AIAA JOURNAL
Vol. 33, No. 10, October 1995

Dryden Lectureship in Research

Perspective on Computational Fluid Dynamics Validation

Joseph G. Marvin
NASA Ames Research Center, Moffett Field, California 94035

A comprehensive approach to computational fluid dynamics (CFD) validation is presented. Requirements from computational and experimental perspectives are given. Experimental validation is emphasized because it ultimately determines the accuracy of CFD modeling and its application to complex problems. The concepts of building block and benchmark experiments are introduced. The types of measurements required of these experiments and their accuracy determination are explained. Contributions from such experiments toward the development and validation of CFD are reviewed and examples provided. Future challenges and strategies for validation are discussed.

Introduction

COMPUTATIONAL fluid dynamics (CFD) has matured over the past decade to the point where it is having a major impact on aerospace engineering and education.[1] Flowfields around vehicles can be generated numerically and studied like experiments to help understand flow features and determine performance trends. However, the accuracy of the numerical solutions is often uncertain because the numerical techniques and flow modeling assumptions are not adequately validated, and so the solutions are not relied on to provide absolute performance values. As a result, validation, flow physics understanding, and modeling still pace the development of CFD.

Experiment has always played a prominent role in the validation of aerospace design tools and the understanding of fluid dynamics. Although it has been highlighted less frequently, experimental fluid dynamics (EFD) has already played a key role in the development of CFD.[2,3] However, the expectations for use of CFD as an accurate engineering and flow analysis tool places more stringent requirements on experiments supporting code development and its validation.

CFD validation in this paper will refer to the establishment of a correspondence between fluid flows produced numerically by computation and the actual flows produced in nature. To achieve validation, two aspects must be considered: numerical and physical.[4] Considering the numerical aspect is necessary because codes provide approximate solutions to the governing equations, they use discrete grids, they employ algorithms that contain numerical dissipation, and they may have lack-of-convergence errors. Considering the physical aspect is necessary because errors resulting from physical modeling assumptions need to be determined.

The process of achieving CFD validation, often viewed from differing and sometimes broader perspectives, continues to be debated within the CFD Community. Research organizations such as NASA view it from a more fundamental level stemming from a long history in CFD research- and pilot-code development, and they often refer to building block and benchmark experiments as elements of the process. On the other hand, industry views it in a broader sense, having to consider, in addition, the total design process within which the code is used. They often refer to additional experiments performed without the specific motivation of validating CFD as elements in their process.

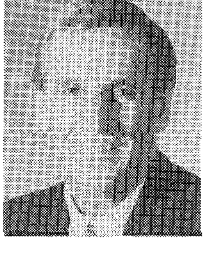
In this paper the validation process will be viewed primarily from the more fundamental perspective. The role of experiment in developing and validating CFD research and pilot codes will be emphasized. Examples of experiments that represent the work of the author and his colleagues at NASA will be given to illustrate progress and the new evolving role of experiment in the validation process. Some discussion on future challenges, strategies, and new instrumentation developments that brighten the future for flow modeling and validation will be discussed.

CFD Validation

Background

Experiment has historically provided the underpinning of validation for aerospace engineering tools and designs, except for high-speed flows where test facilities do not duplicate flight enthalpy and scale. Theoretical and computational approaches together with relevant ground test experiments were essential for such situations. Validation of the theoretical and computational approaches was accomplished in steps. First, by performing small-scale experiments in ground-based facilities to validate the approaches and any related extrapolation techniques and, later, by data from flight experiments with design hardware or prototypes. Some illustrative examples follow.

Theories based on closed-form solutions or numerical integration of limiting forms of the governing equations of fluid motion were verified through small-scale experiments and applied to guide the development and design of aerospace vehicles in the 1960s. An



Joseph G. Marvin is presently the Technical Manager for NASA's Turbulence and Transition Research Program. His professional accomplishments include development of nationally unique experimental facilities and the development, direction, and management of major NASA programs in turbulence modeling and computational fluid dynamics (CFD) code validation. He has served meritoriously on important national technology teams: Space Shuttle Aerothermodynamics team, NASA Boundary Layer Transition team, NASA CFD Validation Team, the National Aerospace Plane (NASP) Aerodynamics Team, and as Chairman of NASP CFD Validation Task Force. He is an Associate Fellow of AIAA.

Presented as Paper 93-0002 at the AIAA 31st Aerospace Sciences Meeting, Reno, NV, Jan. 11–14, 1993; received March 26, 1993; revision received Feb. 27, 1995; accepted for publication March 16, 1995.

example is the validation of a theory developed to predict planetary entry heating.[5] Numerical solutions to the laminar boundary-layer equations for the stagnation point region were computed for air, nitrogen, carbon dioxide, and argon utilizing curvefit models for the equilibrium thermodynamic and transport properties. The computed results were compared with shock tube experiments to establish validation of the numerical technique and the thermodynamic gas chemistry modeling. Theoretical correlations were then developed to enable engineering predictions of heating in planetary atmospheres composed of mixtures of these gases. Figure 1 shows typical comparisons of computation and experiment for argon and carbon dioxide. In the case of carbon dioxide, computations from Hoshizaki[6] were available to provide computation-to-computation validation for the numerical technique. (For this and all remaining examples, error bounds on the data and computations are not provided. The reader is referred to the relevant reference citations from which the material was extracted for this information.)

Later in the decade conceptual and preliminary design of the Space Shuttle vehicle was initiated. The vehicle shape presented formidable challenges to theoretical development and computation due to its three-dimensional geometry and the lack of large, powerful computers. Techniques were devised to circumvent these challenges and simple numerical design tools developed and validated experimentally. Pressures were predicted by inviscid "equivalent" elliptic cone theory and the heating by solutions to the three-dimensional boundary-layer equations employing streamline divergence values obtained from the cone theory. Results were verified[7] with data obtained in a low-enthalpy, high-Mach-number wind tunnel as shown in Fig. 2. Such concepts found their way into industry extrapolation to flight procedures as shown pictorially in Fig. 3. Designers included equilibrium gas chemistry in their procedure for extrapolation of wind tunnel test data to the entry flight conditions.

Emerging computer technology in 1970 enabled more complex computations. It became feasible, for example, to solve the inviscid blunt body and three-dimensional method of characteristics equations for high temperature, chemically reacting airflows. Coupled with three-dimensional boundary-layer computations, flight predictions at various speeds and altitudes along the Space Shuttle vehicle entry trajectory were made.[8] The results could only be verified at the low-enthalpy conditions achieved in wind-tunnel experiments as shown in Fig. 4a. A shortcoming of the computations occurred when the flow became subsonic in the region where the bow shock intersected the wing ($y/L = 0.10$ and $x/L > 0.6$). Nevertheless, these computations provided designers with additional confidence in the extrapolation procedures mentioned previously. Subsequent comparisons of Shuttle windward thermal protection system temperatures measured during flight as shown in Fig. 4b validated the real-gas modeling, including surface catalysis.

The advent of the supercomputer era in the 1970s brought about other major advances. Emphasis in CFD development switched from hypersonic to transonic and supersonic speeds. Solutions to the

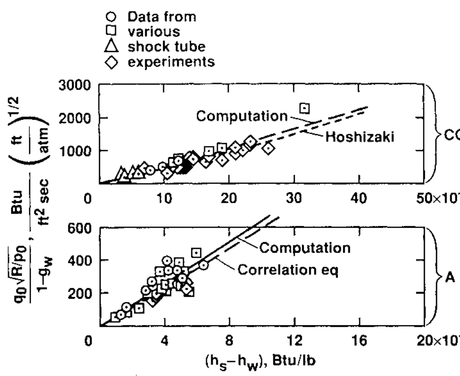


Fig. 1 Verification of stagnation point heating rate computations for planetary atmospheres.

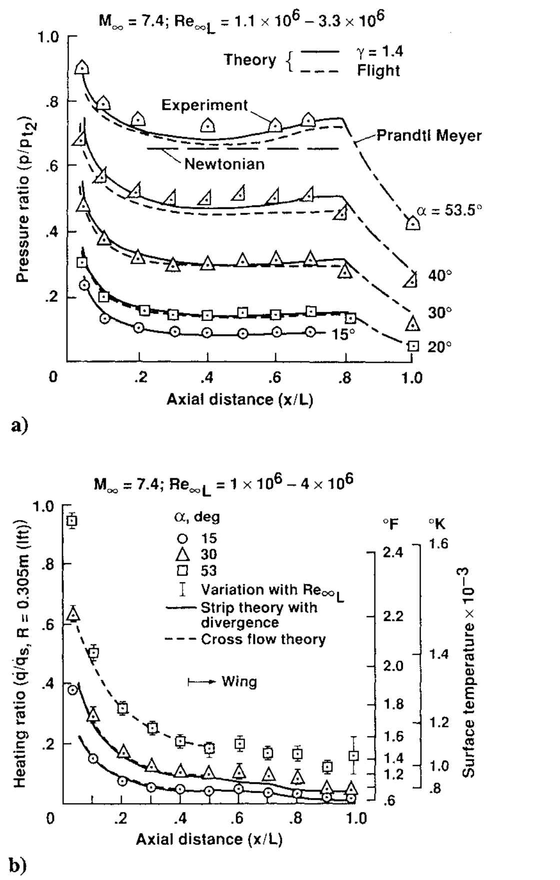


Fig. 2 Verification of approximate CFD methods for predicting Space Shuttle vehicle pressures and heating: a) pressures and b) heating ratio.

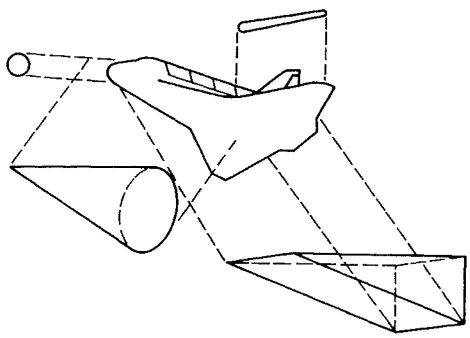


Fig. 3 Sketch illustrating equivalent shapes used in applying approximate CFD methods for extrapolation of wind tunnel data to flight.

transonic small perturbation equations[9] and the laminar form of the Navier–Stokes equations at supersonic speeds[10] were demonstrated. The latter were followed quickly by solutions for turbulent flows employing the Reynolds-averaged Navier–Stokes equations.[11,12] Experiment was used in the computational demonstrations to confirm their legitimacy. However, the database lacked sufficient detail to validate viscous flow code modeling for turbulent flows. At NASA, modeling validation experiments, specifically to support that development, were initiated.[2]

One cooperative experimental and computational study is worth mentioning because it illustrates some of the synergistic payoff that transpired during this period. A circular arc airfoil was tested at transonic speeds[13] in a specially constructed solid wall wind tunnel. Airfoil pressures, tunnel wall pressures, and flowfield velocity profiles obtained with a laser anemometer system were measured. Depending on the experimental Mach number, steady flow or unforced buffet flow developed. The buffet onset condition evidenced hysteresis, and the buffet Mach number domain was different for experiments during which the Mach number was in-

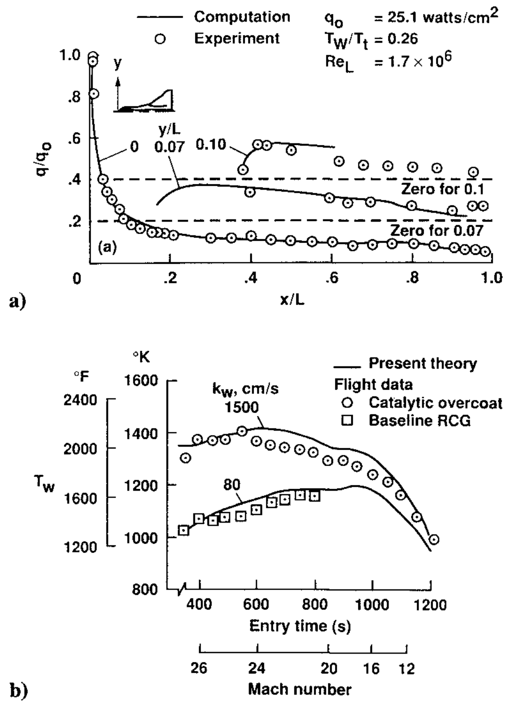


Fig. 4 Validation of Space Shuttle vehicle heating rates and temperatures predicted by a coupled inviscid-viscid computation procedure: a) heating rates at wind tunnel conditions and b) surface temperatures at flight conditions.

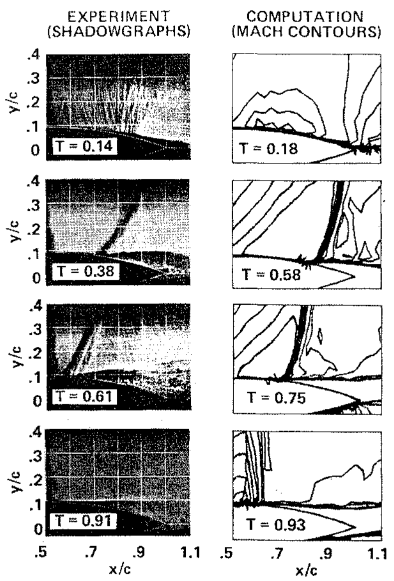


Fig. 5 First demonstration of airfoil buffet computations employing the Reynolds-averaged Navier–Stokes equations.

creased or decreased. Computations,[14,15] using an explicit, time-accurate Navier–Stokes code and a simple turbulence model, were undertaken at three Mach numbers where steady-attached, -attached weak shock, and -shock induced separated flow occurred in those experiments conducted at fixed Mach numbers. At the intermediate Mach number (0.755) the computation was not converging. The experimentalists pointed out that the flow was unsteady at this Mach number for experiments during which the Mach number was decreased once buffet was present and that perhaps the computation was reproducing the correct physics. The computations were continued and run for a time equivalent to several chords of flow travel and periodic buffet flow developed. Figure 5 shows a comparison of a time sequence of experimental shadowgraphs compared with Mach contours from the computations. The magnitude and frequency of the computed buffet compared well with experiment, although the frequency was found to be about 20% less than the experimental value. These studies confirmed for the first time that time-accurate solution methods used to solve the Reynolds-averaged Navier–Stokes equations had the potential for predicting buffet.[15]

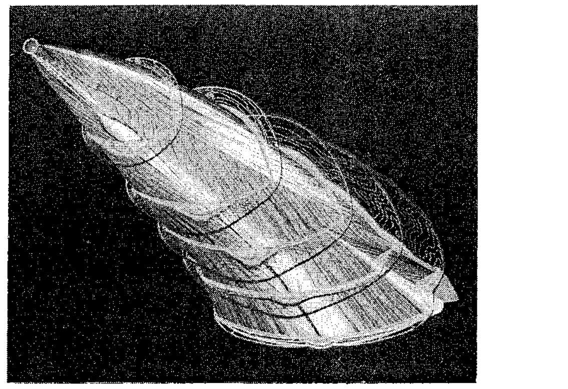


Fig. 6 Mach contours around a generic aerospace plane vehicle; M = 16 and Re/in. = 2 × 10^6.

During the 1980s, CFD matured significantly. Applications became more complex and more design oriented, and industry was deciding on strategies for implementing it in design cycles for new aircraft and aerospace concepts. Furthermore, direct and large eddy numerical simulations of the time-dependent Navier–Stokes equations[16] and complete solutions to the quantum mechanical equations[17] were demonstrated for simple fluid flows and chemical systems, providing new possibilities for flow physics understanding and extraction of new modeling ideas.

CFD developers were being challenged to provide accuracy estimates for their application computations. An illustration of the challenge is aided by Fig. 6. Mach contours around a generic National Aerospace Plane configuration generated with a CFD code[18] that solved the parabolized form of the Navier–Stokes equations are shown for a flight condition. The solutions provided designers with an opportunity to visualize the flowfield and study its impact on design. In this case, a low-momentum region of the flow developed near the lower surface centerline, as indicated by the bulges in the contours, and became severe approaching the inlet region of a propulsion system. Undesirable performance resulted. However, the designers could not be assured of the accuracy of solutions. Therefore, although the computation was useful in sorting out possible design flaws, it could not be relied on for accurate performance estimates.

Dilemmas of this nature were commonplace for many CFD applications, and so NASA initiated a study of a process for CFD validation. Recommendations for a more comprehensive approach to accomplish validation were put forward.

NASA Validation Concepts

The substance of the study was summarized by Bradley.[4] He introduced definitions for code validation and calibration as follows. CFD code validation: Detailed surface- and flowfield comparisons with experimental data to verify the codes ability to accurately model the critical physics of the flow. Validation can occur only when the accuracy and limitations of the experimental data are known and thoroughly understood and only when the accuracy and limitations of the code's numerical algorithms, grid density effects, and physical basis are equally known and understood over a range of specified parameters. CFD code calibration: The comparison of CFD code results with experimental data for realistic geometries that are similar to the ones of interest, made to provide a measure of the code's ability to predict specific parameters that are of importance to the design objectives without necessarily verifying that all of the features of the flow are correctly modeled. During the intervening years since these definitions were formulated, it has been argued that the definition of validation is too restrictive, especially for the complex applications associated with realistic geometries. Nevertheless NASA has maintained the definition as a goal of its validation process.

Science discipline	Enabling technologies	Research codes	Pilot codes	Production codes
CFD	• Algorithms • Grid generation • Computer Power	• Technology integration • Limited pioneering applications	• Wide range of applications	• Applied in design environment • Cost effective
		Building block experiments	Benchmark experiments	Design experiments
EFD	• Facilities • Instrumentation • Data acquisition	• Flow physics including numerical simulations • Flow modeling	• Calibration • Validation	• Configuration • Performance • System integration

Fig. 7 Stages of CFD development and corresponding experiments.

CFD Requirements

The accuracy of a computation depends on two principal considerations: 1) the physical realism of the governing equations and boundary conditions and 2) the accuracy of the numerical solution of these equations. Inaccuracies introduced by these must be evaluated through a systematic CFD validation process.[4] Errors associated with the algorithm, including the order of time and space discretizations and numerical dissipation, and the level of grid refinement, including the rate of grid stretching employed in regions of steep gradients, may be evaluated in the absence of experiment. Errors associated with modeling and application to complex configurations require experiments to evaluate them.

EFD Requirements

The stages of code development and the corresponding role of experiments[19] is illustrated in Fig. 7. Experiments are required at each stage of code development, i.e., research, pilot, and production phases. Building block experiments provide the data for flow physics understanding and modeling guidance required in the research phase. Bradley subdivided these into flow physics and flow modeling experiments. Today, direct and large eddy numerical solutions of the time-dependent Navier–Stokes equations and the quantum mechanical equations are also considered as building blocks. Benchmark experiments are utilized during pilot code development, and they define the parameter space over which codes apply. Bradley referred to these as calibration and validation experiments. Design experiments provide the performance data required to validate new designs and substantiate CFD performance estimates. They also are the source of many of the additional experiments used in industry's validation process referred to previously. Our attention herein is focused on the two former categories.

Measurements. Each type of experiment requires specific information that will enable a critical assessment of a code's capabilities. Examples of typical measurements and the test conditions where they are needed are listed in Fig. 8. The measurements listed are germane to the development of Reynolds-averaged Navier–Stokes codes for fully developed turbulent flow.[20]

Building block experiments require measurements of surface and flowfield variables, including turbulence data. Although flow physics experiments and numerical simulations may be carried out for simple flows at incompressible conditions, ultimately, flow modeling validation experiments need to be conducted at representative flight Mach and Reynolds numbers where the codes are actually applied. Benchmark experiments also require measurements of surface and flowfield variables. However, flowfield data at critical locations are needed since the objective is to ensure only that the code represents the correct physics. To clearly identify the applicable operational envelope of the code, parametric testing over a range of flight Mach and Reynolds numbers is necessary.

For each category of experiment careful measurements of boundary conditions are required because they may influence the flowfield around test models or flight vehicles. Moreover, they may be needed to initiate computations. Freestream or initial conditions, wall boundary physical location, and precise model lines are examples of these measurement requirements.

Accuracy. Accuracy assessments for experimental data are essential; otherwise there is no quantitative means for determining the validated range of a code. Error estimates for test geometry dimensions, test operating and freestream conditions, model and flowfield measured variables, and instrumentation should all be specified. Uncertainty analysis is a well established method for determining experimental data accuracy, and it should be a prerequisite for all validation databases. See, e.g., the method described in Ref. 21.

Reliance on single experiments or measurement procedures during the code validation process should be viewed with caution because of facility and instrumentation limitations. Therefore, redundant measurements and similar experiments performed in more than one facility are desirable.

Instrumentation. Advances in instrumentation and data acquisition over the last two decades have made the prospects promising for obtaining the measurements required for the various experiments. Some of the advances showing the trends away from intrusive measurement techniques are illustrated in Fig. 9. Measurements of flowfield velocity, density, temperature, gas concentrations, skin friction, and surface variables such as pressure and heating have been demonstrated.

One example of how these advanced techniques are already being employed in combustor flow studies[22] performed to investigate flow physics and modeling is given in Fig. 10. Mole fraction contours obtained with planar-laser-induced-iodine-fluorescence for staged injection of a seeded gas from two normal jets into a supersonic flow downstream of a step are compared with computations from a solution to the Reynolds-averaged Navier–Stokes equations. Measurements in various cross planes provide a three-dimensional map of the mole fractions. The computations compare well with experiment and validation of the code is still being carried out.

Experiment	Measurements (Representative for turbulence modeling)	Test conditions
Building block (Phenomenological)	Surface quantities include transition pts. Flow field quantities Turbulence Individual stresses Correlation lengths Structure Boundary conditions Free stream Tunnel walls Model shape	Representative flight M_∞, Re_∞
Benchmark (Parametrical)	Surface quantities include transition pts. Flow field quantities (Selected locations) Boundary conditions (See above)	Vary M_∞, Re, α over flight ranges
Design (Confirgurational)	Drag, lift, moments, heat loads, shear loads Boundary conditions (See above)	As close to flight M_∞, Re, α as practical

Fig. 8 Experimental requirements germane to code development for turbulent flows.

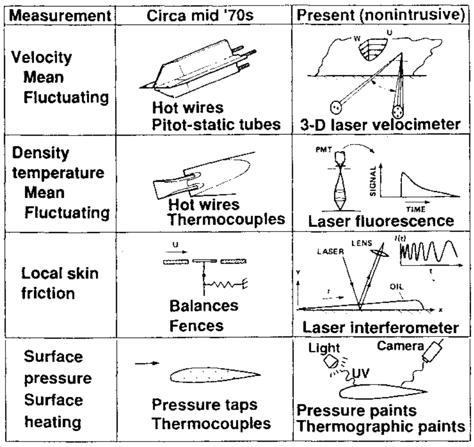


Fig. 9 Advances in measurement capabilities.

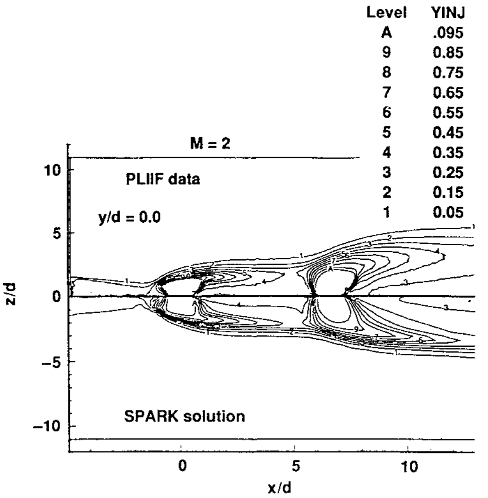


Fig. 10 Plane of symmetry mole fraction contours for staged injection measured with PLIIF and computed with the SPARK code; T = 167 K, M = 2, and M_j = 1.35.

Progress

Building block and benchmark experiments initiated by NASA have already been accomplished, and others are under way. Some examples provided by the author's colleagues at the Ames Research Center that highlight contributions toward the development and validation of CFD for turbulent and chemically reacting aerodynamic flows follow.

Flow Physics

Direct and large eddy numerical simulations of the time-dependent Navier–Stokes equations have the potential for providing new understanding of the physics of turbulent flows. Simulations have been carried out for simple, low-Reynolds-number flows using periodic boundary conditions. The next step is to provide simulations for more complex flows, but some questions have been raised regarding the assumptions employed, the procedures for treating spatially developing flows, and the extrapolation of the results to realistic Reynolds numbers. Experiments were initiated to provide answers to these questions.

An underlying assumption of direct numerical simulation of turbulent flows is that small-scale turbulence is isotropic. Measurements that substantiate this assumption have recently been made in the very high Reynolds number, thick boundary layer developing on the wall of the 80 × 120 ft wind tunnel at Ames Research Center.[23] The maximum Reynolds number based on the Taylor microscale was 1.5×10^3, one of the highest values achieved in a laboratory flow, allowing turbulence spectra to be measured over five decades of wave number. The Kolmorogorov-scaled spectra are compared in Fig. 11a with an updated experimental compilation first presented by Chapman.[24] The ratios of the measured transverse spectra are shown Fig. 11b. In the inertial and dissipation wave number ranges the ratio is 1.0. Furthermore, the calculated values of these cross spectra using the streamwise spectra agree with measured values.

Direct numerical simulations of a more complex nature such as adverse pressure gradient flows and separated flows are now being performed. In these simulations, boundary conditions for the spatially developing inflow represent formidable challenges. Further, low-Reynolds-number effects may limit their use for flow physics modeling at high Reynolds numbers. Experiments have been initiated to clarify these issues.[25,26] Figures 12a and 12b show some of the results from a pressure gradient flow study that were used to substantiate the "fringe method"[25] for specifying inflow and outflow boundary conditions. The wall pressure distribution and the turbulent stresses from the numerical simulation compare very well with the measurements for the same Reynolds number. These and other comparisons of statistical turbulence quantities confirmed the appropriateness of the "fringe method."

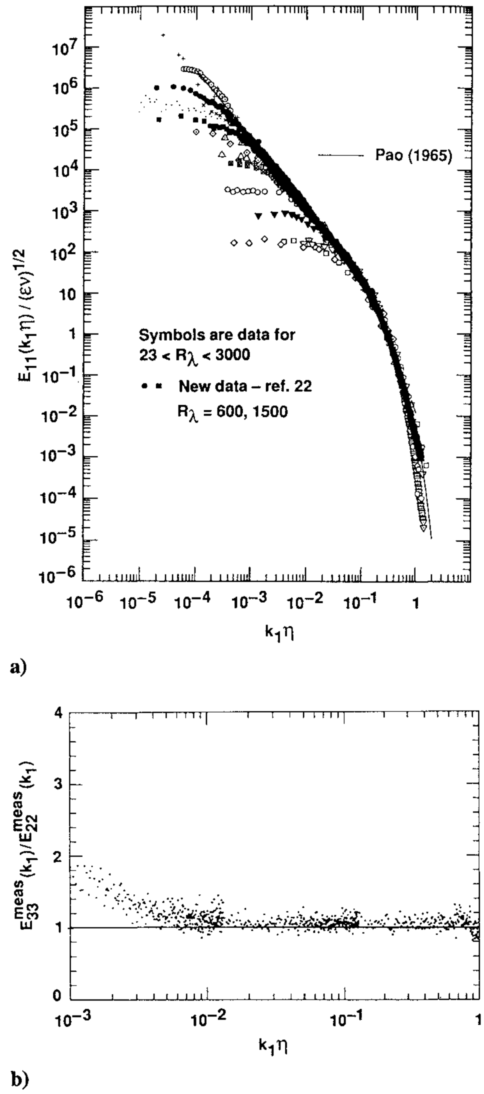


Fig. 11 Turbulence spectra measured to evaluate small-scale isotropy assumption in DNS: a) streamwise component and b) cross spectra ratios.

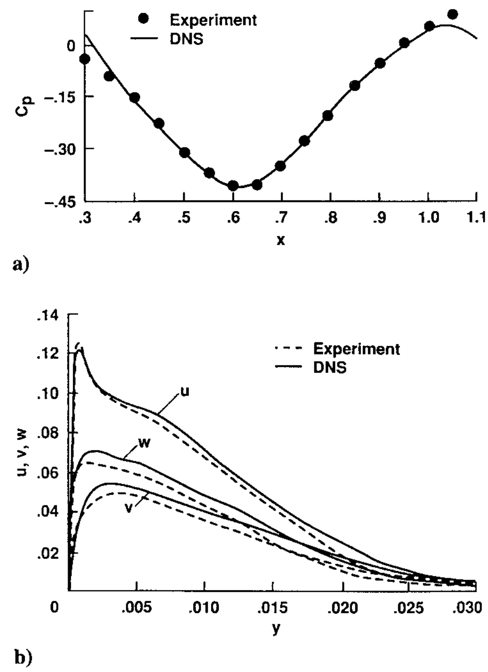


Fig. 12 Flow physics data compared with DNS for adverse pressure gradient flow: a) pressure coefficient and b) normal stresses.

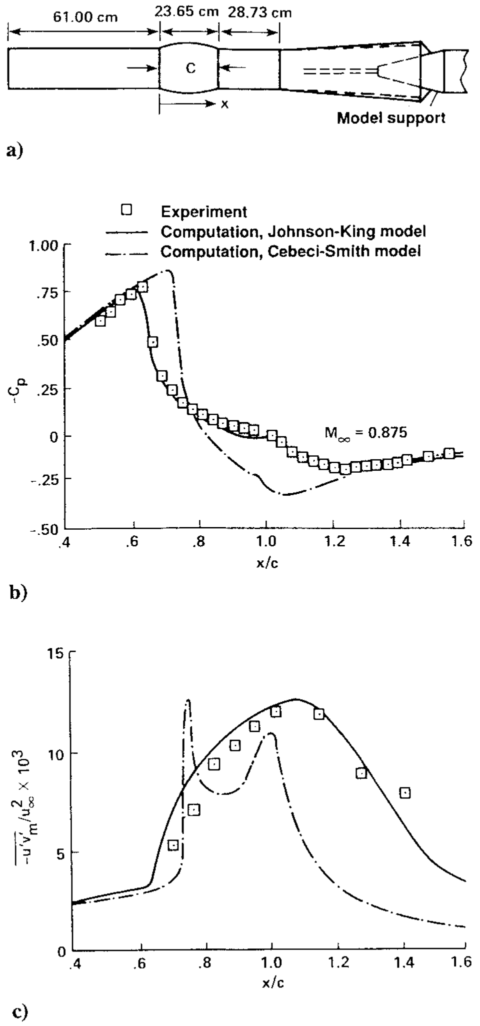


Fig. 13 Flow modeling experiment that led to the development of the Johnson–King turbulence model: a) test model, b) pressure coefficient, and c) maximum turbulent shear stress.

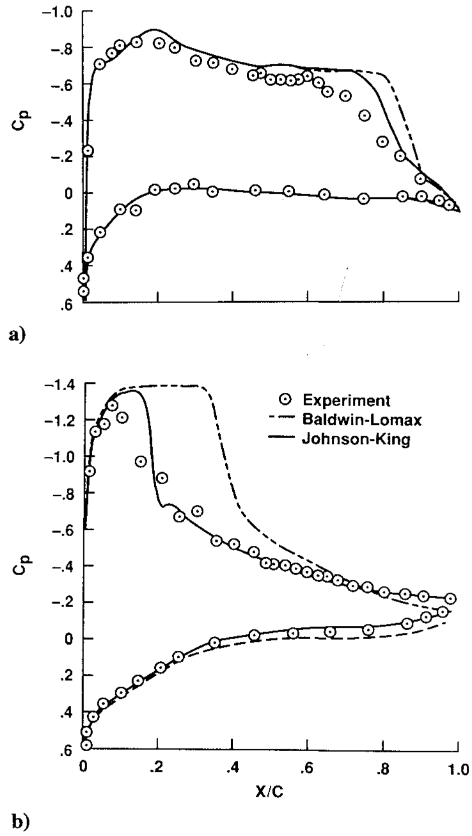


Fig. 14 Validation of the Johnson–King model for a transport wing application: a) inboard station pressures and b) outboard station pressures; $M = 0.855$ and $\alpha = 5.7$ deg.

Flow Modeling

A status of turbulence modeling and related experimental validation experiments for aerodynamic flows was presented in Ref. 27. Improvements in modeling have taken place since then due in large measure through the addition of turbulence quantities to the experimental database.

The Johnson–King turbulence model development[28] typifies the impact experiment has had on turbulence model improvements. Extensive experimental data and concurrent computations were used in its development. The experiment[29] shown in Fig. 13 provided much of the data for guidance and validation of their model for transonic airfoil and wing applications. The test configuration, designed to avoid undesirable three-dimensional flow, was an axisymmetric cylinder aligned with the freestream and fitted with a circular arc section. Surface pressures and flowfield mean and turbulence quantities obtained with a laser anemometer system were measured for various Mach numbers. The turbulence model derived made use of experimental evidence on the development of the turbulent shear stresses through shock waves of varying strength that developed on the circular arc section as the freestream Mach number was varied. An ordinary differential equation that accounts for the development of the maximum shear stress was derived and used to modify the eddy viscosity predicted with algebraic turbulence models. The modification results in better predictions of pressure recovery, separation location and its extent, and velocity profile development.

Improvements to the model[30] have been made since its original development. The model has been validated with a number of experiments, successfully implemented in various CFD codes, and is being used in advanced wing designs for commercial aircraft.[31] Pressure coefficients on two sections of the 747-200 wing are compared with computations using the model in Fig. 14 to illustrate its application in computing high-aspect-ratio wing flows.

More general, higher order turbulence modeling formulations and improvements are needed for more complex CFD applications. Experiments for a number of different building block flows wherein measurements of turbulence quantities were obtained are being used to guide and validate improvements. Menter[32] describes an improvement to a two-equation model formulation that was based on such experiments, particularly those of Driver.[33] The model that improves predictions of adverse pressure gradient and separating flows does not require near wall low-Reynolds-number terms, and it is only weakly dependent on freestream turbulence boundary conditions. Some results demonstrating the models ability to predict a separated flow are given in Fig. 15. The computations with the shear stress transport (SST) model agree with the measurements of the mean velocity profiles and the maximum eddy viscosity development, a sensitive discriminator of turbulence model improvement.

Numerical simulation databases are also being used to improve higher order turbulence models. Most of the original model formulations (an exception was mentioned earlier) are unable to describe the near wall behavior of turbulence accurately, and the simulations are providing information that cannot be obtained by experiment. Figure 16 shows an example of how the data from the simulations have been used to evaluate near wall model formulations.[34]

Calibration

Code parameters such as turbulence models may need to be adjusted to accommodate applications for geometries and conditions outside the envelope of their original validation. Experiments intended to support this activity can be referred to as calibration experiments.

An example of how experiment[35] was used to calibrate turbulence models for compressible flow conditions not accounted for in their original formulations is shown in Fig. 17. The test model

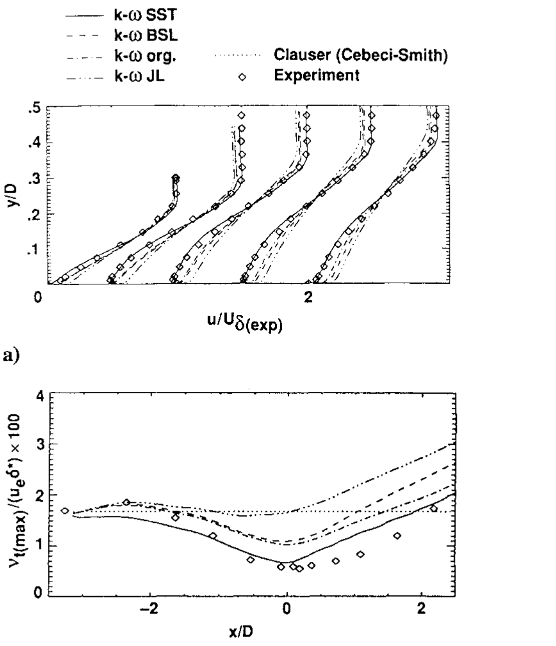


Fig. 15 Flow modeling experimental data used to guide and develop a shear stress transport turbulence model: a) velocity profiles and b) maximum eddy viscosity development.

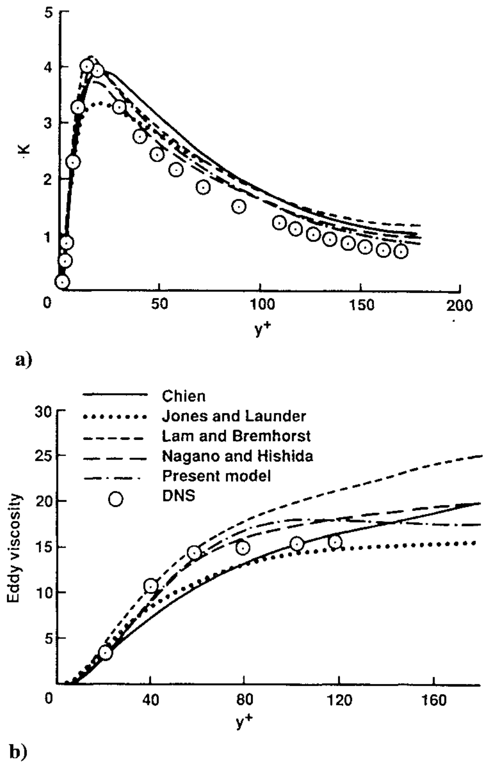


Fig. 16 Near wall turbulence model derived and validated from channel flow DNS: a) kinetic energy and b) eddy viscosity.

consisted of an ogive-cylinder-flare aligned with the freestream. Mean surface and flowfield data were obtained at $M = 7$. Pressures and heating rates are compared with computations[36] using uncorrected models and the same models corrected for compressibility and length scale. Good agreement with the data was obtained with the corrected models. Subsequently, comparisons were made for other hypersonic experimental shock interaction data specifically documented for turbulence model calibration,[37] and similar good agreement was achieved. It was concluded that these corrections, which can be used with any two-equation model formulation,

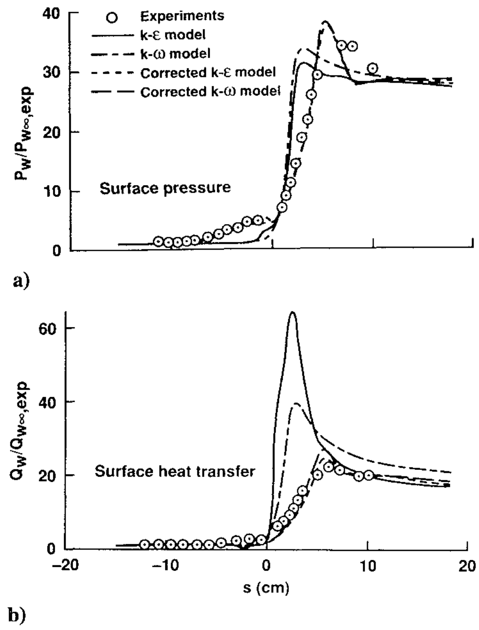


Fig. 17 Experimental calibration of turbulence model corrections for hypersonic shock-wave boundary-layer interactions: a) surface pressure and b) surface heat transfer.

extend their useful range of application to the hypersonic regime. Recently these compressibility model corrections have been substantiated for three-dimensional interactions[38] with calibration experiment data.[39]

Another experiment, used to assess and calibrate an air chemistry model, is shown in Fig. 18. Sharp and blunt cone models were fired down the Ames hypervelocity gun range at speeds in excess of 5 km/s and Reynolds numbers between 10^5 and 10^6. High enthalpy reacting laminar flow was established over the models. Aerodynamic coefficient and flow visualization data were measured, and those for a 5-deg blunt cone fitted with ring shock generators are presented. The data for drag are insensitive to chemistry modeling since it is composed mainly of forebody pressure drag. However, the pitching moment and conical flowfield are sensitive to the air chemistry, and the data were used to calibrate an air chemistry model in a code originally validated with data from high-Mach-number, low-enthalpy experiments. Results[40] from a Navier–Stokes code using a seven species air chemistry model, strongly coupled to the fluid dynamics equations, are compared with the data in Fig. 18 and good agreement achieved.

Validation

Subsequent to the NASA study,[4] experiments were initiated toward achieving the goal of validation discussed previously. Some of these experiments have recently been reported.[41–44]

The transonic low-aspect-ratio wing experiment[42] illustrates some of the features of a well-posed validation experiment when transonic conditions can lead to comparative uncertainties between experiment and computation. The experimental layout is shown in Fig. 19a. The approach is unique in that the model was tested in a solid wall wind tunnel so that the issues of corrections for wall ventilation and interference usually associated with transonic testing are eliminated. The upper and lower tunnel walls were diverged slightly to accommodate boundary-layer growth on all four walls, and CFD solutions can be accomplished by specifying straight solid wall boundary conditions with slip. Wing surface pressures, tunnel wall pressures on all four walls, side wall boundary-layer velocity profiles, and flowfield velocity profile data, including the freestream inflow values, were measured for Reynolds numbers of 8 and 14×10^6 million and Mach numbers from 0.6 to 0.8. Turbulence and velocity profiles in the viscous layer over the model surface were not measured, however. Strict attention was paid to identifying and quantifying experimental uncertainty. Pressure coefficient data at various span locations for $M = 0.8$ and the two Reynolds numbers

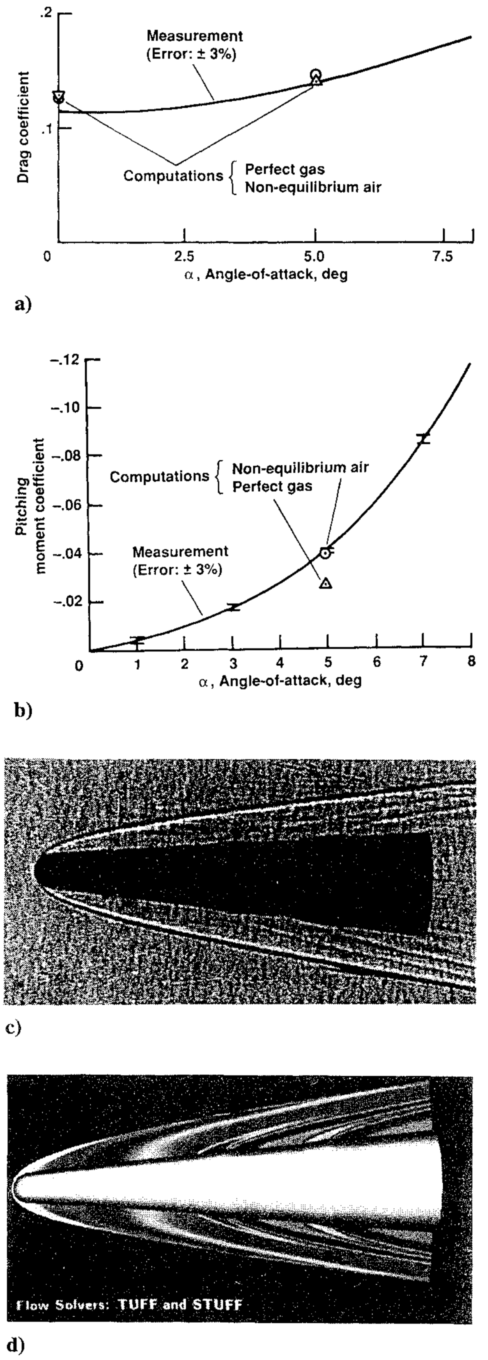


Fig. 18 Experiment performed to calibrate an air chemistry model: a) drag coefficient, b) pitching moment, c) shadowgraph, and d) computed pressure contours.

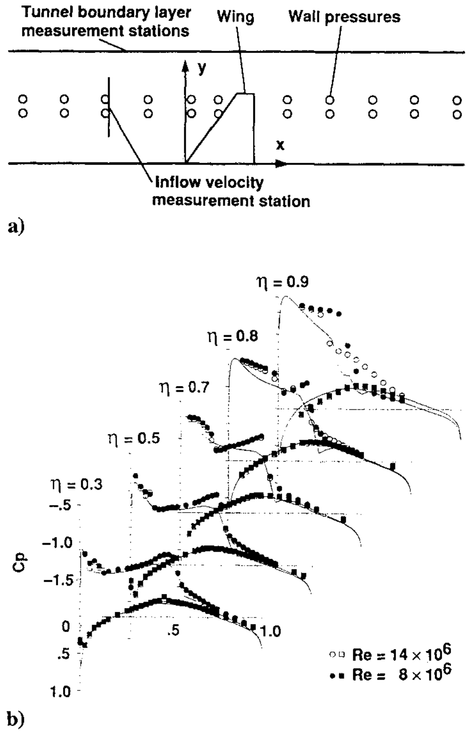


Fig. 19 Transonic wing validation experiment: a) plan view of model installation and b) pressure coefficients; $M = 0.8$, $\alpha = 5$ deg, and $Re = 8$ and 14×10^6.

are shown in Fig. 19b. The flow on the upper surface at these conditions shows complex shock structures and Reynolds number effects that will challenge the capabilities of newly emerging Navier–Stokes codes and turbulence models.

Hypersonic flow conditions also present formidable challenges to validation, but progress is being made in this regard. For example, a generic hypersonic vehicle model was recently tested to provide benchmark data for verifying forebody codes developed for use in integrating air-breathing propulsion systems with vehicle airframes.[41] Experimental data on flow visualization, surface pressures, surface convective heat transfer, and pitot pressure flowfield surveys were obtained for various Reynolds and Mach numbers in a low-enthalpy facility. A photo of the model and some typical pressure, heating rate, and pitot pressure profile data are shown in Fig. 20. Concurrent with the experiment, computations using the Ames UPS code (an upwind Parabolized Navier–Stokes solver) were initiated and are shown compared with the data. There is generally good agreement. These comparisons and others for a variety of different hypersonic experiments have been used to establish validation of the Ames UPS code for wind tunnel conditions.

Future Challenges and Strategies

Thus far, CFD code validation has been discussed primarily using experiments with generic geometries and performed mostly in small-scale, ground-based facilities. CFD provides a far wider range of possibilities for simulating flows over more complete geometries, including complete aircraft at flight conditions, and multidisciplinary applications. Validation of such capabilities is significantly less well developed and presents a formidable challenge. An approach to performing any such computations for engineering purposes would be to incorporate validated flow physics modeling and perform adequate grid resolution studies, but even this approach does not guarantee accuracy because multiple and interacting physical phenomena may be present. Therefore experiments will ultimately be required.

Obtaining validation measurements such as surface and flowfield data on aircraft shapes in large wind tunnels is extremely difficult and costly due to, for example, the lack of dedicated nonintrusive instrumentation, limited tunnel viewing access in large-scale facilities, the general lack of flight scale and enthalpy simulation, and wind tunnel wall interference effects at transonic conditions. Alternatively, flight experiments designed with validation as an integral part of their motivation are attractive. Of course, these also are difficult and costly due to, for example, geometric shape changes caused by flight loads, uncertainties in freestream conditions and aircraft attitude, and the inability to obtain the necessary flowfield and physics validation measurements.

A plausible strategy would be to perform a few well-conceived ground-based/flight experiments designed with validation as part of their purpose. More quantitative results on somewhat simpler geometries obtained in ground-based facilities could be used to perform the bulk of application code validation followed by the ground-based/flight validation on realistic vehicle configurations. A few such studies have already been initiated.[45–48] An example of computations and flight data for an aircraft at high angle of attack[45] are shown in Fig. 21. The agreement of the computations with the

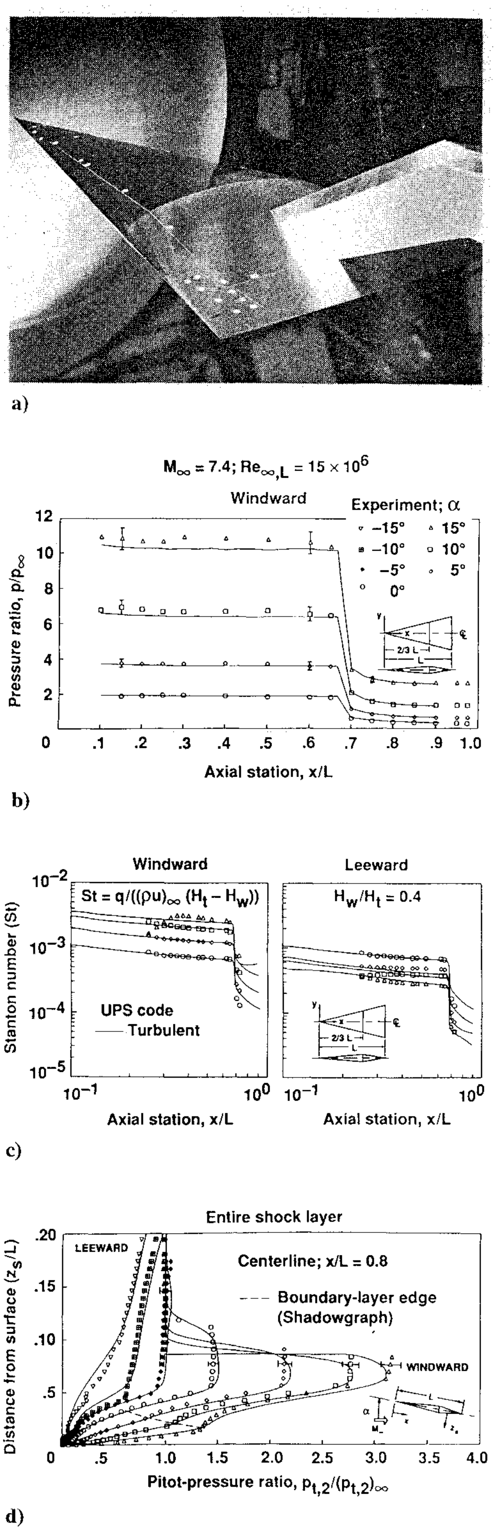


Fig. 20 Hypersonic "all body" validation experiment: a) model photograph, b) pressures, c) heating rates, and d) pitot profiles.

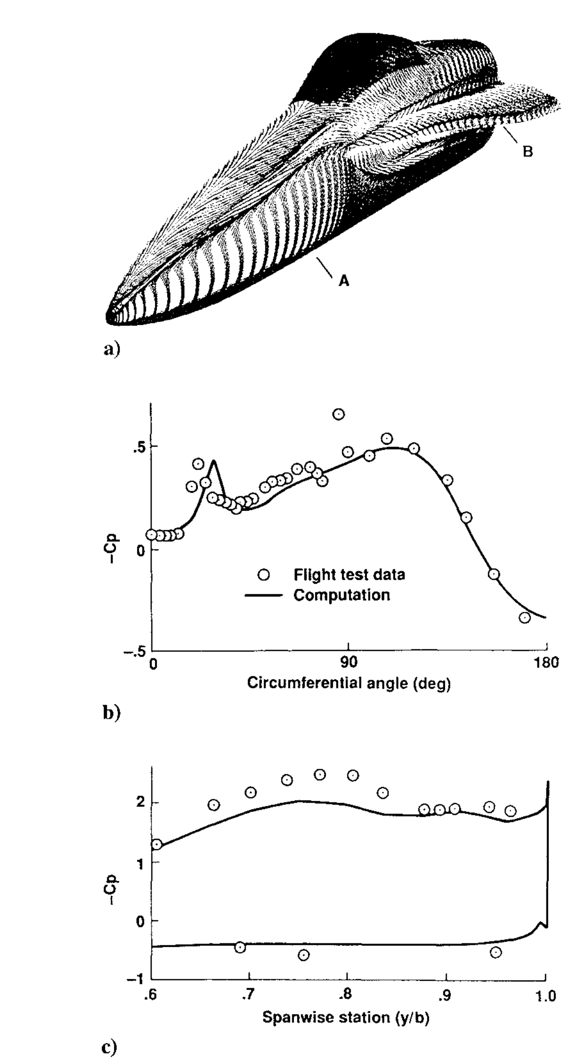


Fig. 21 Comparisons of "oil flow" and surface pressures computed for an F-18 forebody with flight data, $M = 0.2$, $\alpha = 30$ deg, and $Re_D = 11.5 \times 10^6$: a) oil flow, b) pressures at A, and c) pressures at B.

data is quite remarkable. As the level and quality of data in both ground-based and flight test experiments improve in the future, the prospects for developing validated codes across a broad spectrum seem to be bright.

Another important aspect of validation strategy is the cataloging, documentation, and assessment of experimental databases. Such information can be used in workshops to assess code development and validation, by CFD design groups, who must validate codes used in the design process, and by planners of future experimental programs, who should be aware of existing data. Precedents for such undertakings have already been established.[37,49–51]

Concluding Remarks

The process of CFD validation was discussed. Requirements from both the computational and experimental points of view were presented. The determination of a code's accuracy related to its algorithm and grid refinement can proceed in most cases in the absence of experimental data because the sources of error are numerical in nature. Ultimately, however, the accuracy related to a code's chemistry and physical modeling and its application to complex aerodynamic flows requires comparison with experiment to support development and validation.

Building block and benchmark experiments were described that can provide the necessary measurements needed for validation during research and pilot code development stages. Examples were given to show how these experiments are contributing toward the development of CFD in regards to flow physics understanding, flow modeling, calibration, and validation. The requirement for surface and flowfield data was stressed along with the need for measurement accuracy determination. Progress in the overall validation of CFD codes was shown to be improving because more well-posed experiments are becoming available, some improvements in turbulence modeling have been developed, and algorithm efficiency and computer speed have enabled more appropriate spatial resolution.

Future challenges and strategies were discussed. Full and large eddy simulations used to guide model development should be performed for more complex aerodynamic flows. Experiments for unsteady and multidisciplinary CFD applications should proceed now to provide timely validation data. And the importance and need for a few well-designed, ground-based and flight validation experiments on realistic aircraft geometries were discussed and proposed as the ultimate test of a code's ability to simulate real aerodynamic flows at flight conditions. Implementing instrumentation advances into validation experiments performed in both large-scale wind tunnel facilities and flight is essential to the success of this strategy.

References

[1]Anon., "Computational Fluid Dynamics," *Aerospace America*, Pt. 1, Jan. 1992, pp. 16–48; Pt. 2, Feb. 1992, pp. 18–48.

[2]Marvin, J. G., "Advancing Computational Aerodynamics Through Wind Tunnel Experimentation," AGARD Fluid Dynamics Panel Meeting, Chattanooga, TN, Sverdrup/ARO, Inc., Arnold AFB, TN, Sept. 1980.

[3]Marvin, J. G., "Accuracy Requirements and Benchmark Experiments for CFD Validation," *Validation of Computational Fluid Dynamics* (Lisbon, Portugal), AGARD-CP 437, Vol. 1, Paper No. 2, Dec. 1988.

[4]Bradley, R. G., "CFD Validation Philosophy," *Validation of Computational Fluid Dynamics* (Lisbon, Portugal), AGARD-CP, 437, Vol. 1, Paper No. 1, Dec. 1988.

[5]Marvin, J. G., and Deiwert, G. S., "Convective Heat Transfer in Planetary Gases," NASA TR 224, July 1965.

[6]Hoshizaki, H., "Heat Transfer in Planetary Atmospheres at Super-Satellite Speeds," *ARS Journal*, Vol. 32, Oct. 1962, pp. 1544–1551.

[7]Marvin, J. G., Seegmiller, H. L., Lockman, W. K., Mateer, G. G., Pappas, C. C., and DeRose, C. E., "Surface Flow Patterns and Aerodynamic Heating on Space Shuttle Vehicles," AIAA Paper 71-594, June 1971.

[8]Rakich, J. V., Stewart, D. A., and Lanfranco, M. J., "Catalytic Efficiency of the Space Shuttle Heat Shield," *Entry Vehicle Heating and Thermal Protection Systems: Space Shuttle, Solar Starprobe, Jupiter Galileo Probe*, Vol. 85, Progress in Astronautics and Aeronautics, AIAA, New York, 1983, pp. 249–272.

[9]Murman, E. M., and Cole, J. G., "Calculation of Plane Steady Transonic Flows," *AIAA Journal*, Vol. 9, No. 1, 1971, pp. 114–121.

[10]MacCormack, R. W., "Numerical Solution of the Interaction of a Shock Wave with a Laminar Boundary Layer," *Lecture Notes in Physics*, Vol. 8, Springer–Verlag, New York, 1971, pp. 151–163.

[11]Baldwin, B. S., MacCormack, R. W., and Deiwert, G. S., "Numerical Techniques of the Compressible Navier–Stokes Equations and Implementation of Turbulence Models, AGARD-LSP-73, March 1975.

[12]Deiwert, G. S., "Numerical Simulation of High Reynolds Number Transonic Flows," *AIAA Journal*, Vol. 13, No. 10, 1975, pp. 1354–1359.

[13]McDevitt, J. B., Levy, L. L., and Deiwert, G. S., "Transonic Flow About a Thick Circular-Arc Airfoil," *AIAA Journal*, Vol. 14, No. 5, 1976, pp. 606–613.

[14]Levy, L. L., Jr., "Experimental and Computational Steady and Unsteady Transonic Flows About a Thick Airfoil," *AIAA Journal*, Vol. 16, No. 6, 1978, pp. 564–572.

[15]Marvin, J. G., Levy, L. L., and Seegmiller, H. L., "On Turbulence Modeling for Unsteady Transonic Flows," *AIAA Journal*, Vol. 18, No. 5, 1980, pp. 489–496.

[16]Moin, P., and Kim, J., "The Structure of the Vorticity Field in Turbulent Channel Flow, Part I—Analysis of Instantaneous Fields and Statistical Correlations," *Journal of Fluid Mechanics*, Vol. 155, June 1985, pp. 441–464.

[17]Cooper, D. M., Jaffe, R. L., and Arnold, J. O., "Computational Chemistry and Aeroassisted Orbital Transfer Vehicles," *Journal of Spacecraft and Rockets*, Vol. 22, No. 1, 1985, p. 60.

[18]Lawrence, S. L., Chaussee, D. S., and Tannehill, J. C., "Application of an Upwind Algorithm to the Three-Dimensional Parabolized Navier–Stokes Equations," AIAA Paper 87-1112, June 1987.

[19]Marvin, J. G., and Holst, T. L., "CFD Validation for Aerodynamic Flows—Challenge for the '90's," AIAA Paper 90-2995, Aug. 1990.

[20]Marvin, J. G., "Wind Tunnel Requirements for Computational Fluid Dynamics Code Verification," *Aerodynamic Data Accuracy and Quality: Requirements and Capabilities in Wind Tunnel Testing*, AGARD CP-429, Paper No. 34, July 1988.

[21]Moffatt, R. J., "Contributions to the Theory of Uncertainty Analysis for Single-Sample Experiments," *Objectives, Evaluation of Data, Specification of Tests Cases, Discussion, and Position Papers*, edited by S. J. Kline, B. J. Cantwell, and G. M. Lilley, Vol. 1, *The 1980-81 AFOSR/HTTM-Stanford Conference on Complex Turbulent Flows*, Thermo Sciences Div., ME Dept., Stanford, CA.

[22]Eklund, D. R., Northern, B. G., and Fletcher, D. G., "A Validation Study of the SPARK Navier–Stokes Code for Nonreacting Scramjet Combustor Flow Fields," AIAA Paper 90-2360, July 1990.

[23]Saddoughi, S. J., "Local Isotropy in High-Reynolds Number Turbulent Shear Flows," Annual Research Briefs—1992, Center for Turbulence Research, NASA Ames Research Center and Stanford Univ., Stanford, CA, Dec. 1992.

[24]Chapman, D. R., "Dryden Lectureship in Research, Computational Aerodynamics Development and Outlook," *AIAA Journal*, Vol. 17, No. 12, 1979, pp. 1293–1313.

[25]Spalart, P. R., and Watmuff, J. H., "Experimental and Numerical Study of a Turbulent Boundary Layer with Pressure Gradients," *Journal of Fluid Mechanics*, Vol. 249, April 1993, pp. 337–371.

[26]Jovic, S., and Browne, W. B., "Coherent Structures in a Boundary Layer and Shear Layer of a Turbulent Backward-Facing Step Flow," Seventh Symposium on Turbulent Shear Flows, Stanford Univ., Paper 24-3, Aug. 1989.

[27]Marvin, J. G., "Turbulence Modeling for Computational Aerodynamics," *AIAA Journal*, Vol. 212, No. 7, 1983, pp. 941–955.

[28]Johnson, D. A., and King, L. S., "A Mathematically Simple Turbulence Closure Model for Attached and Separated Turbulent Boundary Layers," *AIAA Journal*, Vol. 23, No. 11, 1985, pp. 1684–1692.

[29]Bachalo, W. D., and Johnson, D. A., "An Investigation of Transonic Turbulent Boundary Layer Separation Generated on an Axisymmetric Flow Model," AIAA Paper 79-1479, July 1979.

[30]Johnson, D. A., and Coakley, T. J., "Improvements to a Nonequilibrium Algebraic Turbulence Model," *AIAA Journal*, Vol. 28, No. 11, 1990, pp. 2000–2003.

[31]Yu, N., "Transonic Airfoil and Wing Design Using Navier–Stokes Codes," AIAA Paper 92-2651, June 1992.

[32]Menter, F. R., "Improved Two-Equation k-w Turbulence Models for Aerodynamic Flows," NASA TM 103975, Oct. 1992.

[33]Driver, D. M., "Reynolds Shear Stress Measurements in a Separated Boundary Layer," AIAA Paper 91-1787, June 1991.

[34]Shih, T. H., "Turbulence Modeling: Near-Wall Turbulence and Effects of Rotation on Turbulence," Annual Research Briefs, Center for Turbulence Research, NASA Ames Research Center and Stanford Univ., Stanford, CA, 1989, pp. 12–26.

[35]Kussoy, M. I., and Horstman, C. C., "Documentation of Two- and Three-Dimensional Hypersonic Shock Wave/Turbulent Boundary Layer Interaction Flows," NASA TM-101075, Jan. 1989.

[36]Coakley, T. J., and Huang, P. G., "Turbulence Modeling for High Speed Flows," AIAA Paper 92-0436, Jan. 1992.

[37]Settles, G. S., and Dodson, L. J., "Hypersonic Shock/Boundary-Layer Interaction Database," NASA CR-177577, April 1991.

[38]Bardina, J., Coakley, T. J., and Marvin, J. G., "Two-Equation Turbulence Modeling for 3-D Hypersonic Flows," AIAA Paper 92-5064, Dec. 1992.

[39]Kussoy, M. I., and Horstman, K. C., "Documentation of Two- and Three-Dimensional Shock-Wave/Turbulent-Boundary Layer Interaction Flows at Mach 8.2," NASA TM-1038938, May 1991.

[40]Strawa, A. W., Molvik, G., Yates, L., and Cornelison, C., "Experimental and Computational Results for 5 Degree Blunt Cones with Shock Generators at High Velocity," AIAA Paper 89-3377, Aug. 1989.

[41]Lockman, W. K., Lawrence, S. L., and Cleary, J. W., "Flow Over an All-Body Hypersonic Aircraft: Experiment and Computation," *Journal of Spacecraft and Rockets*, Vol. 29, No. 1, 1992, pp. 7–15.

[42]Olsen, M. E., and Seegmiller, H. L., "Low Aspect Ratio Wing Code Validation Experiment," *AIAA Journal*, Vol. 31, No. 10, 1993, pp. 1744–1752.

[43]Monson, D. J., and Seegmiller, H. L., "An Experimental Investigation of Subsonic Flow in a Two-Dimensional U-Duct," NASA TM 103931, May 1992.

[44]Mateer, G., Seegmiller, H. L., Hand, L. A., and Szodruch, J., "An Experimental Investigation of a Supercritical Airfoil at Transonic Speeds," NASA TM 103933, July 1992.

[45]Cummings, R. M., Rizk, Y. M., Schiff, L. B., and Chaderjian, N. M., "Navier–Stokes Prediction of the Flow Field Around the F-18 (HARV) Wing and Fuselage at Large Incidence," AIAA Paper 90-0099, Jan. 1990.

[46]Fisher, D. F., Banks, D. W., and Richwine, D. M., "F-18 High Alpha Research Vehicle Surface Pressures: Correlation with Flow Visualization and Wind-Tunnel Data," NASA TM 101724, Aug. 1990.

[47]Rizk, Y. M., and Gee, K., "Numerical Prediction of the Unsteady Flowfield Around the F-18 Aircraft at Large Incidence," AIAA Paper 91-0020, Jan. 1991.

[48]Meyn, L., and Lanser, W., "Full Scale High-Angle-of-Attack Tests of an F/A-18," AIAA Paper 92-2676, June 1992.

[49]Anon., "Experimental Data Base for Computer Program Assessment," Fluid Dynamics Panel Working Group 04, AGARD-AR-138, May 1979.

[50]Kline, S. J., Cantwell, B. J., and Lilley, G. M. (eds.), *Objectives, Evaluation of Data, Specifications of Test Cases, Discussion, and Position Papers, The 1980-81 AFOSR/HTTM Stanford Conference on Complex Turbulent Flows*, Vol. 1, Thermo Sciences Div., ME Dept., Stanford, CA.

[51]Anon., "Test Cases for Inviscid Flow Field Methods," Fluid Dynamics Panel Working Group 07, AGARD AR-211, May 1985.