

CFD Uncertainty and Validation for Commercial Aircraft Applications

Edward N. Tinoco

P.O. Box 3707
Boeing Commercial Airplanes
Seattle, WA 98124-2207, USA
edward.n.tinoco@boeing.com

ABSTRACT

Effective use of Computational Fluid Dynamics (CFD) is a key ingredient in successful design of modern commercial aircraft. Like any source of information for engineering design, uncertainties exist in the predictions obtained from CFD tools. And, as in any engineering endeavour, success is attained by understanding the impact of the uncertainties in CFD predictions on the final product. Processes and techniques used at Boeing to bound CFD uncertainties and limit their impact on design decisions will be described. Key ingredients for success include a clear understanding of the requirements of each design problem, extensive experience in the mechanics and use of CFD modeling tools, availability of a diverse set of CFD modeling tools, standardized methods of using the tools, access to both appropriate test data and computing hardware adequate to define the limits of computational simulation capability and continuous ongoing efforts to establish, refine and extend our collective understanding of both capabilities and limits of CFD simulations.

NOMENCLATURE

α	= angle of attack
AR	= aspect ratio
C_D	= drag coefficient
C_L	= lift coefficient
C_N	= normal force coefficient
C_M	= pitching moment coefficient
C_P	= pressure coefficient
Eta	= span fraction
FNS	= full Navier-Stokes
GRIDSIZE	= number of grid cells
N	= number of grid cells
NS	= Navier-Stokes
Re	= Reynolds number based on mean aerodynamic chord
SA	= Spalart-Allmaras turbulence model
SST	= Menter's Shear-Stress-Transport turbulence model
TLNS	= thin-layer Navier-Stokes
X/C	= wing chord fraction
YEDGE	= Y(buttline) extent of separation bubble

Tinoco, E.N. (2007) CFD Uncertainty and Validation for Commercial Aircraft Applications. In *Computational Uncertainty in Military Vehicle Design* (pp. 20-1 – 20-36). Meeting Proceedings RTO-MP-AVT-147, Paper 20. Neuilly-sur-Seine, France: RTO. Available from: <http://www.rto.nato.int>.

CFD Uncertainty and Validation for Commercial Aircraft Applications

1.0 INTRODUCTION

Effective use of Computational Fluid Dynamics (CFD) is a key ingredient in successful design of modern commercial aircraft. The combined pressures of market competitiveness, dedication to the highest of safety standards, and desires to remain a profitable business enterprise all contribute to make intelligent, extensive and careful use of CFD a major strategy for product development at Boeing. The purpose of CFD in industry is to produce a better product. To this end, CFD is used when the design engineers and their management believe that its use will lead to that better product within the allotted budget of time, money, and effort allocated for the project. The application of CFD today has revolutionized the process of aerodynamic design. CFD has joined the wind tunnel and flight test as primary tools of the trade.¹⁻⁴ Each has its strengths and limitations. Because of the tremendous cost involved (and potential risk) in flight testing, modern aircraft development places major focus on the use of CFD and the wind tunnel prior to flight. The wind tunnel has the advantage of dealing with a “real” fluid and can produce global data, cheaper, and over a far greater range of the flight envelope than can CFD. It is best suited for validation and database building within acceptable limits of a development program’s cost and schedule. Historically, CFD has been considered unsuited for such as task. However, the wind tunnel typically does not produce data at flight Reynolds number, is subject to sometime significant wall and mounting system corrections, and is not well suited to provide flow details. The strength of CFD is its ability to quickly produce a small number of simulations leading to understanding necessary for design. Of great utility in this connection is the fact that CFD can be used in an “inverse design” or optimization mode, predicting the necessary geometry shape changes to optimize certain flow characteristics or a payoff function (e.g., drag). Beyond this, CFD is heavily used to provide corrections for the extrapolation of data acquired experimentally (typically from testing a reduced scale model of the vehicle in a wind tunnel) to conditions that characterize the full-scale flight vehicle. Finally, CFD is used to provide understanding and insight as to the source of undesirable flight characteristics, whether they are observed in subscale model testing or in the full-scale flight testing. Collectively, flight test, wind tunnel testing, and CFD all contribute to minimizing risk and uncertainty in a new airplane product.

Experience to date at Boeing Commercial Airplanes has shown that CFD has had its greatest effect in the aerodynamic design of the high-speed cruise configuration of a transport aircraft. The advances in computing technology over the years have allowed CFD methods to affect the solution of problems of greater and greater relevance to aircraft design. Use of these methods allowed a more thorough aerodynamic design earlier in the development process, permitting greater concentration on operational and safety-related features.

The 777, being an all new design, allowed designers’ substantial freedom in the early 1990’s to exploit the advances in CFD and aerodynamic design. High-speed cruise wing design and propulsion/airframe integration consumed the bulk of the CFD applications. Without the aerodynamic performance improvements enabled by the application of “inverse” wing design, the 777 would not have achieved the market dominance it enjoys today. The “inverse” wing design process required confidence that CFD could adequately predict wing pressure distributions in sufficient detail. Airframe/engine integration required an adequate prediction of installed powered exhaust effects. Many other features of the aircraft design were also influenced by CFD. For example, CFD was instrumental in design of the fuselage cab. CFD augmented wind tunnel testing for aft body and wing/body fairing shape design. Here CFD provided insight and guided the design process through the calculation of pressure distributions and streamlines. In a similar fashion, CFD augmented wind tunnel testing for the design of the flap support fairings. The wind tunnel was used to assess the resulting drag characteristics. CFD was used to identify prime locations for static source, sideslip ports, and angle-of-attack vanes for the air data system. CFD was used for design of the environmental control system (ECS) inlet and exhaust ports and to plan an unusual wind tunnel evaluation of the inlet. The cabin (pressurization) outflow valves were positioned with the aid of CFD. Collaboration between the wind tunnel and CFD involved the

CFD Uncertainty and Validation for Commercial Aircraft Applications

use of CFD to determine and refine the corrections applied to the experimental data due to the presence of the wind tunnel walls and model mounting system. Today, CFD is routinely used to provide such corrections to wind tunnel data. More recently in the 777 program multipoint CFD design/optimization was used to design the fan cowl for a new engine variant. The design was accepted and committed to production prior to any wind tunnel confirmation!

The Next-Generation 737-700/600/800/900, being a derivative of earlier 737s, presented a much more constrained design problem. Again the bulk of the CFD focused on cruise wing design and engine/airframe integration. Although the wing was new, its design was still constrained by the existing wing-body intersection and by the need to maintain manual control of the ailerons in case of a complete hydraulic failure. As with the 777, CFD was used in conjunction with the wind tunnel in the design of the wing-body fairing, modifications to the aft body, and design of the flap track fairings and the high-lift system. Recently CFD was used to develop a redesign of the thrust reverser cascades to reduce the ground spray into the landing gear wells during roll-out upon landing on snowy/wet runways. Full scale ground testing proved the adequacy of the new design which is now certified and being incorporated into new airplane production.

The Sonic Cruiser program would have been inconceivable without the use of CFD. Wind tunnel model aeroelastics, lack of space for extensive pressure instrumentation, and tunnel interference near Mach one were forcing unprecedented reliance of CFD by the Loads and Stability and Control communities. John Roundhill, then vice-president of marketing, was quoted in the popular press as saying, “The use of CFD has enabled Boeing to overcome the greatest single roadblock to building a fast plane.” Unfortunately CFD alone could not overcome the real world market economics of that concept.

The new 787 Dreamliner development has seen an unprecedented level of CFD involvement as illustrated in Figure 1⁴. Multipoint optimization employing CFD was used extensively in the design of the wing, nacelles and forebody. To quote Walt Gillette, then vice-president of 787 engineering, speaking about 787 efficiency⁵ “the company will eke 2-3% out of the new airplane’s aerodynamics thanks to advanced computational fluid dynamics ... Now CFD codes are advanced enough to analyze airflow over the entire aircraft, and aerodynamicists can propose the result they want and let the computer propose the shape.”

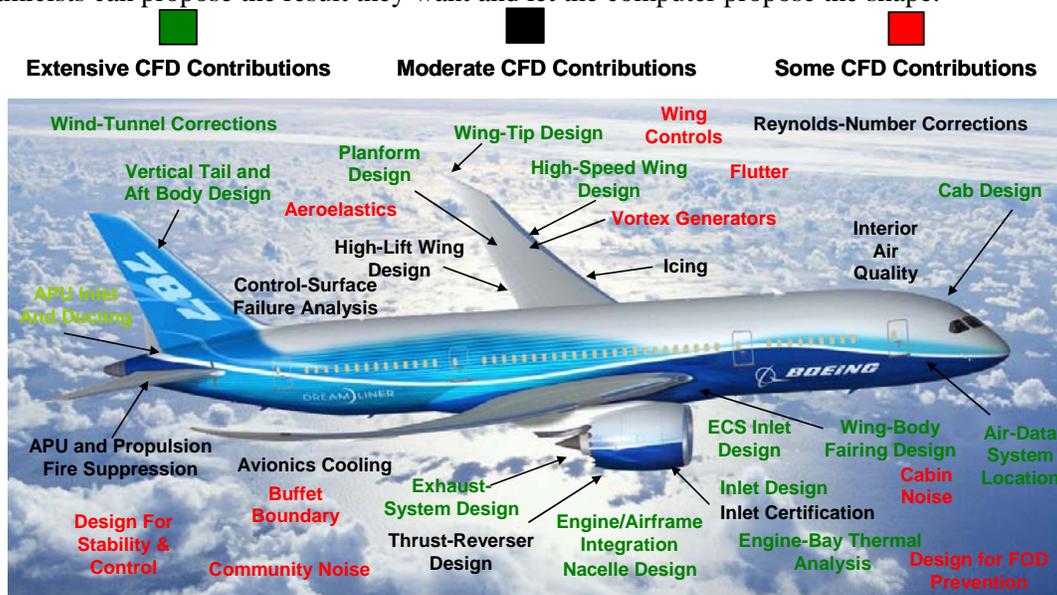


Figure 1. CFD Contributions to the 787.

CFD Uncertainty and Validation for Commercial Aircraft Applications

The role of CFD in aircraft development only continues to increase as evidenced by its use in the major derivative 747-8 program. The constraints of a derivative program and the pressures on development cycle time and cost compound the demands on CFD. Having an efficient CFD design process allows for frequent updates in response to the ever changing requirements as a new airplane design matures. The advances in CFD are now allowing a greater penetration in the airplane development process than just high-speed lines design⁶. While the wind tunnel is still used primarily for the extensive database collection over the entire flight envelope, CFD is being used for wind-tunnel corrections, dynamic derivatives, aeroelastics, Reynolds-number effects, configuration trade studies, hinge moments, engine inlet and exhaust nozzle design, thrust-reverser/airframe compatibility, etc.

Since the development of the 767 airliner in the late 1970's, the necessary wind tunnel occupancy hours for an all new airplane development program have been cut essentially in half.⁴ Together CFD and the wind tunnel are being used to limit risk and bound the aerodynamic uncertainties of a new airplane development. CFD enables more focused wind testing. Test matrices are reduced and CFD used to "fill-in" the gaps. In some instances wind tunnel tests are eliminated entirely relying instead on the predictive capabilities of CFD! This reduction in testing time has largely been enabled by the maturity and "calibration" of advanced CFD methods, and by the realization that the risk and uncertainty with these applications is on par or lower than just relying on testing alone. These advances in developing and using CFD tools for commercial airplane development have produced significant saving for Boeing over the past 20 years. The reduction of wind tunnel testing not only reduces cost but more importantly reduces development cycle time which enables faster market readiness. However, as significant as these savings are, they are only a small fraction of the value CFD delivered to the company.

A much greater value of CFD in the commercial arena is the added value of the product (the airplane) due to the use of CFD. Value to the airline customer is what sells airplanes! Value is added to the airplane product by achieving design solutions that are otherwise unreachable during the fast-paced development of a new airplane. Value is added by shortening the design development process. Time to market is critical in the commercial world, particularly when starting after a competitor has committed a similar product to market, or in maintaining an advantage after an aggressive competitor enters a market. Aircraft development in the large commercial aircraft industry is somewhat different from that in the military arena. The time scales from program launch to in-service deliveries are much shorter, frequently less than five years!

Very important in the commercial world is getting it right the first time. No prototypes are built. From first flight to revenue service can be less than one year! Any deficiencies discovered during flight test must be rectified sufficiently for government certification and acceptance by the airline customer based on a schedule set years before. Any delays in meeting this schedule may result in substantial penalties and jeopardize future market success. The competitive market place allows your customers to go elsewhere if your product does not meet their expectations. The McDonnell-Douglas MD-11 was not a bad airplane. It missed its commitments by what used to be considered a small margin. By the time it had made up most of its performance shortfall it had lost the market to the Boeing 777 and the Airbus A340. To be of value, CFD must be able to play a role in reducing development cycle time and "getting it right the first time" so that schedule and performance commitments can be met at the time of delivery into service. The added value to the airplane product will produce increased sales and may even open up completely new markets. The result is more profit to both the buyer and seller (who does not have to discount the product as much to make the sale). All this translates into greater market share.

2.0 PREREQUISITES TO THE USE OF CFD IN AIRCRAFT DEVELOPMENT

As previously, stated the purpose of CFD in industry is to produce a better product. To this end, CFD is used when the design engineers and their management believe in the need, and that its use will lead to that better product within the allotted budget of time, money, and effort for the project. In other words, **desperation** is frequently the catalyst for a new application of CFD! Desperation led to the development and adaptation of “inverse” wing design in the late 1980’s. The existing “cut and try” wing design along with CFD assessment was not meeting the performance goals set out for a new airplane development at that time. “Inverse” design required “validation” of the adequate predictive capability of wing pressure distributions. Desperation led to the development and adaptation of wing design/optimization in the late 1990’s. The existing “inverse” wing design process was not meeting the performance goals set out for the four-engine aircraft configurations being investigated at that time. Design/optimization required “validation” of the predictive drag of complete aircraft configurations. A tremendous amount of money and experience has been invested in the existing processes for aircraft design so changes to these methods are not taken lightly.

Before committing to the use of CFD, management must believe that it makes economic sense; it must deliver the “most bang for the buck.” It must reside in the engineering unit responsible for the task at hand. This means that the CFD must be usable by the engineers on the project and that the management has confidence that the both the CFD codes and the engineers are “validated” for the task. This requires that the CFD tools and the ability to use them is in place when needed. Achieving this takes more than just codes. Product Development engineers must be able to focus on engineering processes and have little time for manipulating the CFD “process”, i.e. codes must be very user oriented. What is needed are engineering solutions: Stable, packaged software solutions that promote and enable consistent, repeatable processes. These not only put CFD into the hands of the Product Development / Product Support engineers but also allow the “expert” user to get fast results with reduced variation. This requires a long term relationship between CFD tool supplier and customer. Future needs must be anticipated in order to have pre- and post-processing tools, validation, documentation, and user training and experience available when needed.

2.1 Verification, Validation, Calibration, and Certification?

For engineers and their management to have the confidence to “bet” a multi-billion dollar airplane development program on CFD being “close-enough” requires an ongoing process of verification, validation, calibration, and certification. Verification and validation to some extent have been consistently defined in several references including an AIAA Standards Guide.⁷ From a recent update from the AIAA committee on Standards for computational fluid dynamics⁸ we see *verification and validation of CFD codes and calculations as the process of determining the level of confidence that can be placed in the resulting CFD data where:*

- **Verification:** The process of determining that a model implementation accurately represents the developer’s conceptual description of the model and the solution to the model.
- **Validation:** The process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model.

Here the definition of validation is too general and yet not encompassing enough! It is too general in that every intended use of a code can have a different level of required accuracy which is specific not only to the use but also to the user. The definition is not encompassing enough because it is not just the code that needs validation, but the entire CFD process including the user! Cosner, et. al. in reference 8 suggest that the

CFD Uncertainty and Validation for Commercial Aircraft Applications

meaning of “validation” be limited to the process of determining the degree to which CFD results agree with experimental data. In this sense, verification and validation are the necessary first steps to evoke an interest in a specific CFD code possibly being of value for specific intended use. The follow-on work to determine the degree to which results from a CFD code are an accurate representation of the real world from the perspective of the intended uses of that code has been referred to as “calibration”, “certification”, or just further validation. Melnik, et. al, in Reference 9 describe a process for “certification” which is indicative of our current practices. The reality is that these processes for determining usability of a CFD process for an intended purpose are “...ongoing activities that do not have a clearly defined technical completion point.”⁸ Rather than referring to these processes as “calibration” or “certification” we will just continue to call them “validation” – validation for an intended purpose. The challenge is to get from the stage where the initial verification and validation has shown a new CFD code to have potential in meeting some engineering need to actually making it part of your engineering process.

2.2 Validation for an Intended Purpose.

The aerodynamic development of an airplane configuration is not an exact science; it is rather a compromise involving several competing disciplines and interests. The challenge is to provide sufficient credible information so that intelligent trade-offs, decisions, and compromises can be made. CFD tools, like wind tunnels can be used to explore and bound that design space. Both CFD and the wind tunnel use models to develop the required engineering data. Neither one represents the reality of the flight article. Decades of experience have gone into wind tunnel testing to develop the expertise to bound the test uncertainties and limit their impact on the design decisions. In similar fashion that same level of expertise must be developed and applied to CFD modeling.

An essential criterion for use of CFD is user confidence, a characteristic that is difficult to attach a numerical value. "User confidence" means not only that the CFD code can adequately model the flow physics of interest, but also produce solutions that are “close enough”, and it must be able to do it rapidly. Validation of a “CFD process” cannot be limited to the solver alone but must include the entire process of geometry, grid generation, solver, and post-processing for going from “lofts to plots.” For this paper we will limit our discussion of CFD validation to include just the grid generation / flow solver combination, although for the user the entire “lofts to plots” process must be usable in a timely manner. In addition we will also limit our discussion to transonic flight conditions of cruise type commercial configurations.

The definition of "close enough" depends on the application. Like any source of information for engineering design, uncertainties exist in the predictions obtained from CFD tools. And, as in any engineering endeavour, success is attained by understanding the impact of the uncertainties in CFD predictions on the final product. The type of generic validation described in References 7 and 8 is of little value for instilling that user confidence. It is, however, very necessary for sparking the interest for doing the follow-on validation to gain that user confidence in the first place. The end user must have the confidence of “close enough” in predicting the variables of interest on the vehicles of interest. This “close enough” of CFD will be assessed in comparison to experimental data which the user already believes is “close enough.”

In assessing the accuracy of CFD, wind tunnel results are frequently used as the “Gold Standard.” However, even if we had the “perfect CFD code” we would not, nor should we get perfect agreement with wind tunnel results because the wind tunnel is also imperfect. Each wind tunnel experiment has its own imperfections which also must be understood or at least recognized in order to assess or validate the CFD. Effective CFD

validation requires intimate knowledge of both the CFD and the experimental data being compared. CFD validation cannot consist of the comparison of the results of one code to those of one experiment. Rather it is the agglomeration of comparisons at multiple conditions, code to code comparisons, an understanding of the wind tunnel corrections, etc. that leads to the understanding of the CFD uncertainty and validation of its use as an engineering tool¹⁰.

Making pointwise comparisons and looking for a close correlation in the results is not a good approach to understand and validate the correlation between the experimental and computational CFD models. And yet that is the context in which nearly every CFD comparison with test data is made. The CFD models define Reynolds number, Mach number and angle of attack to fifteen or more digits of precision and then look for correspondence of pressures and/or forces and moments to four significant figures when knowledge of the parametric test conditions (Mach, angle-of-attack, etc.) may only be good to two or three significant figures. The result is of course highly configuration and test dependent. If the sensitivity of the quantity that is being compared is very low to changes of this order in the parameter values, very good correlations may be experienced. If the sensitivity is very high, the correlation may be very poor. An example of a case where shock position was very sensitive to flow parameters will be shown later in this paper in section 3.2.2.

It is important to realize that the experimental data represents a sample of the behavior of the system (model and test facility) at some imprecisely known values of test parameters. If the experimental data are taken over a range of these parametric values and the CFD calculations are also performed over the same range, we have an opportunity to look for correlations in the experimental/computational models on a region-by-region basis and possibly expose likely systematic errors. In other words, you want to make a lot of comparisons over a range of flight conditions of interest. The question should be are the CFD results “close enough” over a range of conditions, not how well do they agree on a point by point basis for a limited number of comparisons. The lack of perfect agreement by the CFD with the experiment does not mean that the CFD is solely in error. The experiment is never absolute and is subject to its own set of errors. If the CFD correlations are “close enough” over a range of conditions and if these “close enough” correlations are seen for other similar sets of similar configurations and tests which the user believes are “close enough” then the user can have the necessary confidence in the CFD to make engineering decisions that can impact millions of dollars.

A CFD code by itself can never be validated for an intended use because there are just too many variables. These variables include those user accessible “knobs” within the solver that control the discretization, dissipation, convergence strategy, turbulence model, etc. and the BIG external variable called the grid. As a minimum it is the process of how a particular CFD solver is run along with its related grid generation that must be validated. And as previously stated, using wind tunnel data for CFD validation is best done on a trend and increment basis which means we will be running CFD to compare those trends and increments. The question is not can CFD give me a great answer for one or two test cases, but can the CFD “processes” give me good answers for a range of cases when run by a competent engineer? This is what validation for an intended purpose is all about¹¹.

3.0 ADVENTURES IN VALIDATION

The following examples illustrate some of our experiences in minimizing CFD uncertainty and validation.

3.1 The CFD Codes

It is extremely helpful to have several CFD methods available in any validation exercise. Code-to-code comparisons can be particularly illuminating. The user must have a good understanding of the underlying flow

CFD Uncertainty and Validation for Commercial Aircraft Applications

physics modeling of the methods to make meaningful code-to-code comparisons and understand to what extent the results should agree with each other. Code-to-code comparisons give some measure of the variation to be expected due to the underlying numerical solution process, turbulence modeling, convergence characteristics, etc.

The dominant CFD methods currently in use in Boeing Commercial Airplanes for most transonic cruise design and analysis that meet the "close enough" and usability criteria by the product development community are briefly described below. The validation issues with these methods apply to the whole range of CFD for the most part.

TRANAIR – TRANAIR^{12,13} is a three-dimensional finite element full potential flow code directly coupled with an integral boundary layer method. The code employs adaptive field grids that are locally refined Cartesian meshes that intersect, rather than conform to configuration surfaces. The boundary layer method is two-dimensional with wing sweep and taper modifications that allow it to be applied in strips for three-dimensional problems. A lag-dissipation turbulence model, which employs a half-equation approximation to the Reynolds stress-transport equation, is used. Equilibrium values of the shear stress coefficient have been calibrated on both favorable and adverse equilibrium boundary layer flows. Nonequilibrium (lag) behavior has been calibrated with measured boundary layer thickness/mixing length ratios and turbulent kinetic energy/Reynolds stress ratios. An inexact Newton method is applied to the full system of equations and results in rapid convergence of the coupled system. TRANAIR is at the core of a powerful multipoint design/optimization process. A unique feature of TRANAIR that contributes to its reliability and accuracy and makes it extremely useful for CFD validation is its solution adaptive grid capability. It knows where to put the grid thereby largely eliminating grid placement as a source of variation in the solution. However, the limitations of full potential with coupled boundary layer limit its use to essentially attached flows.

CFL3D/TLNS3D - CFL3D originated from the NASA Langley Research Center.¹⁴ The Reynolds-averaged Navier-Stokes (RANS) equations are solved using Roe's upwind scheme. A finite-volume method is employed to construct both viscous and inviscid fluxes, and a multigrid scheme is used to accelerate convergence rate. The code was originally developed with the thin-layer approximation, but it has since been extended to include the full Navier-Stokes terms. TLNS3D also originated from the NASA Langley Research Center.¹⁵ Thin layer Reynolds averaged Navier-Stokes equations are solved using a five stage Runge-Kutta time integration scheme. A finite volume scheme based on Jameson's approach is employed to construct both viscous and inviscid fluxes. Matrix dissipation is added to enhance numerical stability, and a multigrid scheme is used to accelerate convergence rate. We find CFL3D to be the more accurate code but TLNS3D is more tolerant of the gridding details. Due to the use of a higher order upwind scheme in the CFL3D code, this option requires a high-quality Navier-Stokes grid in order to obtain a reliable converged solution. Hence, a lot of effort over the years has gone into learning how to generate these high-quality multi-block structured grids for the type of configurations of interest¹⁶.

OVERFLOW – OVERFLOW¹⁷ was developed by NASA with numerous contributions from academia and industry. It is a node-based RANS code specifically designed for structured overset grids. Like CFL3D, it is a general purpose 2D/3D solver capable of simulating steady or unsteady flows. The code can be run with central differencing with Jameson 4/2 dissipation or Roe upwinding. A number of turbulence models are available. Grid sequencing and multigrid are implemented for convergence acceleration. The use of an overset "chimera" grid structure allows a significant flexibility of adding new geometric features or including geometric details that are difficult to accommodate in the structured grid CFL3D/Zeus system.

We typically use either the Spalart-Allmaras (SA)¹⁸, or Menter's two-equation Shear-Stress-Transport (SST)¹⁹

turbulence models with the RANS codes. The RANS formulation enables solutions with significant flow separation extending its applicability to flight conditions beyond the capability of TRANAIR⁶.

Some form of these four CFD codes have been in use at Boeing for at least 15 or more years. These codes work! The RANS codes have been run thousands of times in Boeing Commercial and TRANAIR over one hundred thousand times for 3-D viscous solutions over transport airplane configurations. This level of code usage is a result of the user community believing that a validated “system” or “process” going from “lofts to plots” is in place for the intended use, that results will be “good enough”, and that sufficient expertise is available to guide the user if necessary.

3.2 Knowing the test data

In Boeing Commercial Airplanes we are blessed with an abundance of wind tunnel data on airplane configurations of interest to our user community. Because we do our own testing and make most of our wind tunnel models we have available detailed knowledge of the entire testing process. These data have been deemed “close enough” for their intended engineering use in the product development or support activities. For several configurations, multiple data sets have been taken, in separate tunnel entries in the same facility, and in multiple facilities. The availability of these multiple data sets allows us to develop a deep understanding of that data and the variability due to the testing environment. During the development of a new production airplane, several different yet similar wing designs will be tested on the same body during the same wind tunnel entry. This allows for the experimental determination of incremental effects between the different configurations with a high degree of accuracy. Test conditions and the corrections applied to the wind tunnel data are also well known. One drawback however is that these tests have been conducted to support product development or support activities and not to acquire data for CFD validation. Configuration force and moments, wing pressure data, sometimes body, horizontal and vertical tail pressure data, and some surface flow visualization are usually available. Detailed boundary layer information is essentially non-existent.

Characteristics and some testing experiences in four transonic wind tunnels are described in References 20 to 24. These four wind tunnels, NASA Ames 11-by 11-Foot Transonic Pressure Tunnel, 2.5 meter National Transonic Facility at the NASA Langley Research Center, Arnold Engineering and Development Center 16T (16 foot transonic tunnel), and the Boeing 8 x 12 foot Transonic Wind Tunnel are considered among the best in the world. Drag prediction intervals have been demonstrated to be in the order of 1 to 2 drag counts, and with sufficient repeat runs, increments between similar configurations are believable down to tenths of a drag count! However, testing the same model in different wind tunnels will still show differences in drag levels of several drag counts²⁰. From these and other references and from personal experience it is evident that these premier wind tunnel facilities do not deliver pristine, perfectly uniform airflow to the model in the test section. Measurements made in the empty tunnel show variations in flow directions and pressure in all three directions^{21,22}. The inclusion of the wind tunnel model in the test section adds further tunnel induced perturbations.

3.2.1 Wind Tunnel Data Corrections

Typical wind tunnel data corrections applied to test data include corrections to model attitude (angle-of-attack), corrections to the model forces and moments, and corrections to the wind tunnel operating conditions^{20, 25}. These corrections are facility dependent. The angle-of-attack seen by the model is not a direct measurement but a derived quantity that includes an upflow correction and a lift interference correction which are added to the measured quantity which, in turn, can have its own variability. The upflow correction

CFD Uncertainty and Validation for Commercial Aircraft Applications

is model specific and represents the integrated flow perturbation in the lift direction. The upflow correction usually varies with Mach number. The lift interference correction is due to the lift induced flow angularity due to the presence of the wind tunnel walls. This correction is better described in Reference 25 and in its simplest form consists of a lift interference parameter, the ratio of wind tunnel model reference area to wind tunnel cross-sectional area, and lift coefficient. The lift interference parameter can be computed with CFD²⁵, and is a function of Mach number and the details of wall porosity in a ventilated transonic wind tunnel. These two corrections can easily total 0.2 to 0.4 degrees at cruise lift coefficients. Because angle-of-attack is a derived quantity we will frequently compare CFD and wind tunnel model parameters such as wing pressure distributions on an equal lift basis rather than equal angle of attack. Lift or normal force is more a direct measurement than is angle-of-attack.

Mach number in the tunnel is determined by measuring total and static pressure. The static pressure measurement may not correspond to that of the centerline of the test section and is therefore subject to a correction that may vary with Mach. The addition of the test model may also further change the static pressure measurement and require a “Mach blockage” correction.

Corrections to model forces and moments include but are not limited to; clear tunnel buoyancy (affects drag), solid blockage induced buoyancy (affects drag), and model cavity pressure effects (affects drag, lift, and pitching moment). These corrections which are model and tunnel specific can add up to several drag counts. Clear tunnel buoyancy accounts for the clear tunnel variation of static pressure over the length of the model. Solid blockage induced buoyancy is due to the additional variation of static pressure induced by the model in the tunnel (that would not exist in free air). Model cavity pressures induce forces on a wind tunnel internal strain gauge force balance.

The model support mechanisms add their own flow perturbations. These too are facility and model dependent. These tare and interference effects require testing the model with different combinations of mounting systems and are not typically done for most testing due to its high cost. CFD has been used to give first order estimates of these effects. In addition to the facility and model support induced flow perturbations there is question about the exact shape of the model that is being tested. Although high speed model wings are usually made of high strength steel and the entire model must have a significant strength safety factor before being allowed in the wind tunnel they are nevertheless flexible. We do have systems to measure deflections during a test but the accuracy of these system decreases as we approach the tip of a high aspect ratio swept tapered wing. It is toward the tip where the aeroelastic deflections will be the greatest and most significant.

All of this is not to say that wind tunnel testing is too inexact. Years of experience have taught us how to correct and compensate for these deficiencies. What this does say is that wind tunnel results do not represent a “free-air” solution about a rigid airplane, which is what we usually represent by our CFD. There should not be perfect agreement between the CFD and wind tunnel measurements! Can we improve the agreement by including the wind tunnel and mounting system in our CFD model? The answer is yes if we knew all the necessary boundary conditions, but is it worth it? Certainly not as a routine way of doing CFD. However, it has been useful as a way of developing better corrections to apply to the wind tunnel results²⁵. Decades of experience, extreme attention to details of model design, data acquisition, correction, and tunnel operation has led to high quality, consistent, repeatable flow simulations that are accepted with a high degree of confidence. Wind tunnels do not produce absolute estimates of aerodynamic characteristics. They do a very good job of producing trends and increments. **Using wind tunnel data for CFD validation is best done on a trend and increment basis.** As previously stated, effective CFD validation requires intimate knowledge of both the CFD and the experimental data being compared.

3.2.2 Poor Validation Choice

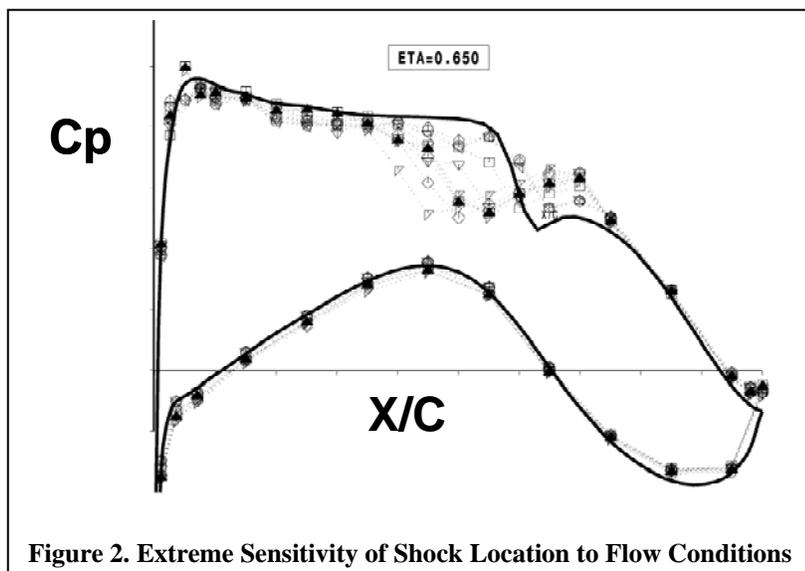


Figure 2. Extreme Sensitivity of Shock Location to Flow Conditions

During the early 1990's a new CFD code under development was being evaluated for transonic cruise analysis and potential use in inverse wing design. A key requirement for use in inverse wing design is the ability to accurately predict wing pressure distributions including the “rooftop” suction levels and the shock location. In keeping with our validation philosophy, comparisons were made with wind tunnel data from several different models. The agreement with test data was “good enough” on all but one case, Wing-Body Configuration A. Development and improvements to the code continued for another year but the resulting solution of

this particular case was still missing the shock location by more than 10% chord. The code developers, who had picked this case as one that should be of interest to demonstrate the wonderfulness of their new creation, were in a deep state of despair when after a year they still could not show agreement with the test data. This particular wind tunnel model had been tested four different times. In each entry there were usually repeat runs at essentially the same flow conditions, angle-of-attack increments, etc. Looking at the test data from these multiple repeats revealed that the code developers had made a very unlucky choice in picking the flow condition for comparison from one of the test entries. Pressure distributions were typically acquired at 0.2 to 0.4 degree increments at a given Mach number. The angle-of-attack chosen for the comparison happened to be one for which the shock location was extremely sensitive to the exact flow conditions (and who knows what else) as illustrated in Figure 2. This characteristic was not at all evident from looking at data from just one run (angle-of-attack series). Only by looking at data from other repeats did we see just how unlucky the particular choice of that case was for CFD validation. For that nominal condition of Mach and angle-of-attack, the shock location on the outboard part of the wing could vary by nearly 30% chord! At half a degree over or under the shock location from the entire set of test runs at the same nominal condition varied by less than 2-3% chord. The force data showed good repeatability over the four tunnel entries. This case was a very poor choice for validation due its extreme sensitivity to the exact flow conditions in the tunnel.

3.2.3 Mach Blockage

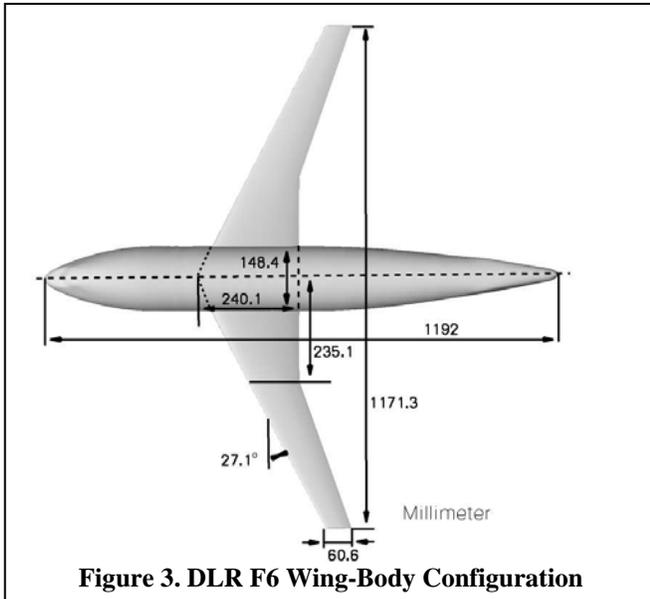
In another example CFD was found to agree better with the test data if it were run at a Mach number 0.01 higher than the “free air corrected” test data. The initial attempt at CFD correlation showed poor agreement with the test data for the large half model being tested. The large model size had been chosen to maximize the chord Reynolds number. A multi-organizational team including CFD developers, members of the wing design group, and members of the wind tunnel testing support staff was formed to look at this issue. Only after a lot of investigation and with the aid of CFD was it shown that the process for determining the “Mach Blockage” - the correction to Mach number needed to be revised. Applying the revised “Mach Blockage” correction to the entire data set resulted in improved agreement with CFD across the board. The “free air corrected” Mach number should have been 0.01 higher at the condition being considered¹⁰. The investigation

CFD Uncertainty and Validation for Commercial Aircraft Applications

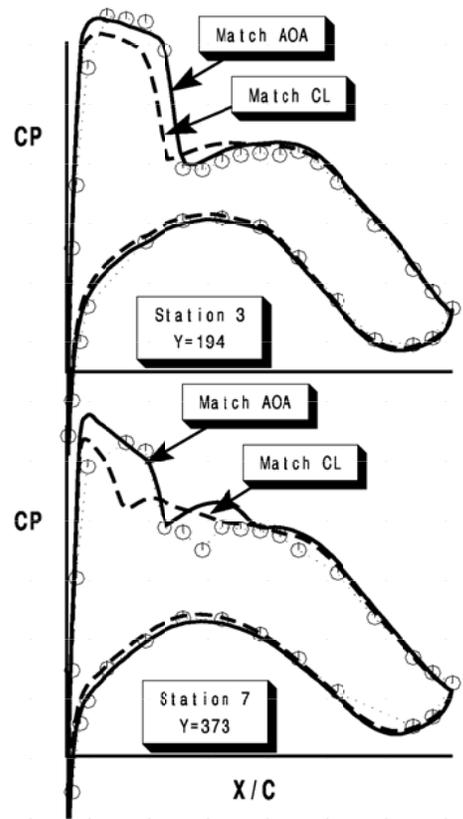
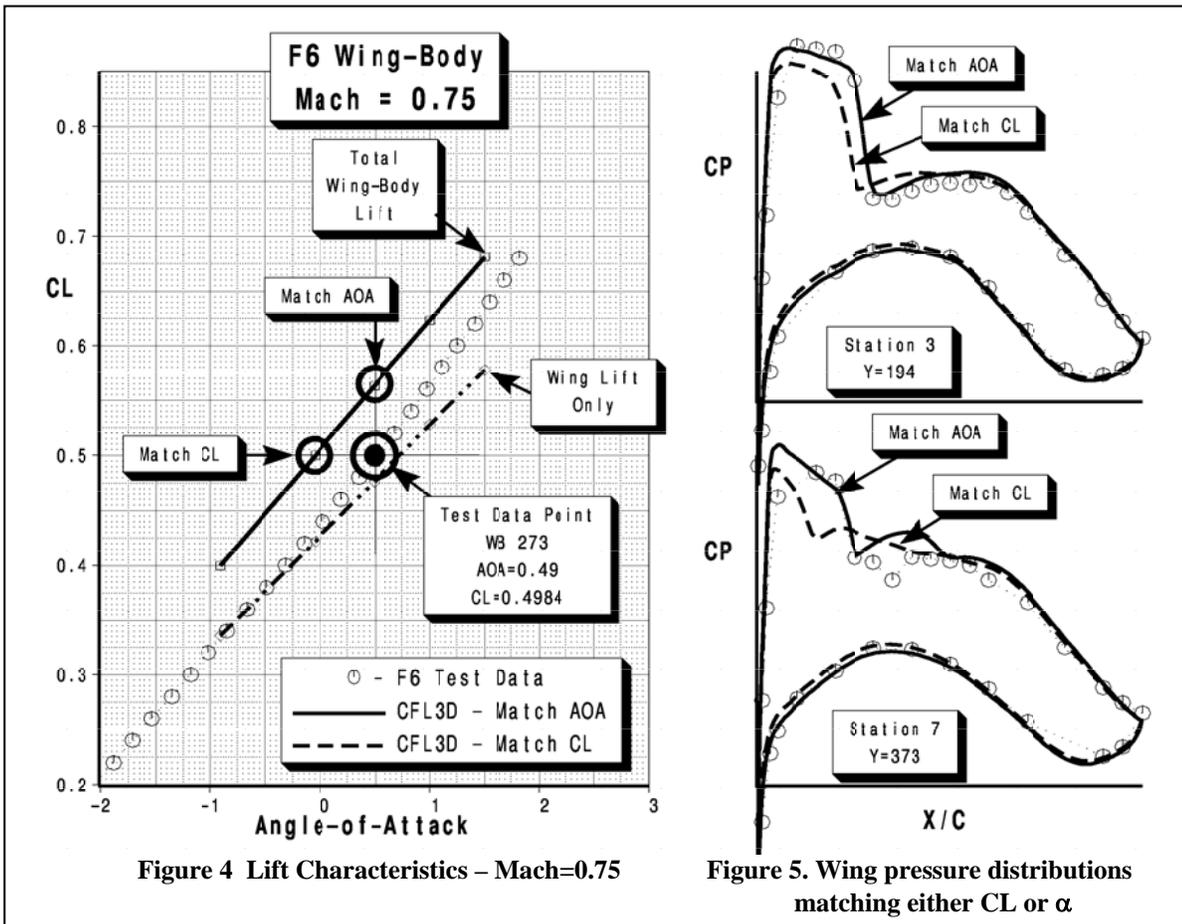
was successful because it involved all the principle parties and included detailed access to the wind tunnel data. The lessons learned from this investigation included an improved process for determining the “Mach Blockage” correction to Mach number, as well as the realization that the model size may have been excessive for the wind tunnel.

3.2.4 Angle of Attack or Lift Coefficient?

When only limited data are available from a wind tunnel test it might not be possible to resolve the discrepancies with CFD. The Second AIAA Drag Prediction Workshop²⁶ asked for comparisons of CFD results with test data taken in the ONERA S2M Wind Tunnel on the DLR F6 configuration illustrated in Figure 3. Geometry of the configuration and a very limited set of wind tunnel data were provided by the workshop sponsors. The test data included all the standard corrections for that facility. Comparisons of lift coefficient vs. angle-of-attack showed that at fixed lift, most participants’ under-predicted angle-of-attack by nearly 0.4 degrees²⁷. Why? A 0.1 to 0.2 degree discrepancy is not unusual given the uncertainty of wind tunnel corrections, CFD grids, turbulence models, convergence, etc. A 0.4 discrepancy seemed rather large. In any validation exercise all the correlations should make sense. Pressure measurements and correlations should be consistent with the force measurements and correlations.



Wing pressure distributions were compared to test data at the primary test point matching either the experimental configuration lift coefficient ($CL=0.50$) or matching the stated angle-of-attack ($\alpha=0.49$)²⁸. Pressure distributions at two wing stations and the corresponding lift data are shown in Figures 4 and 5. Matching lift coefficient resulted in a pressure distribution significantly different from the test data and resulted in a predicted angle of attack 0.365 degrees lower than stated for the experiment. Matching angle of attack resulted in a very good match with the experimental pressures but yielded a lift coefficient of $CL=0.541$; 0.041 higher than indicated by the test data. Note that typically we would get a better match with pressure distributions if we match lift coefficient since that is usually the more direct measurement. The good agreement with the wing pressure distribution would indicate that the wing lift was correctly predicted and that the error must be in the predicted body lift. The predicted wing lift is 0.476 indicating that the remaining body lift should only be 5% of the total. However, body lift on a configuration like this is typically 10-12% of the total indicating that the total lift cannot be only 0.50, but should be closer to the predicted amount! This same inconsistency was also found at the two other test conditions for which pressure data were available. At $\alpha=-0.9$ degrees the wing lift alone matched the wind tunnel value for the entire configuration. The experimental wing pressure distributions do not seem to be consistent with the measured forces! In this case with only limited test data and lacking experience with test data from the ONERA facility we were not able to resolve the discrepancy. *The recently completed wind tunnel test at the NASA NTF facility will provide substantially more test data that should bring better understanding of this discrepancy.*



3.3 Validation for Drag

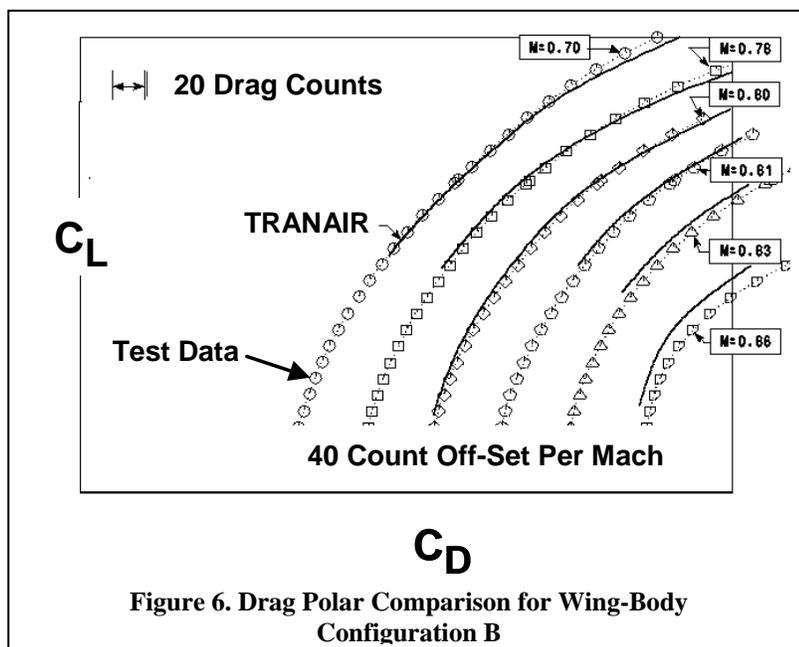
The use of CFD for multi-point design/optimization to minimize drag of a transport configuration requires a high degree of confidence in the drag predictive capabilities of a CFD code. Design/optimization was developed in the TRANAIR code in the early 1990's. It has been refined and expanded over the years and is still used extensively in any airplane design project. The physical model in the code, full potential plus coupled boundary layer, is adequate for modeling essentially attached transonic flows. The real strength of the TRANAIR system is its robustness and reliability in part due to its solution adaptive grid capability. The confidence in the code's predictive capability did not come easily, but is a result of a continuous validation effort with every new configuration that becomes of interest. Test to CFD comparisons were not limited to a few discrete points but covered a range of flight conditions. Not only is lift and drag compared, but also pitching moment, pressure distributions, and wing section forces. All comparisons have to make sense to build confidence.

3.3.1 Simple Wing-Body Configurations

Initial validation focused on wing-body configurations, thereby avoiding the complications of struts, nacelles, and tails. Drag comparisons for Wing-Body Configuration B are shown in Figure 6. These CFD results from

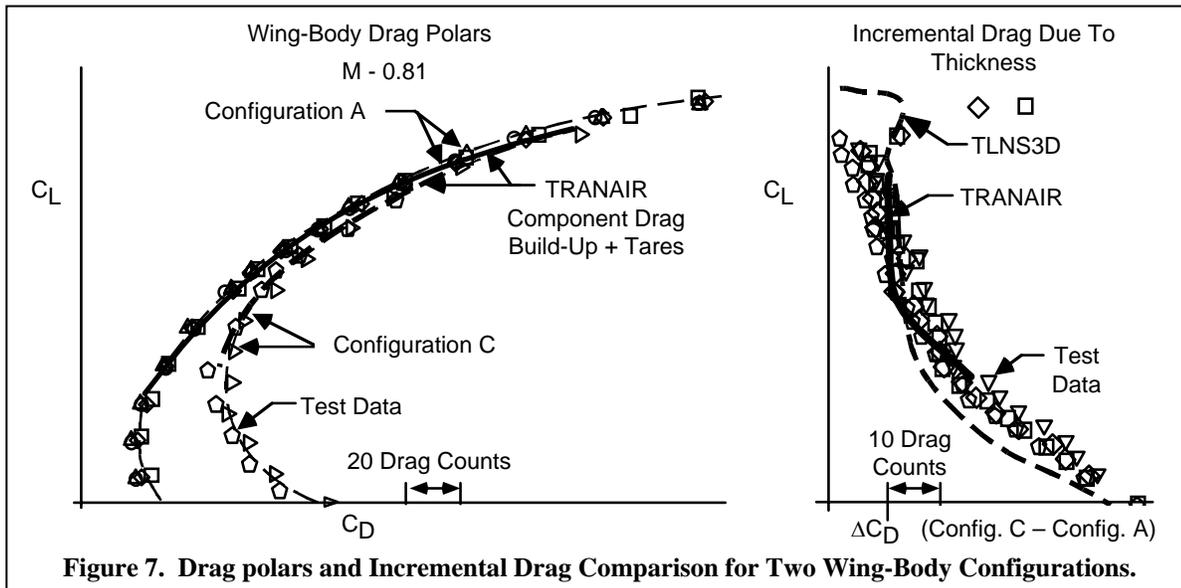
CFD Uncertainty and Validation for Commercial Aircraft Applications

the 90's are compared to wind tunnel data²⁹. A single arbitrary tare (shift) has been added to the TRANAIR results to better align them with the test data. There are always enough issues with both the CFD modeling and with the corrections applied to the test data that question if an absolute drag level can be computed much less measured. This tare compensates for those unknown deficiencies. The increments between Mach numbers and the polar shapes are in good agreement with the test data. In addition to the drag polars, lift and pitching moment characteristics were also compared to test data across the Mach number range²⁹. Pressure distributions at each flight condition were examined to assess roof top level and shock location. The higher Mach numbers away from the design cruise Mach really stress the assumptions of full potential plus boundary layer modeling and reveal some of its shortcomings. By looking at the forces, moments, and pressure distributions over a range of flight conditions one can get a feel for the range of conditions which warrant a high degree of confidence in the CFD solutions as well as the conditions which warrant a higher degree of uncertainty. These comparisons highlighted the need to improve the wave drag computations which was later accomplished.



During the 90's the speed and size of available computers limited the grid size for routine use of Navier-Stokes solvers. Careful grid generation practices to ensure consistent size, topology, and spacing are essential to ensure good drag prediction results as illustrated in Figure 7. Here predictions from TRANAIR and the TLNS3D Navier-Stokes code are compared to test data for two wing-body configurations, Configurations A and C²⁹. These two configurations had the same body and wing planform but featured different wing thickness distributions. Also shown is the incremental drag difference between the two configurations. The differences between several different test repeats are shown for the test data. A different

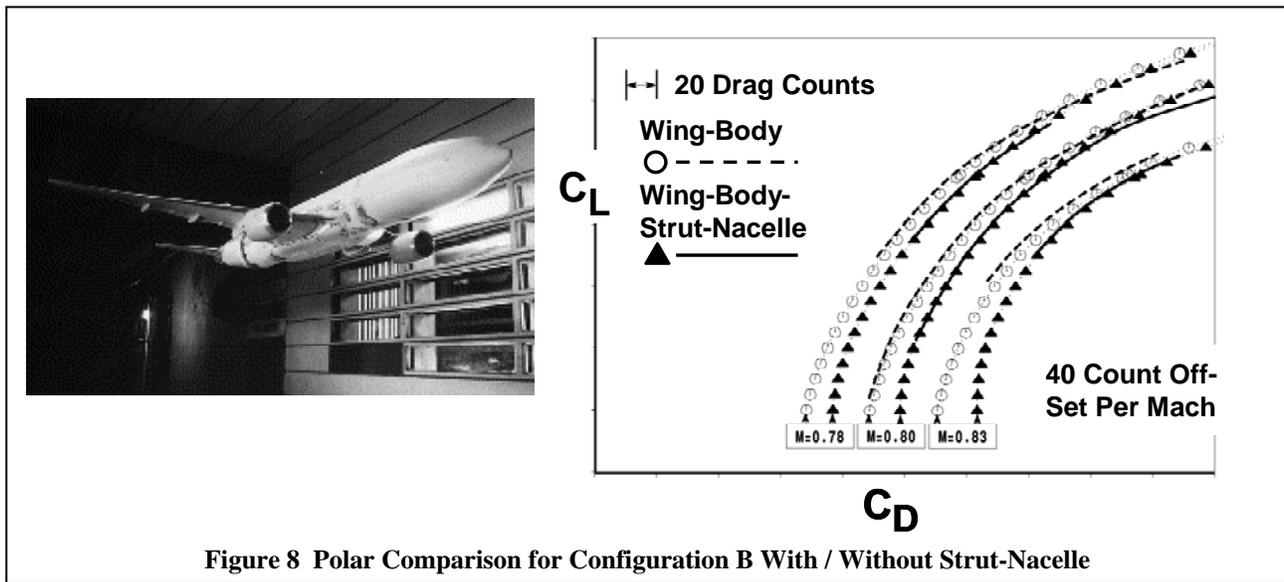
tare for each code was added to the CFD results to better match the levels of the test data. The incremental drag difference between the two configurations predicted by the two CFD codes show excellent agreement with the test data. Note that the tares cancel out when comparing increments. Again, comparisons included lift and pitching moment characteristics and pressure distributions. Code-to-code comparisons as well as code to test comparison yield more insight into their respective characteristics. The advantage of the Navier-Stokes solvers is that they are valid for flow conditions beyond the capability of TRANAIR. However these conditions are generally away from the design cruise flight conditions where drag is less critical. In addition the Navier-Stokes solvers are significantly more computationally expensive compared to TRANAIR.



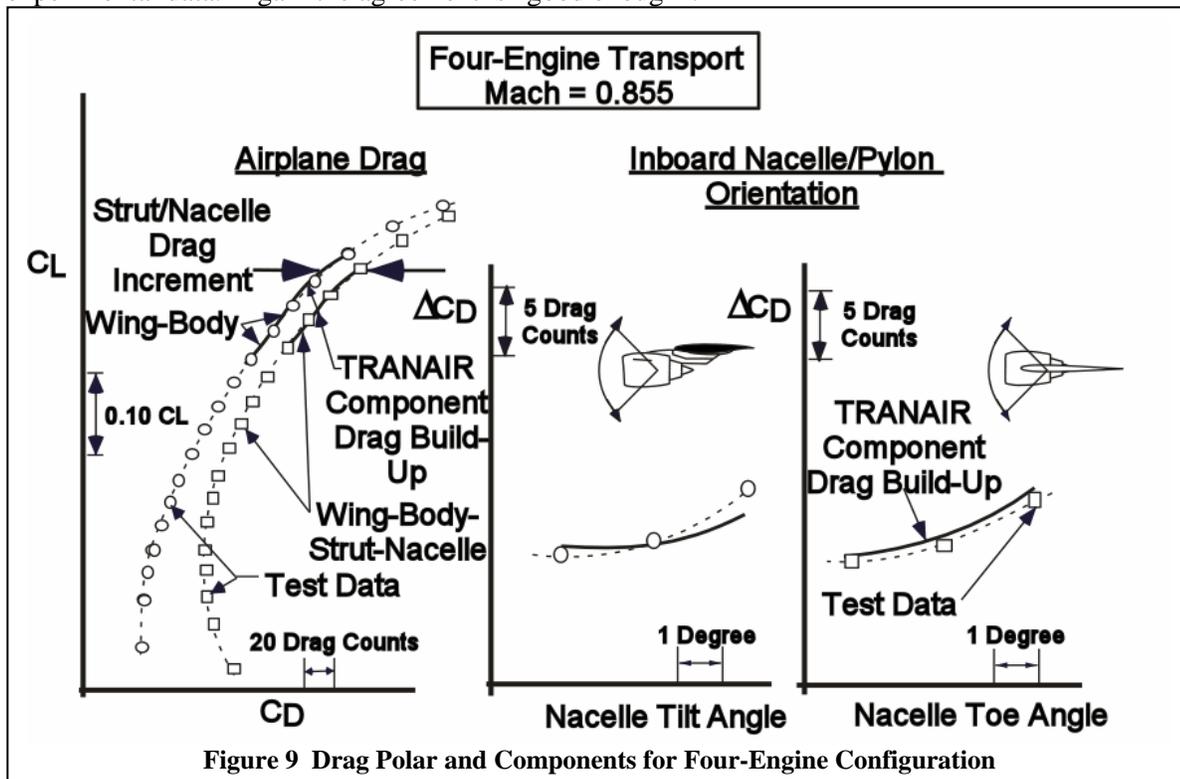
3.3.2 Adding Engine Struts and Nacelles

While interesting, a wing-body configuration is not representative of transport aircraft. These initial correlations were necessary to establish the feasibility of using CFD to adequately predict drag before progressing to a more complex configuration. The presence of engine strut and nacelle can have a dominant effect on the wing and must be included in any design/optimization study. A flow-through nacelle and strut were added to wing-body Configuration B for the next level of validation²⁹. The internal nacelle drag has been removed from the test data. The correlations would now test the CFD modeling of the addition of the strut nacelle and its impact on drag. Again good agreement is seen between the TRANAIR results and the test data which is shown for both the wing-body and the wing-body-strut-nacelle configurations in Figure 8. Of particular interest in this comparison is the incremental difference in drag with the addition of the strut-nacelle. The impact of adding the nacelle and strut on lift, pitching moment, wing pressure, span load, and section pitching moment distributions were all scrutinized. In any validation study one has to look beyond the total configuration forces. Hopefully detail pressure measurements on the wing are available. Do the roof-top pressure levels and shock locations agree with the test data? How well do the integrated section lifts and pitching moments agree? Keep in mind that the sparseness of the experimental data can result in a significant difference when compared to the integrated CFD data. A comparison of the spanwise distribution of section lift can give an indication of how well the wing twist of the CFD model matches that of the wind tunnel model at that particular test condition. Validation has to make sense for all the quantities that can be compared between the CFD and test data. These comparisons frequently will reveal the areas in which the CFD needs improvement.

CFD Uncertainty and Validation for Commercial Aircraft Applications



The initial impetus for design/optimization was to be able to apply it to 4-engine configurations. The drag prediction capability for TRANAIR on a 4-engine configuration is shown in Figure 9. Computational data and experimental results are shown for both a wing-body alone configuration and the configuration with the addition of four pylons and flow-through nacelles². The computational results are in good agreement with the wind tunnel data. Also shown is the effect of nacelle toe and pitch on installed drag compared with experimental data. Again the agreement is “good enough”.



3.3.3 Adding the Tail – Trim Drag

Trim drag, drag associated with trimming the horizontal tail to zero pitching moment about the center of gravity, can have a significant impact in comparing similar wing designs. A trimmed drag polar typically requires interpolating data from drag polars from a configuration with three different horizontal tail deflections to develop the relationship between drag and pitching moment changes. TRANAIR allows the user to specify both lift coefficient and pitching moment for a solution, greatly simplifying the generation of a trimmed drag polar. Figure 10 shows a comparison of polars with experimental data for three different wing-body-strut-nacelle configurations, D, E, and F. These three configurations featured the same body, strut, nacelle, and tails, but wings of different planform and span. Results are shown for both the tail-off configuration and with the tail-on trimmed to zero pitching moment. A single tare has been added to all the CFD data to better align it with the test data. Other than the value of the tare, these are predictive CFD results generated before the wind tunnel test!

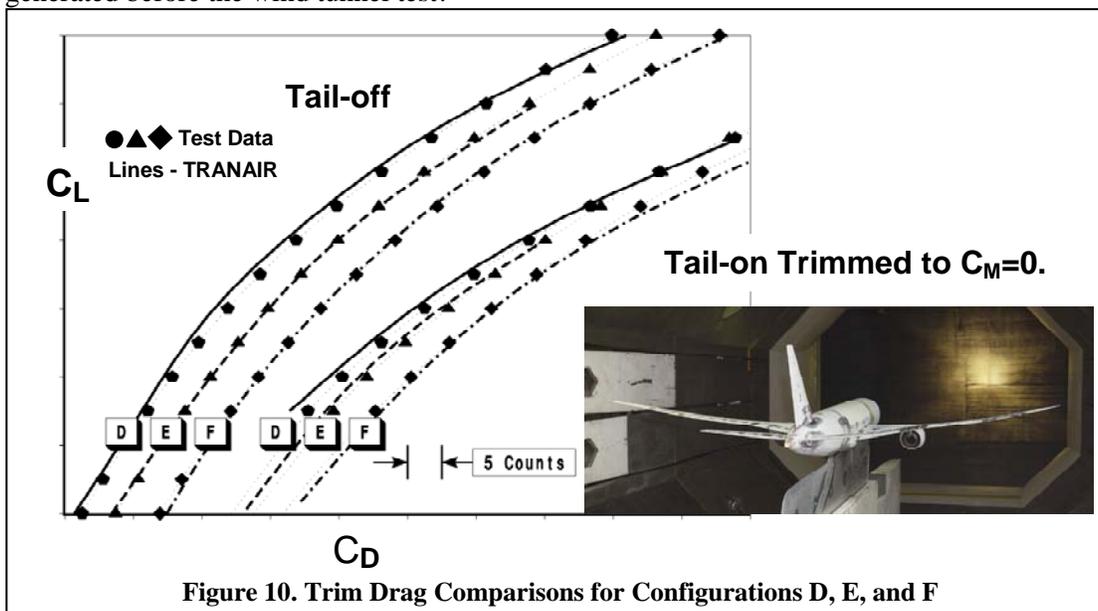


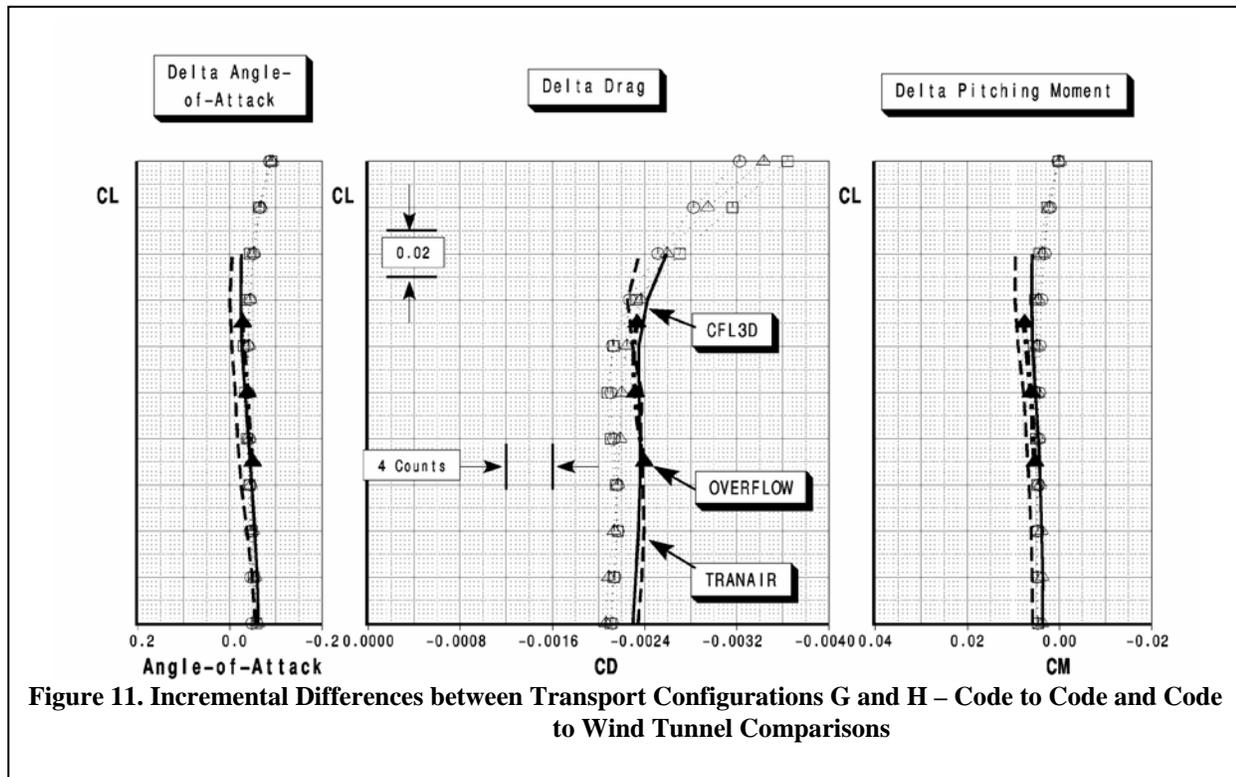
Figure 10. Trim Drag Comparisons for Configurations D, E, and F

3.3.4 Navier-Stokes Drag Increments

While most of the design work is based on TRANAIR, the Navier-Stokes methods play an important role in providing a second opinion, exploring viscous dominated details that TRANAIR is incapable of resolving, and extending the range of Mach and angle-of-attack beyond which TRANAIR is capable. Although TRANAIR, CFL3D, and OVERFLOW may deliver somewhat different solutions for a given configuration, the codes will give very similar incremental differences between configurations. Shown in Figure 11 are the incremental differences between two wing-body-strut-nacelle transport configurations, G and H, as computed by the three codes compared to increments derived from wind tunnel data. These two configurations featured identical bodies and nacelles but had wings of different planform area and span. The increments computed by the three codes are almost identical and agree quite well with the wind tunnel data. Given that three very different CFD codes, using different turbulence models, give essentially the same incremental drag value which disagrees with the wind tunnel increment by two drag counts, one might be inclined to question the wind tunnel results in this particular case. This example illustrates the overwhelming need for consistency between CFD models when trying to predict drag increments. The codes will yield different drag levels depending on choice of grid, turbulence models, and all the “knobs” available to the user. Only by minimizing these

CFD Uncertainty and Validation for Commercial Aircraft Applications

differences between CFD models within the same code can one hope to deliver meaningful increments between configurations. Consistency is the key to minimizing uncertainty!



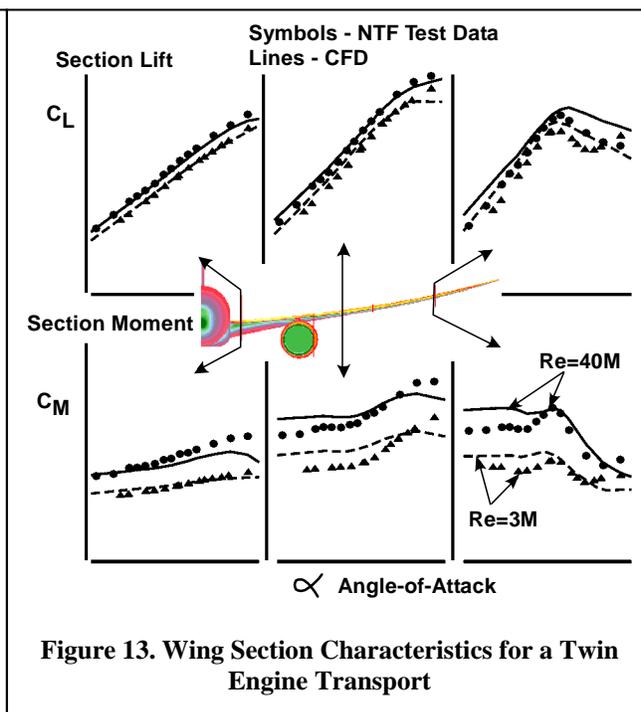
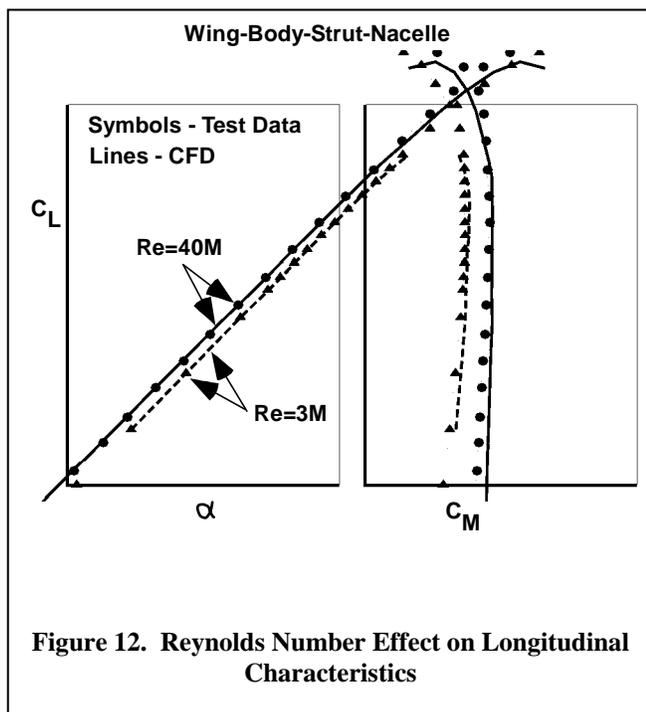
CFD validation for drag is a never-ending process of refinement and discovery. Each new configuration tested is a new source for validation. If the new configuration is just an iteration of the “tube and wing” transport topology no surprises are expected nor have any been found in recent years. Even the all-wing Blended Wing Body configuration requires only minor modifications to the processes in place for the conventional “tube and wing” transports. An all-new type of configuration and flight conditions such as the Sonic Cruiser with its $M=0.98$ design Mach number did need new adjustments to the gridding and solution process. The generally good agreement shown in these examples did have one thing in common. All these configurations exhibited essentially very little or no flow separation at their design cruise conditions. With significant flow separation the drag prediction process is no longer as accurate. For the most part, the refinements in drag prediction have dealt with improving the robustness of the gridding process for both TRANAIR and the Navier-Stokes solvers. Taking advantage of the faster and multiple processors for more geometric resolution and streamlining the entire “lofts-to-plots” process continues the agenda.

3.4 Validation for Loads

Aerodynamic loads prediction is one of the more time consuming items in an aircraft development program. Large aerodynamic loads data bases are needed before the detail structural design of the airframe can proceed. These data bases can number ten’s of thousands of points when control surface deflections over a wide range of flight conditions are included. Critical design loads are generally found at the extremes of the flight envelope. These loads can be subjected to significant Reynolds number effects that the typical conventional wind tunnel may not catch. Progress is being made in the ability to use CFD to provide Reynolds number

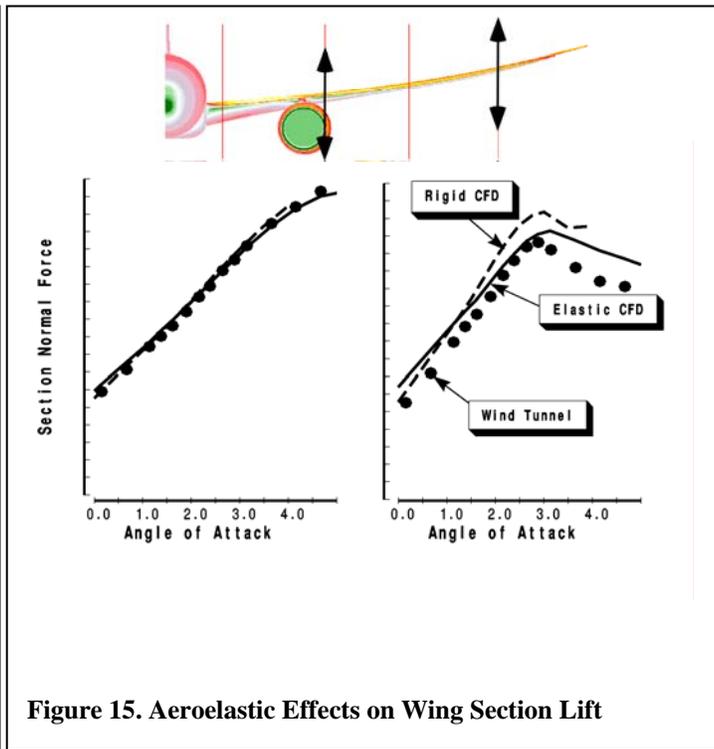
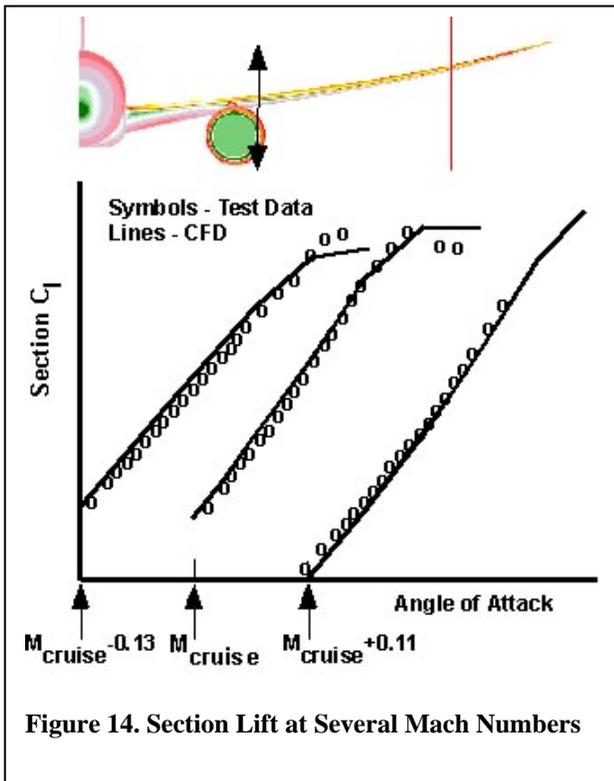
corrections to conventional wind tunnel data, and to use CFD to determine initial loads prior to a wind tunnel entry⁶. One challenge here is identifying the appropriate experimental data sets for use in CFD validation. Conventional wind tunnels offer only modest Reynolds number changes and remain far lower than flight Reynolds number for large commercial aircraft. Cryogenic wind tunnels offer a more appropriate Reynolds number range but are still limited on the variety of data currently available. Appropriate detailed flight data is even scarcer.

Figure 12 shows a comparison of airplane lift and pitching moment characteristics at wind tunnel and flight Reynolds number. The CFD predictions capture the incremental differences due to Reynolds number and show reasonable agreement with the test data. A tare of +0.3 degrees in angle-of-attack and +0.01 in pitching moment was added to the CFD results to better illustrate the incremental predictions with Reynolds number. Of greater interest to the Loads engineer is the distribution of these forces and moments across the wing. Figure 13 shows wing section lift and pitching moment coefficients as a function of angle-of-attack at several stations along the wing span of a twin engine transport. CFD results capture the incremental differences due to Reynolds number change. These type of comparisons must be made over a wide range of Mach numbers to give the Loads engineers sufficient confidence to use CFD in flight loads predictions.

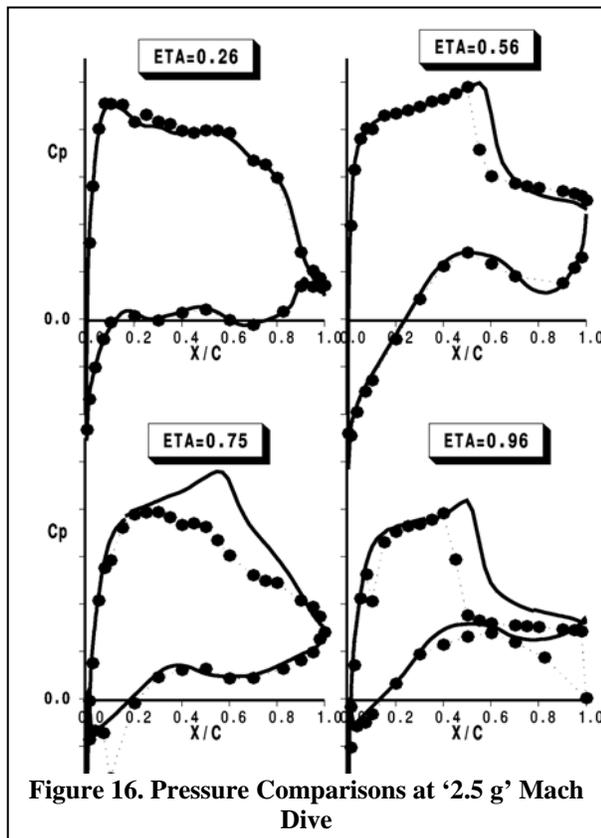


Prior to the availability of wind tunnel data, CFD can provide initial wing load characteristics over a wide range of Mach numbers. Figure 14 shows a comparison of CFD results with test data at a wing span station outboard of the engine nacelle. Section lift coefficients as a function of angle-of-attack are shown for three Mach numbers – cruise Mach, cruise Mach minus 0.13 Mach, and cruise Mach plus 0.11 Mach. The CFD results well capture the loading variation with Mach number at this wing station. Further outboard on the wing the wind tunnel model aeroelastics effects are more significant and must be taken into account in trying to validate the CFD results. Figure 15 show a comparison of section normal force at two spanwise stations on the wing. CFL3D was used for the CFD calculations. The “rigid CFD” results are based on the 1-g cruise shape for the entire angle-of-attack range. For the “elastic CFD” solutions CFL3D has been loosely coupled with a NASTRAN finite element model of the wind tunnel model wing in an iterative fashion to generate a

CFD Uncertainty and Validation for Commercial Aircraft Applications



solution at each specified angle of attack. The impact of wind tunnel model aeroelastic is dramatic on the outboard part of the wing. For this type of validation wind tunnel model aeroelastics must be taken into account.



Critical loads and stability and control issues are frequently encountered at the extremes of the flight envelope where significant flow separation over the wing may be encountered. Validation of CFD by comparison to experiment is further complicated at these conditions. Figure 16 shows a comparison of wing pressure distributions at four span stations with wind tunnel data for a multi-engine configuration at an approximately 2.5 g Mach dive pull-up. Loads encountered by the wind tunnel model are significantly different from the design '1-g' load resulting in significant aeroelastic changes which should be accounted for. They are not in this comparison for lack of an appropriate finite element model. These wind tunnel data were taken close to Mach one and at high angle of attack so one might question the validity of the data representing "free-air" conditions. Similarly one can question the validity of CFD at these conditions with massive flow separation. What can be expected of the turbulence models at these conditions? Are the

grids appropriately distributed? Is steady RANS sufficient or do we need unsteady RANS or DES (detached eddy simulation). These are questions that need further research. Remember we need reliable predictive CFD capability over a broad range of flight conditions, not just tuning of CFD to match one set of experiments.

In the development of a large commercial transport the determination of “final” aerodynamic flight loads demands a very high degree of accuracy. Too much conservatism in the loads prediction results in unacceptable excess structural weight. Under-predicting the flight loads may result in an un-certifiable airplane or certifying to a lower weight with significant performance penalties until the structure is improved. CFD data can be used in determining initial structural sizing, to support wing trade studies, and to help scale the wind tunnel data base to flight conditions. However, CFD is not yet able to deliver the required degree of accuracy and throughput necessary to completely replace the wind tunnel. The challenge today is how to best combine the use of CFD with wind tunnel testing to improve the prediction of aerodynamic flight loads while reducing the development cycle time.

5.0 MINIMIZING CFD UNCERTAINTY

The overriding engineering requirement for the effective use of CFD is one of consistency of results, rather than absolute accuracy. In comparison to the small, tight-knit, community of wind tunnel test practitioners, who have developed a relatively standard and consistent process for acquiring data at their respective facilities, the CFD community is wide open. There are several levels of flow simulation. Results depend on decisions regarding which physical model or models are used (e.g., viscous or inviscid, linear or non-linear, incompressible or compressible, full potential or Euler or Reynolds Averaged Navier Stokes, boundary layer models, selection of turbulence models and transition models, etc.). We started with linear potential (panel methods) in the '60's and still use it today. For transonic flows our major focus today is full potential with coupled boundary layer and the Reynolds Averaged Navier-Stokes equations. Each code has its own set of user defined “knobs” which can greatly affect the solution - the biggest knob of all is grid generation. The grids have to adequately and consistently resolve the configuration's geometric features, viscous boundary layers, shear layers, shock waves, etc. The major focus of making CFD effective in commercial aircraft applications has been to minimize uncertainty by assuring the utmost consistency in results via a standardization of the way CFD modeling is done.

To achieve this goal, stable “productionized” packaged software solutions have been developed to enable and promote consistent processes. These not only put CFD into the hands of the Product Development / Product Support engineers, but also allow the “expert” user to get results more quickly with reduced variation. These integrated packaged software solutions combine the various CFD components to go from “lofts to plots” in the time scale consistent with a fast paced engineering program. These packages include: scripted packages for “Standard” configurations; geometry and grid/paneling generation components; flow solvers; and post-processing components for analyzing the results. By “Standard” we mean those configuration types that have been scripted in the various components that make up the particular CFD process. Configurations not included in the “Standard” can still be analyzed but will not benefit from the same degree of automation. These processes are all placed under some form of software version control to maintain consistency.

A key component of CFD and most engineering processes is geometry. CAD systems, such as CATIA, dominate most geometry engineering needs. However, these systems are designed for component design and definition and are not necessarily well suited for CFD use. A key component of many BCA CFD processes is AGPS – Aero Grid and Paneling System^{30,31}. AGPS is a geometry software tool implemented as a programming language with an interactive graphical user interface. It allows the engineer to create,

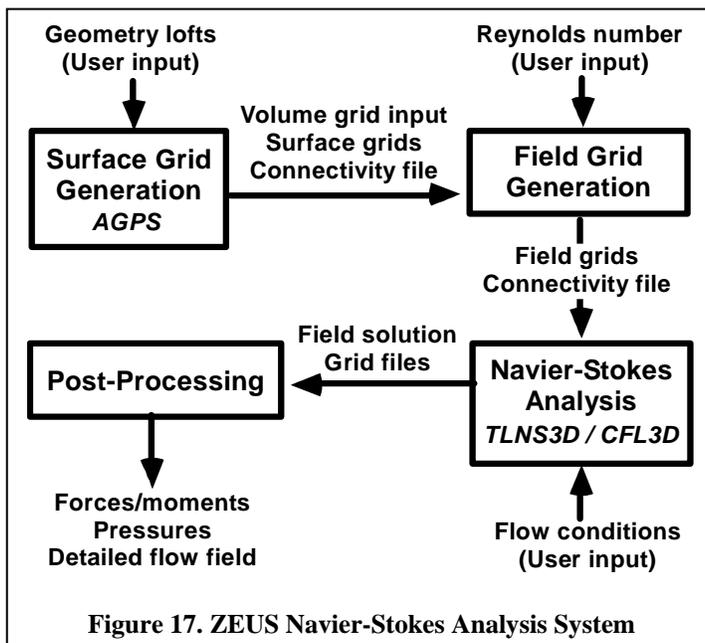
CFD Uncertainty and Validation for Commercial Aircraft Applications

manipulate, interrogate, or visualize geometry of any type. It features a journaling/replay capability to automate tasks such as surface grid generation and the creation of the complete input set-up to the CFD process.

Grid generation practices have been an evolving process over the years with an aim of improving consistency, robustness, and accuracy. Each new application can bring lessons to improve the CFD process. As computers have gotten bigger and faster there has been the tendency to increase grid sizes. Changes to the topology and density of the grids have been guided by the lessons learned from the detailed comparison of forces, moments, and pressures with wind tunnel data for each case evaluated. These changes become standard for the next case. Rarely is any formal type of grid refinement study carried out. Rather refinement is restricted to local areas of the configuration. We have developed processes that work for TRANAIR, CFL3D, and OVERFLOW. The changes enacted make the process and resulting solutions a little more accurate, but mainly improve robustness and reliability, all of which improve user confidence. These processes will continue to evolve. This evolutionary process works in part because the airplane configurations we apply CFD to are also evolutionary. Large jet aircraft have not really changed much from tube and wing configuration of the Boeing B-47 which first flew nearly 60 years ago!

Two formal CFD processes, in use in Boeing Commercial Airplanes for transport cruise configuration analysis, that feature significant automation for fast turn-around are ZEUS for structured grid Navier-Stokes analysis, and TRANAIR for full potential plus boundary layer analysis and design/optimization. A less formal process to insure consistent application of the OVERFLOW code transport type configurations has also been established. Lessons learned are continuously being captured and incorporated into these processes. It is through these standardized processes that variation and uncertainty are minimized.

5.1 ZEUS System for Structured Grid Navier-Stokes

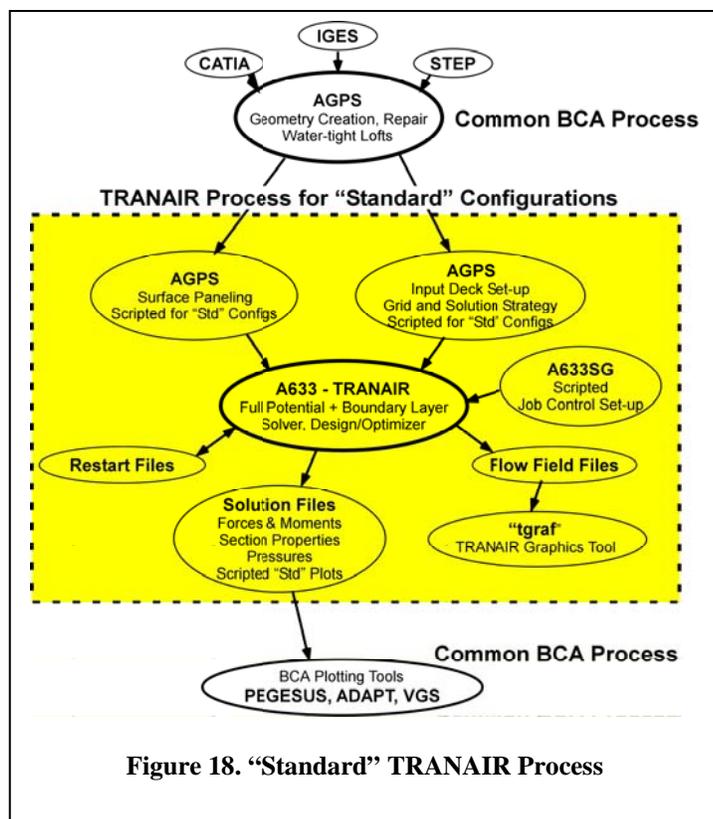


The *ZEUS* analysis system, illustrated in Figure 17, is a series of UNIX shell scripts that coordinate surface grid generation, field grid generation, analysis, and post-processing for structured grid Navier-Stokes CFD codes - CFL3D and TLNSD. For a given type of configuration, a specific grid generation process is developed and packaged into the system. Boeing CFD developers have rewritten a 3D grid generation code through the use of an advancing front approach¹⁶, so that a precise control on grid quality, such as grid spacing, stretching ratio, and grid orthogonality near configuration surfaces can be achieved. This is an important requirement for accurate resolution of viscous flow regions for all existing Navier-Stokes solvers. The matched/patched multiblock grid approach subdivides the flowfield into a number of topologically simple regions, such that in each region high quality grid can be generated. This is a rather time-consuming and tedious process for complex configuration analysis. However, once this “blocking” process is done for one configuration and has been captured, a similar configuration can

be done easily through the use of script or command files.

To analyze a configuration using the *ZEUS* analysis system, users are only required to input water tight geometry in the form of AGPS lofts and flow conditions, the system takes care of the rest of the analysis processes. The output from the system includes automatic plots of sectional data, convergence history, configuration force and moment information, together with detailed flow field files for flow visualization and diagnosis. Current configurations supported by *ZEUS* include wing/body, wing/body/pylon/nacelle with either twin or four-engine flow-through or powered nacelles, and wing/body/pylon/nacelle/horizontal-vertical tail. Options to support winglets and vortex generators are also available. Configurations not included in the “Standard”, such as the addition of control surfaces, or spoilers can still be analyzed but will not benefit from the same degree of automation.

5.2 The TRANAIR System



The TRANAIR Process for “Standard” Configurations is illustrated in Figure 18. Similar to the *ZEUS* system, the package is compatible and takes advantage of common BCA processes for geometry and post-processing. At the center of this process is the TRANAIR flow solver. AGPS scripts have been developed to automate the paneling of “standard” configurations from AGPS lofts. AGPS scripts have also been developed to generate the input deck for the TRANAIR solver. These inputs define the flight conditions, solution adaptive gridding strategy, and the boundary layer inputs for “standard” configurations. A UNIX script is available to generate the various job control files to execute the solver on several types of computing systems. The TRANAIR solver generates several files for restarts of the solver and output processor, output files for various aerodynamic parameters, and a file for flowfield parameters. A special purpose code, compatible with the unique structure of the TRANAIR code is available to view the

flowfield properties. The package enables setting up and submitting for solution a “standard” configuration from AGPS lofts in one or two hours. Complete solutions from “lofts to plots” are frequently available in less than 4 hours. “Standard” configurations include conventional transport configurations including 4-engine 747-like aircraft with under-wing struts and nacelles and vertical and horizontal stabilizer with boundary layer on the major configuration components. “Standard” packages for innovative configurations such as the Sonic Cruiser have also been developed. The various components have been assembled to not only fit, but also to form the engineering process.

6.0 CASE STUDY – THE AIAA 3RD DRAG PREDICTION WORKSHOP

The AIAA Third Drag Prediction Workshop^{32,33} (DPW3) serves as an interesting case study. The workshop presented us with the opportunity (excuse) to do a grid refinement study and now to illustrate the predictive nature of CFD. All the calculations and the bulk of this paper were written prior to the validation wind tunnel test that was completed in October 2007. The subject configuration was the DLR F6 previously used in the AIAA Second Drag Prediction Workshop and subject of Figures 3 and 4. This configuration is a simple wing-body which lends itself to a systematic grid refinement. (Note that a systematic grid refinement becomes much more challenging with the addition of a strut and engine nacelle.) Two configurations were considered – the basic F6, and the F6 with the FX2B fairing on the side of body in the region of the wing trailing edge, shown in Figure 19. The purpose of the fairing is to eliminate a large pocket of separated flow in the wing-body intersection around the wing trailing edge.

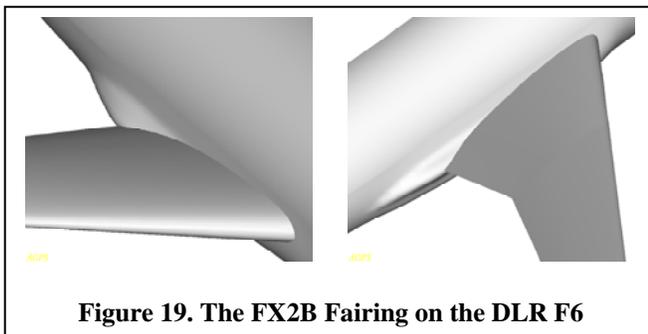


Figure 19. The FX2B Fairing on the DLR F6

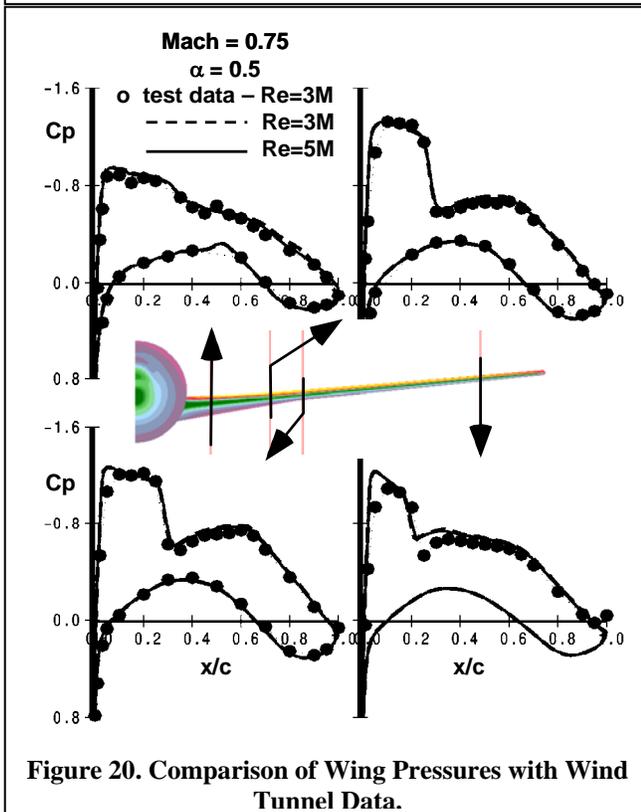


Figure 20. Comparison of Wing Pressures with Wind Tunnel Data.

made at a fixed angle of attack, $\alpha=0.50$ degrees, closely matching that of the experimental data ($\alpha=0.49$).

We constructed four sets of point-matched structured multiblock grids for each configuration³⁴. These ranged in size from just under 3 million cells to over 31 million cells. The grid sizes were increased in all three directions by approximately 1.5 from each preceding grid. Because our goal is getting good drag increments between the configurations we always strive to minimize the differences between the two configuration grids – topologies, surface point distribution, and grid size. For these cases the number of grid points and distributions were identical between configurations. These cases were run in CFL3D. (Note that cases were also run with two unstructured grid codes. However, our experiences with these methods are still very limited and do not yet lend themselves to this discussion.)

6.1 Comparison with Existing Test Data

The existence of wind tunnel test data²⁶ for the basic DLR-F6 wing-body configuration from the 2nd AIAA DPW suggested that an initial check of solution quality could be made by comparison to that data. As previously noted test data were acquired in the ONERA S2M wind tunnel, at a Reynolds number of 3 million based on the mean aerodynamic chord. The specified Reynolds number for the 3rd AIAA DPW calculations was 5 million in anticipation of acquiring new wind tunnel test data at that value. Solutions at 3 million and 5 million Reynolds numbers were computed using the medium grid (9 million cells). A comparison of wing pressures from these results with test data is shown in Figure 20. The comparison is

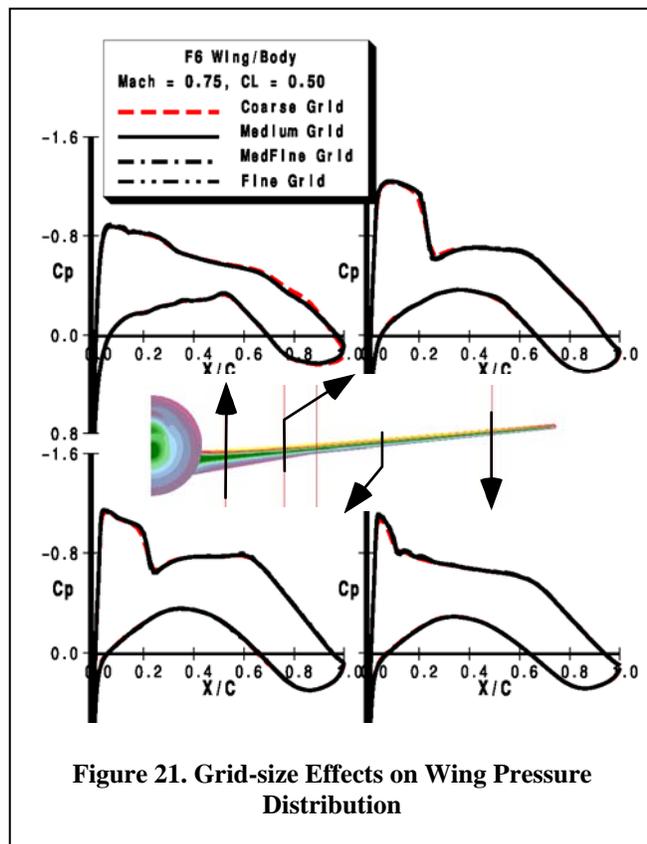
Increasing Reynolds number has little visible effect on the solution, resulting in a slight aft movement of the shock wave. Overall agreement with the test data is very good.

6.2 Grid Convergence Studies

A series of solutions were obtained to investigate the influence of grid refinement on the predicted drag characteristics of the two wing-body configurations. Solutions were obtained at a constant lift coefficient, $CL=0.50$, using the four available grids with:

- CFL3D with thin-layer Navier-Stokes terms using the SST turbulence model.
- CFL3D with thin-layer Navier-Stokes terms using the SA turbulence model.
- CFL3D with the full Navier-Stokes terms using the SST turbulence model.
- CFL3D with the full Navier-Stokes terms using the SA turbulence model.

Multigrid and grid sequencing were used to accelerate solution convergence. In addition to monitoring residual reduction, drag and lift variation with iteration was monitored. Solutions were converged to $CL=0.5 \pm 0.0001$, with drag varying less than 0.05 count over the last 400 iterations, except for some coarse grid cases where drag settled into a steady oscillation of ± 0.1 counts or less.



Wing pressure distributions from the four grids on the baseline DLR-F6 wing-body using the SST turbulence model and the thin-layer approximation are shown in Figure 21. With the exception of the coarse grid solution, the distributions are indistinguishable. This was typical of both configurations, turbulence models, and the thin-layer and full Navier-Stokes approximations. For a fixed angle of attack, the shock location was slightly farther aft for the SA turbulence model compared to the SST model.

One other piece of information that we had about the basic F6 configuration was an oil-flow showing the side-of-body separation bubble. An oil-flow picture from the ONERA wind tunnel test at 3 million Reynolds number is shown in Figure 22. Overlaid on this picture are streamlines from a CFL3D thin-layer solution using the SST turbulence model at 5 million Reynolds number. The computed streamlines are seen to be in reasonable agreement with the oil flow. A solution of the medium grid at 3 million Reynolds number produced a separation bubble only 1.5 mm wider. The resolution of the overlay is no better than that!

CFD Uncertainty and Validation for Commercial Aircraft Applications

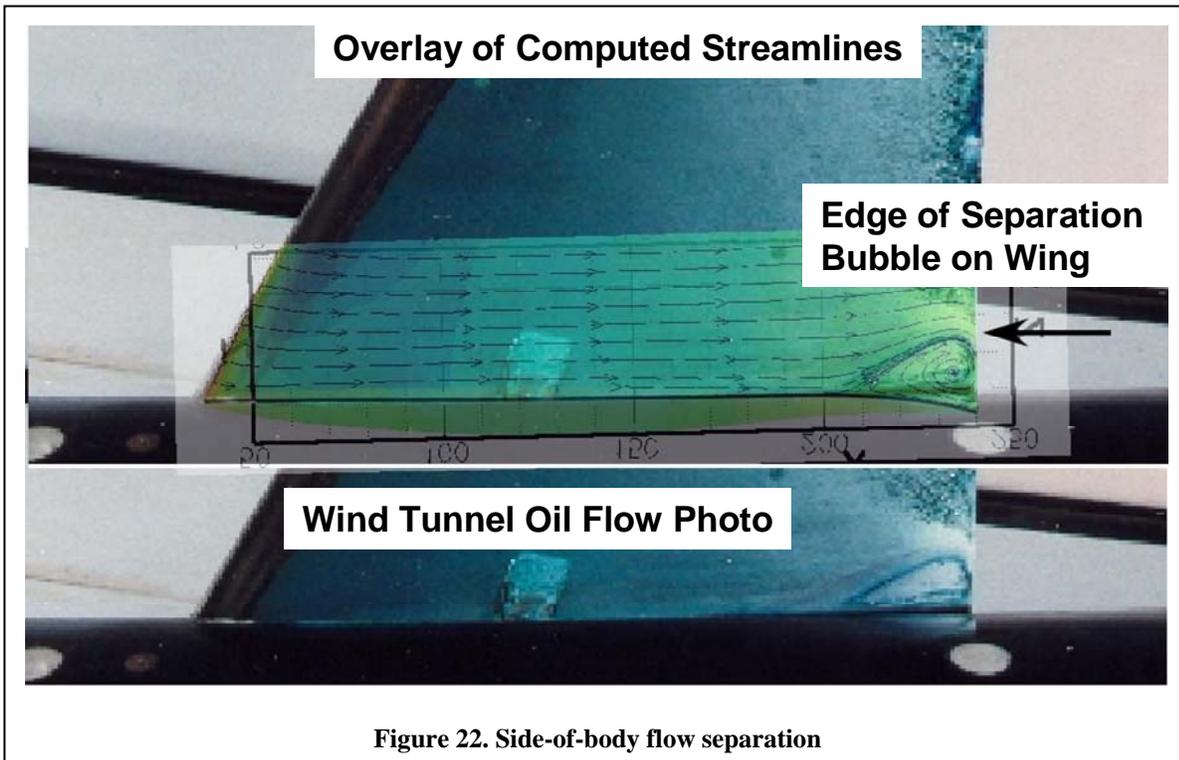


Figure 22. Side-of-body flow separation

Figure 23 shows a measure of the spanwise extent of the side-of-body separation bubble from solutions from CFL3D with either full or thin-layer Navier-Stokes terms using the SA and SST turbulence models for the four grids. The values are plotted as a function of (GRIDFAC) $1/N^{2/3}$, where N is the total number of grid cells in the solution. A value of 0 would correspond to infinite grid size. The size of the separation bubble varies little for the three largest grid sizes. The use of the SA turbulence model resulted in a separation bubble slightly smaller than with the SST turbulence model. Use of the full Navier-Stokes terms resulted in separation bubbles 2 to 3 mm wider. Note that the extent of the separation did not encroach on the first row of pressure taps on the wing. This was also evident in the pressure distributions shown in Fig. 20. Although the variation of the predicted bubble size is small for the larger grid sizes, it can be expected that this variation will impact the calculation of drag and in particular the calculation of the drag increment between the baseline F6 with the separation and the configuration with FX2B fairing which eliminates the separation.

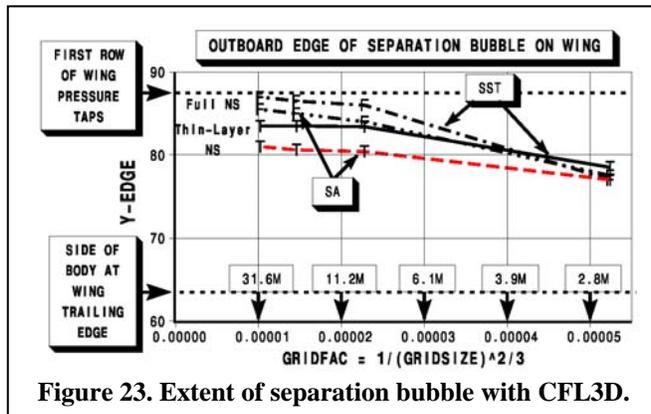


Figure 23. Extent of separation bubble with CFL3D.

Drag results from CFL3D with thin-layer Navier-Stokes terms using the SA and SST turbulence models are shown in Figure 24. Drag is shown for both the basic F6 wing-body configuration and for the configuration with the FX2B fairing. Total drag is plotted as a function of (GRIDFAC) $1/N^{2/3}$. For a second-order accurate method, results from successively refined grids should form a straight line extrapolation to an infinite grid size. In actuality, successively refining grids by doubling them in every direction in 3D for complex

configurations is still not vary practical, although we believe that this set of grids is a reasonable approximation. As previously stated, the solutions were converged until the drag varied by less than 0.05 counts over the last 400 iterations. However, as is typical of most Navier-Stokes codes, it was not practical to drive the residuals to machine zero. It is interesting to note that while using the SA model results in a more linear variation of drag with grid size; it also results in a greater variation of the absolute value of drag with grid size. The lack of linearity of the drag convergence using the SST model is puzzling, particularly on the configuration with the FX2B fairing. On this configuration, the flow is everywhere attached; the significant flow features have been resolved. It is the skin friction that continues to change with grid refinement when using the SST model. The behaviour of the drag convergence at the finest grids is particular and not yet understood. Solutions using CFL3D with the full Navier-Stokes terms were of similar character.

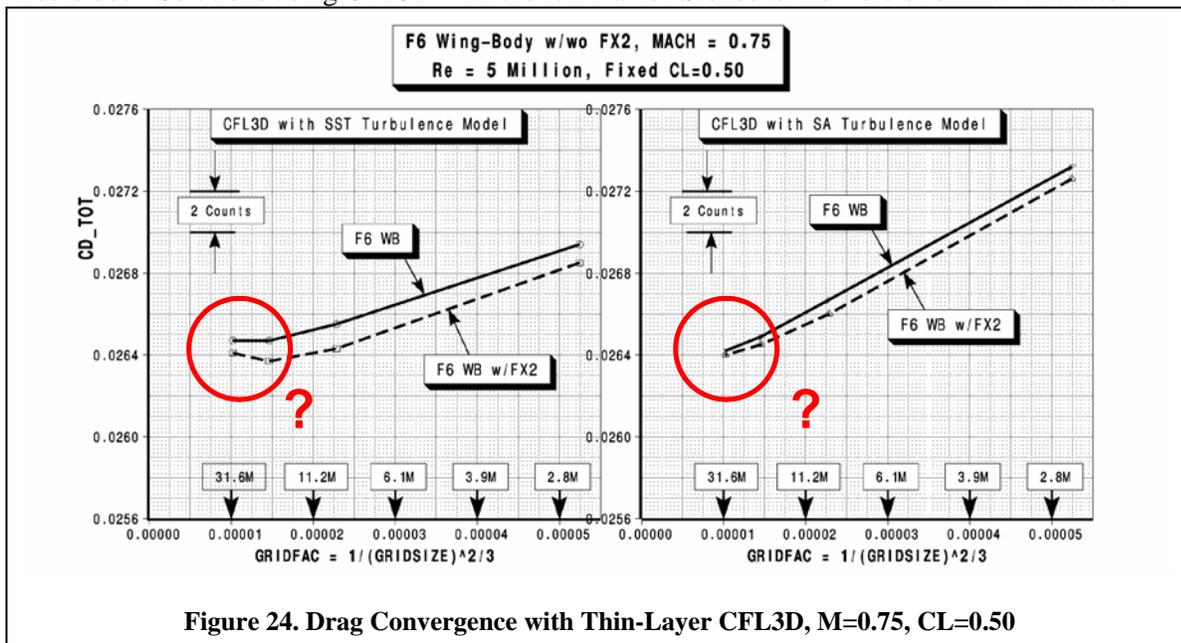


Figure 24. Drag Convergence with Thin-Layer CFL3D, M=0.75, CL=0.50

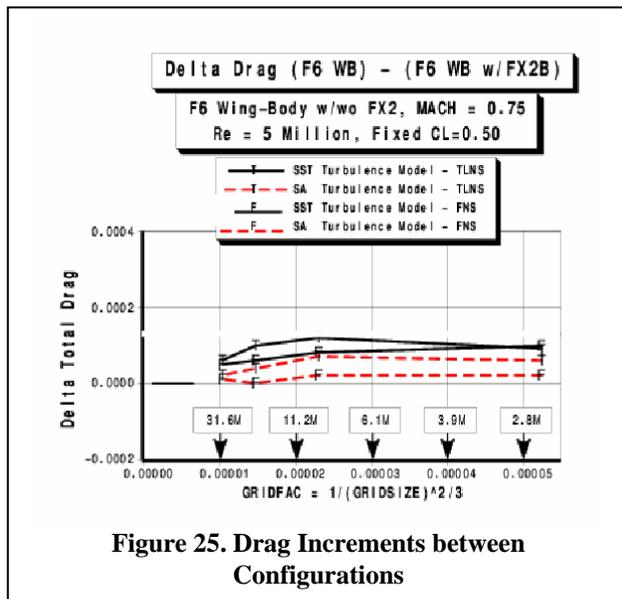


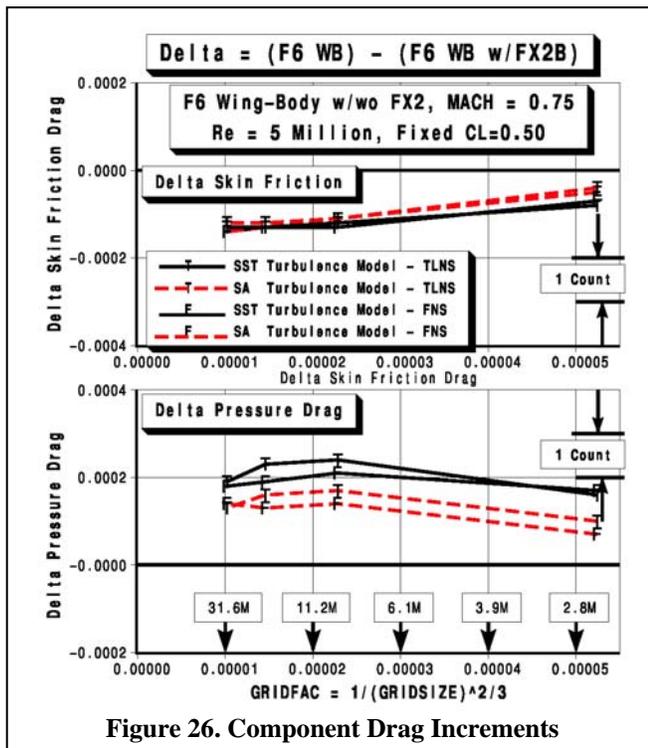
Figure 25. Drag Increments between Configurations

The incremental drag between the two configurations at a lift coefficient of CL=0.50 (basic F6 minus F6 with FX2B fairing) is shown in Figure 25. Increments are from the two turbulence models, calculated using either the thin-layer or full Navier-Stokes approximation. Using the full Navier-Stokes terms in CFL3D yields a similar variation but only half the increment values compared to the thin-layer terms. The incremental drag difference between the two configurations is very small—approximately one drag count or less regardless of the turbulence model or choice of approximation. The incremental values vary little for a wide range of grid sizes. Extrapolating to infinite grid is dangerous.

The completion of a wind tunnel test of these two configurations in the NASA National Transonic Facility in October 2007 will provide the necessary

CFD Uncertainty and Validation for Commercial Aircraft Applications

test data to evaluate the accuracy of these predictions.

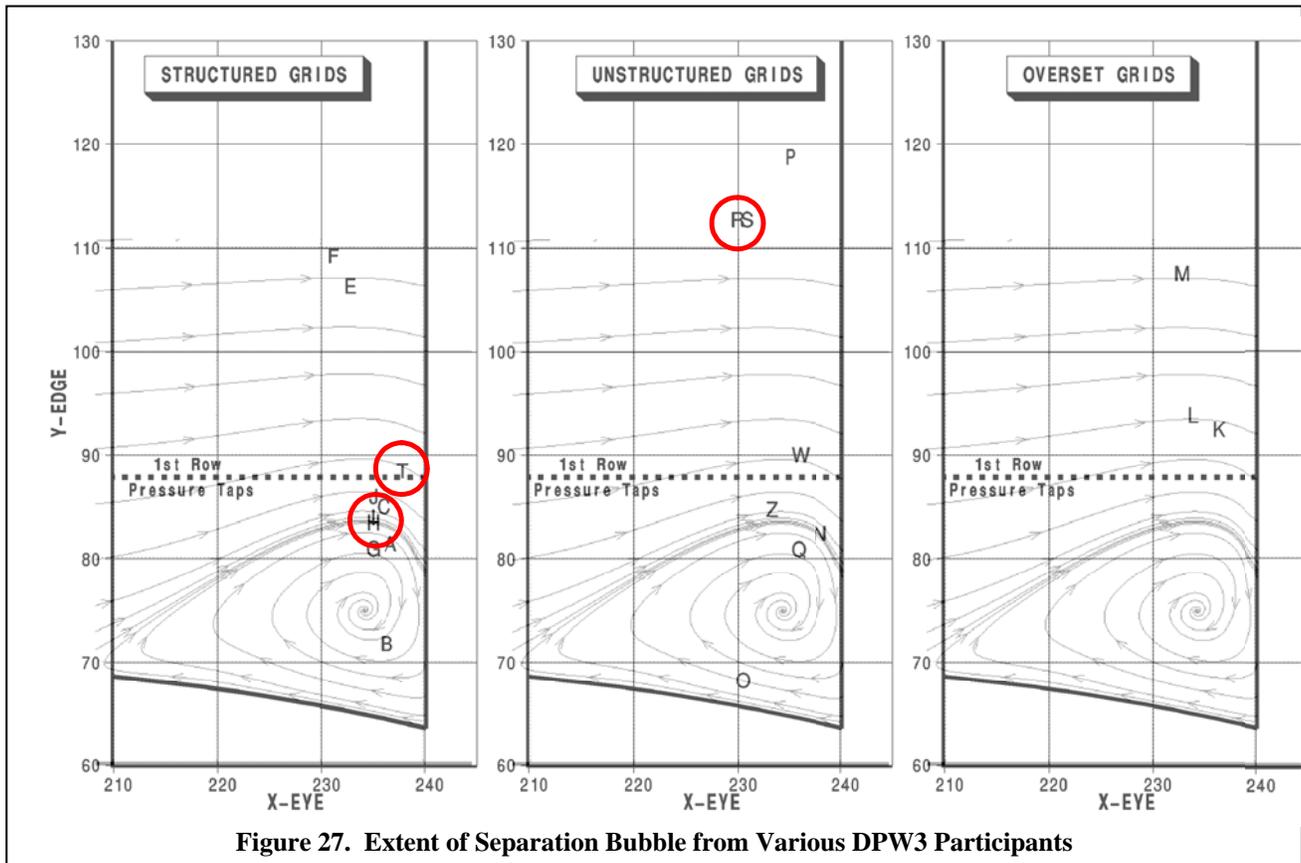


Further insight into the computational drag difference is given by looking at the component increments shown in Figure 26. The skin friction drag increment shows good convergence to the same value with increasing grid size. The configuration with the fairing has higher drag, which is consistent with the greater wetted area. The variation of the incremental drag is essentially due to the pressure drag, and this is where we see the biggest variation. The larger pressure drag increment with the SST turbulence model is consistent with the larger size of the separation bubble compared to that obtained with the SA turbulence model in the basic F6 configuration. The addition of the fairing eliminates the side-of-body separation, reducing the pressure drag at the expense of higher skin friction drag. However, the effect of using either the full or thin-layer Navier-Stokes formulation is just the opposite! The solutions using the full Navier-Stokes terms result in a smaller drag increment compared to using the thin-layer approximation although the predicted extent of the separation bubble is larger by a few millimetres.

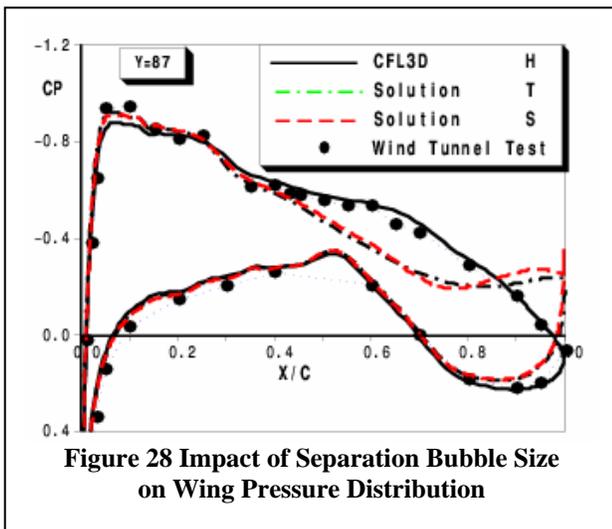
We have no explanation for this difference. The size of the drag increment and the differences shown here would make it very difficult to resolve the superior approach on the basis of this comparison alone. This goes to show just how sensitive the pressure drag is to the details of the structure and extent of the separation bubble – a flow feature that steady Reynolds-Average Navier-Stokes may not be able to properly discern.

6.3 Workshop Results – Prediction of Separation Pocket and Drag Increments

The pocket of separation in the baseline F6 configuration turns out to be a very challenging test for CFD validation. This is illustrated by the wide range of solutions submitted to the 3rd AIAA Drag Prediction Workshop³³. Figure 27 illustrates the size of the side-of-body separation as computed by the DPW3 participants using their medium size grids. For reference these data are overlaid on an oil-flow pattern taken from the ONERA wind tunnel test. *Flow visualization images from the recently completed wind tunnel test in October 2007 should provide a more direct comparison when they become available.* For clarity, these data are organized into subplots by grid type. The A-Z symbols plotted in this figure represent the edge coordinates of the computed separation. The A-Z symbols correspond to different solutions and are identified in Reference 33. Symbols G, H, I, & J correspond to solutions we submitted from CFL3D using the point matched multiblock grids we generated. Remember that our results showed little variation with grid size, turbulence model, or discretization and agree reasonably well the oil flow picture. Solutions A, B, and C by a different researcher made use of these same grids; however these solutions exhibited convergence difficulties that compromised their drag prediction. Assuming that the extent of the separation bubble at 5 million Reynolds number will be little changed from that observed at 3 million as our calculations with CFL3D indicate, we see that most other solutions predicted a much larger separation extent. There is a large scatter in the computed sized of the separation bubble.



The impact of predicting too large a separation bubble can clearly be seen in the predicted wing pressure distributions on the row of wing pressure taps closest to the body. The location of these taps relative to the predicted bubble extent was shown in Figure 27. Figure 28 shows a comparison between results from CFL3D and Solutions T and S whose predicted separation bubble extent is outside the row of pressure taps. The difference is dramatic. Worse yet is the resulting drag increment predicted between the two configurations



from all the various codes, shown in Figure 29. In most cases the increments are much larger than predicted by CFL3D. In many cases the variation with grid size is not even close to being monotonic. We know that pressure drag appears to be very sensitive to bubble size and that bubble size varied significantly with grid from some of these solutions. If we filter the solutions to include only those whose predicted bubble size was inside the first row of pressure taps, and further filter out the solutions whose pressure distributions for the F6 configuration were not close to the character exhibited by the original ONERA test data, we significantly reduce the size and variation of the predicted drag increment to those shown in Figure 30. *The recently completed wind tunnel test in October 2007 for these two configurations should confirm the small magnitude of this drag increment.*

CFD Uncertainty and Validation for Commercial Aircraft Applications

Unlike the case presented in section 3.2.2 which was a very poor choice for validation due to its extreme sensitivity to the exact flow conditions in the wind tunnel this case is probably a very good choice because it really stresses the CFD solution. I do not believe that the flow separation pocket is overly sensitive to flow conditions in the wind tunnel. We did not see any indication in the pressure distributions that were available from DPW2, *nor in the recent NTF test*. It is obvious that today's state-of-the-art Navier-Stokes solvers are having difficulty with this flow feature. The disturbing thing is we don't know exactly why. Murayama³⁵ computed a large separation bubble using their UPACS structured solver and grids they generated. Running their code with the SA turbulence model and the thin-layer approximation on our (Boeing) medium grid resulted in a small bubble comparable to our solutions. This might lead one to conclude that details of the grid topology, distribution, and density are important. They are! Running with the full Navier-Stokes terms gave a large separation. Why? Is the thin-layer and full Navier-Stokes discretization implemented differently in CFL3D and UPACS? We don't know. We just recently added the full terms to CFL3D and have not had much experience using them. We do know that the solutions with the full Navier-Stokes terms are more sensitive to grid.

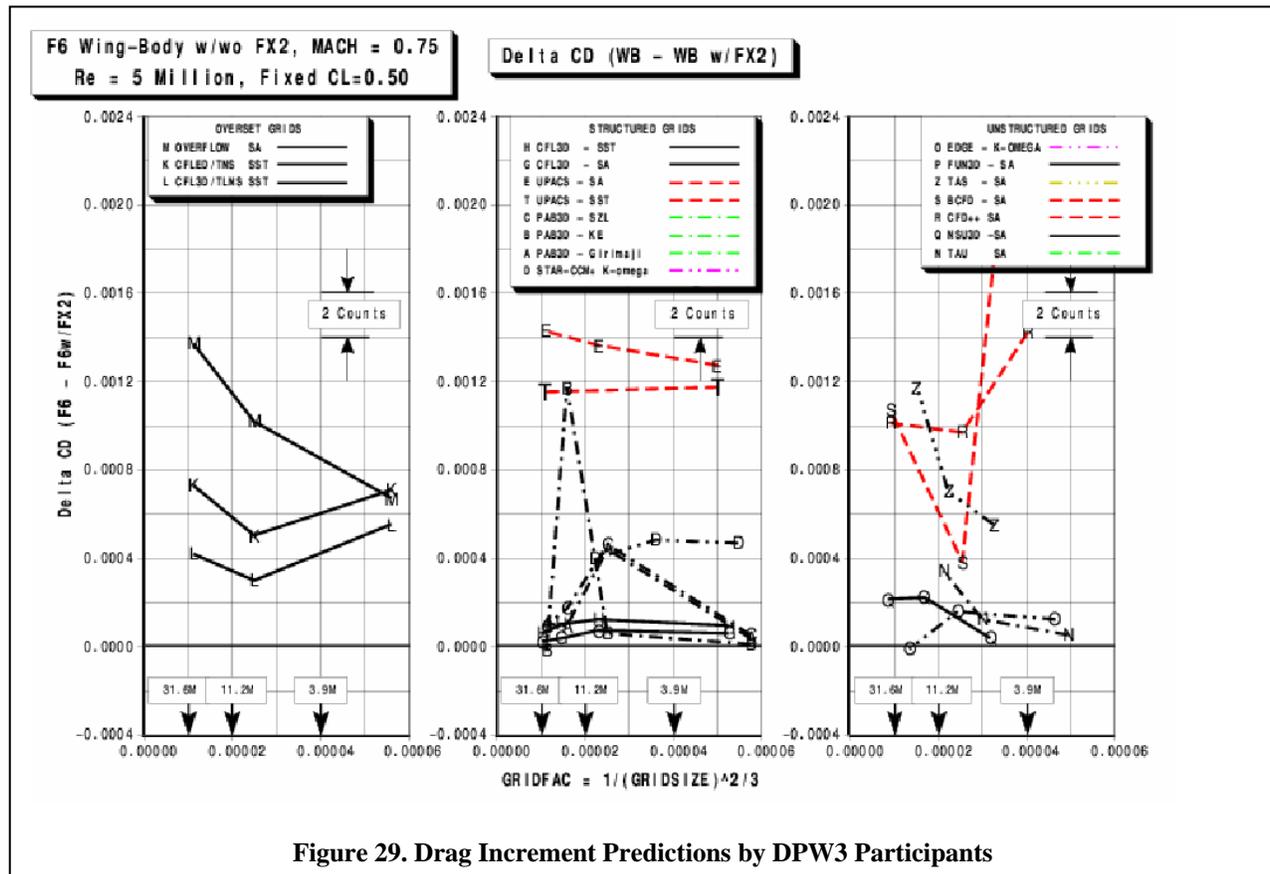
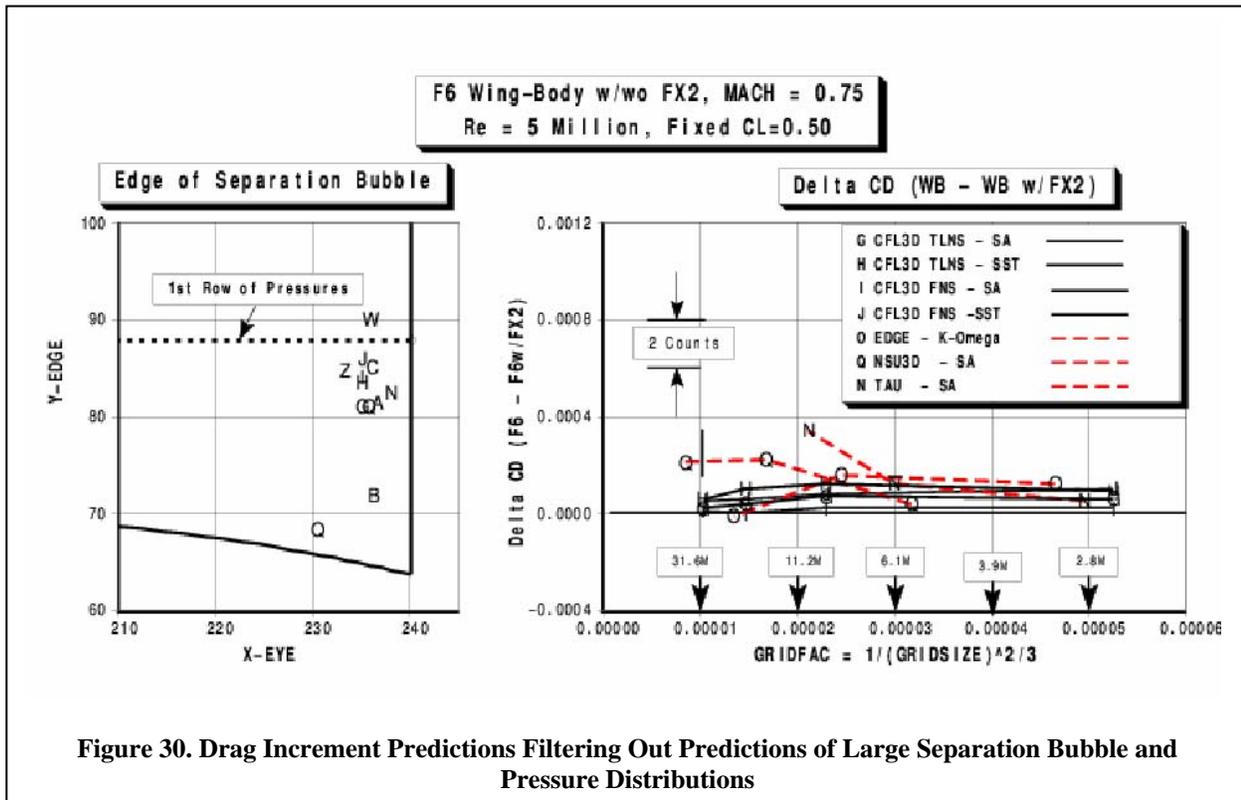


Figure 29. Drag Increment Predictions by DPW3 Participants



Based on these results (and a lot of other history) we choose to take the pragmatic view that we will rely on the incremental drag values. The values vary little over a wide range of grid size and appear to be reasonable. Extrapolating to infinite grid size is neither practical nor believable. That said, experience still dictates that it is imperative for making incremental drag comparisons, that gridding be kept as consistent as possible between configurations. Like use of the wind tunnel, great care must be taken with CFD to minimize extraneous influences that are not relative to the differences being sought. This is how you reduce uncertainty!

7.0 CONCLUDING REMARKS

The challenge is to get from the stage where the initial verification and validation has shown a new CFD code to have potential in meeting some engineering needs to actually making it part of your engineering process. Frequently this initial verification and validation is reported in the form of conference papers, or vendor brochures which tend just to show the application successes. Getting from this stage to where CFD is being used with confidence in a predictive manner to make airplane development decisions is a long and costly endeavor, frequently costing much more than the original code development or acquisition cost. The reason for this is that it is not just the code that must be validated for its intended purpose, but also the entire process of geometry, grid generation, solver, post-processing of results, and even the user that must be validated. The overriding key to validation for an intended purpose is consistency. Processes and techniques used at Boeing Commercial Airplanes to provide the needed consistency, to bound CFD uncertainties and limit their impact on design decisions have been developed in the form of “productionized” CFD packages: Stable, packaged software solutions that promote and enable consistent, repeatable processes. These integrated packaged software solutions combine the various components to go from “lofts to plots” in the time scale consistent

CFD Uncertainty and Validation for Commercial Aircraft Applications

with a fast paced engineering program, while minimizing variation. These packages include: scripted packages for “Standard” configurations; geometry and grid/paneling generation components; flow solvers; and post-processing components for analyzing the results. We are not yet smart enough to achieve the required levels of accuracy and consistency for completely general geometries so we limit our processes to the type of configurations primarily dealt with. Key to this is the on-going, never-ending, validation process. As new types of configurations become of interest they too are soon added to the stable of “Standard” configurations.

Programs such as the AIAA Drag Prediction Workshop Series^{26,31} and the CAWAPI F-16XL project^{36,37} in which several different organizations apply CFD to a common geometry and present their results in an open forum are very valuable in providing new insight to the validation process. Here we have gotten to better see the CFD shortcomings as well as the successes. These open programs provide great opportunities for information exchange on validation as well as providing new sources of data. The significant separation bubble on the DLR F6 configuration, which we may not have encountered otherwise, provided valuable lessons on the sensitivity of adequately resolving such phenomenon.

Effective CFD validation requires intimate knowledge of both the CFD and the experimental data being compared. CFD validation cannot consist of the comparison of the results of one code to those of one experiment. Examples in this paper have shown that limited comparisons can and do lead to erroneous conclusions. Rather it is the agglomeration of comparisons at multiple conditions, code to code comparisons, an understanding of the wind tunnel corrections, etc. that leads to the understanding of the CFD uncertainty and validation of its use as an engineering tool. Throughout this paper we have spoken of data, whether from wind tunnel, flight, or CFD, being “good enough” for its intended purpose. The CFD developer can not define “good enough.” Only the engineers who will be using the data can define what “good enough” is. Its definition will change depending on the stage of a program going from preliminary design studies to final data for production. Likewise the CFD developer cannot state when validation for an intended purpose has been accomplished. The engineers and their management must decide when they have the confidence in the CFD and all its related processes to get a specific job done. Achieving this requires a very close working relationship between developer, validator, and end-user. You cannot just throw validation over-the-fence. The use of CFD must make economic sense. It must deliver the “most bang for the buck,” and do it in the time constraints of an airplane development or support program.

It is difficult, if not impossible, to put a precise numerical definition on what is CFD validation and when CFD is “good enough”; but I know it when I see it. And to know it requires seeing a lot of it in order to develop the confidence that it is “good enough.”

8.0 ACKNOWLEDGEMENT

I would like to acknowledge the contributions of N. J. Yu, T-J Kao, M. Hong, J.E. Bussoletti, F. T. Johnson, T. K. Matoi, M. F. Smith and many other members of Boeing Commercial Airplanes Enabling Technology and Development Group. Work presented here was supported by various independent sources of Boeing Commercial Airplanes.

9.0 REFERENCES

1. Rubbert, P. E., "AIAA Wright Brothers Lecture: CFD and the Changing World of Airplane Design," ICAS-94-0.2, September 1994.
2. Tinoco, E. N., "The Changing Role of Computational Fluid Dynamics in Aircraft Development," AIAA-98-2512, June 1998.
3. Johnson, Forrester T., Tinoco Edward N., and Yu, N. Jong, "Thirty Years of Development and Application of CFD at Boeing Commercial Airplanes, Seattle," AIAA-2003-3439, June 2003.
4. Goldhammer, M. I., "Boeing 787 – Design for Optimal Airplane Performance," CEAS/KATnet Conference On Key Aerodynamic Technologies, Bremen, Germany, June 2005.
5. Gillette, Walt, *Aerospace America*, page 34, May, 2004.
6. Tinoco, Edward N., Bogue, David R., Kao, Tsong-Jhy, Yu, Neng J., Li, Pei, and Ball, Douglas N., "Progress Toward CFD for Full Flight Envelope," Vol. 109, No. 1100, *The Aeronautical Journal*, October 2005.
7. Guide for the Verification and Validation of Computational Fluid Dynamics Simulations, AIAA G-077-1998.
8. Cosner, Raymond R., Oberkampf, William L., Rumsey, Christopher L., Rahaim, Christopher P. and Shih, Tom I-P, "AIAA Committee on Standards for Computational Fluid Dynamics: Status and Plans," AIAA-2005-0568, January 2005.
9. Melnik, R.E., Siclari, M.J., Barber, T., and Verhof, V., "A Process for Industry Certification of Physical Simulation Codes," AIAA-1994-2235, June 1994.
10. Bussoletti, John E. "CFD Calibration and Validation: The Challenges Of Correlating Computational Model Results With Test Data", AIAA-1994-2542, June 1994.
11. Tinoco, Edward N. and Bussoletti, John E., "Minimizing CFD Uncertainty for Commercial Aircraft Applications (Invited)," AIAA-2003-407, January 2003.
12. Young, D. P., Melvin, R. G., Bieterman, M. B., Johnson, F. T., Samant, S. S., and Bussoletti, J. E., "A Locally Refined Rectangular Grid Finite Element Method: Application To Computational Fluid Dynamics and Computational Physics", *J. Comp. Phys.*, 92, 1991, pp. 1–66.
13. Jou, W. H., Huffman, W. P., Young, D. P., Melvin, R. G., Bieterman, M. B., Hilmes, C. L., and Johnson, F. T., "Practical Considerations in Aerodynamic Design Optimization" AIAA Paper 95-1730-CP, 12th Computational Fluid Dynamics Conference, June 19–22 1995, San Diego, CA.
14. Biedron, R. T., and Rumsey, C. L., "CFL3D User's Manual", NASA/TM-1998-208444, June 1998.
15. Vatsa, V. N., Sanetrik, M. D., and Parlette, E. B., "Development of a Flexible and Efficient Multigrid-Based MultiBlock Flow Solver," AIAA-93-0677, January 1993.

CFD Uncertainty and Validation for Commercial Aircraft Applications

16. Yu, N. J., Su, T. Y., and Wilkinson, W. M., "Multiblock Grid Generation Process for Complex Configuration Analysis Using Navier-Stokes Codes," AIAA-96-1995, June 1995.
17. Buning, P.G., Jepersen, D.C., Pulliam, T.H., Chan, W.M., Slotnick, J.P., Krist, S.E., and Renze, K.J., "OVERFLOW User's Manual, Version 1.8L," NASA Report, 1999.
18. Spalart, P. R. and Allmaras, S. R., "A One-Equation Turbulence Model for Aerodynamic Flows," AIAA paper 92-0439, 1992.
19. Menter, F. R., "Zonal Two Equation k-w Turbulence Models for Aerodynamic Flows," AIAA paper 93-2906, 1993.
20. Kmak, F, Hudgins, M, and Hergert, D., "Revalidation of the NASA Ames 11-By-11-Foot Transonic Wind Tunnel with a Commercial Airplane Model (Invited), AIAA-2001-454, January 2001.
21. Amaya, M.A. and Murthy, S.V., "Flow Quality Measurements in the NASA Ames Upgraded 11-by 11-ft Transonic Wind Tunnel (Invited Paper), AIAA-2000-2681, June 2000.
22. Krynytzky, Alexander J. and Hergert, Dennis W., "Boeing Transonic Wind Tunnel Upgrade Assessment," AIAA-2002-2782, June 2002.
23. Belter, Dale, L, Mejia, Kevin M, and Okada, Dustin S., "A Review of Standards Model Testing in The Boeing Transonic Wind Tunnel (Invited), AIAA-2002-2789, June 2002.
24. Wahls, R.A., Adcock, J.B., Witkowski, D.P., and Wright, F.L., "A Longitudinal Aerodynamic Data repeatability Study for a Commercial Transport Model Test in the National Transonic Facility," NASA Technical Paper 3522, August 1995.
25. Krynytzky, Alexander J. "Parametric Model Size Study of Wall Interference in Boeing Transonic Wind Tunnel Using TRANAIR," AIAA-2004-2310, June 2004.
26. 2nd AIAA Drag Prediction Workshop, <http://aac.larc.nasa.gov/tsab/cfdlarc/aiaa-dpw/Workshop2/workshop2.html>
27. Laflin, K.R, et al. "Data Summary from Second AIAA Computational Fluid Dynamics Drag Prediction Workshop, Journal of Aircraft, Vol. 42, No. 5, pp 1165-1178.
28. Tinoco, E.N., and Su, Tsu-Yi, "Drag Prediction with the Zeus/CFL3D System (Invited)," AIAA-2004-0552, Jan. 2004
29. Tinoco, Edward N., "An Assessment of CFD Prediction of Drag and Other Longitudinal Characteristics," AIAA-2001-1002, January 2001.
30. Capron, W. K., and Smit, K. L., "Advanced Aerodynamic Applications of an Interactive Geometry and Visualization System," AIAA-91-0800, January 1991.
31. AGPS – Calmar Research Corporation http://www.calmarresearch.com/software_tech/agps.htm

32. 3rd AIAA CFD Drag Prediction Workshop, <http://aaac.larc.nasa.gov/tsab/cfdlarc/aiaa-dpw/>
33. Vassberg, John C., et.al. "Summary of the Third AIAA CFD Drag Prediction Workshop," AVT147-58, December 2007
34. Tinoco, Edward N., Venkatakrishnan, Venkat, Winkler, Chad, and Mani, Mori, "Structured and Unstructured Solvers for the 3rd CFD Drag Prediction Workshop (Invited)," AIAA-2007-255, January 2007.
35. Murayama, Mitsuhiro, and Yamamoto, Kazuomi, "Comparison Study of Drag Prediction for the 3rd CFD Drag Prediction Workshop by Structured and Unstructured Mesh Method," AIAA-2007-258, January 2007.
36. Rizzi, A., Boelems, O., Jirasek, A., and Badcock, K., "What Was Learned from Numerical Simulations of F-16XL (CAWAPI) at Flight Conditions, AIAA-2007-683.
37. Boelens, Okko J., "Feature-based code validation using F-16XL flight test and wind tunnel data," AVT147-A02, October 2007



CFD Uncertainty and Validation for Commercial Aircraft Applications

Paper No. 20

Discussor's Name: A. Cenko

Question: Small differences in Mach number can have large changes in wind tunnel forces and moments at transonic speeds. Have you tried running the CFD solution at slightly different Mach numbers to see if that gives a better match to the wind tunnel results?

Author's Reply: I always like to deal with test data that has a lot of repeats to ensure that I haven't chosen a case that shows excessive sensitivity to flight conditions. If the data show this sensitivity, we can try to duplicate it.