is a collaborative effort between the Office of the Secretary of Defense and the U.S. Army, Navy, and Air Force, was formed because the current investment by each service in computational tool development does not adequately address DoD-wide acquisition challenges, nor does it optimize the ability to utilize current high-end computer hardware. The CREATE projects will deliver fully mature tools at the end of a 12-year project schedule. Case studies of physics-based computational engineering projects with similar goals and scope indicate that nearly 10 years are needed for a team of 30 to 40 professionals to develop and deploy a fully mature tool. The cost of each project is about $10 million per year, plus some infrastructure support, for a three-project total of $360 million.

CREATE has a two-stage development and deployment strategy. The first stage involves the integration and upgrade of existing physics-based design tools to make them easier to use and to provide more accurate analyses and better exploit existing and near-term supercomputers. These first-stage tools will serve as prototypes to gain experience and develop requirements for the design and development of the next generation of tools that can fully exploit the next generation of supercomputers (i.e., the exaflops systems) and treat the major physical effects that will affect system performance.

Under the CREATE program, the air-vehicle computational design tools will include:

- **Kestrel:** a next-generation high-fidelity multiphysics simulation tool for fixed-wing air vehicles.
- **Helios:** a next-generation high-fidelity multiphysics simulation tool for rotary-wing air vehicles.
- A next-generation software tool to enable high-fidelity analysis of airframe-propulsion integration.

A CREATE-funded engineering application team will apply and transition computational tools to acquisition program problems and will gather requirements from the acquisition programs.

At maturity, Kestrel will facilitate full-aircraft high-fidelity simulation, including stores/cargo carriage and release, at realistic flight conditions in the subsonic, transonic, and supersonic flight regimes. Key functional attributes of Kestrel will include the capability to simulate complex maneuvers, propulsion effects, moving control surfaces, and aeroelastic effects, as well as the incorporation of realistic inner-loop and outer-loop control laws. Together, these software attributes allow for early assessment for air vehicles subjected to problematic environments and conditions that have historically required late-phase redesign. Kestrel will facilitate store/cargo release simulations with the associated need for accurate trajectory (i.e., translation and rotation) prediction. Kestrel will enable operational applications testing, including multivehicle proximate flight and maneuvers (e.g., refueling events) and environmental impact assessments (i.e., flight in proximity to the ground, runway, or ship deck).

The software tool Helios will facilitate (1) full rotorcraft high-fidelity simulation, including direct simulation of the relative motion between the rotors and the airframe, (2) engineering models of rotor systems, (3) stores/cargo carriage and release for realistic flight conditions (i.e., hover, forward flight, and transition and conversion for vehicle concepts that employ such technology), and (4) operational conditions such as refueling maneuvers or takeoff and landing maneuvers in benign and harsh environments (e.g., pitching/heaving ship decks). Helios provides for the coupling of various physical effects, such as rotor aeroelastic effects (i.e., flapping, lead/lag, and torsional), rotor/wake and airframe interactional dynamics, and propulsion effects (e.g., aerodynamics and inlet performance, exhaust re-ingestion during hover, and exhaust plume dynamics). Together, these software attributes allow for early assessment for rotorcraft designs (at both the preliminary design and final design stages) with corresponding opportunities for fault detection prior to fabrication of scale models or full-scale prototypes.

A key issue for all of the CREATE tools is the ability to rapidly and easily generate the numerical representations for the geometry of complex weapons systems (i.e. the meshes/grids), which are the starting point for design analyses. The CREATE program has initiated another project to develop the geometry and mesh generation capabilities that are required by the three major projects.

To be successful, the CREATE program must overcome many challenges including:

- The integration of many strongly coupled physics effects with solution algorithms that can exploit massively parallel computers.
- The use of large, multidisciplinary, multi-institutional teams in a collaborative, distributed environment to develop complex engineering software.
- The development of robust and accurate tools that meet the needs of the end users and provide results quickly enough to influence design decisions.
- The assurance of adequate software quality, to include a strong verification and validation (V&V) program.
- The provision of convenient and easy-to-use capabilities for generating problems and analyzing the results.

- The accomplishment of these goals by using the next generation of highly complex supercomputers.

Meeting these challenges requires (1) analysis and experimentation to identify optimal solution algorithms, (2) a distributed collaborative development environment, (3) intense focus on customer needs and requirements, (4) a strong V&V program, (5) an advanced geometry and mesh-generation capability, and (6) a strong focus on identifying and exploiting advances in software and computer architectures and engineering.

When the end-to-end flow times for complex (i.e., realistic) computational fluid dynamics (CFD) studies are analyzed, one can clearly see that instantaneous computing can only reduce the block time by 30 to 50 percent. Other elements of the process require a skilled person to make judgments, and therein is the bottleneck—setting up the geometry, generating the grid, setting up the flow problem, stopping and reviewing intermediate results, making adjustments as necessary, and post-processing (i.e., extracting knowledge from data). The issue of quality management must be addressed, which at present is handled by specialist involvement throughout the process. This issue must be addressed on a total-system basis. This is one of the objectives of the CREATE program, which will provide a framework for rapid multidisciplinary analysis and design through the use of high-fidelity physics-based tools. While NASA should keep a close watch on the CREATE development, the program is not aimed at understanding and bridging the gaps in knowledge and modeling of unsteady complex turbulent flows, which was the focus of the discussion in section 8. The reader is referred to reference [120] for more recent developments in the CREATE project.

Appendix F. What is at Stake?

Although computational fluid dynamics (CFD) is not expected to replace wind tunnels in the next 20 years, the continued improvement in computational capabilities in terms of physics modeling and the increase in efficiency due to the availability of much faster computing hardware will facilitate the increased use of CFD in the design of aerospace vehicles. Furthermore, wind-tunnel usage will continue to decline because of cost, facility age/capability, and lack of developmental programs. Given the massive advances in computing that are expected, the question is not if but when CFD will essentially replace nearly all wind-tunnel testing. This question will be faced by all aeronautical testing facilities, if not in two decades, then in three to five decades. The potential consequences to closing wind tunnels are both foreseen and unforeseen because such facilities provide the primary reason for the existence of some centers.

Note that Department of Energy (DoE) laboratories, such as Los Alamos, Livermore, and Oak Ridge have gone through a transition of sorts in the past two decades from nuclear-weapon development facilities to major high-performance-computing (HPC) centers. Reference [121] provides a brief account of the contributions in computational technology that have been made in support of DoE's Stockpile Stewardship program, while recognizing the need for laboratory experiments for validation. Existing aeronautical research centers may also go through a similar change in the coming decades, as they are the best positioned and most logical places to invest in future aerospace computational disciplines, ensuring their survival as vibrant aeronautics research centers.

REPORT DOCUMENTATION PAGE	*Form Approved* *OMB No. 0704-0188*

1. REPORT DATE *(DD-MM-YYYY)* 01- 09 - 2012	**2. REPORT TYPE** Technical Publication	**3. DATES COVERED** *(From - To)*

4. TITLE AND SUBTITLE Role of Computational Fluid Dynamics and Wind Tunnels in Aeronautics R&D	**5a. CONTRACT NUMBER**
	5b. GRANT NUMBER
	5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S) Malik, Mujeeb R.; Bushnell, Dennis M.	**5d. PROJECT NUMBER**
	5e. TASK NUMBER
	5f. WORK UNIT NUMBER 561581.02.08.07.48
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) NASA Langley Research Center Hampton, VA 23681-2199	**8. PERFORMING ORGANIZATION REPORT NUMBER** L-20144
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001	**10. SPONSOR/MONITOR'S ACRONYM(S)** NASA
	11. SPONSOR/MONITOR'S REPORT NUMBER(S) NASA/TP-2012-217602

12. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified - Unlimited
Subject Category 01
Availability: NASA CASI (443) 757-5802

13. SUPPLEMENTARY NOTES

14. ABSTRACT

The purpose of this report is to investigate the status and future projections for the question of supplantation of wind tunnels by computation in design and to intuit the potential impact of computation approaches on wind-tunnel utilization—all with an eye toward reducing the infrastructure cost at aeronautics R&D centers. Wind tunnels have been closing for myriad reasons, and such closings have reduced infrastructure costs. Further cost reductions are desired, and the work herein attempts to project which wind-tunnel capabilities can be replaced in the future and, if possible, the timing of such. If the possibility exists to project when a facility could be closed, then maintenance and other associated costs could be rescheduled accordingly (i.e., before the fact) to obtain an even greater infrastructure cost reduction.

15. SUBJECT TERMS
Aeronautics; Computational fluid dynamics; Flow separation; Turbulence modeling; Wind tunnels

16. SECURITY CLASSIFICATION OF:			**17. LIMITATION OF ABSTRACT**	**18. NUMBER OF PAGES**	**19a. NAME OF RESPONSIBLE PERSON**
a. REPORT	**b. ABSTRACT**	**c. THIS PAGE**			STI Help Desk (email: help@sti.nasa.gov)
					19b. TELEPHONE NUMBER *(Include area code)*
U	U	U	UU	73	(443) 757-5802

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std. Z39.18