Integrated CFD Validation Experiments for Prediction of Turbulent Separated Flows for Subsonic Transport Aircraft

Jeffrey P. Slotnick
Boeing Commercial Airplanes
Seattle, WA, 98012
UNITED STATES OF AMERICA

jeffrey.p.slotnick@boeing.com

ABSTRACT

Efforts are underway to develop and mature an integrated set of wind tunnel experiments targeted at obtaining high-quality data to improve computational fluid dynamics (CFD) modelling of turbulent separated flows for the accurate aerodynamic prediction of low-speed high-lift performance for transport airplane configurations and similar systems. Two specific configurations have been developed: a three-dimensional tapered bump and the High-Lift Common Research Model (CRM-HL). The geometric definitions of these configurations have been specifically designed to be open-source and publically available, and are driving internationally-coordinated test and simulation campaigns to advance flow physics modelling of turbulent separated flows. This paper will begin with an overview of the current state-of-the-art in CFD prediction of high-lift flows, particularly through the lens of results obtained from recent community workshops. Technical shortcomings in predictive capability will be highlighted, and motivation for new experiments will be discussed. Then, details of each of the two test configurations will be provided, including comparison of selected experimental and computational results from recent test campaigns. Finally, plans for future testing will be addressed.

1.0 INTRODUCTION

The design of safe and competitive commercial transport aircraft requires the acquisition and use of accurate aerodynamic data throughout the entire airplane development cycle. Obtaining accurate aerodynamic data is especially critical at the edges of the flight envelope, where complex flow physics and non-linear effects often size airplane structure and systems [1]. Aerodynamic performance at low-speed maximum lift is particularly important, and sets stall margin, characteristics, approach speed, and warning parameters [2]. In current practice, data to develop aerodynamic characteristics for low-speed performance during the aircraft design phase are primarily obtained in the wind tunnel, and corrected to flight-level conditions based on established empirical and experience-based procedures. Similarly, data to satisfy regulatory certification requirements are largely obtained from airplane flight testing. The rapid maturation of computational aerodynamic analysis tools and processes, such as those based on Reynolds-Averaged Navier-Stokes (RANS) CFD methods, is enabling the augmentation of these airplane aerodynamic databases with numerical data. Of particular difficulty is the numerical prediction of low-speed aerodynamic performance at high-lift conditions, when the airplane is in a take-off or landing configuration.

Airplane configurations in the take-off (TO) and landing phases of flight are characterized by wings with increased camber obtained from the deployment of leading edge (e.g. slat) and trailing edge (e.g. flap) aerodynamic devices. Chord-wise slots between these high-lift elements provide a mechanism to re-energize the boundary layer on downstream surfaces, allowing for operations at higher lift conditions. Inclusion of the pylon and engine cowl, landing gear, as well as required external slat and flap support brackets and fairings, with associated surface cut-outs, greatly increase the geometric complexity of the high-lift configuration. As a result, high-lift flow physics is dominated by a variety of interacting flow phenomena, such as confluent and merging wakes, discrete vortices, laminar and turbulent boundary layers, and unsteady separated flow

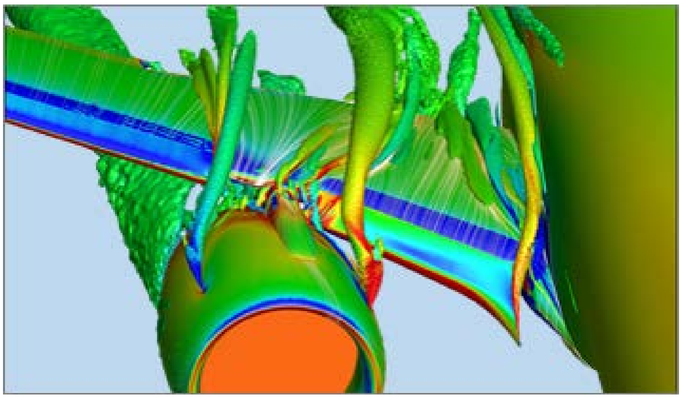

Figure 6-1. Representative flow features of a transport airplane in high lift configuration

[3], as depicted in Figure 6-1. Effective use of CFD to supplement low-speed performance databases is predicated on the ability of such methods to accurately capture these relevant flow features when modelling complete airplane configurations, which often include a wide range of geometric scales.

A major finding of the NASA-sponsored CFD Vision 2030 report [4] is that "…the use of CFD… is severely limited by the inability to accurately and reliably predict turbulent flows with significant regions of separation." Furthermore, "well-known limitations of RANS methods for separated flows have confined reliable use of CFD to a small region of the flight envelope or operating design space." [5] This is particularly true for separation-dominated high-lift flow-fields. Modelling the complex high-lift flow physics for full airplane configurations remains a challenge for CFD predictive capability today, particularly using current eddy-viscosity-based turbulence models in production RANS CFD flow solvers.

A large body of evidence, collected over the past decade as part of the High Lift Prediction Workshop (HiLiftPW) series [6-8] of open community forums, has proven this out. The primary goal of the workshops is to assess the numerical prediction capability of CFD technology for swept, medium/high-aspect-ratio wings in landing/TO (high-lift) configurations. For the workshops, participants first generate CFD results on a representative subsonic transport test configuration, and compare their results directly to experimental wind tunnel data collected on the test configuration. CFD results from participating aerospace organizations, including academia, industry, and government research laboratories, are then aggregated and used to highlight overall trends in CFD predictive capability and identify areas of further improvement. The majority of CFD simulations are RANS, but an increasing number of entrants are employing emerging solvers that employ more advanced eddy-resolving models. In the latest workshop (HiLiftPW-3) held in Denver, CO in 2017 [9], the JAXA Standard Model (JSM) configuration with nacelle/pylon (case2c), as depicted in Figure 6-2, was computational analysed. All workshop participant CFD results are compared with experimental data. A comparison plot for lift coefficient (CL) versus angle-of-attack (alpha) for this test case, at the selected flow condition, is shown in Figure 6-3. These results show that the lift prediction of CFD simulations generally agrees well with experimental data in the linear portion of the lift curve, but that CFD lift prediction deviates dramatically from test data approaching maximum lift ($C_{L,\,max}$) and at post-stall. Near the stall region, there is also evidence of multiple solutions, where CFD solvers can predict results along different lift "branches" [10]. Although there are exceptions, results from RANS solvers tend to predict pre-mature (early) stall due to inadequate modelling of separated flow behind high lift support brackets (e.g. slat), and incorrect prediction of the stall breakdown mechanism, which, for this configuration, is driven by flow separation on the inboard wing upper surface at the side-of-body. These failures are vividly illustrated by the wall-streamline patterns. Similar discrepancies in results for drag and moment coefficients are reported [8]. Subtler effects such boundary-layer transition, or of using a half-model against a wind-tunnel wall, still require investigations; another issue is the present inability to obtain grid convergence on full configurations. Outcomes from the first two workshops established similar trends. [6-7]

Figure 6-2. JAXA Standard Model (JSM) high-lift configuration with nacelle/pylon used for HiLiftPW-3

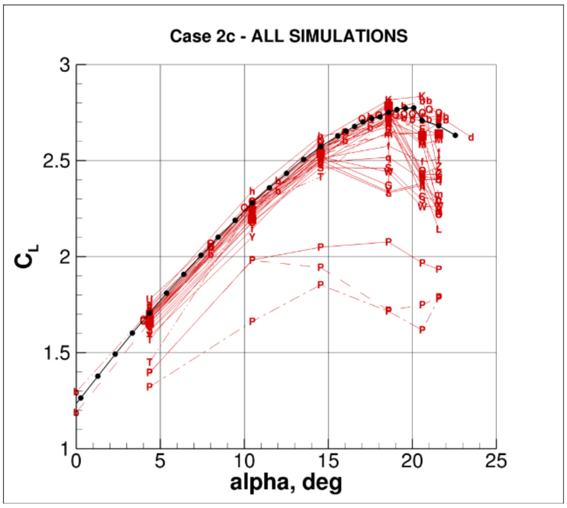

Figure 6-3. Comparison of lift coefficient results (CFD: red lines with symbols; experimental data: black solid circles) for JSM configuration with nacelle/pylon at Mach=0.2, Rec=3.6M

To significantly expand the use of CFD for low-speed airplane development at off-design conditions, as well as to augment and reduce airplane certification flight testing performed near the edges of the flight envelope using CFD analysis methods, improvement of CFD prediction for flows dominated by turbulent separated flow is critically needed. To this end, integrated experimental testing of key validation cases, ranging from simpler, canonical 3D configurations to complex high-lift geometries, will be required to systematically study complex high lift flow physics. Insights from data collected from well-designed experimental test campaigns will help inform the development and validation of next generation physical models (e.g. turbulence, transition, etc.) to improve CFD predictive capability, particularly when implemented in efficient flow solvers optimized for emerging High Performance Computing (HPC) platforms. For this purpose, two test geometries have been designed and developed to drive focused wind tunnel test campaigns to obtain experimental aerodynamic and flow physics data at nominal subsonic flow conditions to characterize turbulent flow separation associated with low-speed, high-lift airplane flight configurations. With this data, coordinated CFD validation activities within the aerospace community are expected to help accelerate CFD technology development, both of the RANS and the turbulence-resolving types.

We begin with a description of each experimental configuration, highlighting preliminary learnings from initial testing, including comparisons of test data with CFD results. Then, we discuss upcoming testing efforts planned using both configurations, including the potential utilization of future experimental data for community workshops to accelerate progress in advancing predictive CFD modelling capability.

2.0 TEST CONFIGURATIONS

2.1 Three-dimensional tapered bump

2.1.1 Motivation

Over the past several decades, several fundamental CFD flow physics experiments have provided a great deal of information to help assess the predictive capability of CFD. Several studies have considered two-dimensional bump and three-dimensional hill geometries [11-15], where the characteristics of flow separation at subsonic speeds were carefully obtained to generate experimental databases for CFD assessment and validation purposes. Although rich with critical flow information, these experimental databases are not sufficient for the purpose of decisively advancing the state-of-the-art in CFD turbulence modelling for several reasons:

- Two-dimensional geometries suffer from side-wall and streamline-divergence effects, which cannot be controlled to the fine level now expected in such studies. The axisymmetric geometries utilized by Driver [12] and Bachalo [13] at NASA Ames are notable exceptions to this problem.
- Some of the test configurations did cause separation, but that separation was strongly controlled by the geometry, so that the challenge was more on post-separation and reattachment physics, rather than on the separation location. Incipient separation, which typically affects maximum lift, was not created.
- A test configuration used in a comprehensive experimental campaign aimed at critical flow physics (e.g., incipient separation) would ideally have a control parameter, similar to an angle or attack, allowing the flow to be studied over a range of that parameter. Previous studies typically did not have this, although the Bachalo-Johnson flow had Mach number, and the recent Notre-Dame experiment has an adjustable ceiling [16].
- Shortcomings in measurement techniques utilized in many well-designed experiments conducted in the 1970s and 1980s limited the use of the experimental data to guide turbulence model design. Modern instrumentation offers better accuracy and capability of measuring closer to the wall, and is particularly critical when highly precise boundary-layer physics are the focus.
- Experimental datasets alone will not be sufficient to advance the accuracy of turbulence models. Turbulence modellers request terms (pressure-velocity, skin-friction, and dissipation in particular) which have never been measured in flows relevant to the current interest. To assist the systematic development of turbulence models, combined experimental-computational programs must be organized, where ideally the computations cover the entire spectrum from RANS to DNS.

As a result of these deficiencies, a new three-dimensional tapered bump (also known as the “Speed Bump”) configuration has been designed by Boeing, and is being tested to obtain a much richer set of experimental data that will be used, in conjunction with careful computational studies, to advance the state-of-the-art in CFD turbulence modelling for separated flows.

2.1.2 Geometry

The Speed Bump geometry is an analytically-defined, elongated bump that is tapered to minimize side-wall separation when installed and tested in the wind tunnel. The bump shape is given by the following formula:

$$z(x,y) = h\,\frac{1 + erf\left(\left(\frac{L}{2} - 2y_0 - |y|\right)/y_0\right)}{2}\,exp\left(-\left(\frac{x}{x_0}\right)^2\right)$$

where L is the square cross-section width, x is stream-wise, y lateral, and z wall-normal. The origin is at the center of the bump, so that the y interval is [-L/2, L/2]. Three dimensions define the bump shape, a height h in z, and two widths x_0 and y_0. The taper was designed into the configuration to keep the side-wall boundary layers attached. The other three parameters are: the ceiling height, the Reynolds number based on L, and the incoming boundary-layer thickness. The concept is for six numbers to fully define the problem, whether proposed for experiments or simulations. RANS CFD tools were used to design the baseline shape, which is given by $x_0 = 0.195L$, $y_0 = 0.06L$, and $h = 0.085L$. Contours of the bump in both the x and y planes are given in Figure 6-4.

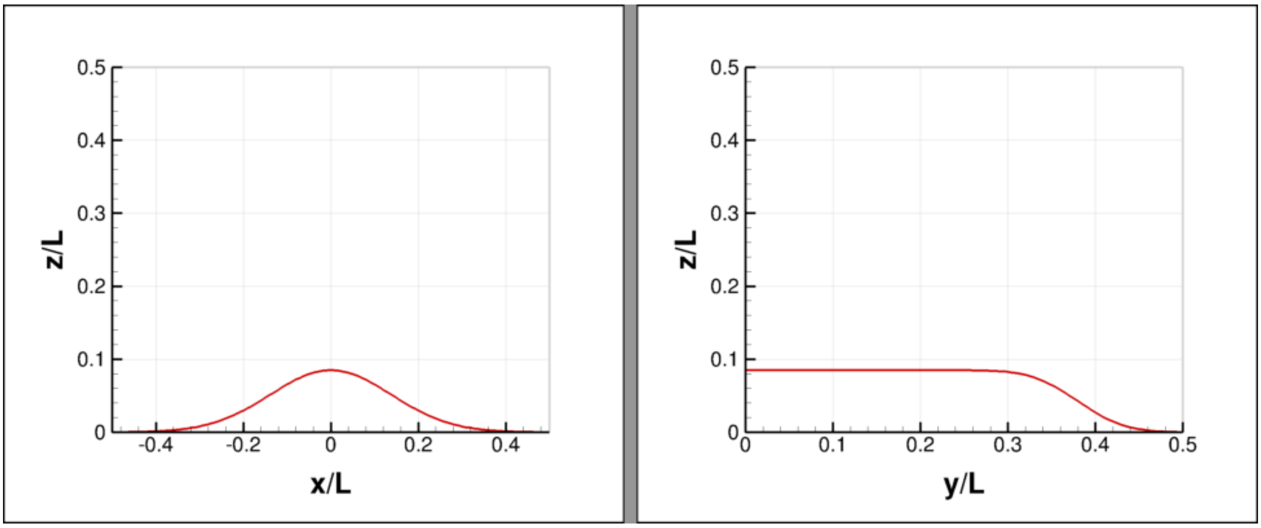

Figure 6-4. Contour views of baseline Speed Bump configuration – Left: side view; right: front view. Symmetry plane at y/L=0, side-wall at y/L=0.5, ceiling at z/L=0.5.

2.1.3 Initial Testing and Preliminary Results

Risk mitigation testing was recently performed at the Boeing North American Aerodynamic Research Tunnel (NAART) facility in Huntington Beach, CA and at the University of Washington (UW) in the 3x3 Foot Subsonic Wind Tunnel in Seattle, WA. This testing has confirmed the essential characteristics of the separated flow field aft of the bump. Figure 6-5 shows the test article in the UW facility. Limited data was collected at a range of speeds, ranging from 10 meters/second to 60 meters/second with a 36'' tunnel width. In the NAART, oilflow images were obtained. In the UW tunnel, oilflow images and a limited amount of surface pressures measurements were collected. Preliminary CFD simulations using both the Spalart-Allmaras turbulence model (SA) with curvature and rotation correction terms (RC) [17] and the Menter SST-RC model [18] from the NTS code [19] were obtained. A comparison of surface flow features including the oilflow images from the NAART and UW facilities, as well as surface streamlines from the CFD results, is shown in Figure 6-6. Results from the two experiments show two counter-rotating vortices, shed from the spanwise edges of the bump, which are roughly in the same position both streamwise relative to the top centreline of the Bump, as well as laterally between the tunnel side walls. CFD results using the two turbulence models indicate significant differences in flow development at this condition, with the SST-RC

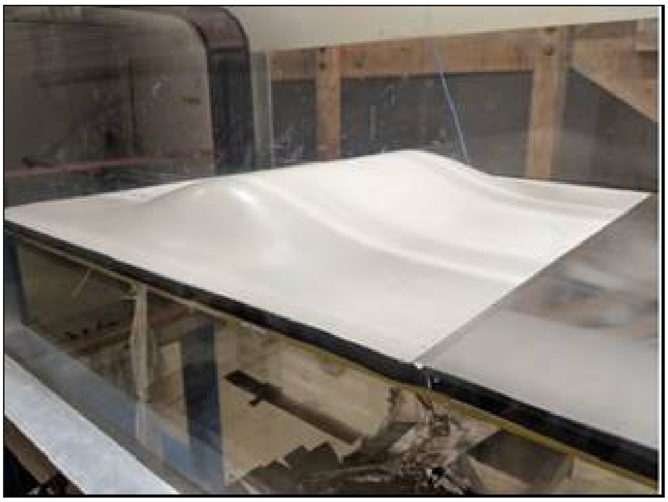

Figure 6-5. Speed Bump configuration in UW 3x3 Foot Wind Tunnel

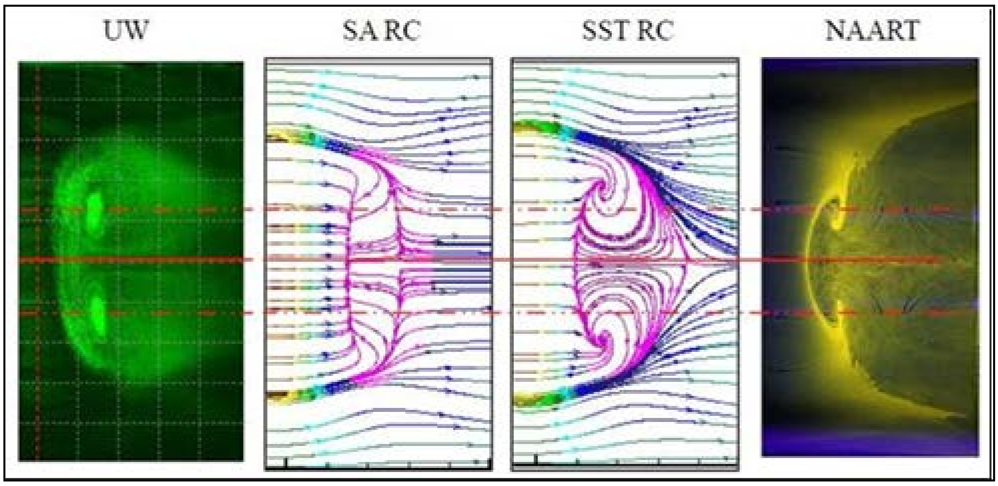

Figure 6-6. Comparison of surface flow features, V=60m/s flow condition.

model showing a vortex rollup and streamwise extent of the separation region more consistent with experimental results. These initial results confirm that the separated flow field generated by the Speed Bump is discriminating and exposes significant differences in the predictive capability of the RANS CFD simulations. A more complete summary of the UW experimental results, including comparisons with CFD simulations, will be available at the AIAA SciTech conference in January 2020.

2.1.4 Future Plans

Plans are maturing to perform additional experiments in multiple facilities to collect additional data from state-of-the-art, high-quality measurements, including velocity profiles, Reynolds stresses, oil flow, surface pressures, and skin friction. Stereoscopic Fluorescent Particle Image Velocimetry (PIV) and some limited profiles of three-velocity component Laser Doppler Velocimetry (LDV) will be used to get detailed, time-resolved, planar measurements of near-surface and off-body three-component velocity fields. Hot wire anemometry may also be employed. For oil flow, standard colored and/or fluorescent pigments will be used. The intent is to capture the full range of flow patterns at as many test conditions as possible. Skin friction may be indirectly inferred from the measured velocity profiles obtained from PIV, LDV, and/or hot wire anemometry by fitting the sub- and buffer layers profiles. A desirable option is Oil-Film Interferometry, especially since strong pressure gradients and separation defeat the usual buffer-layer profiles.

It is anticipated that results from systematic CFD studies will be compared to test data to assess the current state-of-the-art in predictive capability, and to identify key technology gaps and shortcomings, through community workshops. A key objective is to utilize high fidelity numerical simulations, such as Direct Numerical Simulation (DNS) or Large Eddy Simulation (LES), depending on the Reynolds number, to augment the experimental data in order to provide a complete set of information to use in improving turbulence models. Careful and methodical numerical simulations of the test configurations using a trusted CFD capability will be required to augment field data, such as pressure-strain and dissipation tensors, that are not measurable using current measurement techniques. Some early DNS simulations are underway on a two-dimensional slice of the Bump geometry by a number of researchers.

2.2 High Lift Common Research Model (CRM-HL)

2.2.1 Motivation

Over the past decade, three high-lift CFD prediction workshops have utilized configuration geometry and experimental results from pre-existing test campaigns. HiLiftPW-1 used the NASA Trapezoidal Wing configuration [6], HiLiftPW-2 used the DLR-F11 configuration [7], and HiLiftPW-3 used the JAXA Standard Model configuration [8]. In all cases, high-quality experimental data, usually in the form of integrated forces and moments and surface pressures, was utilized to benchmark the capability of CFD flow solvers to predict the aerodynamic characteristics of industrial class high-lift configurations. However, only a limited amount of data from these testing campaigns either existed or were made available for the community workshops. And, more importantly, because these datasets are pre-existing, little, if any, ability to augment the datasets with additional test data is possible. For example, a common focus within the workshop community is to better understand how the flow breaks down at stall, since integrated CFD results may show good correlation in lift, but miss in pitching moment. Having flow visualization information, both on- and off-body, would help in characterizing the actual physical mechanism associated with stall on these configurations. These additional types of measurements were not typically obtained since the testing was originally performed for other purposes.

As a result of these limitations in available experimental data, among other considerations, a comprehensive geometry and configuration "ecosystem" is being developed for the low-speed (high-lift) variant of the Common Research Model. The intent of the ecosystem is to design and build a series of CRM-HL configurations that can be tested in a range of wind tunnels, from small academic atmospheric tunnels to large pressurized and/or cryogenic facilities, to collect a rich set of both unique and complementary data to fully characterize the aerodynamic characteristics associated with a representative high-lift configuration. There are many expected benefits of the ecosystem:

- Allows for the assessment of the accuracy, efficiency, and robustness of current and future CFD tools and technologies on industry-consistent configuration(s).
- Provides a common, well-understood, fully-documented, and relevant dataset (geometry and test data) that enables direct assessment and comparison between CFD flow solvers and modelling approaches.
- Provides a common standard to assess the predictive capabilities of emerging computational tools, primarily through open community workshops, employing newly-acquired test data (some of which could facilitate "blind" comparisons).
- With proper controls, enables the design and fabrication of nearly identical models in multiple facilities – establishes data repeatability, a critical component of high-quality experimental datasets,

as well as provides opportunities to explore issues such as scale effects.

- Provides a challenging open-source configuration(s) to demonstrate advanced measurement and sensing techniques to collect new, critical data needed for CFD model development – energizes new and continuing efforts within academia, hardware vendors, and other collaborators.
- Provides a freely-sharable geometry, which enables new, and strengthens existing, partnerships to accelerate technology development.
- Provides a geometrically-relevant testing platform to jointly develop, assess, and share emerging aerodynamic technology (e.g. Active Flow Control, noise, etc.) with external partners (e.g. NASA, etc.)
- Drives development of enabling technologies which provide indirect benefits, like improved test facility capability/utilization and workforce development (e.g. industry/university collaboration).

2.2.2 Geometry

Based largely on the original high-speed Common Research Model [20], the CRM-HL geometry was designed and developed to be representative of a conventional transport high lift configuration [21]. As shown in Figure 6-7, the base CRM-HL geometry includes a wing, body, nacelle, pylon, and horizontal tail (not shown). The high lift configuration includes inboard and outboard leading edge slats and inboard and outboard single-slotted flaps. Recommended nominal TO and landing positions have been established for all devices. Additional devices, such as nacelle chines, have also been developed.

Figure 6-7. High Lift Common Research Model geometry

2.2.3 Initial Testing and Preliminary Results

Initial testing of a 10% scale semi-span CRM-HL model has been completed at the NASA Langley 14x22 foot wind tunnel. The main focus of the testing was on Active Flow Control (AFC), but some limited tested of the conventional high-lift configuration at the nominal landing positioning (30° slat deflection and 37° flap deflection) was also accomplished [22-24]. Figure 6-8 depicts the model in the tunnel. All testing was performed at Mach=0.2. Some limited CFD simulations were obtained [25, 26], leading to planned testing for aero-acoustics research objectives in 2020 [27, 28]. Experimental testing of the conventional high-lift system configuration confirmed the as-designed behaviour of the flow breakdown mechanism at stall, namely inboard upper surface flow separation aft of the nacelle/pylon installation, rather than flow separation that initiates at the outboard wing. Figure 6-9 shows the lift curve obtained in the experiment for the configuration with and without the nacelle/pylon, as well as the configuration with a nacelle chine, which improved the flow characteristics near maximum lift. Surface flow features captured with fluorescent mini-tufts [29] confirms the flow breakdown aft of the nacelle/pylon at a pre-stall angle-of-attack (see Fig. 6-10).

2.2.4 Future Plans

Additional testing of the NASA semi-span model will occur in the QinetiQ 5-meter (Q5m) wind tunnel in October 2019. The primary objectives of this testing will be to (a) confirm the essential flow features of the conventional high-lift system, particularly at stall, for the nominal landing configuration tested in the NASA

Figure 6-8. Semi-span CRM-HL model installed in the NASA LaRC 14x22 Foot Wind Tunnel

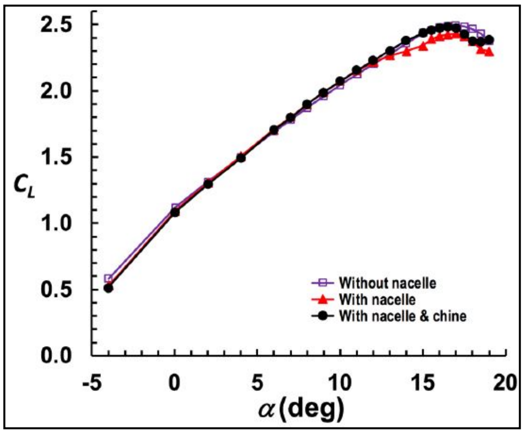

Figure 6-9. Experimental lift curve data from NASA test

tunnel, and to (b) explore high-lift element positioning beyond the original settings to establish a nominal reference landing configuration which will then become the baseline configuration for all future testing within the CRM-HL ecosystem. For the reference landing configuration, the desire is to optimize the performance at maximum landing flaps, ensuring representative flow physics from low angle-of-attack through stall, while considering positioning variations for a wide range of Reynolds numbers expected to be obtained in various testing facilities. The reference landing configuration definition is expected to be available to the user community in January 2020.

Parallel to the experimental testing activities currently underway, development of the CRM-HL ecosystem continues. A spectrum of CRM-HL models is envisioned for testing in various facilities. The plan is for each testing campaign to generate key flow information, taking advantage of unique testing capabilities at selected facilities (e.g. high Reynolds number testing in cryogenic facilities, advanced measurement techniques, etc.), in order to build a comprehensive database of CFD validation data that will be broadly available to the user community. Guidance on what experimental data will be acquired from the testing campaigns will be driven from a set of CFD requirements that is being compiled from discussions with key stakeholders. Specifically, the approach is to identify key knowledge gaps which currently exist in four key areas: CRM-HL configuration (e.g. position sensitivity, aero-elastic deformation, etc.), high-lift flow physics (e.g. stall

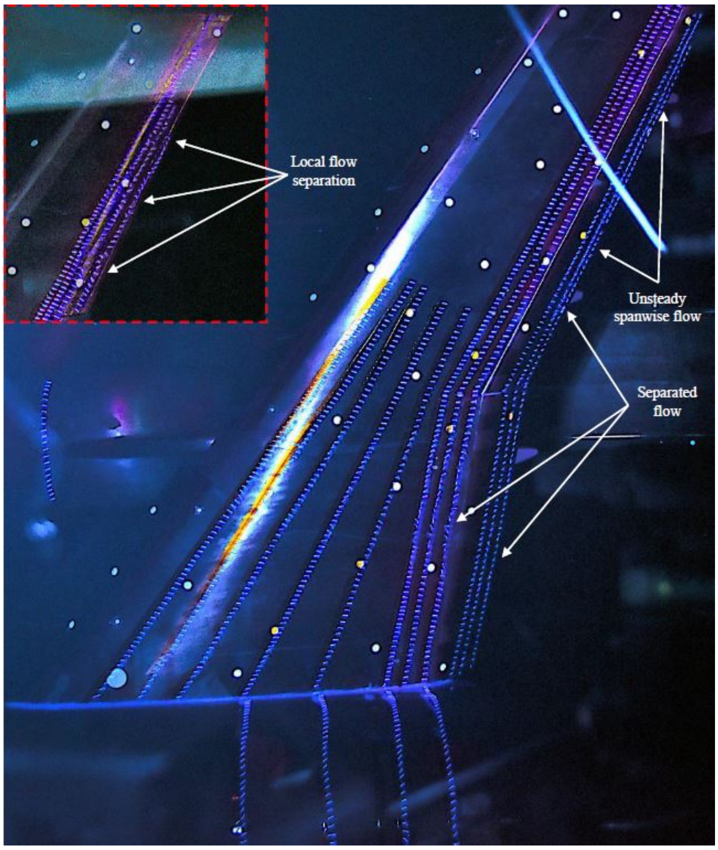

Figure 6-10. Experimental mini-tuft data showing flow separation behind the nacelle/pylon installation – Mach =0.2, a=16°

mechanism, flow transition, effect of hysteresis, flow behind brackets, etc.), tunnel/installation effects (e.g. mounting, state of inflow velocity field, etc.), and uncertainty quantification (e.g. repeatability, data corrections, time dependency for forces, moments, pressures, etc.). Each of these gaps will inform collection of specific experimental data at specific facilities, which will then drive systematic CFD validation studies.

Many organizations have expressed strong interest in participating in the design, development, fabrication, and/or testing of CRM-HL models. Boeing, QinetiQ, and Innovate UK (through the Aerospace Technology Institute) will be designing, fabricating, and testing a full-span CRM-HL model optimized for the pressurized QinetiQ 5-meter facility in 2021. NASA has committed to doing initial design work on the development of three separate cryogenic models that would eventually be tested in the National Transonic Facility (NTF). ONERA has recently confirmed their plans to build a full-span model for the F1 subsonic, pressurized facility in Le Fauga-Mauzac. In all cases, efforts will be coordinated to ensure that the set of CRM-HL models will be complementary and consistent to the maximum extent possible, with most, if not all, experimental data collected eventually becoming part of the public domain.

3.0 SUMMARY

Numerical prediction of high-lift aerodynamics is critical in the design and development of efficient and safe commercial transports. To this end, two specific CFD validation efforts are underway, which when considered together as an integrated activity, will provide a rich and extensive set of experimental data to be used to improve the predictive capability of CFD tools for high-lift flows. Testing of the Boeing Speed Bump and CRM-HL configurations promises to provide a wealth of information, ranging from fundamental flow physics data in turbulent boundary layers to the integrated effects of high-lift flow physics on realistic transport geometries. These activities are being well coordinated to facilitate key technical collaborations within the aerospace community, including industry, academia, and government laboratories.

ACKNOWLEDGMENTS

The author would like to thank the following people for their help with this manuscript: A. Clark, D. Lacy, and P. Spalart at the Boeing Company; O. Williams, A. Ferrante, M. Samuell, and S. Sarwas at the University of Washington; R. Wahls, J. Lin and M. Rivers at NASA Langley Research Centre; I. Smith, D. Seamarks and S. Rowley at QinetiQ; and S. Gates at the Aerospace Technology Institute.

REFERENCES

[1] Slotnick, J. P., and Heller, G., “Emerging Opportunities for Predictive CFD for Off-Design Commercial Airplane Flight Characteristics,” 54th 3AF Conference, Paris, France, 2019.

[2] Ross, H., “Flight Test of Transport Category Airplanes”, University of Washington MAE Colloquium, March 2014, https://www.aa.washington.edu/students/academics/mae/colloquium/2014winter

[3] Rumsey, C. L. and Ying, S. X., “Prediction of High Lift: Review of Present CFD Capability,” Progress in Aerospace Sciences, Vol. 38, 2002, pp. 145–180.

[4] Slotnick, J. P., et al., “CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences”, NASA/CR–2014-218178, 2014. https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20140003093.pdf

[5] Tinoco, E. N., et al., “Progress toward CFD for Full Flight Envelope”, the Aeronautical Journal, Vol. 109, pp. 451-460, October 2005.

[6] Rumsey, C.L., et al., “Summary of the First AIAA CFD High Lift Prediction Workshop”, Journal of Aircraft, Vol. 48, No. 6 (2011), pp. 2068-2079.

[7] Rumsey, C.L., et al., "Overview and Summary of the Second AIAA High Lift Prediction Workshop", AIAA 2014-0747, January 2014.

[8] Rumsey, C. L., et al., "Overview and Summary of the Third AIAA High Lift Prediction Workshop", https://doi.org/10.2514/1.C034940

[9] https://hiliftpw.larc.nasa.gov

[10] Kamenetskiy, D. S., et al., “Numerical Evidence of Multiple Solutions for the Reynolds-Averaged

Navier–Stokes Equations”, AIAA Journal Vol. 52, No. 8, August 2014.

[11] Greenblatt, D., Paschal, K., Schaeffler, N., Washburn, A., and Yao, C., “Separation Control over a Wall-Mounted Hump: A CFD Validation Test Case,” AIAA Paper 2004-2219, June 2004.

[12] Driver, D. M., “Reynolds Shear Stress Measurements in a Separated Boundary Layer Flow,” AIAA-1991-1787, 22nd Fluid Dynamics, Plasma Dynamics and Lasers Conference, Honolulu, HI, June 1991.

[13] Bachalo, W. D. and Johnson, D. A., “Transonic Turbulent Boundary-Layer Separation Generated on an Axisymmetric Flow Model,” AIAA-1979-1479, AIAA 12th Fluid and Plasma Dynamics Conference, Williamsburg, VA, July 1979.

[14] Bell, J. H., Heineck, J. T., Zilliac, G., Mehta, R. D., and Long, K. R., “Surface and Flow Field Measurements on the FAITH Hill Model,” AIAA-2012-0704, 50th AIAA Aerospace Sciences Meeting, Nashville, TN, January 2012.

[15] Byun, G., Simpson, R. L., and Long, C. H., “Study of Vortical Separation from Three-Dimensional Symmetric Bumps,” AIAA Journal, Vol. 42, No. 4, April 2004.

[16] Simmons, D. J., et al., “A Smooth Body, Large-Scale Flow Separation Experiment,” AIAA-2018-0572 Aerospace Sciences Meeting, Kissimmee, Florida, 2018 https://doi.org/10.2514/6.2018-0572

[17] Shur, M. L., Strelets, M. K., Travin, A. K., Spalart, P. R., "Turbulence Modeling in Rotating and Curved Channels: Assessing the Spalart-Shur Correction," AIAA Journal Vol. 38, No. 5, 2000, pp. 784-792.

[18] Smirnov, P. E., Menter, F. R., "Sensitization of the SST Turbulence Model to Rotation and Curvature by Applying the Spalart-Shur Correction Term," ASME Journal of Turbomachinery, Vol. 131, October 2009, 041010.

[19] Kravchenko, S. V., et al., “Fifteen Years of Boeing–Russia Collaboration in CFD and Turbulence Modeling/Simulation”, Computational Fluid Dynamics 2010 – Proceedings of the Sixth International Conference on Computational Fluid Dynamics, ICCFD6, St Petersburg, Russia, on July 12-16, 2010, pp. 155-162.

[20] Vassberg, J.C., et al., “Development of a Common Research Model for Applied CFD Validation Studies,” AIAA Paper 2008-6919, AIAA Applied Aerodynamics Conference, Honolulu, HI, August 2008. https://doi.org/10.2514/6.2008-6919

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

[22] Lin, J. C., et al., “High Lift Common Research Model for Wind Tunnel Testing: An Active Flow Control Perspective”, AIAA-2017-0319, January 2017.

[23] Lin, J. C., et al., “Wind Tunnel Testing of Active Flow Control on the High Lift Common Research Model”, AIAA 2019-3723, June 2019.

[24] Melton, L. P., et al., “Sweeping Jet Flow Control on the Simplified High-Lift Version of the Common Research Model”, AIAA 2019-3726, June 2019.

[25] Rivers, M. S., et al., “Computational Fluid Dynamics Analyses for the High-Lift Common Research

Model Using the USM3D and FUN3D Flow Solvers", AIAA 2017-0320, January 2017.

[26] Alauzet, F., et al., "3D RANS anisotropic mesh adaptation on the high-lift version of NASA's Common Research Model (HL-CRM)", AIAA 2019-2947, June 2019.

[27] Lockard, D. P., et al., "Noise Simulations of the High-Lift Common Research Model", AIAA 2017-3362, June 2017.

[28] Lockard, D. P., et al., "Assessment of Aeroacoustic Simulations of the High-Lift Common Research Model", AIAA 2019-2460, May 2019.

[29] Koklu, M., et al., "Surface Flow Visualization of the High Lift Common Research Model", AIAA 2019-3727, June 2019.