Utilization of Computation and Experiment for Airframe Propulsion Integration Development

Douglas L. Bowers and Capt. Carl H. Berdahl
USAF Wright Aeronautical Laboratories (AFWAL/FIMM)
Wright-Patterson AFB, Ohio 45433-6553 USA

ABSTRACT

Computational and experimental engineers must combine efforts to assure efficient use of both computational and experimental resources during future aircraft development. The objective of this cooperative approach is to develop the best possible integration of the propulsion system into the airframe and thereby reduce the enormous risk of committing limited resources to build an unproven design. While a few excellent examples of the joint application of computation and experiment to propulsion integration can be found, a conscious directed effort will be required to satisfy future design problems. The experiment must be used to assist the computational development and the computation must be used to assist experimental accuracy and efficiency. Other factors which currently block cooperative progress such as a lack of code user friendliness and weak management committment to computations must also be removed. Both the computationalist and the experimentalist should seek to understand the physics of the propulsion component flowfields. Credible progress toward designing optimum propulsion system integrations with the most efficient use of resources will only be achieved through cooperative efforts between both disciplines.

SYMBOLS

A_o/A_i	inlet capture area ratio
$C_{D_{ADD}}$	additive drag
$R_{NOSE(in)}$	scramjet inlet sphere-cone nose radius (inches)

I Introduction - The Nature of the Airframe Propulsion Integration Flowfield.

It is unfortunate that one of the most complex technical areas is also one of the most critical to overall aircraft performance. Airframe/inlet and airframe/nozzle flows (Fig. 1) are characterized by mixed transonic/supersonic regions dominated by inviscid/viscous strong shock interactions, pressure discontinuities, separated regions, temperature variations, and local flow unsteadiness. The physical engine installation is strongly three-dimensional and is often complicated by variable geometry. The complex flowfield and the three-dimensional geometry create technical problems in which the detailed flow physics are often poorly understood. In spite of the difficulties, the installed propulsion system is required to operate with high inlet recovery, nozzle thrust, and low drag for all flight conditions. Transports, for example, have finely tuned performance requirements which are complicated by atmospheric conditions such as water ingestion and icing. For Short Takeoff and Landing (STOL) aircraft, the thrust reverser plume interacts with the external flow over local control surfaces, affecting aircraft stability and control and surface heating and acoustics. For Vertical/Short Takeoff and Landing (V/STOL) aircraft, the installed propulsion system must function over three diverse flight modes, i.e. hover, transition, and cruise. The JVX vertical lift aircraft (Ref. 1) is a prime example. In the hover mode, the nacelle is at high angle-of-attack, the aircraft is at a slow forward speed,and the engine airflow is high. In transition, the nacelle angle-of-attack and forward velocity are changing rapidly and the engine airflow is high. In cruise, the nacelle angle-of-attack is low, the forward velocity is high, and the engine airflow varies from low to high. Added to these diverse flow conditions, the vehicle is required to operate in a sand environment ... a very complex but realistic design problem.

How does the designer satisfy typically diverse performance requirements? Historically, an empirical data base plus additional wind tunnel data were utilized with minimum use of computational tools. Now however, factors beyond the designers control, such as limited resources and the need to be timely, are forcing changes in the distribution of the resources allocated for the wind tunnel and computation, with increasing emphasis now being placed on the computation. The wind tunnel and the computational tools should ideally be used in a complementary manner to provide the best possible propulsion integration design (Fig. 2). For a variety of reasons, including design tool availability and credibility, this is often not consistently done. A conscious cooperative effort to integrate these techniques must be made. In the long term, this effort will enable the designer to successfully satisfy and reduce the risks of the diverse airframe propulsion integration design demands. This paper will present current and potential uses of experimental and computational fluid dynamic (CFD) techniques for airframe propulsion integration (API) in aircraft development. It will also present areas where these technologies must assist each other, other factors which inhibit development of design tools, a look to the future and recommendations to expedite further successes.

II The Use of Experimental and Computational Techniques for Airframe Propulsion Integration during Aircraft Development

The wind tunnel has been and is now the accepted state-of-the-art tool for the evaluation of candidate airframe propulsion system integrations. The very nature of the flow field has mandated use of experiment to simulate the individual component interferences. The amount of data taken in these experiments have been necessarily large to develop the level of confidence required during the development process. The prediction of the basic performance characteristics of the F-15 aircraft (Ref. 2) was based primarily on the data obtained in an extensive wind tunnel test program. Adequate numerical tools to handle the complex configuration geometry and mixed subsonic/supersonic flow were not available. By proposal submittal, 10,000 wind tunnel hours had been completed. By contract award, nearly 13,000 wind tunnel hours had been recorded. An appropriate model in a suitable wind tunnel can provide basic installation effects (e.g. the wing/nacelle interference), can identify by flow visualization areas of complex interactions (e.g. separated regions, shock location, flow direction) in both steady and unsteady flows, and can be used to optimize the propulsion integration for best performance. Together with an accurate bookkeeping system, wind tunnel data can be used to predict values of most forces expected in flight. If the experimenter is willing to accept the complexity of an engine-simulator equipped model, then the simultaneous representation of both inlet and cold nozzle flow is possible. These factors are the impetus for continued high interest in the use and development of improved testing techniques for airframe propulsion integration problems.

The benefits of the experiment are obtained at an enormous cost during aircraft development. Flax (Ref. 3) states that typical major aircraft programs in the 1964-1974 time period incurred development costs of between $1 and $2 billion (of the same magnitude as the manufacturer's net worth). The costs to verify the designs include wind tunnel occupancy time, which have been between 10,000 to 20,000 hours at an average cost of $1500/hr, or $15 to $30 million for each program. Test models and associated hardware add an additional $10 million. This information was presented in 1974 and today's costs are substantially higher. In addition, even extensive wind tunnel testing has historically not avoided the problems that various aircraft development projects later encountered in flight test. The solution of these problems often results in time delays and costly aircraft modifications (Ref. 4). Any means available for the designer to reduce risks in the system design must be utilized. This opens the door for the strong future use of computational fluid dynamics. CFD assistance in aircraft development can be applied over the full spectrum from configuration sorting prior to test, to data analysis assistance during post-test. Computational tools can assist during pretest planning and test execution. Ranges and location of instrumentation can be decided. Precomputed or online computations can be compared with online experimental data to verify data trends and approximate performance and/or data levels to identify testing errors. CFD can be used to extrapolate an existing data base to the applicable design environment. The data base can also be completed by CFD in areas where experimental parameters were not investigated in detail. Tinoco and Chen (Ref. 5) cite a potentially large payoff (millions of dollars) if CFD codes could reliably determine incremental geometry effects before model fabrication and testing. CFD gives the designer the freedom to look at many different design options early and decide if a change is beneficial. The best and most promising changes can then be targeted for always limited resources, resulting in fewer designs going to test, with subsequent overall cost savings. If an aircraft is later re-engined, for example, CFD could potentially assess the performance impacts without a tunnel re-entry. In addition, innovative designs, considered too risky to commit resources for a wind tunnel model and test, could be assessed by CFD.

During post-flight test data analysis, CFD can act as a referee to evaluate discrepancies between the wind tunnel and flight test data avoiding an expensive pursuit of a problem that may not exist. Computational tools can also be used to extrapolate data from wind tunnel Reynolds number conditions to full scale Reynolds number conditions and in this way complement low Reynolds numbers wind tunnel facilities. CFD can provide greater detail of the flowfield than is possible in a wind tunnel since all aerodynamic parameters are computed at every grid point (Ref. 6). This information, displayed by an intensive post-processor with three dimensional color graphics, for example, can permit a review and analysis of data in a reasonable time.

The use of CFD in the process of hardware development is summarized by Barton (Ref. 7) as follows:

1) minimize parameters in model testing

2) reduce risks by permitting more configurations to be evaluated numerically than experimentally

3) remove the constraints of the data base limitations

The potential for the cooperative use of both experiment and computation for airframe propulsion integration during aircraft development is best illustrated by several examples.

Paynter, et. al. (Ref. 1) demonstrates an extensive use of CFD in conjunction with the wind tunnel for installed inlet design and development. CFD was utilized for parametric geometry refinement which was subsequently verified by the wind tunnel. The inlet was designed by Boeing for the V-22 Osprey (JVX), a tilt rotor, vertical lift aircraft (Fig. 3). The engine inlets were required to supply air to the engine with high recovery and low distortion over a wide range of operational conditions (described in Section I) and had to prevent ingested sand from entering the engine. Given the desired stringent performance characteristics, the inlet had to be carefully designed with acceptable compromises. The strategy was to select an inlet concept from preliminary design, refine the concept with the CFD code, and then verify the design by a wind tunnel test.

After preliminary design concept selection, a zonal modeling strategy with an inexpensive potential flow panel and boundary layer analysis was used to "eliminate" by contour re-design, regions of separation in critical inlet flow areas (Fig. 3) at the specified flight conditions. The CFD analysis also later provided the internal performance for the final design such as pressure recovery, distortion, and drag to assess the configuration's success at critical mission points. Subsequent wind tunnel data verified that the inlet met the required recovery and distortion goals and that no separation was present. This effort demonstrated the practicality of the use of CFD and the experiment as complementary tools in airframe propulsion integration development.

Rubbert and Tinoco (Ref. 8) clearly show the benefits of paneling codes in establishing the transport nacelle position relative to the wing and in defining a nacelle shape that minimizes drag. Based on a large empirical wind tunnel data set for many transport nacelle placements, the nacelle was deliberately kept away from the wing to avoid a large "interference drag" (the shaded area, Fig. 4). This interference drag problem was attacked computationally using panel methods. The result was an understanding of the flow phenomenon involved (i.e. an induced or vortex drag caused by a change in the wing span loading due to the presence of the nacelle and strut) and an opening of the available design space. The nacelle/wing installations for the 707/CFM 56, 737-300/CFM 56, 757-200/RB211 and 767-200/JT9D aircraft are, as a result of this CFD application, much more closely coupled than previously believed possible. The difference in the nacelle installation for the original and CFD opened design space for the 737-300 is obvious in Fig. 5.

Also from Reference 8, the Boeing 707 aircraft, when re-engined with the larger diameter CFM 56 engine, experienced an unexpected 10% reduction of the maximum lift coefficient in flight. It was found that the wind tunnel model and the flight vehicle flowfields were dominated by entirely different flow mechanisms. Rather than enter into an expensive and time-consuming flight test fix program, CFD was used to design a non-standard leading edge device to force the wind tunnel flow to model the flight environment. The resolution of the flight lift loss was a vortex control device designed in the CFD modified wind tunnel flowfield. CFD and the experiment were used as complements to satisfy a flight propulsion integration design problem.

For high speed applications, White (Ref. 9) describes CFD's prominent role. Inviscid codes for scramjet inlets are available for routine parametric screening of concepts. Parameters such as forebody bluntness and cowl angle can be qualitatively assessed. These tools can also address inlet capture ratio, additive drag, cowl wave drag, and the internal kinetic energy efficiency (Figs. 6,7). More robust numerical tools include parabolized Navier-Stokes (PNS) codes for a preliminary assessment of viscous effects on performance and full, time dependent Navier-Stokes codes for a quantitative performance analysis. CFD technology for scramjet inlet design and analysis provides a valuable tool for all phases of the design process. The rapidly increasing emphasis on hypersonic flows will demand more development of CFD codes with varying degrees of sophistication, especially in light of limited test facility capability at hypersonic speeds. As a minimum, codes which handle non-equilibrium chemical kinetics and high speed, high temperature flows will be needed. As in other technology areas, the wind tunnel will not be eliminated as an essential tool for inlet development but CFD will lessen the dependence on testing by reducing the need for parametric experimental investigations in the preliminary design phase.

References are available which describe the state-of-the-art of CFD for airframe propulsion integration problems specifically and CFD in general. Shang (Ref. 10) presents an excellent summary of various Navier-Stokes applications and presents an extensive reference list. Boppe (Ref. 11) summarizes CFD as applied directly to propulsion integration. Specific components are addressed by Howlett (Ref. 12) for inlets, Vadyak (Ref. 13) and Lee (Ref. 14) for subsonic diffusers, Chen (Ref. 15) for nacelles, Putnam (Ref. 16) for nozzles, and for complete aircraft solutions, Jameson (Ref. 17, 747-inviscid transonic solution with nacelles) and Shang (Ref. 18, M=6.0 lifting body by Navier-Stokes).

The CFD tools to independently solve many propulsion integration design problems are not available at this time, but the examples presented demonstrate the potential for using CFD and the experiment together in API development. Projected benefits for use of these technologies are so large that the impetus for development must continue to be strong now and in the future. The rate of this development will be set by the resolution of the technical and human factors discussed in later sections.

III CFD Assistance in the API Experiment

The limitations of experimental testing are particularly acute for airframe propulsion integration problems due to the complexity of the flowfield. Integration of surface pressures to determine forces, for example, can be difficult due to complex three dimensional geometry. Local turbulence and measuring device interference make wake and base region flowfields difficult to measure. The following examples are representative of API problems where CFD can assist the development of experimental testing techniques.

Korkegi (Ref. 6) cites two critical areas where the wind tunnel test can be improved by CFD, i.e. data quality and facility operational efficiency. Improvements in data quality are focused on corrections for the effects of the model support, wall interference, and instrumentation sizing and placement. CFD should be utilized to correct for the interference of the model strut/sting. Wall interference which is often cited as the cause of data disagreement between different wind tunnels, can be computationally determined to identify how the presence of the tunnel walls affects the data. CFD can provide loads estimates to properly select force balances and can also provide flow field insights so that a test engineer can make judgments concerning important flow parameters and flow phenomena. Instrumentation can then be concentrated in the important areas resulting in more efficient testing and more significant test data. Test facility operators can use CFD to increase efficiency and productivity by using computed results to identify anomalous test results on-line. Knowledge of the flowfield provided by CFD can guide selection of boundaries of survey regions such as those measured by laser doppler velocimeters (LDV) or pressure probes.

CFD can assist in development of experimental techniques in many phases: test planning,test execution, post test analysis, test facilities studies, and in instrumentation development (Ref. 7). The role of computers and the experimental test facility has been and is the emphasis of international working groups (Refs. 19, 20, 21). The full blending of CFD and the API experiment will some day provide data not before available, reduce overall test time, reduce post test analysis, and in general, solve API problems during aircraft development more accurately and efficiently.

IV. Experimental Assistance in API Computations

While it can be shown that CFD has many uses, it can't provide all the answers. In computing flows characteristic of aircraft-propulsion integration (API), the best success has been achieved using inviscid codes to predict pressures over surfaces with subsonic, attached flow. For example, inlet flow can be calculated at small angles of attack. Calculation of the mutual effects of a wing-pylon-nacelle can allow tailoring the design of the strut and nacelle to reap benefits in reduced "interference" drag. Future development and use of CFD depends in part on assistance from the experiment. Well defined, very accurate experimental data can be used to verify and expand the accepted range of application of existing and developing CFD codes.

One good type of experimental flow to study for API's needs is the flow in the vicinity of a separation bubble induced by a shock wave, such as in the aftbody area of a fighter (Fig. 8). Both Marvin (Ref. 20) and an AGARD working group (Ref. 22) identified this flow as a troublesome one for turbulence models used in CFD codes. In addition, the experimental study of simple symmetric nozzle flows with jets directed straight aft or slightly deflected would help to establish data useful for CFD code validation. A further move towards three-dimensional geometries such as the "2-D" nozzles being considered for thrust vectoring would give data for use in validating CFD codes necessary for advanced aircraft applications.

To see specifically what kind of data is needed to validate a CFD code, consider the CFD engineer's perspective when approaching an analysis problem. The engineer must define a computational domain around the aircraft and specify boundary conditions in order to run the computation (Fig. 9). Unfortunately, as the AGARD group showed, most of the experimental data taken in the past on nozzles has been force and moment data intended to understand how to minimize aftbody drag. Upstream conditions, tunnel wall conditions, downstream conditions and model surface conditions were not considered to any great extent. Unfortunately, these are the boundary conditions the computationalist must have to do the computations. The experimental data needed for validation of codes computing one of the most difficult flowfields, the nozzle/afterbody, is as follows:

1) detailed flow surveys at the nozzle exit including the base area, and internal and external flows;

2) boundary conditions upstream on the model, on the tunnel walls and at downstream pressure boundaries;

3) shear layer details between inviscid plume and freestream flow;

4) an exhaust jet with a temperature representative of a real engine;

5) an accurate 3-D geometry description...with a progression from axisymmetric models over an angle-of-attack range, to twin jets with close spacing, to generic twin jet models with empennage. (Ref. 23)

Once experimental flow conditions have been selected for CFD code validation and the boundary conditions have been recorded, the next step for the experimentalist would be to record not only model force and moment data but also detailed pressure information both on and off the model surface. A flowfield survey of velocity with hot wire anemometry and laser doppler velocimetry is also helpful. Flow visualization will give a qualitative feel for how well the CFD code is capturing the basic features of the flow.

This kind of detailed information describing the flow field is far superior to simple force and moment data. As Barton (Ref. 7) points out, some CFD codes may compute surface pressures that are well off the actual values but will predict fairly good forces and moments by integrating these pressures. Such a fortuitous result leaves us uncertain as to whether we would be as lucky on the next computation. Consequently, both detailed flowfield information and the force and moment data are needed. A statement of the current use of the wind tunnel to verify CFD codes is presented by Marvin in Ref. 24.

Obtaining the experimental data to assist the computational development does not naturally occur in aircraft development programs. The experimentalist may not be aware of the type of data or the level of accuracy required to satisfy the CFD needs. Management whenever possible can help obtain necessary data by directing dedicated experiments or by "piggybacking" on other experiments in a minimal interference manner. The experimentalist can also be educated to become aware of CFD validation and development needs. The experimentalist and the computationalist must work together to understand each others unique problems. Effort short of these steps will postpone the availability of the required design tools for API problems.

V. Other Limiting Factors

Beyond the technical issues, other factors such as "user friendliness", CFD acceptance, and management support can limit the use of CFD for design. Code developers have focused much attention on new CFD algorithms, innovative uses of CFD codes, and the like, but few understand the critical role "user friendliness" has in obtaining high productivity from CFD. Note that here "user friendly" is defined as how quickly and conveniently a user can accomplish an acceptable CFD analysis. The problems in the scientific computations arena are contrasted with the use of the computer in the business world. While the software used by an accountant has standard notations, concepts, and conventions, no standards have been established for scientific software (Ref. 25).

Often "user friendliness" receives "lip service" during code design or is added band-aid fashion after the CFD code is written. Conscious acceptance of the impact of "user friendliness" up front in code design would save a great deal of wasted manpower later. Unfriendly codes or systems for readying input to the codes cause users to make modeling mistakes. Users typically have to have a lot of experience with the code to be productive with it. Complicated options without default values or poor understanding of the implications of using certain input values create user confusion, encourage mistakes, and make the code much harder to debug. Poor quality user's manuals complicate the problem. Couple these problems with slow turnaround from an organization's computer and it can take a long time before a user correctly analyzes an aircraft with a CFD code (Fig. 10). Ironically, one of the premier strengths of computers, a quick analysis of data, is lost in a morass of little problems that force the user to submit and resubmit jobs to the computer until the input is right.

Having stated that improving "user friendliness" would greatly increase the productivity of CFD codes, note that the attractive aspect of this CFD technical development need is that, in most cases, the technology is already in hand to greatly improve this area. Pre- and post-processing of CFD data by using advanced graphics terminals and by developing or acquiring geometry manipulation software will accelerate the tedious procedure of developing computer-understandable models of the configurations to be analyzed. The user also reaps big dividends through a quicker understanding of the computed flow field. The most effective way to use this combination of hardware and software appears to be through the use of a data management system such as that proposed by LaBozzetta and Cole (Ref. 26). With a packaged system, the user doesn't have to confront a myriad of time-consuming details as he or she would if the system were not available to standardize tasks like geometry manipulation.

Please note that implementation of "user friendliness" will never result in "black boxes" for highly sophisticated flow codes. The need will not be diminished for engineers highly experienced in programming and handling codes on supercomputers, for mathematicians trained to develop new and efficient algorithms for Euler and Navier-Stokes equations, or for a high degree of physical understanding of the flow phenomena to interpret the numerical results and to develop accurate numerical models (Ref. 23). The consideration of "user friendliness" will assure, however, more general use of CFD codes in the total design process.

Although perhaps not often consciously identified as a CFD development need, people problems have a marked affect on CFD's utility. In considering the issues of CFD confidence and acceptance, an interesting analogy to the use of experimental techniques can be drawn. Because experimental techniques have been around for so long (dating back at least to the Wright brothers) and because these techniques provided the only tool for assisting aircraft design, experimental testing in wind tunnels has an established place in aircraft design. CFD will likely follow a similar evolutionary path to acceptance like experimental testing. CFD will only be used if pressing needs (e.g. avoiding expensive experiments) convince us to use it.

This "convincing" question orbits around a "Which comes first, the chicken or the egg?" dilemma. If CFD is to become viable in an organization, high-level management must support it. However, before management supports it, they want to know if it can be viable. Management must believe CFD can help produce a quicker, more economical system development to the required level of excellence than could be achieved in experimental facilities alone (Ref. 8). In this dilemma, high-level management must use visionary leadership to support CFD for its promise. The most comforting answer to a manager's question about CFD's viability may be that a CFD-experimental integration will reduce the risk of a design mistake by providing two predictions of a design's performance instead of just one (experiment). Without high-level support, CFD will not survive in an organization.

The technical personnel structure set by an organization's management also determines the success of CFD development. Setting up a CFD group separate to some extent from the more traditional experimental testing groups has advantages (Ref. 27). A separate group provides a focus for developing CFD expertise and because of its corporate memory, its members will learn what codes work well and which ones don't. They learn what modeling techniques worked and which ones didn't. They are often in the best position to take basic research (user unfriendly) type codes and build them into a user friendly system that will increase their likelihood of being used throughout their organization. They can identify and campaign for specialized equipment required by CFD, an important consideration for any group competing with other groups for limited funds.

Existence of a CFD group separate from other groups can create some communication barriers, though. "Such barriers are in part a consequence of the existence of departments the very separation of which, in one way or another, may inhibit information flow or communication." (Ref. 27). A solution to this problem is to have both experimental and CFD groups report to a person with a sound background in both, a kind of fluid dynamics general practitioner who can refer the patient (aircraft designer) to the right specialist and can recognize opportunities for productive collaboration between experimentalists and computationalists.

Both issues, i.e. lack of "user friendliness" and management acceptance and committment to the computations, are in some instances now blocking the complementary use of CFD and the experiment. Progress to the resolution of these items must be emphasized to have the tools available to satisfy future API design problems.

VII. Outlook and Recommendations

Use of CFD and experiment in aircraft development is at a critical stage, and the future course of both technologies will be determined by the near term actions of experts in both specialities. The wind tunnel as an aircraft development tool has matured to a point of acceptance even though it's techniques have well documented flaws. These flaws, however, offer the areas of most promise for future work. Primary tunnel flow problems such as blockage, bouyancy, and support interference will be calculated by CFD and incorporated in the data corrections. The high costs often associated with testing, both financial and time, will be reduced with pre-test predicted data allowing higher accuracy, finer placement of instrumentation, and on-site determination of erroneous data. Much parametric testing previously conducted in the wind tunnel will be done computationally resulting in testing of only the most promising configurations.

The major pacing items for CFD development are computer capability (speed and storage), algorithm sophistication (Ref. 9) and validation and confidence in code use. The speed and storage issues show strong progress toward solutions. The ETA^{10} (Ref. 28) will be 16 times more powerful than the current state-of-the-art Cray XMP/48 and new computers are expected by 1990 which will be three to ten times faster than the ETA^{10}. Concerns to be addressed include computer saturation and computation costs. New large computers are often backlogged with users on the first day of operation with turn-around times approaching one week. Increased numbers of very large, very fast computers will alleviate this situation though computer application is by all measures expected to increase. Computer cost, that is, cost to purchase (or lease) a mainframe and to support or run a unit will continue to grow. This is now a carefully conserved resource in corporations with separate computing organizations.

The application of CFD to practical engineering design problems, especially for very difficult flow phenomena associated with API, will be determined by algorithm development and the validation/verification (confidence) procedures used. The development cycle for code acceptance (Fig. 11) is lengthy with Hankey (Ref. 29) projecting a 20 year cycle from the basic research until the method reaches production. Both Korkegi (Ref. 6) and Boppe (Ref. 30) state that the calculation of difficult separated and interacting flows, a maneuvering aircraft at high angle-of-attack, for example, will depend on full Navier-Stokes equations with very good flow separation and turbulence models. This capability is not expected for 15-20 years. CFD may follow the evolutionary path taken in wind tunnel facility development, that is, slow acceptance over time as an alternative to even more costly wind tunnel or flight test. While the acceptance of CFD may be forced on the industry by declining budgets or lack of suitable test facilities, the time of introduction of CFD as a design tool for a wide range of problems will depend on a conscious validation/verification process by comparison with data from ground test facilities and flight test. As confidence in CFD matures, greater complementary use of the computational and the experimental result will be seen. Ballhaus (Ref. 4) suggests that the time in the wind tunnel will not be reduced but will be used more intelligently and effectively.

Much must be done in the coming years to ensure adequate development of both CFD and the experiment to attain the capabilities projected 20 years in the future. This process may be long and tedious with no one group primarily responsible. Those who develop the codes, for example, do not have the time or are generally not always interested in the application and verification of the codes or making them "user friendly" to the extent engineers can use them with confidence. Who will do this? Too often the codes are always developing and never moving into the useful stage (Ref. 6). Boppe (Ref. 11) and Marvin (Ref. 20) cite the need for more detailed experimental studies along with a closer analytical/experimental link to advance the capabilities of CFD. The critical assessment of CFD through comparison with experiment provides the validation and resulting confidence in the methods to simulate complex flowfields. In general, high quality experimental data is required for unsteady, separated flow in both two and three dimensions. More must be done experimentally to determine the flowfield structures and critical parameters to gain further understanding for CFD modeling and to provide well documented benchmark tests against which progress can be gauged (Marvin, Ref. 20). Consistent with the findings cited here and to expedite the development of a complementary CFD/experimental capability, the following recommendations are proposed:

the computationalist should:

1) establish the accuracy and applicability of his numerical approach to the propulsion component flowfield. This should be done with existing verified data (where available) to calibrate the method over specific ranges of test conditions;

2) apply existing methods to practical propulsion integration problems to establish the utility of these computational techniques;

3) strengthen efforts that are developing three dimensional, viscous, high angle-of-attack CFD methods and high temperature chemical kinetics codes

the experimentalist should:

1) recognize the data needed to predict the test configuration, and to upgrade and verify the CFD code, and <u>then</u> <u>facilitate</u> acquiring this data in ongoing and future tests,

2) assist the computationalist in developing reference data cases of sufficient accuracy and elements;

3) utilize available computational methods to assist in test planning, execution and analysis.

The management (people) development of CFD and the experiment to address complex API flow problems is as important as the technical development. As both capabilities develop, there will be inherent problems in modeling for both CFD and the wind tunnel. These modeling problems may be only partially understood within the respective disciplines and completely misunderstood outside the discipline. In this situation, further advancement is only possible when specialists in both fields are aware of the procedures, lines of thought, and problem areas of the different specialties (Ref. 19). This will require new connections between the two groups of specialists. Dornier, for example, has instituted a "code application" group which conducts analyses for the design engineer and then helps interpret the computational results (Ref. 8). A better appreciation for CFD by the experimentalists and for the experiment by the computationalist will accelerate progress by both groups. It is recommended that:

1) a qualified group of specialists examine the inherent problems in both CFD and the experiment and determine the required data accuracy and flow quality needed for wind tunnels to support CFD development and validation and to determine the specific areas where CFD can assist the experimental fluid dynamics research and development (Ref. 4);

2) the CFD technical community strongly consider the "human" factor in use of CFD codes. "User friendly" is not a trivial consideration in determining the use of advanced CFD codes. This task should be assigned in the organization to ensure it is accomplished;

3) industry management review its committment to the development of both CFD and the experiment and critically assess practices which hinder communications between both groups;

4) government agencies through contract clauses, and industry in their internal research and development, require CFD prediction of test configurations prior to test and the installation of instrumentation to obtain useful CFD development data during the test.

5) all groups, government, industry, and academia, should strive to acquire to the maximum extent possible a basic understanding of the flowfield physics. This understanding encourages development of accurate computational models and experimental simulations contributing to solving the complex API design problem.

References

1. Paynter, G.C., Koncsek, J.L., Turczeniuk, B., Dvorak, F.A., "Extension of CFD Technology Used to Design the JVX Inlet", AIAA 85-1215, AIAA/SAE/ASME/ASEE 21st Joint Propulsion Conference, Monterey, California, July 1985.

2. Abercrombie, J.M., "Flight Test Verification of F-15 Performance Predictions", AGARD CP-242, paper 17.

3. Flax, A.H., "Aeronautics--A Study in Technological and Economic Growth and Form", The Aeronautical Journal, Dec 1974, Vol 78, pp 537-552.

4. "New Computers Will Aid Advanced Designs", Aviation Week and Space Technology, August 1983.

5. Tinoco,E.N., Chen,A.W. , "Transonic CFD Applications to Engine/Airframe Integration", AIAA 84-0381,January 1984.

6. Korkegi, R.H., "Impact of Computational Fluid Dynamics on Development Test Facilities", Journal of Aircraft, Vol 22, No 3.

7. Barton, J.M., "The Role of Computational Fluid Dynamics in Aeropropulsion Ground Testing", Journal of Aircraft, Vol 21, No 10, October 1984.

8. Rubbert, P.E., Tinoco, E.N. "Impact of Computational Methods on Aircraft Design", AIAA 83-2060, August 1983.

9. White, M.E., Drummond, J.P., and Kumar, A., "Evolution and Status of CFD Techniques for Scramjet Applications," AIAA 86-0160, 6-9 Jan 1986.

10. Shang, J.S., "An Assessment of Numerical Solutions of the Compressible Navier-Stokes Equations", AIAA 84-1549.

11. Boppe, C.W., "Elements of Computational Engine-Airframe Integration", AIAA-84-0117, AIAA 22nd Aerospace Sciences Meeting, Reno, Nevada, January 9-12, 1984.

12. Howlett, D.G., Hunter L.G., "A Study of a Supersonic Axisymmetric Spiked Inlet at Angle of Attack Using the 3-D Navier Stokes Equations", AIAA 86-0308, January 1986.

13. Vadyak, Joseph, "Simulation of Diffuser Duct Flowfields Using a Three-Dimensional Euler/Navier-Stokes Algorithm", AIAA 86-0310, January 1986.

14. Lee, C.C., Boedicker, C.,"Subsonic Diffuser Design and Performance for Advanced Fighter Aircraft", AIAA 85-3073, October 1985.

15. Chen, H.C., et. al. "Flow Simulations for General Nacelle Configurations Using Euler Equations", AIAA 83-0539, January 1983.

16. Putnam, L.E., Bissinger, Dr. N.C., "Results of AGARD Assessment of Prediction Capabilities for Nozzle Afterbody Flows", AIAA 85-1464, July 1985.

17. Jameson, A. et. al., "Calculation of Inviscid Transonic Flow over a Complete Aircraft", AIAA 86-0103, January 1986.

18. Shang, J.S., Scherr, S.J., "Navier-Stokes Solution of the Flow Field Around a Complete Aircraft", AIAA 85-1509-cp, 1985

19. "Windtunnel Capability Related to Test Sections, Cryogenics, and Computer-Windtunnel Integration", AGARD Advisory Report No. 174, 1982.

20. Marvin, J.G., "Advancing Computational Aerodynamics Through Wind-Tunnel Experimentation," AGARD Fluid Dynamics Panel Meeting, Chattanooga, TN, 24-25 Sept 1980.

21. Marvin, J.G., "Experiments Planned Specifically for Developing Turbulence Models in Computations of Flow Fields Around Aerodynamic Shapes," AGARD Conference Proceedings No. 210 on Numerical Methods and Windtunnel Testing.

22. "Aerodynamics of Aircraft Afterbodies", AGARD Working Group 08, final report, 1986.

23. Sacher, Peter, Messerschmitt Bölkow Blohm, personal conversation.

24. Marvin, J.G., "Future Requirements of Wind Tunnels for Computational Fluid Dynamics Code Verification", AIAA 86-0752-CP, March 1986.

25. Denning Peter J., Adams III, George B., "Wanted: supercomputers for engineering design", Aerospace America, January 1985.

26. LaBozzetta, Walter F. and Cole, Paul E., "Interactive Graphics for Geometry Generation - A Program with a Contemporary Design," J. Aircraft, Vol. 22, No. 12, pp. 1054-1058, Dec 1985.

27. Bradbury, F., Jervis, P., Johnston, R., Pearson, A., Transfer Processes In Technical Change, Sijthoff and Noordhoff International Publishers B. V., Alphen aan den Rijn, The Netherlands, 1978.

28. "The Future Is Now," Forbes, Kindel, S., 27 Jan 1986, pp. 88-92.

29. "USAF Studies Non-Linear, Vortex Flows", Aviation Week and Space Technology, August 29, 1983.

30. "Modeling Methods Aid Airframe Design", Aviation Week and Space Technology, August 29, 1983.

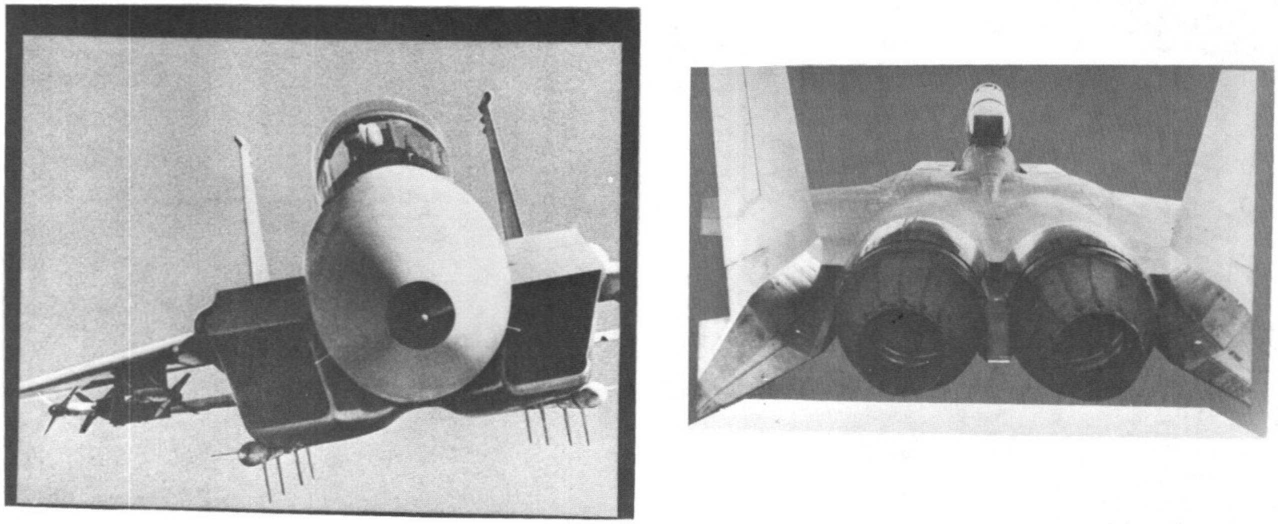


FIG. 1 CHARACTERISTIC AIRFRAME/INLET AND AIRFRAME/NOZZLE INTEGRATIONS

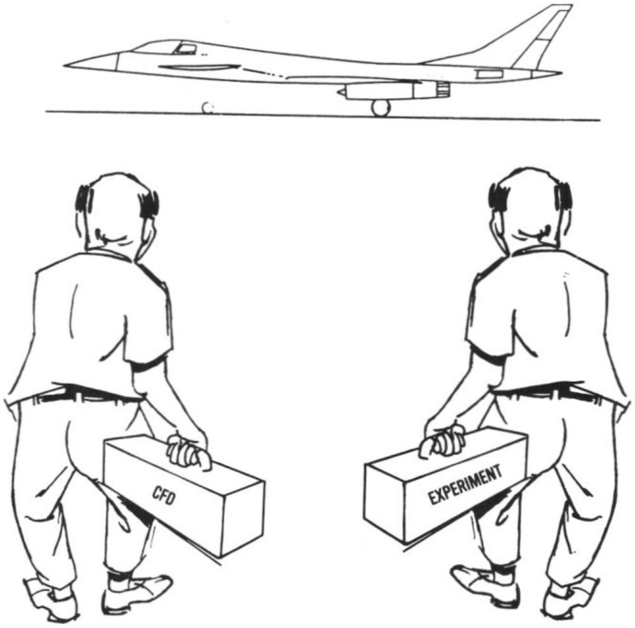


FIG. 2 AIRCRAFT DESIGN TOOLS

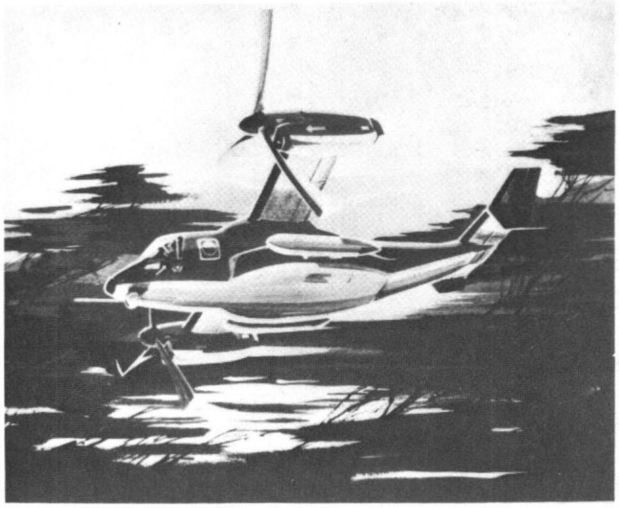
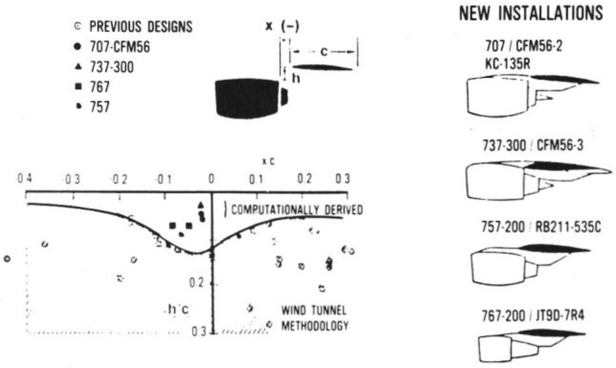
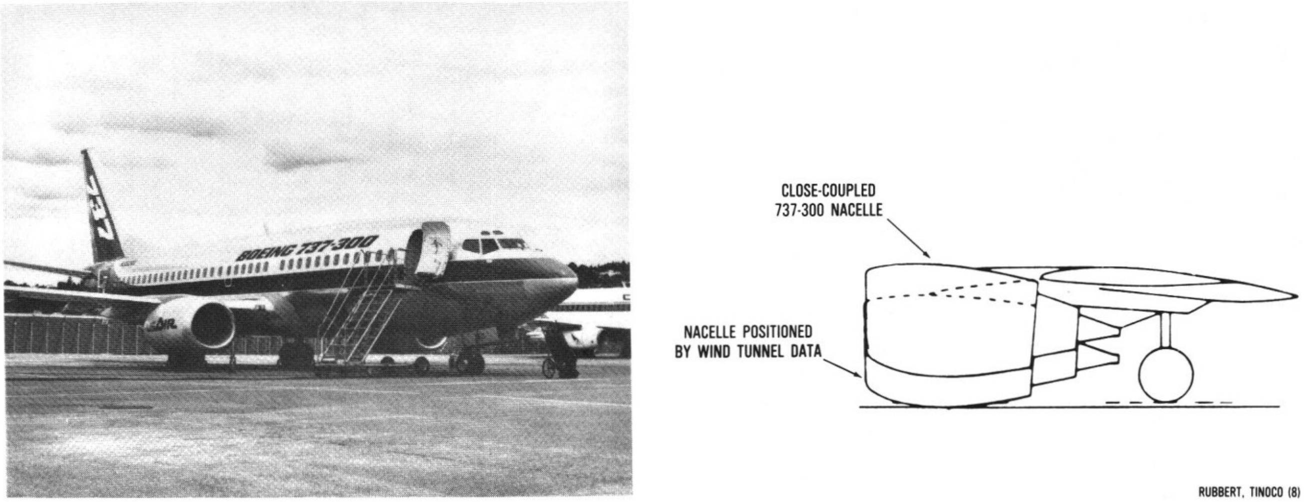


FIG. 3 JVX AIRCRAFT AND CRITICAL INLET FLOW REGIONS



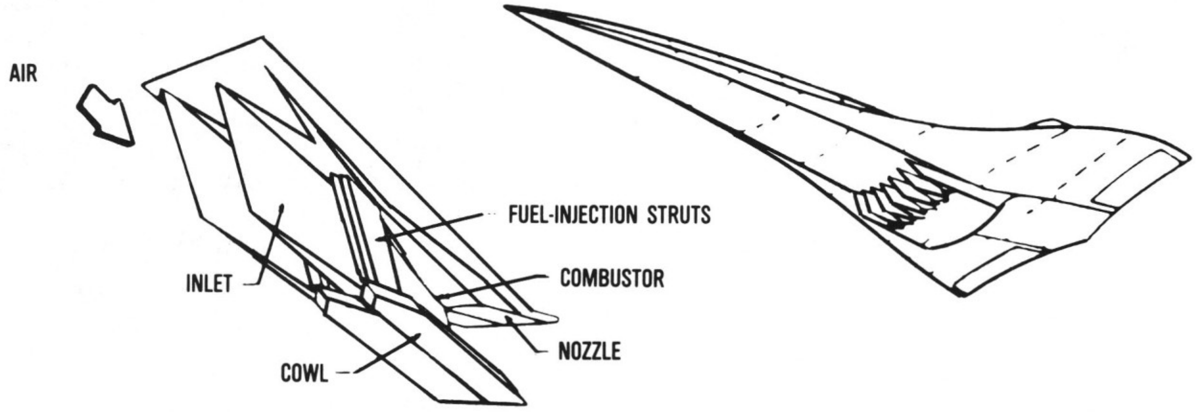
RUBBERT, TINOCO (8)

FIG. 4 COMPUTATIONALLY DERIVED CLOSE-COUPLED NACELLE POSITIONS



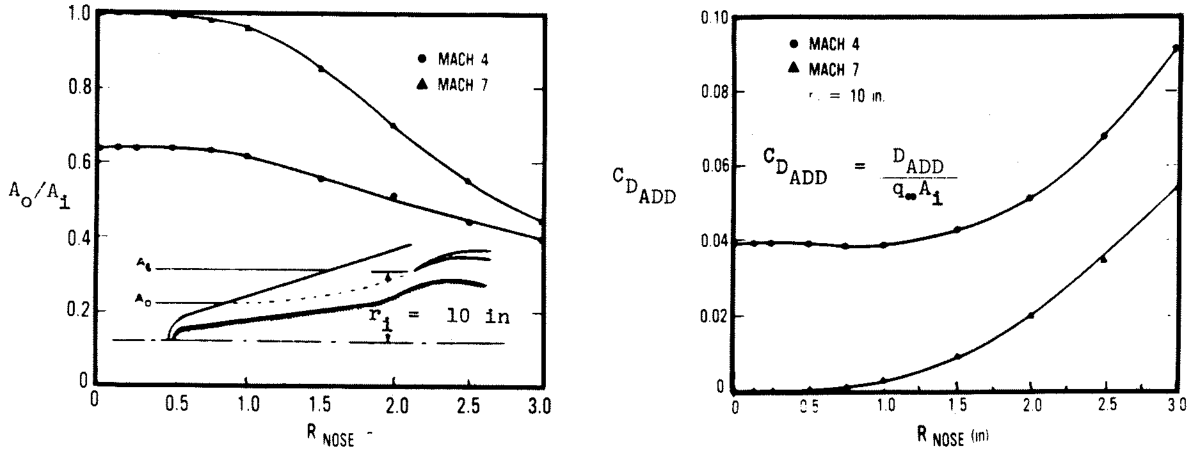
RUBBERT, TINOCO (8)

FIG. 5 BOEING 737-300 AIRCRAFT AND IMPACT OF CLOSE NACELLE COUPLING ON ENGINE INSTALLATION



OBLIQUE VIEW OF SCRAMJET MODULES.

FIG. 6 HYPERSONIC ENGINE/AIRFRAME INTEGRATION



AIR CAPTURE VERSUS SPHERE-CONE NOSE RADIUS FOR A TYPICAL SCRAMJET INLET.

ADDITIVE DRAG VERSUS SPHERE-CONE NOSE RADIUS FOR A TYPICAL SCRAMJET INLET.

WHITE (9)

FIG. 7 CFD FOR SCRAMJET INLETS

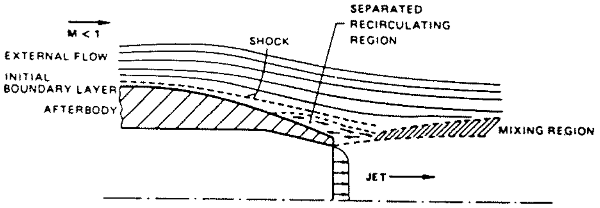


FIG. 8 AFTERBODY/NOZZLE FLOW CHARACTERISTICS

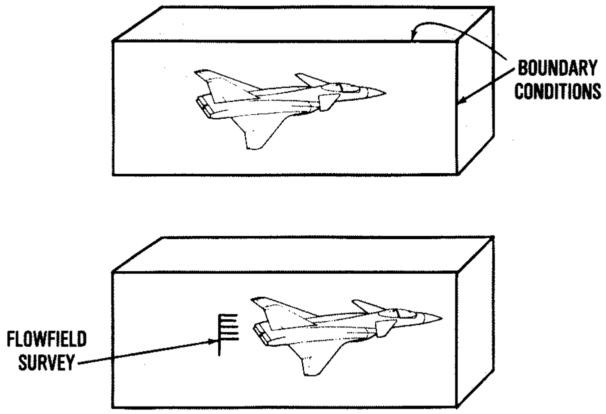


FIG. 9 EXPERIMENTAL DATA TO VERIFY CFD

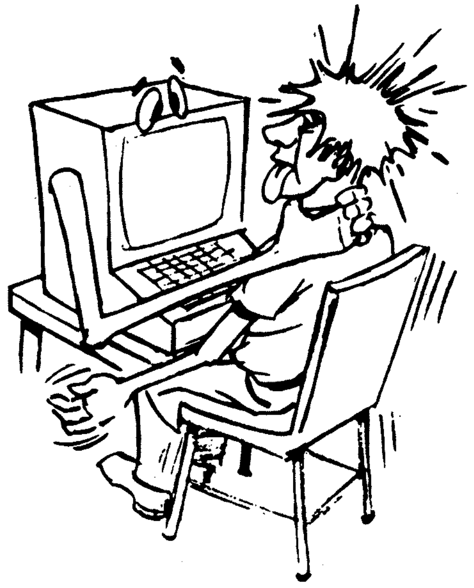
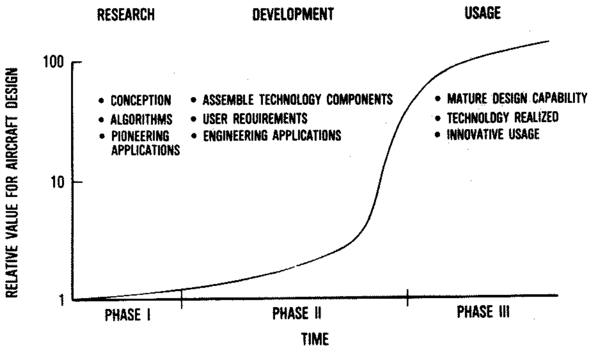


FIG. 10 UNFRIENDLY CFD SYSTEMS



KORKEGI (6)

FIG. 11 DEVELOPMENT CYCLE FOR A MAJOR COMPUTATIONAL CAPABILITY