NASA Technical Memorandum 100087

Accuracy Requirements and Benchmark Experiments for CFD Validation

Joseph G. Marvin

(NASA-TM-100087) ACCURACY REQUIREMENTS AND BENCHMARK EXPERIMENTS FOR CFD VALIDATION (NASA) 18 p CSCL 20D
N88-21423
Unclas
G3/34 0140259

May 1988

NASA
National Aeronautics and
Space Administration

NASA Technical Memorandum 100087

Accuracy Requirements and Benchmark Experiments for CFD Validation

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

May 1988

National Aeronautics and
Space Administration

Ames Research Center
Moffett Field, California 94035

ACCURACY REQUIREMENTS AND BENCHMARK EXPERIMENTS FOR CFD VALIDATION

Joseph G. Marvin
Chief, Experimental Fluid Dynamics Branch
NASA Ames Research Center
Moffett Field, CA 94035

SUMMARY

The role of experiment in the development of Computational Fluid Dynamics (CFD) for aerodynamic flow prediction is discussed. CFD verification is a concept that depends on closely coordinated planning between computational and experimental disciplines. Because code applications are becoming more complex and their potential for design more feasible, it no longer suffices to use experimental data from surface or integral measurements alone to provide the required verification. Flow physics and modeling, flow field, and boundary condition measurements are emerging as critical data. Four types of experiments are introduced and examples given that meet the challenge of validation: (1) flow physics experiments; (2) flow modeling experiments; (3) calibration experiments; and (4) verification experiments. Measurement and accuracy requirements for each of these differ and are discussed. A comprehensive program of validation is described, some examples given, and it is concluded that the future prospects are encouraging.

1. INTRODUCTION

Mathematical approximations, limited computer capacity, and lack of understanding of physical modeling lead to uncertainties in the application of Computational Fluid Dynamics (CFD). Consequently, the pace of introduction and the extent of reliance on CFD in the design process depends on validation[1]; and experiments that verify CFD have become an essential element of its evolutionary development.[2]

Experimental validation is required for a number of different aerodynamic flows that occur over the full range of flight speeds. Any effective, timely program to provide the necessary data will require good planning and cooperation between various aerospace disciplines. Because of this situation the topic of validation has been intensely debated within NASA during the past year. An outgrowth of that debate resulted in the concepts of CFD validation and calibration and categories of experiments recommended by a NASA ad hoc Committee on Validation introduced by Bradley.[3] And the first NASA CFD Validation Workshop made further recommendations: (1) provide closer cooperation between CFD developers and experimentalists; (2) provide detailed measurements of the flow field and boundary conditions in addition to model surface and integral quantities; (3) provide new or improved nonintrusive measurement capabilities, especially for hypersonic or reacting flow conditions; (4) provide redundancy in both measurements and experiments whenever practical so as to clarify accuracy and credibility; (5) provide dedicated large facilities for validation research activities; and (6) provide standardized test cases with accessible data bases.

The intent of the present paper is to provide a perspective on validation using these ideas and to introduce a synergistic approach for timely accomplishment of validation. Details on experimental and accuracy requirements will be discussed using the concepts of validation, calibration, and categories of experimentation as defined in Ref. 3.

CFD code validation: Detailed surface- and flow-field comparisons with experimental data to verify the code's 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 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 design interest, made in order 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 the features of the flow are correctly modeled.

Categories of experimentation: (1) Experiments designed to understand flow physics; (2) experiments designed to develop physical models; (3) experiments designed to calibrate CFD; (4) experiments designed to validate CFD.

The categories of experiments will be explained first with the aid of some examples that represent the work of the author and his colleagues at the Ames Research Center. Their cooperation in the use and preparation of this material is greatly appreciated. Following that, accuracy, instrumentation and facility requirements, and future prospects for validation experiments will be discussed.

2. EXPERIMENTAL REQUIREMENTS

2.1 Role of Experiments

A framework for describing the connection between experiment and computation was presented in Ref. 4. That framework can be depicted with the aid of Fig. 1, taken from Ref. 4, and extended to reflect new developments and the various categories of experiments defined in Ref. 3. The stages of code development are shown in ascending order of maturity and each is linked to a type(s) of experiment.

Research codes refer to those developed by integrating new enabling technology such as supercomputers, algorithms, grid methodology, and new understanding of physical modeling to solve specific problems. One or two researchers are involved in developing the code, and limited documentation is available. Experiments utilized at this stage are referred to as building blocks. These provide the data required to understand flow physics, to guide flow modeling processes, and to validate the computations for a particular problem. Two types of experiments make up the building blocks leading to the development of the research code. They are flow physics and flow modeling experiments. An additional new development at this level is the use of full and large-eddy numerical simulations[5] (FS and LES) and computational chemistry[6] to develop data bases for understanding phenomena such as a transition, turbulence, and reaction rate chemistry.

Pilot codes refer to a more mature stage of development. Documentation is more complete, the code is operated by others besides those involved in the research code development, and the envelope of application is expanded in recognition of the potential advances afforded by the research code. Benchmark experiments are the key to this stage of development. They provide the parametric information leading to the identification of the range of applicability of the code. Calibration and verification are the objectives of these experiments.

Subsequently the code would advance to its ultimate development stage when it could be used alone or in combination with codes from other disciplines such as structures or propulsion and applied confidently in the design process. Configurational, performance and system integration experimental data would be needed for verification at this stage.

The delineation of the various stages of development outlined above is idealized, and not always evident in practice, because of the dynamic nature of CFD and its wide-ranging possibilities for solving such a variety and complexity of problems; but the framework depicts how experiment and computation, working together, could accelerate the pace of development. Without question the success of such a framework depends on close coordination between experimental and computational fluid dynamicists and instrumentation developers. For this paper, the emphasis will be on the requirements for the first two stages of development.

2.2 Flow Physics Experiments

The lack of understanding of fundamental physical phenomena is limiting the pace of CFD development. Important expamples are transition from laminar to turbulent flow, turbulence, and high temperature gas physics related to hypersonic flows. Flow physics experiments are defined as those experiments that provide fundamental understanding of such phenomena so that they can be accurately modeled in the codes. As mentioned previously, computation itself is beginning to supplement data from flow physics experiments through numerical simulations that do not require any modeling of the physics. The following examples show how full simulations of the Navier-Stokes equations are being combined with experiment to provide a more fundamental understanding of turbulent boundary layers. These examples will suffice to illustrate what is meant by flow physics experiments.

The physical insight obtainable from direct Navier-Stokes simulation[7] is shown in Fig. 2. The figure highlights several important aspects of the structure of turbulence in a simulated flat-plate boundary layer. The simulation[8] was developed by P. R. Spalart of Ames, and employs no turbulence models of any kind. The complete, time-dependent Navier-Stokes equations were solved using spectral methods at each of the 9.4 million grid points in the computational domain. The Reynolds number based on momentum thickness is approximately 670. In the figure, elongated white surfaces identify the low-pressure cores of vortices. Shaded regions show where significant contributions to the Reynolds shear stress, $-\overline{uv}$, are occurring. Regions of low-speed fluid ejected outwards, and high-speed fluid swept wallward are labeled. These, and other realizations show that large hook-shaped vortical structures are clearly present in the numerical turbulent boundary layer, and the loactions of significant Reynolds stress contribution are seen to occur adjacent to these vortical structures. This level of understanding of the fundamental processes involved in the Reynolds stress generation is eventually expected to aid the development of improved statistical models that accurately reflect the underlying physical behavior of turbulence.

To gain similar physical insight, experimental techniques must generally employ multipoint measurement schemes. An example [9] of such an approach in a flow that is beyond the current capability of full simulation is shown in Fig. 3(a). Here, the large-eddy structure of a high Reynolds number, compressible turbulent boundary layer was investigated by mounting a fixed hot-wire at the wall in conjunction with another, traversing hot-wire mounted directly above the first. The objective was to map the spatial character and extent of the coherent eddies in a Mach 3 axisymmetric boundary layer, and to compare the results with low Reynolds number, incompressible flow experiments and simulations.

The long-time-averaged, space-time-cross-correlation functions between the near-wall sensor and various positions of the outer sensor are shown in Fig. 3(b). These curves show that measurable correlation between the two wires occurs up to a separation distance of at least half the boundary-layer thickness. These data suggest the presence of coherent outer-layer structures that extend well into the near-wall region, which may provide an energy transfer path between the free-stream flow and the near-wall, turbulence-producing region. The slope of these structures can be deduced from the correlation curves by using an eddy convection velocity, and is shown in Fig. 3(c) to vary from 5° near the wall to 30° in the outer layer.

The nature of these large disturbances can be studied in more detail by computing ensemble-averaged, mass-flow histories around strong, rapid accelerations and decelerations. The results for accelerations, shown in Fig. 3(d), closely resemble those of similar investigations performed in low-speed flows and in numerical simulations. This suggests confirmation of Morkovin's hypothesis that the basic structure of turbulent boundary layers is not fundamentally changed by compressibility, at least for moderate Mach numbers.

2.3 Physical Modeling Experiments

Practical CFD applications involving complex turbulent flows rely on statistical modeling of turbulence.[10] Physical modeling experiments are defined as experiments that provide guidance for and verification of the modeling process.

An example of a physical modeling experiment[11] used to improve turbulence modeling for transonic flows with strong shock-wave/boundary-layer interaction is shown in Fig. 4. The test model consisted of a cylindrical body fitted with a circular arc section similar to that of an airfoil. Shock-wave interactions of varying strengths were studied by varying free-stream Mach number. The choice of an axisymmetric geometry was made to eliminate three-dimensional effects. Mean-flow velocity and turbulence profiles, obtained with a Laser Doppler Anemometer System (LDA), and surface quantities such as pressure and oil-streak data were documented.

Computations of the flow field from a Reynolds-averaged, Navier-Stokes code revealed deficiencies in the turbulence modeling. By using a model developed primarily for attached boundary layers, the shock wave location was predicted incorrectly and consequently the pressure recovery was seriously overpredicted. The mean- and turbulence-profile data were used to explain the differences and to guide modeling improvement. The primary cause of the pressure recovery overprediction was the failure of the eddy viscosity model to adequately reflect the lag of turbulence adjustment through the shock wave. Using new modeling concepts in conjunction with the turbulence data resulted in a significant model improvement.[12] In particular, the "history effects" of the turbulence changes through the shock wave were accounted for by prescribing and solving an ordinary differential equation for the maximum shear stress development. The improved model results are shown.

2.4 Calibration Experiments

Calibration experiments are intended to reveal a code's ability to predict specific parameters. The data, in most instances, are limited with respect to their ability to determine the completeness of the flow modeling. Code calibration is prevalent and important to developing codes for real gas hypersonic applications because in this flight regime it is extremely difficult to provide ground test data for exact flight conditions and their attendant chemical and length scales. For example, facilities may duplicate flight energy levels but not match the air chemistry, or they may duplicate flight Mach number but not match the energy level.

An example of a calibration experiment intended to determine the applicability of the air chemistry model used in a parabolized Navier-Stokes code[13] is shown in Fig. 5. Drag data from 10° sharp cones fired down a Ballistic Range are shown as a function of angle of attack.[14] The angle of attack range represents the variation (uncertainty) in launch and flightpath angle of the cones from various firings done nominally at zero angle of attack. For these test conditions the flow is laminar; viscous-inviscid interaction is small; and the temperature in the viscous layer is sufficiently high to cause dissociation of the air. Drag owing to friction and pressure is about the same magnitude, so comparisons of the data with integrated pressures and skin friction from the computations provide a sensitive measure of how well the code predicts skin friction in a high-speed boundary layer. The favorable comparison with the computations performed by A. W. Strawa serves to illustrate that the code can predict drag in this chemically reacting flow field. More discussion on this experiment and its results are presented in Ref. 15.

2.5 Verification Experiments

Verification experiments provide the final validation of the codes. As such they require flow-field and surface measurements over a range of conditions and in sufficient detail to ensure that the flow physics is properly represented. The following example illustrates this category of experiment.

The improved turbulence model shown previously has recently been introduced into a transonic Navier-Stokes code and compared with data from an airfoil section. The airfoil was mounted in a specially designed test section with solid walls. Boundary-layer suction was applied upstream of the airfoil on the sidewalls to minimize interference. To further minimize wall interference, the upper and lower walls were contoured to streamline shapes that were predetermined by computation to account for the presence of the model, which further minimized interference. Tests[16] were performed at chord Reynolds number of 6×10^6 and angle of attack and Mach number were varied over a range sufficient to produce transonic flow covering weak and strong shock-wave/boundary-layer interaction and attendant displacement effects. The boundary layer was tripped on the upper and lower model surface to ensure turbulent flow beyond 7% chord. Model pressures, wall-boundary shapes and pressures, total drag, lift, and flow-field and wake velocities from an LDA system were documented. A data base of this type with minimal interference from a tunnel with solid walls provides an ideal basis for evaluating the development of codes for the transonic speed range because the codes can include wall-boundary conditions more precisely than interference corrections can be made to the data sets.

An example of some of the comparisons is shown in Fig. 6. At present the code does not include the solid wall-boundary conditions, but a preliminary assessment using these benchmark data indicates that the code provides very good simulation for the strong interaction cases when the improved turbulence model developed by Johnson and King[12] is employed. Results of the comparisons for one strong interaction case (where separation occurred at the trailing edge) are shown. The airfoil pressures, flow field velocities at constant heights above the model, and a wake profile at the trailing edge are compared with computations using two different turbulence models, a two-equation model,[17] and the Johnson-King model. The comparison shows that the computations using the improved turbulence model simulate the measurements very well. It is important to emphasize that this conclusion could not have been drawn without the complete data set composed of total drag, lift, boundary conditions and flow-field surveys. (See Ref. 16 for further discussion.)

3. MEASUREMENT REQUIREMENTS

3.1 Completeness

Each of the types of experiment discussed previously requires specific information that will enable a critical assessment of the code's capabilities at each stage of its development. Some examples of these measurements and the test conditions where they are needed are listed in Fig. 7 taken from Ref. 4. In these examples the measurements are representative and are germane to the development of Reynolds-averaged Navier-Stokes codes for fully developed turbulent flow.

Building block experiments must provide the data required for phenomenological understanding and/or modeling guidance and enable a critical test of the research code's ability to simulate important aerodynamic flows (e.g., shock-induced separation). Surface variables and flow-field variables, including turbulence data, are essential measurements. For the turbulence modeling problem, flow physics experiments and full numerical simulation of the Navier-Stokes equations carried out for simple flows at incompressible and compressible conditions can be very helpful in providing fundamental understanding and guidance of statistical modeling. But the flow modeling data must be obtained at representative flight Mach and Reynolds numbers where the codes are to be applied to ensure that the physics is modeled adequately.

Benchmark experiments must provide the parametric measurements necessary to calibrate or verify pilot code development. Surface and flow-field data at critical locations are the essential information since the objective of verification is to ensure that the code represents the correct physics or for calibration to ensure that the code adequately predicts some particular flow quantities. In order to clearly identify the applicable range of the code, parametric testing over as wide a range of flight Mach and Reynolds numbers is necessary. Experiments at extremes in such conditions are now often limited by instrumentation and facility development, as in hypersonic or high Reynolds number regimes.

Design experiments at the final stage provide the optimal configuration data necessary for performance evaluation and the experiments should be carried out as close to flight conditions as practical. CFD is expected to expedite the execution of these by eliminating the need for fine increments in parametric variations, by helping to resolve anomalous data sets, and by extrapolating the design performance data to flight conditions when facilities are unable to achieve them.

For each category of experiment careful measurements of boundary conditions are required because they may influence the flow field around test models. Moreover, they may be needed to initiate computations. Free-stream or initial conditions, wall-boundary physical location and necessary measurement variables, and precise model lines are examples of these measurement requirements.

3.2 Accuracy

Accuracy assessments for both computational procedures and experiments are essential. Otherwise there is no quantitative means for determining the limits and ranges of applicability for the codes. Uncertainty analysis is a well-established method for determining experimental data accuracy and should be a prerequisite for all levels of experiment used to develop CFD. It is useful during the planning and developmental phases of experiments, for evaluating data obtained with different instruments, and for comparing data from different experiments. (See Ref. 18 for more discussion on accuracy.)

Error estimates for test geometry dimensions, test operating and free-stream conditions, model and flow-field measured variables, and instrumentation should all be specified and the method used documented sufficiently to allow independent assessment.

Reliance on single experiments or measurement procedures for code validation purposes should be viewed with caution because of the current limitations of facilities and instrumentation needed to accomplish validation. (These limitations are especially present in hypersonic experiments.) Therefore, redundant measurement techniques and similar experiments performed in more than one facility may be required. In every case, careful substantiation and specification of experimental accuracy limits is crucial.

4. WIND TUNNEL REQUIREMENTS

The requirements for test facilities used to validate CFD were discussed in Ref. 4. The most important of these requirements are: (1) versatility, along with well-defined test and boundary conditions;

(2) appropriate scale and speed range; (3) accessibility of nonintrusive instrumentation; (4) provision for high-speed data systems; and (5) dedication of use to verification-experimentation.

5. FUTURE PROSPECTS

During the past year NASA has embarked on a comprehensive CFD validation program. Coordinated experimental and computational studies have been initiated at each of the NASA OAST Research Centers by teams comprised of computational and experimental research scientists.

At the Ames Research Center, the major thrust of the activity is supporting the development of codes employing the Reynolds-averaged Navier-Stokes equations. Data from in-house and university-funded experiments are expected to be published in the public domain and made available to other computational fluid dynamicists carrying out CFD validation. Some examples of the benchmark experiments that illustrate the scope of the program follow.

Turn-around-duct experiment: The experiment shown in Fig. 8 is under way to help guide the development of a 3-D incompressible Navier-Stokes code (INS-3D),[19] including its turbulence model. The application of the code is to study the axisymmetric flow in the Space Shuttle Main Engine turn-around-duct. The geometry consists of a constant area aspect-ratio 10 duct which turns an air flow, at high Reynolds number, through a 180° bend. The bend radius is equal to the duct height and some separation of the flow occurs on the inner corner wall near the end of the turn. A planer rather than axisymmetric geometry was chosen to permit access for nonintrusive laser instrumentation. Surface pressures, skin friction, velocity profiles, and Reynolds-averaged normal- and shear-stress profiles are being documented for a range of Reynolds numbers. Companion computations for this geometry are planned to verify the range of applicability of the code and various turbulence modeling approximations.

Transonic Wing and Wing-Body Experiments: Transonic experiments have been performed and others are now under way to guide the development of a transonic Navier-Stokes code (TNS).[20] The approach to the experiments is unique in that they are deliberately performed in solid-wall wind tunnel facilities. This test technique was chosen because the code can use the tunnel walls as boundary conditions and eliminate uncertain corrections to the data for wall interference. Once the code has been validated, it can confidently be used for free-air computations by appropriately changing the boundary conditions.

The first phase of the experimental activity was conducted several years ago.[21] A low-aspect ratio wing with a NACA 0012 profile section in the stream direction was mounted on the sidewall of a high Reynolds number facility and tested over a range of Mach numbers from 0.5 to 0.84, Reynolds numbers from 2×10^6 to 8×10^6, and angle of attack from 0° to 2°. Solid, straight, wind tunnel walls, sloped to correct for "tunnel empty" boundary-layer growth and instrumented with pressure taps were employed. Inviscid, no-slip boundary conditions along all walls were assumed for the computations, but that may not be entirely adequate as discussed later. Model pressures, wall-boundary pressures, surface oil flows, and limited velocity profiles obtained with an LDA were documented. Thus far, the data have been used by computational groups at the NASA Ames and Langley research centers.

The Ames group used comparisons with the data at the lower Mach numbers and angles of attack to establish confidence in the zonal techniques employed in the TNS code. At the higher Mach numbers and angles of attack they used comparisons with the data to sort out grid refinement and turbulence modeling issues.[22] Results of the comparisons with the high Mach number data were satisfactory only in the sense that they reproduced many of the complex flow features, but it could not be determined whether the turbulence model was solely responsible for the differences with the data. Recently the Langley group showed the importance of including the viscous, no-slip condition along the mounting wall. Their results, taken from Ref. 23, are shown in Fig. 9. A perspective view of the surface streamlines shows the influence of the viscous sidewall. The streamline patterns, especially in the side-wall region, are remarkably similar to the experimental oil flows. The comparison of computed and measured pressures on the tunnel walls and the wing shows good agreement except on the wing at the span location where a strong shock forms. These differences reflect the inadequacy of the turbulence model. Efforts are under way to improve the modeling.

A follow-on experiment conducted in a solid wall transonic test section is under way. This experiment eliminates some of the shortcomings of the one previously described: the model and the test facility are larger; the Reynolds number range can be extended; a more realistic, low-aspect, high-taper-ratio wing geometry is being used; and the sidewall boundary layer will be measured. Moreover, provision is made to test a wing-body combination. A photograph of the wing-body model mounted in the tunnel is shown in Fig. 10. The measurements to be made are also listed. The half-model body is mounted on the sidewall. The TNS computations will employ no-slip boundary conditions along the mounting wall and slip conditions on the other walls. Preliminary wing-alone and wall pressure data have been obtained recently.

3-D Supersonic Shock Interaction Experiments--Several experiments are under way to study the interaction of shock waves with turbulent boundary layers. Reference 24 presented data for a series of asymmetric separated flows on an ogive-cylinder-flare model. Shock unsteadiness was a major issue in the experiments and the reader is referred to Ref. 24 for further discussion.

Another series of experiments on a swept-wedge plate are being conducted by Settles. Figure 11 shows the geometry, test conditions and some recently published measurements.[25] The surface skin friction on the plate has been measured and compared with a computation solving the Reynolds-averaged Navier-Stokes equations. A two-equation turbulence model with wall functions was employed and the results compare well with the data. In Ref. 25, comparisons with data for other wedge angles using both two-equation and algebraic turbulence models show that turbulence modeling is not critical to resolving the structure physics of these flows, probably because they are dominated by inviscid effects. However, the effects of viscosity are essential to reproducing the structures, and Euler codes probably cannot represent these flows adequately.

Hypersonic All-Body Experiment: The experiment depicted in Fig. 12 is being performed to guide the development of a 3-D Parabolized Navier-Stokes code[26] that uses up-wind differencing to obtain sharp shocks. The geometry is a 70° swept delta with an elliptical cross section. At the two-thirds body length station, an expansion surface forms the upper part on the model. Some recent experimental results taken from Ref. 27 are also shown in Fig. 11. Spanwise pressure distributions for 15° angle of attack and $M = 10.3$ over the forebody region ahead of the expansion are shown compared with the computations for a single streamwise station, assuming either laminar or turbulent flow from the leading edge. The agreement is good with either assumption because viscous-inviscid interaction has a small influence on the pressure distribution at this Reynolds number. When the remaining measurements of heating and velocity profiles are completed, other validation issues such as aerodynamic heating will be addressed.

6. CONCLUDING REMARKS

Experiments play a critical role in the development of CFD. They provide phenomenological data to help understand the physics of complex flows; they provide guidance in the modeling process where the physics is unknown or so complex that computational procedures are not practical; and ultimately they provide the verification necessary to establish the limits of applicability to various aerodynamic flows.

Four types of experiments supporting the development of CFD were described: (1) flow physics experiments, (2) flow modeling experiments, (3) calibration experiments, and (4) validation experiments. The first two types were broadly categorized as building block experiments. They provide the phenomenological and modeling data required for research code development. An additional new technological advance contributing to the building block data base is full- and large-eddy simulations and computational chemistry. The building block data base is more detailed and often requires sophisticated instrumentation and test techniques. The second two types were broadly categorized as benchmark experiments. These experiments provide the data needed to identify the accuracy and limitations on the code's ability to compute complex aerodynamic flows. The data requirements differ from the building block experiments in the sense that phenomenological and modeling issues are not investigated in detail.

The categories of experiments and corresponding measurements lead to specific requirements for facilities used for validation. Versatility, appropriate scale and speed range, accessibility for nonintrusive instrumentation, computerized data systems, and dedicated use for verification are the important requirements.

A synergistic, comprehensive approach to validation was introduced. A program is under way to provide validation experiments that can guide the development of advanced computational procedures for application to complex flows. Both computational and experimental fluid dynamicists are focusing on key aerodynamic problems whose solutions are paced by the lack of adequate understanding of the flow physics and modeling and by the lack of adequate validation data to verify code development. The major challenge for success of the program depends on timely accomplishment of the experiments, development and implementation of new instrumentation, and development of appropriate high Reynolds number and high Mach number, high-enthalpy facilities.

REFERENCES

1. Bradley, R. G., Bhateley, I. C., and Howell, G. A., "Computational Fluid Dynamics - Transition to Design Applications," Supercomputing in Aerospace, NASA Conference, Publication 2454, pp. 69-767, Mar. 1987.

2. Current Capabilities and Future Directions in Computational Fluid Dynamics, National Research Council, National Academy Press, Washington, D.C., 1986.

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

4. Marvin, J. G., "Wind Tunnel Requirements for Computational Fluid Dynamics Code Verification," Paper 34, AGARD Symposium on Aerodynamic Data Accuracy and Quality, Requirements and Capabilities in Wind Tunnel Testing, Naples, Italy, Sep. 28-Oct. 1, 1987.

5. Moin, P. and Kim, J., "The Structure of the Vorticity Field in Turbulent Channel Flow, Part I - Analysis of Instantaneous Fields and Statistical Correlations," J. Fluid Mechanics, Vol. 155, p. 441, 1985.

6. 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, p. 60, Jan.-Feb. 1985.

7. Kline, S. J., and Robinson, S. K., "Turbulence-Producing Coherent Structures in the Turbulent Boundary Layer: Progress of a Cooperative Evaluation," to be presented at the Zaric Memorial International Seminar on Wall Turbulence, Dubrovnik, Yugoslavia, May 1988.

8. Spalart, P. R., "Direct Simulation of a Turbulent Boundary Layer up to Rtheta = 1410," NASA TM 89407, 1986.

9. Robinson, S. K., "Space-Time Correlation Measurements in a Compressible Turbulent Boundary Layer," AIAA Paper 86-1130, Atlanta, GA, May 1986.

10. Marvin, J. G., "Turbulence Modeling for Computational Aerodynamics," AIAA Journal, Vol. 21, No. 7, pp. 941-955, Jul. 1983.

11. 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, Williamsburg, VA, 1979.

12. Johnson, D. A., and King, L. S., "Transonic Separated Flow Predictions Based on a Mathematically Simple, Nonequilibrium Turbulence Closure Model," IUTAM Symp. on Turbulent Shear Layer/Shock Wave Interaction, Paris, France, Sep. 1985, NASA TM 86826, Oct. 1985.

13. Prabhu, D. K., Tannehill, J. C., and Marvin, J. G., "A New PNS Code for Three-Dimensional Chemically Reacting Flows," (U), AIAA 87-1472, Honolulu, HI, Jun. 1987.

14. Intiere, P. F., Kirk, B. K., Chapman, G. T., and Terry, J. E., "Ballistic Range Tests of Ablating and Nonablating Slender Cones," (U), AIAA Journal, Vol. 8, No. 3, pp. 558-564, Mar. 1970.

15. Deiwert, G. S., Strawa, A. W., Sharma, S. P., and Park, C., "Experimental Program for Real Gas Flow Codes at NASA Ames Research Center," Paper No. 21, AGARD Symposium on Validation of Computational Fluid Dynamics, Lisbon, Portugal, May 1988.

16. Mateer, G. G., Seegmiller, H. L., Coakley, T. J., and Hand, L. A., "An Experimental Investigation of a Supercritical Airfoil at Transonic Speeds," AIAA Paper 87-1241, Honolulu, HI, Jun. 1987.

17. Coakley, T. J., "Numerical Simulation of Viscous Transonic Airfoil Flows," AIAA Paper 87-0416, Jan. 1987.

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

19. Chang, J. L. C., and Kwak, D., "A Three-Dimensional Incompressible Flow Simulation Method and Its Application to the Space Shuttle Maine Engine, Part II--Turbulent Flow," AIAA Paper 85-1670, Reno, NV, Jan. 1984.

20. Holst, T. L., Kaynak, U., Gundy, K. L., Thomas, S. D., Flores, J., and Chaderjian, N. M., "Numerical Solution of Transonic Wing Flows Using an Euler Navier-Stokes Zonal Approach," AIAA Paper 85-1640, Cincinnati, Jul. 1985.

21. Lockman, W. K., and Seegmiller, H. L., "An Experimental Investigation of the Subcritical and Supercritical Flow About a Swept Semispan Wing," NASA TM-84367, Jun. 1983.

22. Kaynak, U., Flores, J., "Advances in the Computation of Transonic Separated Flows over Finite Wings," AIAA Paper 87-1195, Honolulu, HI, Jun. 1987.

23. Vasta, V. N., and Wedan, B. W., "Navier-Stokes Solutions for Transonic Flow Over a Wing Mounted in a Tunnel," AIAA Paper 88-0102, Reno NV, Jan. 1988.

24. Brown, J. D., Brown, J. L., Kussoy, M. I., Holt, M., and Horstman, C. C., "Two-Component LDV Investigation of 3-Dimensional Shock/Turbulent Boundary Layer Interactions," AIAA Paper 87-0553, Reno, NV, Jan. 1987.

25. Kim, K. S., and Settles, G. S., "Skin Friction Measurements by Laser Interferometry in Swept Shock Wave/Turbulent Boundary-Layer Interactions."

26. Lawrence, S. L., Chaussee, D. S., and Tannehill, J. C. "Application of an Upwind Algorithm to the PNS Equations," (U), AIAA Paper 87-1112, Honolulu, HI, Jun. 1987.

27. Lockman, W. K., Cleary, J. W., and Lawrence, S. L., "Flow Visualization and Pressure Distributions for an All-Body Hypersonic Aircraft," (U), Paper No. 53, Fourth National Aero Space Plane Technology Symposium, Monterey, CA, Feb. 1988.

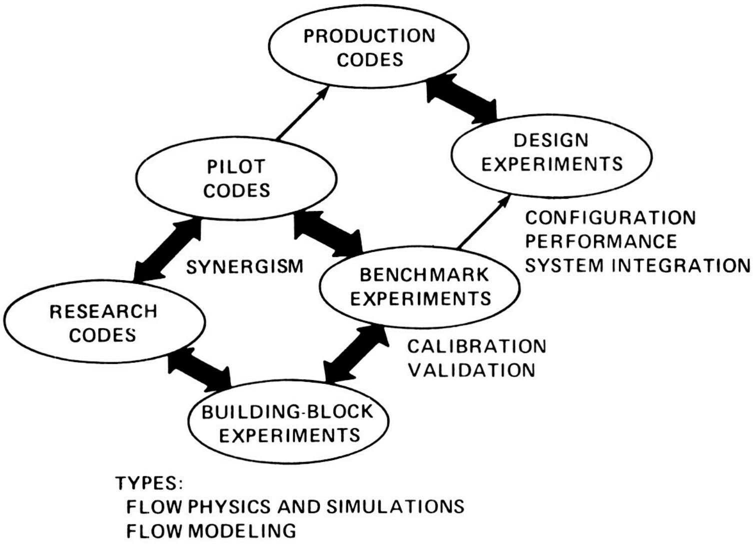
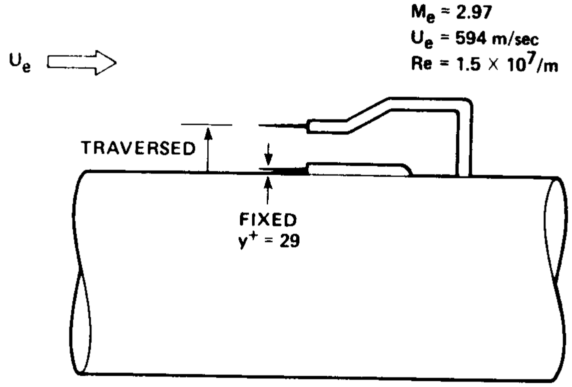
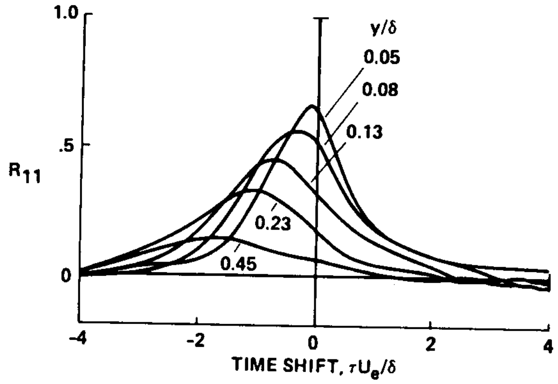


Figure 1. The role of experiment in developing CFD.

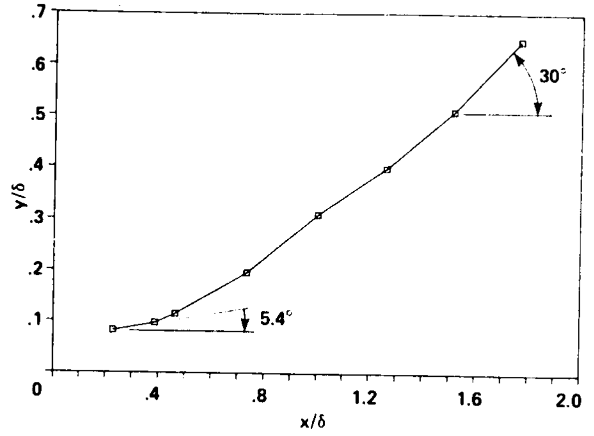
Figure 2. Turbulent flow-physics obtained from a full simulation of the Navier-Stokes equations. $M_\infty = 0$, $Re_\theta = 670$.



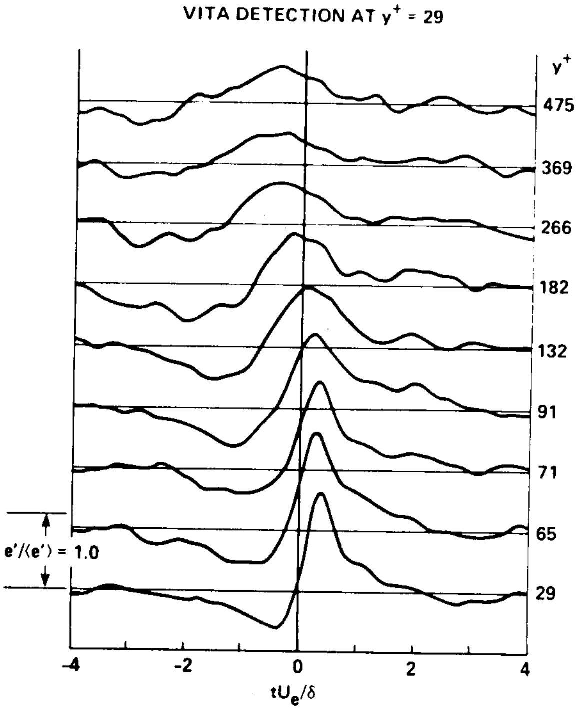
(a) HOT-WIRE PROBE INSTALLATION



(b) CROSS-CORRELATIONS BETWEEN TWO HOT-WIRES

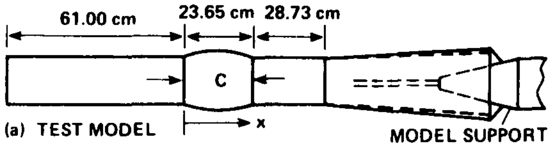


(c) MEAN ANGLES OF DISTURBANCE FRONT

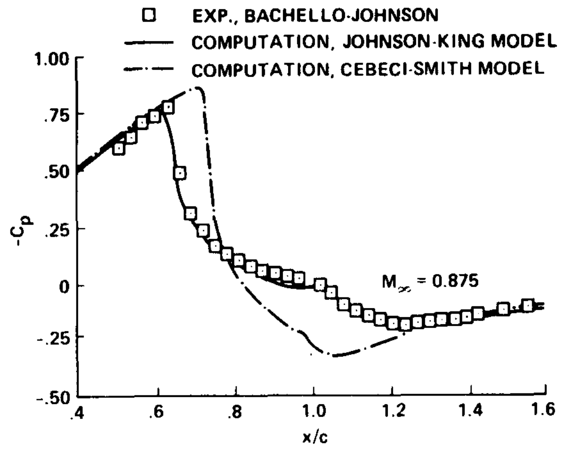


(d) ENSEMBLE-AVERAGED VOLTAGES FOR LARGE POSITIVE EVENTS

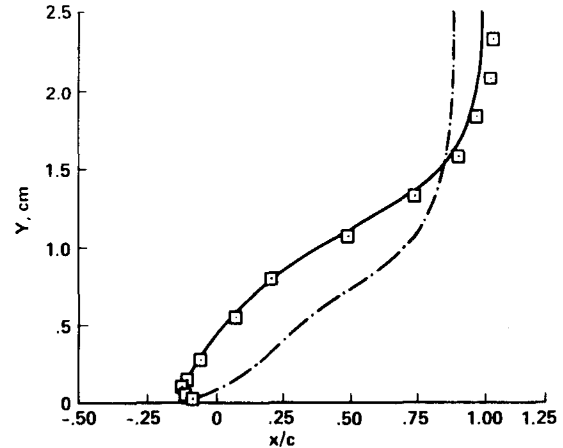
Figure 3. Turbulent flow-physics obtained from the experiment of Ref. 9 employing multiple hot-wires. M_∞ = 3; Re = 1.5 × 10^7/m.



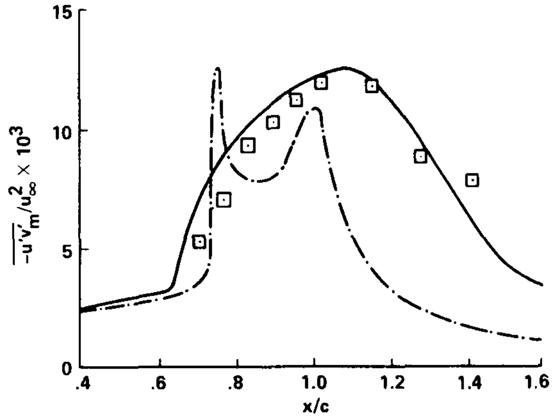
(a) TEST MODEL



(b) PRESSURE COEFFICIENT

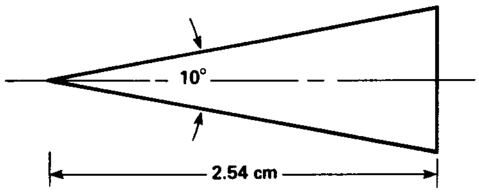


(c) VELOCITY, x/c = 1.0

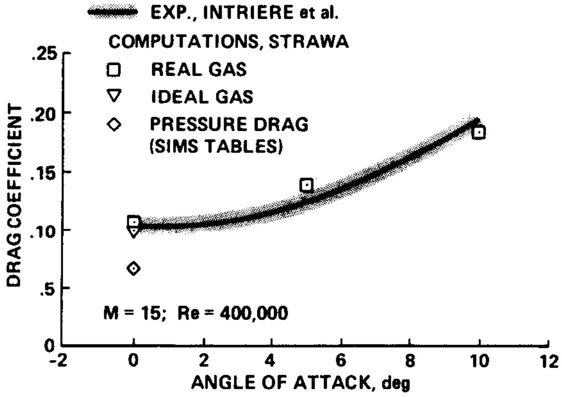


(d) MAXIMUM TURBULENT SHEAR STRESS

Figure 4. A flow-modeling experiment used to develop an improved turbulence model.

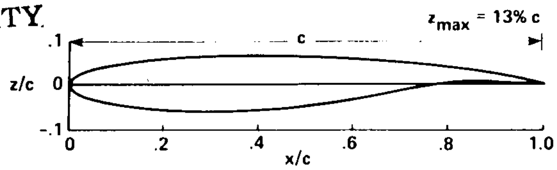


(a) GEOMETRY

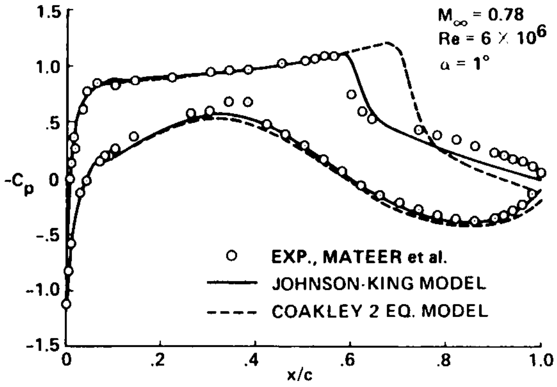


(b) DRAG DATA FOR 10° CONE

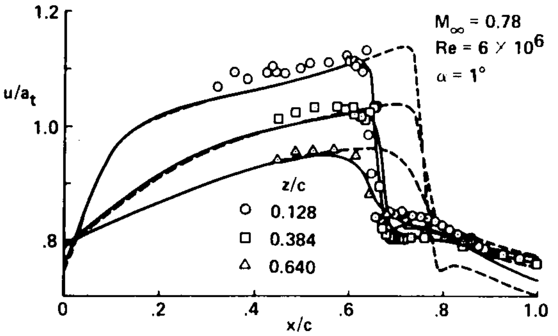
Figure 5. A calibration experiment used to evaluate a real gas chemical model in a parabolized Navier-Stokes code. 10° cone; M_∞ = 15; and Re_L = 400,000.



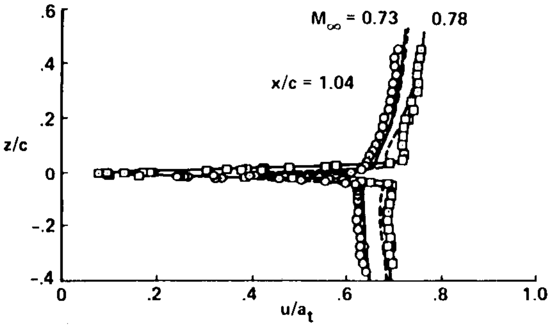
(a) TEST MODEL – MBB VA-2 AIRFOIL
TRAILING EDGE THICKNESS = 0.522% CHORD



(b) PRESSURE COEFFICIENT



(c) FLOW FIELD VELOCITIES

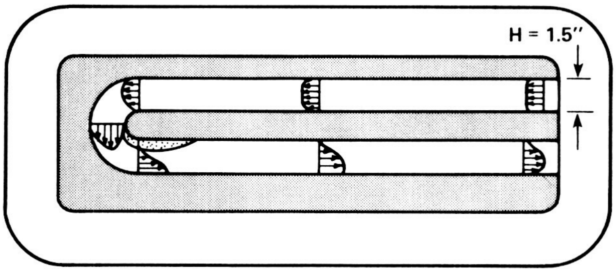


(d) AXIAL VELOCITY PROFILES

Figure 6. A benchmark airfoil experiment used to verify development of an improved turbulence model.

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 (CONFIGURATIONAL)	DRAG, LIFT, MOMENTS, HEAT LOADS, SHEAR LOADS BOUNDARY CONDITIONS (SEE ABOVE)	AS CLOSE TO FLIGHT M_∞, Re, α AS PRACTICAL

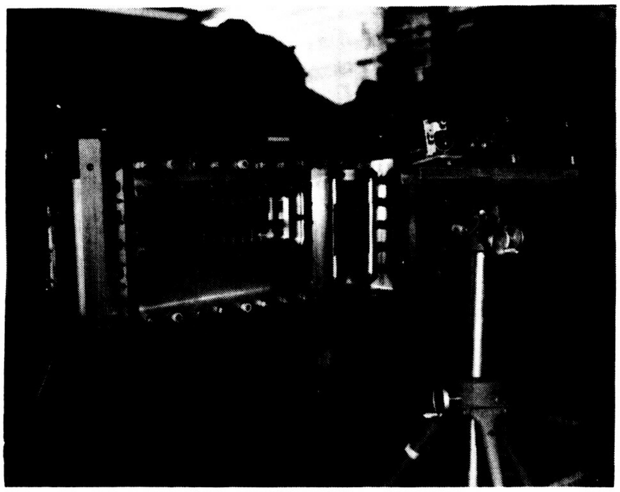
Figure 7. Experimental requirements.



(a) FLOW FIELD AND GEOMETRY

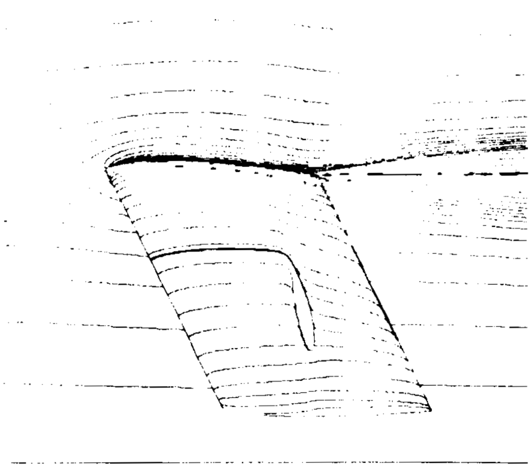
SURFACE: p_w, c_f, OIL FLOW
FLOW FIELD: U, V, $\overline{u'}$, $\overline{v'}$, $\overline{u'v'}$
$RE_H = 0.1 - 3.0 \times 10^6$, $M = 0.1 - 0.3$

(b) MEASUREMENTS AND TEST CONDITIONS

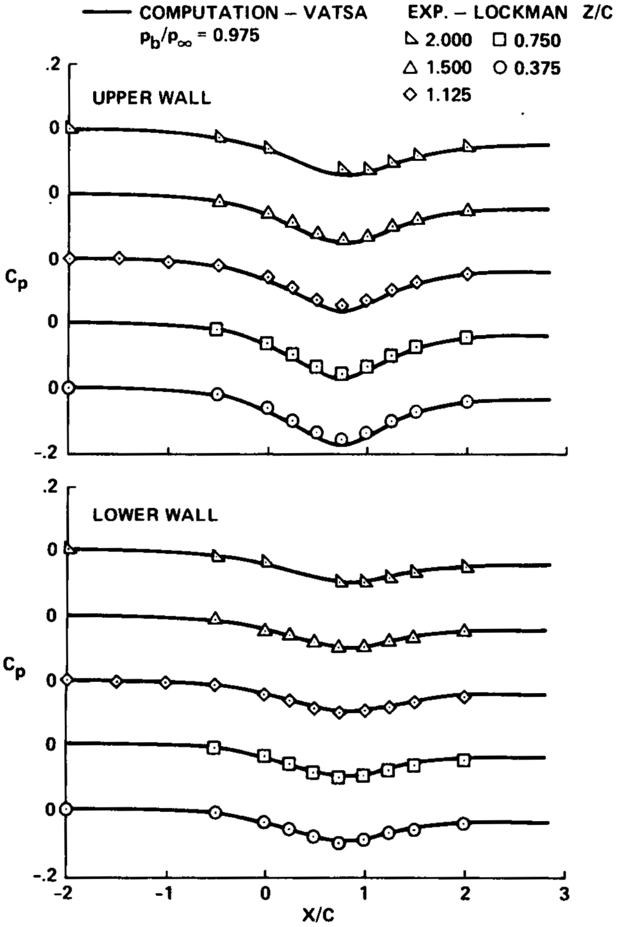


(c) VIEW SHOWING LASER SKIN-FRICTION INTERFEROMETER

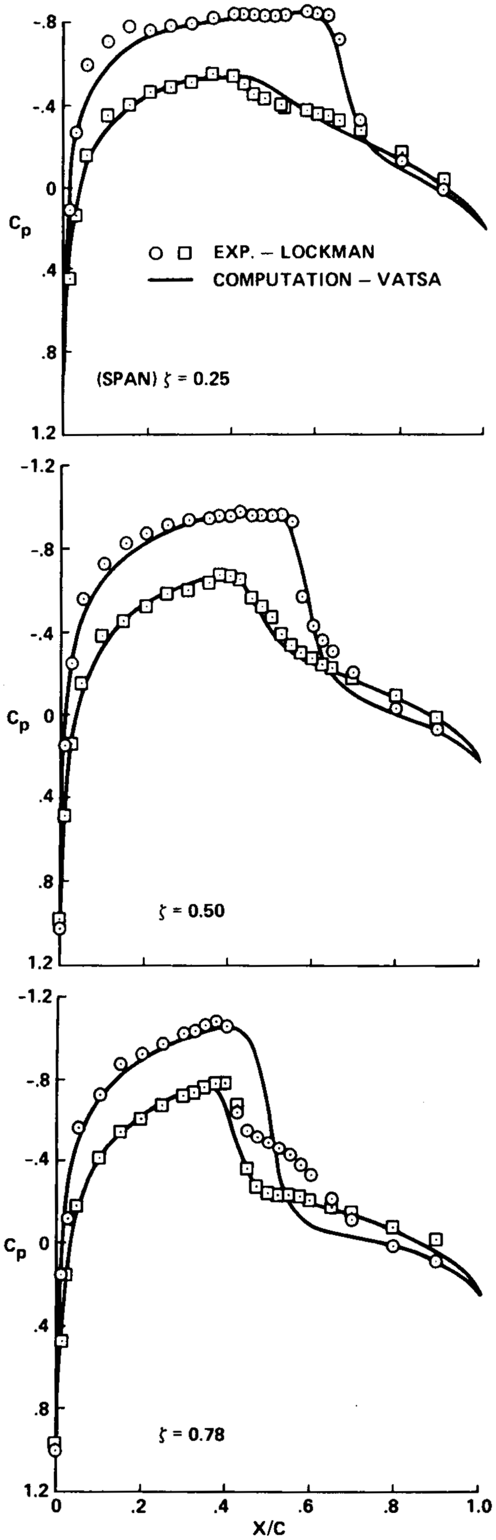
Figure 8. 2-D turn-around-duct experiment.



(a) PERSPECTIVE VIEW OF COMPUTED STREAMLINES ON WING AND WALL



(b) TUNNEL WALL PRESSURE COMPARISONS



(c) WING PRESSURE COMPARISONS

Figure 9. Comparison of Reynolds-averaged Navier-Stokes computations and data from a verification experiment. $M_\infty = 0.826$, $Re = 8 \times 10^6$, $\alpha = 2°$, A/R = 3.

GEOMETRY

- OGIVE-CYLINDER HALF-BODY
- WING
  - NACA 64A008 STREAMWISE SECTION
  - ASPECT RATIO = 3.2
  - TAPER RATIO = 0.25
  - L.E. SWEEP ANGLE = 36.9°

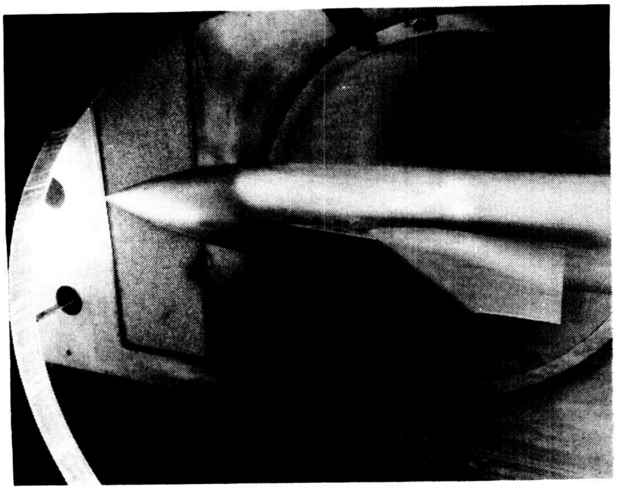
TEST CONDITIONS

- $M_\infty = 0.5$ TO 0.8
- $Re_{\infty,\bar{c}} = 1 \times 10^6$ TO 10×10^6
- $\alpha = 0°$ TO 15°

MEASUREMENTS

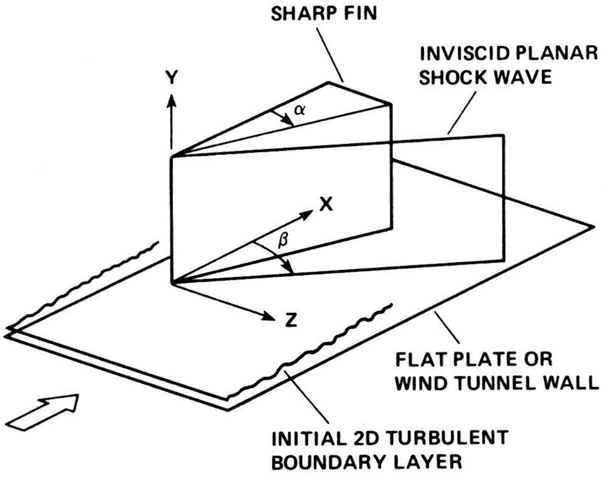
- FLOW VISUALIZATION
  - SURFACE (OIL FLOW)
  - FLOW FIELD (VAPOR SCREEN)
- SURFACE PRESSURES (WING, TUNNEL WALLS)
- MEAN VELOCITY – FLOW FIELD (LDV AND PROBES)

(a) GEOMETRY, TEST CONDITIONS AND MEASUREMENTS



(b) PHOTO OF MODEL MOUNTED IN TUNNEL

Figure 10. A low-aspect-ratio wing-body experiment.



PENN STATE EXPERIMENT:

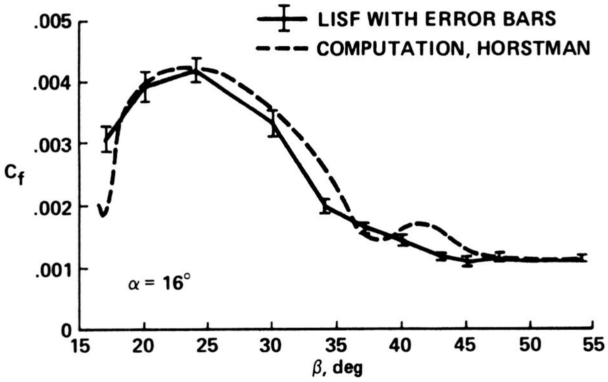
$M_\infty \approx 2.4 - 4$, $\alpha = 4 - 22°$, $Re_x \approx 10 \times 10^6$

MEASUREMENTS

SURFACE: OIL FLOW, p_w, c_f

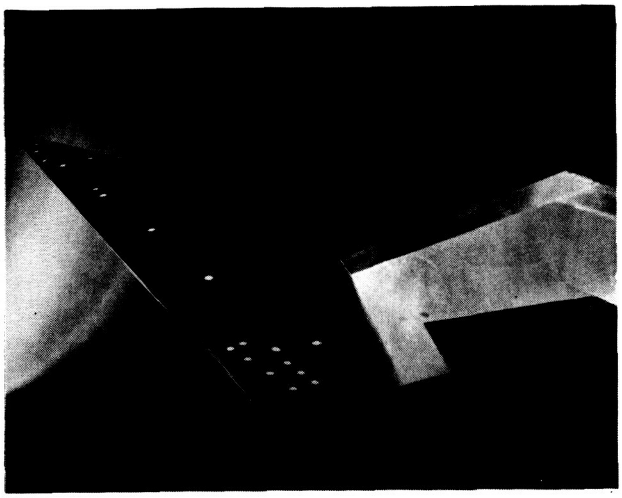
FLOW FIELD: VAPOR SCREEN, p_{t_2}, YAW ANGLE

(a) FLOW GEOMETRY

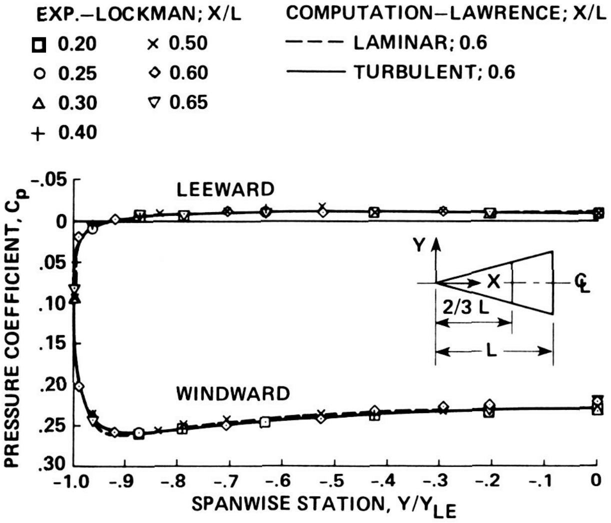


(b) SKIN FRICTION ON THE PLATE, x = 3.5 in.

Figure 11. A 3-D shock-wave boundary layer interaction experiment.



(a) ALL-BODY HYPERSONIC AIRCRAFT MODEL IN NASA/ AMES 3.5-ft HWT; LENGTH = 3 ft



(b) SPANWISE PRESSURE DISTRIBUTIONS FOR FOREBODY $\alpha = 15°$; $M_\infty = 10.3$; $Re_{\infty,L} = 5 \times 10^6$

Figure 12. A hypersonic all-body experiment.

NASA
National Aeronautics and
Space Administration

Report Documentation Page

1. Report No. NASA TM-100087	2. Government Accession No.	3. Recipient's Catalog No.

4. Title and Subtitle Accuracy Requirements and Benchmark Experiments for CFD Validation	5. Report Date May 1988
	6. Performing Organization Code
7. Author(s) Joseph G. Marvin	8. Performing Organization Report No. A-88123
	10. Work Unit No. 505-60-11
9. Performing Organization Name and Address Ames Research Center Moffett Field, CA 94035	11. Contract or Grant No.
	13. Type of Report and Period Covered Technical Memorandum
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546-0001	14. Sponsoring Agency Code

15. Supplementary Notes

Point of Contact: Joseph G. Marvin, Ames Research Center, MS 229-1
Moffett Field, CA 94035 (415) 694-5390 or FTS 464-5390

16. Abstract

The role of experiment in the development of Computational Fluid Dynamics (CFD) for aerodynamic flow prediction is discussed. CFD verification is a concept that depends on closely coordinated planning between computational and experimental disciplines. Because code applications are becoming more complex and their potential for design more feasible, it no longer suffices to use experimental data from surface or integral measurements alone to provide the required verification. Flow physics and modeling, flow field, and boundary condition measurements are emerging as critical data. Four types of experiments are introduced and examples given that meet the challenge of validation: (1) flow physics experiments; (2) flow modeling experiments; (3) calibration experiments; and (4) verification experiments. Measurement and accuracy requirements for each of these differ and are discussed. A comprehensive program of validation is described, some examples given, and it is concluded that the future prospects are encouraging.

17. Key Words (Suggested by Author(s)) Computer code validation Experiments Turbulent flows	18. Distribution Statement Unclassified-Unlimited Subject Category - 34

19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of pages 16	22. Price A02

NASA FORM 1626 OCT 86 For sale by the National Technical Information Service, Springfield, Virginia 22161