Continuing Evolution of Aerodynamic Concept Development Using Collaborative Numerical and Experimental Evaluations

Russell M. Cummings* and Scott A. Morton†
United States Air Force Academy
USAF Academy, CO 80840 USA
russ.cummings@usafa.edu

Abstract
Traditionally, computational predictions and experimental evaluations of aerodynamic concepts have been conducted separately, with little collaboration other than post priori comparisons of results. This has led to distrust and even antagonism between the computational and experimental communities. These difficulties probably began when early CFD practitioners boasted that wind tunnels would become secondary in aerodynamic concept development within a few short years, a prediction that has not come true. On the contrary, we believe that a great deal of synergy can be cultivated when computational and experimental evaluations are conducted in an integrative fashion. A variety of projects where this has been done will be reviewed, including a pitching UCAV, a delta wing with periodic suction and blowing for aerodynamic control, a missile with drag brakes that caused excessive unsteady flow, a C-130 aircraft configured for air drop, and closed loop flow control. Further evolution of the numerical/experimental collaboration will be discussed showing results from flow control research where the dividing line between numerical predictions and experimental evaluations is becoming blurred. Suggestions for future directions in collaboration will also be made.

Keywords: CFD, experimental aerodynamics, unsteady aerodynamics

Introduction
“It could be argued that the process of aerodynamic investigation would be significantly enhanced if the integration of CFD and experiments was much stronger. In particular, the design and reliability of experiments could be significantly enhanced by CFD, the scope of experimental measurements extended through CFD and the credibility of the simulation results enhanced by the availability of suitable measurements from experiments. This sort of closer integration is however rare.”

At a major conference at NASA Langley in 1975, Dean R. Chapman drew on the rapid advances in computer capability to present a chart that could be used to project that computational aerodynamics would be fully developed by 1985, as shown in Fig. 1 [1]. This event, and other similar events, created a rift between CFD and experimentalists that continues to the present day! Chapman made a statement that in particular upset a great many people [1]: “. . . If history is a guide, the wind tunnels can be expected to play a secondary role to the computers in aerodynamics . . .”

*Professor, Department of Aeronautics
†Professor, Department of Aeronautics and Director, Modelling & Simulation Research Center
Experimentalists, naturally, reacted by buttressing their fortress and attacking CFD. CFD’ers went off and found their own way. No one, or very few people, collaborated. CFD’ers only used experiments to validate codes. An opportunity for advancing aerodynamic knowledge was lost.

This led to a great deal of derision and animosity between the two research communities, resulting in, among other things, jokes made at the expense of the other camp. Here is joke by the CFD researchers at NASA Ames in the mid 1970s that reflects the attitude of the time: Question: “What do you use wind tunnels for?” Answer: “They are places with lots of space, where you store your computer output.”

Advantages and Disadvantages of Experiments and Computations

We believe that it is imperative that people conducting joint computational/experimental research should be well versed in the advantages and disadvantages of BOTH the computational approach and the experimental procedure (including error analysis). Without this knowledge level at hand, collaboration is more difficult and a great deal of time can be wasted trying to resolve discrepancies in results. We will briefly describe some of the advantages and disadvantages of both experiments and computations.

Experiments

While we all could make our own lists of the advantages and disadvantages of experimental methods, there are a number of issues that come to mind immediately. Certainly, any list has its own deficiencies, so I invite the reader to add their own ideas to these lists.

Some of the strong advantages of wind tunnel testing include:

- Well known and understood capabilities
- Usually easy to set and verify free stream conditions
- Forces and moments are relatively easy to obtain
- Flowfield properties are readily available (from probes, hot wire, LDV, PIV, PSP, etc.)

Some of the disadvantages of wind tunnel testing include:

- Many measurements are intrusive and modify the flow
- Wall corrections are often required and difficult to make
- Support system corrections are often required and difficult to make
- Blockage issues must be addressed
- Model fidelity is often a challenge
Matching flight conditions can be difficult (Reynolds number, transition, etc.)

Transonic flow is especially troublesome due to nearly normal shocks

**Computations**

Perhaps the biggest disadvantage to CFD predictions is the over-optimism of the earliest users, as described in the Introduction. In fact, CFDb non-acceptance by many people led to a common lament among CFD practitioners: “No one believes CFD results except the person who ran the code, and everyone believes wind tunnel results except the person who conducted the test.” There is certainly a great deal of truth to this, but in reality there are a number of advantages and some important disadvantages to CFD.

Some of the advantages of CFD include:

- Complete flowfield prediction (all properties are predicted throughout the flow)
- Matching flight conditions is fairly straightforward
- Nonintrusive flowfield “measurements” can be made
- Steady or time accurate results are possible
- Flow visualization is easy

Some of the disadvantages of CFD include:

- Turbulence models
- Transition prediction
- Numerical dissipation
- Numerical error
- “Black box” syndrome (GIGO is still a common CFD problem)

Without understanding the strengths and weaknesses of both approaches, researchers are left to “grop” in the dark at gaining understanding into various aerodynamic phenomenon—using both approaches is often enlightening and beneficial to understanding. Several examples of collaboration will be shown that detail how experiments and CFD can be used together, and how an evolution is taking place that utilizes both approaches to their fullest capability in aerodynamic design.

**Delta Wing with Periodic Suction & Blowing for Flow Control**

The first example of a situation where close collaboration between experimentalists and computationalists paid dividends is for a delta wing study conducted at the U.S. Air Force Academy. The purpose was to determine the feasibility of using periodic suction and blowing (PSB) along the leading edge of the wing [2,3]. The 70° delta wing configuration was tested in US Air Force Academy water tunnel at $\alpha = 35^\circ$ and $Re = 40,700$ (see Fig. 2). Two dimensional PIV measurements were taken of the flow over the upper surface of the delta wing, but no force and moment data were taken.

Figure 2. PIV measurements of delta wing in wind tunnel [2,3].

To perturb the shear layer originating at the leading edge of the delta wing, a semi-spherical rubber cap was used as an oscillatory blowing and suction flow actuator. It was moved back and forth by a connecting rod, eccentrically mounted on a disk that was driven by a 560 W DC motor. The water displacement produced by the moving cap was channelled through a tube 2 cm in diameter to the hollow wing and to the length of the slot in its leading edge. With this setup, as with any oscillatory flow control
method, fluid is drawn into the actuator over half of the sinusoidal cycle, and ejected over the other half ($V = V_o \sin \omega t$). The phase during the forcing cycle is determined by the position of the rotating disk flywheel, which features an adjustable optical pickup to synchronize the data acquisition with a particular phase of the forcing cycle. A forcing cycle starts at 0° with the blowing phase which extends to 180°. The suction portion between 180° and 360° completes the cycle.

One of the problems encountered during the experimental phase of the investigation was that it did not appear that the suction phase was equally as effective as the blowing phase of the periodic cycle. While this observation was important to the experiment, no direct reasons for the apparent anomaly were known, leaving the experimentalists to wonder if their apparatus was operating correctly, or if there was some fluid dynamic interaction at work. Another difficulty realized by the researchers was that while they knew the impact of the suction and blowing on the flowfield, they did not know the impact on the aerodynamic forces of the delta wing.

Double frequencies for the converged PSB case are more easily seen in Fig. 3, which shows the normal force variation for 17,000 iterations (over ten cycles of the suction and blowing). The suction and blowing frequency is obvious, but overlayed on that frequency is the shear-layer instability frequency, constantly oscillating around the lower frequency. Also notice that the blowing portion of the suction/blowing cycle is more effective, as evidenced by the amount of time the normal force remains at the highest levels. When the suction cycle takes place, decreasing the normal force, the force spikes to a minimum value but then quickly rises again as the suction phase ends. This explains why, during the experimental portion of this work, it appeared that the suction was incomplete (or possibly working incorrectly). Even the numerical simulation clearly shows that the suction phase is not as effective in altering the normal force acting on the delta wing. There is also a slight dwell as suction begins, which was also not explained by the PIV results.

![Figure 3. Normal force variation for periodic suction and blowing, $\Delta t^*=0.006$ [2,3].](image)

Again, the CFD results were able to answer some of these questions due to the ability to interrogate the flowfield at all locations and at all times. Figure 4 shows the velocity vectors in the vicinity of the delta wing leading edge at the 60% chord location. During the blowing phase (90 deg.), Fig. 4a shows that the fluid is able to expel directly into the surrounding flow and have a direct impact on the shear layer region. However, during the suction phase (270 deg.), Fig. 4b shows that the flow in the vicinity of the leading edge of the delta wing is not able to turn the sharp corner and be fully brought into the PSB channel. This explains the difference seen in Fig. 4 between suction and blowing, and may also explain the dwell during the blowing phase, since the flow is attached and fully formed during this phase.
Figure 4. Velocity vectors near delta wing leading edge showing difference between suction and blowing [2,3].

Pitching UCAV Configuration

A full-scale model for the Boeing 1301 UCAV configuration is shown in Fig. 5; the configuration has many similar features to the X-45A UCAV configuration. The 1301 configuration has a straight, 50° sweep leading edge, an aspect ratio of 3.1, rounded leading edges, a top-mounted engine inlet, and a blended wing/body planform. A 1:46.2 scale model of the configuration was tested in the USAF Academy 3 ft × 3 ft (0.914 m × 0.914 m) open return low-speed wind tunnel [4]. The scaled model has a mean aerodynamic chord of 5.24 in (0.133 m) and a reference area (wing planform area) of 46.82 in² (302.1 cm²). The tunnel has less than 0.05% free stream turbulence levels at all speeds. The test was conducted at a free stream velocity of 65.4 ft/s (20 m/s), which corresponds to a chord-based Reynolds number of 1.42 × 10⁵. The model was sting-mounted from the rear, and forces and moments were measured with a six-component force balance. Both static and dynamic testing was done; forces during the dynamic runs were obtained by subtracting the force history with the tunnel off from the dynamic data. The dynamic pitching was done with a shifted cosine oscillation, starting at a certain angle of attack and pitching up to twice the peak amplitude of the cosine wave, then back to the original angle of attack.

\[ \alpha(t) = \alpha_o + m \cdot \cos(\omega t) \]

where \( \alpha_o \) and \( m \) were varied to obtain results for \( 0^\circ \leq \alpha \leq 45^\circ \) in three pitching cycles. This pitching function was used since it produces a motion without any discontinuities in acceleration or velocity at the beginning and end of the motion, thus being easier to implement in an experiment or a CFD code.

Figure 5. Boeing 1301 UCAV configuration [4].
One of the limitations and difficulties encountered during the experiment was that the experiment only was able to measure forces. This is a common problem during wind tunnel tests, where tests are either of the force and moment variety or of the flowfield property variety, but rarely does an experiment include both sets of measurements. Because of this, researchers are usually unsure of the fluid dynamic causes of various results, being left to make educated guesses about unusual or unexpected results. For example in the case of the UCAV wind tunnel tests, the vehicle lift coefficient showed linear lift characteristics up to 10° to 12° angle of attack, as shown in Fig. 6. Wing stall was evident at about 20° angle of attack, with the lift being re-established up to 32°, after which an abrupt loss of lift takes place. But what is the cause of the poor lift characteristics? Are the results a direct effect of leading-edge vortices and vortex breakdown? The experimentalist is left to hypothesize and wonder, but the numerical researcher and add insight into the problem.

Figure 6. Numerical (time accurate) and experimental (static) force coefficient comparison [4]. Also shown in Fig. 6 are the results of the CFD predictions. Perhaps the most important result of the CFD simulation was the realization of just how unsteady the flowfields in the post-stall region were. Time-accurate results matched the experiment fairly well, with fairly good modeling of the flowfield, including drag, up to \( \alpha = 45^\circ \). However, there was a difference in lift from \( \alpha = 20^\circ \) to \( \alpha = 30^\circ \), which could have been caused by the presence of the sting, surface roughness, transition, or a host of other phenomenon.

Figure 7 shows representative numerical simulations of the configuration at \( \alpha = 5^\circ, 10^\circ, 15^\circ, \) and \( 20^\circ \), with the flowfield being visualized with streamlines and the surface coloured with pressure. The leading-edge vortices are clearly visible closely following the 50° sweep, until approximately \( x/l = 0.40 \) when vortex breakdown is evident. Low surface pressures are visible beneath the vortex prior to breakdown; these low pressures account for the lift on the configuration at \( \alpha = 20^\circ \). After breakdown, the vortex wake quickly moves up and behind the leading-edge, leading to higher pressures on the upper surface of the wing. The vortices are very wide compared with their height, most likely due to the rounded leading edges of the wing. Secondary vortices are also visible beneath the primary vortices. The primary vortex is seen splitting into two flow structures after the breakdown location.
Numerical flowfield predictions for various angles of attack (surface coloured by pressure) [4].

These numerical simulations help to answer some of the questions raised by the wind tunnel tests. The rounded leading edges and mid-range leading-edge sweep yield weak leading-edge vortices that do not produce very much lift. The vortices are only just beginning to form (and are very weak) when breakdown takes place and reduces lifting benefit of the vortices. These are common characteristics of “lambda” type wings, but seeing the CFD simulation helps to place understanding into the wind tunnel tests data.

ARGUS Missile Configuration

The Advanced Remote Ground Unattended Sensor (ARGUS) missile configuration was tested in the subsonic wind tunnel at the US Air Force Academy (as shown in Fig. 8), where a companion CFD study was also completed [5, 6]. A 61.5% model tested in was tested for $M_\infty = 0.2$ to 0.5, at angles of attack ranging from $\alpha = -5^\circ$ to 20$^\circ$, and for a length-based Reynolds number of $Re_L = 3.5 \times 10^6$ to 8.7 $\times$ $10^6$. The missile has drag brakes that are used to control velocity, which is critical to the overall mission of the vehicle.

Figure 8. ARGUS configuration in USAFA Subsonic Wind Tunnel.

The configuration had a number of unusual aerodynamic features, including a production of negative lift at positive angles of attack and a substantial coning motion during flight. While the wind tunnel test could give results that verified these behaviors, the test could not explain why the missile behaved as it did. Figure 9 shows the CFD predictions for the configuration at a free stream Mach number of 0.5.
While the normal force coefficient variation with angle of attack looks reasonable, the CFD also predicted that the configuration produced negative lift as a function of angle of attack up to about $\alpha = 15^\circ$. Again, the wind tunnel results did not shed any light on why this was happening.

One of the unique features of CFD is the ability to account for forces on a variety of surfaces in the flow. For example, using Cobalt, it was possible to account for forces on the missile body as well as the drag brakes, which might shed light on the cause of the negative lift coefficients. Figure 10 shows the total lift coefficient, as well as the components due to the body and the fins. It is fairly obvious that the body is producing lift normally, and that the negative lift is coming from the fins. What is not so obvious is why the fins are producing negative lift, but an evaluation of the flowfield can quickly answer this question.

Flow visualization in Fig. 11 begins to shine light on the situation, as the curved drag brake extension arm is seen creating a region of low pressure on the upper surface of the fin at $\alpha = 20^\circ$. The vorticity contours in the vicinity of the drag brake show flow separation over the support arm which extends over most of the upper half of the brake. Therefore, the lower surface of the brake has attached flow, but the curved nature of the brake creates a negative lift coefficient, while the upper surface of the brake has separated flow and does not counter the force or moment created on the lower surface.
One of the unforeseen problems with the drag brakes were the unsteady forces and moments on the configuration, leading to yaw-roll coupling and coning motion during flight. To alleviate this problem, perforations were added to the drag brakes, but a basic understanding of the fluid dynamic reasons for the flow were not known from the wind tunnel test. Figure 12a shows the DES simulation of the baseline drag brakes, and when animated it becomes clear that the blunt drag brakes are created vortex rings which shed from each brake at different times—a classic vortex shedding flowfield. Figure 12b shows the results for perforated drag brakes (the same iso-surface levels were used for both figures) where now vortex shedding is evident, even though there is still low levels of flow unsteadiness. The perforations solved the vortex shedding problem and alleviated the coning motion of the configuration.

C-130 Airdrop Configuration

This is a case where CFD and experiments were being conducted in a collaborative fashion. A wind tunnel model of a C-130 was being tested at various wind tunnels, with different types of flowfield tests being conducted at each tunnel (force and moments, surface flow visualization, etc.). As the wind tunnel tests proceeded, the results were in disagreement with the CFD simulations being performed, leading to a great deal of hand wringing and difficulty. Finally, after a lot of hard work, it was discovered that the wind tunnel model had been degrading in shape as the various tests were being performed (see Fig. 13), leading to more and more configuration mismatch between the model and the original CAD description which had been used to create the CFD grids.

This led to a study of the model material’s chemical compatibility with products to be used in the wind tunnel tests, including (possibly):
- Fluorescent viscous wall coatings
- Acetone
- Black paint
- Fog generating liquid, etc.
- Filling of holes and gaps with putty, etc.

Figure 13. C-130 wind tunnel model part showing decomposition of surface shape.

A 3D optical digitizing of the wind tunnel model was performed at ENSICA (achieved by GOM Company, as shown in Fig. 14) in Toulouse and comparisons were made with the original CAD geometry. Figure 15 shows the differences between the original CAD geometry and the actual model being tested in the ENSICA wind tunnels, with large variations evident at various locations around the fuselage and horizontal tails. In fact the tips of the horizontal tails appear to be bent as much as 2mm away from the original CAD shape. This shows another example of how CFD and experiments can be used together to insure accuracy of results in aerodynamic evaluation.

Figure 14. 3D optical digitizing of C-130 wind tunnel model.

Figure 15. Differences between C-130 wind tunnel model and original CAD definition: coloured scale: -2 to +2 mm.
Closed Loop Flow Control

A novel combination of numerical and experimental evaluation is being conducted to show the effect of feedback flow control on the wake of a circular cylinder at a Reynolds number of 120, as shown in Fig. 16 [7,8,9]. An initial 2D numerical simulation of the laminar flow was investigated in direct numerical simulation using proper orthogonal decomposition (POD) by placing sensors at various locations downstream of the cylinder as shown in Fig. 17. The flow was computed using the commercial Navier-Stokes solver Cobalt, and the POD analysis was done with MatLab. Also shown in Fig. 17 is the feedback loop after information from MatLab is used to oscillate the cylinder normal to the free stream flow in order to excite or dissipate the vortex wake. The CFD was used to determine optimal number and location of the sensors in order to accurately (to required levels) describe the flow.

![Figure 16. Flow Visualization of the cylinder wake at Re = 120, forced at the natural shedding frequency with an amplitude of 30% of the cylinder diameter [7-9].](image)

In the unforced flow, the vortices roll up between 1 and 2 diameters downstream of the cylinder, while in the feedback controlled situation the rollup occurs between 3 and 4 diameters downstream, as shown in Figure 18. Simultaneous with the lengthening of the recirculation zone, the researchers observed a reduction in the vortex shedding frequency. In the low drag state, the near wake is entirely steady, while the far wake exhibits vortex shedding at a reduced intensity. The forced case achieved a drag reduction of close to 90% of the vortex-induced drag, and lowered the unsteady lift force by the same amount.

![Figure 17. Flow geometry around a circular cylinder including sensor placement and control concept [7-9].](image)
This success of the low dimensional feedback control of the circular cylinder wake in the two dimensional CFD simulation led to the implementation of the control approach in a water tunnel experiment. Reynolds number and actuation of the experiment match the simulation exactly. An in-house developed real time PIV system was used to provide sensor information at the same downstream and flow normal locations used in the CFD simulation, using a grid of 35 off-body sensors. The main difference is that the simulation was two-dimensional, while the water tunnel model features a three-dimensional model and flow field with an aspect ratio of more than 40. With these experimental findings, 3D numerical simulations were performed to gather quantitative data along the span of the model, which is not possible with current state of the art experimental measurement techniques. The simulation setup shown in Figure 19 resembles the water tunnel experiment in terms of aspect ratio, Reynolds number and feedback control method employed. This approach showcases the highest level of integration of CFD and experiments that the authors have seen and shows what can be accomplished with the best attributes of each approach are used together.

**Conclusions**

Five cases of CFD/experimental interaction have been presented, with each case showing a different way in which research is improved by collaboration. While some of the examples merely show companion studies of CFD and wind tunnel tests, even in this simplest form of collaboration a great deal is to be learned. Unfortunately, most research is only performed from an experimental or computational viewpoint, leaving us to wonder how much improvement in aerodynamic design improved collaboration could bring about.

Here are just some examples of what could be done with collaboration (these are partially based on the Glasgow symposium):

- knowledge of the flow before deciding what should be measured
- knowledge of model geometry accuracy and fidelity
- having checks in place on the experimental measurements as they are taken
- overcoming difficulty in making certain important measurements
assessment of the influence of the experimental techniques on the measurements
- the ability of CFD to provide detailed flow information and sensitivity at a reasonable cost for some cases
- CFD validation and improvement enhanced by good experiments

Unfortunately, there are still a number of problems that exist when trying to collaborate:
- the large cost (and time) of CFD calculations for certain cases (especially massively separated flows)
- the lack of credibility for CFD results for some flow categories (transition, shock/boundary layer interactions, chemically reacting flows, etc.)
- the large cost (and time) in conducting certain experiments

We believe that collaboration is only limited by our imagination and level of determination. If we can visualize experiments and CFD as being complimentary they can be used to bring out the best attributes of each. In fact, the strengths and weaknesses of the two approaches

Acknowledgments

The authors wish to thank Stefan Siegel of the U.S. Air Force Academy for the flow control simulation information, and Bruno Vignes and Yannick Bury of ENSICA for providing the C-130 information. We also thank Boeing Military Aircraft in St. Louis, MO for providing the UCAV 1301 geometry.

References


