_1995122304_

REQUIREMENTS FOR EFFECTIVE USE OF CFD IN AEROSPACE DESIGN

Pradeep Raj*
Lockheed Aeronautical Systems Company
Marietta, Georgia 30063-0685

SUMMARY

This paper presents a perspective on the requirements that Computational Fluid Dynamics (CFD) technology must meet for its effective use in aerospace design. General observations are made on current aerospace design practices and deficiencies are noted that must be rectified for the U.S. aerospace industry to maintain its leadership position in the global marketplace. In order to rectify deficiencies, industry is transitioning to an integrated product and process development (IPPD) environment and design processes are undergoing radical changes. The role of CFD in producing data that design teams need to support flight vehicle development is briefly discussed. An overview of the current state of the art in CFD is given to provide an assessment of strengths and weaknesses of the variety of methods currently available, or under development, to produce aerodynamic data. Effectiveness requirements are examined from a customer/supplier view point with design team as customer and CFD practitioner as supplier. Partnership between the design team and CFD team is identified as an essential requirement for effective use of CFD. Rapid turnaround, reliable accuracy, and affordability are offered as three key requirements that CFD community must address if CFD is to play its rightful role in supporting the IPPD design environment needed to produce high quality yet affordable designs.

INTRODUCTION

Over the last thirty years, we have seen a phenomenal growth in speed and memory of digital computers with estimates ranging from three to four or more orders of magnitude. Scientists and engineers have successfully exploited this growth to significantly advance the frontiers of science and technology. In the present context, advances in our ability to model complex flow fields are of most interest. The corresponding enabling technology, widely known as computational fluid dynamics or CFD, is now an integral part of all science and engineering disciplines where fluid dynamic interactions play an important role. From a scientific view point, the critical importance of CFD is obvious from the role it is playing in providing a better understanding of the more complex flow physics in general and turbulence in particular (ref. 1, 2). From the engineering vantage point, CFD holds considerable promise to revolutionize the design of flight vehicles, automobiles, turbomachinery, etc., provided that its potential is successfully harnessed. Even a cursory glance at the ever growing list of technical publications documenting CFD applications should be enough to convince even skeptics of CFD's potential. It is impractical to include an exhaustive list of CFD publications in this paper; interested readers should consult Reference 3 for a representative sampling of the variety and complexity of geometries and flow fields that can be modeled using modern CFD techniques. However, it would be a mistake to consider the number and volume of publications on CFD as a testimony to its effective use in the aerospace design environment. With this basic premise, the present paper examines the issue of CFD effectiveness in aerospace design and identifies some of the key requirements that CFD must meet in order to be fully effective. The outcome of any examination is generally a function of the examiner's level of knowledge, past experiences and personal biases. The reader should be forewarned that the present effort is also subject to the same influences.

The remainder of the paper is organized along the following lines. The section on Design Process and Role of CFD immediately follows this Introduction section and contains author's observations on the general nature of the aerospace design process and where CFD fits in. Both conventional design practices and transformations taking place to accommodate the emerging integrated product and process development environment are considered. The following section provides an overview of the current state of the art in CFD and the direction it appears to be heading. Requirements for effective use of CFD are discussed in the next section. The paper then concludes with a few summary observations in the Concluding Remarks section.

---

*Technical Fellow

DESIGN PROCESS AND ROLE OF CFD

In this section, some general observations are presented about current trends in aerospace design. What follows is not a comprehensive discussion of all relevant issues; such a detailed discussion is probably outside the scope of the paper and certainly beyond the limited abilities of the author. Instead, observations are presented mainly to help set the stage for discussing the role of CFD and partly for the sake of completeness. While the author fully recognizes subtle and sometimes not so subtle differences among the design processes of individual companies, it is hoped that what follows will faithfully represent important aspects of the current trends at a majority of companies and thereby provide a valid basis for the rest of the discussion. Readers are strongly encouraged to read many interesting and thought-provoking articles that have appeared in literature over the past few years including but not limited to a survey paper by Miranda (ref. 4) on application of CFD to airplane design, the Lanchester Memorial Lecture by Hancock (ref. 5) on the role of computer in aerodynamics, a paper by Miranda (ref. 6) on challenges and opportunities for CFD in fighter design, a paper by Cosner (ref. 7) on issues in aerospace application of CFD, and the Wright Brothers Lecture by Rubbert (ref. 8) on the role of CFD in the changing world of airplane design.

Conventional Design Practices

A schematic of the aircraft design process, shown in Figure 1, forms the basis for general observations about the conventional design practices. The process is divided into three phases, (1) Conceptual, (2) Preliminary, and (3) Production, that are carried out in sequence. In the conceptual phase, a set of candidate configurations is defined that is expected to meet customer specifications and requirements. Following trade-off studies using estimates of performance, weight, cost, etc., a single configuration is selected for further development. The design typically undergoes numerous modifications during the conceptual and preliminary design phases. The goal is to create an "optimum" design that satisfies all customer requirements. In the production design phase, the final layout and more extensive validation are carried out prior to releasing the design for manufacturing. In each design phase, the myriad of activities that take place can be broadly placed into synthesis or analysis categories. Synthesis covers defining, refining, and altering concepts and configurations; analysis encompasses methods, tools and expertise to produce data and its use in evaluating concepts and configurations. Their roles are illustrated in Figure 2.

Although highly sophisticated tools and techniques have evolved to support synthesis and analysis activities, completing a design cycle with comprehensive and extensive evaluations of competing concepts and configurations can take several months and many, many labor hours. Most of the time and effort goes into generating data for different disciplines that design teams need to reach design closure. Many times, data from different disciplines create conflicting demands on the direction in which a configuration should be altered. Such conflicts cannot be easily reconciled without the timely availability of accurate quantitative information about the interdisciplinary relations among the design variables. A simple example is that of wing design. The "best" set of geometric parameters obtained from purely aerodynamic considerations may not look so good when structural integrity aspects are taken into account. The real challenge then is to guide the design in a direction that offers the "best" balance between aerodynamic and structural efficiencies. This can be accomplished only through a good understanding of the interrelationship of aerodynamic and structural design variables. Adding more disciplines such as producibility, manufacturability, maintainability, cost, etc., further compound the problem but they have to be taken into account before a design can be finalized. The same basic principles apply to the design of a complete aerospace system of which wing may be just one component. However, the challenge grows nearly exponentially with increasing complexity of the system. At present, procedures for generating quantitative data on interdisciplinary relationships are less than satisfactory at best, and nonexistent at worst. Design team's decisions are therefore highly dependent on the intuition and experience of its members, especially in the early stages of product development.

The large time and effort associated with a complete design cycle limits the number of cycles that can be conducted to explore a wider spectrum of alternatives within schedule and cost. It cannot be overemphasized that schedule and cost constraints are central to all industrial design processes; sometimes they get lost in discussions of advanced technologies. It should also be noted that decisions made in the early stages of design have far-reaching consequences for the life-cycle cost of the final design. It has been variously estimated that 70% to 90% of the life-cycle cost of an airplane is locked in during the conceptual and preliminary design phases. Tools and techniques used in the early phases are typically not as advanced or sophisticated as needed to produce highly accurate and reliable data. As the product evolves over time and data from more detailed investigations come in, design teams face the prospect of either changing the design at the expense of increased cost or retaining a design that may not meet all

customer requirements and specifications.

Transition to IPPD Design Processes

It is obvious from the discussion above that the deficiencies in conventional design processes make it extremely challenging for design teams to produce *high quality* designs at *affordable* prices. The challenge is not either quality or affordability, but both simultaneously. Without successfully meeting this challenge, the U.S aerospace industry will have a difficult time in maintaining its leadership position in the increasingly competitive market place of the '90s and beyond. To address this concern, the industry and government jointly initiated many studies during the 1980s which led to a widely accepted conclusion that industry must transition to an integrated product and process development (IPPD) environment. IPPD is characterized by integration of all aspects of product development including design, marketing, manufacturing, and product support. The IPPD approach relies on considering *all* requirements and constraints from the start rather than altering a design in its later stages to facilitate manufacturing or accommodate product support needs. Proper trade-offs can therefore be made early and the need for design changes later on is considerably reduced. The result is improved quality and increased productivity of the entire development process.

In the IPPD context, design is viewed as an integrated multidisciplinary process. A key distinguishing feature of the integrated process is that it incorporates fast, accurate and cost-effective means of generating data for each contributing discipline as well as for complex interdisciplinary relationships among design variables. Availability of such data is critical to driving the design in the right direction. The integrated process must not be construed as an "automated design process." It cannot substitute for human creativity and unique synthesis ability. What it can do well is to shorten the design cycle time by expeditiously providing design teams with data needed to make more informed decisions and thereby alleviate the serious shortcomings of conventional design processes. Design teams can then devote more time and effort to considering a broader set of options with attendant improvements in quality and productivity. It must also be noted that the integrated process does not in any significant way differ from the conventional process in *what* activities are actually carried out, the significant difference is in *how*.

Role of CFD

In the opinion of the author, the CFD technology will play a pivotal role in the implementation of the integrated design process and in its eventual success in improving quality and reducing cost of aerospace designs. Why? Because accurate estimation of aerodynamic data is essential to any flight vehicle design. Force and moment data are needed to evaluate performance and flying qualities; surface pressures provide inputs for structural design; and flow-field data facilitate systems integration, such as the integration of propulsion system with airframe. Using wind-tunnels alone to produce the desired aerodynamic data is too costly and time consuming to meet the basic requirements of the integrated process. A judicious mix of wind-tunnels and CFD is already beginning to pay off in design projects; the paper by Bangert et al (ref. 9) on F-22 tactical fighter design being a case in point. With continuing advances in CFD, there is ample reason to believe that an even stronger partnership with wind tunnels will emerge to produce aerodynamic data in a more timely and cost-effective manner.

Additionally, there are two areas where CFD can play an important role because it holds an edge over wind tunnels. First, CFD affords a means of computationally defining and/or refining geometric shapes to produce certain specified flow characteristics while satisfying some prescribed constraints; this is not feasible in a wind tunnel. Second, a combination of CFD and advanced computational methods from other disciplines, such as structures, controls, propulsion, etc., offers the only practical means of generating interdisciplinary relationships among design variables which are a cornerstone of the IPPD design process. However, full benefits of CFD can only be realized if we can use it effectively in the aerospace design processes. Before discussing the requirements for effectiveness, the current state of the art in CFD is briefly reviewed in the next section.

CFD STATE OF THE ART

A variety of CFD codes are presently available to generate aerodynamic data for a given design. The codes can be broadly categorized into four levels shown in Figure 3. Basic characteristics of each level of codes are highlighted in this section. The categorization into four levels is based on a number of factors including the timeframe

of introduction of the methodology to the flight-vehicle design environment, the nature of mathematical formulation, and capabilities of the codes. The lowest level codes, introduced back in the mid to late 1960s, are now widely used and accepted; the highest-level codes, introduced more recently, are still struggling to find their place. It has long been known that the Level IV codes, based on the Navier-Stokes (N-S) equations, can in principle simulate nearly all flow phenomena of interest to aerospace community for which the continuum assumption is valid. (The Boltzmann equations based on the kinetic theory of gases need to be solved for modeling molecular flows; the related numerical techniques will not be covered in this overview.) However, adequate computer power and efficient numerical algorithms to solve the N-S equations were not available in the 1960s. This forced researchers to explore alternatives based on inviscid approximations to the N-S equations; the first three levels correspond to codes based on a hierarchy of inviscid approximations. Different mathematical formulations largely dictate the capabilities and limitations of the codes in modeling flow physics, and the associated numerical formulations have a strong bearing on the usability and applicability of the codes. In comparing the capabilities of different levels of codes, the focus is deliberately on complex geometries as they dominate the aerospace design landscape.

Level I: Linear Potential Codes

The linear potential codes are based on the Prandtl-Glauert or Laplace equations which form the lowest level of inviscid approximation to the N-S equations. Most of the codes employ the boundary integral approach to solve the governing partial differential equations (PDEs). The equations along with the boundary conditions are cast in a surface-integral form using Green's theorem. The solution is constructed by discretizing the geometry into small elements and assigning a type of singularity (sources, doublets, or vortex filaments) to each element. The singularity strengths are determined by satisfying the no-normal-flow condition at a control point on each element. Depending upon the approximations used in surface discretization (mean surface or actual surface) and the type and functional form of singularities (constant source, constant doublet, linear doublet, etc.), codes with different characteristics (ref. 4) can be developed. The simplest codes, widely known as vortex-lattice methods, employ mean-surface representation of geometry and vortex-filament singularities, e.g., the VORLAX code (ref. 10). When the actual surface geometry is used, the methods are commonly referred to as panel methods. Low-order singularity distributions, constant on each element, have been employed in the QUADPAN code (ref. 11) and higher-order distributions, linear or quadratic functions, in the PANAIR code (ref. 12).

Although the simplicity of mathematical formulation of the linear potential codes inherently restricts their validity to purely subsonic and supersonic attached flows, they are quite extensively used in design efforts due to the ease of use, computational efficiency, and a high level of confidence built upon years of use. An experienced user can set up a computational model in a matter of hours even for relatively complex configurations like a complete aircraft. The computational times are small ranging from a few seconds on supercomputers to a few minutes on workstations. However, user expertise is crucial to ensure that results are correctly interpreted. The vortex-lattice methods and panel methods generally provide good estimates of lift, induced drag, moment coefficients and pressures for steady flight conditions. This data usually form the basis for performance and weight estimations in the early stages of design. Some of the codes also offer a design option that can be used to determine geometric characteristics (like twist and camber of a wing) for a prescribed set of aerodynamic parameters. To meet the aerodynamic data needs of the aeroelastic and flutter disciplines, versions of doublet-lattice method (ref. 13) are the codes of choice. The linear potential codes were first introduced into the aircraft design environment in the late 1960s and the entire class of codes reached a high level of maturity in the early '80s. With the possible exception of the oscillatory aerodynamic codes (ref. 14, 15), very little effort is presently going into research and development of this level of codes.

Level II: Nonlinear Potential Codes

The nonlinear potential methods are based on either transonic small perturbation (TSP) equations or full-potential equations (FPE). Their ability to model transonic flows with shocks is the most significant benefit over the Level I codes. However, this benefit comes at the expense of added complexity stemming from the need to resort to a field approach to solve the nonlinear PDEs. The field approach requires that a region of the flow field surrounding a given configuration be divided into small elementary volumes; it is no longer enough to just divide the surface. In practice, TSP codes are easier to use than FPE codes, especially for complex geometries, because of the differences in boundary condition treatment. The TSP approach permits a simplified treatment based on the application of the no-normal-flow condition at a mean surface. In contrast, the FPE approach requires application to the actual surface. Consequently, Cartesian field-grid systems suffice for a TSP code whereas the FPE codes need boundary-conforming

grids. Cartesian grids are considerably easier to set up as compared to the boundary-conforming grids. Of course, the TSP codes suffer from limitations on the class of geometries and flow conditions that they can model accurately—a direct result of their simplified boundary-condition treatment.

Considerable progress was made throughout the 1970s towards developing a practical transonic-flow analysis capability based on FPE and TSP approaches. For steady-flow computations, the TSP code of Boppe (ref. 16) and the FLO-series of FPE codes of Jameson and Caughey (ref. 17) are representative examples. The promise and excitement of the newly-found ability of computing transonic flows were so strong that even wing design procedures (ref. 18) were developed while the analysis methods were still evolving. Reference 19 is a good source of additional details of progress made during the '70s. Since transonic flows are particularly susceptible to viscous effects associated with shock/boundary-layer interaction, considerable research was also done in coupling inviscid TSP and FPE codes with boundary-layer codes. In addition, an aeroelastic analysis capability based on the TSP formulation (ref. 20) was developed.

Although the Level II codes offered the much needed capability of modeling transonic flows, they did not find the same level of widespread acceptance as the Level I codes. A variety of factors contributed to this situation including the then level of grid generation technology which was not conducive to applying the codes on a regular basis to anything more complicated than wing or wing-body configurations, and the limited region of flight envelope (transonic cruise) where the codes could produce data of acceptable accuracy. Applications of the codes indicated, as one might have suspected, that solution accuracy deteriorated if the actual flow being modeled contained strong shock waves or large regions of vorticity (e.g., leading-edge vortices). Usefulness of the codes was therefore severely limited. For instance, they could not adequately handle a whole class of aerodynamic problems associated with fighter design. In the author's opinion, nonlinear potential codes were basically taken over by the rapid pace of advances in Euler codes in the early '80s. Development of the TRANAIR code (ref. 21) was an exception to this trend. TRANAIR adopts an unconventional hybrid approach combining the flexibility of panel methods to handle complex geometries with the ability of FPE formulations implemented on Cartesian grids to handle nonlinearities of transonic flows. Considerable success has been reported (ref. 3, chapters 15 and 19) in applying this code to aerospace design problems.

Level III: Euler Codes

The Euler equations, which form the basis of Level III codes, represent the highest-level of inviscid approximation to the N-S equations. By permitting nonisentropic shocks and rotational flows to be part of the solution, Euler codes alleviate the major limitations of potential-flow methods albeit at the cost of additional computational expense. The added expense comes from the need to solve at least four and generally five coupled first-order PDEs instead of one second-order PDE. However, two factors at the dawn of the eighties convinced most researchers to shift their focus to Euler equations. These factors were: (1) projected growth in computer power, and (2) development of more efficient numerical algorithms to solve the Euler equations (ref. 22, 23). In addition, the accelerated pace of boundary-conforming grid generation technology combined with the use of finite-volume concept to decouple flow solvers from grid mappings held considerable promise for realizing CFDers dream of analyzing realistic geometries, such as a complete aircraft, on a regular basis. A synopsis of the impressive progress made so far is presented here; details can be found in many publications including Reference 3 and a recent AGARD report (ref. 24).

Two distinct development paths can be identified for Euler codes: one based on hexahedral structured grids and the other on tetrahedral unstructured grids. During the early part of the eighties, most researchers focused their energies on structured-grid methods whereas the interest shifted considerably towards unstructured-grid methods from mid-eighties onwards. This shift was prompted by the realization that unstructured grids afforded greater flexibility in handling complex geometries and promised to "automate" the grid-generation process. Structured-grid advocates pursued a multiblock strategy to overcome the difficulties encountered in handling complex geometries, and codes based on patched or overset multiblock grids evolved to a high degree of sophistication. In spite of many publications detailing the virtues of one approach over the other and considerable advances in grid-generation techniques, the fact remains that constructing multiblock grids for complex geometries continues to be a labor-intensive and time-consuming task and unstructured-grid generation is not yet sufficiently automated although it certainly requires less time and effort. The recent resurgence in Cartesian-grid methods (ref. 25, 26) offers an attractive alternative because it essentially dispenses with the difficulties of grid generation leading to a considerable reduction in time and effort of applying them.

Two other aspects of Euler code development deserve mention. First, shock-capturing rather than shock-fitting has become the preferred approach. Both upwind and adaptive-dissipation schemes have been employed to a great degree of success on all kinds of grid systems. (Although the battle between the advocates of each scheme rages on, intensity has gone down considerably compared to the early years.) Second, most codes solve time-dependent form of the Euler equations even for modeling steady flows. Convergence acceleration techniques, such as local time step and multigrid, are employed to obtain time-asymptotic steady-state solutions in a computationally efficient manner. Both explicit and implicit time-marching schemes have been effectively utilized. Due to the use of time-dependent equations, modeling of unsteady flows is relatively straightforward, and this aspect has been exploited to develop aeroelastic analysis methods (ref. 27). Recent attempts at developing inverse design (ref. 28) and aerodynamic design optimization (ref. 29) methodologies are also noteworthy; their progress is being carefully watched.

Euler codes give us a powerful tool to analyze configurations of interest throughout the subsonic to hypersonic flight regime. This, combined with their demonstrated ability to automatically capture rotational flow regions (such as wakes shed behind wings and vortices emanating from sharp, highly-swept leading edges of delta wings), requiring no explicit a priori definition of such regions, renders them significantly more useful than the Level I or II codes. They are also beginning to make inroads into supporting the data needs of airplane design but in later stages (ref. 9). However, the implications of neglecting viscosity should be clearly recognized. Whereas the Euler codes are superior to the nonlinear potential codes in modeling strong shocks, their solutions are not necessarily closer to the actual flow which is likely to exhibit the effects of shock-induced separation. Some researchers have combined Euler codes with boundary-layer codes to more accurately model transonic flows on wing and wing-body configurations. The Euler codes also have an edge over potential-flow methods in capturing leading-edge vortices. But the location and strength of the primary vortices may not be accurate in cases where the secondary and/or tertiary vortices exert considerable influence. Also, the codes cannot provide an estimate of total drag (including skin-friction) or model flow separation from smooth surfaces. It is, therefore, not surprising that development of N-S codes has been aggressively pursued in parallel.

Level IV: Navier-Stokes Codes

Navier-Stokes codes have a great deal in common with Euler codes. In practice, a single code usually serves the need of solving both Euler and N-S equations. This follows directly from the similarities between the two sets of equations. Elimination of diffusion terms readily converts the N-S equations to the Euler equations; they both share a common set of convective terms. However, the practical implications of this seemingly minor difference are enormous. For example, size of the computational model grows considerably due to the need of accurately resolving the diffusion terms which require highly clustered grids close to solid surfaces (as well as in other regions where viscous stresses are large). This has a bearing on grid generation, numerical algorithms, and computational resources. With appropriate grid clustering, we can solve the N-S equations to simulate laminar flows in a relatively straightforward fashion. But using these equations to directly model even simple turbulent flows stretches the current supercomputers to their limits. At present, the Reynolds-averaged Navier-Stokes (RANS) equations are used almost exclusively to simulate complex turbulent flows. For a large majority of problems, the thin-layer approximation to the RANS equations is employed to reduce the problem to a manageable size. But these simplifications impose a heavy toll; we now require a turbulence model!

A variety of turbulence models have emerged in recent years ranging from relatively simple algebraic models to more sophisticated Reynolds-stress models. They have been implemented into various codes. Impressive results have been obtained using multiblock structured-grid methods, both patched (ref. 30) and overset (ref. 31). Unstructured-grid techniques are also advancing at an accelerated pace (ref. 32). Many competing approaches are evolving ranging from tetrahedral grids to hybrid grids (combining prismatic grids in close vicinity of configurations with tetrahedral or Cartesian meshes elsewhere). In general, experiences to date in modeling turbulent flows have produced rather mixed results. There have been many successes in using simple models for relatively complex flows and some failures in using the more sophisticated ones for relatively simple flows. Attempts to refine existing models and develop new, improved ones continue unabated. Considerable research effort is also being devoted to developing models for laminar to turbulent transition, another area of great significance.

While progress is being made on many fronts, CFD practitioners dilemma is quite clear. For the foreseeable future, they will have to use RANS methods for modeling engineering problems of interest in aerospace design. Yet, the accuracy and reliability of the solutions for turbulent flows will continue to be subject to the inadequacies of

turbulence models. The prospects of a universal model are rather bleak; capturing the complex nature of turbulence in its entirety into a model with a few free parameters is a long shot indeed. Nevertheless, Level IV codes will continue to find increasing use in the years to come not because the turbulence and transition modeling difficulties will be fully resolved but to meet specific engineering needs. There are enough problems where viscous effects dominate and they can be properly simulated only by solving the N-S equations. Internal flow problems (inlet, diffusers, nozzles, etc.) and high-lift systems (multi-element wings) are two prime examples. Probably the best rationale for continuing use of Level IV codes, in spite of their limitations, may be taken from Bradshaw (ref. 33):

> "...we cannot calculate all flows of engineering interest to engineering accuracy. However, the best modern methods allow *almost* all flows to be calculated to higher accuracy than the best-informed guess, which means that the methods are genuinely useful even if they cannot replace experiments."

This brief overview of Level IV codes will not be complete without mentioning the recent emergence of Digital Physics™ technology (ref. 34, 35), developed and marketed by Exa Corporation, Cambridge, Massachusetts. Developers have shown preliminary incompressible-flow results on two test cases, backward-facing step and cylinder, for which good correlation with measurements were obtained without any turbulence modeling! The technology is claimed to have a fundamental advantage over conventional RANS CFD codes because it is free from the artifice of discretization. Depending upon the success in extending the technology to compressible flows and additional demonstrations, Digital Physics™ could provide a very attractive means of circumventing the turbulence modeling problem altogether for engineering applications.

REQUIREMENTS FOR EFFECTIVE USE OF CFD

Webster's New Collegiate Dictionary defines effective as "producing a desired effect." In the context of using CFD in an industrial setting, the important questions are: Whose desires? What is desired? It is instructive to look at the whole issue from a customer/supplier viewpoint. The customer in our case is the design team and suppliers are the CFD practitioners. Then, CFD use can be considered effective only if the desires and expectations of the design team are met. (Note that your use of CFD is effective if the design team calls upon you the next time they have a need!) Design teams need a variety of data ranging from integrated quantities like forces and moments to detailed flow features like shocks and vortices. Their natural desire is to obtain data of highest fidelity within schedule and cost. It follows that CFD use would be effective if the CFD team can produce the highest-fidelity data while meeting the schedule and cost milestones of the design team.

The customer/supplier viewpoint also simplifies the issue of requirements for effective CFD use. It forces the practitioners to look at CFD from the customer's angle—and the view turns out to be quite different. What we then find is that the customer wants good quality aerodynamic data to help him do his job better, at the time that fits his milestones, and at an affordable price; the customer does *not* want CFD per se. For example, engineers involved in performance estimation want drag polars, stability and control engineers want derivatives of force and moment coefficients, structural loads engineers want surface pressures or aerodynamic influence coefficient matrices. To engineers in the design team, the important things are engineering data, schedule, and cost. If CFD can provide them with data they need when they need it, then they consider the use of CFD to be effective. To them, CFD is just another tool. They do not always, nor should they be expected to, understand subtle differences among various techniques upon which the plethora of CFD methods are built. Many times, CFD practitioners have "oversold" CFD to customers without fully appreciating each others point of view. This lack of appreciation has led to rather unpleasant situations when CFD practitioners had difficulty in satisfying customer expectations.

For effective use of CFD in aerospace design environment, the most essential requirement is to have a partnership between the design team and the CFD team. Before embarking on any task, it is crucial for design teams to clearly define their data needs and the associated schedule and cost constraints. CFD teams should then devise appropriate strategies and define a set of feasible options. The two teams should jointly select the option that best fits the needs. Without such a partnership, a design team all by itself is most likely to select an option based on past experiences and least likely to take advantage of new advances in technology. By the same token, CFD teams are likely to resort to their favorite method to address every demand of data without fully analyzing the potential quality, schedule, and cost implications.

Having dispensed with the CFD effectiveness issue from the customer's viewpoint, let us examine the key requirements from the viewpoint of CFD suppliers or code developers. This aspect has received considerable attention in the past with notable contributions by Miranda (ref. 4), Bradley (ref. 3, Chapter 25), Cosner (ref. 7), and Rubbert (ref. 8). The cited articles contain opinions and observations of industry leaders from Lockheed, McDonnell-Douglas and Boeing. Although the articles appeared at different times over the span of over ten years, many common themes run through them. As far as the relationship between CFD effectiveness and code characteristics is concerned, all views can be condensed into the following expression due to Miranda:

effectiveness = quality × acceptance

Here, quality refers to accuracy and realism of the solution, and acceptance includes usability, applicability, and affordability. This expression impresses upon CFD developers the importance of the simple fact that focusing on either quality or acceptance alone is not desirable; our approaches must enhance *both* simultaneously if we wish to increase the overall effectiveness of CFD in an aerospace design environment. The basic premise of this expression is as true today as it was when originally proposed. Combining this expression with the role of CFD in the aerospace design process discussed earlier, the author proposes three key requirements that CFD must meet in order to be effective in an aerospace design environment: rapid turnaround, reliable accuracy, and affordability. A word of caution is in order before we discuss each of them. All three must all be considered *together* and not separately in evaluating the effectiveness of a particular CFD methodology.

Rapid Turnaround

The first and foremost requirement is rapid turnaround. "Minimizing calendar time" is identified by Bangert et al (ref. 9) as one of the primary requirements based upon their assessment of CFD applications to F-22 design. Turnaround is meant to cover the entire time it takes from the initial go-ahead to the final delivery of data to the customer. A typical CFD application process requires three steps: (1) Pre-processing or acquisition of geometry and setting up of a suitable computational model, (2) Running a flow solver, and (3) Post-processing or extraction of desired aerodynamic quantities by processing the flow-solver output and delivery of data to the customer. In order to reduce the total turnaround time, each step must be carried out with utmost efficiency. Of course, the level of the selected CFD code, i.e., Level I or Level IV, has a strong influence on turnaround time; the lower-level codes offer quick turnaround and the higher-level codes take longer. This is obviously not a desirable state of affairs when we compare the regions of relevance among different levels of CFD in a simplified two-parameter design space as shown in Figure 4. The large extent of the Euler and N-S regions clearly points to the potential of significant payoffs if their turnaround time can be made comparable to that of the lower-level codes.

At present, the higher-level codes are considerably slower than the lower-level codes in producing aerodynamic data. Days of geometry acquisition, weeks of grid-generation, hours of execution time on supercomputers, and days of time-consuming and labor-intensive postprocessing, all contribute to the present situation. Pre- and post-processing steps are the primary culprits when only few analyses are conducted using a single model. If a large number of runs are made on a single model, the total amount of computer time (in wall-clock hours or elapsed time) can be substantial and may even overwhelm the pre- and post-processing times. The most challenging situation arises when the configuration geometry undergoes changes and multiple analyses have to be performed for each variation. However, that is precisely what the IPPD design environment demands of CFD! An integrated design process that significantly reduces design-cycle time depends on methods that are fast. The current higher-level CFD methods are just not up to the challenge, except perhaps for component-level design for some limited region of the flight envelope. The challenge for the CFD community is clear: develop appropriate technologies and integrate them in a manner that brings the turnaround time for each analysis to a matter of minutes. The list of potential enabling technologies includes: streamlined interfaces to computer-aided design (CAD) systems based on standard data-exchange protocols; nearly automated grid generation; parallel processing of flow solver software; intelligent systems for data analysis and management, to name a few. Ongoing research and development, some of it reported in the proceedings of this workshop, gives considerable hope and encouragement to CFD practitioners that the target is achievable.

Reliable Accuracy

Although reducing turnaround time is crucial, producing data of reliable accuracy is of equal importance. A solution of reliable accuracy is one that comes with a known and acceptable error band on all quantities of interest to the customer. As pointed out by Bangert et al (ref. 9), F-22 design team relies primarily on wind-tunnel data due to

the limitations of current CFD codes in modeling viscous effects, especially when applied to complex geometries and very large speed, altitude, and maneuver envelope. An interesting thing to note here is that wind tunnels are not necessarily the best tools to generate desired aerodynamic data. They have limitations of their own such as support and wall interference effects, scale effects, etc. But design teams have built-in confidence in data coming from tunnels which have been used for almost as long as aeronautics has been around. Wind-tunnel test teams know their customer as well as understand the limitations of their tool. They have developed elaborate procedures to compensate for most, if not all, sources of error in data. In contrast, CFD methods do not have the same limitations as wind tunnels; flow analysis can, in principle, be conducted for arbitrary flight conditions. But, in practice, CFD teams have a good deal of difficulty in attesting to the reliability of their data. The situation must be rectified because without reliable accuracy of CFD predictions, producing "optimum" designs in an IPPD environment will remain an elusive goal.

Accuracy of computed solutions has two components: numerical and physical. A solution may be considered accurate in a numerical sense if it shows little or no sensitivity to changes in grids as well as other numerical parameters related to the algorithm. (It is assumed that the code in question has been verified as to the adequacy of its numerical formulation in solving the governing equations.) At present, there are few, if any, *practical* means of estimating the effect of grid resolution, truncation error, numerical parameters such as dissipation and dispersion, etc. Schedule and cost constraints of a typical design effort do not permit extensive investigations to determine the optimal grids and parameters. CFD teams usually rely upon previous experience and expertise but the situation is not totally satisfactory. What is really needed is built-in means of quantifying the level of accuracy. The problem is admittedly difficult but a solution is urgently needed if CFD is to be utilized effectively in the IPPD environment. In combining CFD with methods from other disciplines to produce interdisciplinary relationships among design variables, an assessment of the level of accuracy and associated error bounds of the solutions is even more critical. Incorporation of solution-adaptive techniques based on truncation error and/or numerical dissipation is one possible approach to address the problem of estimating as well as minimizing numerical errors. Some of the approaches, such as unstructured grids or Cartesian grids, are inherently more suitable to addressing this aspect.

Even if a code produces a numerically accurate solution on a given model, it is not trivial to determine how well the solution stacks up against the *real* flow—a measure of the physical accuracy. Keep in mind that when CFD is used in a predictive mode in a design environment, CFD teams do not have the luxury of comparing results with other data to determine the level of accuracy! To date, CFD community has advocated and conducted extensive "validation" exercises to generate correlations that can be used to substantiate claimed levels of physical accuracy. In practice, we have been able to barely "calibrate" the codes for specific applications of interest. (See ref. 3, chapter 25 for definitions of validation and calibration.) Why? Because major difficulties arise in planning a comprehensive validation effort. For example, how many test cases, what combination of flow conditions for each test case, and what range of values for each condition must we consider before a code can be declared as fully validated? A matrix of runs using a reasonable set of test cases and conditions quickly grows into a monumental task. Even if we assume that adequate resources as well as measured data are available for carrying out such a task, we run against the tide of technology dynamics. Rapid pace of advances in hardware, numerical algorithms, and models of turbulence and transition fosters an environment where codes are never quite "finished." Sometimes the changes are nominal, many times not. Cost/benefit assessment of any plan of allocating huge resources to validate a code that might be superseded the next day by a "new and improved" method does not support the validation route. CFDers inevitably fall back upon calibration to meet the immediate needs for a class of problems of greatest interest. This situation is likely to persist as long as we rely on RANS codes that require turbulence and transition models. For most applications, a judicious mix of CFD and wind tunnels will be the most effective strategy. Experience shows that a properly calibrated code can go a long way in enhancing the overall cost-effectiveness of CFD in aerospace design.

Affordability

The third and final requirement is that of affordability. Costs associated with CFD use include both labor and computing expenses. At present, labor expenses are mainly connected with pre- and post-processing steps. For higher-level CFD codes, the labor expenses are still beyond the acceptable range. For example, the use of structured-grid methods requires several person-weeks of pre-processing effort whereas a desirable value is closer to a few person-hours. Unstructured-grid (tetrahedral and Cartesian) methods appear to be quite promising in reducing the level of effort. Progress in developing streamlined interfaces between grid-generation methods and CAD systems is crucial to reducing the geometry acquisition time. These improvements will also help in evaluating design changes in an inexpensive manner. As a matter of fact, technologies needed to reduce labor hours are essentially identical to those mentioned earlier for reducing turnaround time.

Computing costs mainly relate to running the flow solver and may include grid generation in some instances. We typically need computers with high processing speeds and large memory. Without access to such machines, it is very difficult to produce the desired set of data on schedule. A typical design cycle can require hundreds of runs before a sufficient amount of data is generated. Since shortening the design cycle is one of the key objectives of the design teams, data must be generated over a matter of days and not months. Computing expenses to generate the desired data in this kind of timeframe must not be so large that the total product development cost will actually increase rather than decrease. Consequently, cost and computational efficiency of the entire hardware and software system are very important considerations for effective use of CFD. Strategies to increasing computational efficiency and reducing cost must be an integral part of all CFD development and applications planning.

CONCLUDING REMARKS

CFD is a key enabling technology for the successful implementation of an IPPD environment needed for producing high quality yet affordable designs. Key requirements identified in the previous section must be addressed if CFD is to play its rightful role in the integrated multidisciplinary design process that is part of the emerging IPPD environment. The three key requirements are: *rapid turnaround, reliable accuracy,* and *affordability.* Unless they are met, the technology will not get *fully* incorporated into the industry design processes. Considering CFD as a tool—a means to an end—is necessary to evaluating and selecting the "right" technologies for building future capabilities. We must take a system-level approach to CFD; increasing the effectiveness of the overall CFD application process is more important to realizing the full benefits of CFD than enhancing the state of the art in some selected constituent elements. For example, development of a faster flow-solver will have the desired payoffs only if the associated pre- and post-processing tools are also speeded-up to permit a significant reduction in the overall turnaround time.

The challenge facing the CFD community today is to channel their efforts and resources in a manner that makes CFD fully responsive to the design needs. Numerous benefits will accrue from incorporating advanced CFD methods into the design processes. Using CFD methods that offer rapid turnaround capability will reduce design cycle time. Design teams can then explore a wider spectrum of alternatives within the schedule and cost constraints of a typical product development effort than is currently feasible. *Fast, accurate* and *affordable methods* will increase the productivity of the design process and reduce the number of expensive tests needed to support design data needs. The use of advanced methods may also reduce the number of cycles required for design closure. Design teams will be able to conduct extensive trade-offs needed to guide the evolution of a configuration in a direction that minimizes both acquisition and life-cycle costs. Improved understanding of component interactions will permit design changes to be made early and thereby reduce risk and increase the probability of meeting all customer requirements.

REFERENCES

1. Landahl, M., "CFD and Turbulence," ICAS-90-0.1, The Daniel & Florence Guggenheim Memorial Lecture, 17th Congress of the International Council of the Aeronautical Sciences, Stockholm, Sweden, September 9-14, 1990.

2. Rai, M., Gatski, T., and Erlbacher, G., "Direct Simulation of Spatially Evolving Compressible Turbulent Boundary Layers, " AIAA-95-0583, 33rd Aerospace Sciences Meeting & Exhibit, Reno, Nevada, January 9-12, 1995.

3. "Applied Computational Aerodynamics," *Progress in Astronautics and Aeronautics,* Vol. 125, American Institute of Aeronautics and Astronautics, Washington DC, 1990, Henne, P.A. (ed.).

4. Miranda, L.R., "Application of Computational Aerodynamics to Airplane Design," *Journal of Aircraft,* Vol. 21, No. 6, 1984, pp 355-370.

5. Hancock, G.J., "Aerodynamics—the role of the computer," The 25th Lanchester Memorial Lecture, *Aeronautical Journal,* August/September 1985, pp. 269-279.

6. Miranda, L.R., "Transonics and Fighter Aircraft: Challenges and Opportunities for CFD," NASA CP 3020, Vol. I, Part 1, Transonic Symposium: Theory, Application, and Experiment, April 19-21, 1988, pp. 153-173.

7. Cosner, R.R., "Issues in Aerospace Applications of CFD Analysis," AIAA Paper 94-0464, 32nd Aerospace Sciences Meeting & Exhibit, Reno, Nevada, January 10-13, 1994.

8. Rubbert, P.E., "CFD and the Changing World of Airplane Design," ICAS-94-0.2, The Wright Brothers Lecture, 19th Congress of the International Council of the Aeronautical Sciences, Anaheim, California, USA, 18-23 September, 1994.

9. Bangert, L.H., Johnston, C.E., and Schoop, M.J., "CFD Applications in F-22 Design," AIAA Paper 93-3055, AIAA 24th Fluid Dynamics Conference, Orlando, Florida, July 6-9, 1993.

10. Miranda, L.R., Elliott, R.D., and Baker, W.M., "A Generalized Vortex Lattice Method for Subsonic and Supersonic Flow Applications," NASA-CR 2865, December 1977.

11. Youngren, H.H., Bouchard, E.E., Coopersmith, R.M., and Miranda, L.R.,"Comparison of Panel Method Formulations and Its Influence on the Development of QUADPAN, an Advanced Low-Order Panel Method," AIAA Paper 83-1827, Applied Aerodynamics Conference, Danvers, Massachusetts, July 1983.

12. Magnus, A.E., Ehlers, F.E., and Epton, M.A., "PANAIR - A Computer Program for Predicting Subsonic or Supersonic Linear Potential Flow About Arbitrary Configurations Using a Higher-Order Panel Method," NASA CR-3251, April 1980.

13. Giesing, J.P., Kalman, T.P., and Rodden, W.P., "Subsonic Steady and Oscillatory Aerodynamics for Multiple Interfering Wings and Bodies," *Journal of Aircraft*, Vol. 9, No. 10, 1972, pp 693-702.

14. Chen, P.C., Lee, H.W., and Liu, D.D., "Unsteady Subsonic Aerodynamics for Bodies and Wings with External Stores Including Wake Effects," *Journal of Aircraft*, Vol. 30, No. 5, 1993, pp 618-628.

15. Chen, P.C. and Liu, D.D., "Unsteady Supersonic Computations of Arbitrary Wing-Body Configurations Including External Stores," *Journal of Aircraft*, Vol. 27, No. 2, 1990, pp. 108-116.

16. Boppe, C., "Transonic Flowfield Analysis for Wing Fuselage Configurations," NASA CR 3243, 1980.

17. Caughey, D.A., and Jameson, A., "Progress in Finite-Volume Calculations for Wing-Fuselage Combinations," *AIAA Journal*, Vol. 18, No. 11, 1980, pp 1281-1288.

18. Hicks, R.M. and Henne, P.A., "Wing Design by Numerical Optimization," AIAA Paper 77-1247, 1977.

19. "Transonic Aerodynamics," *Progress in Astronautics and Aeronautics*, Vol. 81, American Institute of Aeronautics and Astronautics, Washington DC, 1982, Nixon, D. (ed.)

20. Borland, C.J. and Rizetta, D.P., "Transonic Unsteady Aerodynamics for Aeroelastic Applications, Vol. I—Technical Development Summary for XTRANS3S," AFWAL-TR-80-3107, 1982.

21. Samant, S.S., Bussoletti, J.E., Johnson, F.T., Burkhart, R.H., Everson, B.L., Melvin, R.G., Young, D.P., Erickson, L.L., Madson, M.D., and Woo, A.C., "TRANAIR: A Computer Code for Transonic Analysis of Arbitrary Configuration," AIAA Paper 87-0034, Aerospace Sciences Meeting, Reno, Nevada, January 1987.

22. Jameson, A., Schmidt, W. and Turkel, E., "Numerical Solution of the Euler Equations by Finite-Volume Methods Using Runge-Kutta Time-Stepping Schemes," AIAA Paper 81-1259, 1981.

23. Rizzi, A., "Damped Euler Equation Method to Compute Transonic Flow About Wing-Body Combinations," *AIAA Journal*, Vol. 20, No. 10, pp 1321-1328.

24. "Computational Aerodynamics Based on the Euler Equations," AGARD-AG-325, September 1994, Sloof, J.W. and Schmidt, W. (eds.)

25. Tidd, D.M., Strash, D.J., Epstein, B., Luntz, A., Nachshon, A., and Rubin, T., "Application of an Efficient 3-D Multigrid Euler Method (MGAERO) to Complete Aircraft Configurations," AIAA Paper 91-3236-CP, 1991.

26. Karman, S., "SPLITFLOW: A 3D Unstructured Cartesian/Prismatic Grid CFD Code for Complex Geometries," AIAA Paper 95-0343, 33rd Aerospace Sciences Meeting, Reno, Nevada, January 1995.

27. Guruswamy, G.P., "Unsteady Aerodynamics and Aeroelastic Calculations for Wings Using Euler Equations," *AIAA Journal*, Vol. 28, 1990, pp. 461-469.

28. Campbell, R.L. and Smith, L.A., "A Hybrid Algorithm for Transonic Airfoil and Wing Design," AIAA Paper 87-2552, 1987.

29. Reuther, J.J., Cliff, S.E., Hicks, R.M., and van Dam, C.P., "Practical Design Optimization of Wing/Body Configurations Using Euler Equation," AIAA Paper 92-2633, 1992.

30. Ghaffari, F., "Navier-Stokes, Flight, and Wind Tunnel Flow Analysis for the F/A-18 Aircraft," NASA Technical Paper 3478, December 1994.

31. Gomez, R.J. and Ma, E.C., "Validation of a Large Scale Chimera Grid System for the Space Shuttle Launch Vehicle," AIAA Paper 94-1859, CP 946, 1994.

32. Venkatakrishnan, V., "A Perspective on Unstructured Grid Flow Solvers," AIAA Paper 95-0667, 1995.

33. Bradshaw, P., "Turbulent Secondary Flows," *Annual Review of Fluid Mechanics*, Vol. 19, 1987, pp. 53-74.

34. "Digital Physics," in *Machine Design*, Vol. 66, No. 23, December 1994, pp. 96-100.

35. Teixeira, C., "Continuum Limit of Lattice Gas Fluid Dynamics," PhD. Thesis, Massachusetts Institute of Technology, Cambridge, MA, September 1992.
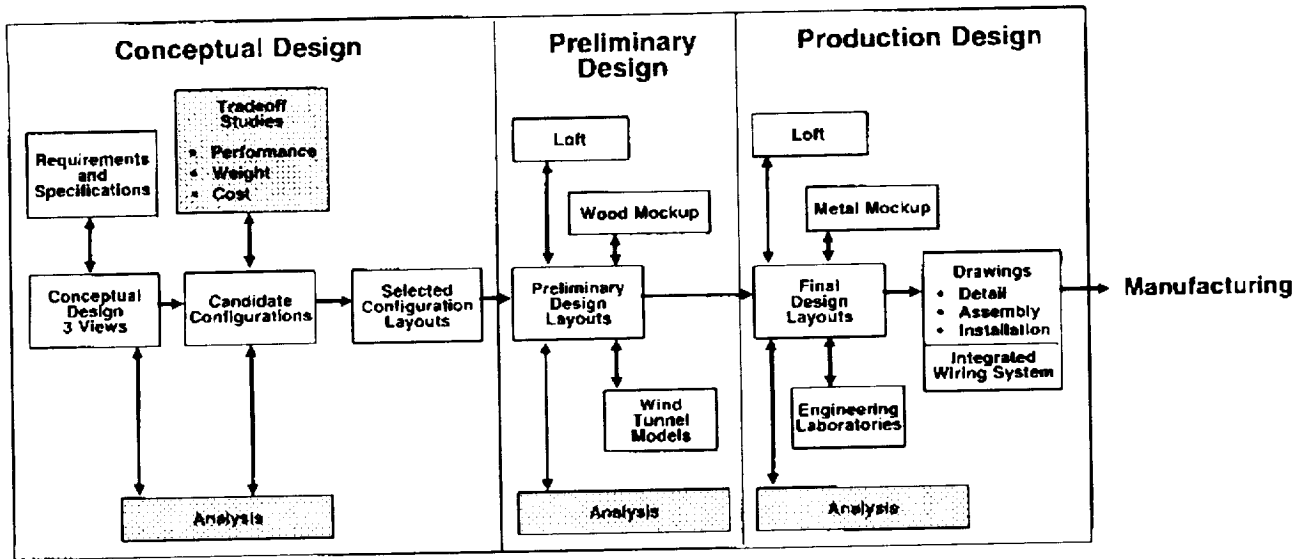
/

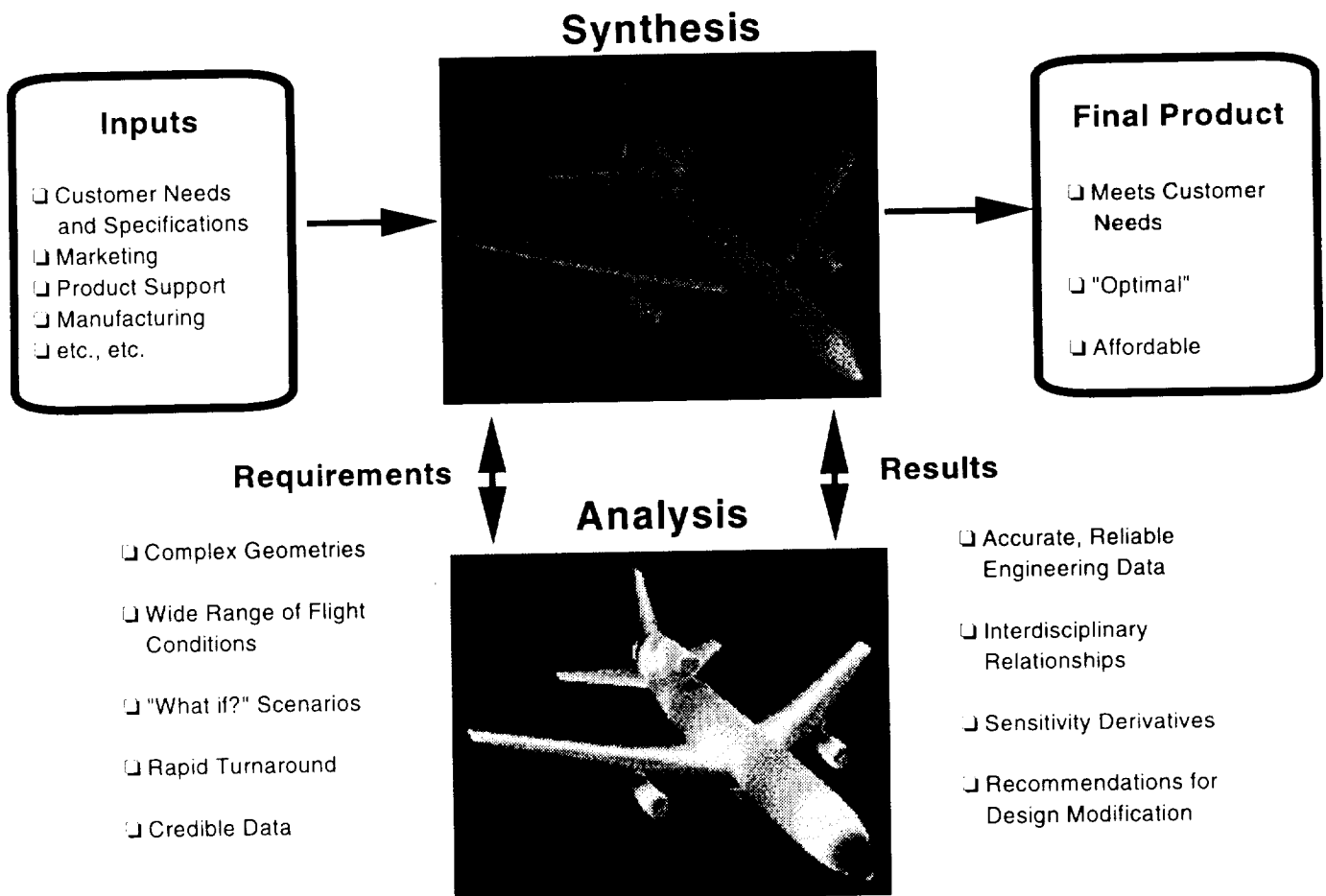**Figure 1.—Schematic of conventional aircraft design process**

Figure 2.—Role of synthesis and analysis in aerospace product development
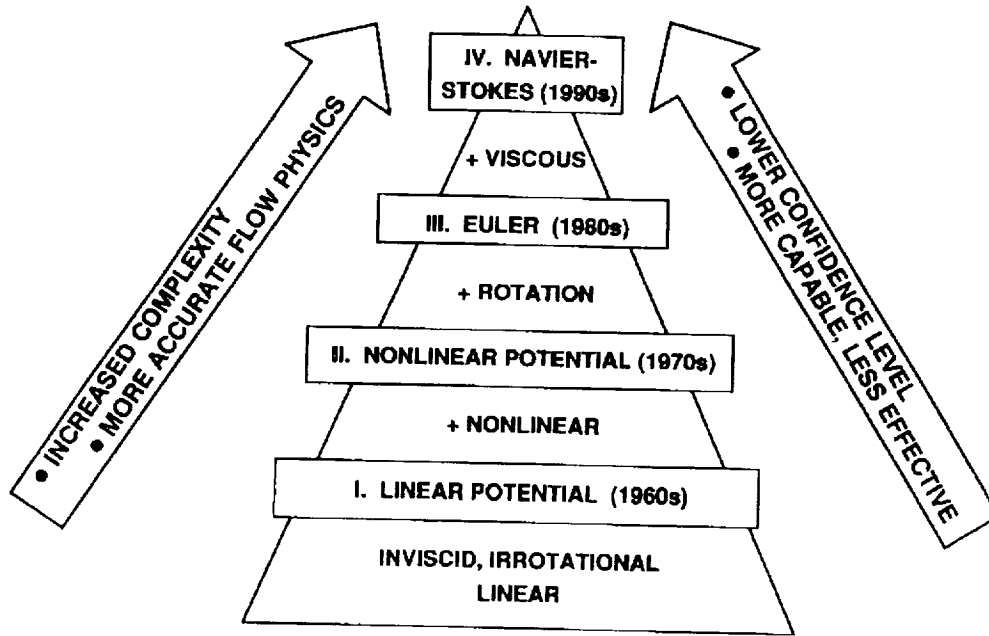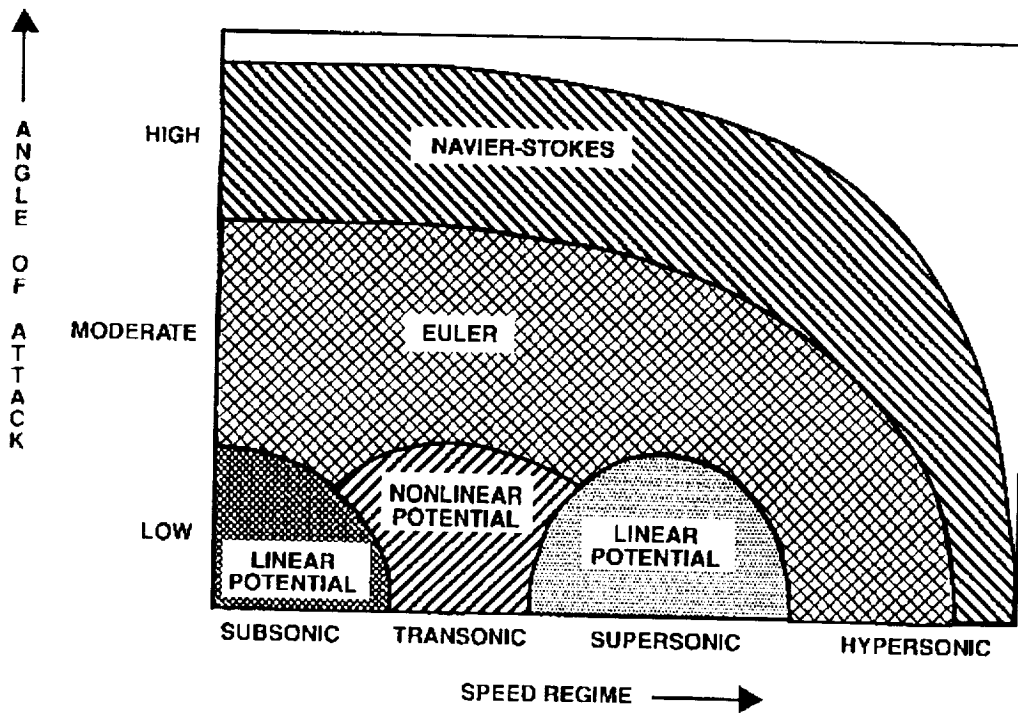
Figure 3.—Four levels of CFD

Figure 4.—Regions of relevant CFD levels in a two-parameter design space