THE MODERN ROLE OF CFD IN ADDRESSING AIRFRAME/ENGINE INTEGRATION ISSUES FOR SUBSONIC TRANSPORTS

F. T. Lynch
MDC Fellow
Program Manager, Integrated Wing Design
and
G. A. Intemann
Manager, Transonic Aerodynamics Technology
McDonnell Douglas Aerospace - Transport Aircraft
Long Beach, California

Abstract

The ever-increasing role that CFD has played in the development of effective, integrated wing-mounted engine installations for subsonic transports is reviewed. Three examples are used to illustrate the role that CFD is currently playing in the aerodynamic design process for airframe/engine integration at transonic cruise conditions. First, an Euler method, in conjunction with an inverse design capability, is employed to design a pylon-leading-edge fairing that eliminates a known flow separation. Second, the use of a Navier-Stokes method to define the wing flowfield influence on core-exhaust flow characteristics is illustrated. The third example illustrates the use of an Euler/IBL method to accurately depict the nacelle/pylon influence on the wing upper-surface transonic flow development. The near-term future role of CFD in addressing airframe/engine integration issues for large high-bypass-ratio engines will be to avoid adverse interference at cruise conditions by permitting the design of wing, pylon, and nacelle concepts accounting for mutual flowfield interferences. The potentially most-significant new role for CFD will be for guiding the effective installation of these large engines at high-lift conditions.

Introduction

Historically, interference drag (and other effects) associated with engine installations on subsonic transports has contributed a significant part of the performance shortfalls encountered in flight.(1) No airframe manufacturer has escaped this situation. Probably best known was the Convair 990,(2) where excessive nacelle/pylon/wing interference drag encountered in their quest for an 0.89 cruise Mach number eventually led to their demise as a prime builder of transport aircraft. Douglas encountered significant interference drag on early versions of the DC-8, and had to finally resort to the forward-located long-duct nacelle/cut-back pylon design on the -62 and -63 to provide a low-interference-drag installation. However, even then it was not easy because the initial prototype long-duct nacelle installation (located further aft) encountered much greater interference drag at flight Reynolds numbers than experienced at typical (lower) wind-tunnel Reynolds numbers (see Figure 1). This was one of the earlier documented(1) examples of significant adverse Reynolds number effects which, unfortunately, have subsequently become more numerous, and are now clearly recognized as a threat to be dealt with.(3,4,5)

Much progress has been made since these earlier days in our knowledge of what leads to an effective <u>integrated</u> wing-mounted engine installation. For example, the following installation concepts were introduced on the DC-10:

- Upper-external-cowling designed to avoid flow separation at engine-out second-segment climb conditions.

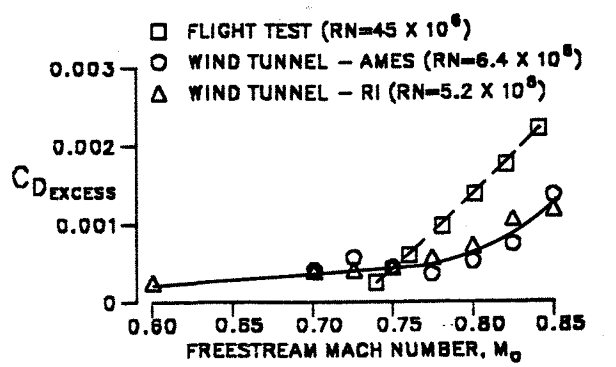

Figure 1. Interference Drag of DC-8 Prototype Long-Duct Nacelle Installation.

- Lower-internal-inlet contours designed to preclude flow separation at high angle of attack, high engine-mass-flow conditions.
- Minimizing nacelle lift (total induced drag).
- Nacelle/pylon toe-in to account for sweep effects.
- Use of nacelle-mounted strakes to help alleviate loss in high-lift performance due to nacelle/pylon installation.

However, on the continuous learning side, another type of adverse Reynolds number effect was observed on the DC-10 in that a separation was observed in flight at the trailing edge of the nacelle-pylon juncture region (on the inboard side) as illustrated in Figure 2 that was not observed in powered-nacelle wind-tunnel testing at significantly-less-than-flight Reynolds numbers. This was one of the earlier documented examples of adverse Reynolds number effects on separation onset in juncture flow regions.(1) There have been several more since.

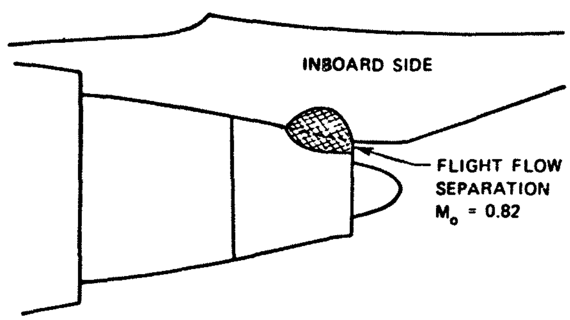

Figure 2. Flow Separation at Nacelle-Pylon Juncture.

In the 20-plus years since the DC-10 development, many additional installation concepts have been developed and implemented.(6-9) These include:

- Nacelle-external-cowling profiles designed to minimize drag at Extended Range Operations (ETOPS), i.e., cruising with a failed engine at high speeds for extended times.
- More-closely-coupled nacelle/wing installations with (proportionally) larger engines.
- Wing-lower-surface contouring to minimize nacelle/pylon/under-wing interference.
- Nacelle placement/designs to avoid adverse jet interference effects.
- Accounting for nacelle/pylon effect on wing span loading (and related wave drag).
- Pylon-leading-edge shaping to eliminate/ minimize separated flow regions.

An example of the latter was the development of a pylon-leading-edge fairing for the MD-11(9) that eliminated a region of separated flow on the outboard side of the pylon near the leading edge in the wing juncture region (that had undoubtedly been there since the first DC-10). The failure to discover the excess drag on the DC-10 provides another example where testing at typical wind-tunnel Reynolds numbers has not revealed excess interference drag encountered in flight. In this case, it is theorized that the separation at wind-tunnel Reynolds numbers involved a laminar boundary-layer separation with much less dissipation (drag) than would be experienced with the turbulent boundary-layer separation at flight conditions.

Throughout the foregoing development of more efficient (and sophisticated) wing-mounted engine installations, the overall design philosophy has not significantly changed. That is, with the exception of the wing lower surface modifications, the wing (particularly the upper surface) has been designed ignoring the influence of the engine/nacelle/pylon. Likewise, the nacelle has been designed only accounting for first-order effects of the wing flowfield, i.e., average local Mach number and flow angularity, involving a lot of isolated nacelle analysis and testing. Then, an extensive (usually low Reynolds number) experimental/CFD-guided effort is mounted to assess and minimize (to the extent possible) the nacelle/pylon interference drag (and other) effects. Unfortunately, this process has not been overly effective to date, and it is certainly not adequate for addressing the next major challenge facing the airframe and engine manufacturers, which is the installation of larger, higher bypass-ratio engines on advanced-technology wing designs (see Figure 3). These installations pose a greater challenge because these new wing designs, either all turbulent or laminar flow control (LFC) concepts, have an increased sensitivity of the flow development to small flow-condition changes. Similarly, LFC concepts are being considered for the fan cowls of the larger nacelles, leading to the requirement for much greater accuracy in achieving target pressure distributions on the nacelle.

Consequently, a new process is required wherein all the potential sources of interference

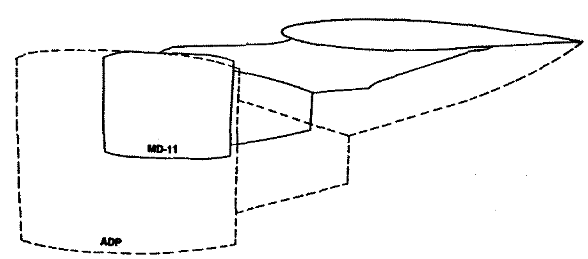

Figure 3. ADP vs. MD-11 Nacelle Size Comparison.

(drag and other) are considered, but, in addition, the wing (and high-lift system) is designed accounting for the presence of the engine/nacelle/pylon installation, and the nacelle/pylon is designed accounting for the presence of the wing. It will be shown that the design methods required for this process must be CFD-based, as an experimental development process would be neither practical nor successful. However, high Reynolds number validation testing is an essential part of this new process.

Emerging CFD Role

Use of CFD methods to assist in the design of nacelle installations started decades ago. Linear potential-flow methods were used to define (average) wing flowfield effects at the inlet location, and these linear methods were also used in conjunction with early boundary-layer prediction methods to assist in the design of (isolated) inlet and nacelle geometries. For example, a typical CFD application has involved the development and validation of design changes required after testing has been completed. Such methods were used to predict Reynolds number effects on the DC-8-50 inlet performance, and these predictions were subsequently validated by flight-test results. Similarly, these methods were used to develop external cowling shapes that avoided flow separation at engine-out second-segment climb conditions, starting with the DC-10. These linear full-potential methods have also provided useful guidance for the development of more closely coupled nacelle installations[10] by assisting in the shaping of individual components (such as the wing lower surface[7]) to avoid significant adverse interference effects caused by super-velocity additions (see Figure 4). However, there have been several cases encountered where these linear methods (not unexpectedly) significantly underpredicted supersonic flow regions around nacelle/pylon/wing installations.[10]

The introduction of transonic full-potential methods (coupled to boundary-layer methods) provided a major advancement in the ability to predict such nonlinear supersonic flow regions. Best known of these was TRANAIR,[11] a finite-element full-potential code which avoided the requirement to generate surface-fitted grids to the complex geometries involved by use of a hierarchically refined rectangular grid. Other, surface-fitted structured-grid, methods have also been utilized to address transonic (isolated) nacelle design problems[12] and nacelle/pylon/wing interference flow situations.[13] While some

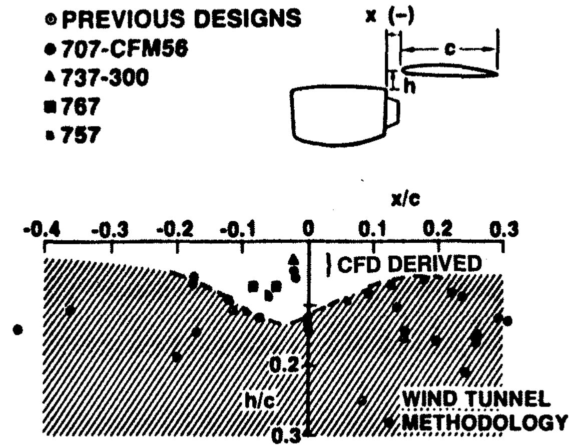

Figure 4. CFD-Derived Close-Coupled Nacelle Positions (Ref. 10).

notable successes (considering the time period) were achieved with these methods, there have been a number of examples encountered where the inability to accurately represent shock characteristics has limited the confidence in these methods. This then led us to the current class of methods, involving Euler, Euler/IBL, and Navier Stokes, which are presently being applied to address wing-mounted (and other) nacelle installation design problems.

Structured-grid Euler and Euler/IBL[14] methods were used most initially for simpler geometries. However, these methods are now being shunned in favor of unstructured-grid Euler and Euler/IBL methods because of the much-easier grid generation process, particularly for application to more complex (full configuration) geometries. Most notable of the unstructured-grid Euler codes are Jameson's AIRPLANE[15] and the NASA Langley-developed USM3D.[16] Most current utilizations of these codes now incorporate IBL corrections.[17] However, some flow situations are encountered in nacelle/pylon/wing installations where the Euler/IBL methods are inadequate. Most conspicuous of these are exhaust jet (mixing) effects and complex separated flow situations. To address these problems, Navier-Stokes methods are now utilized, although the elapsed time to acquire an initial solution for a single complex geometry still takes significantly longer than desired. The multi-block Navier-Stokes code, NASTD,[18] developed by McDonnell Douglas, has been the most widely used code for addressing airframe/engine integration issues, especially for jet-interference effects. More recently, the NASA Ames-developed overlapping-grid OVERFLOW code,[19,20] and versions of the NASA Langley-developed TLNS3D[21,22] and CFL3D[23-25] codes, are being utilized to address interference flow situations not involving jet effects.

The role of CFD in the development of effective airframe/engine installations has increased dramatically in the last decade or so as the modeling of the prevailing flow physics has become more representative with the advanced methods. One of the major contributions of CFD has been in furnishing flow diagnostic capabilities to supplement experimental results in order to provide the understanding necessary to allow deficient characteristics to be addressed and corrected, or to guide the design of concepts with improved aerodynamic characteristics. Years ago, CFD applications typically involved a lot of post-test data comparisons in order to provide insight into the test results. CFD was often applied to develop and validate design changes after testing was complete, although only minor geometry excursions were considered. In the next stage, CFD was employed in pretest studies to minimize the amount of testing by screening potential wind-tunnel-test geometry concepts to eliminate noncompetitive designs, and to help formulate wind-tunnel-test instrumentation and test condition requirements. This was followed shortly by the use of inverse CFD methods to develop (relatively simple) geometries that achieved selected target pressure distributions, including isolated nacelle fan-cowls. Now, however, with the current state of CFD, involving the use of Euler/IBL and Navier-Stokes methods, a new role for CFD is emerging as a consequence of the wealth of information now provided for complex nacelle installations. In this new role, CFD is being used to address nacelle/pylon/wing installation issues that heretofore could not be dealt with in a practical (experimental) way. Also, concepts can be studied quickly without the time lag of wind-tunnel testing. These new capabilities should permit the design of even more efficient nacelle installations with much less testing, but also permit the efficient integration of more-efficient higher-bypass-ratio engines than might otherwise be possible. Some examples of current CFD capabilities applied to wing-mounted nacelle/pylon installations are shown in the following section in order to illustrate how CFD can be used to identify adverse interference effects which could be very significant, and, concurrently, provide a basis for developing geometries and installation concepts that avoid these interference effects.

Current CFD Role

Three examples are used to illustrate the growing role of (need for) CFD in effectively addressing airframe/engine integration issues. They include an exercise in pylon shaping to avoid flow separation, an illustration of the importance of accounting for wing flowfield effects in the design of exhaust systems, and a depiction of the importance of the nacelle/pylon influence on the wing upper-surface flow development (i.e., performance).

Pylon Shaping - In the course of a study to identify potential areas for drag reduction for the MD-11, predictions from the NASTD multiblock Navier-Stokes code[18] revealed the likely existence of a separation at the leading edge on the outboard side of the pylon at the wing juncture. Off-surface streamline traces (see Figure 5) indicated a large recirculating separation bubble in this region. Flight-test tuft observations confirmed the existence of this separated flow region. A subsequent CFD analysis[9] using an unstructured grid Euler (inviscid) code[15] indicated the existence of very high local Mach numbers (~2.0) in this region followed by a rapid recompression which no boundary layer could sustain without separating. Ensuing cut-and-try development of a pylon leading-edge fairing using an early version of the unstructured grid Euler code yielded a shape illustrated in Figure 6 which was predicted to eliminate the large adverse pressure gradient responsible for the separation on the baseline (see Figure 7). To our chagrin, flight tests of this modified shape did not produce the expected drag reduction.

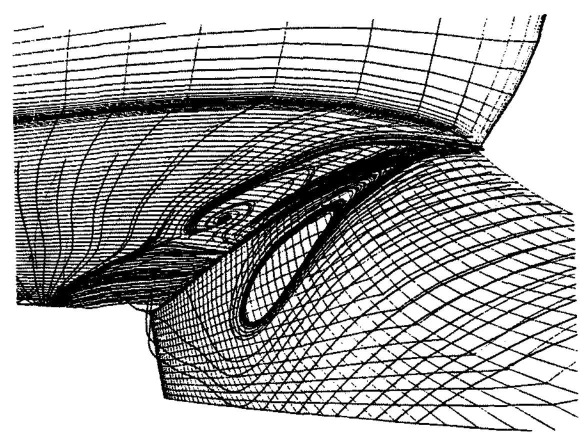

Figure 5. NASTD-Predicted Off-Surface Streamline Traces for MD-11 Baseline Pylon.

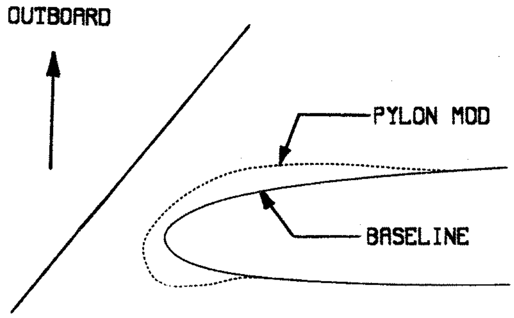

Figure 6. First Flight-Test Pylon Fairing.

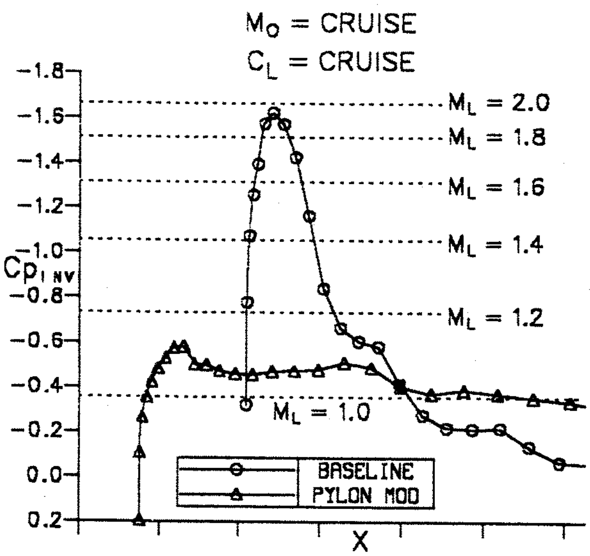

Figure 7. Initial Predicted Pressures on Outboard Side of Pylon at Pylon/Wing Juncture.

In order to understand the failure of the pylon modification (developed with a central-difference Euler code) to produce the desired results, an additional prediction of the (inviscid) flow characteristics of the flight-tested geometry was obtained(9) using the USM3D upwind unstructured grid Euler code.(16). The two predictions are compared in Figure 8, where it can be seen that the upwind code, with a grid density only about 20-percent of the central-difference method, indicated the existence of much higher local Mach numbers. The indicated level with the upwind method was certainly high enough to explain the failure of the tested fairing to eliminate the flow separation in the juncture region. This disagreement between the USM3D and AIRPLANE solutions prompted an investigation of the effects of artificial dissipation in the central-difference AIRPLANE code. The findings were dramatic.(9) It can be seen from Figure 9 that progressively reducing the dissipation levels led to predictions with significantly greater local Mach numbers. Considering the differences in surface (grid) resolution, the predictions at the reduced dissipation levels were in good agreement with the upwind code predictions.

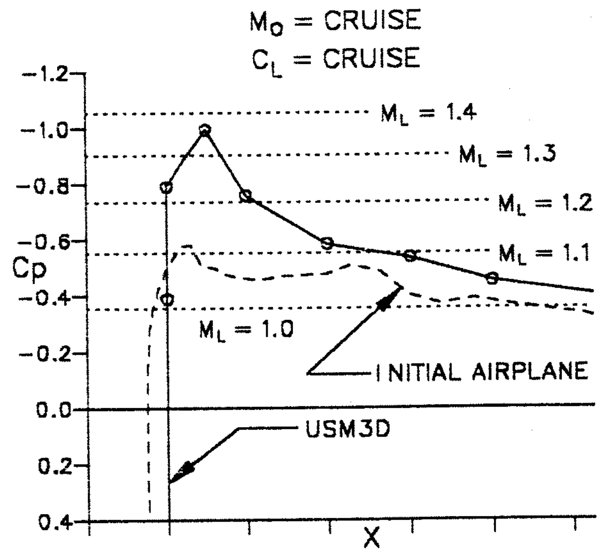

Figure 8. USM3D vs. Initial AIRPLANE Predicted Pressures on Outboard Side of Pylon.

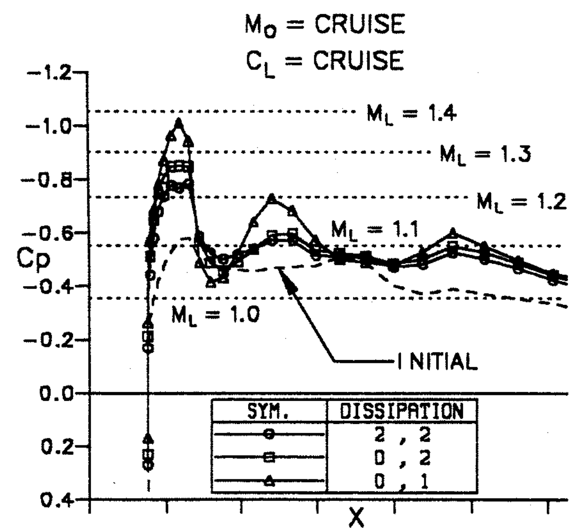

Figure 9. Effect of Reduced Dissipation Levels on Predicted Pressures on Outboard Side of Pylon.

Utilizing the lessons learned from the preceding studies, the design of a new pylon fairing for flight evaluation was undertaken. Initially, the DISC inverse design method(26,27) was used in conjunction with USM3D to provide insight into geometries which reduced the peak pressure (coefficient)/shock strength. Based on these results, both the USM3D and AIRPLANE (with greatly reduced dissipation levels) Euler codes were used in a direct design and analysis mode to define a new pylon fairing with significantly reduced velocity levels relative to the initial flight-tested configuration (see Figure 10). An analysis of this design with the OVERFLOW Navier-Stokes code(29) produced the off-surface streamline traces shown in Figure 11, which indicated the separation bubble had been eliminated.

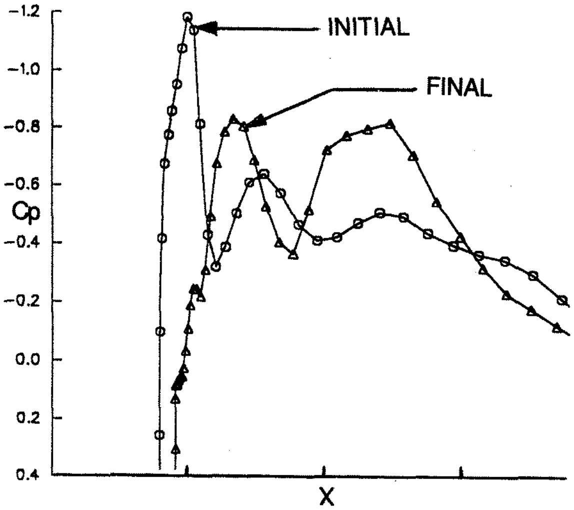

Figure 10. AIRPLANE-Predicted Pressures on Outboard Side of Pylon for Initial and Final MD-11 Pylon Fairings.

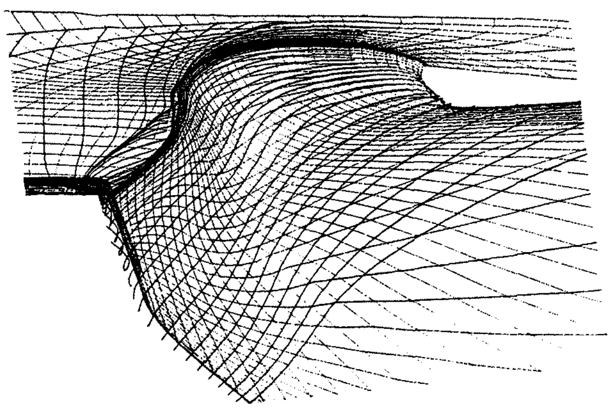

Figure 11. OVERFLOW-Predicted Off-Surface Streamline Traces for MD-11 Final Pylon Fairing.

Flight test results for this configuration did show that an important drag reduction had been achieved.

Several important lessons have been derived from this example. First, the development of such a (successful) modification would likely not have been possible at all if a solely experimental approach had been employed, but, most certainly, not in a practical time period. The inviscid Euler code predictions were invaluable in guiding the systematic development of a geometry that would not induce flow separation. Results from an experimental approach would not provide such systematic information, but would only provide success or failure indications, since the (inviscid) peak pressure region collapses whenever the flow separates. Furthermore, any experimental development would be lengthy and costly since it would have to be carried out at flight Reynolds numbers (i.e., either on a flight airplane or in a cryogenic wind tunnel such as the NTF or ETW) because typical lower-Reynolds-number experiments where laminar boundary layers exist in these leading-edge regions do not provide meaningful results for separation indication.

A second important lesson learned from this study was that the application of inverse design methods which prescribe target pressure distributions is much more complicated for flow situations such as this complex three-dimensional flowfield than it was for earlier applications involving simpler flowfields (i.e. airfoils, wings, isolated nacelles, etc.). Specification of achievable target pressures along unknown streamline directions poses a major dilemma because of the lack of an adequate database to provide the needed guidance. However, CFD applied in the direct mode does offer the capability/opportunity to generate the required databases.

Jet Interference - The use of CFD as an alternative to expensive and lengthy wind-tunnel testing involving the use of small turbine-powered engine simulators to develop airframe/ engine integration designs that avoid adverse jet interference effects has been a goal of transport aircraft developers for some time.(28) The first objective has been to use CFD to more efficiently address any adverse jet interference effects at cruise conditions in the channel between the wing lower surface, core cowl (ahead of the core exit), and the pylon (for separate fan/core exhaust flow engines). Application of the NASTD Navier-Stokes method to an MD-11 configuration at typical cruise conditions predicted no adverse jet interference (separated flow or strengthened shocks) in this area. These predictions were substantiated by flight test tuft observations. The next step is to validate these CFD capabilities for installations which are known to, or designed to, produce measurable adverse jet interference effects in this region. This validation process will require additional wind-tunnel testing of geometries specifically designed to produce the range of interference flow situations needed to assure that the CFD methodology can provide adequate warning for potentially adverse installation concepts.

The foregoing potential application of CFD is very important because of the cost- and design-cycle-time reductions it will provide. This is a situation where CFD is being viewed as a more efficient alternative approach, since this particular jet interference situation can be addressed in wind-tunnel testing utilizing turbine-powered engine simulators. There is, however, an additional jet interference issue/situation that can only be practically addressed using CFD. This involves the region aft of the core nozzle exit, including the core plug and the pylon shelf/heat shield, illustrated in Figure 12. To properly represent the airplane flowfield in this region requires simulating both the fan and core flows (flow conditions and mass flow). This has not been feasible with the small air-driven turbine powered simulators. Consequently, the design of geometry components in this area has traditionally relied on a combined experimental/ CFD approach, with validation testing for cruise conditions being carried out using an isolated (i.e., no wing) nacelle/pylon mounted on the end of an air supply pipe. Unfortunately, in this case, any influence of the wing flowfield is neglected/ ignored.

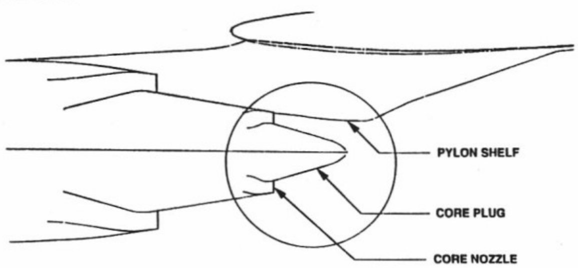

Figure 12. Core-Exhaust Interference Region.

To assess the adequacy of this traditional approach, NASTD predictions for a MD-11 (engine) configuration at cruise conditions were compared for the installed nacelle/ pylon and a case with no wing present. Two significant areas of disagreement were observed. One involved flow conditions on the core-exhaust-scrubbed pylon shelf/heat shield as illustrated in Figure 13. As can be seen, predictions for the isolated nacelle/pylon indicated completely attached flow on the pylon shelf, which is in agreement with corresponding experimental results. However, when the wing flowfield influence is accounted for, a flow separation (i.e., drag/thrust loss) is predicted to occur, clearly indicating the inadequacy of the traditional design approach. Unfortunately, due to the high temperatures in this region, no flight test flow visualization data has yet been obtained to substantiate this prediction, although the CFD/experimental agreement for the isolated nacelle/pylon is encouraging.

The second area of disagreement between the isolated and installed predictions involves flow characteristics on the core plug. As can be seen from Figure 14, the predicted pressures on the

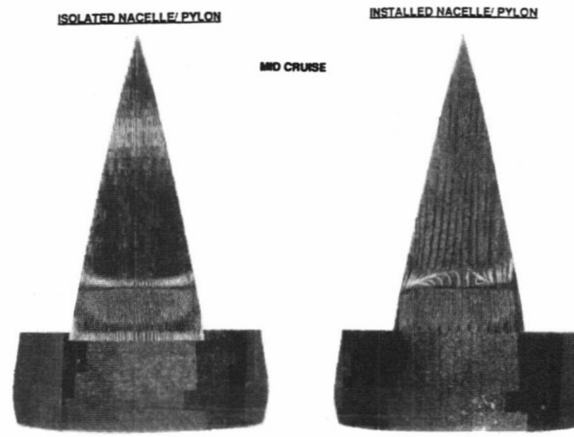

Figure 13. Pylon Shelf Predictions with and without Wing.

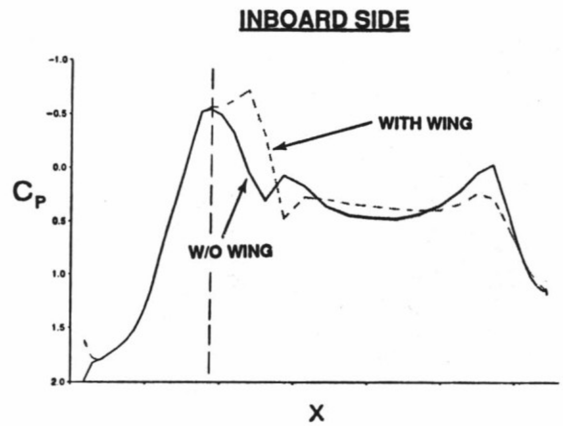

Figure 14. Predicted Wing Influence on Core Plug Pressures.

inboard side of the plug change from a gradual recompression without the wing influence to a distinct shock when the wing influence is incorporated. However, no flow separation is indicated, but a drag/thrust-loss penalty can accrue due to the shock.

The obvious lesson learned from this example is that the influence of the wing flowfield must be accounted for in the design of component geometries in the core exhaust region. Since there is no practical experimental approach to achieve this, short of prohibitively expensive full-scale flight testing, CFD methods must be further validated, and then applied to define geometries that do not contribute avoidable performance penalties.

Wing Development - The wing-upper-surface flow development on subsonic transport aircraft is significantly altered by the nacelle/pylon installation at cruise conditions. While the extent of the nacelle/pylon influence has often been related to the pylon/wing leading-edge geometry arrangement, which is what led to the cutback pylon designs starting with the DC-8-62, there is still a major influence of the nacelle/pylon on the wing-upper-surface flow development even when cutback pylons are employed. And, as would be expected, the influence is even more dramatic with less aerodynamically-desirable pylon designs.

There are two primary effects of a typical wing-mounted nacelle/cutback pylon installation on the wing-upper-surface flow development (and hence drag, etc.) that must be dealt with. These are illustrated in the CFD-predicted isobars shown in Figure 15 for an advanced wing installation at cruise conditions with and without a nacelle/pylon at high Reynolds numbers. The first occurs inboard of the nacelle/pylon where the often-typical isentropic recompression on the basic wing is changed into a drag-producing unswept shock in the presence of the nacelle/pylon. This effect has been observed to a similar extent on both contemporary and advanced wing designs.

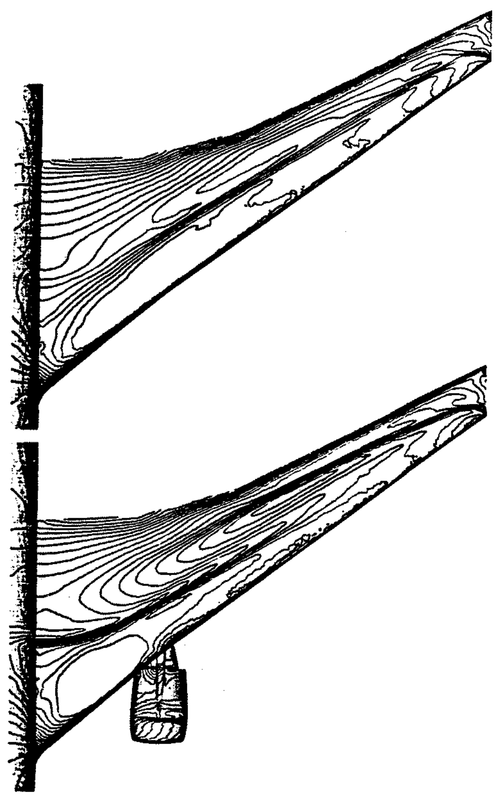

Figure 15. Predicted Nacelle/Pylon Influence on Advanced Wing Isobars.

Outboard of the nacelle/pylon, the nacelle/pylon effect tends to change a single-shock (or isentropic) recompression into a potentially much less efficient double shock, with the second shock located quite far aft. This effect has also been observed on both contemporary and advanced wing designs, but to a somewhat lesser degree on contemporary wing designs. The increased propensity of advanced wing designs toward this behavior is associated with an increased sensitivity of the flow development on these wings to small flow-condition changes. Furthermore, as illustrated in Figure 16, the development and strengthening of the aft shock on advanced wings is further accelerated when the engine size (i.e., bypass ratio) increases. The validity of the CFD-predicted nacelle/pylon impact on the wing flow development is demonstrated by the excellent comparison of the predicted and wind-tunnel measured pressures on the wing with and without the larger nacelle/pylon as shown in Figure 17. The CFD predictions shown in Figures 15-17 were obtained using the unstructured-grid Euler AIRPLANE/IBL code, and the test results shown in Figure 17 were obtained in the NASA Langley 16-Ft TWT at a (mac) Reynolds number of 4 x 10^6 (as are the CFD predictions in Figure 17).

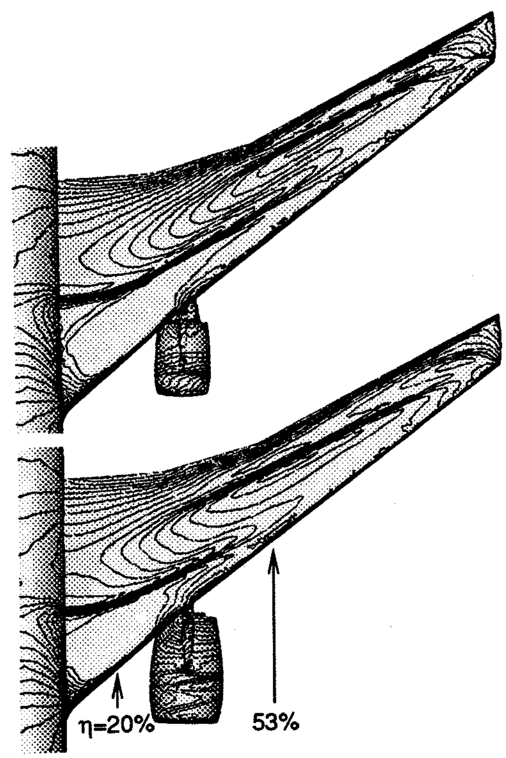

Figure 16. Predicted Nacelle Size Effect on Advanced Wing Isobars.

Some important lessons are apparent from these studies. First, and foremost, it is essential that the nacelle/pylon influence be accounted for in the aerodynamic design of effective subsonic transport wings. Otherwise, the resulting wing flowfield (shock development, etc.) will produce

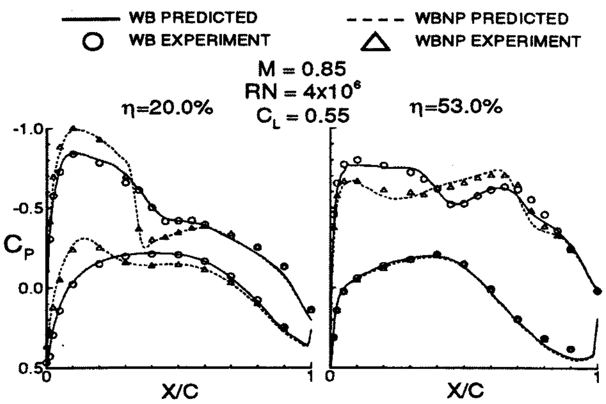

Figure 17. Predicted vs. Wind-Tunnel-Measured Influence of Nacelle/Pylon.

less-than-desired/or possible performance characteristics. This is true for both turbulent and laminar flow control (LFC) wings. And, the only practical means of accounting for the nacelle/pylon influence in the design process will have to be based on the use of inverse CFD methods. However, for the use of inverse methods to be feasible, a sizable CFD-generated database will have to be developed to provide the aerodynamic designer with the information needed to specify attainable target pressure distributions for practical wing designs.

A second important lesson that can be derived from these studies has to do with the importance of simulating flight Reynolds numbers in all phases of the wing design process. Or, if laminar flow designs are desired, appropriate simulation of the laminar flow region is crucial. The use of current CFD capabilities does permit this modeling for (hopefully) attached-flow cruise conditions. In addition, experimental validation of resulting turbulent wing designs must also be conducted at flight Reynolds number conditions (i.e., in either the NTF or ETW) if the development risk is to be satisfactorily controlled.

Future CFD Role

The near-future role of CFD in addressing airframe/engine integration issues for the installation of large, high-bypass-ratio engines on subsonic transports will be to guide the aerodynamic design at cruise conditions by permitting the design of installations that preclude adverse interference effects, and by allowing Reynolds number effects to be estimated (for attached-flow conditions). These CFD applications will require extensive use of inverse design methods for the wing, nacelle, and pylon, and, therefore, will require the establishment of CFD-derived (and experimentally validated) databases that will provide the aerodynamic designer with the guidance needed to specify achievable target pressure distributions for the complex flow situations at hand. This is especially crucial for the successful introduction of any LFC wing or nacelle designs. Two other prerequisites that are required to permit this very practical and useful application of CFD are the establishment of guidelines for avoiding separation onset, and the validation of turbulence models that adequately represent the jet flow mixing process. For the former, this guidance is needed for juncture flows as well.

In addition to the foregoing role of guiding the design of low-drag engine installations at cruise conditions, there are additional roles desired for CFD, such as in multidisciplinary optimization (MDO) to define the best overall aircraft/engine-installation design considering all aspects, and for guiding the design of effective engine/nacelle installations at low-speed, high-lift conditions. However, some major technical obstacles/challenges must be overcome before CFD can be effectively applied for either of these purposes.

Development of the capabilities required to effectively use CFD-derived inputs for MDO involving airframe/engine integration issues for subsonic transports is not viewed as being a practical endeavor for the foreseeable future. There are two major fundamental limitations that will severely limit the usefulness of CFD-derived aerodynamic inputs for MDO at transonic cruise Mach number conditions. The first is the inability to calculate drag with sufficient accuracy and reliability for representative (complex) cruise-configuration geometries. The main culprits in this are the inaccuracies introduced by the artificial dissipation incorporated to facilitate convergence, and the enormous number of grid points required to obtain an accurate surface pressure integration for these 3-D geometries. Unfortunately, there are also additional culprits as well. The second major fundamental limitations is the inability to predict the aerodynamic characteristics at conditions involving flow separation, which typically are most of the critical aerodynamic loads conditions. The main culprit here is the inadequacy of known turbulence models to adequately represent 3-D separated flow conditions, at either low or high Reynolds numbers. Neither of these limitations is expected to be overcome in the foreseeable future.

Although the use of CFD to accurately predict either maximum lift or drag characteristics for 3-D high-lift geometries including installed engine effects is a much greater challenge than for corresponding cruise conditions (which in itself is not achievable in the foreseeable future), the use of CFD to provide useful guidance for the effective installation of large, high-bypass-ratio engines at high-lift conditions is viewed as an achievable objective with very large payoffs. This would, in essence, be the same role that CFD has played in the design of effective transonic installations, although the flow physics to be considered at high-lift conditions are definitely more complex. However, the potential payoff in terms of cutting down on the amount of very expensive and time-consuming high-lift development testing at high Reynolds numbers, including power simulation, would provide significant relief in perhaps the most time-critical element in the subsonic transport design-cycle process. The type of guidance desired would be in terms of assessing possible slat tailoring concepts, nacelle position effects, nacelle strake/location effectiveness, and jet exhaust/flap interaction effects. Euler/IBL methods should be effective for simulating vortex flows and compressibility effects. Navier-Stokes methods will have to be employed to address jet exhaust/flap interaction effects.

Conclusions

The ever-increasing role that CFD has played in the development of effective, integrated wing-mounted engine installations for subsonic transports has been reviewed. Conclusions arising from this review include the following:

- Use of linear full-potential CFD methods provided useful early guidance for the design of nacelle inlet and cowling concepts at low-speed conditions, and for the development of more closely coupled nacelle/wing installations. However, several cases were encountered where these linear methods significantly underpredicted supersonic flow regions.
- Transonic full-potential methods (coupled to boundary-layer methods) provided a major advancement in the ability to predict such nonlinear supersonic flow regions, but a number of examples were encountered where the inability to accurately represent shock characteristics limited the confidence in these methods.
- Unstructured-grid Euler and Euler/IBL methods are seeing a much-increased usage for addressing complex interference problems because of the much-easier (than structured) grid generation process, and the success in accurately representing a wide range of flow situations. Notable exceptions to this success are in addressing exhaust jet (mixing) effects and complex separated flow situations.
- Navier-Stokes methods have been successful in addressing complex airframe/engine integration issues, especially for jet-interference effects, although the elapsed time to obtain a solution for a single geometry still takes significantly longer than desired.
- Current CFD capabilities, involving Euler, Euler/IBL, and Navier-Stokes methods, provide a unique capability to address nacelle/pylon/wing installation issues involving mutual interference effects that heretofore could not be dealt with in a practical way. Three examples of this capability were provided which illustrated the various roles requiring each of the three methods.
- A major emphasis is required to generate the (CFD-developed) databases needed to guide the aerodynamic designer in specifying achievable target pressure distributions when applying the inverse design methods necessary to deal with the mutual interference effects. Most significant of these situations is the ability to account for the influence of the nacelle/pylon on the wing upper-surface flow development.
- Development of the capabilities required to effectively use CFD-derived inputs for multidisciplinary optimization (MDO) involving airframe/engine integration issues for subsonic transports is not viewed as being a practical endeavor for the foreseeable future. Fundamental limitations in CFD capabilities for drag prediction and separated flow representation lead to this conclusion.
- Developments necessary to permit application of CFD to provide useful guidance for the effective installation of large, high-bypass-ratio engines at high-lift conditions is a high priority because of the very large potential payoffs in the subsonic transport design-cycle process.

Acknowledgment

Acknowledgment is given to all of those that have contributed to this on-going effort. Specifically, thanks are extended to the Ray Cosner-led team at MDA-East responsible for the very unique NASTD applications and predictions. Acknowledgment is also given to those at MDA-West responsible for the Euler/IBL and Navier-Stokes PAI applications. Thanks are also extended to the Transonic Aerodynamics Branch at NASA LaRC for their contributions on the pylon fairing study. Finally, the authors also would like to acknowledge the computational resources made available at the Numerical Aerodynamics

Simulator (NAS) and at NASA LaRC to support these efforts.

References

1. Lynch, F.T., "Commercial Transports - Aerodynamic Design for Cruise Performance Efficiency," in Transonic Aerodynamics (D. Nixon, ed.), *Progress in Astronautics and Aeronautics*, Vol. 8, 1982.
2. Kutney, J.T. and Piszkin, S.P., "Reduction of Drag Rise on the Convair 990 Airplane," AIAA Paper No. 63-276, June 1963.
3. Lynch, F.T., "Experimental Necessities for Subsonic Transport Configuration Development," AIAA Paper No. 92-0158, Jan. 1992.
4. Mack, M.D. and McMasters, J.H., "High Reynolds Number Testing in Support of Transport Aircraft Development," AIAA Paper No. 92-3982, July 1992.
5. Haines, A.B., "Scale Effect in Transonic Flow," *Aeronautical Journal*, pp. 291-313, Aug-Sept 1987.
6. Berry, D.L., "Civil Aircraft Propulsion Integration - Current & Future," SAE 932624, Sept 1993.
7. Goldhammer, M.I., "Nacelle and Wing Assembly," US. Pat. No. 4,815,680, Mar 1989.
8. Bengelink, R.L. and Wickemeyer, R.H., "Nacelle-Pylon Configuration for an Aircraft and Method of Using the Same," US. Pat. No. 4,489,905, Sept 1981.
9. Potsdam, M.A., Intemann, G.A., Frink, N.T., Campbell, R.L. and Smith, L.A., "Wing/Pylon Fillet Design Using Unstructured Mesh Euler Solvers," AIAA Paper No. 93-3500, Aug 1993.
10. Rubbert, P.E. and Tinoco, E.N., "Impact of Computational Methods on Aircraft Design," AIAA Paper No. 83-2060, Aug 1983.
11. Chen A.W., Curtin, M.M., Carlson, R.B. and Tinoco, E.N., "TRANAIR Applications to Engine/Airframe Integration," *Journal of Aircraft*, Vol. 27, No. 8, pp. 716-721, Aug 1990.
12. Shmilovich, A., "Calculation of Transonic Flows with Separation Past Arbitrary Inlets at Incidence," *AIAA Journal*, Vol. 28, No. 4, pp. 601-609, Apr 1990.
13. Clem, B.C., Elliott, J.K., Tamigniaux, T.L.B. and Tinoco, E.N., "Recent CFD Applications on Jet Transport Aircraft," AIAA Paper No. 92-2658, 1992.
14. Shmilovich, A. and Halsey, N.D., "Calculation of Transonic Flows for Novel Engine-Airframe Installations," In *Numerical and Physical Aspects of Aerodynamic Flows IV* (ed. T. Cebeci), Springer-Verlag, 1990.
15. Jameson, A. and Baker, T.J., "Improvements to the Aircraft Euler Method," AIAA Paper 87-0452, Jan. 1987.
16. Frink, N.T., "Three-Dimensional Upwind Scheme for Solving the Euler Equations on Unstructured Tetrahedral Grids," Ph.D. Dissertation, Virginia Polytechnic Institute and State University, Sept 1991.
17. Potsdam, M.A., "An Unstructured Mesh Euler and Interactive Boundary-Layer Method for Complex Geometries," AIAA Paper No. 94-184, June 1994.
18. Bush, R.H., "A Three-Dimensional Zonal Navier-Stokes Code for Subsonic Through Hypersonic Propulsion Flow Fields," AIAA Paper No. 88-2830, July 1988.
19. Renze, K.J., Buning, P.G. and Rajagoplan, R., "A Comparative Study of Turbulence Models for Overset Grids," AIAA Paper 92-0437, Jan 1992.
20. Gea, L.M., Intemann, G.A., Halsey, N.D. and Buning, P.G., "Applications of the Navier-Stokes Code OVERFLOW for Analyzing Propulsion-Airframe Integration Related Issues on Subsonic Transports," ICAS 94-2.2.3, Sept 1994.
21. Vatsa, V.N. and Sedan, B.W., "Development of an Efficient Multigrid Code for 3-D Navier-Stokes Equations," AIAA Paper No. 89-1791, 1989.
22. Kao, T.J., Su, T.Y. and Yu, N.J., "Navier-Stokes Calculations for Transport Wing-Body Configurations with Nacelles and Struts," AIAA Paper No. 93-2945, July 1993.
23. Thomas, J.L., Van Leer, B. and Walters, R.W., "Implicit Flux-Split Scheme for the Euler Equations," AIAA Paper No. 85-1680, July 1985.
24. Ostrander, M.J. and Cedar, R.D., "Analysis of a High Bypass Ratio Engine Installation Using the Chimera Domain Decomposition Technique," AIAA Paper No. 93-1808, June 1993.
25. Cedar, R.D., Dietrich, D.A. and Ostrander, M.J., "Engine/Airframe Installation CFD for Commercial Transports: An Engine Manufacturer's Perspective," SAE 932623, Sept 1993.
26. Campbell, R.L. and Smith, L.A., "A Hybrid Algorithm for Transonic Airfoil and Wing Design," AIAA Paper 87-2552, 1987.
27. Smith, L.A. and Campbell, R.L., "A Method for the Design of Transonic Flexible Wings," NASA TP-3045, 1990.
28. Rubbert, P. and Goldhammer, M., "CFD in Design: An Airframe Perspective," AIAA Paper No. 89-0092, Jan 1989.