

CFD to Flight: Some Recent Success Stories of
X-plane Design to Flight Test at the NASA
Dryden Flight Research Center

May 25, 2007

Contents

1	CFD to Flight: Some Recent Success Stories of X-plane Design to Flight Test at the NASA Dryden Flight Research Center	1
1.1	Introduction	1
1.2	Background	2
1.3	Computational Fluid Dynamics and the Aircraft Design Process	4
1.4	Comparisons and Contrasts with Wind Tunnel Data	6
1.5	Difficult Areas for Application of Computational Fluid Dynamics	8
1.6	Application of Computational Fluid Dynamics to Internal Flows	9
1.7	Computational Fluid Dynamics Analysis Yields Results Directly Applicable to Flight Test Success	10
1.8	X-36	10
1.9	X-45A UCAV	12
1.10	X-48B Blended Wing Body Demonstrator	13
1.11	Summary	15
1.12	Acknowledgements	16

Chapter 1

CFD to Flight: Some Recent Success Stories of X-plane Design to Flight Test at the NASA Dryden Flight Research Center

Gary B. Cosentino¹

1.1 Introduction

Several examples from the past decade of success stories involving the design and flight test of three true X-planes will be described: in particular, X-plane design techniques that relied heavily upon computational fluid dynamics (CFD). Three specific examples chosen from the authors personal experience are presented: the X-36 Tailless Fighter Agility Research Aircraft, the X-45A Unmanned Combat Air Vehicle, and, most recently, the X-48B Blended Wing Body Demonstrator Aircraft. An overview will be presented of the uses of CFD analysis, comparisons and contrasts with wind tunnel testing, and information derived from the CFD analysis that directly related to successful flight test. Some lessons learned on the proper application, and misapplication, of CFD are illustrated. Finally, some highlights of the flight-test results of the three example X-planes will be presented. This overview paper will discuss some of the authors experience with

¹X-48B Blended Wing Body Project Manager.

taking an aircraft shape from early concept and three-dimensional modeling through CFD analysis, wind tunnel testing, further refined CFD analysis, and, finally, flight. An overview of the key roles in which CFD plays well during this process, and some other roles in which it does not, are discussed. How wind tunnel testing complements, calibrates, and verifies CFD analysis is also covered. Lessons learned on where CFD results can be misleading are also given. Strengths and weaknesses of the various types of flow solvers, including panel methods, Euler, and Navier-Stokes techniques, are discussed. The paper concludes with the three specific examples, including some flight test video footage of the X-36, the X-45A, and the X-48B.

1.2 Background

During the decade of the 1980s the birth of the supercomputer, and the enabled application of CFD techniques, both took off in a dedicated development effort within the aircraft research and design industry. In particular, the efforts of the National Aeronautics and Space Administration (NASA) in cultivating the development of both the supercomputer and CFD were unsurpassed. At the NASA Ames Research Center (Moffet Field, California), an entire division of the organization was dedicated to obtaining and operating what was at that time the state-of-the-art supercomputer, with another division of extremely talented individuals immersed in the development, application, and validation of CFD techniques. These early CFD algorithms were targeted for and made great use of the newly procured supercomputers computing away in close proximity just across the parking lot. It was a productive, interdependent relationship. The gains made not only at Ames but across the country during that decade were unprecedented; one technology complementing and enabling the other. Because of the tremendous increases in computing speed and memory storage that were occurring almost on a quarterly basis the ambitions and abilities of the researchers grew at a rate to match. What took days or perhaps weeks to compute (and therefore was not undertaken as being impractical) in the late 1970s could be done in just several hours with the advent of the Cray-1 computer (Cray Research, Incorporated, Bloomington, Minnesota) in the 1981 timeframe. Once this computing speed became possible, the ability to increase the scope of the computations became available to the researchers, which then yielded CFD codes that required the next generation of supercomputer, and so on. Both computing speed and calculation fidelity involving ever-increasing fluid physics representations grew rapidly throughout the decade. Interestingly, the newly-emerging computer architectures required new and unique ways of actually measuring their speed and throughput. It was obvious that computing power was increasing rapidly, but evaluating that power quantitatively required the development of a set of standardized benchmarks that could be uniformly applied to the new supercomputers, and tasked them in ways representative

of the computational algorithms that were desired to be run in the day. The group at NASA Ames that was dedicated to obtaining and running the fastest computers available put together just such a set of benchmarks that measured, fairly and repeatedly, the power that was available to the researcher with each new generation of supercomputer operated at Ames (Reference 1). In the early 1980s, CFD techniques were developed for calculation of flow fields about airfoils (2D) and wings (3D) using potential flow theory. Complex fluid physics including viscosity were neglected with this formulation. Nonetheless, very useful aerodynamic solutions were obtained within reasonable amounts of central processing unit (CPU) time. With the advent of true supercomputers such as the Cray-1s in 1981, the CPU time required for some 3D potential solutions became so small (in some cases just 10 seconds), a new possibility for CFD application emerged: performing computational design optimization. Toward that end, various researchers began looking at optimization algorithms and identified one method, the so-called quasi-Newton method, as a robust and general means of driving some objective function to a local (hopefully, global) minimum. Combining this with a fast CFD flow solver able to compute the lift-to-drag (L/D) ratio of a wing geometry in seconds or minutes of computer time rather than hours allowed the new supercomputers to not only analyze aerodynamics, but alter the geometry to optimize them, within some suitable constraints. One successful example of this is given in Reference 2. As the transformational decade of the 1980s progressed, the trend toward faster processors, multiple processors, and more memory (speed) continued. Periodic upgrades of computing power could be counted on by the many researchers who were developing the complementary CFD algorithms for the new capability. The researchers, in turn, were then ready to challenge the capacity of the latest installed supercomputer to cope with these new, more complex algorithms. Increasing fidelity of fluid physics were modeled by the new CFD codes, surpassing the relatively simple panel and potential equation solutions with the non-viscous Euler equations of motion (adding vorticity effects) and finally, the full viscous Navier-Stokes equations. In parallel with the increasing fidelity of fluid physics came the increasing scope of geometry modeling. More and more dense 3D grids about complex geometries were created. Multiple body problems, internal flow problems, and even moving grids (store separation problems) were undertaken. Computational times of hours became the status quo; it seemed as though the problems undertaken were matched to the supercomputer capacity available at the time that would result in turnaround times of a few to several hours. It was as though this was the threshold of pain of the researchers and engineers for the time they could wait to see their answers. Complex problems requiring days were generally avoided, if for no other reason than it could not be reasonably assured that the computer would remain up for that length of time. A crash, reset, required maintenance, or reboot would cause such a long-running problem to be lost. Thus, a balance was struck between the computing power available,

and scope and ambition of the problem to be solved. As was stated, both areas enjoyed periodic upgrades throughout the decade.

1.3 Computational Fluid Dynamics and the Aircraft Design Process

When starting with a clean sheet of paper, the typical aircraft design process usually begins with a configurator laying out a rough sketch of the outer lines of the new shape on the computer aided design (CAD) system at hand. After the overall planform and other shaping characteristics are decided upon, the designer will usually have enough information to create a three-dimensional model of the aircraft model suitable for early, lower-order CFD analysis. Efficient, flexible CFD tools including panel and potential equation codes and other linear methods are available and are extremely useful at this early stage of design to generate preliminary aerodynamics for the new shape. This information allows redesign and refinement to proceed quickly, maturing the aircraft design almost in real time. At this point, there is sufficient detail in the CAD system to begin discretizing the shape and preparing the electronic model of the aircraft for more detailed and refined analysis, including both CFD and finite-element structural codes to be brought into the process. It is generally at this point that some state-of-the-art CFD methods are applied and detailed fluid dynamics generated. Based on these results, further design refinement can occur, going back to the CAD model from which the original analysis was based. This iterative process can continue until the designers agree that the new aircraft shape meets initial criteria and performance characteristics, and is worthy of still more complex analysis, to include building a model of the aircraft for wind tunnel testing. The author feels that it is at this point in the design process that CFD plays its most important role. Wind tunnel models are generally very expensive, costing perhaps hundreds of thousands of dollars or even more. Wind tunnel test time is a significant cost driver in a project. Viscous CFD methods applied to the candidate geometry before cutting metal for the model is generally time and effort extremely well spent. It can make, and has made, the difference between building a costly, disappointing model and one that simply verifies the adequacy of the design as predicted by the CFD methods. A no surprises wind tunnel test is generally the goal at this stage of the design process. As part of the wind tunnel test/CFD analysis stage, CFD can provide a link from the model configuration and data to the actual aircraft configuration. To support the model in the wind tunnel test section, a modification is generally made at the aft end of the shape to allow the wind tunnel sting to be inserted in the rear of the model, which in turn is connected to the strain gage balance inside the model for force and moment resolution, and at the other end, affixed to the tunnel test section itself. The modification of the shape at the aft end of the model to

accommodate the typically cylindrical sting is called the sting distortion. Obviously, the data taken is then for the shape with this distortion, not of the actual aircraft (undistorted shape). Computational fluid dynamics can provide a unique and generally quite accurate sting distortion correction, allowing the designers to correlate, using this correction, the forces and moments measured in the tunnel to what they would be on the actual, undistorted aircraft shape. In fact, one way to accomplish this is to model the aircraft shape in CFD with a perfectly expanded jet plume shape emanating from what would be the engine nozzle, which should give the forces and moments on the actual aircraft shape as it would be in flight with the engine plume. This can then be correlated to the wind tunnel measurements, which represent a distorted body shape and a cylindrical solid plume (sting). As a double check, one can also model this sting/distortion shape in CFD, giving all of the increments for correlation and prediction of actual aircraft performance. In addition to forces and moments, wind tunnel models typically have several pressure orifices in order to measure the pressure distribution on the surface of the model. By their very nature, CFD solutions provide surface pressures everywhere on the surface of the model, limited only by the grid density (discretization) of the surface. Therefore, good correlations can be made between the surface pressure measurements at the few locations on the model, and the overall pressure distribution predicted by CFD. If the correlations are good, the designer can be relatively confident that the forces and moments predicted by CFD are good as well. If not, the wind tunnel data can be used to calibrate or even improve the CFD method, by making improvements to the calculation method and/or grid density, and then having the solutions rerun. The relationship of CFD and wind tunnel testing is synergistic and complementary they are not exclusionary. In the authors experience, the best usage of CFD early in the aircraft design phase is two-fold: first, it can assist the designer (and configurator) in shaping the aircraft in a preliminary way to meet early performance criteria. Second, it can greatly aid in designing a wind tunnel model that can be reasonably expected to perform well. Gross errors in design are usually predicted well enough by CFD to make corrections with confidence in the model design. Once a model design is committed to, and cutting the metal begins, CFD analysis can continue before entry into the wind tunnel to fill out the database of flow conditions for later reference once the wind tunnel data begins to be generated. Indeed, it can be extremely useful to have the CFD-generated aerodynamic database available in the wind tunnel to correlate immediately with the data coming out of the test in real time. In the authors experience, there was even one occurrence in which the wind tunnel testing was stopped because the data did not at all correlate with the CFD predictions, and in fact, an error in the wind tunnel data reduction parameters was found and corrected. Without this capability, the tunnel testing and data acquisition would have continued with this error unnoticed, thus requiring a total recalculation of the tunnel database after the test once the error, hopefully,

was found.

1.4 Comparisons and Contrasts with Wind Tunnel Data

Using CFD effectively in the manner described in the previous section, and upon detailed comparisons of the CFD predictions to the wind tunnel data, several conclusions are generally evident. Assuming a robust and accurate Navier-Stokes flow solver (and sufficiently dense flow field grid) were used, typical strengths and weaknesses of the CFD analysis are brought forth. For flow conditions of low angle of attack and/or sideslip (benign flow with little or no separation), the CFD-predicted forces and moments as well as surface pressure distributions are usually found to be in good to very good correlation with the wind tunnel data. Once a few conditions of this nature are checked, the CFD method can be used with good confidence for examining the flow field in detail, perhaps with an eye toward minor redesign of the shape. This can help tremendously in taking the design further along the path to a prototype aircraft post-wind-tunnel test. Because the CFD analysis was found to be in good agreement with the wind tunnel data at these benign conditions, one could conclude it might be safe to trust the method to aid in refining the shape further, without, perhaps, building a new wind tunnel model and retesting. This therefore saves a step, and an expensive and time-consuming one, for the next iteration of the design process. Another strength of CFD analysis, again for these benign flow conditions, would be to extract the increment to the results of the sting distortion mentioned earlier. The overall predicted lift-to-drag (L/D) ratio of the design will be altered in the wind tunnel data due to the presence of the sting and the attendant distortion of the aft section of the aircraft. Computational fluid dynamics analysis of the undistorted shape will produce the increment of this alteration in performance. Thus, the wind tunnel data, with CFD-generated corrections, can be recalculated to better represent the aircraft design in flight. This is routinely done for aircraft configurations, especially those in which the sting is insert at the aft end (engine nozzle end) of the aircraft. It is an extremely useful correction technique. The success with which CFD can be compared to wind tunnel data is usually confined to the benign flow conditions of small to moderate angle of attack. For the higher transonic Mach numbers, where strong shocks are prominent on the aircraft shape, the ability of CFD to cope with and accurately predict flow field separation, shock-induced separation, or massive flow separation at very high angles of attack is limited at best. Highly viscous-dominated flow conditions, wherein many simplifying assumptions used in formulating CFD codes are not valid, create the limitations evidenced upon comparing both the forces and moments and the surface pressures to the wind tunnel data. What is generally found is that, for

example, the lift curve slope of the aircraft configuration is followed quite well over the small to moderate angle of attack range. Above this range, near the onset of lift breakdown, the slope generated by the CFD methods will diverge from the data. The CFD methods can still give an approximation of where the lift curve knee will appear, but likely will not be very accurate in the calculation of maximum lift coefficient (CL_{max}), for example. It may also be found that upon looking at the surface pressure distributions, the shock location, strength, and sharpness will be inaccurate. The stronger the shock waves, generally the poorer the CFD calculations of their strength will be. Specifically, over the range of approximately Mach numbers 0.90 to 1.10, the strength of the shocks and their ability to separate the boundary layer are difficult for CFD methods to model accurately. Increase the Mach numbers to, for instance, 1.2 and above, however, and the calculations again become more accurate in terms of forces and moments and surface pressure distribution predictions. Drag calculation is another general area of CFD weakness. Since the drag is usually, for most aircraft configurations, small when compared to the lift and moment forces, inaccuracies play a larger role in the values obtained. Also, since drag onset due to separation is largely a viscous-dominated flow characteristic, the extent of separation is difficult for CFD to compute well. This weakness will manifest itself when comparing drag polars of the configuration to the wind tunnel data. It should be mentioned here that even for the benign flow conditions and lower angles of attack, the absolute drag computed by the CFD method may be off by an almost constant increment over the entire range of the data, diverging finally at the more severe conditions. This increment may come about from the difference between the calculated skin friction drag and the wind tunnel data. The increment can sometimes be determined to be fairly constant at a given Mach number, therefore, upon examination of the data, it may be possible to correct this incremental difference by adding a constant to the CFD calculated drag via post-processing of the computed data, and then replotting the CFD results with the wind tunnel data superimposed, effectively compensating for this incremental error in the absolute drag numbers. Other major areas of comparison include the moment curves and the derivation of stability derivatives. Often, again for the more moderate flow conditions, these curves are fairly linear, and CFD can do a very good job of predicting these quantities. Asymmetrical calculations of a model at sideslip conditions in CFD require a full grid, without taking advantage of a symmetry plane. Therefore, these calculations double the time and computing resources required. Once comparisons with wind tunnel data are made and found to be favorable, investing these resources to compute asymmetric stability derivatives becomes worthwhile. Finally, surface pressure distributions are easily compared (if the wind tunnel model is so equipped), and for flow conditions where good comparisons exist, the CFD pressures can then be used for other detailed analysis of the configuration. For example, the dense surface pressure data can be used in conjunction with a finite-element struc-

tural model for calculation of loads and moments about the aircraft. Lastly, as was mentioned earlier, the all-important calibration of the sting distortion increment is usefully provided by CFD methods. It allows extrapolation of the characteristics of the wind tunnel model (with sting distortion) to the actual aircraft configuration with the correct aft shaping. This correction makes the wind tunnel data even more useful for prediction of full-scale aircraft performance parameters. Again, one of the best uses of CFD is to ensure that the wind tunnel model that is built yields a largely no surprises wind tunnel test. Once the CFD methods are so calibrated from one test, it is possible to apply them with even greater confidence to the next design iteration, even perhaps allowing refinements to be made to the design without the need for a subsequent wind tunnel reentry. This is clearly where investment in CFD pays dividends.

1.5 Difficult Areas for Application of Computational Fluid Dynamics

The discussion in the previous section often referred to benign or moderate flow conditions. These conditions are typically the low to medium angles of attack (or sideslip), and the low transonic or lower supersonic Mach numbers. Once significant flow separation is present, or at high transonic Mach numbers (approximately 0.90 to 1.10) where very strong shocks are present, discrepancies with test data are likely to be prominent. In typical CFD codes used for full aircraft configuration analysis, turbulence is generally modeled to some approximation in order to provide a reasonably sized problem. The various turbulence models do a fairly good job for areas of no to small separated flow. Once the separation becomes significant, with large areas of stagnated and recirculating flow, these models generally break down. The result is the under- or overprediction of the separated regions, with the attendant inaccuracies in the surface pressure distribution and integrated forces and moments. Where very strong shocks are present, first the shock strength and location are usually poorly predicted, and then the resulting flow separation and recirculation regions are accordingly mispredicted. When applying CFD under these conditions, great caution should be taken unless there are test data to either validate the results, or to calibrate the errors of the computations. Even under benign flow conditions, CFD can still be misleading when applied to certain regions of the aircraft shapes flow field. For example, applying CFD in a boattail region, perhaps in an aft-facing step area or in area of the exhaust nozzle, significant flow separation can exist even for benign flow conditions. Drag calculations for a configuration with aft-facing steps will likely be inaccurate. Configurations with landing gear in the flow stream are similarly troublesome. Landing gear are often complex shapes, both difficult to model in the computational grid, and difficult to compute for the CFD flow solver. It is often desired to evaluate the increment of drag with

landing gear down versus landing gear retracted, and thus the temptation to use CFD methods to evaluate this early in the design stage. Again, caution should be exercised in these areas of interest unless wind tunnel data is available to calibrate and correct the results.

1.6 Application of Computational Fluid Dynamics to Internal Flows

Most of the discussion thus far has been made in the context of CFD analysis of the external shape of an aircraft configuration, for the purposes of force and moment calculation and surface pressure distribution. Computational fluid dynamics can be and has been applied very effectively to the calculation of internal flows, specifically the calculation of inlet and nozzle flows including the ducting before and aft of a simulated engine. In fact, many more detailed wind tunnel models have provision for so-called flow-through configurations, wherein an inlet is uncovered and flow is allowed to enter the inlet, flow through the model, and exit at the aft end of the model near the sting area. To control the flow through this ducting, different inserts can be fabricated to choke down the flow at the exit, thus giving the effects of varying mass flow. To correctly compute the flow about (and through) such a model, it is necessary to grid and model the internal flow, including the effects of varying mass flow. This allows calculation and prediction of inlet drag and inlet spill. These effects clearly show up in the surface pressures near and immediately behind the inlet, and of course affect the drag and base pressures near the sting. Including these in the CFD model can greatly improve the agreement of the computations with the wind tunnel data. In fact, if the CFD analysis is seriously used in the detailed design of the wind tunnel model, these effects should be included. In addition to flow-through models, purely propulsion-related internal flows can also be effectively computed. Inlet and ducting designs can be fairly accurately assessed provided the attendant flow separations are reasonably subdued. Reference 3 outlines some detailed internal flow calculations and their comparisons with experimental data. Similarly, nozzle flow paths can be designed using internal CFD computations with care toward and knowledge of the inherent limitations. These are generally more complex flow problems than purely external calculations, yet very good results can be obtained that aid greatly the designers efforts to refine a configuration before committing it to costly metal fabrication and testing. In addition, CFD analysis can help direct where wind tunnel model instrumentation (e.g., pressure taps) should be placed on the surface for the best results.

1.7 Computational Fluid Dynamics Analysis Yields Results Directly Applicable to Flight Test Success

The processes of applying CFD analysis to aircraft design in the manner illustrated above, with all of its limitations, has been instrumental in first allowing a well-performing wind tunnel model to be built, and second, to allow the extrapolation of both computed and measured quantities to full-scale flight test articles. Many of the corrections to wind tunnel data, extrapolation of wind tunnel data, and modeling of physical features of an actual aircraft can play a major role in ensuring flight test success. Often, the X plane aircraft design will undergo many design changes after the wind tunnel model (and its testing) are completed. These may be minor changes, but nevertheless some quantification of the effect on aerodynamics and/or performance must be made in order to develop the flight control software. Computational fluid dynamics analysis, especially after having been compared (and somewhat calibrated) to the wind tunnel test data, can be confidently used in assessing these changes to the flight article, and the increments used to modify the database for inclusion into the flight control law development. This was done repeatedly during the design and testing process for each of the three X plane examples given in this paper, and the flight test results reflect the benefits of such an approach. In short, the CFD corrections, when properly applied where they are valid, can be more accurate than simple extrapolation. In particular, stability derivatives can be computed from the CFD forces and moments, and over the regions of validity (benign to moderate flow conditions), have been found to be quite accurate during flight test. For neutrally to highly unstable aircraft configurations, accurate calculation of these stability derivatives is critical.

1.8 X-36

First of the three examples cited in this paper, the X-36 design, followed the path described herein almost exactly, and was the authors first end-to-end success story of CFD-to-flight. The X-36 was a tailless fighter agility demonstrator aircraft that proved, both via full-scale simulation and subscale flight test, that it is possible to achieve fighter-class agility with a configuration without any vertical tail surfaces. Yaw control was provided entirely by using split ailerons (drag rudders) and/or thrust vectoring. Extensive analysis was performed on both this and precursor (similar) configurations using the full spectrum of computational analysis tools available to the designer. Each of the various types of CFD methods were used at the appropriate time during the configuration development, and each refinement of the design was aided significantly by the CFD analysis that was performed. As the design matured, more sophisticated



Figure 1.1: The X-36 Tailless Fighter Agility Research Aircraft in flight over the Mojave Desert.

CFD methods were employed. Critical to this process was an assessment of the stability and control derivatives of the design, as instability was a given byproduct of the goals of the aircraft. Thus, candidate configurations were assessed using the various analytical methods that were applied in determining the levels of instability, making sure that they were tractable given the capabilities of the modern digital flight control systems of the day. Occasionally, the analysis tools showed levels of instability that could not be tolerated, and design adjustments had to be made to bring the configuration back within the acceptable limits of stability and control.

Before designing the X-36 wind tunnel model, extensive full Navier-Stokes CFD analysis was performed on the wind tunnel configuration (including the sting distortion), again providing valuable data to fine-tune the shape and help locate instrumentation on the model. Once committed to metal, further extensive CFD runs were performed, pre-running many of the cases to be run in the wind tunnel, and producing a database similar to what was to be generated in the wind tunnel testing. Therefore, when the actual data acquisition began in the tunnel, the CFD database was already in place for early comparisons with the data, almost in real-time as it came out of the tunnel data system. It was because of this that an early error in the data reduction scheme in the wind tunnel calibrations was found and corrected before too much time had passed.



Figure 1.2: The X-36 Tailless Fighter Agility Research Aircraft in flight.

As a result of the preliminary analysis using full Navier-Stokes CFD, the initial results of the wind tunnel data indicated, as desired, no surprises. This meant that the model designed was performing exactly as had been hoped, and as had been evident from the CFD database. Further, the piloted simulation of the aircraft using the stability derivatives of the configuration as analyzed by CFD was not only entirely flyable by the test pilot, but also met the maneuverability and agility goals that were defined early in the program.

From the successes noted above, the decision was made to take the next step, and actually build a subscale flyable prototype that would be remotely piloted from a fixed ground station, and hand flown by the same pilots who evaluated the handling characteristics in the simulations performed earlier using the combination of CFD and wind tunnel derived stability coefficients. The success of the flight test was exemplary, as the X-36 flew 33 safe and successful research flights from 1997 to 1998. An assessment of the success of the flight test and stability and control characteristics is found in Reference 4.

1.9 X-45A UCAV

The second example cited in this paper is the X-45A Unmanned Combat Air Vehicle (UCAV). As can be seen in the photo below, the X-45A followed in the



Figure 1.3: Figure 3. The X-45A Unmanned Combat Air Vehicle in flight over the Mojave Desert.

footsteps of the X-36 in that it too was configured without any vertical control surfaces. In this case, oppositely deflected outboard elevons, in a so-called crow mix fashion, along with thrust vectoring, were used for yaw stability and control. The design evolution of the X-45A followed a very similar, if abbreviated, path of CFD analysis, design refinement, wind tunnel model design, and finally flight article design and fabrication. Much of success of the X-36 approach to tailless, highly unstable configuration design and control led to another tremendously successful X-45A flight test program. No fewer than 64 safe and successful research flights were conducted on two X-45A demonstrators, some of which were dual vehicle flights. Much more about the X-45A UCAV program and flight test can be found in References 5 and 6.

1.10 X-48B Blended Wing Body Demonstrator

The third and final example cited in this paper is the X-48B Blended Wing Body research aircraft. In this particular case, the configuration has been of interest and under investigation for more than 20 years. Extensive CFD analysis encompassing all methods described in this paper have been utilized to establish and refine the aerodynamics of this unique configuration. Until just recently,



Figure 1.4: Figure 4. The X-45A Unmanned Combat Air Vehicle in flight.

no actual flight test data has been obtained. The X-48B represents the first attempt to obtain quality, scalable data from both wind tunnel and flight test. The Boeing Company has subcontracted to a British firm, Cranfield Aerospace (Cranfield, Bedford, United Kingdom), the design and fabrication of two high-fidelity 8.5 percent scale models of a notional prototype (full-scale) aircraft. Careful attention has been paid to the scalability of data in order to infer, as much as possible, the flight characteristics of the full-size aircraft from the subscale flight test. The design of the shape of this aircraft has been developed and refined over many years using both CFD analysis and limited wind tunnel test data. Upon completion of the first of the two Cranfield Aerospace-built models, a wind tunnel entry at NASA Langley's Full-Scale Wind Tunnel was performed in March of 2006. This vehicle matched identically the second, flight-worthy model, which would undergo subsequent flight test. An extremely high-quality wind tunnel database for the X-48B was obtained in over five straight weeks of testing. This database, along with CFD supplements, was used to produce the flight control law derivations that would later be used to control and stabilize the aircraft in flight.

REST OF THIS SECTION TBD NOT FLOWN YET



Figure 1.5: Figure 5. The X-48B Blended Wing Body research aircraft at rest on the Rogers Dry Lake. bed

1.11 Summary

This paper has described and given examples of successful CFD application to the design process of three true X planes. The process of conceptual design, CAD modeling and refinement, followed by CFD methods application and further refinement has been described. Specifically, how CFD can aid in the design of a wind tunnel model to yield few if any surprises during wind tunnel testing was explained. Once in the wind tunnel, data can then be directly correlated to the computed CFD database, thus calibrating the CFD methodology and in some cases ensuring that the wind tunnel data reduction is being performed correctly. CFD can be and has been an enabling technology on the path to getting a new aircraft shape to flight. Controlling an inherently unstable configuration is critically dependent on determination of its aerodynamics and stability derivatives; CFD can provide preliminary estimates of these quantities accurately enough for the development of early control laws and a flyable simulation. Configuration assessments and incremental redesign can then be accomplished in a deliberate fashion, with the goal of arriving at a final configuration to be committed to more detailed (and expensive) analysis leading toward a flight model, with greatly improved chances of success.

1.12 Acknowledgements

The author is greatly indebted to two individuals who, more than anyone else, influenced and motivated me to pursue the career that I have enjoyed and benefited from so much. The author would like to thank both the late Professor A. Richard Seebass of the University of Colorado and Dr. Helmut Sobieczky of the German Deutsche Forschungs- und Versuchsanstalt für Luft- und Raumfahrt (DFVLR) for guiding and inspiring me while I was a sophomore at the University of Arizona, and later with Dean Seebass at the University of Colorado. No two individuals did more to chart my career path than they, and I will forever remember the wonderful times we had when we were all together, exploring life and CFD.

Bibliography

- [1] Bailey, D. H., et al., The NAS Parallel Benchmarks Summary and Preliminary Results, *ACM Paper 1991-89791-459-7/91/0158*.
- [2] Cosentino, Gary B. & Holst, Terry L., Numerical Optimization Design of Advanced Transonic Wing Configurations, *NASA-TM-85950*, 1. May 1984.
- [3] Won, Mark J., Wong, Michael D., Fletcher, Michael J., & Cosentino, Gary B., Application of Various Flowsolvers to a Subsonic Serpentine Diffuser Geometry *AIAA Paper 96-2648*.
- [4] Balough, Dwight L., Determination of X-36 Stability Margins Using Real-Time Frequency Response Techniques, *AIAA Paper 98-4154*.
- [5] Cosentino, Gary B. & Hirschberg, Michael J., Flight Testing Transformational Air Power: The Joint Unmanned Combat Air System (J-UCAS) X-45A Demonstrator, *ITEA Journal of Test and Evaluation*, December 2004-January 2005.
- [6] Wise, Kevin A., X-45 Program Overview and Flight Test Status, *Appl. Math. Comput. 13*, AIAA Paper 2003-6645.