31st Congress of the International Council
of the Aeronautical Sciences

Belo Horizonte, Brazil; September 09-14, 2018

CFD VERIFICATION AND VALIDATION FOR INDUSTRIAL APPLICATIONS

Mori Mani* , Andrew Cary*
*The Boeing Company

Keywords: *CFD, Verification, Validation*

Abstract

This paper highlights CFD verification and validation (V&V) at Boeing Research & Technology (BR&T). As CFD becomes a significant source of data at Boeing for design and certification it is important that V&V is performed and documented for each code. This article particularly highlights solution and code verifications and provides a present list of preferred test cases for verification of internal and external flows used within Boeing. In this manuscript we distinguish between benchmarking and validation and provide a number of example cases in how BR&T conduct each at Boeing. At Boeing majority of the code evaluation cases fall within benchmarking due to the lack of experimental data that meets the validation criteria.

1 Introduction

Computational Fluid Dynamics (CFD) is in wide use at Boeing for design, analysis, optimization, and certification. Over the past decade, CFD has played a major role in providing the necessary data for preliminary design and optimization and now is in process of being used for certification. As the paradigm shifts from experimental data exclusively to the use of experimental and computational data for design, optimization, and certification, the accuracy of the CFD code and process must be quantified. There are two parts to CFD code and process accuracy quantification: verification and validation. Verification is establishing that the correct equation is being solved and the right process is being applied. In another word, it is the process of quantifying the accuracy of the discretized solution relative to the exact solution of the model equation. Validation is the process of quantifying the simulation results against "physically-realizable" experimental results. An important question is to assess how accurately the discrete solution should represent the physical reality. Validation as stated above answers how well the model equations represent the real world physics of the considered phenomena, while verification answers how accurately the governing equations are discretized and solved. Therefore, both verification and validations are necessary to gain confidence in the CFD code being used to obtain simulation results.

Verification splits to two parts, code verification and solution verification. Code verification is concerned with how accurately the model equation is discretized and implemented as well as ensuring that no bugs have been introduced, while solution verification addresses how well the solution to the governing equations is resolved. Solution verification relates itself to estimating numerical errors associated with grid resolution, time-step, relaxation parameters, convergence order, and other parameters of the solution process.[1, 2, 3, 4]

Uncertainty Quantification (UQ) is another important element of the CFD process to quantify the confidence in the CFD solution. This topic is beyond the scope of this paper. The readers are referred to References[5, 6].

2 Verification and Validation at Boeing

2.1 Verification at Boeing

A closed form solution of the Euler and Navier-Stokes equations for 3D configurations with some complexity does not exist for many situations of interest. For rigorous verification it is possible to use Method of Manufactured Solution (MMS). However, such verification requires the ability to introduce a source distribution, usually through source code modification. At Boeing, internally developed, NASA developed, and commercially available CFD codes are used due to diversity of applications. To avoid the MMS requirement for ability to specify a source distribution, the CFD codes are verified using exact solutions and rigorously established benchmark cases that have been evaluated using codes previously verified with MMS[3, 7]. At Boeing, code verification is performed prior to release of a new version of an internally developed code or when a new version is acquired from external sources like NASA or commercial vendors. This responsibility primarily belongs to the code developer or Technical Lead Engineer (TLE) in the appropriate subject matter. The verification is performed on a number of different cases, as identified in Fig. 1, for example. These cases have been selected as such to exercise various parts of the code including discretization of different terms and different interactions. The results obtained are compared against exact solutions or other rigorously established numerical results, such as the NASA Turbulence Modeling Resource (TMR)[8]. The order of accuracy of solution is also compared against the code formal order of accuracy. While verification assesses all elements of the analysis process from grid generation to post-processing, including all functional reductions being used in intended program applications, only flow solution verification will be focused on in this section.

For industrial CFD codes, the number of features and potential interactions of terms are quite large, so a careful set of simulations has been developed to test many of the different features simultaneously, with failures in particular combinations of cases being used to highlight potential root causes. As an example of this, the Mach 0.5 Joukowski airfoil case is solved for inviscid, laminar, and turbulent flows with particular turbulence models. The inviscid solution helps to ensure that the pressure field is computed appropriately, the laminar solution (Re=1000) provides evidence that the viscous discretization is appropriate, and the turbulent simulation (Re=1E6) demonstrates appropriate solution and coupling of the turbulence model. This case features a Joukowski airfoil to reduce the singularity in the pressure field at the trailing edge. Not all codes in use have been verified for all of these cases due to limited resources, identification of a confounding issue, or lack of need/capability for the code use.

Case Name	BCFD	OFLOW	CFD++	GGNS	FUN3D
2D Inviscid Bump	✓	✓	✓		✓
Inviscid Oblique Shock	✓		✓		✓
Joukowski Airfoil, Inviscid					
Joukowski Airfoil, Laminar					
Joukowski Airfoil, Turbulent	✓		✓		
2D Airfoil Near-wake	✓	✓	✓	✓	✓
2D Bump	✓	✓	✓		✓
Taylor-Green Vortex	✓		X	X	X

Fig. 1 : Example of verification cases at Boeing.

2.2 General verification requirements

Oberkampf and Trucano[4] perform an extensive review of verification and identify several characteristics of algorithm testing and strong benchmarks that are applicable to the present effort. They argue for "exact, standardized, frozen, and promulgated" definitions, statement of purpose, requirements for code comparison, and success criteria. The following examples attempt to do these (at least within Boeing) for a given problem. In this context, the definition of the case provides all relevant information to state the mathematical problem. The statement of purpose is used to identify what features/attributes are expected to be tested by the case. Comparison of results to the reference solution need to be specifically defined to ensure that results are being properly assessed. For example, while drag could be defined by looking at a momentum deficit at the far field or by integrating on the surface, a specific methodology is specified for the verification problem. Finally, a clear acceptance criteria needs to be defined. These metrics should be based on engineering significance, but since verification is a mathematical exercise, tighter tolerances should be applied.

A standard template has been generated to define the verification problem and document the results. Note that these definitions are sufficiently general that they do not have to be applied only to the flow solver but could also be applied for grid generation or post-processing (or other) codes as a means for documenting correctness of calculations. To properly complete the verification, solutions are required to be generated on one or more families of grids.

2.3 Example verification case: inviscid bump

The 2D inviscid bump was used to establish steady subsonic solution convergence in the presence of curved wall boundaries. The compressible Euler equations (steady, 2D) with calorically perfect gas was employed for this verification. The computational domain extends from $-1.5 \leq x \leq 1.5$ and $0.0625\exp(-25x^2) \leq y \leq 0.8$. The upper and lower boundaries are slip walls. The inflow boundary is specified as uniform flow at a nominal Mach number of 0.5 parallel to the x axis at atmospheric pressure with fixed total pressure and total temperature. The outflow boundary is set to free-stream static pressure. The acceptance criteria is that the convergence of entropy error on nested family of hexahedral meshes should be consistent with other solvers of the same order (within $\pm 5\%$).Sample results are shown in Fig. 2 for an integrated entropy norm convergence. Observe that second order finite volume flow solvers demonstrate an order of accuracy greater than the theoretical value of two for this metric on this problem as a result of the non-linearities. Similar results are seen for other metrics. Because this is an inviscid shock-free solution, the entropy should be uniform; increasing grid resolution shows this trend. Some of the codes tested failed this evaluation for reasons suspected to be

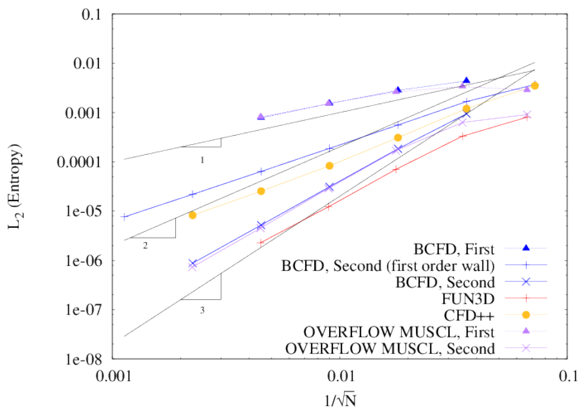

Fig. 2 : L2 norm of entropy convergence with grid length metric. BCFD (with second order inviscid wall boundary condition) has consistent observed order of accuracy with second order OVERFLOW and FUN3D.

inconsistent order of accuracy for the inviscid wall boundary condition. This behavior was duplicated using BCFD to illustrate the cause. This example highlights how multiple aspects of simulation codes interact even on simple cases and can create verification challenges.

2.4 Verification case findings

In addition to the inviscid cases described above, a range of turbulent cases have also been utilized, leveraging the AIAA Higher order CFD workshop[9] and the TMR[8]. While many of these cases are two-dimensional, multiple grid topologies contribute to the confidence that the CFD codes are properly verified.

General observations associated with performing these verification cases indicate that it can be a challenging proposition in that a significant number of details must be properly executed to get the appropriate results and seemingly minor approximations can lead to significant differences in the results. Some of these experiences compared to linear codes or academic problems have been outlined above where the assumed order of accuracy is not observed even when answers are changing below engineering tolerance and discontinuities in the flow can disrupt the expected convergence rate in some quantities. Other cases have demonstrated how very minor differences in implementation of characteristic boundary conditions can lead to different results. Since a user does not always have access to this level of detail, attempting verification without code access can be challenging. If code is available, MMS provides a mechanism to more thoroughly isolate different components for testing and would generally be recommended. Code developers/vendors should provide adequate evidence of verification to support the end user's confidence to perform validation and benchmarking.

3 Validation/Benchmarking

3.1 Validation and Benchmarking at Boeing

The purpose of validation is to ensure that the CFD code is appropriate for the intended use. At Boeing that means to be able to use the CFD data with confidence in design, analysis, and certification. Therefore, the goal of validation is to assess the validity of the model in accurately simulating the flow-field and predicting functional of interest for intended applications. At Boeing, the validation contains the CFD process from geometry fidelity to post-processing. To validate our CFD code we perform a grid resolution study to demonstrate the grid convergence, and utilize Richardson extrapolation [1, 3] to obtain the functional interest at infinite grid resolution for comparison with experimental data. A validation process has two key elements: the validation experiment and model accuracy assessment. These are define as follows in Reference[1]

- Validation Experiment: experiments performed expressly for the purpose of validating the model.
- Accuracy Assessment: quantify how well the experiment and simulation results compare.

Note that most of the experimental data typically collected in an engineering campaign do not meet the "Validation Experiment" requirements

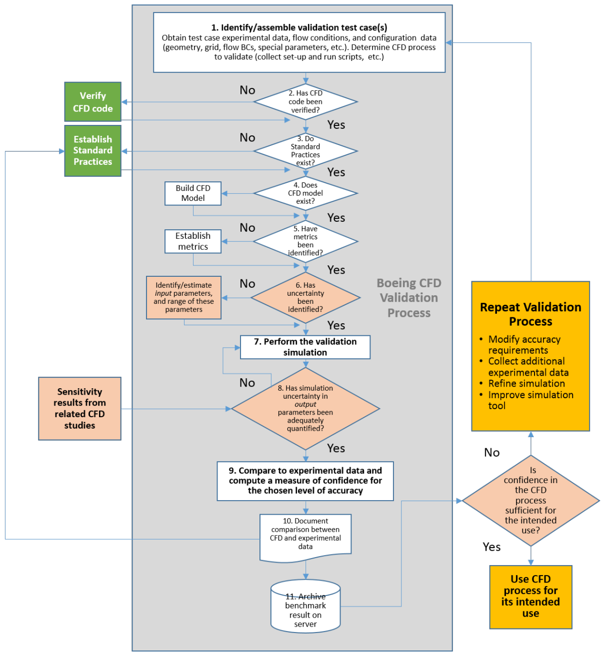

Fig. 3 : Boeing validation process.

due to their intended use being different than obtaining data for validation and a general incompleteness in measurements and error/uncertainty quantification. However, experimental data with error quantifications are highly valuable in CFD benchmarking. The distinction between validation and benchmarking is the level of rigor. In benchmarking cases, portions of the validation process such as grid resolution studies may be skipped and experimental data used that were not obtained with the rigors required for the "validation" purpose.

The procedure that Boeing follows is similar to the one developed by the AIAA[2] with slight modifications and is shown in Fig. 3. Some of the key additions compared to the AIAA process are associated with the use or development of best practices, an emphasis on the entire process that will be used in practice including grid generation, and specific guidance on assessing input uncertainty and comparing results to experimental data.

3.2 Example Benchmarking Case: Supersonic Square Duct

Supersonic flow through a square duct is altered by secondary vortical flow developing from the corners. These secondary flows are generated by Reynolds stress gradients acting in the corner region and appear to have similar structure to those found in subsonic flow through square ducts[10, 11, 12, 13]. Such flow is representative of various airplane inlets and therefore it is important to understand and predict the impact of this secondary flow on inlet pressure recovery and distortion. An experimental study was performed at the University of Washington to gain a better understanding of how the secondary flow associated with corners affects local flow conditions in a square duct over the boundary layer development length[12]. This configuration was utilized in this study to verify the implementation of Quadratic Constitutive Relation (QCR) into SA turbulence model, as well as validate the SA-QCR turbulence model in BCFD for corner flow simulation at Boeing.

Supersonic flow through the duct employed in this study was tested by Davis and Gessner[12] at the University of Washington. The square duct tested has a length-to-width ratio of 50, with dimensions of 1x1x50 inches (Fig. 4). The computational domain consists of two zones: an upstream zone to ensure clean flow entering duct, and the test section highlighted in red in Fig. 4. A grid resolution study was performed to ensure the results presented in this manuscript are mesh-converged. Two different grid families were used in this analysis: a nested structured grid family with appropriate boundary layer resolution and a family of prismatic-tetrahedral meshes (mixed-element mesh) that used the same set of surface nodes as the corresponding structured grid resolution. This grid study is not discussed further in this paper, but this use of multiple meshes reflect the expected use of unstructured mixed-element meshes in practice. Based on the experiment performed at the University of Washington, the flow is steady and symmetric and therefore only 1/4 of the configuration was used to perform the body

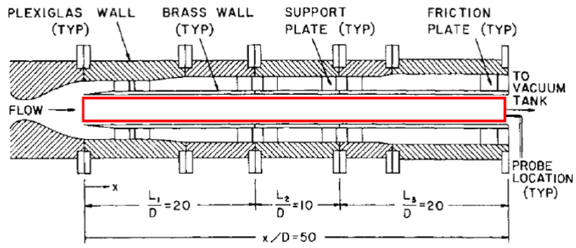

Fig. 4 : Supersonic square duct benchmarking case.

of the work as well as the mesh resolution study. A periodic boundary condition was used to connect the quadrants. A CFD run was obtained on the full configuration to ensure the flow is steady and symmetric, as was predicted in the experiment.

The standard SST, SA, and SA-RC turbulence models were selected for the purpose of this study as they are the primary turbulence models used at Boeing for design and analysis. The CFD results obtained in this study are compared with the experimental measurements of total pressure and mean velocity which were collected at four axial locations, X/D of 5.37, 20, 40, and 50. In this paper we show only the results at $X/D = 50$, for other location see Reference[13].

All turbulence models predicted the wall bisector velocity profile with reasonable accuracy at $X/D = 50$ where the flow is fully developed. However, none of the standard turbulence models demonstrated good comparison to the velocity distribution at the corner bisector as shown in Fig. 5.

The skin friction results are shown in Fig. 6 for $X/D = 50$ and none of the eddy viscosity models predict the experimental profile well.

These results clearly indicate the shortcomings of eddy viscosity models due to the lack of ability to predict the aspects of anisotropy, namely the differences between streamwise, wall-normal, and lateral Reynolds stresses, which create the secondary flow. This result is not new and was expected, as it has been known for many decades that eddy viscosity models will not predict square duct flow. The QCR model developed by Spalart[14] has been implemented into

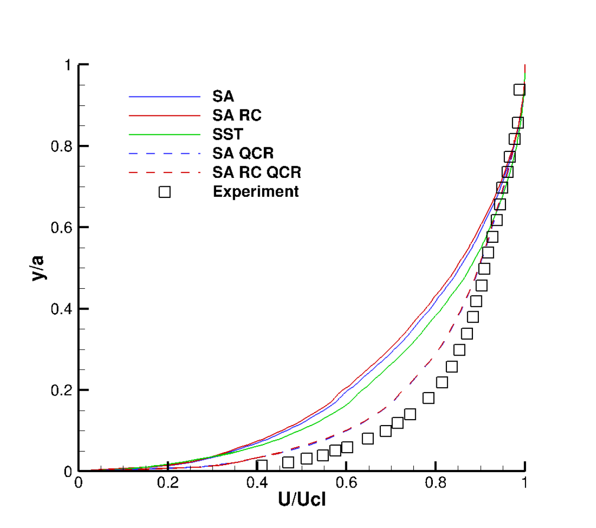

Fig. 5 : Corner bisector velocity profile at $X/D = 50$.

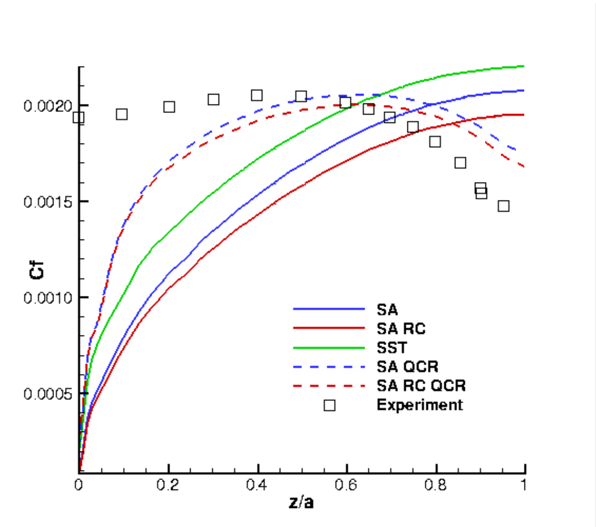

Fig. 6 : Skin friction distribution at $X/D = 50$.

BCFD[15] for the SA and SST turbulence models. The same case was rerun to investigate and document the improvements associated with the QCR for supersonic turbulent flow in a square duct with significantly better agreement to the experiment, as shown with dashed lines in Fig. 5 and 6.

3.3 Examples of Validation Cases

Compared to benchmarking, validation cases take the CFD assessment further in comparison to experiment by including uncertainty analysis and more rigorous discretization error analysis. Including these steps enable a comparison to careful experiments that can highlight capabilities and shortcomings associated with physical modeling. These validation experiments typically require additional measurements than typical engineering experiments not only to develop data to corroborate with simulation, but also to inform potential experimental uncertainties associated with geometry or inflow conditions.

The following sections describe two example cases where this level of rigor has been applied and identifies some of the challenges associated with performing this type of analysis.

3.3.1 Backward-Facing Step

The backward-facing step validation case from TMR[8] is based on an experiment by Driver and Seegmiller[16] where the exit of the channel is 9 times the step height and the Reynolds number based on step height is $Re_H = 36,000$. The specific upstream conditions of the experiment were determined by measuring the temperature and the velocity profile 4 step heights upstream of the step. For the CFD simulation, the exit pressure was adjusted to achieve a peak Mach number of 0.128 at this location, consistent with experimental observation. The inflow length of the channel was established to provide a naturally developed turbulent boundary layer of approximately the same thickness as observed experimentally prior to the step ($1.5H$). One of the key metrics for this test case is the location of the reattachment bubble which was determined experimentally to be $x/H_{\text{reattach}} = 6.26 \pm 0.1$ by interpolating laser oil-flow interferometer measurements of skin friction to zero. This feature was observed to be largely invariant with respect to span of the tunnel. In addition to the measurements of reattachment length, lower wall pressure and skin friction measurements were made, as well as velocity profiles at several stations both upstream and downstream of the step.

To assess discretization error, 5 uniformly-refined structured grids are defined on the TMR ranging in size from $5K$ cells to $1.3M$ cells. Present BCFD simulations with the Spalart-Allmaras turbulence model suggested that the solution was not yet asymptotic, so three additional grids were generated by further refinement with the finest grid being $82M$ cells (in 2D). Each of these simulations were converged to machine-zero to eliminate concerns of convergence error. The resulting grid convergence of the reattachment length is shown in Fig. 7. One of the reasons for the abrupt shifts in these results are that as the region near the backward-facing step is resolved, additional recirculation vortices appear that result in a shift in the overall reattachment length. The additions of these features significantly delay reaching asymptotic limits for convergence and highlight one of the challenges of performing validation analysis on non-trivial cases. Note that skin friction upstream of the step and far downstream of the step show smooth convergence with grid, emphasizing the importance to evaluate multiple metrics for grid convergence.

Because of the use of the experimental measurements to develop the boundary conditions for the CFD simulations, no further uncertainty was associated with these conditions in this study. However, uncertainty in the turbulence model coefficients was investigated using non-intrusive polynomial chaos similar to the approach taken in [17]. As in the previous study, the uncertainty in three coefficients were given by $\sigma \in [0.6, 1.0]$, $\kappa \in [0.38, 0.42]$, and $c_{w3} \in [1.75, 2.50]$. This analysis was performed at two different grid levels with very consistent results, suggesting that the discretization error is much smaller than this uncertainty for this particular case. In terms of reattachment length, the uncertainty interval associated with the turbulence model coefficients is about $0.595H$, whereas the discretization uncertainty is about $0.011H$. Fig. 8 depicts the flow field and identifies the experimental attachment length location, as well as the nominal turbulence model result and the associated uncertainties including both discretization error and turbulence model coefficient uncertainty. Since the experimental range is contained within the numerical estimation of the range, there is no evidence that the turbulence model is incorrect for this quantity. However, comparisons between the simulation and the experiment in Fig. 9 for the pressure coefficient within the bubble illustrate that differences exist in these regions of the flow.

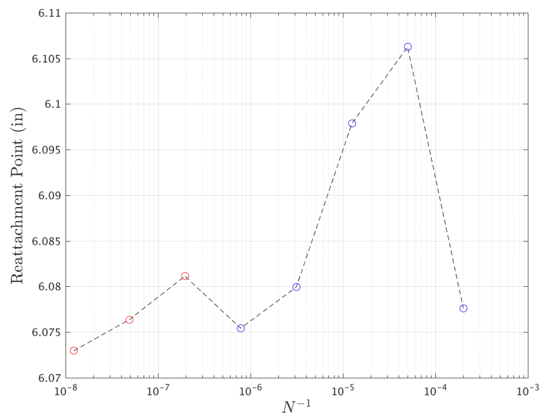

Fig. 7 : Grid convergence of reattachment length of backward-facing step.

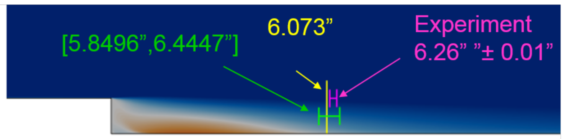

Fig. 8 : Illustration of backward-facing step flow field with reattachment length annotation.

3.3.2 *Common Research Model Example*

The common research model is a representation of a modern commercial transport developed by Boeing and NASA to provide an openly-available relevant geometry. This model was initially used

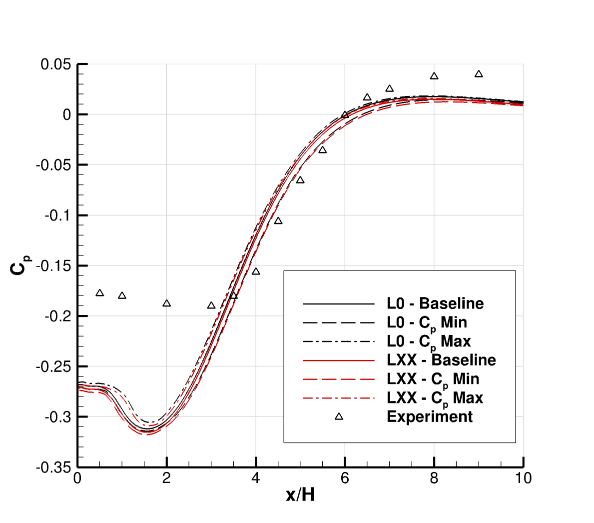

Fig. 9 : Pressure coefficient on lower wall of backward-facing step validation case comparing experiment with CFD (including uncertainty).

as part of the AIAA Drag Prediction Workshop series in DPW-4 and was subsequently tested in multiple NASA facilities. This configuration is used as a demonstration of full-configuration validation at a transonic Mach number of 0.85, a chord Reynolds number of 5×10^6 representing the wind tunnel condition and a lift coefficient $C_L = 0.5$ (with tail incidence at zero). In addition to the discretization error and epistemic uncertainty on the turbulence model described in the previous section on the backward-facing step, aleatory uncertainty associated with expected variations in the wind tunnel test conditions were included based on NTF estimates[18]. The Mach number was taken to be normally distributed about 0.85 with a standard deviation of 0.005. Similarly, the angle of attack and angle of sideslip were assumed to also be normally distributed about the nominal lift coefficient with no side slip with a standard deviation of $0.008°$ and $0.01°$, respectively. Although subsequent investigations have identified impacts from the blade sting mount and the tunnel walls, these effects are assumed to have been corrected out. There is additional evidence that the aeroelastic effects were not properly represented in the geometry used for the computational model, but these effects have been ignored for the present example.

With this combination of uncertainties, a probability box representation of the predicted drag coefficient is depicted in Fig. 10. In addition to identifying the epistemic uncertainty associated with the turbulence model and the additional uncertainty associated with the discretization error, the effect of the aleatory uncertainty in freestream conditions is reflected by the cumulative density function aspect of the probability box. The experimental data, interpolated to $C_L = 0.5$ from multiple tests are also illustrated in the figure as a set of discrete dashed lines. Note that the higher drag results come from experiments performed in the NASA Ames 11foot tunnel compared to the other results from the NASA NTF facility. The predicted uncertainty band for the BCFD simulations has a significant overlap with the experimental observations, but there is a region where the probability boxes do not overlap; this region is the evidence for disagreement between the experimental and numerical results and provides a representation of model error, given the previous assumptions. Fig. 11 and Fig. 12 provide two representative pressure profiles that highlight the comparison between experiment and simulation in this case. The experimental measurement error bars are smaller than the symbols used for plotting and the primary uncertainty associated with the CFD simulations are observed to be associated with the shock location.

4 Summary

In this paper we provided the definitions of verification, validation, and benchmarking that have been adopted by Boeing. The process that is followed and some of the test cases in use for V&V have been provided. The verification and validation is an expensive proposition, however, CFD is an essential element of the design, certification, and risk mitigation at Boeing and it is essential to be performed. The use of CFD data without V&V at best is risky. The CFD technology is still maturing and V&V should remain an essential part of the process. The Uncertainty Quantification was not discussed in this paper, however,

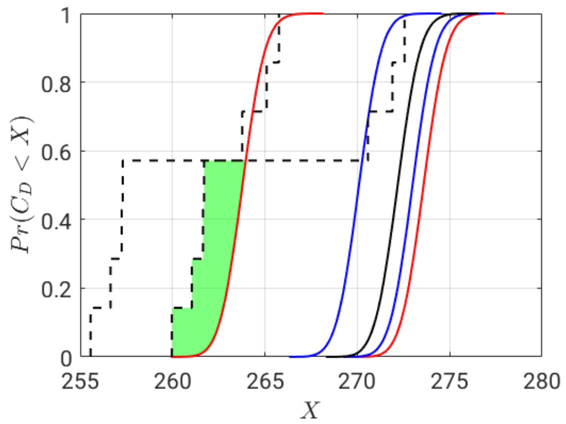

Fig. 10 : Probability box comparing BCFD simulation with experimental observation of drag coefficient on CRM at specified conditions. Solid black line represents aleatory uncertainty predicted from input uncertainties, interval between blue curves is uncertainty associated with free stream conditions and turbulence model coefficients, region between red curves is additional uncertainty associated with discretization error. Black dashed line represents experimental uncertainty from multiple experiments. Green shaded area represents the region of discrepancy between the two methods.

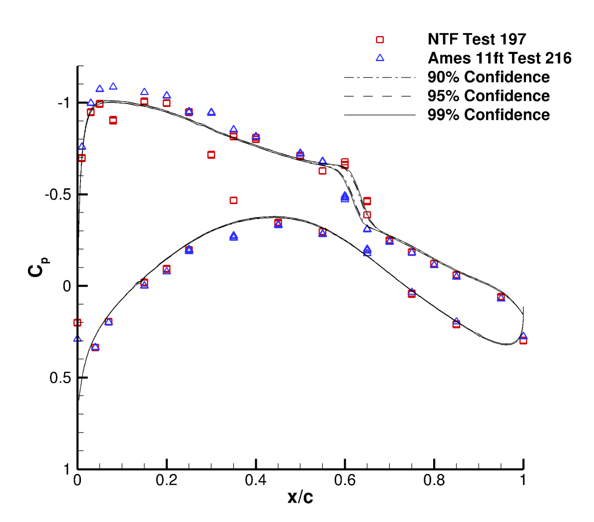

Fig. 11 : Pressure coefficient distribution at 30% span on CRM wing.

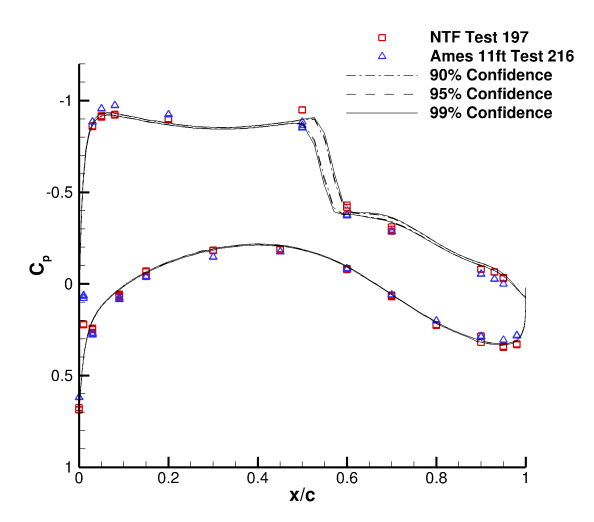

Fig. 12 : Pressure coefficient distribution at 50% span on CRM wing.

it was employed in one case to indicate its importance in validation process.

Acknowledgments

The authors appreciate the contributions of several individuals to the generation of data for this effort. John Schaefer supported many of the verification and uncertainty quantification computations. Matthew Lakebrink performed the inlet simulations. Ari Glezer of Georgia Tech led the supporting experimental effort for the inlet.

References

[1] American Society of Mechanical Engineers. V&V 20-2009. "Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer." 2009. ISBN: 9780791832097.

[2] American Institute of Aerodynamics and Astronautics. "AIAA Standard for Code Verification in Computational Fluid Dynamics." May 2017.

[3] Roache, P. J., *Fundamentals of Verification and Validation*. Hermosa Publishers. Socorro, New Mexico. 2009.

[4] Oberkampf, W. L. and Trucano, T. G. "Verification and Validation Benchmarks." *Nuclear Engineering and Design*. Vo. 238, No. 3, pp. 716-743. 2008.

[5] Roy, C. J. and Oberkampf, W. L. "A Comprehensive Framework for Verification, Validation, and Uncertainty Quantification in Scientific Computing." *Computer Methods in Applied Mechanics and Engineering*. Vol. 200. pp. 2131-2144. 2011.

[6] National Aeronautics and Space Administration (NASA). "Standards for Models and Simulations." NASA Technical Report NASA-STD-7009. Washington DC. 2008.

[7] Oberkampf, W. L. and Roy, C. J. *Verification and Validation in Scientific Computing*. Cambridge University Press. Cambridge, UK. 2010.

[8] Rumsey, C. L. "Langley Research Center Turbulence Modeling Resource." http://turbmodels.larc.nasa.gov.

[9] "4th International Workshop on High-Order CFD Methods." http://how4.cenaero.be. 2016.

[10] Melling, A. *Investigation of Flow in Non-Circular Ducts and Other Configurations by Laser Doppler Anemometry* Ph.D. Thesis. University of London. London. 1975. See also *Journal of Fluid Mechanics*. Vol. 78. Pt 2. pp. 289-315. 1976.

[11] Lund, E. G. "Mean Flow and Turbulence Characteristics in the Near Corner Region of a Square Duct". M.S. Thesis, Dept. of Mechanical Engineering, University of Washington. Seattle, WA. 1977.

[12] Davis, D. O. and Gessner, F. B. "Further Experiments on Supersonic Turbulent Flow in a Square Duct" *AIAA Journal*. Vol. 27. No. 8. August 1989.

[13] Mani, M., Babcock, D., Winkler, C. M., and Spalart, P. R. "Predictions of a Supersonic Turbulent Flow in a Square Duct." AIAA Paper 2013-0860. 2013.

[14] Spalart, P. R. "Strategies for Turbulence Modeling and Simulation." *International Journal of Heat and Fluid Flow*. Vol 21. pp. 252-263. 2000.

[15] Cary, A. W., Dorgan, A. J., and Mani, M. "Towards Accurate Flow Predictions using Unstructured Meshes." AIAA paper 2009-3650. June 2009.

[16] Driver, D.M. and Seegmiller, H.L. "Features of Reattaching Turbulent Shear Layer in Divergent Channel Flow". *AIAA Journal*, Vol. 23, No. 2, pp. 163-171, 1985.

[17] Schaefer, J., Cary, A., Mani, M., Spalart, P. "Uncertainty Quantification and Sensitivity Analysis of SA Turbulence Model Coefficients in Two and Three Dimensions." AIAA Paper 2017-1710. January 2017.

[18] Rivers, M. personal communication. November 2016.

Contact Author Email Address

mailto:mori.mani@boeing.com

Copyright Statement