The essential ingredients needed for CFD can be found in universities, but lack of sufficient support may diminish their contributions

by Antony Jameson
Princeton University

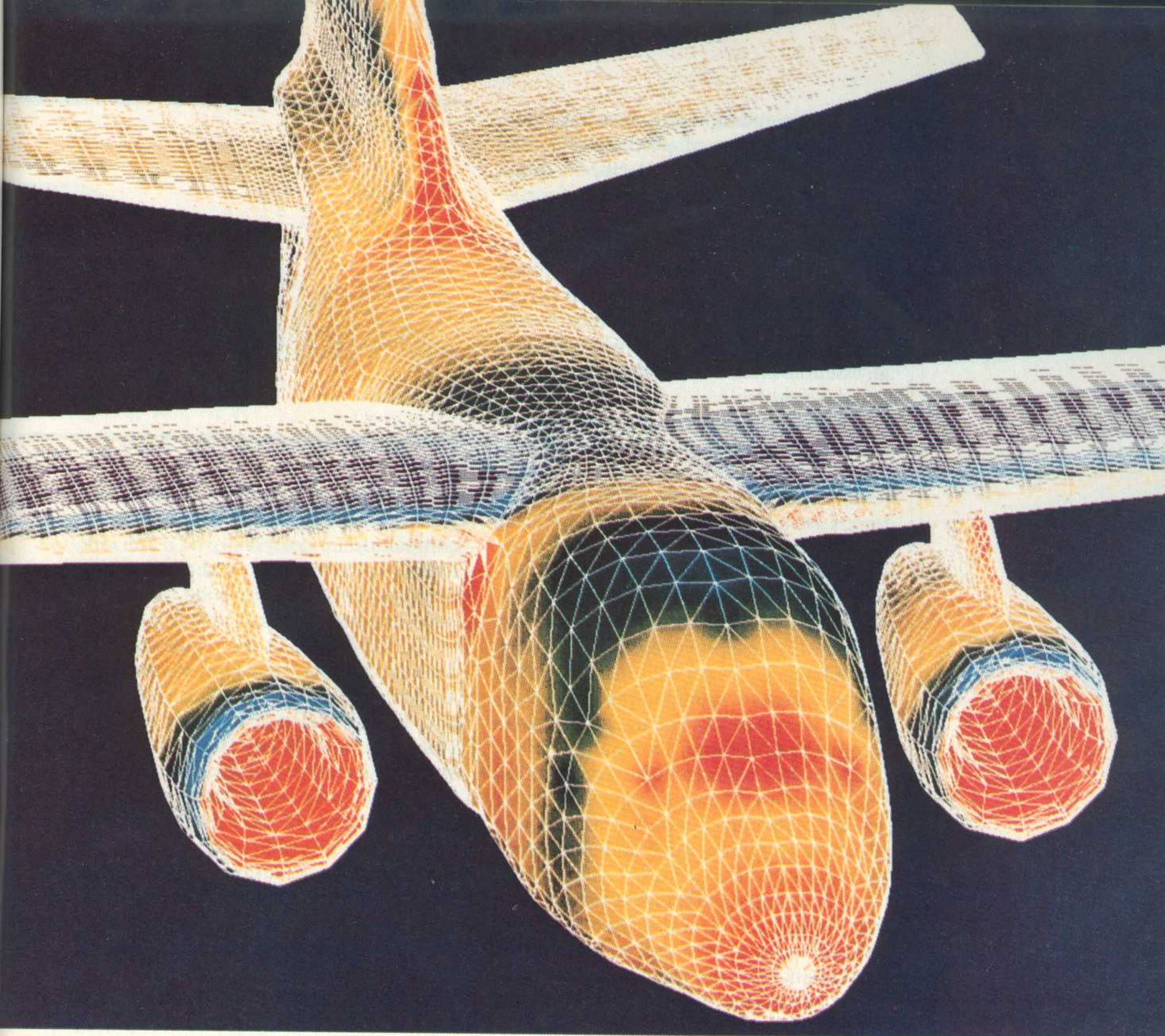
Universities foster CFD growth

The growth of computational fluid dynamics (CFD) has largely occurred since 1965. Before then, computational methods were rarely used in aerodynamic analysis, although they were widely used for structural analysis, and basic theoretical groundwork and fundamental algorithms had been developed, especially at the Courant Institute.

The decade 1970-1980 saw the emergence of practical computer programs for three-dimensional transonic flow, some of which, like FLO22 and FLO27-28 for swept wing calculations, were developed in universities. The latter is incorporated in Boeing's A488 program for wing analysis.

During the 1980s CFD rapidly expanded to more complex applications, which included more complete geometric configurations and higher level flow models. Flexible, efficient methods were developed for solving the Euler and Navier-Stokes equations on arbitrary meshes, for example ARC3D from NASA-Ames and FLO57 from Princeton. FLO57 provided the basis of Euler codes used at Boeing, for the Lockheed TEAM code, and for NASA-Langley's TLNS3D code to solve the thin layer Navier-Stokes equation. Although FLO57 was written at Princeton, the university did not have a computer powerful enough to run the program.

During this decade, multigrid acceleration methods were extended to the Euler equations, which achieve very rapid convergence to a steady state by advancing the solution through a sequence of successively coarser grids, on each of which the correction is determined by the error on the next finer grid. This essentially allows



Surface mesh and pressure distribution for Lockheed S3A were computed using AIRPLANE.

global equilibrium to be achieved simultaneously with local equilibrium. Intensive research efforts were also concentrated on the development of nonoscillatory shock capturing schemes.

Mesh generation has continued to be an obstacle to the treatment of configurations with complex geometry. Here again academia has played a significant role in the development of general techniques, such as those based on the solution of elliptic equations. However, it still may take months to produce an acceptable mesh, and in the last few years methods using unstructured meshes have begun to gain more general acceptance.

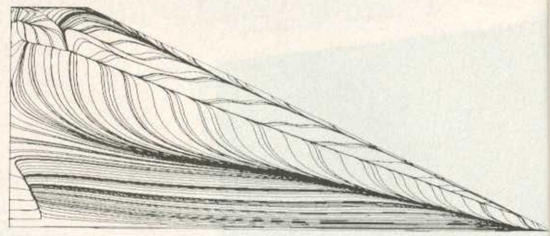
The development of methods using unstructured meshes was a major effort at Princeton throughout the '80s. An Euler solution for a complete Boeing 747 was first obtained there at the end of 1985. Unstructured meshes may offer flexibility in the treatment of very complex configurations and quick turnaround that would be very hard to match with a structured mesh approach. They are also particularly suited to the introduction of automatic procedures for adaptive refinement of the mesh as the calculation proceeds, in order to improve the resolution of features containing high gradients, such as shock waves and vortices.

CFD presents a particularly complex challenge because it lies at the intersection of a number of distinct disciplines—mathematics, computer science, fluid mechanics, and (for useful aeronautical applications) aerospace engineering.

The theoretical underpinnings rest in mathematics and numerical analysis, in-

Antony Jameson is the James S. McDonnell Distinguished University Professor of Aerospace Engineering in the Dept. of Mechanical and Aerospace Engineering at Princeton University.

Using FLO97, the laminar (left) and turbulent flow on the top surface of a cropped delta wing was simulated.



cluding formulation of proper boundary conditions, theory of partial differential equations (such as mixed elliptic, parabolic, and hyperbolic systems of equations), and error estimates and convergence proofs for discrete approximation schemes. These are fundamental topics particularly appropriate for university research, and where universities have historically played a strong role. They are also an essential platform for algorithm design and development.

Computer science has become increasingly important in CFD. With the advent of first vector and then massively parallel computers, algorithms can no longer be developed without considering the architecture on which they are to be implemented. In algorithms using unstructured meshes, for example, loops that exchange data along edges between neighboring mesh points introduce vector dependencies. Graph coloring algorithms can be used to separate the edges into groups such that no two edges of the same color meet at the same mesh point. Dependency is then eliminated by using a separate loop for each group. Computer graphics techniques are vital as well, for effective post-processing and interpretation of the results.

The complexity of fluid flow is such that useful CFD methods cannot be developed without a good understanding of fluid mechanics, and blind use of inappropriate CFD methods by users lacking such understanding can lead to unfortunate results. Numerical dissipation, for example, can result in spurious flow separation at a high angle of attack in what is supposed to be an inviscid solution.

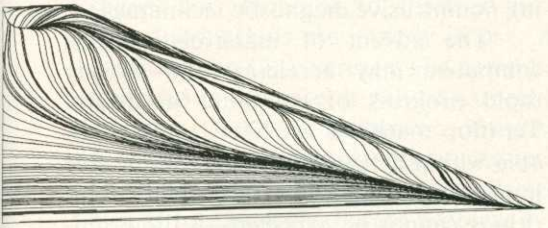
Finally, it is difficult to use CFD methods effectively in the design process without a broad knowledge of aerospace engineering, both to determine which calculations are likely to be useful and cost-effective, and to know how to interpret the results to improve a proposed design.

Five principal stages can be identified in the development of CFD software: choice of a mathematical model, mathematical analysis of the model to ensure that the problem is properly formulated, formulation of a discrete approximation scheme, implementation as a computer program, and validation.

The choice of a model depends on the complexity of the flow, the required level of accuracy, and the computational cost. Since the Reynolds numbers associated with the flight envelopes of full-scale aircraft are very large, the flows that must be predicted are generally turbulent. The computational complexity of a full simulation of a turbulent flow can be estimated as proportional to the cube of the Reynolds number. Thus, a direct simulation of the full Navier-Stokes equations is not feasible, forcing the use of mathematical models with some level of simplification.

These simplifications may be acceptable because efficient flight is generally achieved by the use of streamlined shapes that avoid flow separation and minimize viscous effects. In fact, many useful predictions can be made using inviscid models. Inviscid calculations with boundary layer corrections can provide accurate predictions of lift and drag when the flow remains attached, but iteration between the inviscid outer solution and the inner boundary layer solution becomes increasingly difficult with the onset of separation. Procedures for solving the full viscous equations are likely to be needed for the simulation of arbitrary complex separated flows, which may occur at high angles of attack or with bluff bodies. To treat flows at high Reynolds numbers, estimating turbulence effects by Reynolds averaging the fluctuating components is usually necessary, which requires the introduction of a turbulence model.

Computational costs vary drastically with the choice of math model. Panel methods can be effectively implemented with higher-end personal computers (an Intel 80386 processor and 80387 coprocessor, for example). Studies of the dependency of the result on mesh refinement have demonstrated that inviscid transonic potential-flow or Euler solutions for an airfoil can be accurately calculated on a mesh with 160 cells around the section and 32 cells normal to the section. With multigrid techniques 10-25 cycles are enough to obtain a converged result. Consequently



airfoil calculations can be performed in 3 sec on a Cray-XMP or 18 sec on a Convex C210. Correspondingly accurate three-dimensional inviscid calculations can be performed for a wing on a mesh, say with $192 \times 32 \times 48 = 294{,}912$ cells, in about 5 min on a single-processor Cray-YMP, or less than a minute with eight processors.

Viscous simulations at high Reynolds numbers require vastly greater resources. At Princeton we have found that about 32 mesh intervals are needed to resolve a turbulent boundary layer, in addition to 32 intervals between the boundary layer and the far field, for a total of 64. To prevent degradations in accuracy and convergence due to excessively large aspect ratios (in excess of 1,000) in the surface mesh cells, the chordwise resolution must also be increased. Fairly accurate solutions can be obtained on a 512×64 mesh in 100 multigrid cycles. In three dimensions, this implies the need for meshes with 5-10 million cells ($512 \times 64 \times 256 = 8{,}388{,}608$ cells).

A typical algorithm, such as the multistage scheme used in Princeton, requires on the order of 5,000 floating point operations per mesh point in one multigrid iteration. With 10 million mesh points the operation count is on the order of 0.5×10^{11}/cycle. Given a computer capable of sustaining 10^{11} operations/sec, 200 cycles could be performed in 100 sec. Simulations of unsteady viscous flows would be likely to require 1,000-10,000 time steps. A further progression to large eddy simulation of complex configurations would require even greater resources.

The selection of sufficiently accurate math models and a judgment of their cost effectiveness ultimately rest with industry. In conceptual and preliminary design phases the emphasis is on relatively simple models that provide results with rapid turnaround and low computer cost, in order to evaluate alternative configurations and perform quick parametric studies. The detailed design stage requires the most complete simulation that can be achieved with acceptable cost.

In the past the low level of confidence in numerical predictions forced the extensive use of costly wind tunnel testing early in the design stage. The limited number of models that could be fabricated also limited the range of design variations that could be evaluated. In the future it is anticipated that the role of wind tunnel testing will be more one of verification, although there will be a continuing need for experimental research to improve understanding of the physics of complex flows.

Given a well-formulated mathematical model and a stable, accurate numerical approximation scheme, the difficulties of implementation in computer programs are often underestimated. As simulations become increasingly complex, CFD software becomes more difficult to develop and maintain. Even with a single author it is hard to eliminate programming errors. When code development is undertaken by a number of authors, perhaps changing over time, there may be no one who knows precisely what is contained in a large software package. Structured, carefully controlled programming techniques become essential. Otherwise uncontrolled changes may introduce new errors.

Code validation is increasingly being recognized as being vital to raising confidence in CFD use. In considering this requirement, it is important to distinguish between the correctness of the program and the suitability of the math model to the application. Simply comparing experimental data with numerical results provides no way to distinguish the source of discrepancies, whether they are due to faulty numerical approximation or programming, or to deviations between the math model and the true physics. Among the measures that may be taken to assure program correctness are: modular programming, which enables substitution of logically identical but independently programmed versions of each module (different loop structures, for example) with identical results; mesh refinement studies—does the program give a definite answer as the mesh interval is successively decreased?; and consistency checks for established mathematical properties (for example, does a wing with a symmetric section produce no lift at zero angle of attack?).

Ideally, the numerical scheme and

computer program should be checked and internally validated by tests such as these to the point that discrepancies with experimental data can properly be attributed to inadequacy of the math model and not the program itself.

CFD may now be considered to have reached the point where the key enabling problems for inviscid flow calculations are well understood. A variety of high-resolution nonoscillatory shock capturing schemes are available, for example, while multigrid and other acceleration methods allow converged solutions to be obtained in 10-50 cycles. Effective techniques have been developed to treat complex configurations with both structured and unstructured meshes. Unstructured meshes, such as Princeton's AIRPLANE code, offer the flexibility to treat arbitrarily complex configurations without the need to spend months developing an acceptable mesh.

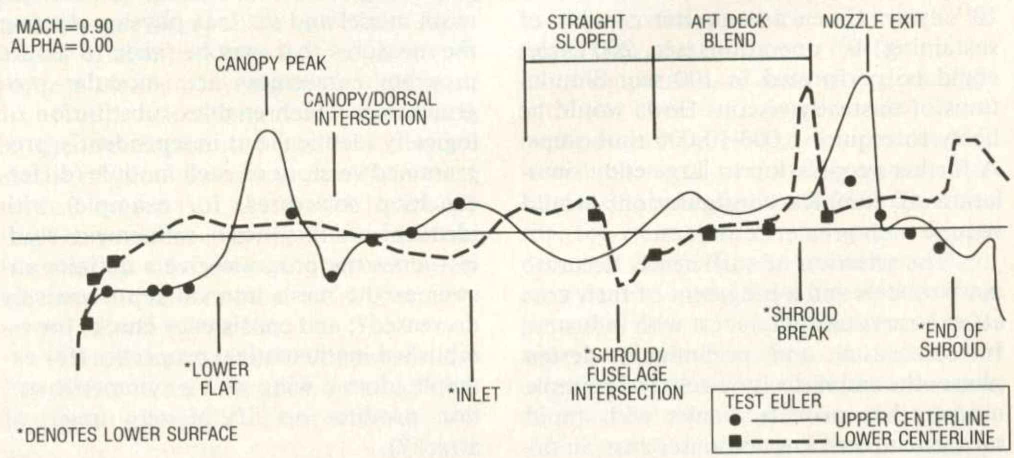
In contrast, viscous flow simulations of full configurations remain too expensive for engineering applications, and the confidence level in the results of viscous predictions is much lower. The greatest difficulties are presented by separated and turbulent flows. Useful calculations can be performed, provided the results are interpreted with care.

For predicting separated flows, a universally reliable turbulence model has yet to be developed. Although a new approach to the derivation of turbulence models based on renormalization group theory, under intensive study at Princeton, has produced promising results, a better understanding of the physics of turbulence is needed to develop both improved turbulence models and better subgrid-scale models for large eddy simulation. This is an area in which universities can play a strong role. Progress may depend on coordinated numerical studies and carefully controlled experiments using nonintrusive diagnostic techniques.

The advent of massively parallel computers may accelerate the already rapid progress of computer hardware. Teraflop machines are likely to be available within a few years. But comparable improvements in the efficiency of algorithms cannot be expected—a thousandfold reduction in the number of cycles in calculations already being completed in less than 100 cycles is clearly impossible.

The only possibility of a radical reduction in the number of arithmetic operations lies in improved discretization methods, such as higher order schemes and adaptive mesh refinement procedures. The value of higher order and spectral methods for appropriate applications (with sufficient smoothness in the solution) has been demonstrated, for example, in computational studies of transition from laminar to turbulent flow. Adaptive mesh refinement is currently the subject of widespread investigation.

Parallel architectures will force a reappraisal of existing algorithms and extensive development of new parallel software. Beyond the shift to more sophisticated algorithms, the present challenge is to extend the effective use of CFD to more complex applications, such as chemically reacting flows and flows with moving boundaries. The analysis of turbomachinery is a particularly challenging arena. Interdisciplinary applications, in which CFD is coupled with computational analysis of other properties of a design, will play an increasing role. These may include structural, thermal, acoustic, and electromagnetic analyses.



A Northrop grid generator and a version of FLO57 computed fuselage pressure distribution on a YF-23 model at Mach 0.9. Results were then compared with measurements.

Research must also be focused on the best way to use CFD to arrive at good designs. Inverse methods that determine shapes corresponding to specified pressure distributions are already in use. As confidence in CFD increases and computational costs decrease, it will be natural to combine CFD with automatic optimization techniques to derive superior designs. One approach, being explored at Princeton, is to regard the design problem as a control problem for a system governed by a partial differential equation (the flow equations) with boundary control. The Frechet derivative of the cost function with respect to the shape can be calculated by solving a single adjoint partial differential equation, at a cost comparable to a flow solution.

This method, which eliminates the need to estimate sensitivity coefficients by varying each parameter in turn, has been successfully applied to the design of transonic airfoils. The technique promises opportunities for improvement in a wide variety of applications. Some of these include the design of a wing for a supersonic transport aircraft or the design of hydrofoils to reduce cavitation.

Progress in CFD will be most rapidly achieved by drawing on all available intellectual and physical resources. Not to make use of academia's resources could delay, or even preclude, significant research breakthroughs, and would jeopardize the university's role in training the next generation of scientists and engineers. Universities must continue to receive financial support for research, access to state-of-the-art computers, and the free flow of information. The widest possible dissemination of resources will result in the greatest rate of progress.

To maximize the potential for long-term payoffs, a balance must be sought in the distribution of funding between a few large-scale projects of key importance and a wide range of smaller ones. Government lab funding should not come at the expense of university research.

It remains in everyone's best interest to foster continuing contact between universities and government labs. There are equally great benefits to be gained from more direct contacts and closer collaboration between academia and industry. Universities gain by exposure to the real problems encountered in industry; industry gains by drawing on the vast university pool of talent.