Recommendations for Future Efforts in RANS Modeling and Simulation

Robert H. Bush[1]
Pratt & Whitney, East Hartford, CT, 06118, USA

Tom Chyczewski[2]
Northrop Grumman Corporation, Melbourne, FL, 32904, USA

Karthik Duraisamy[3]
University of Michigan, Ann Arbor, MI 48109, USA

Bernhard Eisfeld[4]
German Aerospace Center (DLR), D-38108 Braunschweig, Germany

Christopher L. Rumsey[5]
NASA Langley Research Center, Hampton, VA, 23606, USA

and
Brian R. Smith[6]
Lockheed Martin Aeronautics Company, Fort Worth, TX, 76101, USA

The roadmap laid out in the CFD Vision 2030 document [1] suggests that a decision to move away from RANS research needs to be made in the current timeframe (around 2020). This paper outlines industry requirements for improved predictions of turbulent flows and the cost-barrier that is often associated with reliance on scale resolving methods. Capabilities of RANS model accuracy for simple and complex flow flow fields are assessed, and modeling practices that degrade predictive accuracy are identified. Suggested research topics are identified that have the potential to improve the applicability and accuracy of RANS models. We conclude that it is important that some part of a balanced turbulence modeling research portfolio should include RANS efforts.

I. Introduction

Reynolds Averaged Navier Stokes (RANS) turbulence models have been the basis of Computational Fluid Dynamics (CFD) modeling of turbulent flows for many decades. As CFD evolved to predict viscous wall bounded flows, turbulence modeling evolved rapidly from algebraic to one and two equation turbulence models. These models have proven to be robust and accurate for a wide variety of turbulent flows, and provide the basis of the majority of engineering CFD simulations. While there have been advances and generalizations that extend the applicability and accuracy of RANS models for specific applications, there have not been significant advances in predictive accuracy over the past 20 years.

During the July 2017 Turbulence Modeling Workshop held in Ann Arbor, Michigan [2], there was discussion of an "ultimate barrier" with respect to RANS turbulence modeling. This barrier implies that there is an upper bound

---

[1] Nozzle Aerothermal Chief, AIAA Associate Fellow.
[2] Engineering Technical Specialist, AIAA Senior Member.
[3] Associate Professor, Department of Aerospace Engineeering, AIAA Member..
[4] Research Scientist, Institute of Aerodynamics and Flow Technology, Center for Computer Applications in AeroSpace Science and Engineering. Non member.
[5] Senior Research Scientist, Computational AeroSciences Branch, AIAA Fellow.
[6] Lockheed Martin Senior Fellow, Advanced Development Programs, AIAA Fellow

for RANS modeling accuracy that is due to fundamental limitations in the RANS approach, and that for many applications current capabilities may be near this barrier. However, this notion is a matter of significant disagreement. While some feel we are approaching a limit, others believe that if the limit exists, it is problem dependent. In cases where problems are beyond current model capabilities, many believe that significant improvements can extend the accuracy and affordability of RANS computations to advance fluid dynamic design capabilities.

The well documented RANS accuracy limitations have led to a focus of research investment on scale resolving methods for flows that RANS cannot predict well in the mean. The focus on Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) in the CFD Vision 2030 document (Slotnick et al [1]) is a reflection of this emphasis. The document recommends a decision point to phase-out RANS modeling research in the current time frame. We believe that the lack of progress in RANS modeling is partially due to limited funding for this effort over the past 20 years. In this paper, we identify several areas where current turbulence modeling capabilities are immature and where we believe current capabilities are far from an "ultimate barrier." These areas include turbulence chemistry interactions, two phase flow, rotating machinery, hypersonics and uncertainty quantification. We also believe that modeling approaches beyond one and two transport equation turbulence models could provide broad improvement in modeling capabilities. These approaches include expanded development of Reynolds stress closure, data driven modeling, and hybrid RANS/LES .

Given the large industry-wide investments in computer hardware and software, and the person hours devoted to CFD application, investments in improved RANS modeling methods are relatively small, and the return on this investment potentially significant. A collaborative approach to model development may improve the efficiency of these investments. Much of the early progress in turbulence modeling was achieved by individuals or small teams of model developers. To make progress beyond the achievements of these pioneers may require coordinated efforts by teams comprised of experts in turbulence modeling, experimentation, scale resolving simulation and CFD algorithms.

In this paper we define the industrial requirements, current capabilities and limitations, required improvements, and potential avenues for additional research on RANS turbulence models that can provide affordable, engineering capabilities to enable the CFD Vision 2030. For brevity, the discussion is primarily focused on modeling turbulence phenomena that impact the forces and moments generated by the fluid flow, with some discussion on heat transfer. There is also significant value for further investment to extend RANS modeling to other areas such as turbulent chemistry, transition modeling, low frequency unsteady flows and additional turbulent heat transfer topics. Such extensions are beyond the scope of this overview, but further activity in these areas is clearly warranted.

II.2. Industrial requirements

CFD has become an integral part of the design process throughout the aerospace industry. The application of CFD spans the full design cycle, from conceptual to detailed design. In conceptual design, a broad range of configurations must be evaluated for feasibility. In preliminary design, CFD is used to refine performance estimates, perform optimization studies, and enhance the value of physical tests by instrumentation design and data extrapolation. In detailed design, CFD is used to define the environment (e.g., pressures and temperatures) parts will be subjected to throughout the anticipated usage envelope and life of the parts. These applications include predictions in regions that may not be accessible to ground based testing, as well as validation during the product life cycle where CFD is used to troubleshoot shortfalls, evaluate manufacturing deficiencies, and extend the life and operating envelope. Each of these design stages require large numbers of simulations to define the steady or average environment within a severely time constrained design cycle.

Additionally, in many multi-disciplinary applications such as those involving control and trajectory prediction/mission analysis, state-of-the-art tools even consider the most rudimentary CFD analysis as too sophisticated and time consuming at the present time.

RANS models, while imperfect, have been shown to provide useful engineering results, or at least results that can be calibrated for a broad range of problems. In addition, in portions of the design envelope where RANS models give accurate performance trends, they can be extremely valuable even if the predictions of absolute performance levels are imperfect. Improvement of these predictions toward more reliability, providing reduced and quantifiable uncertainty, and extending the range of applicability to more diverse geometries and operating conditions is needed to further improve the design process and maximize the value of physical testing.

Improved RANS models delivering more accurate and rapid predictions with quantifiable accuracy can improve design by minimizing the margins that must be allowed for modeling errors in robust designs of transport aircraft, fighter aircraft, helicopters, turbomachinery, unmanned air systems, industrial flows and space applications. Another significant piece of the time to market for aerospace products is the certification process. Improvements in the accuracy and applicability of RANS results, coupled with advances in uncertainty quantification, have the potential to provide results with enough confidence to enable a simulation-based certification process with an attendant reduction in cycle time.

While problems of interest to engineering and design require primarily steady state data, there is a significant and growing need to quantify the unsteady pressures and temperatures to which structural components are subjected. Although unsteady forces of interest exist across the continuum, the unsteady frequencies of most interest to engineering design (e.g., flutter, vibratory frequencies of aircraft structures subject to acoustic loads, etc.) are low relative to the fluctuating frequencies associated with turbulence. As RANS models improve to more accurately predict the flow in a broader range of problems, models that provide reliable time accurate predictions would be a valuable asset to accurately model at least low frequency pressure content. The development of RANS models based on local flow conditions can provide insight for, and perhaps form the basis of, LES sub-grid models that capture large scale turbulent structures.

Many flow fields of industrial interest have a well-defined and unique time averaged solution. Even for the cases with such a well-defined temporal mean, RANS models in wide use can fail to predict this mean flow. For example, this is generally, but not always true in the case of separated flow fields. High speed cold-wall heat transfer, or mixing layers with either substantial temperature/density variation or compressibility effects are other cases where ad-hoc fixes are required to match the flow physics. The inability to accurately predict these flows is a recognized deficiency of the RANS models in wide use in industry. Although scale-resolved predictions have been shown to give better time average results than current RANS models in many low Reynolds number flow fields of interest, continued efforts to expand the affordable range of applicability and accuracy of RANS models is crucial to the further evolution and application of CFD for industry.

Inherent in the reliance on CFD models is the need to have confidence that the analyses correctly model the underlying physical phenomena. In order to extend RANS models to more accurately predict complex physical phenomena (e.g., adverse pressure gradients and separation, reattachment, flows that drive anisotropic turbulence, free shear layers, turbulent decay, shock boundary-layer interactions, free vortices, etc.) and extend applicability to a broader range of problems (e.g., flows with thermal interactions, turbomachinery cooling, hypersonic phenomena, chemically reactions including combustion, films, droplet breakup, icing, as well as high altitude and multiphase phenomena), there is an increased need for data on basic problems that illustrate and quantify the important physics. Fundamental tests and parallel analyses can be used to inspire new modeling approaches, and quantify the uncertainty of the resulting analyses.

In addition to fundamental testing, there is also a need for continued testing and advanced data collection methodologies for fluid dynamic systems in realistic environments at larger scale (e.g., high lift, transonic flows, fighter aircraft, helicopters, turbomachinery, etc.). The resulting data could be used to validate and calibrate new models and methods in situ. Evaluation of RANS shortfalls in complex systems can drive requirements and inspiration for further model enhancements and allow quantification of uncertainty and certification by analysis.

The aerospace industry needs affordable predictions with defined uncertainty of steady flows in an ever expanding range of conditions and geometric complexity.

III. Current Capabilities and Limitations

The application of CFD simulations in aircraft design is widely used and strongly promoted, envisioning even a simulation-based certification process. This long-term aim is challenged by unquantified uncertainty for flows of engineering interest. RANS models are widely used and provide critical design information particularly at conditions where the flows are relatively well behaved such as at nominal design conditions. Extending this success to the full range of conditions required for engineering design requires knowledge of the capabilities and limitations of the underlying approximations. This section summarizes the capabilities and limitations of RANS model accuracy for simple flow phenomena and more complex flowfields, and identifies the modeling practices that degrade predictive accuracy.

A. Simple flow phenomena

1) Large adverse pressure gradients and separation

RANS models are typically calibrated for the logarithmic law of the wall, covering roughly the innermost 10% to 20% of an attached boundary layer, and provide predictive accuracy for attached boundary layers. This is appropriate as long as the streamwise pressure gradient remains small, but for flows with large adverse pressure gradients that may eventually lead to separation, the predictive accuracy of the models usually degrades. Some improvement has been achieved, e.g., by Wilcox [3] by establishing a transport equation for the scale providing variable, and by Menter [4] by limiting the growth of the Reynolds shear stress. Nevertheless, deviations between predictions and experiments are regularly observed even for the flow around simple airfoils at high angle of attack.

As an example, figure 1, taken from Rumsey [5], shows the flow over an axisymmetric bump at Mach = 0.875. Following a supersonic acceleration, the flow separates on the aft section of the bump and reattaches on the cylindrical section. This flow includes strong favorable and adverse pressure gradients, and most popular turbulence models, including the Spalart-Allmaras [6] and Menter SST [4] models do not predict this flow particularly well. Figure 2 compares the experimental and predicted pressure coefficients, as well as velocity profiles for simulations of this configuration.

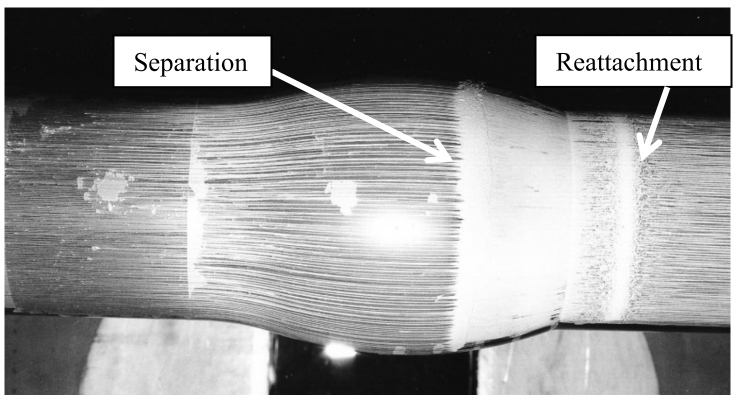

Figure 1 Oil flow for transonic bump experiment [5] at Mach = 0.875

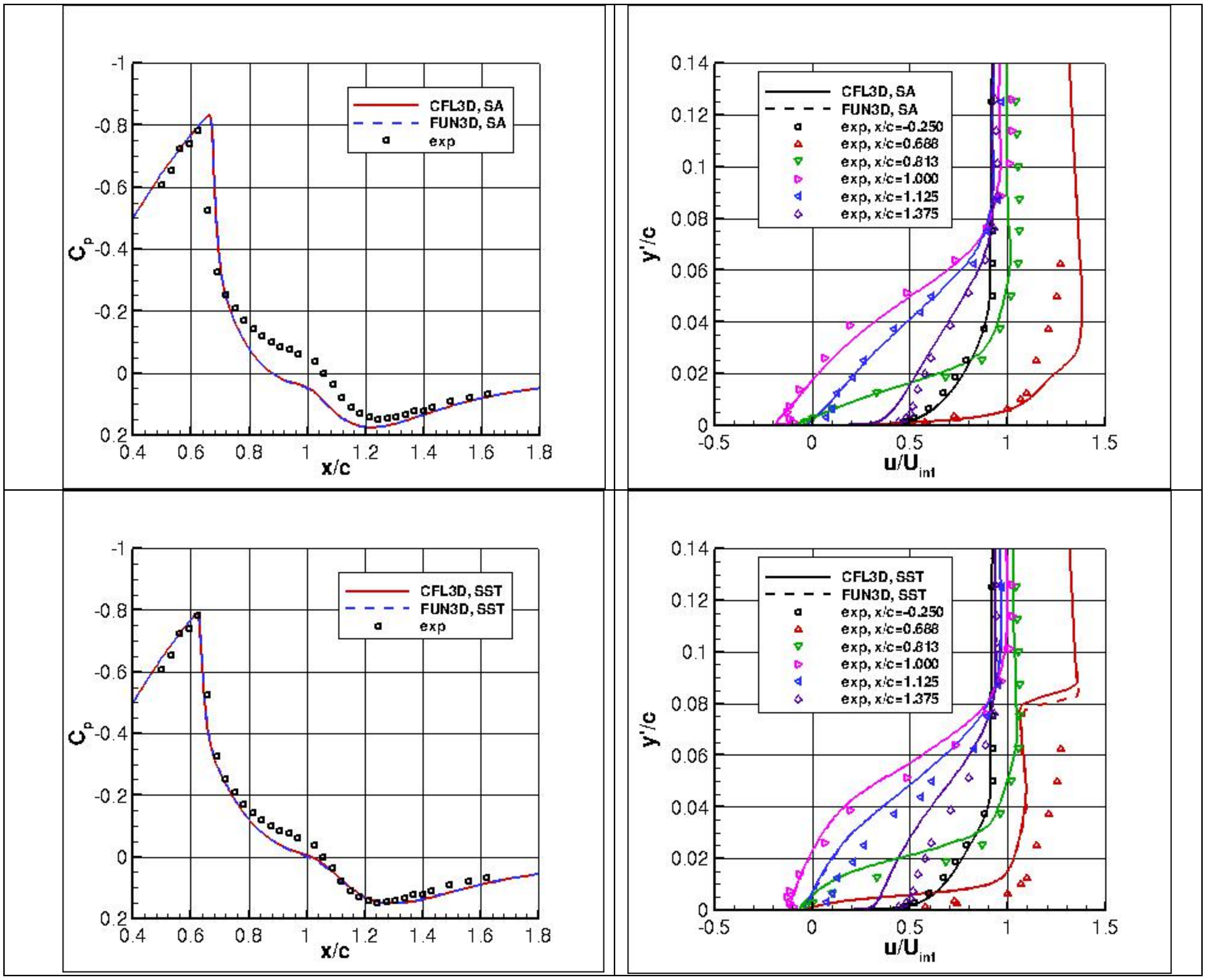

Figure 2 Pressure coefficient and velocity profiles [5] for the Spalart Allmaras (SA) and SST turbulence models

While both models appear to predict the velocity profiles near the separation location well (x/c = 0.813), neither appear to predict the flow near the point of peak acceleration (x/c = 0.688, or in the region beyond the reattachment point (x/c = 1.375). It should be noted that the SA model predicts the velocity profiles better than the SST model in the separated flow region, but the SST model gives a more accurate prediction of the pressure coefficient on the surface. These types of inconsistencies and inaccuracies are still too common for RANS predictions of separated flows.

Note that for flows with incipient or small separation, flow is still attached over the major part of the geometry and scale-resolving simulations requiring high spatial and temporal resolution over the full domain is at present not affordable in routine industrial applications. Zonal approaches may reduce these demands, but depend on a proper definition of the respective sub-domains.

2) Reattachment

Given the correctly predicted location of the separation point, it is the shape of the recirculation zone that defines the effective geometry of the body subjected to the flow. Therefore the location of the reattachment point also needs to be predicted correctly. However it is often observed that models that are sensitive to flow separation predict reattachment too far downstream. The reason seems to be a mismatch of the Reynolds shear stress in the developing shear layer above the recirculation zone (Jakirlic and Maduta [7]).

Figure 3 shows experimentally derived velocity contours for the flow over the NASA wall-mounted hump. Figure 4 compares skin friction and pressure coefficient predictions using the SA and SST models to experimental results. While the pressure coefficients predicted by the two models are inconsistent, the skin friction results clearly show that the reattachment point is predicted to be far downstream of the experimental location. Downstream of the reattachment point, the recovery in the velocity profiles is significantly retarded from the experimental measurements. While this experiment has been shown to have significant 3-D effects which influence the measurements, extensive CFD investigation of the 3-D configuration confirm that the model predictions of this flow field are representative for reattachment and recovery. Scale resolving simulations have been shown to be significantly more accurate for the prediction of reattachment and recovery for flow over this configuration.

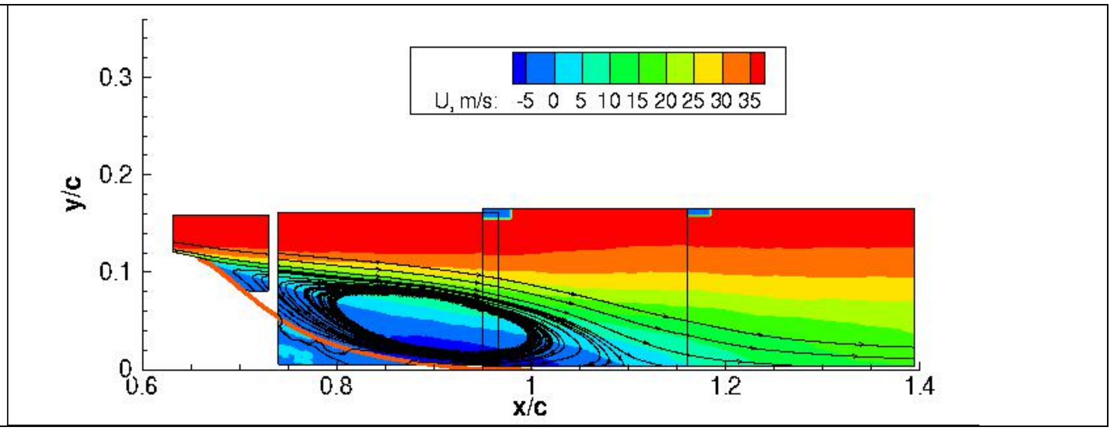

Figure 3 Experimental velocity contours for NASA wall mounted hump model [5]

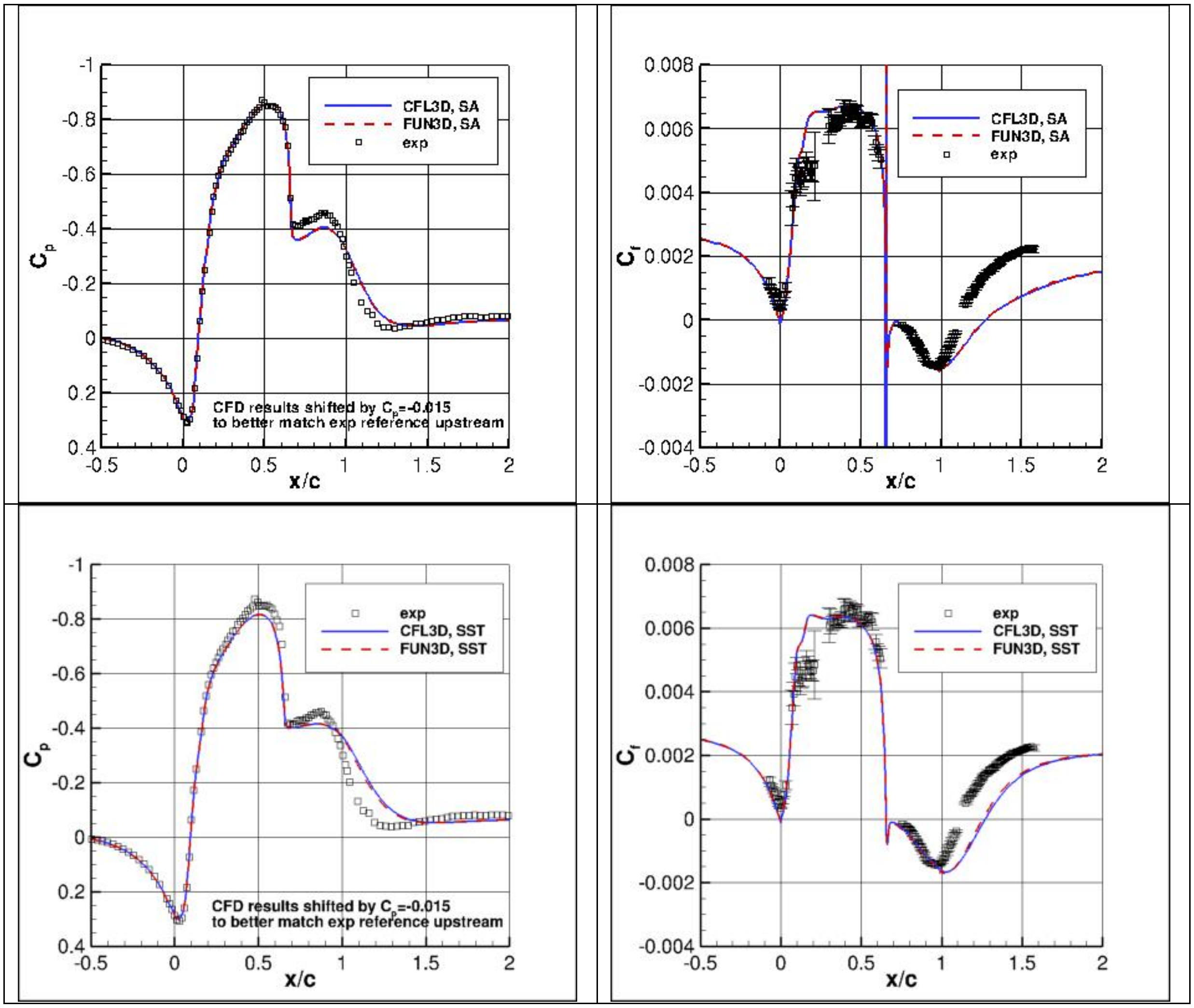

Figure 4 Comparison of SA, SST models and experimental data for pressure and skin friction coefficients [5] for NASA wall mounted hump model.

3) Normal stress anisotropy

Solid walls induce a directional effect on the turbulence structure, leading to differences in the Reynolds-normal stresses, referring to a wall-aligned coordinate system. This so-called normal stress anisotropy induces longitudinal vortices in ducts with rectangular cross-section and influences the separation of the flow along junctions subjected to an adverse pressure gradient. Linear eddy-viscosity models predict identical Reynolds-normal stresses near walls. Hence an accurate RANS-based prediction of such flows requires non-linear extensions, Explicit Algebraic Reynolds Stress, or full Reynolds-Stress Transport models.

Evaluation of turbulence models for the supersonic, Mach = 3.9 flow through a square duct highlights the influence of normal stress anisotropy on the mean flow field. Figure 5 (due to Rumsey [5]) shows velocity profiles and skin friction in a duct 50 heights from the duct entrance, a point where the flow is fully developed. The baseline SA and SST models greatly over-predict the differences between a velocity profile from the corner of the duct and from the duct center of the side to the duct center. The baseline models predict the skin friction near the corner to be too low compared with test data. Thus these widely used models cannot accurately predict the influence of corner vortices, which occur in many practical situations, on the mean velocity. Models that account for normal stress anisotropy, such as the QCR model extension of Spalart [8] can capture these key effects in corner flows significantly better.

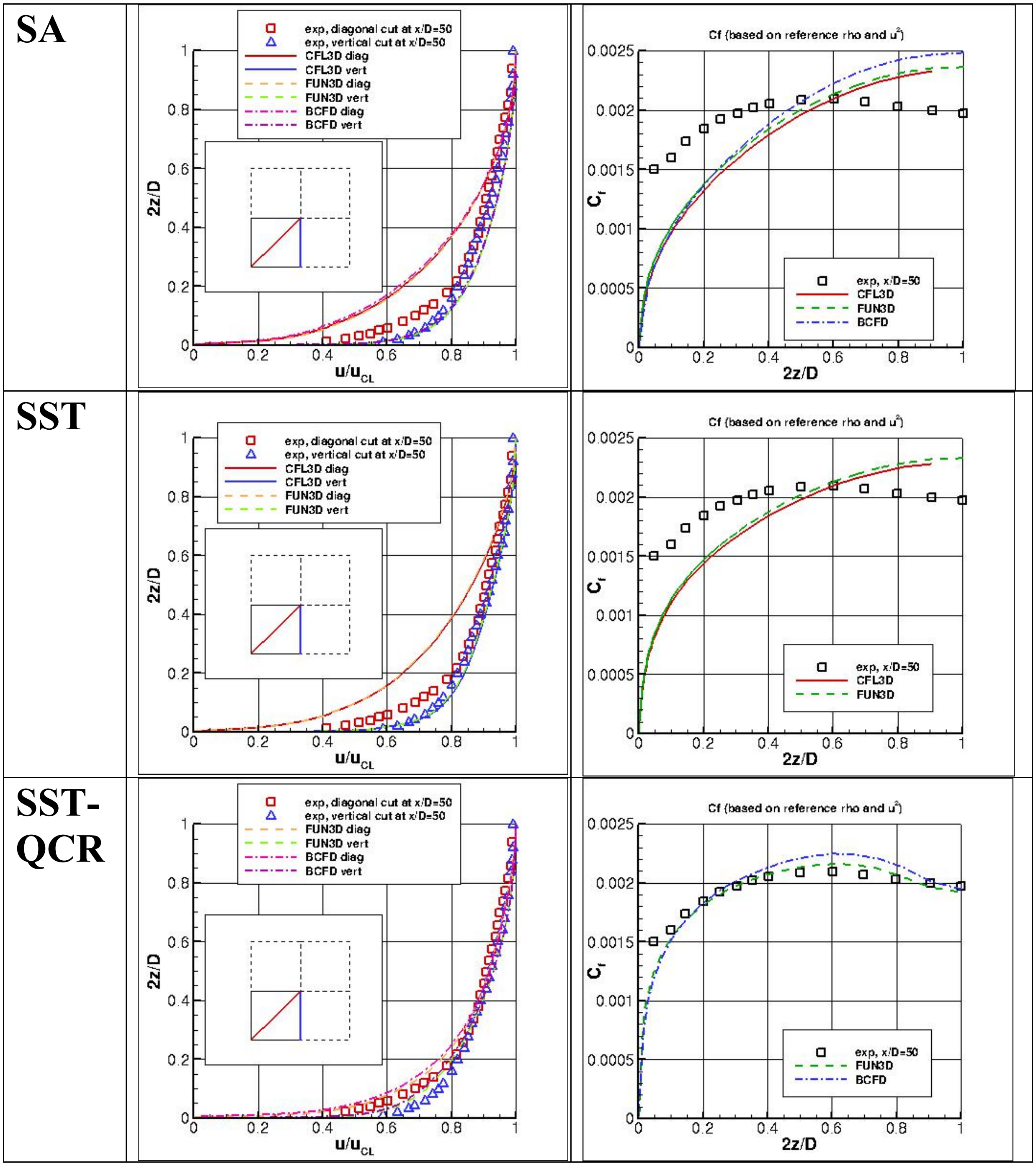

Figure 5 Comparison of SA, SST and SA-QCR models and experimental velocity profiles and skin friction [5] 50 duct half heights from entrance of Mach 3.9 flow through a square duct.

The Reynolds-normal stress anisotropy predicted by most Reynolds-stress models follows theory that may be inaccurate for the Reynolds numbers on current civil transport wings, and more so for the Reynolds numbers of the flows on the fuselage which feed the wing/body junction. For instance, the axial Reynolds-normal stress has been found to grow with increasing Reynolds number, and attains values inconsistent with relations built into most current Reynolds-stress models (Marusic, Uddin, and Perry [9]).

4) Free shear layers

The problem of reattachment behind a backward-facing step is related to the mixing-layer flow with one stream at rest. Indeed, as documented on the Turbulence Modeling Resource (TMR) web-site (Rumsey [5]), most models underestimate the experimental Reynolds-shear stress far downstream of the origin.

The spreading of wakes is important for the interaction between rotors and stators in turbomachinery flows. It is observed that RANS-based CFD simulations underestimate the respective spreading rates.

Similarly, Wilcox [10] reports on the so-called "round jet/plane jet anomaly," stating that models tend to predict a higher spreading rate for the round jet, whereas in experiments the opposite trend is observed. Wilcox [10] refers to a remedy that nevertheless seems not to be included in many models.

RANS models for mixing layer flows with density and/or temperature differences between the two streams also tend to consistently misrepresent the level of mixing. This simple flowfield is at the heart of modeling combustion.

Thus, even simple canonical free shear flows demonstrate RANS-modeling deficits that might be ignored only when restricting oneself to simple boundary-layer flows.

5) Turbulent decay

It is well-established that the turbulent kinetic energy decays in a parallel flow without velocity gradients. Nevertheless, in turbomachinery flows it is observed that the inlet conditions required for eddy-viscosity based RANS models to correctly predict the turbulent decay lead to excessive eddy-viscosity levels, ultimately causing unrealistic losses in the blade passage flow. This highlights another shortcoming of popular RANS models: the Boussinesq eddy viscosity is treated almost interchangeably with the turbulence intensity rather than serving as a transfer function between the principal Reynolds stress and the strain rate tensor. Models typically expect free-stream eddy-viscosity levels to be low, while by strict definition, they should be very large.

6) Shock/boundary-layer interaction

A shock creates an (almost) infinite adverse pressure gradient, potentially inducing separation. Even if its position is fixed, e.g., by a shock generating edge in supersonic flow, the developing separation bubble in terms of wall-shear stress and heat flux is predicted inaccurately by RANS-based CFD simulations. Comparisons with experimental data for different test cases show large deviations of pressure, skin friction and heat flux near the shock location (e.g., Marvin et al. [11]). Figure 6 demonstrates the current accuracy limitations of two standard models (SA and SST) for both the pressure rise and heat transfer profiles in the vicinity of the shock, even though the approach boundary layer is well predicted for this case.

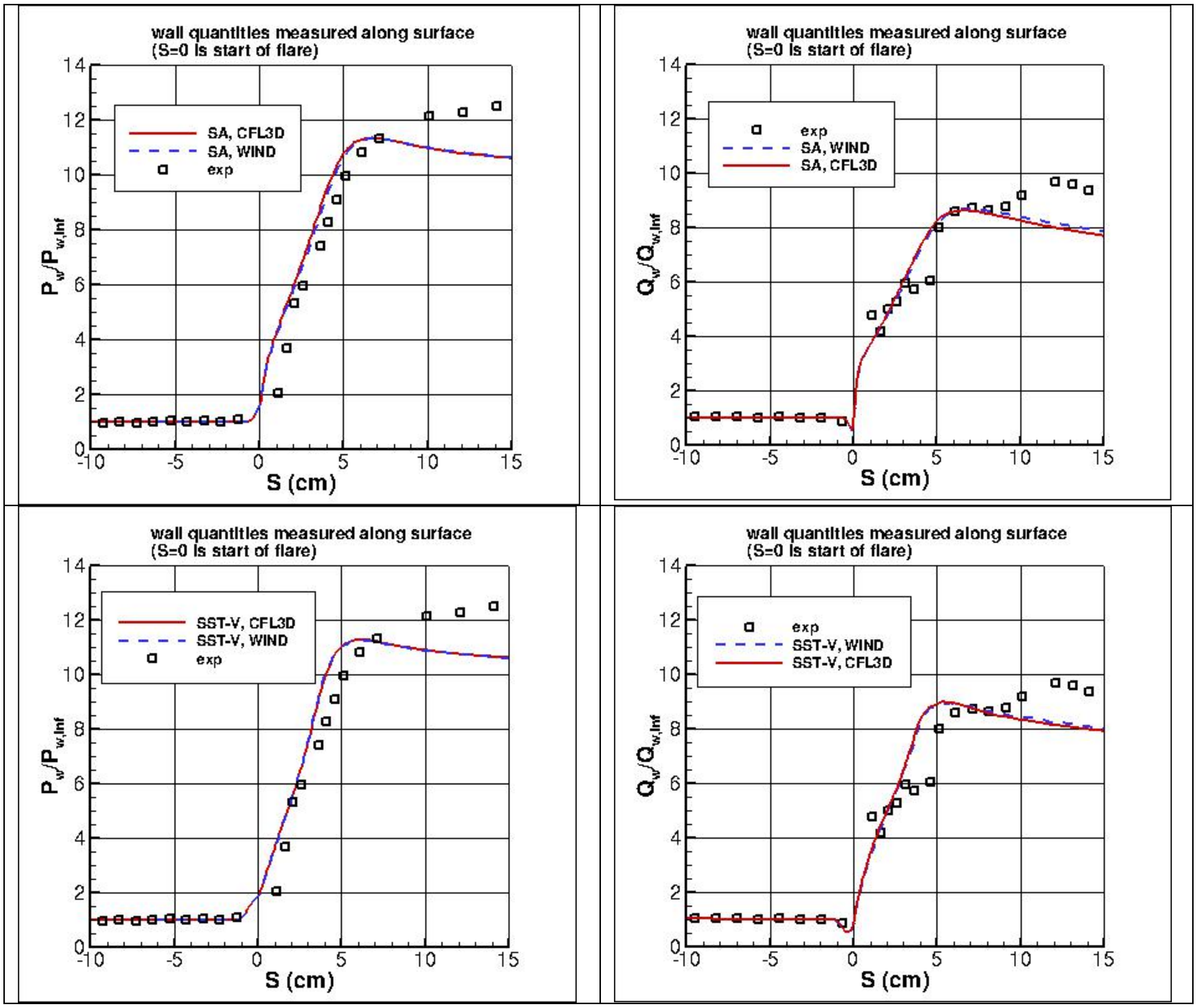

Figure 6 Comparison of SA, SST models and experimental data for pressure and heat transfer coefficients [5] for an Axisymmetric Shock Wave Boundary Layer Interaction near M=7.

7) Free vortices

Eddy-viscosity based models predict a too rapid dissipation of free vortices, which is confirmed by comparison of simulations with experimental data. Shur et al. [12] have devised a rotational correction, which can be combined with various models, but nevertheless aims at compensating for a fundamental deficit.

In contrast, Reynolds Shear Stress Transport models are supposed to predict free vortices in better agreement with experimental observation, based on the argument of an exact production term (and exact advection, (Spalart, private communication, 2017)). Indeed, with these models free vortices are usually maintained for a longer distance. Nevertheless the agreement with experimental data is certainly not perfect, which can be attributed to the additional terms in the transport equations. These terms, like the pressure-strain correlation, are usually not modeled with respect to the physics of vortex flows and therefore supposedly carry modeling deficits.

B. Complex flows

The modeling deficits observed with simple flows enter also into more complex flow situations, where different phenomena occur simultaneously and may interact with one another.

1) Transport aircraft

High-lift flow - High-lift flows on transport aircraft are characterized by significant geometric complexity of the high-lift system. The generated high lift is associated with high suction peaks followed by strong adverse pressure gradients that may lead to separation. Additionally, the wakes emanating from the upstream wing elements spread above the boundary layers of the downstream wing elements, potentially leading to interference effects. Corner flow phenomena are present at the wing-body junction and at the intersections between the wing and the flap-track fairings. Vortices are generated at the tips of the high-lift devices, by engine strakes and potentially in the corners of the flap-track fairings that all travel downstream and interact with the boundary layers. At take-off conditions, additionally the interactions with the engine jet need to be accounted for.

While for take-off conditions systematic investigations of simulation capabilities seem to be still missing, for landing conditions it is obvious that the accuracy of RANS-based CFD methods for predicting the maximum lift coefficient and the associated incidence is generally insufficient. Even below maximum lift, the slope of the lift-curve often does not match the experimental observation.

The reasons for failure are manifold and only partly due to the turbulence model; other factors include geometrical simplifications, insufficient grid resolution, inaccurate numerical modeling, unmodelled transitional flow and lack of convergence. Replacing RANS by scale-resolving methods therefore addresses only part of the problem, while introducing additional requirements with respect to spatial and temporal resolution.

Transonic flow - Modern transport aircraft operate at transonic cruise conditions. At design conditions, characterized by moderate shocks and attached flow, RANS-based CFD predictions are considered reliable. In contrast, at off-design conditions the accuracy of numerical simulations degrades.

Increasing the Mach number or the angle of incidence leads to stronger shocks, finally inducing separation. The developing separation bubble starts interacting with the shock, at some point moving its position upstream. The interplay between shock position and separation is important for predicting buffet onset, where the shock position starts oscillating. It requires the proper prediction of the boundary layer development upstream of the shock and accurate representation of the downstream separation and reattachment. Similar to the observations in high-lift flows, the accuracy of RANS-based CFD simulations is insufficient for these phenomena.

2) Fighter aircraft

The flow around fighter aircraft is characterized by strong vortices generated at the leading edge of highly-swept wings (Schütte, [13]). With increasing angle of attack these vortices are subject to an adverse pressure gradient, finally leading to vortex break-down associated with highly unsteady flow.

While RANS-based CFD is able to predict the position where a vortex is generated at a sharp leading edge, its accuracy degrades when the leading edge is not sharp. Furthermore, a lack of accuracy is observed concerning the prediction of vortex breakdown, vortex-vortex interaction and vortex-shock interaction in terms of position and associated flow conditions, probably requiring scale-resolving simulations (Morton and Cummings [14], Luckring et al [15]).

3) Helicopters

Helicopter flows are by definition unsteady and characterized by large helical vortices generated at the rotor tips. Depending on the flow conditions, interactions of this vortex structure with the following blade may occur. The prediction of such blade-vortex interactions therefore requires the vortices to be preserved for a sufficiently long distance, which is challenging for both the numerical method as well as the turbulence model related dissipation. Current strategies rely on reducing the dissipation by employing higher-order numerical schemes on Cartesian off-body grids, solving the Euler equations (Potsdam et al. [16]), i.e., ignoring turbulence completely, or employing scale-resolving methods, e.g., Potsdam et al. [17].

During its rotation around the vertical axis, a rotor blade section experiences various flow conditions. In particular, the blade is undergoing a pitch oscillation around its spanwise axis, giving rise to dynamically growing separation zones that periodically leave the blade as vortices parallel to the span. This so-called dynamic stall is important with respect to the blade forces and cannot currently be predicted with sufficient accuracy. Scale-resolving methods offer potential improvement, but need to cope with highly energetic reverse flow occurring temporarily close to the wall.

4) Turbomachinery

Some turbomachinery flows, e.g., in low pressure turbines, are characterized by low chord Reynolds numbers, therefore requiring an accurate prediction of transition due to different mechanisms. Furthermore, the unsteady interactions between the flow through rotors and stators take place at frequencies that overlap with the frequencies of turbulent fluctuations. Finally, the CFD-based design of a jet engine additionally requires predicting the interactions between its different components, i.e., compressor, combustion chamber and turbine (Tucker [18]).

The blockage by separated flows and the blade tip vortices determine the surge limit of compressors. As far as can be evaluated from limited experimental data, separation (in particular corner flow) due to an adverse pressure gradient is insufficiently predicted by RANS-based CFD.

Cavities are often present in turbomachines where the local Mach number is significantly lower than in the main flow field. Difficulties are observed predicting the interaction between the flow in the cavities and the main flow by RANS-based CFD.

Turbine blades often require cooling, either by internal cooling channels or by blowing of cool air into the flow field. RANS-based CFD simulations show limited accuracy for the prediction of the turbulent heat transfer. Also cooling jets in cross-flow are not well predicted.

C. Limitations of assessment

The CFD assessment process is often expressed in terms of Verification, Validation and Uncertainty Quantification as outlined in AIAA Standards Document (AIAA [19] and Lee et al. [20]). The assessment of numerical simulation accuracy is based on comparisons between simulation results and experimental data. Meaningful statements therefore require the numerical error to be significantly lower than the model error and must take into account the experimental uncertainties. The difficulties in meeting these requirements, in particular with complex configurations, are not restricted to RANS-based simulations, and are compounded for scale-resolving methods as the grid spacing is typically directly a part of the sub grid models and numerical and modeling errors are of the same order.

1) Numerical limitations

Geometric simplifications - Agreement between numerical simulation results and experimental data can be expected only if identical geometries are considered at identical flow conditions. Nevertheless, validation computations are often carried out under geometrical simplifications.

The flow around nominally two-dimensional configurations is often computed as truly planar, ignoring the effect of lateral wind-tunnel walls, like corner flow at high incidence. Details about wind-tunnel devices like boundary layer fences, guiding vanes or suction panels that are introduced for obtaining a two-dimensional flow near the center of the wind-tunnel are often unknown and generally ignored in this type of simulation. Moreover, numerical simulations often assume free-flight conditions with large far-field distances, whereas the corresponding experiment has been carried out in a wind-tunnel confined by upper and lower walls, where the model causes blockage effects. The effects associated with wind-tunnel walls typically occur in situations where the flow conditions are supposed to be challenging to predict anyway, e.g., close to maximum lift, hence limiting the reliability of statements about model performance.

Similarly, the flow around three-dimensional aircraft configurations is often simulated for only half of the domain, assuming symmetry on the center plane, e.g., Tinoco et al. [21]. Potentially asymmetric stall in the experiment can therefore not be reproduced. In vortex dominated flow fields the symmetry assumption suppresses potential nonsymmetric interactions between the vortices on both wings, particularly at high incidence. Even if experimental data have been obtained on a half-model in the wind tunnel, assuming symmetry on the center plane might be a simplification, since half-models might have been mounted on a peniche or on a support strut with a boundary layer fence, like the ONERA M6 wing (Schmitt and Charpin [22].

In turbomachinery flows, geometric details like seals, cavities or holes for cooling air are often ignored. Furthermore, manufacturing tolerances that might be important on geometrically smaller parts are not accounted for. In the case of a hot geometry, the size of the rotor blade tip gap is often only estimated.

Geometrical uncertainties also occur when comparing to experimental data obtained in pressurized wind-tunnels because the model deforms under high loads. Even if the deformation is measured at certain locations, the corresponding geometry needs to be reconstructed, introducing additional geometric uncertainties into the simulation. Therefore any statement on turbulence model performance based on comparison to experimental data that have been obtained under high loads must be taken with great caution.

Finally, there might be small geometric details that are ignored for reasons of grid resolution, e.g., small gaps and steps on the surface, radii at intersections, etc. Nevertheless, at critical flow conditions these small-scale details may trigger large-scale effects that cannot be captured if the geometry is simplified in the simulation. As an example, Rudnik and Melber-Wilkending [23] have investigated the influence of neglecting pressure tube bundles that have been present in a high-lift experiment.

In industrial design studies such geometrical differences might be acceptable because the major interest is in increments. In contrast, they significantly increase the uncertainty in the validation of turbulence models. For this purpose, the geometry in the simulation needs to be as close to the experimental one as possible.

Grid and time resolution - In principle, the numerical error of any simulation can be reduced by systematic grid refinement, until the quantity of interest is grid independent to within a given threshold. In practice, such studies are typically limited to simple two-dimensional simulations. In complex three-dimensional simulations, such as those over a full aircraft, true grid independence is difficult to reach, as extraordinary computational resources are required to handle billions of cells.

Furthermore, complex geometries are attractive for use with unstructured grids, but these often suffer from cell degradation in corners, in regions where supposedly important flow phenomena need to be resolved. Local mesh adaptation may be used for increasing the resolution, but the results might still depend on the initial grid quality and the refinement strategy. Accordingly, for unsteady problems the physical time step must be sufficiently fine, in order not to introduce additional numerical errors.

Convergence - In principle, numerical simulations should be converged to machine-accuracy, in order to be sure that the numerical error of the discrete solution is close to zero. In practice, this typically is possible only for simple flows, like attached boundary layers on smooth high-quality grids. In contrast, in three-dimensional complex flows, even involving only mild separation, convergence to steady state can be difficult to achieve, due to limitations in the numerical flow solver algorithm, turbulence model instabilities, or because there exists only an unsteady (mean flow) solution. The definition of convergence can also be confounded by non-unique numerical solutions that may arise due to minor changes in the starting grid, initial boundary conditions, non-linear solution path, hysteresis, or multiple physical solutions (e.g., stall cells for high lift configurations).

CFD practitioners often have to accept residual convergence of only few orders of magnitude, averaging pressures or forces over a large number of iterations. In contrast, statements on model performance based on this procedure are questionable, since results obviously involve considerable, but unknown and probably different numerical errors, depending on the respective model and flow solver. Note that the convergence requirement is even more demanding in unsteady simulations, because the result of each physical time step serves as initial condition for the next one, requiring the numerical error of the solution not to accumulate in time (Coder [24]).

Boundary conditions - The solution of the underlying differential equations requires specification of boundary conditions. In particular, the inlet conditions for the turbulence are often associated with high uncertainty. In external aerodynamics, where the far field is at large distance from the region of interest, the influence of the far-field conditions is considered small compared to the influence of the walls; however, improper choice of free-stream turbulence conditions can have a pronounced impact on the solution (Spalart and Rumsey [25]). Additionally, in turbomachinery and other internal flows, the inlet is usually close to the region of interest, leading to a significant influence of the inlet conditions for the turbulence quantities on the numerical solution.

Mode of simulation - Despite being unsteady in character, turbomachinery flows are often simulated as steady, modeling the interaction between blade rows in terms of a so-called mixing plane. This introduces an additional uncertainty, independent of the assumed RANS-model.

2) Experimental limitations

Agreement of numerical simulations with experimental data can only be expected to be within the experimental uncertainty. Unfortunately, error bars are missing in many published data sets, making it difficult to assess whether observed differences between different turbulence models are meaningful or not.

Such experimental uncertainty is present even in well-known canonical flows like the zero-pressure gradient boundary layer, where the local skin-friction coefficient c_f can probably be measured only to within an accuracy of 2-5% (Duraisamy et al. [26]). Similarly, the peak Reynolds stresses in a plane mixing layer deviate by roughly 10% between different experiments considered for reference (cf. the test case data suggested by Benocci et al. [27] for validation of LES).

Experiments on separating flows over nominally two-dimensional geometries show three-dimensional span-wise patterns (e.g., Schewe [28]). Comparing to experimental data that are provided in one section only is therefore somewhat arbitrary. Furthermore, even for relatively simple nominal geometries, there is often only limited parametric data available with detailed turbulence characterization.

Experiments at high aerodynamic loads may lead to deformations that need to be accurately known for proper comparison between numerical simulations and experimental data. Furthermore, the geometry might vibrate due to vortex-shedding, shock-buffet, etc. Comparison with time-averaged experimental data, ignoring the motion of the configuration, is therefore questionable. Note that data on aerodynamic configurations might be obtained experimentally by continuously increasing the incidence, assuming the motion is slow enough to correspond to steady state conditions, as well as raising the question of flow hysteresis. Experimental data in turbomachines, in particular its rotating parts, are even more difficult to obtain. Due to the small geometries, the size of measurement probes is comparatively large, increasing the associated disturbance of the flow field. Optical access requires components manufactured from transparent material and a liquid with adapted refractive index (Uzol et al. [29]). Hence, only limited experimental data are available about the details of the three-dimensional flow within turbomachines, e.g., radial profiles downstream of a blade row.

IV. Required Improvements / Value Proposition

We give a modeling improvement value if it leads to a nontrivial enhancement in flow prediction capability in terms of accuracy and/or robustness, relieving the current limitations outlined in Section 3, for problems faced by a significant portion of current and next generation aerospace engineers as outlined in Section 2. Value is also attributed to activities that facilitate these modeling improvements. A nontrivial enhancement is considered to be one that, at a minimum, advances a capability from unreliable to a reliable workhorse. Many engineers require these enhancements to support the design of future air and space vehicles that address dominant commercial and military needs.

Improvements to turbulence models that can significantly improve the value of CFD for research and engineering applications can be found in extending one and two equation transport models, re-visiting Reynolds Stress Transport (RST) models, leveraging RANS for hybrid LES, and development of a benchmark turbulence database for experimental and fully resolved CFD datasets.

With respect to low speed, nearly isothermal, ideal gas problems, in which all turbulent scales are modeled, we may have reached the point of diminished returns on investment in the development of RANS models characterized by up to two transport equations. The most recent additions to the model toolset that have stood the test of time and are in wide use are the Spalart Allmaras one equation model that first appeared in 1992 (Spalart and Allmaras [6]), and the k-ω SST model (Menter [4]). Between then and now, there have been a number of model corrections and enhancements that extend the applicability of these models to more complex physics, such as compressibility (Spalart [8]), anisotropic turbulence (Mani et al. [30]), and rotation and curvature (Shur et al. [31]). These enhancements address many of the limitations outlined in Section 3.1 and have improved predictions for more general problems. Further extensions that maintain the cost effectiveness of low-cost RANS models while providing improved reliability, robustness and accuracy for more complex physics (e.g., chemistry, combustion, two phase flow, turbomachinery flows, porous surface modeling, compressibility effects, propulsion integration) are still needed to address the complex flow limitations outlined in Section 3.2.

Further investments in the development of new modeling approaches that have the potential to increase the accuracy and extend the range of application beyond the one and two equation models are also needed. One promising approach is the use of Full Reynolds Stress Transport (RST) models (e.g Eisfeld et al. [32] and Cecora et al. [33]). Early thrusts into RST model development withered in the 1990's due to their computational complexity relative to delivered prediction improvement. There were also numerical robustness issues associated with the solution of RST's seven equations. The robustness shortcomings have to some extent been improved by advances in numerical methods and the significant growth in computational resources over the past twenty years. These advancements also facilitated a renewed thrust in this area because they have made higher Reynolds number, complex scale-resolved simulations viable. The detailed data from scale-resolved cases that are more closely aligned with practical flows can be used to provide insights into improved closure expressions and coefficients with the expectation that they will lead to prediction improvements that justify their cost.

Both two equation and RST models typically rely on a single length scale equation. Model turbulent diffusion, turbulent viscosities and turbulent dissipation are assumed to be proportional to this single, scale. This assumption may be a significant limiting factor in RANS model accuracy, and could be investigated more thoroughly using DNS and LES simulations.

A new direction for RANS model development was popularized with the hybrid RANS/LES proposition (termed detached eddy simulation or DES) by Spalart et al. [34] to decompose the computational domain into regions where all turbulent scales are modeled (RANS) and regions where turbulent scales accurately supported by the local grid resolution are time accurately simulated (Large Eddy Simulation - LES). This decomposition introduces additional modeling issues that have been the focus of many research efforts but can still benefit from future investments. This includes RANS to LES region blending functions, LES to RANS region signal reflection mitigation and the anisotropic/reflectionless population of resolved scales as the flow traverses into an LES region. Another issue is the high level of dependence of simulation accuracy on grid resolution, with cell aspect ratio and distribution and cell size distribution having a much larger impact on simulation accuracy than is manifest in a steady-state RANS simulation. Significant returns are possible from further investments in RANS modeling as advances will inform the modeling of the high frequency unsteadiness that is below the LES resolution relevant for the low frequency unsteadiness. RANS model extensions could form the basis of new sub-grid models that provide a hybrid RANS/LES (HRL) or Wall Modeled LES (WMLES) capability that consistently scales with grid and time resolution to provide accurate and cost effective results across a range of frequencies.

An activity that will facilitate these RANS model developments is the creation of a standardized benchmark case data repository populated with experimental and DNS data at relevant Reynolds numbers. This is not a new idea, but the usefulness of a repository can be improved over those currently available. The objective is to maintain accessible and well-defined data for benchmark problems designed specifically to support research efforts on RANS modeling, including the application of data driven modeling techniques to improve the performance of existing models. The database should define a hierarchy of benchmark problems feasible for wall-resolved LES (WRLES) or experimentation, and rank them in terms of value to the RANS modelers. The cataloguing of experimental data will necessitate a reexamination of test and measurement standards from older data sets. In keeping with the premise that experimental validation data are as dated as the measurement tools brought to bear on the flow problem, candidates may benefit from renewed investigation with modern instrumentation capabilities and acquired understanding of CFD validation requirements. Parametric databases facilitating a variation of boundary conditions are particularly important for current modeling tools being researched.

To facilitate the accessibility of simulation data, a standard set of flow statistical data useful to RANS model developers should be stored in addition to raw simulation output. This includes auto/cross spectra and correlations accompanied by error bars due to finite time samples, spatial resolution or measurement uncertainty. Ideally, the utilities used to calculate this data (source code, Matlab scripts, etc.) would also be stored on the repository. If this is not possible, then enough information to allow a researcher to reproduce the processed data from the raw data (spectra/correlation definitions, windowing method, sample sizes and overlaps, spectral averaging, filtering, etc.) should be stored to remove any uncertainty in the interpretation of the data.

This is all for naught without a precise definition of the benchmark problem. CAD files characterizing the geometry, fluid properties, initial conditions or proof of stationarity and a complete characterization of mass flow boundaries (including mean flow, spectra and correlations) are among the items that need to be provided so that the simulation or experimental data can be understood properly. The simulation data should be accompanied by the software and input files used to generate it so that researchers can perform supporting simulations if necessary. Submissions to the repository should be reviewed with the same or greater rigor than is used by refereed journals to certify the archival value of the data.

V. Suggestions for future research topics

This document has provided an overview of the industrial requirements for turbulence modeling, and has detailed the many current shortcomings of RANS methods. Some thoughts regarding possible avenues for pursuing improvements and suggestions for future research topics are outlined.

Problem Specific Models: There may never be a universally accurate model for the range of engineering problems of interest. There should be focused efforts to develop better models for specific phenomena such as wall-bounded flows, free-shear flows, etc. Developing intelligent methods of identifying and blending different regions can broaden the applicability of the focused models for engineering design.

Complex Physical Models: Developing or extending RANS models targeted for more complex physical phenomena (e.g., chemistry, combustion, two phase flow, turbomachinery flows, porous surface modeling, compressibility effects, propulsion integration) can lead to more practical and efficient engineering tools for problems that we currently have no ability to predict except with scale-resolved simulations that are un-affordable for routine engineering studies. Relatively small research activities targeted at modeling specific phenomena can illuminate previously unappreciated physical phenomena that can be exploited for design.

Reynolds Stress Transport (RST) models: RST models have not been demonstrated to be more accurate than simpler models on a consistent basis, perhaps due to computational limitations that would be less constraining today. The community has recently made some effort to systematically evaluate RSTs. However, since RST models (although complex) are based on a somewhat firmer theoretical foundation than simpler models, coordinated research is justified and represents a fruitful area for the application of data-driven turbulence modeling.

Data-augmented modeling: Turbulence modeling has always been data-driven, but using rudimentary techniques such as single or two variable curve-fitting. Recent developments using statistical inference and machine learning have shown that data can be more formally and comprehensively used to improve models. A productive area of research will be to establish that formal data-driven approaches can result in improvements that are generalizable. A comprehensive review of data-driven approaches and uncertainty quantification as applied to turbulence modeling is presented in a recent article (Duraisamy et al. [26]). Our view is that purely data-driven methods will not replace a turbulence model nor can it circumvent theory as the complexity of the problem would place unattainable demands on machine learning algorithms. Incorporation of known physical constraints on the data-driven models will greatly reduce the demands on data and promote generalization. Data-driven modeling should thus be incorporated into existing modeling development frameworks and should evolve symbiotically.

Uncertainty Quantification: Uncertainties in turbulence modeling [26] arise due to the inherent loss of information in the Reynolds averaged representation, operating assumptions (single point/length-scale, etc.), model structure (eddy viscosity), and model parameters. These aspects should be addressed cohesively. In an industrial design setting, reliable uncertainty bounds may actually be more useful than an improved prediction. Having a reasonable estimate of uncertainty will guide the decision making of where to apply additional resources to improve the simulation accuracy. Additionally, minimizing the uncertainty of the turbulence model coefficients [35] or functional forms [36] using high-fidelity simulations and experimental data can lead to improved turbulence models. Certification by analysis will require a much more accurate assessment of the uncertainty due to modeling errors than is currently performed. Extrapolation of model error assessment from well-defined test data on simple or simplified configurations to complex configurations of interest is a major challenge.

Hybrid RANS-LES / Near-wall models (WMLES): Unsteady and/or highly 3D separation may be out of reach of RANS. Also, if one is interested in the fluctuating quantities, Hybrid RANS-LES and wall-modeled LES may be necessary. RANS methodology and approximations can inform LES sub-grid models. This area also includes the need for RANS to LES region blending functions, LES to RANS region signal reflection mitigation and the anisotropic/reflectionless population of resolved scales as the flow traverses into an LES region. Extensions of the RANS methodology and approximations can also provide an avenue to improve the accuracy of Lattice Boltzman and Cartesian Immersed Boundary methods that rely on wall modeling to avoid isotropic resolution to inner layer length scales.

Turbulence Database: Expand experimental and numerical (DNS) databases of turbulence measurements for unit problems to inform turbulence model development. High quality data that includes measurement of turbulent scales and parameters is needed to drive improvements in RANS as well as scale resolving methods. Measurements should be focused on unit problems that illuminate specific phenomena that turbulence models must represent in order to accurately represent more complex flows.

Design of Experiments/High-fidelity simulations: Even though many experiments/simulations have been designed with modeling in mind, they have typically tended towards model validation and not model improvement. To improve models, one has to figure out what exactly is needed to address the specific discrepancy in the model.

Typically, experiments are designed to give the appropriate physics. The focus of high fidelity simulations should be to characterize the modeling discrepancies for a given set of boundary conditions (e.g., Disotell and Rumsey [37]). The field of experimental design (design of experiments) has much to offer in this regard.

Error estimation and control for complex configurations: As computing costs are becoming less of a concern, improvements in solver and grid resolution are available to reduce errors to engineering levels. Discretization error must be quantified and minimized for complex configurations, at least to a level smaller than modeling errors. Turbulence modeling errors (whether RANS or scale resolving) must be identified and quantified to the same level as numerical errors in order to fully characterize solution accuracy. Research to estimate and quantify errors for complex configurations can yield techniques to inform the designers and managers, leading to higher confidence in analyses and reduced design conservatism.

VI. Recommendations

One of the driving forces that inspired the writing of this document was the roadmap laid out in the CFD Vision 2030 document [1], suggesting that a decision to abandon RANS research needs to be made in the current timeframe (around 2020). We feel that such an abandonment would be premature. We have outlined the industry requirements for improved predictions of turbulent flows and the cost-barrier that is associated with reliance on scale resolving methods. Suggested research topics have been identified that have the potential to improve the applicability and accuracy of RANS models. Naturally, with increases in computer capacity and speed, and improvements in modeling, scale-resolving simulations will continue to make inroads in the aerospace industry and research should continue, or even accelerate, in this area. But we feel it important that some part of a balanced research portfolio should still include RANS efforts.

We end with a few recommendations on how we feel future efforts to improve RANS modeling and simulation should be structured.

Community Collaboration: With anticipated funding constraints, we feel it will be important to encourage increased community collaboration in order to address current shortfalls and improve the ability of CFD methods to meet current and future industry requirements.

Research / funding structure: Traditionally, turbulence modeling research has not been a team effort. Instead, much of the research has been funded and performed by individuals. If model improvement is the goal, turbulence modeling research should instead be comprised of an ecosystem (experimentalists, DNS/LES experts, CFD algorithm experts, turbulence modelers, UQ people, statisticians, etc.) who are working in a coordinated fashion.

Turbulence Database Management: A turbulence benchmark repository should be managed by leaders in the field from around the world with input from the community at large. This group of leaders needs to include experts in turbulent flow prediction, RANS modelers, DNS and LES experts and theoreticians, and experimentalists with expertise in collecting high fidelity flow statistics for complex flow fields. One outcome of this effort will be better coordination between experimentalists/DNS/LES and RANS modelers.

CFD Vision 2030 : New paradigms/needs for turbulence model improvement should be incorporated into Vision 2030 (for instance, data-driven modeling was not widely known when the original document was written).

Acknowledgments

The authors would like to thank the members of the AIAA Turbulence Model Discussion Group all of whom made significant contributions to the scope, tenor and in many cases text of this document as well as providing valuable feedback. Other significant contributors include Rob Kunz (Penn State), Klaus Becker (Airbus), Olaf Brodersen (DLR), Jan Willem van der Burg (Airbus),Simone Crippa (Airbus), Anthony Gardner (DLR), Christian Morsbach (DLR), Henning Rosemann (DLR), Ralf Rudnik (DLR), Andreas Schütte (DLR).

References

[1] Slotnick, J., Khodadoust, A., Alonso, J., Darmofal, D., Gropp, W., Lurie, E., Mavriplis, D., "CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences," NASA/CR-2014-218178, March 2014.

[2] Duraisamy, K., Spalart, P. R., Rumsey, C. L. “Status, Emerging Ideas and Future Directions of Turbulence Modeling Research in Aeronautics,” NASA/TM–2017–219682, November 2017.
[3] Wilcox, D.C. “Reassessment of the Scale-Determining Equation for Advanced Turbulence Models,” AIAA Journal, Vol. 26, No. 11, 1988, pp. 1299-1310.
[4] Menter, F.R. “Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications,” AIAA Journal, Vol. 32, No. 8, 1994, pp. 1598-1604.
[5] Rumsey, C.L, Turbulence Modeling Resource, URL: https://turbmodels.larc.nasa.gov/ [retrieved October 2018].
[6] Spalart, P. R. and Allmaras, S. R., “A One-Equation Turbulence Model for Aerodynamic Flows,” Recherche Aerospatiale, No. 1, 1994, pp. 5-21.
[7] Jakirlic, S., Maduta, R. “On ‘Steady’ RANS Modeling for improved Prediction of Wall-bounded Separation,” AIAA Paper 2014-0586, January 2014.
[8] Spalart, P. R. , “Trends in Turbulence Treatments,” AIAA Paper 2000-2306, June 2000.
[9] Marusic, I., Uddin, A. K. M., Perry A. E., “Similarity law for the streamwise turbulence intensity in zero pressure gradient turbulent boundary layers,” Physics of Fluids, Vol. 9, No. 12, 1997, pp. 3718-3726.
[10] Wilcox D. C., “Turbulence Modeling for CFD,” 3rd Edition, DCW Industries, La Canada, CA, 2006.
[11] Marvin, J. G., Brown, J. L., Gnoffo, P. A., “Experimental Database with Baseline CFD Solutions: 2-D and Axisymmetric Hypersonic,” NASA/TM-2013-216604, November 2013.
[12] 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.
[13] Schütte, A., “Numerical Investigations of Vortical Flow on Swept Wings with Round Leading Edges,” Journal of Aircraft, Vol. 54, No. 2, 2017, pp. 572-601.
[14] Morton, S. A., Cummings, R. M., “Computing Unsteady Loads of Fighter Aircraft at High Angles of Attack and in Maneuvering Flight,” RTO-MP-AVT-123, 2005, pp. 5.1-5.20.
[15] Luckring, J. M., Park, M. A. , Hitzel, S. M., Jirásek, A., Lofthouse, A. J., Morton, S. A., McDaniel, D. R., Rizzi, A., Tomac, M., “Synthesis of Hybrid Computational Fluid Dynamics Results for F-16XL Aircraft Aerodynamics,” Journal of Aircraft, Vol. 54, No. 6, 2017, pp. 2100-2114.
[16] Potsdam, M., Datta, A., Jayaraman, B., “Computational Investigation and fundamental understanding of slowed UH-60A Rotor at High Advance Ratios,” 68th AHS Forum, 2012.
[17] Potsdam, M., Cross, Ph., Hill, M., “Assessment of CREATE™-AV Helios for Complex Rotating Hub Wakes,” 73rd AHS Forum, 2017.
[18] Tucker, P.G., “Computation of unsteady turbomachinery flows: Part 2 - LES and hybrids,” Progress in Aerospace Sciences, Vol. 47, No. 7, 2011, pp. 546-569.
[19] AIAA Guide for Verification and Validation of Computational Fluid Dynamics Simulations, AIAA G-077-1998, 2002.
[20] Lee, H. B., Ghia, U., Bayyuk, S., Oberkampf, W., Roy, C. J., Benek, J., Rumsey, C. L., Powers, J., Bush, R. H., and Mani, M., “Development and Use of Engineering Standards for Computational Fluid Dynamics for Complex Aerospace Systems,” AIAA Paper 2016-3811, June 2016.
[21] Tinoco, E. N., Brodersen, O. P., Keye, S., Laflin, K., Feltrop, E., Vassberg, J. C., Mani, M., Rider, B., Wahls, R. A., Morrison, J. H.. Hue, D., Gariepy, M., Roy, C. J., Mavriplis, D. J., Murayama, M., “Summary of Data from the Sixth AIAA CFD Drag Prediction Workshop: CRM Cases 2 to 5,” AIAA Paper 2017-1208, January 2017.
[22] Schmitt, V., Chapin, F., “Pressure Distributions on the ONERA-M6-Wing at Transonic Mach Numbers.” In: Barche, J. (Ed.) Experimental Data Base for Computer Program Assessment, AGARD-AR-138, 1979.
[23] Rudnik, R., Melber-Wilkending, S., “DLR Contribution to the 2nd High Lift Prediction Workshop,” AIAA Paper 2014-0915, January 2014.
[24] Coder, J., “Influence of a Quadratic Constitutive Relation on Detached Eddy Simulations,” Presentation at UMich/NASA Symposium on Advanced Turbulence Modeling, Ann Arbor, 2017.
[25] Spalart, P. R., Rumsey, C. L., “Effective Inflow Conditions for Turbulence Models in Aerodynamic Calculations,” AIAA Journal, Vol. 45, No. 10, 2007, pp. 2544-2553.
[26] Duraisamy, K., Iaccarino, G., and Xiao, H., “Turbulence Modeling in the Age of Data,” Annual Review of Fluid Mechanics, in preparation, 2019.
[27] Benocci, C., Tavoularis, S., Bonnet, J.-P., Leuchter, O., Rodi, W., Onorato, M., van der Vegt, J.J.W., Jimenez, J., Smith, P.D., Cantwell, B., Moser, R.S., Purtell, L.P., “A Selection of Test Cases for the Validation of Large-Eddy Simulations of Turbulent Flows,” AGARD-Report, AGARD-AR-345, 1998.
[28] Schewe, G., “Reynolds-Number Effects in Flow Around More-or-Less Bluff Bodies,” Journal of Wind Engineering and Industrial Aerodynamics, Vol. 89, 2001, pp. 1267-1289.
[29] Uzol, O., Brzozowski, D., Chow, Y.-C., Katz, J., Meneveau, C., “A Database of PIV Measurements Within a Turbomachinery Stage and Sample Comparisons with Unsteady RANS,” Journal of Turbulence, Vol. 8, 2007, DOI: 10.1080/14685240601142867.
[30] Mani, M., Babcock, D. A., Winkler, C. M., and Spalart, P. R., “Predictions of a Supersonic Turbulent Flow in a Square Duct,” AIAA Paper 2013-0860, January 2013.
[31] Shur, M.L. Strelets, M.K., Travin, A.K., Spalart, Ph.R., “Turbulence Modeling in Rotating and Curved Channels: Assessing the Spalart-Shur Correction,” AIAA Journal, Vol. 38, No. 5, 2000, pp. 784-792.

[32] Eisfeld, B., Rumsey, C., and Togiti, V., "Verification and Validation of a Second-Moment-Closure Model," AIAA Journal, Vol. 54, No. 5, 2016, pp. 1524-1541.
[33] Cecora, R.-D., Radespiel, R., Eisfeld, B., Probst, A., "Differential Reynolds-Stress Modeling for Aeronautics," AIAA Journal, Vol. 43, No. 3, 2015, pp. 739-755.
[34] Spalart, P. R., Jou, W.-H., Strelets, M., and Allmaras, S. R., "Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach," Advances in DNS/LES, 1st AFOSR International Conference on DNS/LES, edited by C. Liu and Z. Liu, Greyden Press, Columbus, 1997, pp. 137-148.
[35] Cheung, S. H., Oliver, T. A., Prudencio, E. E., Prudhomme, S., & Moser, "Bayesian Uncertainty Analysis with Applications to Turbulence Modeling," Reliability Engineering & System Safety, 96(9), 2011, pp. 1137-1149.
[36] Singh, A.P., Medida, S., and Duraisamy, K. "Machine-Learning-Augmented Predictive Modeling of Turbulent Separated Flows Over Airfoils." AIAA Journal, Vol. 55, No. 7, 2017, pp. 2215-2227.
[37] Disotell, K.J. and Rumsey, C.L., "Development of an Axisymmetric Afterbody Test Case for Turbulent Flow Separation Validation," NASA TM 2017-219680, November 2017.