CFD's Use in Teaching Fluid Dynamics and Aerodynamics

TERRY L. HOLST

Ames Research Center

Abstract

The field of computational fluid dynamics (CFD) has advanced to the point where it can now be used for the puipose of fluid dynamics physics education. Because of the tremendous wealth of information available from numerical simulation, certain fundamental concepts can be efficiently communicated using an interactive graphi- cal interrogation of the appropriate numerical simulation data base. In other situations, a large amount of aerody- namic information can be communicated to the student by interactive use of simple CFD tools on a workstation or even in a personal computer environment. The emphasis in this presentation is to discuss ideas for how this process might be implemented. Specific examples, taken from previous publications, will be used to highlight the presentation.

Introduction

Use of computational fluid dynamics (CFD) tools by the aerospace field to increase understanding of fluid dynamic and aerodynamic phenomena has been rapidly increasing during the past decade especially the last several years.

The primary reasons for this are the rapidly increasing simulation capabilities in the CFD field and the rapidly expanding capabilities in computer hardware perfor- mance. For example, computer hardware execution speed has increased by a factor of about 15 over the past decade and by over 200 during the past two decades. This rapid advance in computational execution speed is displayed in figure 1. The top curve shows how the theoretical peak execution speed has improved with time and includes effects from both circuit speed and architectural improve- ments, e.g., vectorization speed ups from Cray-type computers. The lower curve represents the improve ment in execution speed due to just circuit speed. The middle curve represents the actual improvement in execution speed performance from a variety of CFD application codes as approximated from the shaded symbols.

Another reason for the dramatic increase in the use of sci- entific computational tools is that industry has discovered the positive influence that computational analysis can Year Imroduced

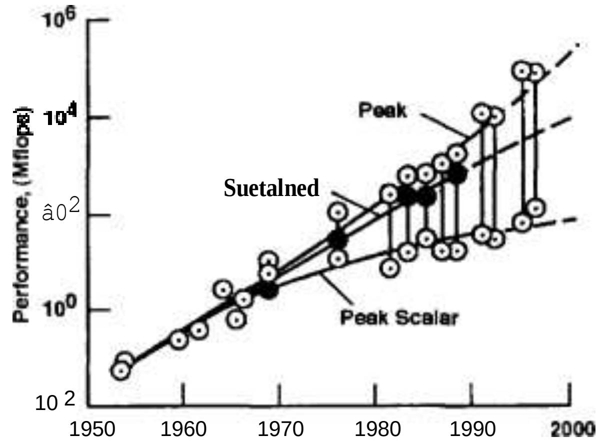


Figure 1. Conventional main-frame computer eKecution speed improvement as a function of time, results taken from references 1—4.

have on aircraft, spacecraft, and missile design. Improved efficiency in aerospace vehicle performance at reduced design cost and risk is a direct result of increased use of computational simulations. Indeed, additional advance- ment in this area is crucial to enable the United States to maintain its technological advantage in the aerospace sciences.

Just as numerical simulation has become a significant and growing aspect of the aircraft design process, the stage is set for a dramatic **increase** in the utilization of **CFD in** the educational arena. In this context, it is not **meant** to imply that the study of CFD will increase dramatically, but that the use of CFD as a teaching tool for other areas or disci- plines of fluid dynamics will increase. This utilization should range from enhancing the understanding of nonlin - ear engineering models, e.g., the aerodynamics of tran - sonic wings, to obtaining a better understanding of fluid physics, e.g., flat plate boundary layer transition. Through the synergistic utilization of CFD coupled with an appro- priate level of experimental validation, students will obtain a better understanding of the physical aspects of aerodynamics and fluid mechanics as well as how to interpret the effects of numerical error associated with CFD solutions.

The remainder of this paper will provide a review of some of the current areas of CFD research that may be utilized in the educational environment in the near future. These areas are especially attractive if improvements in worksta- tion computing continues at today's current fast rate. In addition, areas that may be used more immediately,
i.e., even today, will be presented and discussed.

Review of CFD Applications

The first results used to establish the abilities of **CFD** are a set of full potential solutions for a variety of transonic wing configurations (taken from refs. 5—7). In these simu - lations the nonlinear full potential equation is solved for the in viscid transonic flow field complete with transonic shock waves. Results from two different full potential computer codes, TWING (ref. 5) and FLO28 (ref. 6), are compared with experiment (ref. 7) in figure 2 for the ONERA M6 wing at transonic flow conditions. Although there are some discrepancies in the computed results, both show the same trends, i.e., they both predict a double shock structure on the upper surface including a supersonic-to-supersonic oblique shock swept approximatelyparallel to the wing leading edge. Most of the discrepancies are a direct result of a coarse grid used in the numerical simulations, a direct result of main memory limitations from a decade ago.

An additional full potential result computed with the TWING code and taken from reference 8 is shown in figure 3. In this figure the drag-rise characteristics (CD versus M p) are compared for two transonic wing cases: an original or baseline wing and an optimized wing. The baseline geometry was modified using the QNMDIF optimization code (refs. 9—10) to produce the optimized wing by minimizing the value of cruise DfL. As can be seen from figure 3 the drag-rise characteristics of the optimized wing are significantly improved over the origi - na1 baseline wing. It should be pointed out that the drag values associated with figure 3 are pressure drag values only, i.e., they do not contain skin friction drag.

The most interesting aspect of these simulations, espe- cially in the present context, is the amount of computer time required for a complete simulation. The computing times reported in reference 5 are on the order of 1O—20 sec on a single processor of a Cray Y -MP computer. An entire aerodynamic performance curve, such as the drag rise curve presented herein, requires on the order of only one minute. This machine was in the supercomputer class just a decade ago but now is about even with a high end workstation in execution speed and does not even match a reasonably advanced personal computer in terms of main memory. Such simulations would be easily adapted to the educational environment and will be discussed in more detail in the next section of this paper.

The next example results used to establish the state of the art in CFD applications are a set of Reynolds-averaged

Navier—Stokes (RANS) simulations ahout a canard—wing— fuselage configuration (refs. 1 1 —13). The geometry con - sists of an ogi ve-cylinder fuselage with a canard and wing composed of circular-arc airfoil sections. The canard and wing are closely coupled, have zero-twist, are mid- mounted, and are highly swept and tapered. Precise geo- metric details can be found in references 11—13.

Numerically computed results compared with experimen- tal results taken *from* reference 14 are presented in figures 4 and 5. Figure 4 shows a variety of force and

figuration for both deflected canard (10 deg) and unde- flccted canard cases over a range of angles of attack. Note thc generally good agreement with experiment for these comparisons. Figure 5 shows comparisons of component lift and pitching moment for the deflected canard case.

Again the computed results are in good agreement with experiment. For this set of computations there is a com - pJex canard—wing vortex interaction that exists, especially for the higher angles of attack. Some cases have even been computed that predict wing and canard vortex breakdown, a phenomenon that is also present in the experiment.

The results presented in figures 4 and 5 demonstrate the ability of RANS methods to predict the aerodynamic characteristics of reasonably complex configurations even in the presence of significant viscous effects. Such com - putations would not have been possible with the full potential approach or even an Euler equation approach.

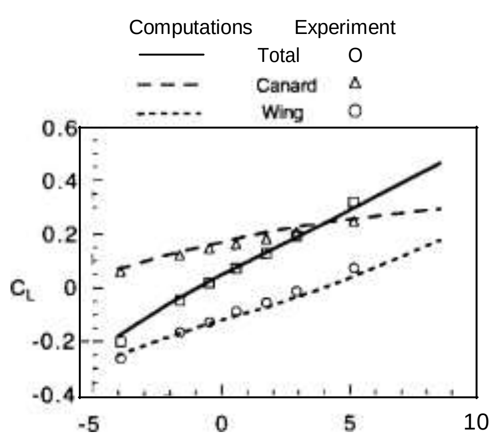
The computer time expense associated with these RANS computations is considerably larger than for the full potential computations presented previously and ranges from about 2 to 15 hours of single processor time on a Cray C-90 supercomputer. The variation in required exe- cution time is due to variations in geometry, flow condi- tions, and grid rernement level. Complete details of this set of computations are given in references 1 l —13.

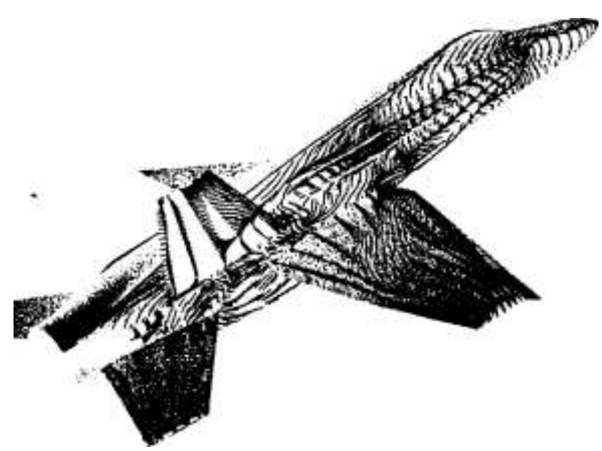
The next example simulation involves the numerical solu- tion of the Reynolds-averaged Navier—Stokes (RANS) equations for the flow field about a nearly complete of flow solutions about several increasingly complete geometrical representations of the F-18. This series staned with forebody flow solutions (refs. 15—17), proceeded

with wing—fuselage simu)ations (refs. 18—19), and to date has concluded with a nearly complete F-18 aircraft that includes the fuselage, tail, and wing with deflected lead- ing edge flap (ref. 20). Results from the last effort utiliz- ing the most complete geometry are presented in figures 6 and 7. This level of geometric complexity is possible in a structured grid approach because of the zonal grid scheme used. The flow field is broken into a series of grid zones

-igure 4. Comparison of computed and experimental forces and moments for a canard—wing—body configuration *for* ***both*** *undeflected and deflected canard* ***cases,***
AT, — ***0.B4. (a)*** *Lift* ***curvas, (b)*** *drag polars,* ***(c) moment curves.***

using an overset or chimera zonal grid approach. Each grid zone is designed to capture the effects of one aspect of the geometry, e.g., vertical tail, wing, or the leading - edge flap, and is generated without regard to the other





a, (deg)

a
)

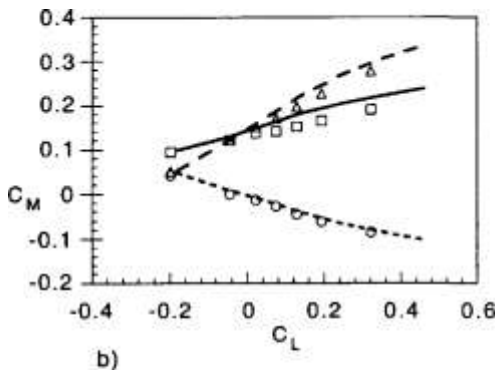


Figure S. Comparison of component lih and pitching moment carves *for a canard—win ody configuration in which the canard is deflected at 10 deg, My —0.84. (a) Lih curves, (b) moment curves.*

geometrical aspects of the overall configuration. During the fiow solution process, information from one grid zone is communicated to another zone using a general interpo- lation procedure.

The present F-18 simulation presented in figures 6 and 7 utilized 10 chimera grid zones in conjunction with a bilateral plane of symmetry, which resulted in a grid of about 900,000 points. The simulation was run for 1,000 iterations in a non-time-accurate mode and an additional 4,100 iterations in a time-accurate mode. The solution required 8 million words of memory on a Cray Y-MP computer and 55 hours of single-processor CPU time.

Figure 6 shows the limiting streamlines that have been computed just off the aircraft surface, and figure 7 shows three-

Figure 6. Computed limiting surface streamlines, F-18 aircrah, My -- 0.243, a — 30.3 dep, Re — 11 x 1 0^ (ref. 25).

Figure 7. Computed particle fraces *showing LEX vortex breakdown and forward surface flow paftem, F-18* **aicra/i',**
— 0.243, a — 30.3 deg, Red -— 11 x 1 0^ (ref. 25).

streamlines on the leading-edge extension (LEX), wing, and deflected leading-edge flap. Each of these figures

dis- plays only an instant in time for this time-dependent solution.

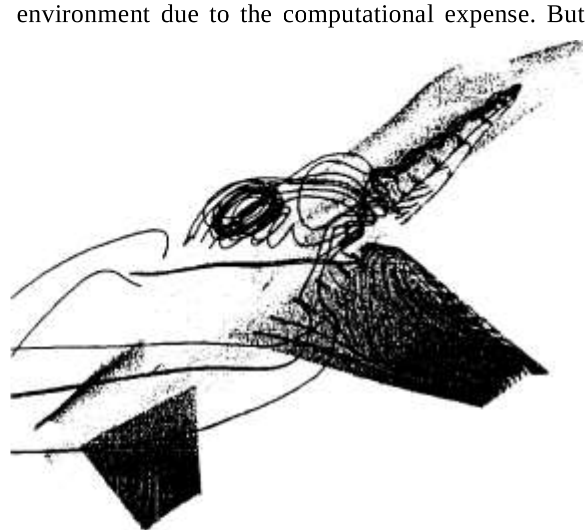
Zonal grid approaches of several different varieties have been used in a number of other applications to solve the RANS equations including Buning et al. (refs. 21—22) for ascent-mode Space Shuttle computations, Meakin and Suhs (ref. 23) and Dougherty et al. (ref. 24) for store- separation computations, Kiris et al. (ref. 25) for biofluid applications, Flores and Chaderjian (refs. 26—28) for a simulation of a reasonably-complete F- l6A aircraft, and

Chawla et a1. (ref. 29) and Smith ct al. (ref. 30) for several powered-lift flow simulations.

An example of a powered lift-flow computation, taken from Smith et al. (ref. 31), is displayed in figure 8. In this figure the viscous flow about a Harrier YAV-8B aircraft in ground effect is displayed via a series of particle traces emanating primarily from the front and rear nozzles of the aircraft. The flow conditions are hover-like in vol ving a low forward speed of 30 knots at an altitude of 30 feet above the ground plane. This computation in volved
2.5 million grid points distributed within 18 chimera grid zones and about 40 CPU hours on a Cray Y-MP computer.

As can be seen from figure 8, there are many flow fea- tures thai exist in this computation including a ground horseshoe vortex generated by the interaction of the

freestream and jet flows in ground effect. Other aspects thai can be studied by analyzing the numerical data base include the fountain effect created by two parallel jets impinging side-by-side on the ground plane, heating on the aft fuselage caused by the aft hot jet fountain effect, the dramatic loss of lift experienced by powered-lift air- craft in ground effect (the so-called suck-down effect), and propulsion efficiency loss due to hot gas ingestion into the propulsion system inlets. Understanding all these complex flow phenomena, even in a qualitative sense, is a difficult task. Such a simulation would be very difficult to perform in today’s educational environment due to the computational expense. But



an interactive interrogation of such a “canned” solution file would yield a wealth of information about a state-of-the-art aerodynamics prob- lem. More discussion on this idea will be presented in the next section of this paper.

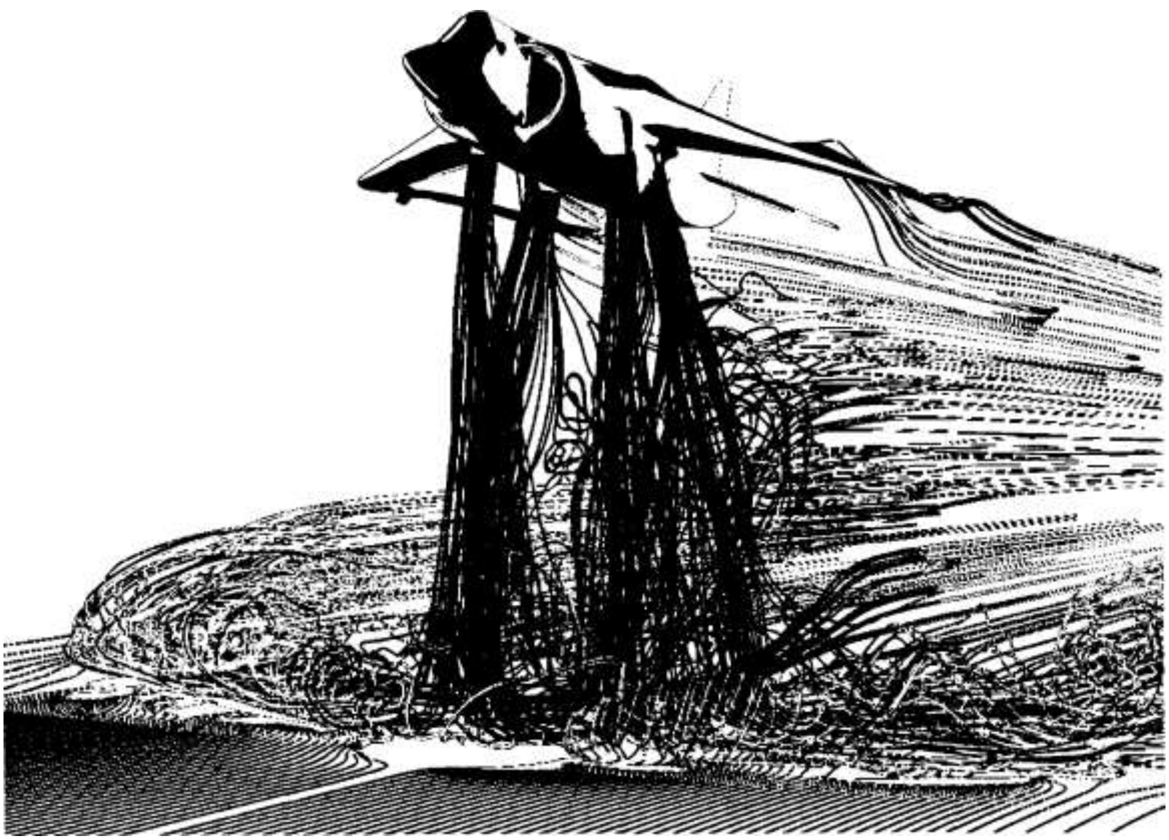


Figure 8. Numerically computed particle traces around a Harrier YAV-8B in ground effect at a speed of 30 knois and an altitude of 30 feet above the ground plane, a -— 8.0 deg (ref. 31).

Educational Uses of CFD

Computer simulations of fluid flows are increasingly being used in the classroom for a variety of reasons. Learning by doing, harnessing the potential of parametric variation, quickly and simply optimizing findings, and visualizing whole unsteady flow fields are just some of the features that may be accessed in the classroom today or in the near future. Hardware cost efficiency and appropriate educationally focused applications software availability are the primary driving issues for further adoption of these capabilities. As can be seen in Figure 1, the ratio of hardware cost-effectiveness to hardware speed is steadily increasing. Rapid advancement for "workstation" class computer hardware, i.e. desktop computers often with strong graphics capabilities, is another facet of quick computer efficiency development that is not illustrated in figure l. This category of computers, whose capabilities have been increasing even more quickly than those of the mainframe machines shown in figure 1, has the potential to have the greatest effect on the classroom.

Applications CFD software may now be executed on modern high-end workstations at 50 MFLOPS and greater. These machines may provide a cost-performance ratio that is an order of magnitude better than that of huge mainframe computers. Several smaller applications, such as inviscid wing analysis or viscous airfoil analysis, may be completed in a fair amount of time using a workstation, despite the fact that their processing power is insufficient for big ones. a

On such a high-end work- station, such calculations often take anything from a few seconds to a few tens of minutes. The amount of time required on a computer will vary greatly depending on the complexity of the simulation and the formulation that is used. There are three main applications of simulations in the classroom: (1) parametric analysis, (2) design (via trial and error), and (3) visualization. We'll go further into each of these topics below.

Parametric Analysis

Parametric demonstrations using computational tools may be used to show many of the prevalent patterns in fluid mechanics, such as the linear connection between lift and angle of attack for an airfoil or the dependency of a fluid's viscosity on its temperature.

linear dependence of the thickness of the laminar boundary layer on the square root of the Reynolds number for flat plates. Several of these aspects of fluid mechanics, which are highly valuable, are often given in the classroom in the form of theory and experiment. The goal of this study is not to advocate replacing these well-established methods but rather to supplement them with a third option, namely, computational fluid dynamics (CFD) simulations. The knowledge of fluid mechanics benefits from the contributions of experiment, theory, and CFD, all of which contribute in their own unique ways. Thus, using all three gives a synergistic effect. CFD adds a degree of parametric freedom to the picture, for example. What occurs if compressibility is included in, or if one reaches the C L point on a C L vs a curve? One question that a CFD analysis can

the answers to these problems; it contributes to the explanation of a discrepancy between theory and experiment in complicated non-linear flow regimes.

The flexibility of computational simulations allows them to potentially serve a wide variety of applications with varying parameter values. The full range of flow situations, from incompressible to compressible flows to rarefied atmospheric entry flows to chemically reacting non-ideal-gas flows to flows involving combustion, and flows over a wide range of Mach numbers and Re, have all been demonstrated in modern applications software (refs. 50-55). Experimentally providing expertise in a wide range of the aforementioned domains would be quite costly for the educational setting, even for the simplest of configurations or challenges. The cost of calculations in some of these domains is high at the moment, but this is expected to change in the near future, particularly for reduced geometry applications..

Design (Learning by Trial and Error)

The second main topic covered here is a subset of the material covered in the section on parametric variation. Hence, the geometry itself serves as the variable. In aerodynamics and fluid dynamics, this is an issue of paramount importance, and as such, it is given unique consideration.

The primary goal of research in this field is to use parametric variation of airfoil shape to provide a desired effect or design, such as a target cruise £JD or maximum cruise LED. Parameters and values that work well with this style of training

Variables such as thickness, angle of attack, leading edge radius, trailing cdgc angle, and more are discussed. Minimum wing volume, maximum wing root bending moment, and maximum unfavorable pressure gradient are all examples of limitations that may be utilized to highlight different aspects of interest. A CFD computer software may be used to show the effects of any or all of these parameters and limitations on airfoil (or wing) L/D performance, which can be both instructive and inspiring. Design contests, in which students compete to see who can create an airfoil with the highest possible cruise lift-to-drag ratio (L/D) within a certain set of limits, may be a great way to get students excited and interested in the subject matter. The usefulness and effectiveness of the educational tool may be greatly increased by adding more complexity, such as many design points, laminar flow control, or high-lift devices. The capacity of computational simulation to quickly alter geometric form is put to use in the design application mentioned above; this feature would be difficult, if not impossible, to create using just an experimental technique.

Visualization

Finally, visualization is the third area where a CFD approach might be useful in a school setting. Accurate simulation of many fluid flow phenomena is possible using CFD, however this takes a lot of time on the computer or in real life. Direct numerical simulations of transitional flow over a flat plate and simulations of flow around a full aeronautical vehicle, as those shown at the beginning of this work, are two examples of such flows. In the classroom, such simulations will not be feasible for a long time.

Nevertheless, "canned" simulations, i.e. solution files that have been previously calculated and placed on disk or some form of CDROM device with a huge storage capacity, make it possible to use the results of such simulations in a classroom or student laboratory setting. This method may be used to store and retrieve data from both computational and experimental studies. These watching sessions might be prerecorded like a movie or they can be student-driven and participatory. This second option is more likely to be exciting and alluring. Relevant information would be extracted from the findings utilizing an interactive graphical workstation environment. The computational model may be directly compared to the experimental model. For the time being, at least, most sessions of this sort would need to focus on stable flows in three dimensions or unsteady flows in two dimensions. Unsteady flows in three dimensions would need too much storage space, and the interactive process would be too time-consuming and expensive to obtain the desired features. As the highly competitive graphical workstation industry continues to evolve, it is possible that even the biggest simulations will be able to be analyzed interactively in a classroom setting.

Concluding Remarks

(2) From simple fuli-potential simulations of a single shape to complex Navier-Stokes modeling of an entire aircraft, CFD technology has been briefly reviewed throughout a wide spectrum of capabilities. There will likely be even more interest in using computational simulations for a wide variety of purposes throughout the coming decade. It is expected that this pattern would be most pronounced in the industry of designing passenger airplanes.

(3) In order to make a difference in the classroom, this presentation was prepared. Numerous fields, including parametric analysis, design (via trial and error), and visualization, are seen to have promise in this respect. The use of computer simulations and the interactivity they enable is widely recognized as a powerful tool for fostering better student-teacher interactions and, by extension, deeper learning..

References

The NASA Computational Aerosciences Program— Toward Teraflops Computing. Holst, T. L., M. D. Salas, and R. W. Claus. Jan. 1992, AIAA Document No. 92-0558.

Second, Chapman (D. R.): Computational Aerodynamics Development and Outlook. Dec. 1979 issue of the AIAA Journal, on pages 1293–1313.

3, Ballhaus, W. F., Jr.: Computational Aerodynamics and Supercomputers. The COMPCON 84 Conference Proceedings were published by the IEEE Computer Society in Silver Spring, Maryland, in February and March 1984.

Four: Kutler, P., J. L. Steger, and F. R. Bailey, The Current State of Computational Fluid Dynamics in the United States. The June 1987 issue of AIAA Paper 87-1135-CP covers this topic.

5-Hoist, T. L., and Thomas, S. D., Numerical Solution of Transonic Wing Flowfields. June 1983 issue of the AIAA Journal, on pages 863–870.

Numerical Calculation of Transonic Potential Flow around Wing–Body Combinations. Caughey, D. A., and A. Jameson. In the February 1979 issue of the AIAA Journal, pages 175–181 were devoted to this topic.

7 The Pressure Distributions of Schnitt, V., and Charpin, F.

Navier—Stokes Prediction of the Flow Field Around the F-18 (HARV) Wing and Fuselage at Large Incidence. Cummings, R. M.; Rizk, Y. M.; Schiff, L. B.; and Chaderjian, N. M. The ONERA M6 Wing at High Mach Numbers: AIAA Document 90-0099, January 1990. May 1979 AGARD Report AR-138.

Numerical Optimisation Design of Advanced Transonic Wing Configurations. By G. B. Cosentino and T. L. Holst. March issue of the AIAA Journal, volume 23 issue 3