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

Role of Code Validation and Certification in the Design Environment

T. J. Barber*
United Technologies Research Center, East Hartford, Connecticut 06108

The question frequently asked after a computational fluid dynamics (CFD) solution has been obtained, How do I know my answer is correct?, is considered. Most engineering organizations using CFD codes in support of their design processes attempt to assess this issue and to reduce the risks incurred by evaluating the usability of the results. Whereas there are several forms of risk, the primary issues focused on in this work are accuracy and reduced variability (robustness). The role of benchmark or validation studies in establishing a code's accuracy is examined. Examples are presented illustrating the difficulty in relying on benchmark studies to validate a code for design usage. Ways of reducing code usage variability are also suggested, including performing numerical experiments and design calibrations.

Nomenclature

C_p = pressure coefficient
C_T = thermal scaling coefficient
D = nozzle diameter
M = Mach number
P = pressure
T = temperature
U = axial velocity component
x = axial coordinate
α = angle of attack
γ = ratio of specific heats
η = normalized spanwise location

Subscripts

b = bypass stream condition
c = convective reference
t = stagnation condition
1 = upstream or stator inlet condition
∞ = freestream condition

Introduction

WITH the maturation of computational fluid dynamics (CFD) techniques and the growth of low-cost computer capabilities, a reasonable question that arises is how to qualify a code for use in support of design processes. The issue that faces design organizations and the CFD community is as follows: Can a code provide accurate and reliable (low-risk) results in a timely and affordable manner? The risk issue addresses the question, How do I know my results are correct when one has to provide these results under time constraints? For many years, researchers have attempted to do this by correlating numerical predictions with experimental data. A significant portion of this effort has been associated with calculations for a few discrete cases, called benchmark experiments. Based on these limited studies and early overstatements of CFD capabilities, a somewhat apocryphal law of research has been frequently cited: Nobody believes the numerical results, except the man who programmed it. Everybody believes the wind-tunnel data, except the man who tested it.

Requirements other than accuracy that can impact the use of CFD codes are the types of design they will be applied to (range); as well, time and cost (turnaround, elapsed, etc.) can be program drivers. The design of conventional products can be heavily influenced by the cost and time factors. More unusual design envelope applications, however, are driven by the inability of ground test facilities to simulate a full range of flight parameters (hypersonic flight). CFD can represent the only feasible or economical means to scale to flight conditions.

During the last few years, several investigators have focused on defining processes for demonstrating or assessing the capabilities of CFD codes to solve benchmark and practical, design-type problems. To provide some structure to these efforts, Melnik et al.[1] suggested that three distinct processes are needed, i.e., code verification, code validation, and code certification. *Code verification* is the process of testing a code to establish its ability to numerically solve the specific set of governing equations and boundary conditions within a defined level of accuracy, e.g., numerical accuracy. This is established by both grid refinement studies and comparisons with exact solutions to model problems. *Code validation* is the process of testing a code to accurately model the critical physics over a specified range. This is established by comparison with benchmark experimental data. Finally, *code certification* is the process of establishing a code's ability to reliably determine the design/performance data of realistic configurations over a clearly defined operating envelope in a timely and easy manner. As in the code validation process, this is established by comparisons with experimental data (on representative configurations). The effectiveness of all three processes is highly dependent on understanding 1) the physics of the problems under consideration, 2) how the experimental data were obtained, and 3) how the model problems relate to the real problem. On the other hand, what the design community needs is a predictive tool that has a proven reliability, and therefore one should recognize that, instead of validating a code, one really is interested in correlating the predicted results with experimental data or designer experience.

The primary focus will be to examine how most engineering organizations attempt to assess the usability and the risks incurred from using CFD codes in support of their design processes, by improving a code's accuracy and reducing its predictive variability. More specifically, this paper will examine how the concepts of risk reduction, applied to the code validation process, can be used to assess the role of performing benchmark studies (establishing a code's accuracy) and how these studies fail to properly address the overall risk issue.

Risk Reduction Process

Risk reduction to an engineering design team is rooted in a determination to deliver reliable engineering data having a specified level of accuracy, in a specified time, and for a specified cost. Risk analysis is a process introduced as an integral part of engineering design programs as a means of examining the probability and consequences of failure, either of any of the components of the processes leading to or the components resulting from the design program. The risk

Presented as Paper 96-2033 at the AIAA 27th Fluid Dynamics Conference, New Orleans, LA, June 17–20, 1996; received Oct. 22, 1996; revision received March 27, 1997; accepted for publication April 7, 1997.

*Manager, Physical and Mathematical Modeling. Associate Fellow AIAA.

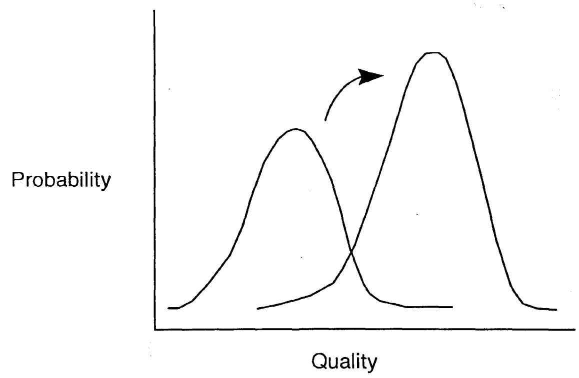

Fig. 1 Improved results through process control.

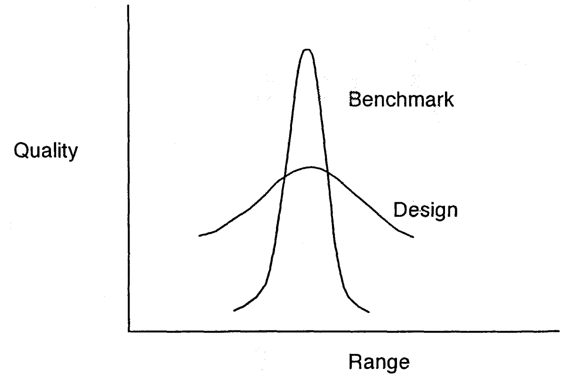

Fig. 2 Schematic view of solution goals.

assessment process is essentially an information evaluation process, where one performs a probability assessment to determine the ability to deliver quality or accurate data at a prescribed cost and in a specified time, a consequence evaluation to determine the effect of errors on the predicted data, and a sensitivity analysis to determine the range of applicability of the data.

To perform these tasks, a requirements-oriented approach is typically followed. One first identifies what the requirements of the predicted data are. This is best determined by conducting expert interviews with end users of the data. These interviews would establish the design issues associated with each requirement, the tolerable error bands (metrics), and the relative risk if the data do not meet the prescribed metric. The risk reduction process should also identify the means for achieving and verifying the desired metrics.

In applying these ideas to CFD codes and their application to the design process, Cosner[2,3] has largely focused on the process-related issues of reliability, time, and cost. He viewed CFD analyses in terms of a user's probability to deliver a quality solution to a design team. Cosner's view, shown schematically in Fig. 1, can be achieved by first understanding the analysis process and then using an optimized application of user experience factors, e.g., standard work, skilled users, robust codes, etc., to achieve high-quality solutions with high probability.

Instead, we shall focus on the issues of accuracy and achieving reduced variability (robustness), wherein a range of physical processes and a variety of desired outcomes can occur. Most organizations seek to achieve this goal by performing validation or certification studies. When the end user is a research group, the metrics for evaluating the accuracy of the code (desired outcomes) are typically fundamental flow variables (streamlines, velocity profiles, etc.). However, when the end user is a member of a design team, the metrics are more performance oriented, e.g., lift and drag coefficients, system efficiency, etc. The central basis of the validation or certification process is benchmarking, whereby a limited number of numerical predictions are made and compared to experimental data. Even though one successfully performs such calculations, the choice of cases frequently is not appropriate for minimizing the risk of providing faulty data for a given design process. Significant risks can be introduced by, for example, poorly defined design requirements. This can occur from a failure to choose the test cases (validation/calibration) from an operational or end-user perspective, instead choosing them from a research perspective.

This paper, however, will not address such occurrences. Instead, it will consider the risks associated with correctly chosen benchmark cases. Several code validation/certification studies, some of which have been cited in the published literature, will be discussed. These studies will be reviewed, and disagreements between experiment and numerical predictions will be discussed.

Benchmark Process

Validation and certification of CFD codes are undertaken to establish the range of expected outcomes and to assess the fitness of those outcomes in providing reliable types of data for either research or design purposes. These processes can be viewed from a point-of-view alternative to Cosner,[2,3] as seen in Fig. 2. Successful benchmark calculations, in reality, provide user confidence only for a narrow range of the design space. The analysis (CFD) outcomes will have varying degrees of success (from good to poor), based on the range of applicability around the benchmark space, on the aggressiveness of the application, and on the expertise of the user. In practice, however, designers are willing to accept some degree of performance degradation for a wider range of problem applicability.

Bobbitt[4] considered CFD code benchmarking from an interesting perspective. He defined a validated code as one that has been used to perform a substantial number of comparisons with data from experimental investigations over a broad range of conditions and obtained good agreement. Bobbitt also reported that, for these comparisons to be meaningful, both the experimental and computational errors need to be assessed or reduced. Bobbitt referred to typical sources of experimental and numerical error, all of which are normally well understood and are frequently assessed. Experimental errors arise from the effects of tunnel-wall interference, sting mounting, Reynolds number mismatch, flow quality, measurement errors, boundary-layer transition, etc. Numerical errors arise from poor grid resolution/distortion, artificial dissipation, round off, truncation, and turbulence and transition models, etc. Freijtas,[5,6] Celik,[7] Karniadakis,[8] and Roache[9] have proposed documentation requirements for identifying, separating out, and estimating spatial (and temporal) grid-induced errors, as well as specifying any stopping criteria for iterative calculations.

A major problem is that any observed discrepancy between experiment and computation is rationalized in terms of the already noted error sources until some positive degree of agreement is achieved or is finally relegated to turbulence modeling deficiencies. The biggest reasons for these observed discrepancies, however, is in the lack of details provided by both the computational and experimental investigator. In performing benchmark calculations, an investigator imposes available experimental data as boundary conditions (BCs) but must frequently assume additional information to properly pose the numerical problem. Solutions are frequently obtained by perturbing BCs in attempts to explain any comparison disagreements. Other missing details, however, may more readily explain these disagreements.

In the next two sections, the text will address several aspects of the benchmarking process, as well as discuss approaches for reducing the reliance (variability) on the expertise of the user. Several examples will be cited wherein missing experimental details have led to poor benchmark calculation comparisons.

Understanding Benchmark Experiments

Even with careful experiments and careful CFD calculations, correspondence is frequently difficult. Published literature cites extensive sources of quality data suited for benchmarking a code's accuracy. CFD analysts are dependent on these data and the details of the experiment to perform their simulations. Analysis differences have been attributed to standard real-world test and code errors. Often, however, these differences may be due to other issues. Experience has shown, however, that the devil is in the details and that these details are typically not documented. This is especially true for the representative configuration necessary for code certification. The following benchmarking studies highlight how the missing details can lead to false conclusions. In the case studies cited, the details discussed are 1) the geometrical definition, 2) the data reduction procedures, and 3) the dominant physics.

Geometry Definition

In performing comparisons with experimental data, it is expected that one must be careful to accurately model the flight or test geometry. Frequently, the tested configuration has not been built to design specifications, and code validation studies need to obtain the model coordinates from direct measurement. A more difficult issue to address, however, is that the flight configuration may not model the static configuration. Few tests are carried out where the shape of the model under the design load is determined. In the turbomachinery community this is frequently called a cold-to-hot part conversion. Rotating machinery deflects and twists under centrifugal loads and grows under thermal loads. Examples of this phenomenon have been encountered in the benchmarking studies for the NASA Rotor 67 and M100 wing/body configurations.

NASA Rotor 67

Rotor 67 is a 22-bladed, low-aspect-ratio fan[10,11] tested at NASA. This configuration has been extensively used for evaluating a turbomachinery code's ability to predict the three-dimensional, transonic flow through a rotating fan blade passage. The fan was tested in a rotor-only configuration to achieve steady flow in the relative frame of reference. Numerical predictions obtained using the ADPAC Navier–Stokes code[12–14] have been compared to NASA measurements. The computational grid for a Rotor 67 calculation is shown on Fig. 3. The geometry was modified from its static state to reflect centrifugal untwist effects. New rotor coordinates (hot or deflected coordinates) were developed using beam theory approximations. Disagreements were noted when one compared the predicted performance of the rotor to the experimental data. The mass flow rate through the fan was substantially lower than the measured value. Laser velocimetry experiments conducted by NASA demonstrated that the blade geometry was not located where expected. New rotor coordinates were developed from a NASTRAN analysis, and the calculations finally achieved the design mass flow rate.

Aircraft Research Association M100 Wing/Body

Similar experiences have been encountered in code verification studies for the M100 wing/body transonic transport configuration.[15] The M100 configuration was extensively studied in the United Kingdom's Aircraft Research Association's (ARA) 9 × 8 ft wind tunnel. The fuselage is 1.65 m long. The wing has a span of 1.81 m and a 5-deg dihedral. A plan view of the configuration and the spanwise distribution of pressure taps is shown in Fig. 4. Data were obtained at a freestream Mach number of about 0.8 for a range of angles of attack (Table 1). Calculations were obtained using the NASA Langley Research Center TLNS3D Navier–Stokes code[16,17] and were compared to the measured data. Initial comparisons showed an overprediction of the wing lift by 15%. A closer examination of the test report indicated that the wing tip showed some deflection under load. An estimated 0.25-deg, nose-down untwist of the tip was applied, resulting in a reduction of the lift overprediction to 7%. The effects of the original or cold geometry and the untwisted or hot geometry of the wing loading are seen in Fig. 5 for three spanwise locations.

Data Reduction Procedures

Benchmark experiments utilize a variety of instrumentation to obtain their cited data. All instrumentation, even those that have been carefully calibrated, have some degree of error associated with them, and the data obtained from a particular measurement technique should have an associated error band. A bigger problem, however, is that the data reduction procedures for a given measurement technique invoke certain approximations that are typically not defined in the published reference. Examples of such data sets and how this misinterpreted data impacted CFD benchmark studies are discussed next.

Table 1 ARA M100 case: M_∞ = 0.8011 and α = 0.148

Parameter	Experiment	Base wing	Untwisted wing
C_L	0.3195	0.3674	0.3418
%, ΔC_L	——	15	7

NASA Rotor 67

Referring back to the Rotor 67 example,[18,19] one finds additional problems in the benchmark calculation, even after the geometry was corrected. The predicted efficiencies were initially found to be in poor agreement with measured data near the peak of the performance characteristic (high flow). Attempts at achieving improved predictions were made, using a finer numerical mesh of 400,000 points, with little improvement. A closer examination of the data indicated that the experimental value of the efficiency was obtained

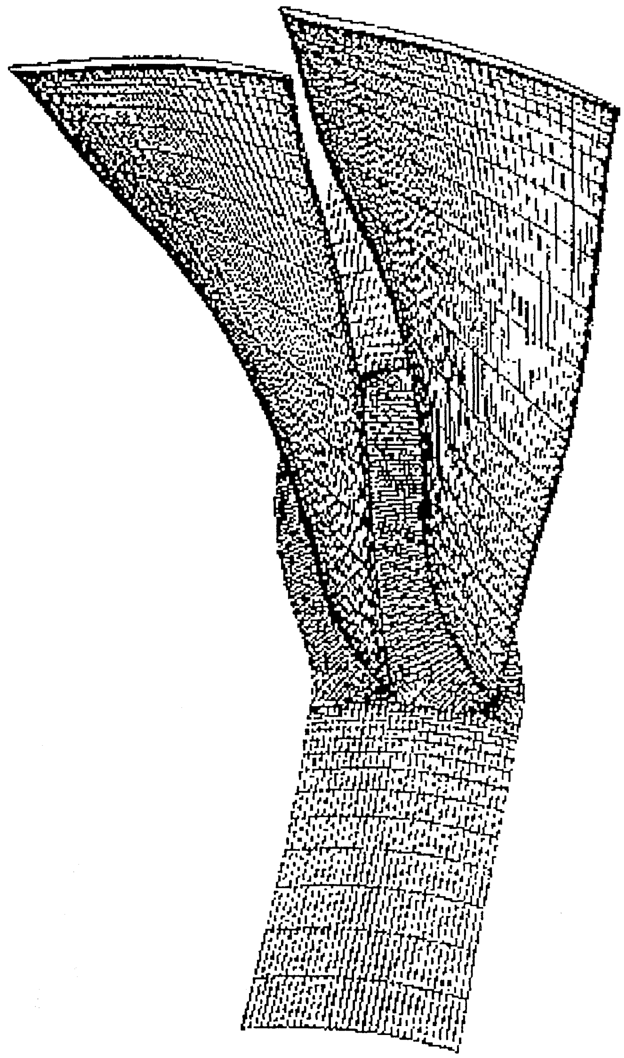

Fig. 3 Computational grid for NASA Rotor 67.

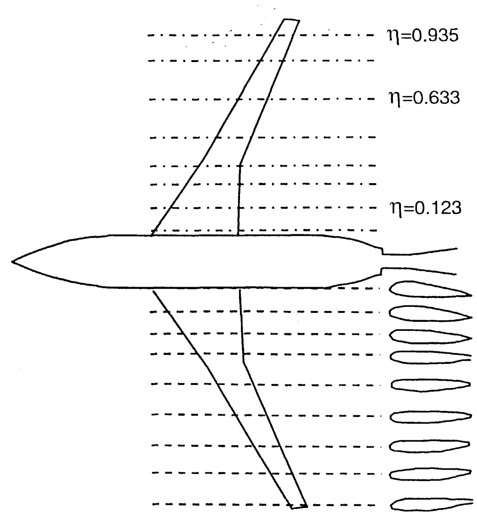

Fig. 4 Plan view of ARA M100 wing/body geometry.

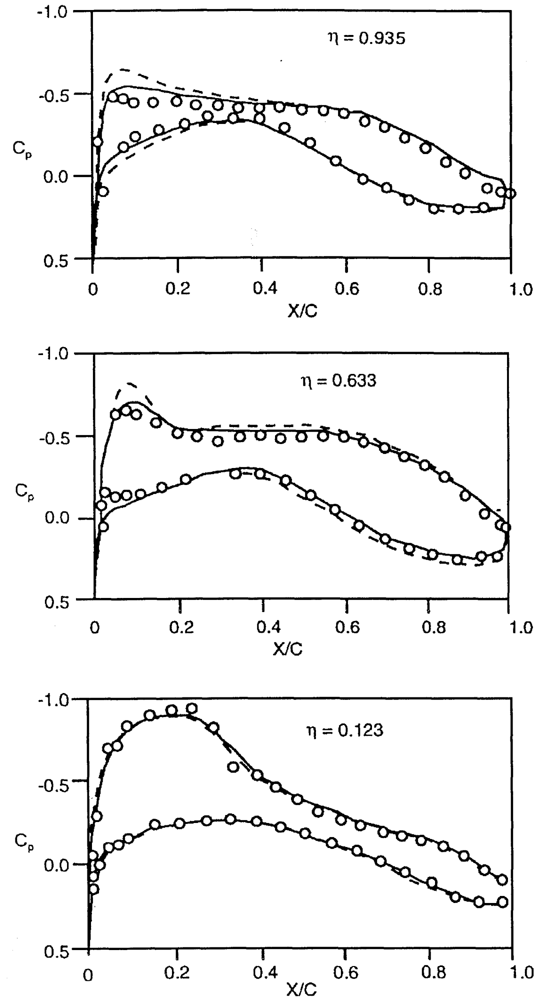

Fig. 5 Surface pressure comparisons for untwisted and base M100 wing configurations for $M_\infty = 0.8$ and $\alpha = 0.148$ deg: ○, ARA data; ---, original wing; and ——, untwisted wing.

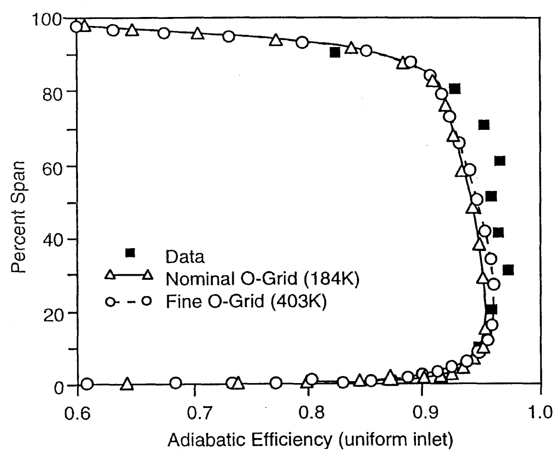

Fig. 6 ADPAC adiabatic efficiency prediction for Rotor 67.

by integrating the profile obtained from an 11-element probe, with no data included from the wall boundary layer. When the numerical results were recalculated by interpolating the computational data to the actual probe locations, efficiencies were obtained that were about 0.8% higher and closer to the experimentally observed values, as seen in Fig. 6. A lesson to be learned from this study, therefore, should be that numerical comparisons with experimental data, conducted in validation/certification studies, should be performed using the same procedures as were used to reduce the experimental data.

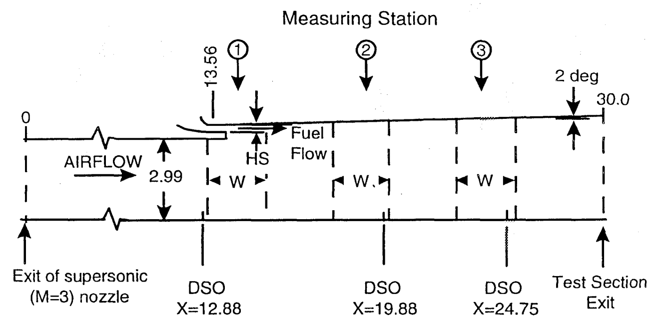

Fig. 7 H_2 scramjet combustor test section.

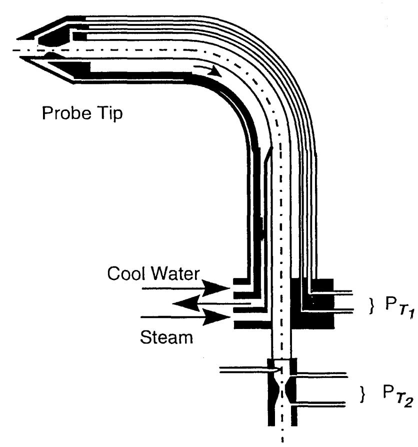

Fig. 8 DSO probe cross section.

United Technologies Research Center H_2-Fueled Scramjet

Experimental and analytical studies of the supersonic mixing and combustion processes in a scramjet combustor[20,21] have been conducted. The numerical simulations were made using the UT-SPARK time-marching Navier–Stokes code.[22,23] Turbulent mixing processes were modeled using the k–ε turbulence model. Initial conditions for the test section simulation were obtained by assuming that the hydrogen, oxygen, and air in the vitiated-air heater reacted completely so that the mass fractions of nitrogen, oxygen, and water vapor at the entrance to the test section were computed from metered flow rates into the heater. Experiments were conducted in the United Technologies Research Center's (UTRC's) jet burner test facility. Gaseous hydrogen was injected tangentially into a Mach 3 (nominal) vitiated airstream (Fig. 7). In the experiment, hydrogen was injected adjacent to the upper wall of the test section. Some of the details of the flow conditions had to be assumed because it was not practical to measure all of the flow boundary conditions.

Stagnation temperature data and pitot pressure data were acquired using the dual-sonic orifice (DSO) probe (Fig. 8). Determination of stagnation temperature using this method is quite difficult, especially where stagnation temperatures within the flame are high. (This prohibits use of thermocouple probes.) Comparisons were obtained for a mixing-only case (primary flow freestream total temperature = 1698°R) having an effective core Mach number of 2.77. The injected or slot fuel flow had a total temperature of 517°R at an equivalence ratio of 0.95. In the mixing-only case, test section calculations were initialized using measured values of P_t and T_t. All other inlet plane BCs were obtained from calculations performed from the subsonic plenum downstream, especially the gas composition entering the test section.

A comparison between computed and measured stagnation temperature profiles (Fig. 9) shows a discrepancy of 60–80 K in the core flow region. However, because the flow is nonreacting, one should expect that the core flow total temperature should match the boundary condition specified at the test section inlet plane. A closer examination of the DSO probe data reduction procedures indicated that an effective specific heat ratio γ was assumed. This value was different from the value determined from the numerical simulation using a finite rate kinetics model. Eliminating this difference

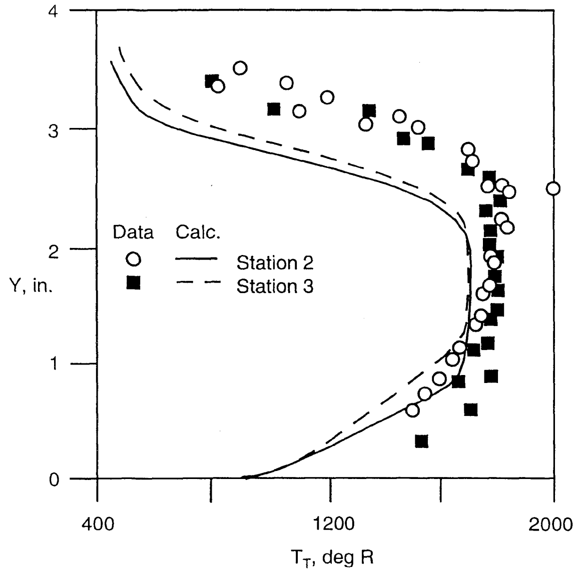

Fig. 9 Stagnation temperature profiles for H_2-fueled scramjet.

resolved the numerical/experimental differences. A lesson from this study, therefore, should be that numerical comparisons with experimental data, conducted in validation/certification studies, should be performed using the same flow approximations as were used to reduce the experimental data.

UTRC Large-Scale Rotating Rig $1\frac{1}{2}$-Stage Turbine

Another example of possible data reduction misinterpretation can be found in the recent comparisons of multiblade row turbomachinery predictions of the impact of a combustor hot spot (streak) on the thermal characteristics of a turbine rotor. Experiments have shown that hot streaks, arising from incomplete mixing in the combustor, can lead to pressure-side hot spots on first-stage turbine rotor blades. Hot streak experiments have been conducted at UTRC for a $1\frac{1}{2}$-stage turbine in the large-scale rotating rig (LSRR).[24,25] A hot streak was introduced through a circular pipe at 40% span, midway between two stator airfoils. The hot streak was seeded with CO_2, and the path of the hot streak was determined by measuring CO_2 concentrations at various locations within the turbine stage. A one-to-one scalar transport correspondence was assumed between the CO_2 concentrations and the hot streak temperature distributions. Several different nondimensional scaling techniques have been developed to relate blade-surface temperatures to the measured concentrations of CO_2. Normalized temperature coefficients $\tilde{C}_T$ (scaling 1) have been defined in terms of time-averaged local T and leading edge T_{le} properties along the rotor surface to display the data, where

$$\tilde{C}_T = \frac{T - T_1}{T_{le} - T_1}$$

Numerical predictions[26] have also been obtained for the same configuration using a derivative form of the NASA Ames Research Center ROTOR Navier–Stokes code. A two-dimensional, constant-span slice through the computational grid is shown in Fig. 10. Discrepancies between data and predictions have in the past been attributed to three-dimensional flowfield effects.

Recently it has been proposed that the assumed correlation between surface temperature and experimentally measured CO_2 concentrations may be incorrect. In the experiment, the CO_2 concentrations were determined by drawing samples of the gas in through surface static pressure taps. The suction force may cause CO_2 gas well above the airfoil surface to be included in the surface sample. Defining a new coefficient (scaling 2) based on the area-averaged local and leading edge temperature through the boundary layer results in remarkably improved agreement between experimental data and numerical predictions. Figures 11 and 12 shows these comparisons for the basic temperature coefficient and for the reinterpreted coefficient.

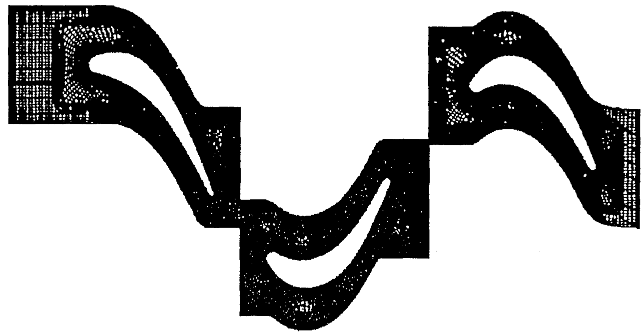

Fig. 10 Computational grid for UTRC $1\frac{1}{2}$-stage LSRR turbine.

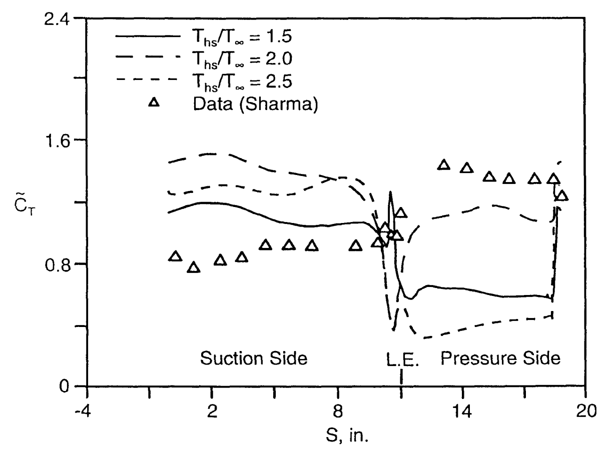

Fig. 11 Temperature coefficient distribution for LSRR turbine rotor, scaling 1.

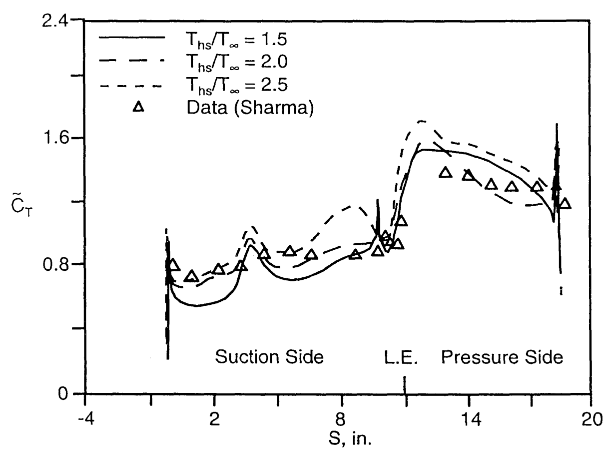

Fig. 12 Temperature coefficient distribution for LSRR turbine rotor, scaling 2.

Dominant Physics Issues

Finally, we address issues associated with benchmarking studies where incomplete or incorrect understanding of the dominant physical processes can lead to incorrect simulations and false conclusions. In the following examples, we are not considering obvious errors, such as using an Euler analysis to model the flow over a stalled airfoil.

Lepicovsky Heated Round Jet

The analysis of jet flow mixing characteristics is a critical issue for many current NASA-sponsored programs. Key issues of interest are the influence of thermal and compressibility effects on the rate of mixing. In most jet flow validation studies, calculations are performed from the nozzle exit plane using ideally expanded flow conditions and an assumed top hat profile or an experimental profile (if available). Lepicovsky[27] performed a detailed experimental study of the temperature and Mach number characteristics of heated round jets into quiescent air over a range of jet Mach numbers from 0.1 to

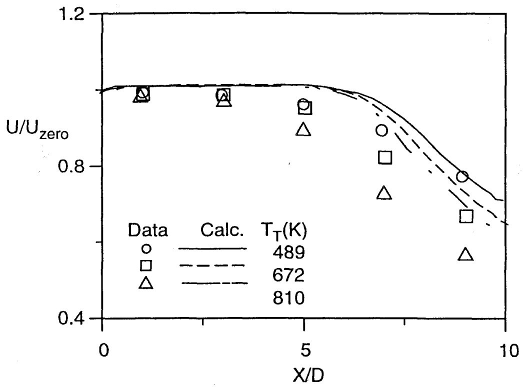

Fig. 13 Centerline Mach number distribution for Lepicovsky[27] jet.

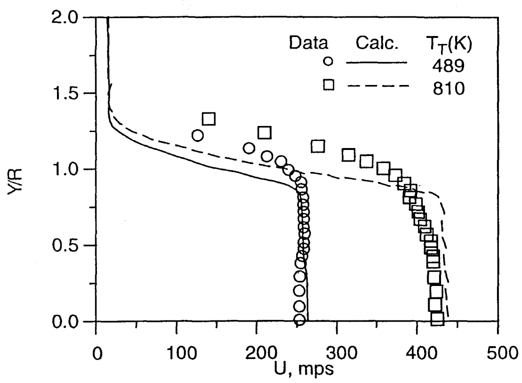

Fig. 14 Nozzle exit plane radial velocity profiles for Lepicovsky[27] jet.

0.9 and jet total temperatures from 290 to 810 K. All experimental data as well as nozzle geometry definition are available.[28] In validation calculations for the $M_j = 0.8$ case performed at UTRC using the NASTAR Navier–Stokes analysis[29] and the Jones–Launder form of the k–ε turbulence model, discrepancies were identified in the centerline mixing characteristics (Fig. 13). The calculations predict a slightly reduced potential core, substantially less than observed. Initial thoughts were that these errors were due to thermal effects not modeled by the turbulence model. In the calculations, a constant turbulent Prandtl number of 0.9 was assumed. A closer look at the experiment, however, indicated that at the cold operating point the jet Reynolds number based on the exit nozzle diameter was about 1.2×10^6, but when the jet was heated, the Reynolds number dropped to 3×10^5. Examination of the exit plane profile verified that the heated jet exit flow was laminar-like (Fig. 14) and had a shape factor of 2.5, whereas the colder jet case profile is clearly turbulent. Clearly an assumed top-hat-type profile or a fully turbulent starting calculation would not model the heated jet flow physics.

High-Speed Civil Transport Compound Choked Ejectors

Quasi-one-dimensional analytical analysis of the mixing processes of a supersonic primary nozzle and an entrained (pumped) subsonic flow allows for two possible mixed-out solutions, one subsonic and one supersonic. The subsonic solution, however, is inconsistent with the restrictions imposed by the second law of thermodynamics. Numerical solutions, obtained by Navier–Stokes solvers for high-speed civil transport, low-noise exhaust system concepts have produced both subsonic and supersonic solutions from arbitrary initializations of the iteration process. Whereas revised initialization procedures have eliminated the subsonic path, one must realize that physically inconsistent but mathematically possible solutions can be predicted by CFD codes.

Reducing Code Usage Variability

As cited earlier, Cosner[2,3] noted that improved data usually can be obtained from analyses executed by expert users of the codes or through attention to process issues. One can state that expert outcomes/results can be established for a selected class of problems by an expert user using a defined set of technology (specific code).

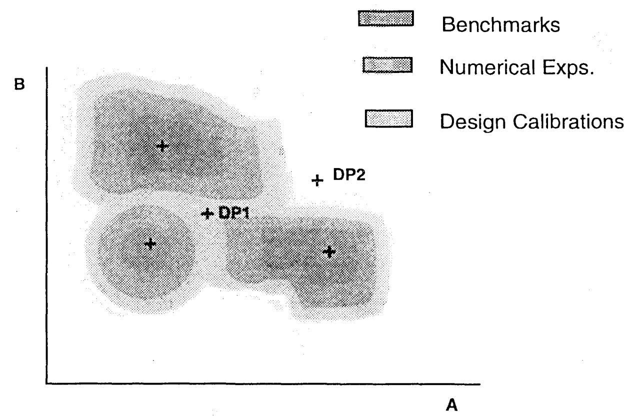

Fig. 15 Schematic view of design space credibility range.

This type of validation has little direct value to industry, where the emphasis is placed on producing results of acceptable quality, cost, and schedule.

The problem, however, is how to eliminate sources of variability/risk without relying on this level of expertise on a day-to-day basis. In essence, the problem translates to eliminating human error, controlling solution management, developing improved models, and providing quality geometry and grid. Typically, this is not possible without an integrated and standardized analysis process.

The outstanding problem still at hand is how a designer is to know whether the code predictions are correct, especially for new design problems. If one views an arbitrarily defined design space in terms of two sample design variables, such as Mach number and angle of attack, or design features, such as shocks and separation bubbles, as denoted by A and B in Fig. 15, the shaded areas represent regions for which various degrees of code validation have been successfully accomplished. DP1 represents a new design point. Implicit is a third axis representing the solution quality function. These regions, however, can be quite localized and, therefore, do not map the complete design space. Mixed experiences resulting from many benchmark studies suggest that the best way to minimize the variability/risk of CFD code use over a range of applications is not to rely solely on the validation process but to complement them with numerical experiments and design calibrations. Numerical experiments include studies in which flow conditions are parametrically varied to ensure a degree of continuity in the design space. These studies develop the iso-confidence contours around the shaded regions.

CFD users frequently conduct numerical experiments to identify the sensitivities of their calculations to their input data, e.g., BCs. For example, whereas calculations may specify inlet values for the total pressure and total temperature based on known data, turbulence kinetic energy k and turbulence dissipation ε are typically not known but are assumed. Reference 30 illustrates one such study, where several CFD analyses were applied to analyze the flow mixing characteristics of an axisymmetric heated supersonic jet operated at perfectly expanded conditions. Initial calculations assumed the primary jet flow to have a jet primary flow turbulence level of 1%. Sensitivity studies determined that specifying turbulence levels of up to 10% had little effect of the downstream mixing of the jet.

Design calibrations are more physically based. Looking for a close correlation with test data has been the context of most CFD validation studies. Because the experimental data represent a sample of the behavior of the system at some imprecisely known set of parameters (Mach number, Reynolds number, angle of attack), a better approach is suggested where one should look for correlations on a regional basis and match trends. This corroborates designer experience and verifies that predicted results follow plausible behavior patterns. Design teams surveyed at Boeing[31] emphasized that validation efforts should focus on the running of a large number of cases, with the intent of correlating CFD predictions with experimental data. Design engineers typically look for comparisons of CFD results to test data to be close enough to their expected design space, especially because most designs are a negotiated compromise of many different technology disciplines (including manufacturability and cost). The main goal of the engineering design process is to provide sufficient data to the decision-making groups so that intelligent tradeoffs and compromises can be made. The issue that still exists is what happens when the intended design point (DP2) occurs outside of the mapped design space defined in Fig. 14.

Practicalities of time and budget frequently limit such efforts; however, regular design experience gained from other analysis and experimental efforts are incorporated as rules to follow to assure that successful simulations will be obtained. An example of this process is applied when an experienced CFD user develops a computational grid to determine the performance characteristics of a complex geometry in a turbulent flow. Near-wall turbulence models are chosen to minimize grid requirements and improve analysis turnaround time. One option is to apply wall integration (low-Reynolds-number model), which is considered more accurate but which requires substantial mesh resources and clustering so that $y+$ (first point off the wall) is of order unity. An alternative and less mesh-intensive option is to apply wall functions (high-Reynolds-number model) with $y+$ of order 50–100. Avva et al.[32] and Georgiadis et al.[33] have shown that both approaches can give accurate wall friction predictions; however, both models can also produce inaccurate results when applied on an inappropriately distributed mesh. Therefore, experience leads the trained user to develop the appropriate grid for the applied turbulence model.

Concluding Remarks

This paper considers the question frequently asked after a CFD solution has been obtained: How do I know my answer is correct? Most engineering organizations using CFD codes in support of their design processes attempt to reduce the risks incurred by evaluating the usability of the results. Risk issues associated with cost and time factors can be minimized through process control approaches. Risks associated with code accuracy are normally addressed through benchmark or validation studies. Examples are presented illustrating the difficulty in relying on benchmark studies to validate a code for design usage. More specifically, shortcomings related to definition of the geometry, to the data reduction procedures, and to the understanding of the dominant physics make it extremely difficult to rely on benchmark studies. Therefore, it has been proposed that benchmark studies be augmented with additional calculation approaches, including performing numerical experiments and design calibrations, and that a balanced program of these represents the best way of reducing risks associated with code usage in a broad design environment context.

Acknowledgments

The author wishes to thank John F. Dannenhoffer for his insightful comments and contributions to the preparation of this paper. The NASTRAN analysis that resulted in new rotor coordinates for the NASA Rotor 67 is due to J. R. Wood, June 1996.

References

[1]Melnik, R. E., Siclari, M. J., Marconi, F., Barber, T. J., and Verhoff, A., "An Overview of a Recent Industry Effort at CFD Code Certification," AIAA Paper 95-2229, June 1995.

[2]Cosner, R. R., "CFD Validation Requirements for Technology Transition," AIAA Paper 95-2227, June 1995.

[3]Cosner, R. R., "CFD Applications Bottlenecks," Workshop for Improving the CFD Applications Process, Stanford Univ., Stanford, CA, Nov. 1994.

[4]Bobbitt, P. J., "The Pros and Cons of Code Validation," AIAA Paper 88-2535, 1988; also NASA TM 100657, July 1988.

[5]Freitas, C. J., "Perspective: Selected Benchmarks from Commercial CFD Codes," *Transactions of the American Society of Mechanical Engineers*, Vol. 117, June 1995, p. 208.

[6]Freitas, C. J., "Editorial and Statement on the Control of Numerical Accuracy," *Journal of Fluids Engineering*, Vol. 115, Sept. 1993, pp. 339, 340.

[7]Celik, I., "Numerical Uncertainty in Fluid Flow Calculations," *Journal of Fluids Engineering*, Vol. 115, June 1993, pp. 194, 195.

[8]Karniadakis, G. E., "Toward a Numerical Error Bar in CFD," *Journal of Fluids Engineering*, Vol. 117, March 1995, pp. 7–9.

[9]Roache, P. J., "Perspective: A Method for Uniform Reporting of Grid Refinement Studies," *Journal of Fluids Engineering*, Vol. 116, Sept. 1994, pp. 405–413.

[10]Strazisar, A. J., Wood, J. R., Hatheway, M. D., and Suder, K. L., "Laser Anemometer Measurements in a Transonic Axial-Flow Fan Rotor," NASA TP 2879, Nov. 1989.

[11]Strazisar, A. J., Wood, J. R., Hatheway, M. D., and Suder, K. L., "Test Cases for Computation of Internal Flows in AeroEngine Components," *Case E/CO-2: Rotor 67*, edited by L. Fottner, AGARD No. 275, 1990.

[12]Hall, E. J., Delaney, R. A., and Bettner, J. L., "Investigation of Advanced Counterrotation Blade Configuration Concepts for High Speed Turboprop Systems: Task I—Ducted Propfan Analysis," NASA Contract NAS3-25270, NASA CR 185217, April 1990.

[13]Hall, E. J., and Delaney, R. A., "Investigation of Advanced Counterrotation Blade Configuration Concepts for High Speed Turboprop Systems: Task V—Counterrotation Ducted Propfan Analysis, Final Report," NASA Contract NAS3-25270, NASA CR 187126, Jan. 1993.

[14]Hall, E. J., and Delaney, R. A., "Investigation of Advanced Counterrotation Blade Configuration Concepts for High Speed Turboprop Systems: Task V—Counterrotation Ducted Propfan Analysis, Computer Program Users Manual," NASA Contract NAS3-25270, NASA CR 187125, Jan. 1993.

[15]Carr, M. P., and Pallister, K. C., "Pressure Distributions Measured on Research Wing M100 Mounted on an Axisymmetric Body," AGARD AR-138 Addendum, July 1984.

[16]Vatsa, V. N., Sanetrik, M. D., and Parlette, E. B., "Development of a Flexible and Efficient Multigrid-Based Multiblock Flow Solver," AIAA Paper 93-0677, Jan. 1993.

[17]Marconi, F., Siclari, M., Carpenter, G., and Chow, R., "Comparison of TLNS3D Computations with Test Data for a Wing/Simple Body Configuration," AIAA Paper 94-2237, June 1994.

[18]Barber, T. J., Choi, D., McNulty, G. S., Hall, E. J., and Delaney, R. A., "Preliminary Findings in the Certification of ADPAC," AIAA Paper 94-2240, June 1994.

[19]Barber, T. J., Choi, D., McNulty, G. S., Hall, E. J., and Delaney, R. A., "Advanced Ducted Profan Analysis (ADPAC) Code Certification," NASA CR (to be published).

[20]Peschke, W. O. T., Chiappetta, L. M., and Barber, T. J., "Analytical and Experimental Study of Supersonic Mixing and Combustion of Hydrogen," 26th JANNAF Combustion Meeting, Pasadena, CA, Oct. 1989.

[21]Peschke, W. T., Barber, T. J., Chiappetta, L., Anderson, T. J., and Eckerle, W. J., "Hydrogen-Fueled Scramjet Investigation," U.S. Air Force Research Lab., Draft Final Rept., Contract F33615-86-C-2659, Wright–Patterson AFB, OH, May 1990.

[22]Drummond, J. P., "Numerical Study of a Ramjet Dump Combustor Flowfield," *AIAA Journal*, Vol 23, No. 4, 1985, pp. 604–611.

[23]Drummond, J. P., Rogers, R. C., and Hussaini, M. Y., "A Detailed Numerical Model of a Supersonic Reacting Mixing Layer," AIAA Paper 86-1427, July 1986.

[24]Dring, R. P., Joslyn, H. D., Hardin, L. W., and Wagner, J. H., "Turbine Rotor–Stator Interaction," *Journal of Engineering for Power*, Vol. 104, Oct. 1982, pp. 729–742.

[25]Dorney, D. J., Davis, R. L., and Sharma, O. P., "Two-Dimensional Inlet Temperature Profile Attenuation in a Turbine Stage," International Gas Turbine and Aeroengine Congress, American Society of Mechanical Engineers, ASME Paper 91-GT-406, Orlando FL, June 1991.

[26]Dorney, D. J., "Numerical Investigation of Hot Streak Temperature Ratio Scaling Effects," AIAA Paper 96-0619, Jan. 1996.

[27]Lepicovsky, J., "Total Temperature Effects on Centerline Mach Number Characteristics of Freejets," *AIAA Journal*, Vol. 28, No. 3, 1990, p. 478.

[28]Lepicovsky, J., Ahuja, K. K., Brown, W. H., Salikuddin, M., and Morris, P. J., "Acoustically Excited Heated Jets, III—Mean Flow Data," NASA CR 4129, June 1988.

[29]Rhie, C. M., and Chow, W. L., "Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation," *AIAA Journal*, Vol. 21, No. 11, 1983, pp. 1525–1532.

[30]Barber, T. J., Chiappetta, L. M., Georgiadis, N. J., and DeBonis, J., "An Assessment of Parameters Influencing Prediction of Shear Layer Mixing," AIAA Paper 97-2639, July 1997.

[31]Bussoletti, J. E., "CFD Calibration and Validation: The Challenges of Correlating Computational Model Results with Test Data," AIAA Paper 94-2542, July 1994.

[32]Avva, R., Smith, C, and Singhal, A., "Comparitive Study of High and Low Reynolds Number Versions of k–ε Models," AIAA Paper 90-0246, Jan. 1990.

[33]Georgiadis, N. J., Dudek, J. C., and Tierney, T. P., "Grid Resolution and Turbulent Inflow Boundary Condition Recommendations for NPARC Calculations," AIAA Paper 95-2613, July 1995.

P. R. Bandyopadhyay
Associate Editor