Goals and Status of the NASA Juncture Flow Experiment

Christopher L. Rumsey and Joseph H. Morrison
NASA Langley Research Center
Hampton, VA 23681
USA

c.l.rumsey@nasa.gov

ABSTRACT

The NASA Juncture Flow experiment is a new effort whose focus is attaining validation data in the juncture region of a wing-body configuration. The experiment is designed specifically for the purpose of CFD validation. Current turbulence models routinely employed by Reynolds-averaged Navier-Stokes CFD are inconsistent in their prediction of corner flow separation in aircraft juncture regions, so experimental data in the near-wall region of such a configuration will be useful both for assessment as well as for turbulence model improvement. This paper summarizes the Juncture Flow effort to date, including preliminary risk-reduction experiments already conducted and planned future experiments. The requirements and challenges associated with conducting a quality validation test are discussed.

1.0 INTRODUCTION

Most turbulence models in wide use in computational fluid dynamics (CFD) are incapable of accurately predicting the flow physics that occur in juncture flow regions (i.e., flow along the intersection of two walls). For example, CFD computations at past Drag Prediction Workshops (e.g., Vassberg et al. [1]) have produced very large variations in the predictions of separation, skin friction, and pressure near the side-of-body (SOB) wing-fuselage juncture, close to the wing trailing edge. While some linear eddy-viscosity turbulence models have shown excessively large regions of side-of-body separation, others have shown small regions of separation that are closer to what is seen in wind tunnel experiments. However, grid and numerical dissipation characteristics are often confounded with model effects. For example, use of a thin-layer approximation for viscous terms tends to yield smaller corner separation than full viscous terms for Reynolds-averaged Navier-Stokes. The widely-used Spalart-Allmaras linear eddy-viscosity model [2] predicts large regions of separation; however, replacing the linear eddy-viscosity relationship with a quadratic constitutive relationship [3] produces small side-of-body separation. Unfortunately, very few experimental details are available for such flows to distinguish submodel effects, in part because of the difficulties inherent in measuring data in the flowfield very close to walls. Some recent efforts have been aimed at improving knowledge of these flow physics [4], but much more remains to be done. Because multiple juncture flows are present on practically all civilian and military air vehicles, there is strong motivation to improve CFD's capabilities to predict them.

Oberkampf and Smith [5] identified four general categories of experiments based on the goal of each experiment: (1) physical discovery experiments, conducted primarily to improve the fundamental understanding of a physical process; (2) model calibration experiments, conducted to improve or determine parameters in mathematical models of moderately well represented physical processes; (3) acceptance or qualification experiments, which determine the reliability, performance, or safety of components, subsystems, or complete systems; and (4) validation experiments.

A validation experiment is conducted for the primary goal of determining the predictive capability of a mathematical model in a physical process. In this context, a computational analyst is the primary customer of a validation experiment. Characterization of the experiment – measuring all of the important characteristics of the experiment that are needed for input to the computational simulation of the experiment – is one of the most important goals of the experiment.

NASA has initiated an effort to perform a validation experiment for a wing-body juncture flow at subsonic conditions. As mentioned above, existing validation data for flow separation along juncture flows are very limited. Building on lessons from over a decade of experience in juncture flows for the Drag Prediction Workshops (DPW), NASA began a collaboration with the DPW Organizing Committee, industry, other government agencies, and academia to develop a new validation experiment that attempted to encompass fully attached, incipient separation, and separated states on a three-dimensional wing-body juncture case using angle of attack to generate these conditions. The design for this experiment was originally undertaken using CFD, but this proved more difficult than originally envisioned, further demonstrating the general lack of knowledge of these flows and ability to predict them. A series of risk-reduction experiments was subsequently undertaken to supplement the CFD predictions, to help guide the geometry selection for the final validation experiment.

The goal of the juncture flow validation experiment is to provide a difficult test to distinguish turbulence model predictions of the onset and progression of turbulent separated flow. The original intent was to design an experiment that had four distinct flow regimes – fully attached flow, incipient separation, a small separation region, and a large separation region – on a fixed geometry by varying a single flow parameter. Most existing experimental data sets have one of these four flow conditions and a turbulence model could be calibrated to match a single condition, but matching across the range of flow conditions would be a significantly more difficult challenge. Additionally, the juncture flow model was planned to have two different fuselage lengths to provide a second parametric variation by changing the incoming boundary layer characteristics.

The juncture flow experiment is designed to measure turbulence quantities that can be used to distinguish between turbulence model and submodel predictions. Detailed measurements in the separated region, and along the full configuration, will be required to distinguish predictions. Additionally, all important characteristics of the experiment that would be used as input to the computational simulation have to be measured, including the mounting system, the inflow conditions at the inlet to the wind tunnel test section, outflow conditions at the outlet of the wind tunnel test section, pressures along the wind tunnel walls in the test section, model deflections, accelerations, etc. The model design has to accommodate an instrumentation approach that can provide detailed mean and turbulent data very near the wall along the fuselage, at the wing leading edge, along the wing, and in the wing-fuselage juncture corner. Instrumentation also has to be included for steady and unsteady pressures, transition location, and mean and turbulence quantities along the wing. The configuration makes use of a low aspect ratio wing to provide a model large enough for the measurements. A general view of the juncture flow model is provided in figure 1-1.

The final validation experiment will make use of an internally mounted laser-Doppler velocimetry (LDV) system installed on a 3-D traverse system inside of the model fuselage; the laser beams will pass through windows on the fuselage wall, and thus will be able to measure flowfield details in the near-corner region (see figure 1-2). Upstream and downstream data, both in the wind tunnel test section and on the configuration fuselage, will also be taken in order to provide unambiguous boundary conditions for CFD. Other measurement techniques – including particle image velocimetry (PIV), steady and unsteady surface pressures, surface shear stress, and oil flows – are also planned. Furthermore, in order to improve characterization of the wind tunnel boundary conditions, efforts are underway to measure those aspects most relevant to CFD. This paper first provides a brief history of the project. It then summarizes the risk-reduction experiments that were conducted during 2015. Details of future experiments are provided next, including (1) plans for preliminary experimental measurements that attempt to characterize the inflow tunnel boundary conditions for CFD, and (2) the final experiment, which will use a model that is 8% of nominal full scale. Finally, the paper concludes with a summary and list of challenges.

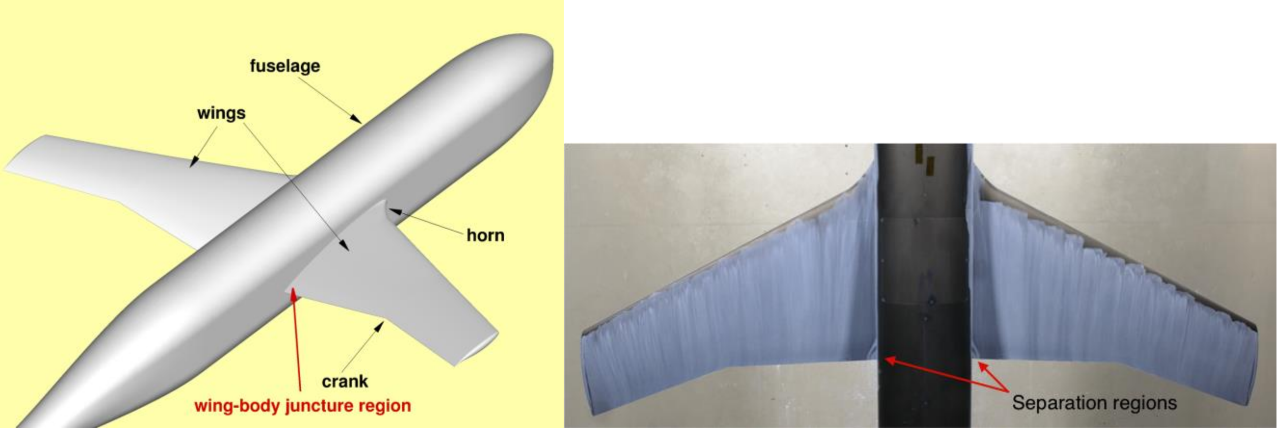

Figure 1-1: General view of the juncture flow model; left: overall shape, with some terminology highlighted; right: photograph of model upper surface showing oil flow separation in the wing-body juncture region.

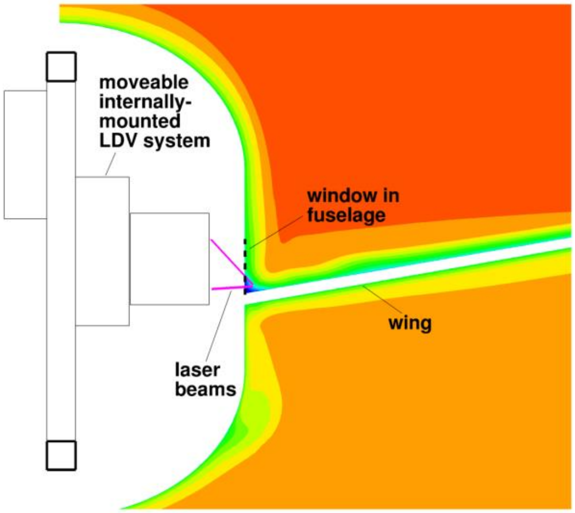

Figure 1-2: Schematic of internal LDV system measuring in the wing-body juncture region. Colors are iso-surfaces of velocity showing the boundary layers on the fuselage and wing and the separation region (shown as dark blue).

2.0 GENERAL SUMMARY OF THE PROJECT TO DATE

NASA’s Juncture Flow (JF) Experiment was conceived over the course of several years, primarily motivated by the wide variability in CFD results near the wing-body trailing-edge junction for most of the Drag Prediction Workshops, and the lack of detailed flow measurements in such regions. By the summer of 2014, the JF project had garnered NASA programmatic interest and support. An informal team was formed, including leadership and primary members from NASA Langley and NASA Ames, as well as additional representation from Boeing, Penn State, and consultants.

Importantly, this effort was conceived from the beginning to be a joint CFD/experimental enterprise. Because the primary customer of the experiment is the computational analyst, the leader was chosen to be someone with a CFD and turbulence background. The team was made up of both experimentalists and CFD practitioners, and was relatively fluid in its makeup, with several core members and many others participating only as their time permitted or as their particular expertise was required. One consultant was retained for his expertise in experimental uncertainty (as related to CFD verification and validation). Another consultant provided expertise in the LDV hardware to be used in the final validation experiment.

From 2014 through spring 2015, the team scoured existing literature and attempted to use CFD to determine an appropriate configuration for the experiment. The original goal was a symmetric (top-bottom and left-right) wing-body configuration that, in transonic flow conditions, would yield fully attached flow through large SOB separation, as a function of the freestream Mach number. This design would be non-lifting, which would minimize model deflection and wind tunnel wall effects on the results. The CFD was unsuccessful at finding such a configuration, and gradually many of the original requirements had to be relaxed. For example, freestream Mach number was replaced by angle-of-attack as the variable flow parameter for achieving different separation bubble sizes. Furthermore, size requirements (at the time) for the internal LDV traverse rig required a larger fuselage than would be practical for most available transonic tunnels, so the requirement for transonic flow was relaxed in favor of subsonic flow.

By spring 2015, the JF team decided that CFD alone could not be relied upon to determine the configuration. The team was heavily influenced by a lesson learned from an experiment at ONERA, for which a configuration that was designed to yield separated flow (based on CFD and theoretical considerations) ended up to be fully attached [6]. The JF team launched several risk-reduction experiments in order to help determine the best candidate configuration for the final experiment and confirm design decisions that were made based on CFD. These risk-reduction experiments are described next.

3.0 RISK-REDUCTION EXPERIMENTS

Three different risk-reduction experiments were conducted for the purpose of down-selecting the wing configuration. A brief summary is given here, but many additional details can be found in Rumsey et al. [7]. Table 1-1 provides flow and model characteristics for the risk-reduction experiments. The test named “Pre 1” was conducted as a semispan test at the NASA Ames Test Cell 2 (TC2) tunnel (with model size 3% of nominal full scale). While this test could be quickly achieved at a low cost, it was considered the riskiest of the tests because the influence of the tunnel’s wall boundary layer on the wing-body juncture region was felt to be potentially significant. The “Pre 2” test was on a 2.5%-scale full-span model in the Virginia Tech 6ft by 6ft Stability Wind Tunnel at relatively low Reynolds numbers [8]. This test would provide a full-span result in the event that the model for the Pre 3 test could not be completed. The “Pre 3” test was on a 6%-scale full-span model in the NASA Langley 14ft by 22ft Wind Tunnel at the same Reynolds number conditions envisioned for the final 8%-scale test and was expected to provide flow conditions closest to the final model conditions.

Table 1-1: Summary of JF Risk-Reduction Experiments

Test	Tunnel	Date	Model Size	Re_c (x 10^6)	M	Approx fuselage length (in)	Crank chord, c (in)	Wing span (in)
Pre 1	TC2	Summer 2015	3%	0.6-0.7	0.14	73	8.2	24.6 (semi-span)
Pre 2	Virginia Tech	August 2015	2.5%	0.6-0.7	0.18	45	6.9	40.9
Pre 3	14x22	December 2015	6%	2.3-2.7	0.27	145	16.5	98.3

Five different wing configurations designed with CFD were chosen for testing. Because the predictive capability of the CFD for turbulent separated flow was considered to be low, these wings were selected/designed in an attempt to span a very large range of possible separation region sizes. The wings were:

1) F6: a truncated version of the DLR-F6 wing [9, 10];

2) 0015-based: an NACA 0015-based wing, with NACA 0015 root shape, blended to an NACA 0012 at the wing crank and 0010 at the wing tip;

3) 0015mod-based: a similar wing with modified NACA 0015 at the root;

4) F6-S12: based on the F6 wing, but symmetrized and modified to increase its maximum thickness and trailing edge included angle near the wing root, then blended to symmetric airfoil shapes derived from the original NASA Common Research Model [11] design; and

5) COCA: a wing that was designed with CDISC [12], with the constraint of achieving a specified skin friction distribution at two stations near the root.

All wings except the F6 were top-bottom symmetric. Additional details concerning the wing shapes can be found in Rumsey et al. [7].

The semi-span test produced results that were significantly different from the full-span tests in terms of SOB separation size [13]. The causes for this difference are not definitively known, but the JF team believed that the mounting of the half-fuselage on the wind tunnel wall created an additional juncture flow that influenced the wing-body juncture flow physics (see figure 1-3). The two full-span tests ("Pre 2" and "Pre 3") produced qualitatively similar results for all wing configurations. Some of the results from the "Pre 2" test are shown in Rumsey et al. [7] and Kuester et al. [8].

A photograph showing the 6%-scale model from “Pre 3” in the 14x22 wind tunnel is given in figure 1-4. In both the Virginia Tech and 14x22 tunnels, the Reynolds number was low enough that tripping the boundary layer (with trip dots) was required to force the boundary layer flow to transition near the wing leading edges (and in some cases to eliminate laminar separation bubbles near the wing leading edges). Trip dots were also employed near the nose of the fuselage. Infrared (IR) thermography proved to be extremely useful for visualizing the transition on the model. Tests were done in some cases both with and without a “horn” (or fillet) near the leading edge of the wing at its intersection with the fuselage. The horn was designed to reduce the impact of the horseshoe vortex formed around the wing-body juncture as modern aircraft have a minimal horseshoe vortex. Generally, including a horn yielded somewhat larger SOB separation at the wing trailing edge.

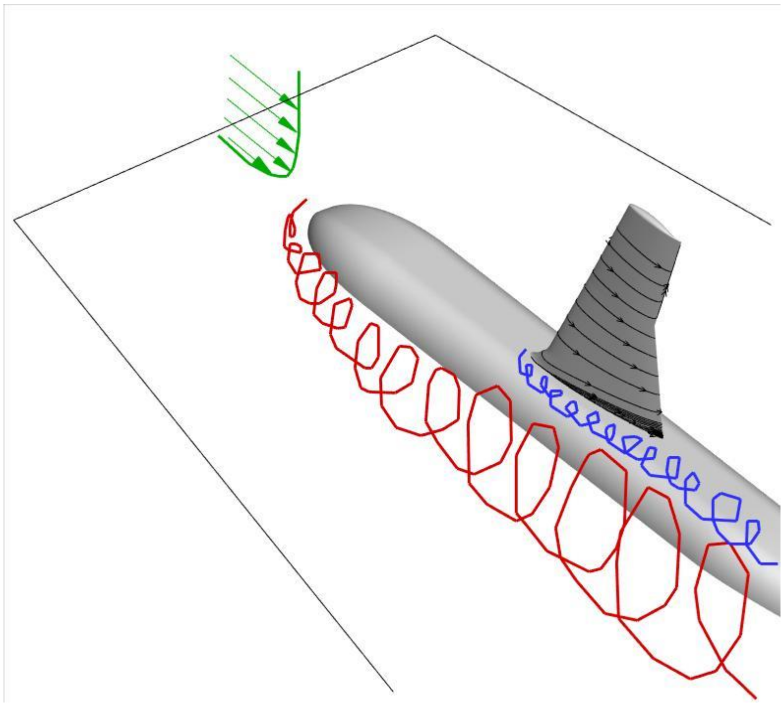

Figure 1-3: Schematic showing conjectured undesirable influence of fuselage-floor juncture vortex (red) on wing-fuselage juncture region (blue), likely to be present in semi-span testing.

In the full-span tests, the different wings produced the following results (in terms of SOB separation). The F6 wing was separated at all angles of attack and produced a range of separation sizes (from roughly 10mm wide to roughly 60mm wide at 6%-scale based on the oil flow patterns) with “classical”-looking oil-flow patterns characteristic of those expected on realistic configurations. The 0015-based wing produced attached flow at low angles of attack, and very small separation patterns (less than 10mm wide) above 5 degrees angle of attack. The 0015mod-based wing produced somewhat smaller separation patterns than the F6; the separation was always present. Both the 0015-based and the 0015mod-based yielded less “classical”-looking separation patterns (with a more triangular shape than the F6). The F6-S12 and COCA wings produced more oddly-shaped separation patterns than the other wings; their SOB separation persisted for all angles of attack. Also, these latter two wings possessed extremely large spanwise geometric slopes near midchord that would have made LDV measurements from inside the fuselage difficult.

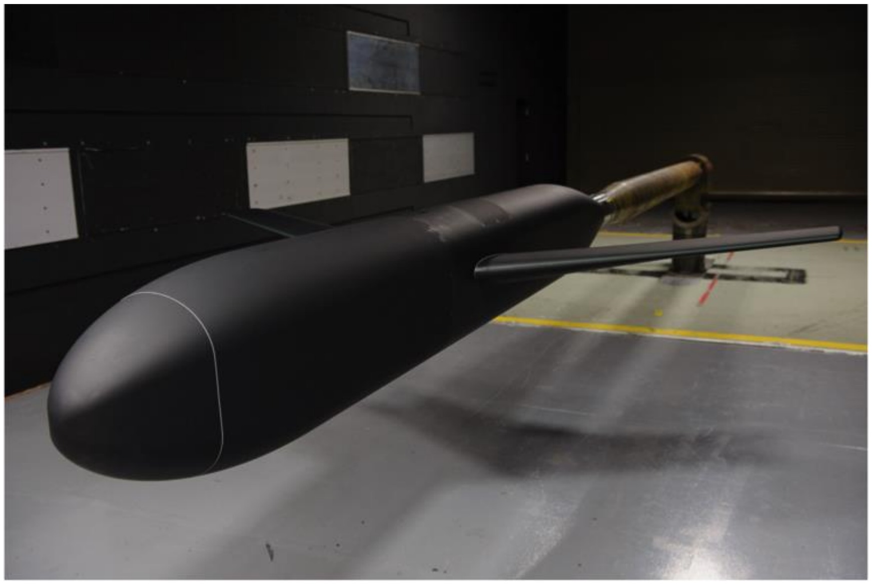

Figure 1-4: Photograph of 6%-scale JF model in the NASA Langley 14x22 wind tunnel.

Some surface oil-flow results near the base of the F6 and 0015-based wings from the 6%-scale test are shown in figures 1-5 and 1-6, respectively. (Note that the position of the camera is not identical for all of the photographs; therefore, the relative sizes of the separation patterns may not be representative of the true relative sizes.) The main items of note from these figures are the general separation shapes, as well as the fact that no separation is visible for the 0015-based wing at angles of attack of 0 and 5 degrees.

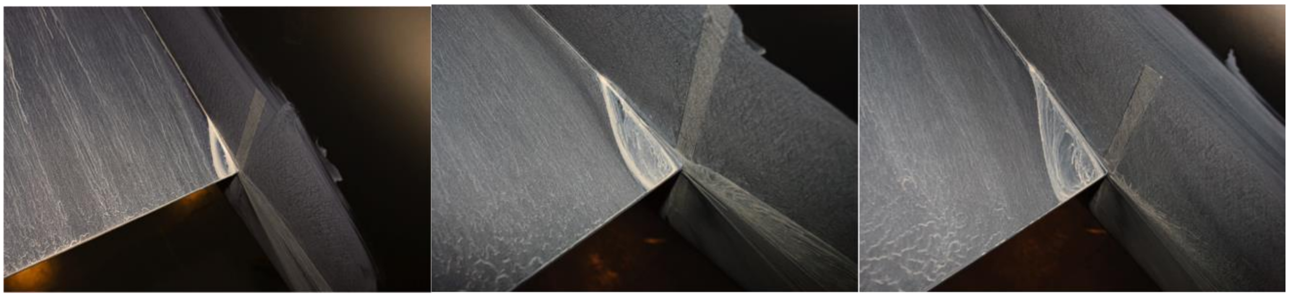

Figure 1-5: Oil flow patterns from the F6 wing (6%-scale model), angles of attack = 0, 5, and 10 from left to right.

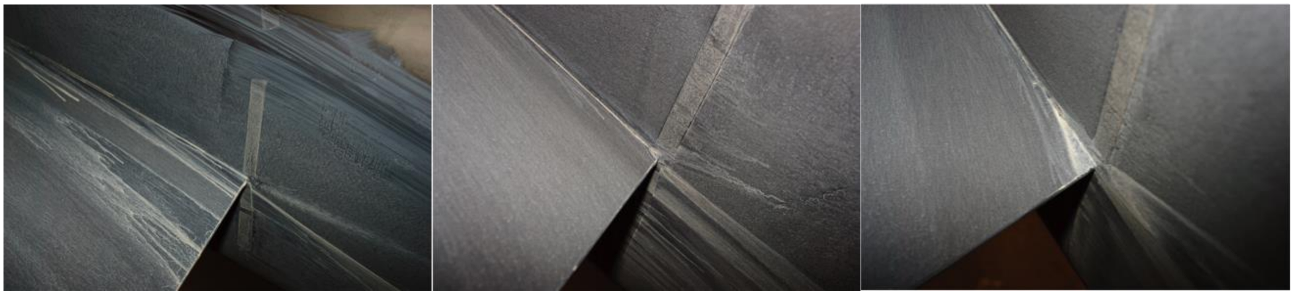

Figure 1-6: Oil flow patterns from the 0015-based wing, angles of attack = 0, 5, and 10 from left to right.

Unfortunately, none of the wing configurations managed to achieve the desired goal of yielding both fully attached flow (at some angles of attack) as well as significant SOB separation (at other angles of attack). Only the NACA 0015-based wing achieved fully-attached flow, but even at high angles of attack it exhibited only a tiny separation. Therefore, because the SOB separation on the F6 wing was believed to be more representative of those seen on realistic aircraft configurations, and because it varied from "small" to "large" over a reasonable range of angles of attack, the final selected wing was the F6-based wing. The JF team also decided to allow for the fabrication of an additional backup 0015-based wing, if it fits within the budget. The fuselage windows and instrumentation will be designed to accommodate both wings. This latter wing could then be substituted and used to explore cases involving incipient SOB separation. The JF team also decided to allow for two fuselage lengths for the model.

Efforts are currently underway to design and build the final model for testing in 2017–2018. Also, a few additional preliminary experiments are being conducted for further risk reduction, as well as for the purpose of learning how to better simulate the wind tunnel environment with CFD. These planned experiments are described next.

4.0 PLANNED EXPERIMENTS

A primary goal of the final experiment is to acquire CFD-validation quality data for wing-juncture trailing edge separation onset and progression on a full-span wing-body model. By "CFD-validation quality," we mean that the experiment should include the measurements of all information – including boundary conditions, geometry information, and quantification of experimental uncertainties – necessary for a thorough and unambiguous CFD validation study [14]. The attainment of this data will then enable the assessment of CFD models. The main purpose of the CFD model assessment is as follows:

> *Assess the ability of existing models to predict the onset and extent of the three-dimensionally separated flow near the Wing Juncture Trailing Edge (WJTE) region of a full-span wing-body configuration, in terms of the surface topology of the flowfield structure. To provide a range of prediction difficulty, a variation of WJTE flow fields are required, including the onset and progression of corner separation.*

However, as discussed in an earlier section, the JF team was unable to find a single configuration that achieved both fully attached flow as well as flow with reasonably large and representative SOB separation. The final compromise was the F6 wing (which achieves the latter), but an auxiliary set of wings (0015-based) may also be built for exploring incipient separation.

At the current time, there are two planned experimental campaigns for the 8%-scale model: one in the summer of 2017 and a second in the spring of 2018. Leading up to these final tests, some additional preliminary experiments are being conducted in the 14x22 wind tunnel during 2016.

4.1 Preliminary Tunnel Measurements and Needs for CFD Validation

The preliminary experiments planned for 2016 have several goals: (1) explore and improve the capability of PIV in the 14x22 wind tunnel, particularly for measuring large areas in the freestream; (2) document the 14x22 wind tunnel boundary conditions, with a primary focus of mapping the tunnel inflow uniformity; and (3) conduct specific preparations for the final test.

At the time of this writing, none of the preliminary experiments have been conducted yet. The PIV empty-tunnel test will be conducted in July 2016. The mechanics of measuring planes in all three axes will be explored. Both 2-D as well as stereo PIV will be used. It is not yet known if current PIV capabilities will have the accuracy necessary to characterize freestream flow non-uniformities (flow angularity), which may be as low as (on the order of) one-tenth of a degree.

The second test planned in September 2016 will focus primarily on tunnel inflow. Both empty tunnel as well as the 6%-scale model F6 wing will be used. The JF team envisions using Boeing's Quantitative Wake-Survey System (QWSS) device [15], which utilizes a 5- or 7-hole probe at the end of a movable arm that swings and maps out a large area. The probe measures static pressure, total pressure, and flow direction at each point in the flowfield; other quantities such as velocity and vorticity can be computed. The results from the QWSS will be compared with those from PIV at the same tunnel inflow streamwise station(s).

From the CFD perspective, inflow conditions would be desired well upstream of the test section, possibly as far forward as the contraction region (to minimize the influence of the installed model on the inflow boundary conditions). However, both measurement methods are currently limited in where they can be used. PIV is primarily limited by optical access, and the QWSS is limited primarily by mounting considerations. The furthest forward measurement station in the upcoming tests will be within about 8 feet downstream of the start of the test section. Like the PIV test, this QWSS test is also considered to be exploratory in nature, as it is not yet known if the QWSS will have the accuracy necessary for measuring freestream flow angularity.

The inflow mapping experiment will also include measurements with the 6%-scale JF model in place. The inflow mapping station will be approximately 5–7 feet forward of the 6%-scale model nose. The QWSS device will have an influence on the flow over the JF configuration itself, but details on the model are not of concern here. Instead, the JF team is looking for the influence of the model (as positioned) on the "freestream" flow at the currently-measured inflow plane.

In October 2016, the 6%-scale JF model will be used for additional preparatory testing in the 14x22 wind tunnel. IR reflection off the floor will be explored using various materials and attachment methods, as the final test requires a method for visualizing the transition on the lower surface of the wings. Also, new unsteady pressure sensors will be tested and a preparatory check of the LDV system (embedded in the tunnel side-wall boundary layer) will be made.

4.2 Experimental Measurements for the 8%-Scale Model

As described earlier, the primary configuration selected for the 8%-scale model testing in 2017–2018 is the F6 wing. The planned freestream Mach number is M=0.2, and the planned Reynolds number based on crank chord length is approximately 2.4 million. At this time, precise measurements and test matrix have not been defined. However, the following represents a prioritized list of desired data, as defined by the JF team (top priority first):

1. Documentation of the test article geometry and surface, as fabricated, as assembled, and as tested in the wind tunnel.

2. Tunnel boundary conditions upstream, around, and downstream of the test article for each angle of attack condition tested.

3. Mean velocity components through the boundary layer on the fuselage and wing (both upstream and within the WJTE region).

4. Second moments (Reynolds stress tensor components) through the boundary layer (same locations as in Item 3).
5. Time-averaged (steady) surface pressure coefficients on the fuselage and wing.
6. If boundary layer trip(s) are used, details regarding their location, type, attachment, etc.
7. Documentation of the surface geometry of the wind tunnel test section and the contraction section upstream of the test section.
8. Documentation of the geometry of the mounting hardware and cabling for the test article in the test section.
9. Time-series measurements of angular position and acceleration on-board the test article.
10. Surface shear stresses on the test article (both upstream and within the WJTE region).
11. Time history and spectra of unsteady velocity and pressure at various locations on the fuselage and wing (both upstream and within the WJTE region).
12. Freestream turbulence intensity and turbulence length scale.
13. Triple velocity correlations through the boundary layer (same locations as in Item 3).

The above items may or may not be possible to attain in the test; the prioritization will be used during testing to help make decisions when time is limited or when particular capabilities are not available. All of the mathematical model input data described above should include a rigorous estimate of total experimental uncertainty, i.e., both random (precision) uncertainty and systematic (bias) uncertainty.

Most likely, several different angles of attack using the F6 wing will be selected, yielding different sized SOB separations. Data will be collected for each. If the NACA 0015-based wing is also built, the same process would be followed for it. The JF team anticipates that existing turbulence models will be compared against the acquired data, and the models' ability to predict the SOB separation size, mean velocities, pressures, and turbulence quantities as a function of angle of attack will be assessed. The experimental data will be of high enough quality to encourage turbulence model development geared toward improving the prediction of these flows.

5.0 SUMMARY AND CHALLENGES

To summarize, experimental testing for a juncture flow model has been initiated through a series of risk-reduction tests, designed to lead up to a set of CFD-validation quality experiments in the 2017–2018 timeframe. These risk-reduction tests are being combined with preliminary CFD analysis as well as with other tunnel flow characterization tests in an effort to insure that the final tests will be useful for both evaluating CFD models as well as for improving them. At the current time, risk reduction tests have helped to define the configuration(s) of the final test, ensuring that wing-body-juncture separation bubbles will be present. The original goal of having a single configuration capable of obtaining both fully attached flow (at low angles of attack) and large side-of-body separation (at high angles of attack) was not realized. The decision was made to build an 8%-scale model with a truncated version of the DLR-F6 wing, since it achieved the most useful bubble sizes (for measurement), and was also representative of separation patterns seen in realistic configurations. Depending on budgetary constraints, a second set of wings with an NACA 0015 root chord shape will be built for testing incipient corner separation.

Some of the challenges of the juncture flow project are listed here. For one, it is not clear how best to fully characterize the sources of uncertainty in this type of experiment. Because of the focus on CFD validation, we desire to document the influence of geometric as well as flowfield uncertainties. The former is fairly easy to do, by performing laser scans of the as-built configuration, for example. But tunnel flowfield uncertainty (and its influence on detailed flowfield measurements such as velocity and Reynolds stress at specific locations) is more difficult to characterize. Therefore, we have initiated an effort to learn how to better measure tunnel inflow boundary conditions, particularly from the perspective of characterizing the flowfield nonuniformity. This is a research task at this stage, as the readily available techniques for use in the 14x22 wind tunnel may not be accurate enough (or able to measure far enough forward) for this purpose.

Another challenge is to encourage effective and persistent collaboration between the CFD practitioners and the experimentalists. A highly collaborative environment is undoubtedly conducive to progress. To date, the two elements of the current team have functioned very well together, but even greater integration and collaboration is always possible. For example, cross-training individuals to some extent may be useful, leading to better understanding of the capabilities and limitations of each discipline. Regular and persistent communication is important because it helps to break down barriers.

Finally, a significant challenge in the juncture flow project is balancing expectations against technical capabilities. This challenge is somewhat related to the barriers challenge (above). Both CFD and wind tunnel experiment have limitations, each of which may not be fully understood by everyone. For example, CFD's models are generally considered inaccurate for separated flows or for juncture-type flows (one of the main reasons for this project), and CFD is also heavily dependent on the quality and size of the grid employed, the convergence level of the particular CFD code employed and on the experience-level of the CFD practitioner running it. On the other hand, experiment often has difficulty fully characterizing the boundary conditions that would be necessary for a true apples-to-apples comparison with CFD, and also has many limitations in terms of measuring specific quantities of interest to CFD. The combination of CFD and experimental methods helps to overcome the limitations of each used alone and increases the knowledge of the flow considered. The current research project should lead to improved methodologies in both disciplines.

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] Spalart, P. R., Allmaras, S. R., "A One-Equation Turbulence Model for Aerodynamic Flows," *Recherche Aerospatiale*, Vol. 1, 1994, pp. 5–21.

[3] Spalart, P. R., "Strategies for Turbulence Modelling and Simulations," *International Journal of Heat and Fluid Flow*, Vol. 21, 2000, pp. 252–263.

[4] Gand, F., Deck, S., Brunet, V., Sagaut, P., “Flow Dynamics Past a Simplified Wing Body Junction,” *Physics of Fluids*, Vol. 22, 115111, 2010.

[5] Oberkampf, W. L., Smith, B. L., “Assessment Criteria for Computational Fluid Dynamics Validation Benchmark Experiments,” AIAA Paper 2014-0205, January 2014.

[6] Gand, F., Brunet, V., Deck, S., “Experimental and Numerical Investigation of a Wing-Body Junction Flow,” *AIAA Journal*, Vol. 50, No. 12, 2012, pp. 2711–2719.

[7] Rumsey, C. L., Neuhart, D. H., Kegerise, M. A., “The NASA Juncture Flow Experiment: Goals, Progress, and Preliminary Testing,” AIAA Paper 2016-1557, January 2016.

[8] Kuester, M. S., Borgoltz, A., Devenport, W., “Experimental Visualization of Junction Separation Bubbles at Low- to Moderate-Reynolds Numbers,” AIAA Paper 2016-3880, June 2016.

[9] Tinoco, E. N., Venkatakrishnan,V., Winkler, C., Mani, M.,Vassberg, J., Kulfan, B. M., “Structured and Unstructured Navier-Stokes Solvers for the Third Drag Prediction Workshop,” *AIAA Journal of Aircraft*, Vol. 45, No. 3, 2008, pp. 738–749.

[10] Brodersen, O., Sturmer, A., “Drag Prediction of Engine-Airframe Interference Effects Using Unstructured Navier-Stokes Calculations,” AIAA Paper 2001-2414, June 2001.

[11] Rivers, M. B., Dittberner, A., “Experimental Investigation of the NASA Common Research Model,” AIAA Paper 2010-4218, June–July 2010.

[12] Campbell, R. L., “Efficient Viscous Design of Realistic Aircraft Configurations,” AIAA Paper 98-2539, June 1998.

[13] Roozeboom, N. H., Lee, H. C., Simurda, L. J., Zilliac, G. G., Pulliam, T. H., “Comparison of Experimental Surface and Flow Field Measurements to Computational Results of the Juncture Flow Model,” AIAA Paper 2016-1558, January 2016.

[14] Aeschliman, D. P., Oberkampf, W. L., “Experimental Methodology for Computational Fluid Dynamics Code Validation,” *AIAA Journal*, Vol. 36, No. 5, 1998, pp. 733–741.

[15] Kusunose, K., Crowder, J. P., “Extension of Wake-Survey Analysis Method to Cover Compressible Flows,” *AIAA Journal of Aircraft*, Vol. 39, No. 6, 2002, pp. 954–963.