AIAA JOURNAL
Vol. 36, No. 5, May 1998

Experimental Methodology for Computational Fluid Dynamics Code Validation

Daniel P. Aeschliman* and William L. Oberkampf*
Sandia National Laboratories, Albuquerque, New Mexico 87185-0825

Validation of computational fluid dynamics (CFD) codes is an essential element of the code development process. Typically, CFD code validation is accomplished through comparison of computed results to previously published experimental data that were obtained for some other purpose, unrelated to code validation. As a result, it is a near certainty that not all of the information required by the code, particularly the boundary conditions, will be available. The common approach is, therefore, unsatisfactory, and a different method is required. A methodology is described developed specifically for experimental validation of CFD codes. The methodology requires teamwork and cooperation between code developers and experimentalists throughout the validation process and takes advantage of certain synergisms between CFD and experiment. The methodology employs a novel uncertainty analysis technique, which helps to define the experimental plan for code validation wind-tunnel experiments and to distinguish between and quantify various types of experimental error. The methodology is demonstrated with an example of surface pressure measurements over a model of varying geometrical complexity in laminar, hypersonic, near perfect gas, three-dimensional flow.

I. Introduction

IN the past, flight vehicle design and development have been based primarily on wind-tunnel experimentation and flight testing. Mathematical methods, primarily approximate analytical solutions, have also made important contributions to design and development, but these methods were commonly directed toward improving the understanding of the flow physics or toward developing approximate engineering solutions. Modern computational fluid dynamics (CFD) has evolved over roughly the past 30 years, tracking the availability of ever more capable computing hardware and algorithms. During much of that time, CFD has concentrated on the development of improved numerical algorithms and the solution of relatively simple research problems. More recently, a broader range of complex flow physics has been addressed along with advanced grid generation techniques for more complex and realistic geometries. As a result of the recent advent of massively parallel (MP) machines, peak computing speeds now exceed one trillion floating point operations per second, and total random access memory approaches 600 Gbytes. Computing speed and memory far exceed projections for the mid-1990s made in 1983 by the National Research Council.[1] However, actual implementation of MP computing has been hindered by the significant effort required in writing efficient code for MP architectures.

CFD has, in specific areas, made important contributions to the design and development of aircraft, missiles, re-entry vehicles, gas turbine engines, and rocket engines, to name a few. In addition, CFD codes are being used increasingly to describe highly complex fluid flow processes, such as chemical vapor deposition, shock/boundary-layer interactions, turbulent reacting flows, and multiphase flows. However, the underlying physics of certain flow processes, e.g., boundary-layer transition and turbulence, is still poorly understood. For such fluid mechanics processes, a predictive capability based on first principles is not available, and it is not certain that simply increasing computing power will lead to valid solutions in those areas.

To some extent ignored by the CFD community in the past, the question of validity (accuracy and reliability) of CFD code predictions is now becoming critically important. CFD is being applied to the design of actual hardware, and a failure to answer quantitatively the question of code validity is increasingly unacceptable. Stated differently, how can the bounds of validity be determined such that, within those bounds, CFD predictions can be trusted without experiment?

Over the past decade, the critical and growing importance of this issue has been noted by numerous researchers.[2–7] Oberkampf[6] presented a proposed framework for evaluating solutions from CFD codes, describing the particular types and classes of problems and the corresponding types of investigations needed to verify, calibrate, or validate codes designed to solve them. He concentrated on the broad philosophy of code verification ("solving the governing equations right") and validation ("solving the right governing equations"), definitions originally suggested by Boehm[8] and popularized by Blottner.[9] The terms *calibration* and *certification* also enter into this discussion. By *calibration* we mean a code's ability to reproduce valid data (not exclusively experimental) over a specified range of flow parameters, for some class of geometry, without necessarily assessing the overall accuracy of the physical models and numerical methods. We consider calibration to be a much less demanding element of validation, and it is addressable experimentally by the same methods. *Certification* was defined by Mehta[10] as the entire process of establishing the credibility of a code, i.e., a certified code has been verified, calibrated, and validated. The term *certification* also has legal implications, such as public safety and liability and possible requirements for competition in a government request for proposals.

Code verification (solving the equations right) involves comparisons to exact analytic solutions, computations from previously verified codes, and codes that address simplified, or specialized, cases. Conversely, CFD code validation (solving the right equations) relies on comparison of computational results to experimental data. Our view is largely consistent with Bradley[2] and Marvin,[3] who consider comparison to experiment as the only acceptable method of generalized CFD code validation. We generally support this view because we believe that validation fundamentally means the demonstration of computational fidelity to reality. However, we differ from them somewhat in that we believe comparison to a previously validated code is also an acceptable means of code validation, provided that certain conditions are met. In particular, a new code can be considered validated only for the class of physics and range of parameters embodied in the original experimental data. Any claim that a code is validated for other physics or parameter ranges is unjustified. As the physical complexity increases, the ability to quantify the class of physics and range of parameters becomes increasingly difficult, if not impossible. For the complex physics case we believe code validation must rely on experimental measurements;

Received Oct. 22, 1996; revision received April 17, 1997; accepted for publication April 23, 1997.

*Distinguished Member of the Technical Staff, Aerosciences and Compressible Fluid Mechanics Department, Mail Stop 0825. Associate Fellow AIAA.

the accuracy of the measurements is an important, but separate, issue.

This paper and a companion paper[11] together describe our approach to the CFD code verification and validation (V&V) process. Reference 11 focuses on the verification of CFD codes. In this paper we concentrate on CFD code validation by experimental means. Our validation methodology consists of general philosophical guidelines and specific procedural recommendations consistent with those guidelines by which the process is implemented. We will start by reviewing some of the historical circumstances that have influenced progress in this area to date and discussing in some detail why we believe the CFD code validation process must be an integral component of CFD code development and not an add-on conducted after the fact. We will note certain inherent synergisms that, if identified and properly utilized, can lead not only to continued improvements in CFD code capability and credibility, but have the potential to improve experimental capabilities as well. We will describe a novel technique for uncertainty analysis and experimental design that serves to distinguish and quantify various sources of experimental error, and then we will present an example to demonstrate the methodology.

II. Historical Background

CFD has evolved more or less in parallel with the development of digital computers over approximately the past 30 years. Because of limitations in computer speed and memory, the early emphasis was on the development of numerical algorithms for simple physical models (inviscid flow over slender bodies of revolution at low angle of attack, for example) and was largely a research exercise. Computing speed has risen, on average, by a factor of 10 every 6 years or so since 1960; cost per compute cycle has fallen by a factor of at least 10^3; and algorithm efficiency has improved moderately. Over the same period, computer memory has grown by a factor of roughly 10^5 for mainframes. This growth has enabled CFD to change from a research activity to an applied technology directed toward solutions to complex fluid engineering problems.

Throughout this period, code development has proceeded along a path largely independent of experimental validation. There are presumably diverse reasons for a lack of perceived need for CFD code validation. Further, there has often existed a competitive and frequently adversarial relationship between computational modelers and experimentalists, which has led to a lack of cooperation between the two groups. Where cooperation has occurred, it seems as often as not to have been due to small teams forming voluntarily. There has, however, over the past decade been a growing awareness[2–7,12,13] that such competition does not best serve the interests of either group. Nevertheless, effective implementation of a cooperative atmosphere, however desirable it may be, remains in general a significant challenge.

Dwoyer[13] has noted that CFD code development has come to a critical juncture and, in the absence of key input from other technical disciplines, is unlikely to make significant advances in attacking the remaining unsolved problems of fluid mechanics, such as transition and turbulent reacting flow. He suggested it will require the contributions of computer scientists, nonlinear mathematical analysts, theoretical and experimental fluid dynamicists, molecular physicists, instrumentation specialists, and facility designers and operators all working closely with computational fluid dynamicists. Dwoyer referred to such an integrated activity as the "science of viscous aerodynamics."

Despite its limitations, the present capabilities of CFD are formidable. The advent of improved gridding techniques in finite element codes and multiblock structured grids has greatly reduced the design cycle time for some problems. Two-dimensional and some three-dimensional airfoils are designed by computer, not parametric wind-tunnel test. Three-dimensional Euler solvers reliably predict steady high Reynolds number flow over wing-body-tail-pylon-engine configurations at low angle of attack. Further, we submit that for a somewhat narrowly restricted, well-defined set of problems, advanced CFD codes are now capable of producing results at least as accurate, if not more so, than can be measured in a wind-tunnel experiment. Depending on the circumstances, the code may also be able to do it faster and more cheaply. (We caution, however, that direct cost comparisons are difficult to make and are often misleading.) An example, which we have used in our own work and which will be described more fully later, is laminar, near perfect gas, hypersonic flow over a slender sphere/cone at low angle of attack. We now have sufficient confidence in certain CFD code predictions for this case that we use the results to provide an in situ calibration of our hypersonic wind-tunnel experiments. It is reasonable to expect that the range of problems that can be very accurately solved by CFD will continue to expand, especially given the barely tapped potential of MP computing.

Assuming that CFD can compute certain flows more quickly, accurately, and cheaply than we can measure them, we see a changing relationship between CFD and wind-tunnel experimentation. This changing relationship has been noted elsewhere[14,15] in regard to wall and model support interference corrections for wind-tunnel data. We believe that through teamwork and cooperation this changing relationship can produce improvements in the capabilities of both computational and experimental fluid dynamics. Conversely, a continued them vs us attitude will surely impede progress in both CFD and experiment and may even serve to accelerate the pace at which aerospace test facilities are being closed.

The consequence of further decline in experimental capability, should it occur, is to us alarming, for it will necessarily imply an increasing dependence on new and unvalidated CFD codes for solutions to the remaining flow problems, which are by definition the most difficult ones. We believe such a consequence is most certainly unwise and is potentially catastrophic. We view it as axiomatic that CFD simply cannot do it alone, now or for the foreseeable future. Likewise, present experimental capabilities cannot provide a complete and simultaneous simulation of certain important flow regimes, for example, high-enthalpy, high Reynolds number, re-entry-type flows. Nevertheless, as Mason,[16] among others, has noted, very real progress in improving experimental capability continues to occur. Mason cites as examples improved connection of subscale testing to full-scale aerodynamics, advanced flow visualization, improvements in unsteady aerodynamics testing capability, and renewed emphasis on experimental foundations for advanced concept development. There is a proposal to examine in detail the technical feasibility of a new hypersonic facility that would provide a complete flow and chemistry simulation capability for Mach 10–20 flight at 100,000–200,000 ft altitude.[17] Given the current funding climate, such a facility, even if feasible, is decades away. We believe CFD is an appropriate tool to tie together experimental results obtained in a piecemeal fashion from separate facilities as they exist now and to aid in advancing experimental capabilities in the future.

No rational computational fluid dynamicist would suggest there is no need for verification or validation of CFD codes. A common view among computational fluid dynamicists, however, is that whereas code V&V are indeed necessary, the process, particularly the validation step, can be accomplished through comparison to existing data documented in reports or archival journals, obtained for some purpose other than CFD code validation. We most strongly disagree. Almost invariably, critical details are missing from published data, particularly for archive journal publications where discussion is limited in the interest of reducing paper length. It is critically important that the boundary and/or initial conditions assumed by the code be accurately known from the experiment.

Rarely, however, is such information presented in sufficient detail to ensure that boundary and initial conditions are matched. Wilcox in a 1995 private communication cites several examples that illustrate this point. In one case, turbulent heat transfer on an ablating nosetip with blowing was computed and compared to earlier published experimental data. Serious disagreement between the experimental data and the code predictions was seen. It later became possible to interact directly with the experimentalists and to discuss the experimental boundary conditions in detail. Once the proper experimental boundary conditions (BCs) had been input to the code, the agreement was excellent. In another case, close agreement was initially observed between experiment and code prediction for a turbulent pipe flow. Subsequent reviews of the experiment and the numerical simulations showed that not only were the experimental results seriously in error, but a substantial deficiency existed in the code physics. The original agreement had been fortuitous. Such later opportunities are unusual and may not suffice even if they can be arranged. Key personnel can become unavailable or forget important details, or there may be political or personal issues involved that make open and honest communication impossible.

III. Philosophical Guidelines

Our CFD code validation methodology is based on a set of philosophical guidelines. These guidelines have evolved from our own work and through our interactions with many others. The underlying framework was presented in Ref. 18. These guidelines are as follows.

1) A CFD code validation experiment should be jointly designed by experimentalists and CFD code developers working closely together throughout the program, from inception to documentation, with complete candor as to the strengths and weaknesses of each approach. No withholding of limitations or deficiencies is permitted, and failure or success of any part of the effort must be shared by all. Without this level of cooperation, openness, and commitment, the process is likely to fail.

2) A CFD code validation experiment should be designed to capture the essential flow physics, including all relevant BCs, assumed by the code. This is especially true for inflow/outflow BCs, which directly impact whether, for example, a two-dimensional calculation is adequate or a full three-dimensional solution is required. In this context, we note that no physical experiment can be truly planar two-dimensional; there are only varying degrees of approximation of the actual three-dimensional flow. Experimentalists must understand the code assumptions and try to determine if the experiment is consistent with those assumptions. If the parameters initially assumed for the calculation cannot be satisfied in the proposed experimental facility, it may be feasible to alter the code inputs so as to meet them, or it may be necessary to look elsewhere for a facility. For example, can the required boundary-layer state on a model be assured? Is the type and quantity of instrumentation appropriate to provide the required data in sufficient quantity and at the required accuracy and spatial resolution? Conversely, computational fluid dynamicists must understand the limitations of the physical experiment, ensure that all of the relevant physics are included, and define physically realizable BCs. As already noted, the level of detailed understanding required can be achieved only if the validation experiment is planned and conducted as part of a team effort.

3) A CFD code validation experiment should strive to emphasize inherent synergisms between the two approaches. For example, if sufficient confidence is available in a code solution for simple flow physics and geometry, computed results can be used as a calibration of the experiment. Other examples of such synergisms are noted later.

4) Although the experimental design must be developed cooperatively, complete independence must be maintained in actually obtaining both the computational and experimental results. Neither side is permitted knobs driving adjustable parameters. Only when the computed and experimental results are in hand is a comparative evaluation permitted, and only then is it appropriate to consider the causes of any differences. We have found that investigating the causes of differences invariably leads to a deeper understanding of the experiment and/or the numerical simulation.

5) CFD code validation must be conducted through a hierarchy of experiments of increasing difficulty and specificity. Start with easier experiments, then proceed to more complex and difficult ones, with each step providing an increasingly difficult challenge to the code. In wind-tunnel experimentation for a flight vehicle, for example, a suggested hierarchy is as follows: a) total body forces and moments; b) control surface forces and moments; c) surface pressure distributions; d) surface heat flux and shear stress; e) flowfield distributions of pressure, temperature, and velocity components; and f) flowfield distributions of Reynolds stresses.

As this hierarchy suggests, body forces and moments are the easiest of the physical quantities both to predict and to measure. It may be argued that total body forces and moments are inadequate for CFD code validation because these data are too gross a measure of code accuracy. We contend that force and moment data are of value for two reasons. First, their value, i.e., their difficulty of prediction, depends directly on both the complexity of the flow physics and of the vehicle geometry. In cases of simple flow physics and simple geometries, we not only agree with the opposing viewpoint but we go beyond it. As stated earlier, these are cases for which we believe the flows can be computed at least as accurately, if not more so, than they can be measured. As a result, such flows can serve as experiment calibration cases for the experimentalist.

However, for more complex flow physics and vehicle geometries, the prediction of forces and moments can be more challenging than is commonly recognized. For example, Walker and Oberkampf[19] experienced substantial difficulty in predicting laminar flow body forces and moments on a re-entry vehicle with a large flap deflection. Computing the large laminar separated flow region and reattachment on the flap proved at or beyond the present state of the art. Second, body forces and moments, as well as control surface forces and moments, can be measured more accurately than, say, surface heat flux. The experimental uncertainty bound on forces and moments is typically 1/10 or less than that of surface heat flux. Therefore, the error tolerance on the CFD result must also be a factor of 10 better on forces and moments as compared to heat flux to fall within the experimental uncertainty.

The general point is that, as one progresses down the list to more difficult quantities for CFD to predict, the experimental uncertainty generally increases also. In the process, knowledge is gained about the experiment, which can lead to improved experimental technique and measurement accuracy in later, more difficult experiments.

6) Employ an uncertainty analysis procedure that delineates and quantifies systematic and random error sources by type. Wind-tunnel data uncertainty analysis as typically practiced[20] attempts to quantify the statistical (random) uncertainty of individual components. It does not normally allow one to distinguish and quantify the contribution of one class of random error from another, nor to identify and quantify systematic errors. These might be random and/or systematic errors due to, for example, flowfield nonuniformity or nonrepeatability, instrumentation uncertainties, and model geometry inaccuracies. Our recommendations for specific steps and an example of this process are presented in Secs. V and VII. Section VII includes an example of the data uncertainty analysis technique applied to our own work.

7) Invest in careful quantification of all relevant experimental parameters needed for comparison of computational predictions to the validation experiment.

Facility flow calibration data are normally available for some parameters, e.g., Mach number, unit Reynolds number, pitot pressure, and total temperature in a wind-tunnel facility, at some level of spatial resolution. However, even for these quantities, absolute accuracy is not easily quantified, and the spatial resolution may be inadequate for the validation experiment. Further, other important parameters are not typically known from the facility calibration but must be measured as a separate step or as part of the validation experiment itself. Acquiring these data can be direct, such as a measurement of the actual base pressure distribution on a model in comparing computed to measured drag, or indirect, such as determining flow angularity from combinations of runs with the model at various pitch and roll angles, as described in Sec. VI.

Experimental parameters that may be important in specific cases for code validation are as follows: freestream static conditions and flow angularity, the inflow/outflow boundary conditions, wall and support interference effects, freestream turbulence intensity (for a transition experiment), and body surface BCs. Code requirements must be incorporated into the validation experiment design to ensure that the needed data can and will be acquired. Conducting this calibration step early may ultimately prove to be faster, less expensive, and more reliable than doing it later when funds may be depleted, the facility or its staff may be unavailable, or experimental parameters may have changed. Obtaining such data can be very expensive and time consuming, however, and it may not prove possible to obtain each one to the level of accuracy or spatial resolution initially desired. However, with each step included, the overall confidence in, and value of, the validation experiment increases.

IV. Synergisms between CFD and Experiment

By a synergism, we mean an activity whose primary intent is to meet a requirement for one approach, whether CFD or experiment, but that generates improvements in capability and/or accuracy of the other, such that both computational and experimental methods benefit. The synergistic use of the strength of one approach to offset a weakness of the other represents a powerful tool in the CFD code validation process. Particular synergisms will vary with the individual situation. Two examples of synergisms follow.

1) If in a wind-tunnel experiment the wind-tunnel model is designed for easy modification from geometrically simple to complex, it becomes possible to produce a wide range of flow conditions. The geometrically simple flows could possibly be calculated with high confidence, whereas the complex geometry flows may exceed the current computational state of the art. As an example, for attached, perfect gas, laminar flow over a slender sphere/cone at low angle of attack, confidence in the computed solutions for flow over the simple model with simple flow physics can be at such a high level that the results are usable for an in situ calibration of the freestream wind-tunnel flow. This type of calibration can provide new, and sometimes surprising, information about the facility. For flow over more complex geometries, the measurements can be used to validate the code.

2) The coupled integration of CFD into the operation of adaptive-wall wind tunnels, and, especially, in correcting for wall and support interference on model aerodynamic data is a synergism that has a large potential payoff. It is desirable to test aircraft configurations at the largest possible scales to maximize Reynolds number, a goal that is in immediate conflict with minimizing interferences. The status of this activity has been assessed by Lynch et al.[14] and Ashill[15] at the AGARD 73rd Fluid Dynamics Panel Symposium. Attempts to apply specific computational methodology are described by several authors at the same symposium. It was noted by Lynch et al.[14] that the CFD capability required to compute interference corrections must advance in concert with the testing requirements. In a similar vein, advances in the use of CFD to compute flows in perforated-wall wind tunnels are retarded by a lack of well-characterized wall BCs. Detailed measurement of the actual wall BCs as a function of test section location and given tunnel operational parameters would directly improve wind-tunnel data accuracy, in addition to providing the needed BCs for a CFD calculation.

V. Recommended CFD Code Validation Procedures

The following procedures are recommended for implementing the methodology proposed here for relatively long run-time wind-tunnel facilities. It is acknowledged that the set of procedures recommended here is highly idealistic. Rarely, if ever, will an individual validation experiment include them all. No one recommendation can ever be satisfied perfectly, and the relative priority of the procedures will change from experiment to experiment. Further, the list is by no means all inclusive; different code validation experiments will necessarily generate different measurement issues, for example, in turbomachinery flows. Use of short-duration facilities, such as shock tubes or shock tunnels, would add a strong temporal response requirement on experimentation procedures and measurements.

1) Obtain detailed, accurate freestream flow calibration data at a spatial resolution consistent with code requirements. Freestream flow calibration at some level of spatial resolution and accuracy is, of course, a requirement for even routine production wind-tunnel testing. However, as already noted, for CFD code validation purposes, flowfield calibration should be done at typically finer spatial resolution; it should include all quantities required by the code as input BCs; and the experimental uncertainties should be quantified. Further, for a boundary-layer transition experiment, it should include a determination of freestream turbulence intensity, scale, and frequencies. It is apparent that most experimental facilities are inadequately calibrated in this context, either because the specific quantities were not needed for normal operations or because of the high cost of acquiring measurements at the desired spatial density. Further, some facility managers may be reluctant to share such detailed flow quality data with users (and competitors). However, for a CFD validation experiment it must be available. This is another argument for having, and using, one's own facilities for code validation research. Having total control over the facility is an invaluable advantage, and in some cases it is absolutely essential.

No wind-tunnel flow is perfectly uniform over the test section volume. The facility calibration may show that the level of nonuniformity is larger than is acceptable within the accuracy requirements of the validation experiment. If so, it will be necessary to use the measured, spatially varying flow properties as location-dependent inflow BCs to the code. (Such a procedure, although conceptually straightforward, to our knowledge has not yet been demonstrated.) Although this approach is probably unnecessary at this stage of CFD code development for validation experiments in typical, i.e., near perfect gas, wind tunnels of high flow quality, it would appear to be an essential requirement for validation experiments in high-enthalpy flow facilities in which rapid expansions combine with finite rate chemistry. In such facilities, the flow is typically highly nonuniform and poorly characterized, making accurate comparisons of experimental data to code predictions extremely difficult, if not impossible.

2) Precisely characterize the model wall boundary conditions, as tested. Differences will exist between the nominal and actual model dimensions, orientations, surface conditions, and locations of instrumentation. These must be known to high accuracy to provide wall BCs for the code. Pretest mechanical inspections of the model as assembled in all its possible configurations should include size; shape, e.g., straightness or out-of-round; surface finish (especially any steps at joints); and surface waviness. Aeroelastic effects must also be considered, because model and sting deflection under aerodynamic load can introduce systematic experimental errors well in excess of measurement precision.

If surface temperature can vary significantly, as in a long-duration hypersonic flow experiment, and computed results are sensitive to surface temperature, then the model surface temperature distribution must be measured. If those temperature changes are both significant and nonuniform, e.g., on a model at angle of attack, then shape change due to thermal expansion must be considered. Model orientation settings (angle of attack, roll and yaw angle) and configuration dimensions must be precisely determined, including the repeatability of these values if the model configuration will be altered routinely. These data will be important input for experimental error assessment.

3) Vary model size in the same facility at the same nominal test conditions. This is a useful strategy to ascertain wall or support interference effects, unsuspected Reynolds number effects such as incipient transition on the model, or variations due to limited core flow size, especially at off-design tunnel conditions. The penalties are added test and model costs, and depending on individual circumstances, not all physical effects, e.g., boundary-layer growth, model dimensions, and tolerances may be directly scalable.

4) Conduct the same experiment in different facilities. If feasible, conduct the same code validation experiment, with the same model, in more than one facility. Satisfactory agreement of results from different facilities lends confidence that there are no inadequately understood facility-related bias errors in the data, e.g., condensation effects, wave focusing, excessive flow angularity, etc. This procedure, especially for simple model geometries, would also serve to uncover inaccuracies and inconsistencies in the flow calibration data for each facility used. The same personnel should oversee the execution of the experiment at each site and also have access to all facility operational and performance data. On the computational side, a recommended corollary is to use the results of different codes to predict the simple flow cases used for any in situ calibrations conducted in the experiment.

5) Apply redundant measurement techniques for critical experimental variables. Inasmuch as no measurement is free of error, and no single measurement technique is best for all applications and ranges of parameters, redundant measurements of critical variables should be performed whenever possible, and certainly if there is a suspicion that a measurement technique is of questionable applicability under some conditions. For example, a pitot-static probe might be used to calibrate the freestream Mach number over the test section. Suppose, however, that the freestream Mach number and probe Reynolds number for some flow conditions are such that probe measurement accuracy is significantly affected by viscous effects. A redundant measurement of freestream Mach number could be obtained by measuring the flow velocity and static temperature independently and computing the Mach number.

6) Develop an uncertainty analysis technique that is able to identify and quantify the significant random and bias errors. Once formulated, use the uncertainty analysis to help define the experimental run matrix. This is central to the method and is distinctly different from, and extends significantly beyond, standard uncertainty analysis. Our methodology does use standard statistical methods but, in addition, incorporates novel extensions of the standard procedures. This is particularly true in the use of repeat runs and reflection of data around pitch and yaw planes in designing the experiment run schedule. This is also important for in situ freestream calibrations based on comparison to code predictions for cases of particularly simple model geometry and flow physics. In this way, random errors can be isolated from certain systematic errors in the course of the data uncertainty analysis, and both types of error can be quantified.

The run matrix should be carefully designed so that combinations of runs yield both statistical and bias error information. Repeat runs should be included that satisfy different objectives. Immediately repeating a particular case yields statistical information on short-term facility repeatability. Repeating runs in varying order, on different days, and in separate facility entries can uncover subtle errors related to facility operations, specific personnel, time of day, etc. Repeat runs require careful introspection in their selection and sequence and are critical to an assessment of the absolute accuracy and statistical precision of the data. Repeat runs are not afterthoughts; they are essential elements in the method and must be incorporated into the experimental plan and the results included in the experimental data set. An example application of our uncertainty analysis is given in Sec. VI.

Detailed methodology for statistical error analysis as it applies to experimental data in general has, of course, been widely available for many years. A recently published treatise, by far the most detailed prescription for dealing with systematic and random errors in wind-tunnel data when the systematic errors have been previously identified and estimated, is presented in Ref. 21. Reference 21 identifies virtually every conceivable source of experimental error in wind-tunnel testing and greatly improves the art of wind-tunnel test data uncertainty analysis. Additional information and insight regarding mathematical treatment of systematic (bias) errors are available in Refs. 22 and 23.

7) Obtain and plot together data for positive angle of attack and negative angle of attack with the model rolled 180 deg. Data obtained with a model at zero roll angle and pitched from, e.g., 0 to +10 deg can be plotted with data for a model at 180-deg roll angle and pitched from $\alpha = 0$ to -10 deg. The result is that errors associated with flowfield nonuniformity and model misalignment in pitch can be identified. This recommended procedure is not new, yet it is done less often in wind-tunnel testing than one might expect. And even if data are obtained in this way, there may be a reluctance to show the results plotted together because the differences are usually larger than the quoted instrumentation uncertainties for the experiment. Estimation of the underlying bias error is discussed in Sec. VI.

8) Take and keep notes that are as careful, detailed, and extensive as possible. Such information will be invaluable when trying to explain any anomalies that may arise during the data analysis. This recommendation is appropriate not only for obviously unusual circumstances or events, but it applies to seemingly routine items as well. Insofar as understanding the experimental data is concerned, it is essentially impossible to record too much annotative information.

Clearly, some of these recommendations are easier to implement than are others. The recommendation to acquire a complete, detailed, finely spaced calibration of the tunnel freestream represents an expensive, time-consuming exercise. For heavily utilized production facilities, interference with higher priority work may make such flowfield calibrations impossible to obtain. Even for research-oriented facilities for which interference with other work may not be a restriction, performing such calibrations almost certainly will require a substantial investment.

For facilities with relatively high flow quality, in situ calibrations based on CFD performance predictions for a simple geometry may provide a technically acceptable alternative at minimal cost for some, if not most, code validation experiments. That is, this approach should be satisfactory if the scale of the model is small relative to the scale of the freestream nonuniformity. Failing that, a possible conclusion may be that some facilities will be dedicated to production testing exclusively, for which existing calibrations and databases are presumably already adequate, and others will be used to provide the needed code validation capability.

VI. Case Study for CFD Code Validation Methodology

In 1990 Sandia National Laboratories initiated a long-term, coupled CFD/experimental effort, referred to as the Joint Computational/ Experimental Aerodynamics Program (JCEAP), to improve the Laboratories' hypersonic wind-tunnel experimentation and CFD simulation capabilities. We will discuss JCEAP briefly to illustrate our code validation methodology and to describe the uncertainty analysis procedure. More detailed descriptions of the experiments and comparisons to computational results are presented in Refs. 12, 19, and 24–29.

The geometry chosen was a spherically blunted cone with a sliced aft region and flaps at the rear of the slice. This geometry generated a wide range of flow complexity, from simple, attached flow to very complex flow with massive separation and strong shock/boundary-layer interactions. At the same time, the geometry was designed to eliminate several potentially troublesome numerical difficulties, which need not be introduced into a validation experiment.

We required that the flow be laminar everywhere on the model to avoid the predictive uncertainty that would be introduced by use of a turbulence model. Flow visualization using shear-stress-sensitive liquid crystals[30] was employed in a preliminary series of experiments with varying freestream Reynolds number to ensure that the boundary layer was laminar over the entire model for all validation experiments. The liquid crystal technique also provided surface-flow characterization data for cases with massively separated flow on the flap.

Wind-Tunnel Conditions

Nominal wind-tunnel conditions for all experiments were as follows: freestream Mach number $M_\infty = 7.84$, stagnation pressure $P_0 = 340$ psia (2.344 MPa), stagnation temperature $T_0 = 1106°$R (614 K), and freestream Reynolds number $Re_\infty = 2.0 \times 10^6$/ft ($6.56 \times 10^6$/m). $Re_L = 1.80 \times 10^6$, based on model length. Angle of attack was varied from -9 to $+18$ deg at nominal 3-deg increments. Roll angle was set at 0 (slice on the windward side), 90, 180, or 270 deg. Model axial location within the test section was also varied to assess errors due to flow axial gradients.

Wind-Tunnel Model

The wind-tunnel models for the force and moment and the pressure experiments were nominally identical in size and shape. The model was a 10.391-in. (0.26393-m)-long, 10% spherically blunted cone with a slice on one side of the body (Fig. 1). The slice is parallel to the axis and begins at 0.7 of the length of the body, measured from the spherical nose tip. The model was designed so that flaps could be attached to the aft portion of the slice, extending to the baseplane and providing deflection angles of 10, 20, and 30 deg. By requiring the flaps to extend to the model baseplane for all flap deflections, a substantial simplification became possible in constructing the grids for the body geometry and for the base flow. This also simplifies setting the outflow BCs across the baseplane in the numerical simulation. The force and moment model was used in conjunction with a precision six-component internal strain gauge balance. The remainder of the discussion relates to the pressure model and experiments.

The pressure model incorporated two 48-port, differential pressure, electronically scanned pressure (ESP) modules, one 0.36-psid (2.5-kPa) and one 1.0-psid (6.9-kPa) unit, mounted internal to the model to minimize pneumatic tubing lengths and pressure lag time. A cylindrical sting cover was used to provide an easily characterized downstream wall BC if needed for CFD simulations at some later time. This is another example of simplifying the geometric design in a validation experiment to eliminate unnecessary complexity in the CFD modeling. The model incorporated nine semiconductor-bridge Kulite gauges to detect any high-frequency surface pressure fluctuations and four coaxial thermocouples in the model wall to provide the wall thermal BC to the code. A detailed mechanical inspection provided precise characterization of all model dimensions and pressure port and thermocouple locations.

Instrumentation

A total of 96 pressure ports of 0.029-in. (0.737-mm) diam were machined in the model surface. These ports were positioned at 15 axial stations along the length of the model. Three axial stations on the cone, 3.200, 5.200, and 7.200 in. (8.138, 13.208, and 18.288 cm) from the nose, each had 16 orifices. Another extensively instrumented area was in the slice/flap region, which contained 40 orifices. The orifices were connected to either the lower or the higher pressure ESP module depending on prior estimates of the pressure level at each port location. Vacuum reference was provided by a high-capacity turbopump. A detailed study was conducted to ensure that errors due to leaks and pressure lag time were negligible. Details of the experimental system are presented in Ref. 25. Approximately 55,000 surface pressure measurements were obtained during the experiment.

A complete run schedule for the experiment is listed in Table 1. A number of schedule features are apparent that are unusual from the traditional perspective of wind-tunnel experimentation. First, repeat runs were scheduled and executed for every configuration. The purpose for this was to obtain a large number of multiple data sets with which to conduct an extensive uncertainty analysis. Second, noting that the run number reflects the chronological order, it can be seen that for flap deflection angle $\delta = 0$ deg, repeat runs were made substantially later during the experiment. For example, runs 20 and 62 were made nearly four weeks apart. Comparing these two runs, as opposed to comparing two consecutive runs on the same day, aids in estimating the overall measurement system repeatability. Third, runs for each configuration were made at the aft axial tunnel station. Comparing the pressure measurements between the forward and aft stations yields quantitative estimates of the effect of changes in the test section flowfield.

VII. Uncertainty Analysis

The uncertainty analysis permits the separation and quantification of random and systematic uncertainties in model surface pressure measurement due to system instrumentation and model alignment errors, flowfield nonuniformity, and model geometry inaccuracies. The force and moment uncertainty analysis is similar but cannot yield the uncertainties due to model inaccuracies because forces and moments are integrated quantities. Additional details on the pressure analysis are presented in Refs. 27 and 29 and on the force and moment analysis in Refs. 12 and 24.

The procedure for statistically estimating these uncertainty components is based on comparisons of measurements obtained from certain types of repeat runs, runs with the model at different locations in the test section, and use of symmetry features of the model geometry. The analysis is an experimentally based statistical estimate of variance components of surface pressure measurements. To take full advantage of this new procedure special attention must be given to constructing the run schedule to maximize information used in the analysis.

Table 1 Run schedule

Roll angle, deg	$\delta = 0$ deg	$\delta = 10$ deg	$\delta = 20$ deg	$\delta = 30$ deg
	Forward tunnel station, 7.6 in. (19.3 cm)			
0	20, 22, 62	42, 43	48, 49	56, 57
90	24, 26, 59, 61	37, 39	46	54
180	30, 32, 58	35, 36	44, 45	50, 53
270	28, 29	40, 41	47	55
	Aft tunnel station, 4.1 in. (10.4 cm)			
0	101, 102	118, 119	124, 126	131, 133
180	103, 112	115, 116	122, 123	127, 129

System Instrumentation and Model Alignment Uncertainty

The total system instrumentation and model alignment uncertainty, hereafter referred to as instrumentation uncertainty, is the experimental uncertainty in surface pressure measurement caused by all of the following and their interaction with each other: pressure sensor hysteresis, nonlinearity, sensitivity drift, and zero shift; reference pressure accuracy and repeatability; analog amplifier system accuracy; data digitizing and recording system accuracy; configuration change repeatability; model pitch, roll, and yaw alignment random errors; variations in freestream Mach number and Reynolds number within a run; and variations in freestream Mach number and Reynolds number from run to run.

It can be seen from this list that all of these error sources produce random errors, i.e., run-to-run variations in each of these sources is expected. No bias errors in instrumentation uncertainty, e.g., an incorrectly set amplifier gain, can be detected by the present analysis. (An error of this type would be detected during the in situ calibration of the flow using the CFD solution for the simple flow physics case.) The instrumentation uncertainty combines all experimental uncertainty in the entire experiment, except that due to test section flowfield nonuniformity and model geometry imperfection uncertainty. To calculate the instrumentation uncertainty, one compares pressure measurements for the same port from different runs with the model at the same physical location and orientation in the test section. For the same angle of attack, roll angle, flap deflection angle, and tunnel location, each pair of ports compared will have the same location in the vehicle-induced flowfield. When differences in pressure port measurements are made in this way the uncertainty due to flowfield nonuniformity and model geometry variation cancels out.

By examining the run schedule (Table 1), one chooses run pairs that have the same roll and flap angles and the same tunnel location; 29 run pairs are found to satisfy these conditions. Of these 29 run number pairs, examples are (20, 22), (24, 61), (103, 112), (42, 43), (124, 126), and (131, 133). For example, the pressure at port 1 of

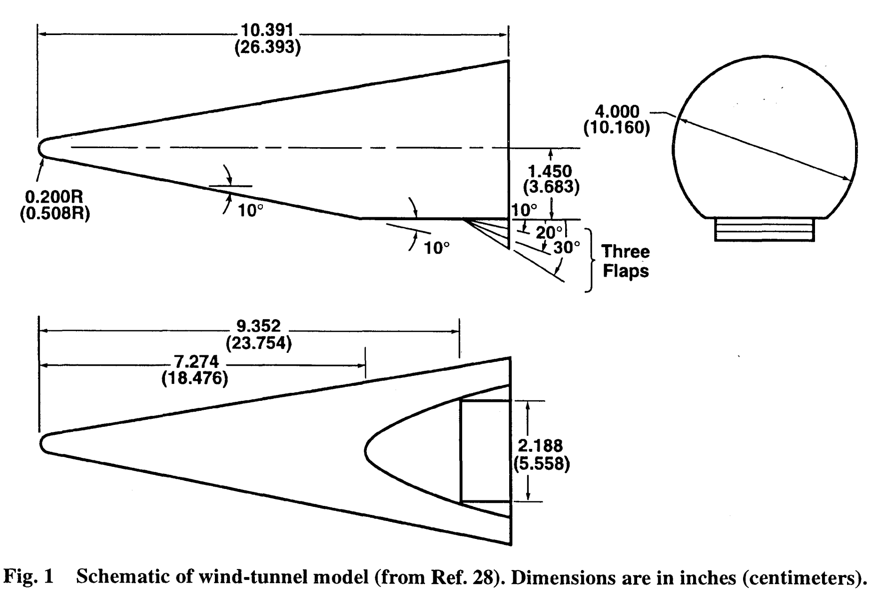

Fig. 1 Schematic of wind-tunnel model (from Ref. 28). Dimensions are in inches (centimeters).

the first run listed is compared with that at port 1 of the second run listed, port 2 of the first run is compared with port 2 of the second run, etc., for each α in common between the two runs. Pressure measurements were obtained for a total of 12 angles of attack for each run, 9 nonzero angles of attack, and 3 measurements at zero α. As a result, there are a total of 18 combinations of α where pressure comparisons can be made (9 nonzero α comparisons plus 9 permutations of zero α measurements). Therefore, an estimate of the total number of possible pressure port comparisons is (96 ports) $\times$ (29 run pairs) $\times$ (18 α pairs) = 50,112 comparisons.

Some ports were overpressured for certain conditions, and so the actual number of available comparisons for instrumentation uncertainty was reduced to 48,164.

Differences in pressure port measurements were computed with the following technique. Let the pressure measurement for port i and angle of attack j be denoted as $(p_i/p_\infty)_j^r$, where the superscript denotes the run number r. Then the average pressure of the port for the two runs being compared is given by

$$\overline{\left(\frac{p_i}{p_\infty}\right)_j}^{r,s} = \frac{1}{2}\left[\left(\frac{p_i}{p_\infty}\right)_j^r + \left(\frac{p_i}{p_\infty}\right)_j^s\right]$$

where $i = 1, 2, \ldots, 96$ and $j = 1, 2, \ldots, 18$, where 18 is the total number of α. Let the absolute value of the difference between a pressure measurement and the average pressure be defined as the residual. Then the residual is given by

$$\left(\frac{\Delta p_i}{p_\infty}\right)_j^{r,s} = \left|\left(\frac{p_i}{p_\infty}\right)_j^r - \overline{\left(\frac{p_i}{p_\infty}\right)_j}^{r,s}\right|$$

Note that the residual can be computed using the pressure measurement from either run r or run s.

To make the pressure port comparisons, it is required that the α of each of the two runs is identical. If they are not the same, then part of the difference in the two measurements will be due to the difference in α caused by nonrepeatability of the model pitch mechanism. Although α is accurately known to ± 0.02 deg, the difference in repeatability from one pitch to another was as large as ± 0.28 deg. To minimize this uncertainty in the analysis, all of the pressure measurements were interpolated to the nominal angles of attack. To accomplish this, a cubic spline interpolation was computed for each pressure port as a function of α for each run so as to obtain pressure data at precisely the nominal values of α.

Test Section Flowfield Nonuniformity Uncertainty

Test section flowfield nonuniformity uncertainty is uncertainty in surface pressure measurements caused by the following.

First is nonuniformity of freestream flow in the test section, which can be due to, for example, inaccurately designed or positioned nozzle wall contours, operation of a fixed nozzle wall wind tunnel at a Reynolds number different from the design condition, and slight changes in the location of nozzle wall boundary-layer transition due to changing wall temperature. Flow nonuniformity due to wave focusing in hypersonic wind tunnels, axisymmetric tunnels especially, is a particularly serious source of uncertainty but is rarely discussed or documented.

Second is bias errors in the alignment of the model in pitch, roll, and yaw, which could be caused, for example, by an improperly calibrated or used bubble level to set the pitch and roll angle, an inaccurately leveled test section, or an inaccurate pitch-sector or model positioning system. Yaw angle alignment of the model in the test section is always a difficult measurement to make due to the lack of an easily definable reference.

The uncertainty in surface pressure measurement due to a combination of test section flowfield nonuniformity uncertainty and instrumentation uncertainty is computed by comparing measurements made at different locations in the test section. The combined flowfield nonuniformity and instrumentation uncertainty is calculated by comparing surface pressure measurements for the same port on the body at the same relative location in the vehicle flowfield but at different locations in the test section. This procedure will not include any uncertainty due to model imperfections because, by using the same ports for both comparisons, this uncertainty component cancels in taking the difference between the two measurements.

By examining Table 1 for combinations of model axial station, roll angle, and flap deflection angle, one finds four types of run pairs that will produce the kinds of residuals desired. These are comparisons between measurements made at different axial locations in the test section, comparisons between different roll angles at the same α, comparisons between positive α with a roll angle of 0 deg and negative α with a roll angle of 180 deg, and comparisons between positive α with a roll angle of 90 deg and negative α with a roll angle of 270 deg. Examples of run pairs for each of these types of comparisons are, respectively, (20, 101), (24, 112), (35, 119), and (46, 47).

The total number of pressure port comparisons for these four types, minus the number of comparisons lost due to overscaled ports, is 101,838 residuals. The residuals for flowfield nonuniformity and instrumentation are computed by the same equations given earlier, but the number of α for each of the types is different.

Model Geometry Uncertainty

Model geometry uncertainty is uncertainty in surface pressure caused by the following.

1) Model geometry deviations are defined as deviations of the physical model from the conceptual, or mathematical, description of the model. These can be due to a variety of sources, for example, model fabrication deviations such as a nonspherical nose, accidental surface damage, time-dependent but reversible bending distortion due to asymmetric aerodynamic heating, and permanent warpage of the model surface or lifting surfaces due to repeated aerodynamic heating in the test section.

2) Model imperfections are defined as model deviations that are not considered part of the geometrical character of the model but do affect the measurement. Examples of these types of deviations are a poorly fabricated or burred pressure orifice and a pressure leak between the orifice and the transducer.

Model geometry uncertainty, along with instrumentation uncertainty, is computed by comparing surface pressure measurements for different ports, with both ports at the same physical location in the test section and at the same relative location in the vehicle flowfield. This requirement can be met on the forward, conical portion of the model but not on the slice or flaps. As a result, pressure port comparisons are made only on the conical section of the model. This procedure will yield the combined model geometry and instrumentation uncertainty but will not include any uncertainty due to flowfield nonuniformity. The total number of pressure port comparisons for model geometry uncertainty is 24,196.

Uncertainty Results

Plotted in Fig. 2 are all of the residuals computed for instrumentation, flowfield nonuniformity, and model geometry uncertainty. It can be seen that the magnitude of the uncertainty steadily increases with the magnitude of the pressure measured. This trend is represented in the residuals by scaling them with the magnitude of pressure measured. A constrained least squares fit to the residuals was computed with the intercept set to zero. The resulting fit was computed to be

$$\Delta p_{\mathrm{ls}}/p_\infty = 0.00773(p_s/p_\infty)$$

where p_s is the surface pressure measured. This fit is also shown in Fig. 2.

The sample variance is now calculated with the local sample scaled according to the least squares fit just given. The equation for estimating each type of variance, normalized by the least squares fit of the residuals, is given by

$$\hat{\sigma}^2 = \frac{1}{N}\sum_{k=1}^{N}\left[\frac{(\Delta p/p_\infty)}{(\Delta p_{\mathrm{ls}}/p_\infty)}\right]_k^2$$

where N is the total number of residuals (or pressure comparisons) and the subscript k indicates the kth residual. The standard deviation

Table 2 Summary of surface pressure uncertainty analysis

Source of uncertainty	No. of residuals	Normalized $\hat{\sigma}$	% of total rms uncertainty
Instrumentation	48,164	0.56	12
Flowfield	101,838	1.28	64
Model	24,196	0.79	24
Total	174,198	1.60	100

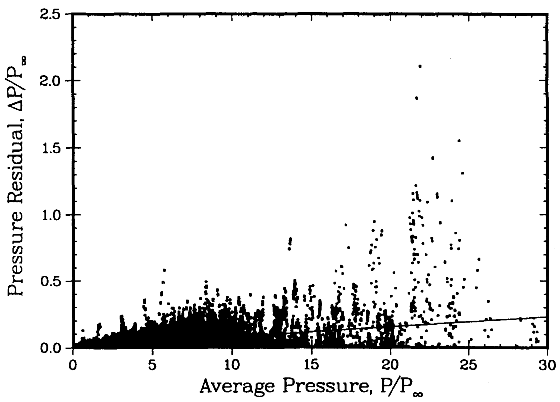

Fig. 2 Combined instrumentation, flowfield, and model residuals vs pressure magnitude (from Ref. 28).

due to flow nonuniformity and model geometry can then be calculated from

$$\hat{\sigma}_{\text{flow}} = \sqrt{\hat{\sigma}^2_{\text{flow + instrumentation}} - \hat{\sigma}^2_{\text{instrumentation}}}$$

$$\hat{\sigma}_{\text{model}} = \sqrt{\hat{\sigma}^2_{\text{model + instrumentation}} - \hat{\sigma}^2_{\text{instrumentation}}}$$

The standard deviation due to all of the uncertainty sources is then given by

$$\hat{\sigma}_{\text{total}} = \sqrt{\hat{\sigma}^2_{\text{instrumentation}} + \hat{\sigma}^2_{\text{flow}} + \hat{\sigma}^2_{\text{model}}}$$

The total estimated standard deviation of each individual measurement is $\hat{\sigma}_{\text{total}}(\Delta p_{\text{ls}}/p_\infty)$. Therefore, the total uncertainty bound on each pressure measurement at the 95% confidence level is

$$(p_s/p_\infty) \pm 2\hat{\sigma}_{\text{total}}\Delta(p_{\text{ls}}/p_\infty) = (p_s/p_\infty) \pm 2\hat{\sigma}_{\text{total}}[0.00773(p_s/p_\infty)]$$

where $\hat{\sigma}_{\text{total}}$ was computed to be 1.82.

Table 2 gives the summary statistics for the uncertainty estimates of the entire experiment. It is seen that the dominant contributor to uncertainty in these surface pressure measurements is due to the nonuniformity of the tunnel test section flowfield. Although we had previously suspected this was the case, the present statistical analysis quantitatively demonstrates it.

The dominant contribution of nonuniform flow to uncertainty in our wind tunnel suggests the question, is this just a characteristic of the present wind tunnel or is it typical? The absolute magnitude (no normalization) of the present results for flowfield uncertainty was compared to those for hypersonic tunnel B at the Arnold Engineering Development Center.[31] This comparison showed that both wind tunnels are comparable in the magnitude of flowfield nonuniformity. We strongly suspect that the largest contribution to measurement uncertainty in most, if not all, near perfect gas hypersonic wind tunnels is due to flowfield nonuniformity. Although this technique has not been applied to transonic wind-tunnel data, we suggest the dominance of flowfield nonuniformity error may also occur in transonic facilities. We encourage others to use the present statistical method to determine if this is the case.

We acknowledge that, in demonstrating the methodology, we ourselves have not followed all of our own recommended procedures. The realities of budgetary and time constraints precluded repeating the experiment in different facilities and at varying physical scales. In addition, the inflow BCs were not experimentally determined to fine spatial resolution. Each of these procedures represents an additional component of variation in the validation process. As an example, if the experiment were completely redone at another facility and the statistical uncertainty analysis repeated on the combined data, we expect that the estimated uncertainty would increase due to the introduction of facility-to-facility variation. The use of additional components of variation in this way will lead to a more reliable assessment of the quality of the validation process and the procedures and equipment used to implement it.

VIII. Conclusions

A methodology for experimental validation of CFD codes has been developed and demonstrated. The methodology incorporates specific experimental procedures that are consistent with, and an outgrowth of, a number of general philosophical guidelines. Two guidelines are key: 1) the use of experiments designed specifically for CFD code validation by computational and experimental fluid dynamicists working closely together from program inception to documentation and 2) implementation of an uncertainty analysis, which guides the experimental design and which permits the delineation and quantification of various classes of both bias and random errors. Because it is our experience base, we have presented the methodology in terms of wind-tunnel experimentation in relatively long-duration aerospace testing facilities, specifically, for hypersonic, near perfect gas flow over a sliced sphere/cone of variable geometry. However, extension of the general recommendations to other experiments should be apparent.

Careful experiments designed and executed specifically for CFD code validation are the recommended source of data for CFD code validation. We consider unsatisfactory the common practice of attempting to validate codes using published data obtained for some purpose unrelated to CFD code validation. Almost inevitably, critical information required by the code, boundary and initial conditions especially, will be unavailable. In addition, experimental investigators should take a more critical view toward measurements obtained for CFD code validation and be willing to identify and quantify components of uncertainty in order to reduce these errors. They should take advantage of numerical simulations to aid in improving the quality of the experiment, particularly in using CFD code solutions for especially simple flow physics and geometries, to provide an in situ calibration of the experiment. Likewise, numerical simulations should routinely include error analyses.

Implementation of some, if not most, of the code validation procedures recommended here is neither inexpensive nor easy. As a result, specific procedures may be technically or economically impractical in particular situations. With each included step, however, the overall experimental uncertainty can be better estimated and the quality of the code validation process improved.

We conclude by noting that the cost of CFD code validation may represent a significant fraction of the total expense of CFD code development, and we understand the reluctance on the part of program managers to use scarce funds for the validation exercise. But we believe that failure to validate complex CFD codes represents false economy. The near-term cost of CFD code validation must be weighed against the future, and potentially much larger, economic and social liability of a system failure whose origin is traceable to erroneous results from an unvalidated code.

Acknowledgments

This work was performed at Sandia National Laboratories, which is operated by Lockheed Martin Corporation for the U.S. Department of Energy under Contract DE-AC04-94AL85000. We thank D. C. Wilcox for the examples in his 1995 private communication illustrating that insufficient detail is provided to ensure boundary and initial conditions are matched.

References

[1]"The Influence of Computational Fluid Dynamics on Experimental Aerospace Facilities: A Fifteen Year Projection," National Research Council, National Academy Press, Washington, DC, 1983.

[2]Bradley, R. G., "CFD Validation Philosophy," AGARD Symposium on Validation of Computational Fluid Dynamics, Paper No. 1, Lisbon, Portugal, May 1988.

[3]Marvin, J. G., "Accuracy Requirements and Benchmark Experiments for CFD Validation," AGARD Symposium on Validation of Computational Fluid Dynamics, Paper No. 2, Lisbon, Portugal, May 1988.

[4]Mehta, U. B., "Some Aspects of Uncertainty in Computational Fluid Dynamics Results," *Journal of Fluids Engineering*, Vol. 113, No. 4, 1991, pp. 519–525.

[5]Cosner, R. R., "Issues in Aerospace Application of CFD Analysis," AIAA Paper 94-0464, Jan. 1994.

[6]Oberkampf, W. L., "A Proposed Framework for Computational Fluid Dynamics Code Calibration/Validation," AIAA Paper 94-2540, June 1994.

[7]Neumann, R. D., "CFD Validation—The Interaction of Experimental Capabilities and Numerical Computations," AIAA Paper 90-3030, Aug. 1990.

[8]Boehm, B. W., *Software Engineering Economics*, Prentice–Hall, Englewood Cliffs, NJ, 1981.

[9]Blottner, F. G., "Accurate Navier–Stokes Results for the Hypersonic Flow over a Spherical Nosetip," *Journal of Spacecraft and Rockets*, Vol. 27, No. 2, 1990, pp. 113–122.

[10]Mehta, U. B., "Flight Performance Estimation Utilizing Computational Fluid Dynamics," *Proceedings of the 5th National Aerospace Plane Technology Symposium*, Vol. 1, NASP CP-5028, Oct. 1988.

[11]Oberkampf, W. L., and Blottner, F. G., "Issues in Computational Fluid Dynamics Code Verification and Validation," *AIAA Journal*, Vol. 36, No. 5, 1998, pp. 687–695.

[12]Oberkampf, W. L., and Aeschliman, D. P., "Joint Computational/Experimental Aerodynamics Research on a Hypersonic Vehicle: Part 1, Experimental Results," *AIAA Journal*, Vol. 30, No. 8, 1992, pp. 2000–2009.

[13]Dwoyer, D., "The Relation Between Computational Fluid Dynamics and Experiment," Invited Presentation, AIAA 17th Ground Testing Conf., Nashville, TN, July 1992.

[14]Lynch, F. T., Crites, R. C., and Spaid, F. W., "The Crucial Role of Wall Interference, Support Interference, and Flow Field Measurements in the Development of Advanced Aircraft Configurations," *Wall Interference, Support Interference, and Flow Field Measurements*, 73rd Fluid Dynamics Panel Symposium (Brussels, Belgium), CP-535, AGARD, 1993, pp. 1.1–38.

[15]Ashill, P. R., "Boundary-Flow Measurement Methods for Wall Interference Assessment and Correction—Classification and Review," *Wall Interference, Support Interference, and Flow Field Measurements*, 73rd Fluid Dynamics Panel Symposium (Brussels, Belgium), CP-535, AGARD, 1993, pp. 12.1–21.

[16]Mason, W. H., "What We Need in Experimental Aerodynamics: One Engineering Educator's View," AIAA Paper 92-0161, Jan. 1992.

[17]Miles, R., Brown, G., Lempert, W., Natelson, D., Yetter, R., Guest, J., Williams, G., and Bogdonoff, S., "Radiatively Driven Hypersonic Wind Tunnel," AIAA Paper 94-2472, June 1994.

[18]Aeschliman, D. P., Oberkampf, W. L., and Blottner, F. G., "A Proposed Methodology for CFD Code Verification, Calibration, and Validation," *Proceedings, 16th International Congress on Instrumentation for Aerospace Simulation Facilities* (Dayton, OH), 1995, pp. 27.1–27.13 (Paper 95-CH3482-7).

[19]Walker, M. M., and Oberkampf, W. L., "Joint Computational/Experimental Aerodynamics Research on a Hypersonic Vehicle: Part 2, Computational Results," *AIAA Journal*, Vol. 30, No. 8, 1992, pp. 2010–2016.

[20]"Aerodynamic Data Accuracy and Quality: Requirements and Capabilities in Wind Tunnel Testing," CP-429, AGARD, Sept. 1987.

[21]"Assessment of Wind Tunnel Data Uncertainty," AGARD Advisory Rept. AGARD-AR-304, 1994.

[22]Coleman, H. W., and Steele, W. G., *Experimentation and Uncertainty Analysis for Engineers*, Wiley, New York, 1989.

[23]Coleman, H. W., Steele, W. G., and Taylor, R. P., "Implications of Correlated Bias Uncertainties in Single and Comparative Tests," *ASME Journal of Fluids Engineering*, Vol. 117, No. 4, 1995, pp. 552–556.

[24]Oberkampf, W. L., Aeschliman, D. P., Tate, R. E., and Henfling, J. F., "Experimental Aerodynamics Research on a Hypersonic Vehicle," Sandia National Lab., SAND92-1411, Albuquerque, NM, April 1993.

[25]Aeschliman, D. P., Oberkampf, W. L., and Henfling, J. F., "Fast-Response, Electronically-Scanned Multi-Port Pressure System for Low-Pressure Hypersonic Wind Tunnel Applications," AIAA Paper 94-2580, June 1994.

[26]Payne, J. L., and Walker, M. A., "Verification of Computational Aerodynamic Predictions for Complex Hypersonic Vehicles Using the INCA™ Code," AIAA Paper 95-0762, Jan. 1995.

[27]Oberkampf, W. L., Aeschliman, D. P., Henfling, J. F., and Larson, D. E., "Surface Pressure Measurements for CFD Code Validation in Hypersonic Flow," AIAA Paper 95-2273, June 1995.

[28]Oberkampf, W. L., Aeschliman, D. P., Henfling, J. F., Larson, D. E., and Payne, J. L., "Surface Pressure Measurements on a Hypersonic Vehicle," AIAA Paper 96-0669, Jan. 1996.

[29]Oberkampf, W. L., Aeschliman, D. P., Henfling, J. F., Larson, D. E., and Payne, J. L., "Surface Pressure Measurements for CFD Code Validation in Hypersonic Flows," Sandia National Lab., Albuquerque, NM (in preparation).

[30]Reda, D. C., and Aeschliman, D. P., "Liquid Crystal Coatings for Surface Shear Stress Visualization in Hypersonic Flows," *Journal of Spacecraft and Rockets*, Vol. 29, No. 2, 1992, pp. 155–158.

[31]Oberkampf, W. L., Martellucci, A., and Kaestner, P. C., "SWERVE Surface Pressure Measurements at Mach Numbers 3 and 8 (U)," Sandia National Labs., Rept. SAND84-2149, SECRET (Formerly Restricted Data), Albuquerque, NM, Feb. 1985.

S. Glegg
Associate Editor