Lessons Learned and Future Goals of the High Lift Prediction Workshops

Christopher L. Rumsey and Elizabeth Lee-Rausch
NASA Langley Research Center
Hampton, VA 23681
USA

Jeffrey P. Slotnick
The Boeing Company
Huntington Beach, CA 92647
USA

c.l.rumsey@nasa.gov

ABSTRACT

The American Institute of Aeronautics and Astronautics (AIAA) High Lift Prediction Workshop series is described. Two workshops have been held to date. Major conclusions are summarized, and plans for future workshops are outlined. A compilation of lessons learned from the first two workshops is provided. This compilation includes a summary of needs for future high-lift experiments that are intended for computational fluid dynamics (CFD) validation.

1.0 INTRODUCTION

Today's computational fluid dynamics (CFD) codes have proven to be reliable and consistent for attached-flow, cruise-type configurations, but the complexities inherent in high-lift configurations add a significant degree of uncertainty. The AIAA High Lift Prediction Workshop (HiLiftPW) series was designed to bring together experts who all run the same high-lift CFD cases, with the expectation that insights will emerge as a result of the common focus and peer interactions. The workshops have been extremely successful to date, with two workshops already held and a third in planning stages (see the website http://hiliftpw.larc.nasa.gov).

To advance the state of the art in predicting high-lift flows, which have relevance for both civilian and military air vehicles, the HiLiftPW series (patterned after the AIAA Drag Prediction Workshop series, e.g., Vassberg et al. [1]) was established with the following long-term objectives: (1) assess the numerical prediction capability (mesh, numerics, turbulence modeling, high-performance computing requirements, etc.) of current-generation CFD technology for swept, medium/high-aspect ratio wings in landing/take-off (high lift) configurations, (2) develop practical modeling guidelines for CFD prediction of high lift flow fields, (3) advance the understanding of high lift flow physics to enable development of more accurate prediction methods and tools, (4) enhance CFD prediction capability for practical high lift aerodynamic design and optimization, (5) provide an impartial forum for evaluating the effectiveness of existing computer codes and modeling techniques, and (6) identify areas needing additional research and development.

Throughout the course of the workshops, the high lift configurations have become progressively more complex and realistic, and the modeling demands on the participants have ramped up as well. The first workshop [2] was held in 2010 and the second [3] in 2013. The first focused on the relatively simple NASA trapezoidal wing configuration, while the second made use of the more realistic DLR-F11 configuration. The third workshop is scheduled for summer 2017, and will make use of both the JAXA high lift configuration [4] (including engine installation effects) as well as a newly-designed Common Research Model high lift configuration [5].

Generally speaking (ignoring outliers), CFD simulation results have been reasonably consistent and reliable at lower angles of attack in the linear portion of the lift curve, far from stall. But very large discrepancies in the data comparisons between the various CFD codes/models and experiment often occur near maximum lift (C_{Lmax}). This area particularly needs greater understanding. Examples of the type of behavior exhibited by the CFD participants on the lift curves for HiLiftPW-1 and HiLiftPW-2 will be given in Section 2.

Like most wind-tunnel campaigns, high lift testing has typically focused on obtaining forces, moments, and surface pressures. These quantities are obviously required, but additional experimental insight into the high-lift flow physics is also desperately needed, especially near C_{Lmax}. Some velocity probe data, transition information, and surface flow visualizations have been obtained for previous high-lift configurations, but have been somewhat limited in scope. Measurements of flow field turbulence information have not yet been made. Not only is more of this type of information needed, but the reliability/uncertainty of such difficult-to-obtain quantities are also paramount.

2.0 SUMMARY OF PAST WORKSHOPS

In this section, we summarize some of the major results and conclusions from the first two high lift workshops.

2.1 HiLiftPW-1

The first AIAA CFD High-Lift Prediction Workshop was held in Chicago, Illinois, in June 2010 [2]. This workshop analyzed the flow over the NASA trapezoidal wing model [6]. It addressed two different slat/flap configurations at a Reynolds number of 4.3 million based on mean aerodynamic chord. The experiment was a semi-span test, but the workshop called for “free-air” fully-turbulent computations. Twenty-one participants from eight countries and 18 organizations submitted a total of 39 data sets of CFD results. A variety of grid systems (both structured and unstructured) were used. Trends due to flap angle were analyzed, and effects of grid family, grid density, solver, and turbulence model were addressed. Some participants also assessed the effects of support brackets used to attach the flap and slat to the main wing for the experiment.

In general, CFD results tended to under-predict lift, drag, and the magnitude of the pitching moment (moment was negative) compared with experiment. Predicting the flow was more difficult at higher angles of attack nearing stall; there was more spread among the solutions, and some participants predicted early stall. See figure 1-1. At the workshop, some participants reported initial condition dependency of their CFD solutions at high angles of attack. In other words, unless a high-angle-of-attack solution was initiated with a converged flow field obtained at a lower angle of attack, sometimes massive separation could result. This dependency was a likely reason for some of the very poor results at high angle of attack. In spite of tending to predict lift somewhat too low in general, many participants were able to predict C_{Lmax} and the angle of attack at which it occurred reasonably well. Participants also generally captured the trend of lift coefficient difference between the two configurations.

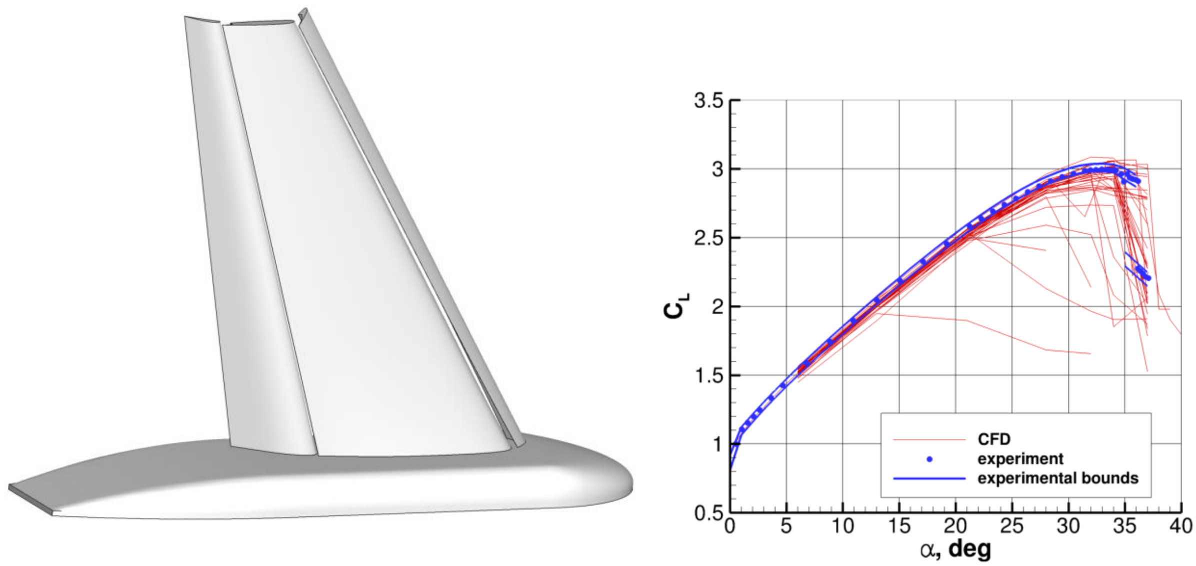

Figure 1-1: NASA trapezoidal wing configuration (left); CFD and experimental results for lift curve (right).

CFD grid convergence studies were only done without support brackets for one of the configurations. Furthermore, by design this first workshop asked for CFD results with no transition modeling or tunnel walls, and these aspects are likely to be important when comparing directly with the experiment.

From the limited pressure results analyzed, the following trends were observed. First, there tended to be greater variation among the CFD results at the outboard trailing-edge region of the flap, and the fully turbulent SST model [7] had a greater tendency to separate there than the fully turbulent SA model [8]. Second, an unstructured tetrahedral grid was found to exhibit greater grid sensitivity than the same grid with its boundary-layer tetrahedra merged into prisms. Third, the flow field near the wing tip was very difficult to predict accurately: all entries except one seriously under-predicted the suction near the wing-tip upper surface. Fourth, neglecting viscous cross-derivative terms yielded even worse predictions near the wing tip than full Navier–Stokes.

Collectively, the SA model yielded higher lift levels than other models near stall, in better agreement with experiment. Two exceptions to this trend were entries that also employed transition modeling or specification. Thus, the possible need to account for transition in the CFD at wind-tunnel-scale Reynolds numbers was highlighted.

At this workshop, the importance of including the support brackets in the CFD computations (particularly nearing C_{Lmax}) was recognized. Figure 1-2 shows an example of the influence of the slat brackets on the flap surface pressure coefficients near mid-span at an angle of attack of 28 degrees. Including the brackets in the CFD computation improved the predictions at this location considerably. The need to reduce variability in the CFD was also highlighted. For example, collecting information regarding the particular version of turbulence model employed, as well as numerical details (such as use of thin-layer vs. full Navier-Stokes) helps reduce code-to-code differences and adds to our collective knowledge regarding the influence of these factors. Furthermore, the need for sufficient iterative convergence (which is typically left to the discretion of the participant) was noted.

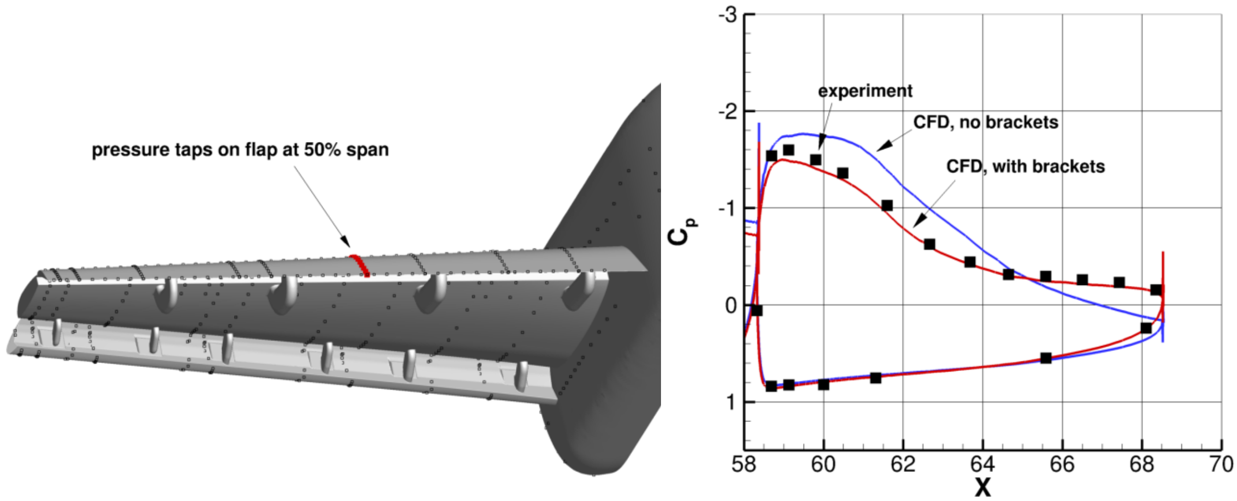

Figure 1-2: NASA trapezoidal wing configuration with brackets and pressure tap locations (left); example CFD surface pressure coefficient results on flap at 50% span station, angle of attack = 28 degrees (right).

2.2 HiLiftPW-2

The Second AIAA CFD High-Lift Prediction Workshop was held in San Diego, California in June 2013 [3]. This workshop analyzed the flow over the DLR-F11 model [9] in landing configuration at two different Reynolds numbers (1.35 million and 15.1 million based on mean aerodynamic chord). The experiments were semi-span tests, but the workshop called for "free-air" fully-turbulent computations (use of transition specification or transition models was optional). Twenty-six participants submitted a total of 48 data sets of computational fluid dynamics results. A variety of grid systems (both structured and unstructured) were used. Trends due to grid density and Reynolds number were analyzed, and effects of support brackets were also included. The DLR-F11 configuration is shown in figure 1-3.

For the DLR-F11 case in general, the importance of including slat and flap brackets, when comparing with the experiment, was established. From previous experimental oil flow, two slat tracks were apparently influential in causing large wedge-shaped regions of separated flow on the main element near stall. Without brackets, the CFD could not possibly capture this type of stall mechanism, and indeed, most entries without brackets tended to predict increasing C_L well past the nominal stall angle. See figure 1-4. There was also some evidence that pressure tube bundles included alongside the slat tracks on the wind-tunnel model had an influence on the flow field near C_{Lmax}.

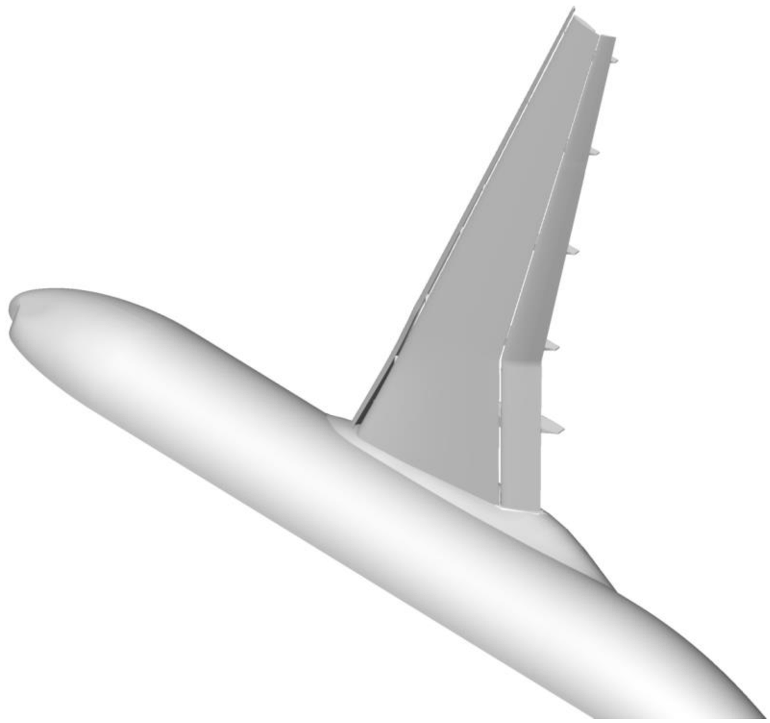

Figure 1-3: DLR-F11 configuration.

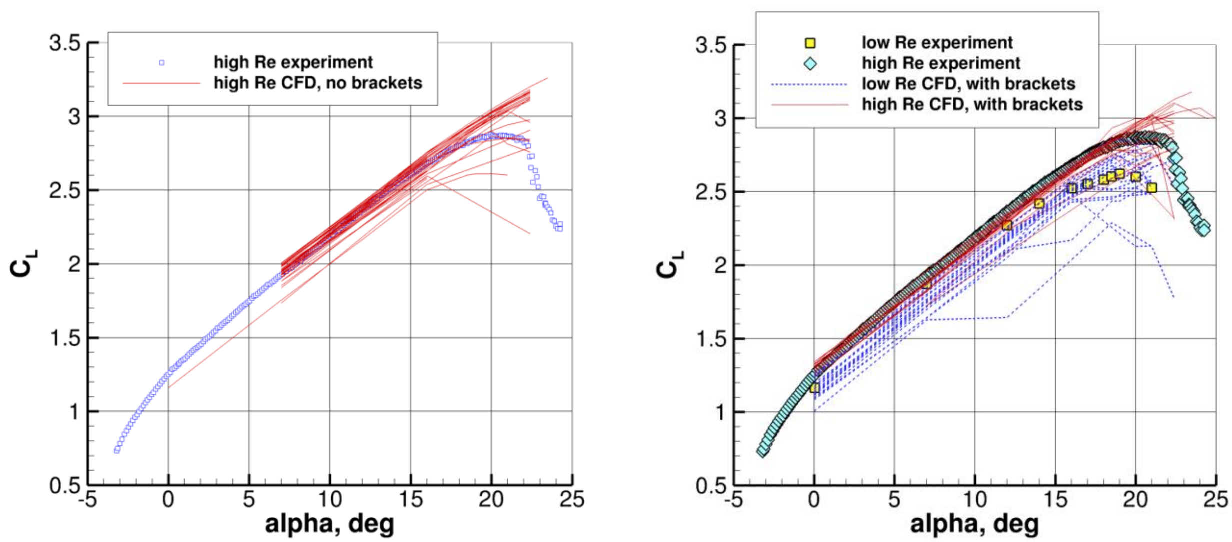

Figure 1-4: DLR-F11 lift curves; high Re with no brackets in the CFD (left); both Re with brackets in the CFD (right).

In terms of consistency, the CFD results exhibited spreads that tended to remain about the same when going from the medium to fine grid levels. In other words, the CFD scatter did not decrease much past a certain grid-refinement level; the reasons for this are not known. Scatter was larger at the angles of attack near stall, as expected (refer to figure 1-4). A small part of the CFD inconsistencies may have been due to poor or insufficient iterative convergence, as the participants had some difficulty fully converging many of the cases. Looking at details, such as pressure coefficients and velocity profiles, the inconsistencies between the CFD results tended to be larger on the flap as well as at the outboard stations of the wing. Some of the unstructured grids were shown to be poor for capturing the wakes of upstream elements, because of insufficient grid resolution in those areas.

No clear trends with respect to turbulence modeling were exhibited in the results. A few participants investigated the use of transition, but unlike the results for the NASA trapezoidal wing of the first high-lift workshop, the results were mixed and no clear trends stood out. The Reynolds-number trends were only qualitatively captured by the CFD. There were some velocity profile data available from the low Reynolds number experiment, but generally agreement between CFD and experiment was fairly poor. An example is shown in figure 1-5 for a line (shown in the figure inset) emanating from the flap. The experiment (that employed particle image velocimetry) was limited in how close it could get to the body, but nonetheless there was a clear offset in the velocity predictions outside of the main element wake. The reason for this offset between CFD and experiment is not known.

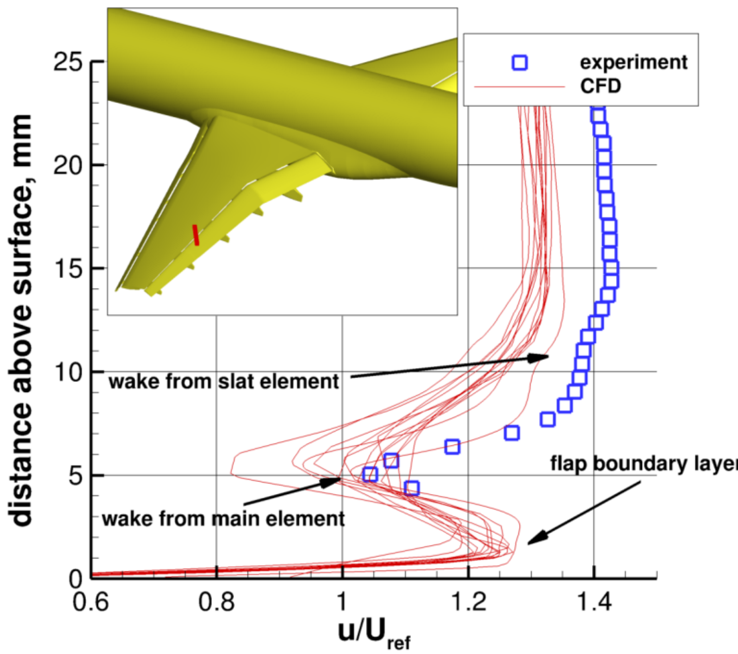

Figure 1-5: Velocity profiles along a line on the flap of the low Re case, configuration angle of attack of 7 deg.

An optional turbulence-model verification case was included in this workshop, and was computed by a few of the participants. For those who used the SA model, there was near-perfect consistency among three entries, one other entry was very close, and two entries exhibited notable differences. These two differences suggest potential problems or inconsistencies in implementation. There was some correlation between the poorer results for the simple verification case and a tendency to be further away from other SA model results for the more complex DLR-F11 case [3].

3.0 PLANS FOR FUTURE WORKSHOPS

The HiLiftPW-3 will be held in Denver, Colorado in June 2017. At this time, the plan is to have three test cases.

The first case will be a grid refinement study on the Common Research Model (CRM) high lift configuration [5]. This will use a simplified baseline landing configuration with no nacelle/pylon, no support brackets, and a single segment slat. The main configuration will include a flap with accentuated (larger) gaps, to facilitate mesh generation. A second optional configuration will include more realistic (smaller) gaps, including partial seals as typically configured in wind tunnel tests. The main purpose of this first case is to explore grid generation issues, including both quality metrics and mesh refinement. A sister workshop on Geometry and Mesh Generation will be held concurrently with HiLiftPW-3 using this same high-lift CRM configuration. We expect there to be a significant amount of synergy between the two workshops.

The second case will be a nacelle installation study on the JAXA Standard Model (JSM). This configuration is at Reynolds number Re=1.9 million based on mean aerodynamic chord, both with and without nacelle. See figure 1-6. Similar to the other high lift workshops, although the experiment used a semi-span model, the workshop will call for "free-air" fully-turbulent computations (with use of transition specification or transition models optional).

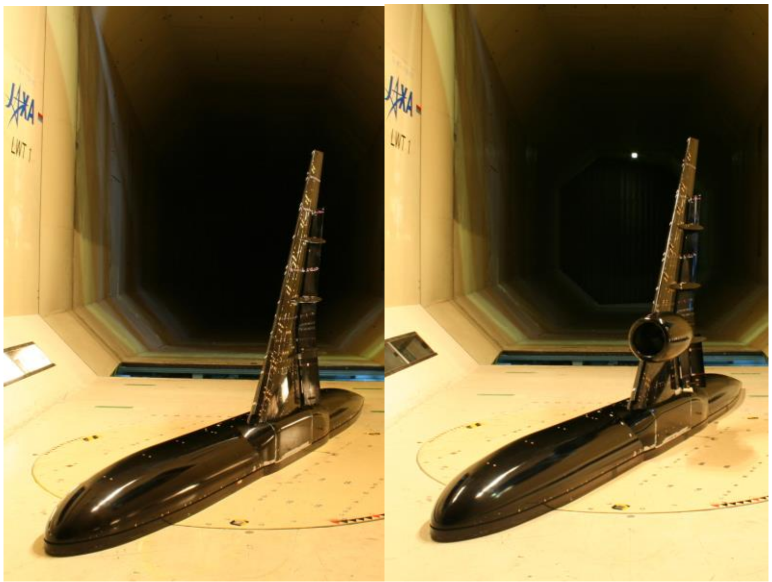

Figure 1-6: Photographs of the JSM semi-span configuration; no nacelle (left), with nacelle (right).

The HiLiftPW-3 will also include a turbulence model verification case. Under current consideration is a 2-D airfoil wake study [10,11]. The purpose of the verification study is to try to ascertain whether different codes with the same turbulence model yield consistent solutions as the grid is refined. (Adequate grid convergence is difficult to insure for three-dimensional cases.) This airfoil case is believed to be relevant because of the importance of accurate wake resolution and modeling for high-lift configurations.

The overall goals for HiLiftPW-3 remain the same as for the earlier workshops, in terms of assessing prediction capability, developing modeling guidelines, advancing understanding, identifying areas needing additional research and development, etc. Some additional focus is now being placed on the process of grid generation, from the Computer-Aided Design (CAD) stage through actual grid generation, by including (and tying in with) a separate new Geometry and Mesh Generation workshop. Also, because of the known difficulties near C_{Lmax} – including greater experimental and CFD uncertainties – some focus will be spent on flows in that regime.

At this time, a HiLiftPW-4 is envisioned for the summer of 2020. Although plans may change, the focus will most likely be on the high lift CRM configuration, which may have been built and tested by NASA by that date.

4.0 LESSONS LEARNED AND SUMMARY OF NEEDS FOR FUTURE HIGH LIFT EXPERIMENTS

From the first two high-lift prediction workshops held to date, the following represent some of the main lessons learned, in terms of conducting a high-quality CFD validation study for these types of configurations. For one thing, from the perspective of *verification*, having high-quality grids of varying density with consistent refinement (so that the grids used in a grid density study are of the same or similar family) is known to be important. For verification, one is not concerned with experiment, but only with consistency (or lack thereof) of CFD results. Presumably, different CFD codes with the same turbulence model should trend toward the same results as the grid is refined. Unfortunately, on today's computers, sufficiently fine grids are difficult to create/run for such complex three-dimensional configurations. A proper family (such as is achieved for structured grids in three dimensions by coarsening through the removal of every other grid point) is very difficult to create while retaining reasonable grid sizes. Typically the refinement/coarsening is somewhat ad hoc for complex configurations, and proper grid families are typically not achieved in these exercises. An excellent summary of grid issues related to the Drag Prediction Workshops (which also apply to the high lift workshops) can be found in Morrison et al. [12]. An alternative to consistent grid refinement is adaptive mesh refinement (AMR), where additional grid points are added automatically only to targeted areas that need them. See, for example, Lee-Rausch et al. [13]. This technology is still relatively immature, and most CFD codes do not possess the capability to perform AMR on a routine basis. For high-lift flows, subject matter experts believe that sufficient grid refinement is required in the wake regions of the slat and main elements, because these wakes pass over and often interact with the boundary layers of the downstream elements. For example, figure 1-5 showed typical velocity profiles over the flap for the DLR-F11 configuration. The flap boundary layer, main element wake, and slat wake clearly interact. There have been conjectures that this type of interaction is particularly influential near C_{Lmax}, but CFD validation studies have never been conclusive in this regard.

From the perspective of *validation*, comparing apples with apples is known to be important; i.e., the CFD configuration and boundary conditions must closely mimic those of the experiment. Unfortunately, to date this requirement is also rarely achieved in practice. In order to ease the process of grid generation, configurations are often simplified (brackets ignored, gaps ignored or widened, etc.). Also, high-lift configurations are usually tested as semi-span models (mounted on the floor or wall of the wind tunnel), but the CFD usually assumes a "free-air" configuration and compares against corrected data (i.e., corrected to attempt to remove the influence of the tunnel walls). One of the reasons for the use of "free-air" CFD is greater ease of acquiring a solution. Running CFD "in a tunnel" is often more difficult. A desired Mach number can be more tedious to attain because the back pressure boundary condition is influential and needs to be iterated. Also, the incoming tunnel boundary layers are likely very important in semi-span testing, so they cannot be ignored. CFD may not achieve accurate boundary layer profiles that match reality, without artificially extending the inflow region of the CFD grid. Furthermore, the wind tunnel inflow boundary conditions (including wall boundary layer characteristics) are often not measured, so one needs to guess many aspects of the inflow conditions of the tunnel. Some earlier work by Fabiano et al. [14] and Konig et al. [15] on the HiLiftPW cases suggest that wind tunnel effects should be modeled in the CFD, rather than attempting to correct the wind tunnel data. In spite of the difficulties, CFD "in-tunnel" simulations will no doubt play an increasingly important role in future validation studies.

In terms of the workshops themselves, one of the important lessons learned to date is the importance of collecting a sufficient amount of information from the participants. For example, from HiLiftPW-1 and

HiLiftPW-2, we learned the importance of achieving sufficient iterative convergence in the CFD runs. If a participant did not run their CFD solutions long enough, they may be reporting incorrect (non-converged) results that skew the workshop statistics for the wrong reason. The only way to discern whether the submitted results have been sufficiently converged is by collecting convergence data. However, the need to collect a lot of data from participants must always be balanced against the risk of making the workshop requirements too difficult/onerous for participants.

Another lesson learned has been the benefit of conducting a well-organized statistical analysis of the workshop results. This type of analysis has benefits from both verification as well as validation perspectives. For verification, statistical analysis helps to identify whether a given solution is statistically similar to the collective set of CFD results, or whether it is an outlier. Analysis of outliers (trying to determine why they are different from the collective) is important to pursue, as it could lead to insights. For validation, statistical analysis adds a level of quantifiable uncertainty to the CFD solutions, providing more information when comparing against experiment.

Both of the high lift workshops held to date have also included special sessions at subsequent AIAA conferences, encouraging participants to perform additional and more detailed analysis, and write complete technical papers. This practice has been very beneficial, since greater insight sometimes emerges only after the community has had time to digest and study the workshop results. After HiLiftPW-2, some authors were also invited to contribute to a special section in the *AIAA Journal of Aircraft*. Many of the known papers from HiLiftPW-1 and HiLiftPW-2 participants are listed in the references [13–69].

There are currently only limited publicly-available experimental data for high-lift configurations. And even among those data that exist, most are limited to forces and moment and surface pressure coefficients. More carefully measured, high quality experimental data is needed, including flow field surveys in boundary layers and wakes (e.g., velocities and Reynolds stresses) and a complete description of the flow within the tunnel. We have already described the inconsistency of comparing corrected semi-span experimental data with “free air” CFD. Either more full-span testing is needed (such as sting-mounted models), or else significantly more inflow boundary condition information in the wind tunnel needs to be collected. Even for full-span testing, tunnel inflow non-uniformity may play a role, and such data could be useful when attempting to conduct CFD comparisons.

Because the largest disagreements between CFD and experiment tend to occur near C_{Lmax}, a focus of future high-lift experiments should be near maximum lift conditions. Sensitivity and uncertainty studies should also be conducted near these conditions as well. For example, what is the influence of small variations in tunnel speed, model angle of attack or sideslip, and model position in the tunnel on C_{Lmax} and the angle at which it occurs? More knowledge of such experimental sensitivities would be useful when conducting CFD validation studies.

Transition to turbulence is also a problematic issue when attempting to compare CFD with wind tunnel tests. Unless one is employing a turbulence model that includes transition, typically a “fully turbulent” run is the only practical choice. (Laminar regions can be specified in a CFD run, but this can be tedious and requires fore-knowledge of the precise transition locations; it also ignores boundary layers that may be transitional, i.e., neither fully laminar nor fully turbulent.) Therefore, the most useful experiments for high lift configurations would typically be at higher Reynolds numbers (e.g., greater than 15–20 million based on mean aerodynamic chord). Such high Reynolds numbers are only possible in a limited number of wind tunnels worldwide. At more typical lower Reynolds numbers, artificial tripping in wind tunnel tests may be called for. In any case, documentation of the transition locations on the model is paramount.

5.0 CONCLUSIONS

The AIAA high lift prediction workshop series has been described. To date, two workshops have been conducted. A brief summary of these workshops was given. HiLiftPW-3 is currently in the planning stages, and is set for June 2017. Its goals and focus were described, including a new effort to explore the impact of geometry and mesh generation in depth, by the addition of a sister workshop to be held concurrently with HiLiftPW-3. Finally, some lessons learned and summary of needs for future high lift experiments were outlined. Currently, conducting an adequate verification and validation study for a (three-dimensional) high-lift configuration is very difficult. Computer limitations (as well as gridding technology itself) often prevents us from running on sufficiently fine grids (and/or grids with adequate resolution in the right places, such as in wakes) that guarantee numerical errors to be significantly smaller than modelling errors. In addition to this, CFD is often severely limited in its ability to adequately match the geometry and boundary conditions in a wind tunnel test. In fact, the wind tunnel boundary conditions are rarely measured to the level required by CFD. Comparing corrected data from semi-span tests with “free air” CFD is fraught with uncertainties. Drawing firm conclusions is difficult when geometry and boundary conditions are not precisely the same. Clearly, the high lift community is faced with many challenges as we attempt to advance the state of the art.

References:

[1] Vassberg, J. C., Tinoco, E. N., Mani, M., Brodersen, O. P., Eisfeld, B., Wahls, R. A., Morrison, J. H., Zickuhr, T., Laflin, K. R., Mavriplis, D. J., “Abridged Summary of the Third AIAA Computational Fluid Dynamics Drag Prediction Workshop,” *AIAA Journal of Aircraft,* Vol. 45, No. 3, May-June 2008, pp. 781–798.

[2] Rumsey, C. L., Slotnick, J. P., Long, M., Stuever, R. A., Wayman, T. R., “Summary of the First AIAA CFD High-Lift Prediction Workshop,” *AIAA Journal of Aircraft*, Vol. 48, No. 6, 2011, pp. 2068–2079.

[3] Rumsey, C. L., Slotnick, J. P., “Overview and Summary of the Second AIAA High-Lift Prediction Workshop,” *AIAA Journal of Aircraft*, Vol. 52, No. 4, 2015, pp. 1006–1025.

[4] Murayama, M., Yokokawa, Y., Yamamoto, K., Ueda, Y., “Evaluation of Computations and Transition Prediction Method for Aircraft High-Lift Configuration,” *AIAA Journal of Aircraft*, Vol. 46, No. 5, 2009, pp. 1487–1499.

[5] Lacy, D. S., Sclafani, A. J., “Development of the High Lift Common Research Model (HL-CRM): A Representative High Lift Configuration for Transonic Transports,” AIAA Paper 2016-0308, January 2016.

[6] Johnson, P. L., Jones, K. M., Madson, M. D., “Experimental Investigation of a Simplified 3D High Lift Configuration in Support of CFD Validation, “ AIAA Paper 2000-4217, August 2000.

[7] Menter, F. R., “Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications,” *AIAA Journal*, Vol. 32, No. 8, 1994, pp. 1598–1605.

[8] Spalart, P. R., Allmaras, S. R., “A One-Equation Turbulence Model for Aerodynamic Flows,” *Recherche Aerospatiale*, No. 1, 1994, pp. 5–21.

[9] Rudnik, R., Huber, K., Melber-Wilkending, S., "Test Case Description for the 2nd High Lift Prediction Workshop," AIAA Paper 2012-2924, June 2012.

[10] Nakayama, A., "Characteristics of the Flow Around Conventional and Supercritical Airfoils," *Journal of Fluid Mechanics*, Vol. 160, 1985, pp. 155–179.

[11] Rumsey, C. L., "The NASA Langley Research Center Turbulence Modeling Resource," http://turbmodels.larc.nasa.gov, cited 07/06/2016.

[12] Morrison, J. H., Kleb, B., Vassberg, J. C., "Observations on CFD Verification and Validation from the AIAA Drag Prediction Workshops," AIAA Paper 2014-0202, January 2014.

[13] Lee-Rausch, E. M., Rumsey, C. L., Park, M. A., "Grid-Adapted FUN3D Computations for the Second High-Lift Prediction Workshop," *AIAA Journal of Aircraft*, Vol. 52, No. 4, 2015, pp. 1098–1111.

[14] Fabiano, E., Fares, E., Nolting, S., "Unsteady Flow Simulation of High-Lift Stall Hysteresis using a Lattice-Boltzmann Approach," AIAA Paper 2012-2922, June 2012.

[15] Konig, B., Fares, E., Nolting, S., "Lattice-Boltzmann Flow Simulations for the HiLiftPW-2," AIAA Paper 2014-0911, January 2014.

[16] Long, M., Mavriplis, D., "NSU3D Results for the First AIAA High Lift Prediction Workshop," AIAA Paper 2011-863, January 2011.

[17] Steed, R. G. F., "High Lift CFD Simulations with an SST-Based Predictive Laminar to Turbulent Transition Model," AIAA Paper 2011-864, January 2011.

[18] Wiart, L., Meunier, M., "Computational Assessment of the HiLiftPW-1 Trap-Wing Model Using the elsA CFD Software," AIAA Paper 2011-865, January 2011.

[19] Sclafani, A. J., Slotnick, J. P., Vassberg, J. C., Pulliam, T. H., Lee, H. C., "OVERFLOW Analysis of the NASA Trap Wing Model from the First High Lift Prediction Workshop," AIAA Paper 2011-866, January 2011.

[20] Eliasson, P., Peng, S.-H., Hanifi, A., "Improving the Prediction for the NASA High-Lift Trap Wing Model," AIAA Paper 2011-867, January 2011.

[21] Reyes, D. A., Girimaji, S. S., Pandya, M. J., Abdol-Hamid, K. S., "Computations of High-Lift Wing-Body Configuration on Unstructured Grids using $k-\omega$ Models," AIAA Paper 2011-868, January 2011.

[22] Fares, E., Nolting, S., "Unsteady Flow Simulation of a High-Lift configuration using a Lattice Boltzmann Approach," AIAA Paper 2011-869, January 2011.

[23] Park, M. A., Lee-Rausch, E. M., Rumsey, C. L., "FUN3D and CFL3D Computations for the First High Lift Prediction Workshop," AIAA Paper 2011-936, January 2011.

[24] Murayama, M., Yamamoto, K., Tanaka, K., "CFD Comparison Study for Trapezoidal High-Lift Wing Configurations by Structured and Unstructured Mesh Method," AIAA Paper 2011-937, January 2011.

[25] Crippa, S., Melber-Wilkending, S., Rudnik, R., "DLR Contribution to the First High Lift Prediction Workshop," AIAA Paper 2011-938, January 2011.

[26] Zhong, B., Tang, K., Drikakis, D., Gou, S., "Numerical Simulations of Turbulent Flows around a High Lift Wing-Body Configuration," AIAA Paper 2011-1043, January 2011.

[27] Antunes, A. P., Azevedo, J. L. F., da Silva, R. G., "Numerical Simulations of Turbulent Flow over a High-Lift Configuration," AIAA Paper 2011-3006, June 2011.

[28] Pandya, M. J., Abdol-Hamid, K. S., Parlette, E. B., "CFD Computations for a Generic High-Lift Configuration Using TetrUSS," AIAA Paper 2011-3008, June 2011.

[29] Eliasson, P., Hanifi, A., Peng, S.-H., "Influence of Transition on High-Lift Prediction for the NASA Trap Wing Model," AIAA Paper 2011-3009, June 2011.

[30] Rumsey, C. L., Lee-Rausch, E. M., "NASA Trapezoidal Wing Computations Including Transition and Advanced Turbulence Modeling," AIAA Paper 2012-2843, June 2012.

[31] Murayama, M., Ito, Y., Tanaka, K., Yamamoto, K., "Computational Studies of the NASA High-Lift Trap Wing Using Structured and Unstructured Grid Solvers," AIAA Paper 2012-2844, June 2012.

[32] Crippa, S., Krimmelbein, N., "Transitional Flow Computations of the NASA Trapezoidal Wing with the DLR TAU Code," AIAA Paper 2012-2845, June 2012.

[33] Antunes, A. P., da Silva, R. G., Azevedo, J. L. F., "Influence of High-Lift Supporting Systems on the Trapezoidal Wing Aerodynamic Coefficients," AIAA Paper 2012-2846, June 2012.

[34] Cavallo, P. A., Feldman, G. M., "Quantification of Grid Refinement Effects for NASA High Lift Trap Wing Using Error Transport Model," AIAA Paper 2012-2847, June 2012.

[35] Moitra, A., Gujar, P., Gupta, P., "Investigation of Solution Methodology for Resolution of Side of Body Flow Separation," AIAA Paper 2012-2848, June 2012.

[36] Ravindra, K., Shende, N. V., Patel, S. K., Balakrishnan, N., "Unstructured Adaptive Mesh Calculations for NASA Trap Wing Using the Code HiFUN," AIAA Paper 2012-2849, June 2012.

[37] Wiart, L., Moens, F., "Numerical Simulation of the NASA High-Lift Trap Wing with the elsA CFD Software," AIAA Paper 2012-2918, June 2012.

[38] Sclafani, A. J., Slotnick, J. P., Vassberg, J. C., Pulliam, T. H., "Extended OVERFLOW Analysis of the NASA Trap Wing Wind Tunnel Model," AIAA Paper 2012-2919, June 2012.

[39] Shankara, P., Snyder, D., "Numerical Simulation of High Lift Trap Wing using STAR-CCM+," AIAA Paper 2012-2920, June 2012.

[40] Lopez, O. D., Ochoa, N., Mahecha, J. C., Leguizamon, S., Escobar, J. A., Ramirez, S., Jimenez, R. A., Giraldo, A. M., Silva, C., "Numerical Simulation of NASA Trap-Wing Model as a Colombian Contribution to the High-Lift Prediction Workshop," AIAA Paper 2012-2921, June 2012.

[41] Holman, D. M., Brionnaud, R. M., Martinez, F. J., Mier-Torrecilla, M., "Advanced Aerodynamic Analysis of the NASA High-Lift Trap Wing with a Moving Flap Configuration," AIAA Paper 2012-2923, June 2012.

[42] Mavriplis, D., Long, M., Lake, T., Langlois, M., "NSU3D Results for the Second AIAA High Lift Prediction Workshop," AIAA Paper 2014-0748, January 2014.

[43] Chitale, K. C., Rasquin, M., Martin, J., Jansen, K. E., "Finite Element Flow Simulations of the EUROLIFT DLR-F11 High Lift Configuration," AIAA Paper 2014-0749, January 2014.

[44] Eliasson, P., Peng, S.-H., "Results from the 2nd AIAA CFD High Lift Prediction Workshop using Edge," AIAA Paper 2014-0750, January 2014.

[45] Allaneau, Y., Goldberg, U., Chakravarthy, S., Peroomian, O., "Contribution from Metacomp Technologies, Inc. to the Second High Lift Prediction Workshop," AIAA Paper 2014-0912, January 2014.

[46] Deloze, T., Laurendeau, E., "NSMB contribution to the 2nd High Lift Prediction Workshop," AIAA Paper 2014-0913, January 2014.

[47] Hanke, J., Shankara, P., Snyder, D., "Numerical Simulation of DLR-F11 High Lift Configuration from HiLiftPW-2 using STAR-CCM+," AIAA Paper 2014-0914, January 2014.

[48] Rudnik, R., Melber-Wilkending, S., "DLR Contribution to the 2nd High Lift Prediction Workshop," AIAA Paper 2014-0915, January 2014.

[49] Cavallo, P. A., "CRUNCH CFD Calculations for HiLiftPW-2 with Discretization Error Predictions," AIAA Paper 2014-0916, January 2014.

[50] Hoffman, J., Jansson, J., Jansson, N., de Abreu, R. V., "Time-Resolved Adaptive FEM Simulation of the DLR-F11 Aircraft Model at High Reynolds Number," AIAA Paper 2014-0917, January 2014.

[51] Lee-Rausch, E. M., Rumsey, C. L., Park, M. A., "Grid-Adapted FUN3D Computations for the Second High Lift Prediction Workshop," AIAA Paper 2014-2395, June 2014.

[52] Murayama, M., Yamamoto, K., Ito, Y., Hirai, T., Tanaka, K., "Summary of JAXA Studies for the Second AIAA CFD High Lift Prediction Workshop Using Structured and Unstructured Mesh CFD," AIAA Paper 2014-2396, June 2014.

[53] Moitra, A., Gujar, P., Gupta, P., Rajvanshi, D., "Investigation of Grid Generation Strategies for Prediction of High Lift Flows," AIAA Paper 2014-2397, June 2014.

[54] Escobar, J. A., Suarez, C. A., Silva, C., Lopez, O. D., Velandia, J. S., Lara, C. A., "Detached Eddy Simulation of the DLR-F11 Wing/Body Configuration as a Contribution to the 2nd AIAA CFD High Lift Prediction Workshop," AIAA Paper 2014-2398, June 2014.

[55] Makarov, V. E., Fedorchenko, Y. P., Shorstov, V. A., Babulin, A. A., Bolshunov, K. J., "CIAM, Sukhoi NCT and Irkut Contribution to HiLiftPW-2," AIAA Paper 2014-2399, June 2014.

[56] Holman, D. M., Brionnaud, R. M., Modena, M. C., Sanchez, E. V., "Aerodynamic Analysis of the 2nd

High Lift Prediction Workshop by a Lattice-Boltzmann Method Solver," AIAA Paper 2014-2568, June 2014.

[57] Gopalakrishna, N, Balakrishnan, N., Patil, Y., Ravindra, K., Shende, N. V., "High lift Flow Computations Using the Code HiFUN," AIAA Paper 2014-2569, June 2014.

[58] Rasquin, M., Chitale, K., Ali, M., Jansen, K., "Parallel Adaptive Detached Eddy Simulations of the EUROLIFT DLR-F11 High Lift Configuration," AIAA Paper 2014-2570, June 2014.

[59] Chen, J., Zhou, N., Zhang, Y., Deng, Y., "CFD Investigations of the HiLiftPW-2 Configuration with MFlow Solver," AIAA Paper 2014-2571, June 2014.

[60] Duque, E. P. N., "Using XDB Workflows to Analyze the 2nd AIAA CFD High Lift Prediction Workshop Simulations," AIAA Paper 2014-2695, June 2014.

[61] Coder, J. G., "OVERFLOW Analysis of the DLR-F11 Configuration from HiLiftPW-2 Including Transition Modeling," AIAA Paper 2014-2696, June 2014.

[62] Pulliam, T. H., Sclafani. A. J., "High-Lift OVERFLOW Analysis of the DLR-F11 Wind Tunnel Model," AIAA Paper 2014-2697, June 2014.

[63] Coder, J. G., "OVERFLOW Analysis of the DLR-F11 High-Lift Configuration Including Transition Modeling," *AIAA Journal of Aircraft*, Vol. 52, No. 4, 2015, pp. 1082–1097.

[64] Chen, J., Zhang, Y., Zhou, N., Deng, Y., "Numerical Investigations of the High-Lift Configuration with MFlow Solver," *AIAA Journal of Aircraft*, Vol. 52, No. 4, 2015, pp. 1051–1062.

[65] Eliasson, P., Peng, S.-H., "Results from the Second AIAA CFD High-Lift Prediction Workshop Using Edge," *AIAA Journal of Aircraft*, Vol. 52, No. 4, 2015, pp. 1042–1050.

[66] Escobar, J. A., Suarez, C. A., Silva, C., Lopez, O. D., Velandia, J. S., Lara, C. A., "Detached-Eddy Simulation of a Wide-Body Commercial Aircraft in High-Lift Configuration," *AIAA Journal of Aircraft*, Vol. 52, No. 4, 2015, pp. 1112–1121.

[67] Holman, D. M., Brionnaud, R. M., Modena, M. C., Sanchez, E. V., "Lattice Boltzmann Method Contribution to the Second High-Lift Prediction Workshop," *AIAA Journal of Aircraft*, Vol. 52, No. 4, 2015, pp. 1122–1135.

[68] Mavriplis, D., Long, M., Lake, T., Langlois, M., "NSU3D Results for the Second AIAA High-Lift Prediction Workshop," *AIAA Journal of Aircraft*, Vol. 52, No. 4, 2015, pp. 1063–1081.

[69] Murayama, M., Yamamoto, K., Ito, Y., Hirai, T., Tanaka, K., "Japan Aerospace Exploration Agency Studies for the Second High-Lift Prediction Workshop," *AIAA Journal of Aircraft*, Vol. 52, No. 4, 2015, pp. 1026–1041.