A00-33784

AIAA 2000-2306

Trends in Turbulence Treatments

Philippe R. Spalart

Boeing Commercial Airplanes
Seattle, WA

Fluids 2000
19-22 June 2000 / Denver, CO

TRENDS IN TURBULENCE TREATMENTS

Philippe R. Spalart

Boeing Commercial Airplanes, Seattle, Washington

ABSTRACT

We comment on turbulence in aeronautical CFD, covering both established and emerging approaches. The hope is to provide food for thought or controversy to fellow modelers, code developers, users, research planners, and providers of experiments, ultimately improving the productivity of the community. The adequacy of today's turbulence models is of course found to be lacking, although they perform better than their great simplicity would portend. Progress is slow, and in some respects is nearly impossible to gauge. Sidewall problems are again mentioned as major in any "2D" flows with strong pressure gradients, which interferes with efforts to finely validate codes and models. Near-wall grid resolution is discussed, with and without wall functions. The impact of recent challenges to the value of the Kármán constant κ is evaluated. Correction terms for the Spalart-Allmaras model are introduced, dealing with compressibility in free shear layers, and with wall roughness. The promise and limitations of a recent technique for massive separation called Detached-Eddy Simulation, and similar concepts, are outlined.

Introduction

This paper should complement our broader review of turbulence strategies.[1] In contrast with that paper, here we focus on aerospace applications, in fact on airframes, and do not aim at predictions eighty years into the future. In return, some issues are examined in more detail, and concrete suggestions are made. Good recent papers are used for support, but we do not claim to have detected all such papers, nor that the text is free of personal opinion.

The essential questions are as follows. Is turbulence modeling satisfactory, or rather for which flows is it satisfactory? Are the accuracy expectations fair, and are the validation exercises conclusive? Do we have a shortage of experiments, or of ideas? Is the turbulence state of the art implemented and tested in the dominant CFD codes? Do they offer a choice of models, broad enough to be instructive, yet manageable for a non-expert? Is modeling progressing well? Is the effort well directed and well funded? Few of the answers are very reassuring. Turbulence will be a permanent problem in most forms of CFD.

If we may start with the last question, it seems that the direction for RANS models is sensible, at least if we give up hope for a "paradigm change". A small number of modelers are conceiving intuitive ideas, and subjecting them to some validation. Some schools of thought prefer not to rely on intuition and hope to establish a "cascade" from complex models to simple models, using systematic approximations.[2] Their success is debatable from a practical point of view, in our opinion.

Extensive validation of a model is very expensive, and is rarely if ever performed by the model originator before the first publication. It is fairly easy to publish a model (the reviewers for the S-A paper[3] were certainly benevolent). It is much more difficult to obtain the help of the community, which brings with it power and impartiality. Good steps for that include clarity and version control, and "under-selling" the model. Funding is certainly limited, particularly in the US. Europe is investing more, especially in careful validation efforts, as well as encouraging collaboration, and keeping some results from non-European scientists. The Academia-Industry gap is also narrower there. Evidence that European industry has benefited is not strong, strictly in terms of models (since US-made models are widely used there), but it would be logical for such evidence to be guarded. Some PhD dissertations in Toulouse contain findings on the S-A model; yet they have been, oddly, unavailable to the author.

Classical models

No model of today is much more universal than those of the 1970's, when the conflicts between simple free shear flows such as plane and round jets made up the principal puzzle. Therefore, no model can be applied to new flows without serious monitoring and quantitative validation in the important flow regions (the same care must be applied to gridding the regions). For instance the S-A model has a glaring deficiency in round jets[4,5] and should not be used in flows that depend on such a jet. Fortunately, few phases of flight pose that challenge. The model is not very good in round wakes either, and these are present in most phases of flight. However, they are often far enough downstream not to matter for performance measures; they matter to noise-prediction tools and to studies of jet blast at airports. In any case a user, or at least a user community, needs to have examined the performance of the model in the class of flows they are calculating. The only safe

area may be that of non-hypersonic attached boundary layers with mild pressure gradient and curvature.

All CFD codes should offer a choice of turbulence models. Modeling will never be a solved problem, and a substantial procedure to size up turbulence-modeling errors must be provided, just like grid refinement is provided to size up discretisation errors. Given this, it is not acceptable for a large supported government code to offer only one model. This also is the case for some codes from CFD vendors. Curiously, some vendors go to the other extreme. Not only do they offer numerous models and fashionable new versions, but they let the user override the empirical constants at the click of a mouse. A window opens to suggest exactly this exercise. An untrained user should not set $c_{\epsilon 1}$ at will, with no regard even for the Kármán constant! Similarly, if a publication uses non-standard constants in "Model X", it is not Model X any more. Admittedly, some model developers set a poor example by modifiying their Model X from conference to conference, implying full continuity between versions but without ensuring that past successes are still achieved by the new version.

Aerospace CFD, at least in the US, is primarily served by two fairly simple transport-equation models, S-A[3] and SST.[6] These models could have been created and used 30 years ago; they are not different from models of that era, either in their mathematical structure and cost level or in their calibration base (except the shock/boundary-layer interactions for SST). On the other hand, both were driven by considerations of sensitivity to free-stream values, which were not as explicit in the past (and surprisingly remain ignored by many modelers[1]). The k-ϵ model sees some use, but its prediction of separation is not accurate enough in many cases. It places more emphasis on free shear flows, possibly because separating flows were numerically too challenging when it was created. The B-B[7] and k-ω[5] models suffer from excessive sensitivity to freestream values, a very serious obstacle to reproducibility and grid convergence; these models now appear to lack "momentum" (in spite of the very favorable performance of k-ω in the near-wall region). Original ideas are in the models of Durbin[8] and Perot,[9] but they have not been implemented in many codes, partly because of their higher number of equations. A consistent accuracy advantage also remains to be proven.[10] Still more complex models such as cubic eddy-viscosity and Reynolds-Stress Transport models are in contention, but again the advantage is unclear, at least for the gross quantities such as lift and drag.[11]

Some Recent Contributions

The article of Barakos and Drikakis[12] contains lessons that are relevant and supported by other studies. They tested a good range of k-ϵ, k-ω, and nonlinear models up to three equations on two transonic "bump" flows

with strong shock/boundary-layer interactions. The stated attraction of the nonlinear models is "to capture effects arising from normal-stress anisotropy". In the event, the nonlinear models were unable to show a sustained superiority over the SST model (the S-A model was not run, but would probably be slightly worse than SST, for the ONERA flow). This suggests again that such flows are dominated by the shear stress and not the normal stresses (although, unless there is a unique set of axes, it is not rigorous to distinguish shear stress and normal-stress differences). The shear stress was the focus of the design of the SST model, and the principal difference between Menter's "second-best" and "best" models (BSL and SST[6]). The differences between normal stresses have a weak impact on these flows (we also found this with an experimental nonlinear version of the S-A model, which gives more-accurate normal stresses[1]). As a result, the elaborate models improve the description of the turbulence, but not necessarily the mean flow in shallow separated regions.

Another finding is that experiment and CFD are far from agreeing on the Reynolds shear stress, even when the velocity profiles agree quite well, and that at several x stations.[12] This strongly suggests an inconsistency with momentum conservation in the experiments. Turbulence measurements in transonic flows are difficult, and these are over fifteen years old. It appears that dramatic improvements will be needed before transonic Reynolds-stress measurements are used as a primary diagnostic. The latest optical techniques may help, but could lead to prohibitive run times in transonic tunnels. Another relevant detail is that the comparison of Reynolds shear stresses depends on which axes are used;[13] a modest change of the axis orientation very noticeably alters the agreement for the quantity $\overline{u'v'}$, especially if the model uses a scalar eddy viscosity.

The work of Forsythe, Hoffmann and Damevin[14] shows that shock-induced separation is not a single problem, for which full confidence could be gained from a few transonic cases. In their 24° compression corner at Mach 2.85, the success of models is largely reversed. Menter's BSL model now surpasses the SST version, and k-ϵ is quite good. The S-A model performs poorly. Later Forsythe[15] added the SARC curvature correction[16] which greatly improved the shock position over S-A, but left the post-shock skin friction much too low. The differences in post-shock skin friction are strikingly large. Unfortunately the calculation domain did not include the full recovery region. None of the models is very successful.

These findings make RANS modeling for supersonic ramps appear very unfinished, and suggest opposite trends, among models, from the transonic ones. The difficulties *may* stem from large-scale unsteadiness in the experiment (indeed, for once, the average-pressure

rise is steeper in the CFD), but this does not exonerate modeling. If an unsteadiness with frequencies much lower and spanwise coherence much larger than those of the internal turbulence of the incoming boundary layer is present, it is legitimate to expect unsteady CFD solutions to capture it, just like they capture vortex shedding from a cylinder. Note that Detached-Eddy Simulation (DES, discussed below) may do just that, in 2D and without extreme grid alterations, since it can considerably lower the eddy viscosity compared with RANS. However this would be a perversion. The basic constant of DES was calibrated in Kolmogorov turbulence, and 2D fields obtained continuously from RANS by lowering eddy viscosity would have nothing to do with such turbulence. Such a study could provide unsteady solutions, but their dependence on the grid spacing could never be suppressed.

The only correct approaches are to find a RANS model which, for valid reasons, provides less damping in this flow, or a full (3D) LES. The latter was performed by Urbin, Knight and Zheltovodov.[17] Their simulations as of early 1999 appeared under-resolved in the wall region, and their 8° ramp angle did not cause separation in the mean; thus, the flow was simpler than with 24°. An LES of this flow could be obtained even at high Reynolds numbers from the DES formulas as we have shown in the channel (discussed below[18]), but this would require a much finer "LES grid" and *resolved* three-dimensional turbulence in the incoming boundary layer. In any case the computing cost would be absurd for an aircraft configuration.[1,19]

The "Very-Large-Eddy Simulation" of Hunt and Nixon on the supersonic ramp[20] is quite promising; the mean and fluctuating pressures are very good. It seems appropriate to describe it as an LES, in the sense that it used a standard SGS model, although with wall functions, and that finer grids would bring it close to mainstream LES. We have not found any follow-on to the work. Interestingly, they attribute the shock motion to incoming boundary-layer eddies (presumably those of size δ); however the existence of a frequency gap between the shock motion and the boundary-layer turbulence is undeniable in the experiments. The phenomenon they hypothesize would *not* be legitimately reproduced by any 2D simulation, especially RANS. Very probably, an unsteady RANS would provide near-harmonic behavior, without the large skewness of the pressure fluctuations seen experimentally by Dolling and Murphy.[21]

The experimental shock motion is more consistent with the passage of large "slugs" (patches with an excess or a deficit of thickness) in the boundary layer. This would be ominous, for both LES and experiments. In LES, the constant need to contain the domain size and the simplistic nature of the turbulent inflow conditions both reduce the chances that realistic slugs can occur. In experiments, the slugs could

well be facility-dependent. In a sense, they are flaws of the incoming boundary layer. Thus, a "cleaner" facility may produce a steeper mean-pressure rise. It would give the same value for the peak pressure standard deviation p_{rms} (near the separation shock), but a narrower peak and/or a different skewness factor. Thus, CFD fails to capture a phenomenon, but that phenomenon may well not be unique (that is, it would not be entirely defined by the two visible parameters, the ramp angle and Mach number).

The backward-facing step has created another surprise, regarding the magnitude of the negative skin friction under the separation bubble. It is found from experiment and DNS to be very Reynolds-number dependent.[22] RANS solutions show a much weaker dependence. As a result, each model matches the accurate results at one Reynolds number, and ranking models based on that quantity will be unreliable, until a model is created that duplicates the correct trend. We may conjecture that this trend is a "low-Reynolds-number effect" and will vanish in the higher range, but we cannot prove this yet.

A useful study by Catris and Aupoix centers on the compressible form of turbulence models, focusing on the logarithmic layer.[23] For most of the five models they considered, which had one or two equations, they were able to obtain agreement with the van-Driest-transformed logarithmic law without additional empiricism, by placing derivatives of the density in "strategic" locations. For example the S-A model (at high Reynolds number) becomes

$$\frac{D\rho\widetilde{\nu}}{Dt} = c_{b1}\widetilde{S}\rho\widetilde{\nu}$$

$$+\frac{1}{\sigma}\left[\nabla\cdot\left(\sqrt{\rho}\widetilde{\nu}\nabla\left(\sqrt{\rho}\widetilde{\nu}\right)\right) + c_{b2}\left(\nabla\left(\sqrt{\rho}\widetilde{\nu}\right)\right)^2\right] - c_{w1}f_w\rho\left[\frac{\widetilde{\nu}}{d}\right]^2$$
(1)

Note that the S-A paper was silent on the subject of large density variations, and therefore the conflict between Equation (1) and the one in that paper is only apparent.

Catris and Aupoix also confirmed that the unmodified k-ω model has the least need for any correction, and that unmodified models typically under-estimate the skin friction at high Mach numbers, beyond about 5. This conflicts with the trend in mixing layers (for which models give too much shear stress), and makes a single "compressibility correction" a difficult proposition. In mixing layers the S-A model is considerably improved by a correction due to Secundov:[24] a term $-C_5\widetilde{\nu}^2 U_{i,j}U_{i,j}/a^2$ is added to $D\widetilde{\nu}/Dt$, where a is the speed of sound and empirically, $C_5 = 3.5$. Thus, the term is based on a turbulent Mach number (a peculiar one, however, since the Reynolds-stress tensor in one-equation models is nearly trace-less, so that the turbulent-kinetic-energy Mach number is not available). It could matter for studies of supersonic cavities

and blunt bases such as on missiles.[25] The Catris modification and the Secundov term have not been tested together.

A warning appears needed regarding coding the S-A model.[3] Its destruction term depends on the wall distance d. As far as we have seen, modifying the S-A model to remove d from the formulation is essentially a return to the Baldwin-Barth model, and the freestream sensitivity becomes unacceptable. This is an essential term, which vanishes in free shear layers, but not in the outer region of boundary layers. Unlike the van-Driest term, it is *not* only a viscous-sublayer damping term (with a reach of the order of 100 wall units). The distance d needs to be accurate, and its coding is not trivial, either for accuracy or speed. It should not be computed "down the grid line" to the wall grid point. It should not be computed "to the nearest wall grid point," but to the projection of the field point on the wall. In multi-block grids, it should not be computed "only to wall points in the same block". If that is done, d will be discontinuous, which does not help convergence. The solution will depend on where the block boundaries are, which is physically incorrect. With overset grid blocks, the two blocks will be solving different equations in the same region of space. Why the solution could be expected to converge in such a situation is beyond us.

Turbulence experiments

The author is often asked for suggestions of experiments to "calibrate" turbulence models, both in terms of which flows should be created, and which quantities should be measured. The first answer can be disappointing, and is that there is very little flexibility left in a particular model after years in service. The position that "modeling is rich in untapped ideas waiting for an experiment good enough for their calibration" is inaccurate. There are few new ideas, even fewer that have a deep reason for working, and many basic experiments, old or new, which have not been used. This is partly because of the effort involved, especially since careful 3D calculations are usually necessary[26] (the discriminating flow conditions, especially with separation, are never truly 2D, and make gridding arduous). This effort includes a thorough appraisal of the accuracy of each experiment, which may not be the modeler's or CFD worker's specialty. Furthermore, standards have risen some since many of the experiments were conducted, both modeling standards and numerical standards. The apparent complacency of some of us also stems from a reluctance in serious modelers to create new versions of models. New versions create cost, dilute validation work, and erode user confidence.

Validation and model comparisons are more realistic goals than calibration. Model comparisons include tests of the same model with and without correc-tions such as streamline-curvature or compressibility terms.[27] These exercises may appear empty of creativity, but except on trivial flows, accurately exercising several models requires numerical talent and effort, for which the modelers are very grateful. These studies also benefit from even a modest involvement of the modelers, for "sanity checks". There is no point in not getting the best out of each model in contention. Modern CFD expects models capable of high accuracy in simple but dominant flow modules, for instance the skin friction on an airliner body, and non-catastrophic performance over a complete geometry, for instance an automobile. Therefore, the broadest experience base is essential.

On the question of which quantities to measure, a balance is sought between accessing instructive quantities such as high-order moments, and obtaining a high level of accuracy. As mentioned above, when mean-flow and Reynolds-stress measurements conflict, the mean flow is usually given much more confidence. As a result, substantiated or even redundant measurements of simple quantities can be more helpful than higher-risk measurements of other quantities. Our view is not shared by all modelers, especially those who develop complex models. For such models, simple quantities are not enough for all steps of the calibration; however, calibrating in a complex flow does not appear wise. Complex models are expected to be set in simple flows, many of which are now accessible to DNS (which provides higher moments with almost the same confidence level as low ones), and generalize to complex flows. Renewed concerns over the uniqueness of even the simplest turbulent free shear flows (in other words, over their memory of inflow conditions or sensitivity to distant boundary conditions) have done nothing to make model validation any more satisfying. Simple-flow experiments should be directed at this issue, or at very high Reynolds numbers for wall-bounded flows.

A consideration that deserves sustained attention is that any new experiment should be thoroughly reproducible by CFD. Many experimentalists understand this well, but not all. In fact, concurrent studies are best. Quantities that have been "forgotten" include opposite-wall geometries, suction and blowing rates in case of flow control, and inflow boundary-layer thicknesses on tunnel walls. Of course, this implies additional measurements, but recording the whole geometry of the wind tunnel can allow a larger blockage ratio, leading to an increase in Reynolds number and decrease in relative probe size.[28]

From the point of view of an airframe company, the experiments that can most advance the value of established CFD are flight tests, at and near the design point. The approach is that CFD is performed at flight Reynolds numbers, and that matching wind-tunnel results at lower Reynolds numbers and higher free-stream turbulence levels is not essential. The

principal difficulty in flight tests is aeroelasticity; transonic flows with incipient separation are the frontier and are exquisitely sensitive to wing twist. Optical methods work well now to provide the exact wing shape in a flight test. Structural models are also improving, which benefits the pre-flight predictions and ultimately the wing design. Modern low-impact pressure-measurement techniques based on MEMS are very welcome in flight tests.

Experiments are desired not just to further refine classical RANS turbulence models, but also to explore other turbulence treatments such as LES and DES, which are at their best in massively-separated conditions. The accuracy requirements are then lower. The candidate flows are numerous: stalled airfoils and wings, flap edges, spoilers and other obstacles, wheels, cavities. We believe that the flight tests of a NASA-Boeing Blended Wing Body model, in 2002, should be preceded by simulations (unsteady RANS or DES if needed), daring enough to predict whether or not the aircraft can "tumble".

The structural-fatigue and community-noise implications of the bluff-body flow modules, as well as unsteady phenomena around the jet engines, are mounting by the year and will provide many fruitful applications for simulation-type CFD. The state of the art may only allow us to obtain correct frequencies for the fluctuations and associated noise, but amplitudes will be obtained very soon. The exploitation by non-empirical tools to calculate noise generation is where the challenge lies, as is well recognized by NASA.

Two-dimensional Tests

For validation purposes, running 2D experiments and CFD offers a lower cost and the best possibilities for grid convergence, and was the only option until about a decade ago.[29] Even today, in 3D it is near-impossible to be sure of grid convergence. For the attached flow on a simple wing, we may converge the wall quantities, but probably not the wake. Grid-convergence is in fact not an easy matter in 2D, when separation takes place. This should change with adaptive unstructured algorithms but in current codes, focusing grid on the separated shear layers until grid convergence is undisputable would be very time-consuming (besides, the code may well not find steady solutions at that point[3]). Nevertheless, the desire to define a test case in only two dimensions remains strong, and new "2D" experiments appear every year. However, it is not acceptable to conduct these experiments without side-wall boundary-layer control, certainly if there is separation.

A constant concern with transonic airfoil test cases has been that the best agreement in pressure distributions was obtained by altering angles of attack.[3,29] This could be attributed to the difference between CFD "free-air" conditions and the slotted wind-tunnel floor

and ceiling, but new CFD results indicate that side-wall separation may have an impact of the same order. Jiang[26] and others have conducted careful CFD studies with viscous side-walls and the suction of the experiment. She first confirms that removing the suction has a drastic effect on the centerline pressure distribution; the effect is as large as the extreme differences between models. Therefore, the suction modeling in CFD cannot be casual, and neither can its description be in the report on the experiment. Axisymmetric experiments are much preferable in this respect, but tend to be limited to smaller sizes and therefore Reynolds numbers. Some tests with side-walls use corner jets in addition to suction; these can benefit the flow quality, but they add another item in the CFD representation, and another flow feature that could be poorly resolved, especially with a simple grid structure.
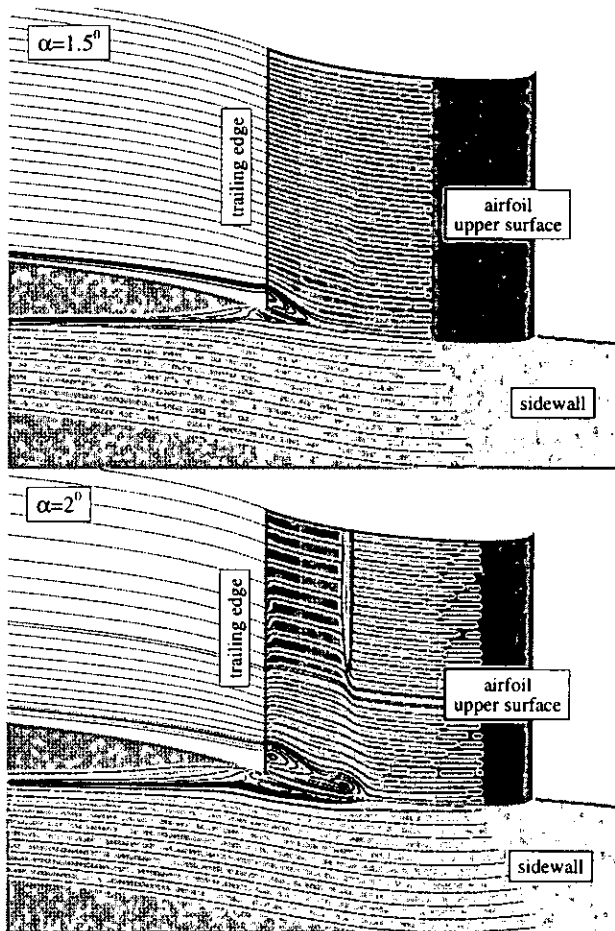

Fig. 1 Side-wall separation in transonic flow with deflected aileron.[26]

Jiang then shows that the separation control by suction can progressively fail as stronger pressure gradients are imposed and the suction level is approximately preserved. Figure 1 is from her paper;[26] the flow features associated with separation are far from uniform across the span. Thus, it is not safe to only verify that the suction is sufficient in the mild cases. Finally she notes that, at least in the experiment she dupli-

cated, the suction was semi-passive and thus was not fully controlled when the wind-tunnel speed was varied. As a result, side-wall effects could masquerade as Reynolds-number effects. But the latter effects are crucial in airliner design.

It appears that 2D tests, even carefully conducted, are not definitive at the level of accuracy that is sought nowadays, for fairly subtle effects such as incipient separation and Reynolds-number dependence. In addition, it is our opinion that many 2D tests even today are *not* carefully conducted. Probably, there are cases in which the experimenter was after rather qualitative or "trend" findings at low cost, but then the CFD workers and their patrons expect to obtain close quantitative agreement.

Rough-wall treatment with the S-A model

This extension of the model may be of help for icing studies, and for estimates of penalties associated with manufacturing or repair methods. We are routinely using it for the ground and for buildings. We follow Schlichting's treatment of rough pipes, and the target is his figure 20.21, which gives the log-law shift as a function of the wall-unit length scale k_s^+.[30] Here, k_s is the conventional Nikuradse sand roughness. Other notation is from Spalart and Allmaras.[3]

We borrow from 1995 work by the group of A. N. Secundov in Moscow (described in a report to Boeing), basically re-tuning their "S-A3" model for better agreement with Schlichting, and for simplicity. The rough model is not local, in that each field point needs the input of a wall roughness k_s (this requires a search for the projection on the wall when k_s is not uniform). The wall boundary condition is not as simple as for smooth walls. On the other hand, the friction velocity u_τ is still not needed.
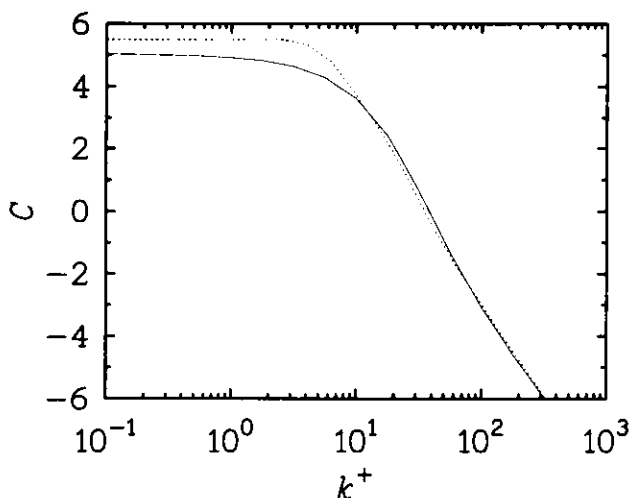

$y = 0$. This requires non-zero wall values of $\tilde{\nu}$ and ν_t. The d function is increased, relative to the minimum distance $d_{\min}$:

$$d \equiv d_{\min} + 0.03\, k_s. \qquad (2)$$

Here, 0.03 is near $\exp(-8.5\kappa)$, and 8.5 is the asymptote of the log-law intercept, in k_s units, as $k_s^+ \to \infty$.[30]

The solution for a standard constant-stress layer is as simple as for smooth walls:

$$\tilde{\nu} = \kappa u_\tau d, \qquad (3)$$

which is obtained with the mixed wall boundary condition

$$\frac{\partial \tilde{\nu}}{\partial n} = \frac{\tilde{\nu}}{d}. \qquad (4)$$

Finally, to adjust the behavior at intermediate values of k_s^+, the f_{v1} viscous damping function is modified by shifting χ:

$$\chi \equiv \frac{\tilde{\nu}}{\nu} + c_{R1} \frac{k_s}{d}, \qquad (5)$$

with $c_{R1} = 0.5$ working well, as seen in Figure 2 (the disagreement for small k_s^+ is unrelated to roughness). The definition of $\tilde{S}$ is adjusted so that we still have $\tilde{S} = u_\tau / \kappa d$ down to the wall, which preserves the budget of $\tilde{\nu}$:

$$\tilde{S} \equiv S + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2}, \qquad f_{v2} \equiv 1 - \frac{\chi - c_{R1} k_s / d}{1 + \chi f_{v1}}. \qquad (6)$$

Note that this term does not use the definition of $\tilde{S}$ based on the f_{v3} function, which we have recommended privately to some users. The f_{v3} formula has an odd effect on transition at low Reynolds numbers, which we never fully understood. It was devised to prevent negative values of $\tilde{S}$. We now recommend, instead, taking for $\tilde{S}$ the larger of the initial f_{v1} definition[3] and $0.3 \times S$.

This term has not seen very much use but our tests indicate fair numerical properties. It could be slightly refined but the incentive is weak, considering how difficult it is to tightly determine the Nikuradse sand-roughness value for a new surface.

Control of transition in turbulence models

It is now a normal desire for a careful CFD user to be able to control where a turbulence model is active and where it is not. It is easiest to set the model in "fully turbulent" mode, typically by setting non-zero inflow values for the turbulence variables ($\tilde{\nu} \approx 3$ is recommended for S-A). When the fluid enters the boundary layers, the variables rapidly amplify (with some models, they amplify prematurely in the irrotational region approaching a stagnation point, but we do not take this up here). However most flows, even at airliner Reynolds numbers, have laminar patches in their boundary layers, and we have seen instances in

Fig. 2 Intercept of log law with rough wall. —, adjusted S-A model; - - -, experiments.[30]

There is no attempt to represent the flow well for $y = O(k_s)$, only to obtain a manageable profile down to

which the level of detail in CFD is now high enough for such patches to influence design decisions.

The S-A model was designed with laminar patches in mind and provided with a "trip term" to activate the turbulence. We found in the meantime that the transition region is too short,[31] but we focus here on the logistical problems, which are more urgent. Unfortunately, these problems are real, and we have been unable to resolve them. The package offered to the code writers is definitely sub-optimal in this respect.

The trip term implies a "trip line" on the surface. It can accept the output of an experimental transition visualization, or of a transition-prediction method. The problems are its cost and its non-local character. The complete application of the term has the following cost. We denote by N the magnitude of the number of points in each direction. The field grid has $O(N^3)$ points, and the trip line has $O(N)$ points. If we allow for transition prediction, the line may move from step to step or iteration to iteration in response to changing pressure gradients. Therefore, at each of these steps we need to find the nearest trip line to each grid point, at a cost of order N^4. This is not practical, and we do not know of a code that implemented this full procedure in 3D (2D costs are very manageable). The term is also non-local in that it uses the vorticity at the trip line to formulate the term at the field point. The consequences are serious for distributed-memory systems or even multi-block grids in which the block-to-block communication is limited.

The response of several code writers has been to use a sort of "blanking" of the turbulence model instead; this appears to be inherited from algebraic models. Blanking can take two forms. In one, the model equations are solved as usual, and the eddy viscosity is not used in the blanked region. This seems dangerous in the sense that the model was designed for a closed-loop situation between the model transport equation and the momentum equation. The model takes meaningless values, but one can hope that these are confined to thin laminar shear layers, and not excessive. In the other form of blanking, the production terms of the model transport equations are "zeroed out". This is more realistic.[11]

The serious problems with blanking are, first, the arbitrariness in the extent of the blanked region in the direction normal to the wall. It is a burden few users are ready for. More importantly, the blanking can well interfere with "legitimate" turbulence coming from upstream, or diffusing, and entering the region. Physically, existing turbulence must be allowed to contaminate a laminar shear region, which has not achieved transition internally. Examples, in a high-lift wing configuration, are the wakes of the slat and main airfoil element over the flap. There is also lateral contamination in boundary layers, leading to "turbulent wedges"; again, these are not particularly accurate

with S-A.[31] Blanking can well suppress some correct physics, and visualizing these phenomena is not easy in 3D.

A much preferable trip system would act only through the boundary condition at the wall. The normal smooth-wall condition is $\tilde{\nu} = 0$ where $\tilde{\nu}$ is the S-A working variable. The ideal procedure would simply set $\tilde{\nu}$ to finite values on a band near the trip line, and that would trigger turbulence across the boundary layer. The cost would be of order N or at most N^2 instead of N^4, and the action would be local. Wilcox proposed essentially such a system for the k-ω model[32] (as published, the tripping system was not local, however, since the boundary-layer thickness and even the streamwise Reynolds number were involved). Unfortunately in the S-A model the destruction term scales with $(\tilde{\nu}/d)^2$ where d is the distance to the wall, so that non-zero wall values are not acceptable.

Versions of some common turbulence models exist for rough walls. It would be quite logical to place a "rough strip" on the wall to set transition. However, as we just saw in (3-6), the roughness length scale propagates into the field equations, at least in the S-A model, so that we again have non-locality (the equation at the field point depends on the roughness at the wall) and the $O(N^4)$ cost. Therefore, existing rough-wall modifications do not solve the problem. The other compromises we have used for tripping, such as searching for a trip only in the same block and near-streamwise grid plane, will probably see years of use. This implies a high level of user awareness and time spent on visualizing the solution. A good starting point is the turbulence index i_t;[3] similar indices can be formulated for models besides S-A.

Another unresolved issue is relaminarisation. A perfect model would return to negligible Reynolds-stress levels when the boundary layer has undergone a strong and sustained enough acceleration or streamline divergence. This is roughly equivalent to its thickness Reynolds number falling below a critical value. Current models do not collapse when they should. This has not received extensive attention in the S-A model, and the transition-control function f_{t2} could have some control, but there is little chance that the task is an easy one and that the performance of the model elsewhere could be preserved. In fact we have had more complaints that the model failed to transition in some low-Reynolds-number boundary layers.

Variations in the logarithmic law

Turbulence models imply scaling laws for the turbulence, and once its constants are set, a model implies values for the empirical constants in the scaling laws. Examples are the decay exponent of isotropic turbulence, and the logarithmic law in wall layers, which is of the highest importance in Aerospace practice. In particular, the Kármán constant κ controls the variation

of the skin-friction coefficient with Reynolds-number. Thus, it has impact not only within CFD, but also in the extrapolation from wind-tunnel to flight Reynolds numbers.

Recent experimental studies have challenged the classical values of the Kármán constant and of the associated "intercept" C in the log law $U^+ = \log(y^+)/\kappa + C$. Here the "+" normalization is in wall units, with friction velocity and viscosity. For many years the author considered that the bracket for κ was $[0.40, 0.41]^{33}$ (ignoring the value in the classic k-ϵ model, which is $\kappa = \sqrt{\sqrt{C_\mu}\sigma_\epsilon(C_{\epsilon 2} - C_{\epsilon 1})} = 0.433$), and he was quite pleased when his Direct Numerical Simulations yielded 0.407 at the highest Reynolds number he could reach.[34] Now the Princeton team after a deep study in a pipe declares 0.436[35] and the joint conclusion of a team in Sweden and another in the US is that their boundary layers indicate 0.386.[36] It is worthwhile to determine the practical importance of such a widened bracket, and it is appropriate for a turbulence-model creator to give users a choice, provided the altered empirical values have credibility.
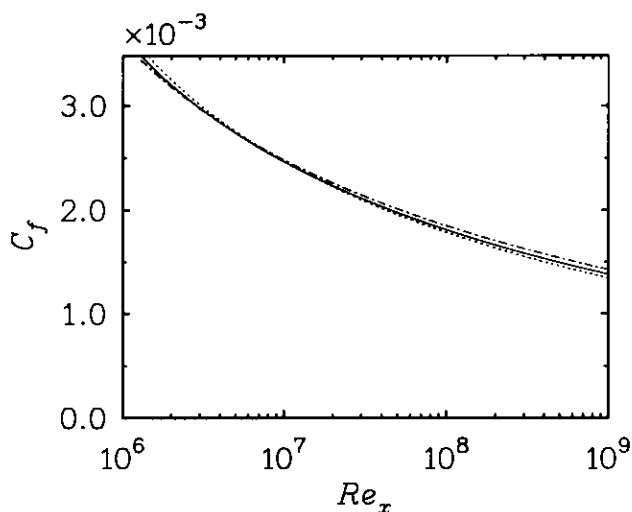

Fig. 3   Local skin friction on a flat plate.
—, $\kappa = 0.41$; - - -, $\kappa = 0.386$; - — -, $\kappa = 0.436$.

We first need a clear theoretical position. We are *not* accepting the idea that both 0.436 and 0.386 are correct, for instance, that there would be a "pipe value" and a "boundary-layer value" for κ. We consider that there is a conflict. It is also unlikely that a turbulence model could be trained to duplicate such a situation (the disagreement applies too close to the pipe wall to be plausibly attributed to transverse curvature). We do accept that the best journals have reviewed and published these papers. Our own analysis of their results does not give exactly the κ values they declare, but that is not crucial. DNS results in the boundary layer indicated that the "apparent Kármán constant" (taken at the inflection point) increased with Reynolds number, strongly suggesting that the final value is

higher than 0.41.[34] The exercise here is to take the papers conclusions at face value, and to conduct a sensitivity test using a simple turbulence model, suitably modified.[3]

Note that both experimental papers support the logarithmic law, only recalibrated, and confirm that alternatives such as power laws were, and still are, obsolete. Power laws are furthermore incompatible with the Galilean invariance that is built, with very good reasons, into all transport-equation turbulence models. No turbulence model is exact, or could be used to *prove* any theory, but giving up invariance would topple all the available transport-equation models.
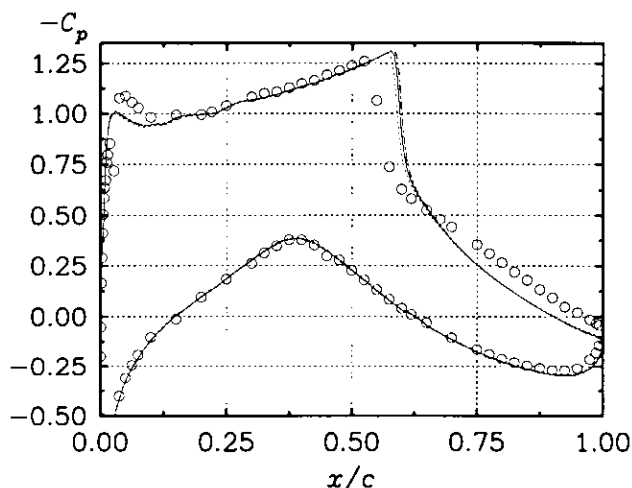
Fig. 4   Pressure coefficient for Case 10, RAE 2822 airfoil. ○, Exp.; —, $\kappa = 0.41$; - - -, $\kappa = 0.386$; - — -, $\kappa = 0.436$.

Within the S-A model, the three constants κ, c_{v1} and c_{w1} are adjusted to adopt a different log law. We preserved the calibration in free shear flows (c_{b1}, c_{b2}, σ), and the requirement for the skin-friction coefficient to be 0.0262 at a Reynolds number $R_\theta = 10^4$.[37] No attempt is made to closely follow the experimental profiles in the buffer layer, below the true log law. This may be fairly easy for the Zagarola profiles,[35] but much less for the Österlund profiles,[36] in which the buffer layer extends to values of y^+ as high as 300. We also give up the exact calibration used in S-A, that $U^+(100) = 16.24$.[37] In any case, we arrive at $\kappa = 0.436$, $c_{v1} = 8.08$, and $c_{w2} = 0.24$ in the first case, and $\kappa = 0.386$, $c_{v1} = 6.05$, and $c_{w2} = 0.35$ in the second. The standard values are $\kappa = 0.41$, $c_{v1} = 7.1$, and $c_{w2} = 0.3$.

We first examine the skin-friction trend at high Reynolds number, in Figure 3. At $Re_x = 10^8$, the Zagarola law gives a 2.2% higher C_f than the standard, while the Österlund law puts C_f 1.4% lower than standard. These differences are modest, but they are not insignificant in regard to the accuracy level that is desired in the airliner industry. The ship and submarine industry probably may have even more interest.[38] We

conclude that the two cited studies are not "academic" in the negative sense of the word. We also conclude that expecting $\pm 1\%$ accuracy from turbulent CFD today would be unfair, since the empirical foundation has not reached that level of accuracy.

Our second test consists in repeating 2D Navier-Stokes calculations over the RAE 2822 airfoil, a classical test case. Especially considering the fallacies of 2D testing for flows with strong pressure gradients,[26] we are not attempting to find the better set of constants, but merely to get a sense for the impact of such changes, which might be magnified relative to the predictable skin-friction change (Fig. 3).

The Reynolds number at the shock is roughly $Re_x = 5 \times 10^6$, which is where the model changes are nearly neutral in Figure 3. Figure 4 shows the pressure distribution for RAE 2822 Case 10.[39] The differences here are quite small. They are much smaller than the disagreement with experiment, which is fairly high because we ran all the cases at $2.56°$ angle of attack. Solutions with matched lift give better leading-edge levels and a better shock position, but not a better post-shock level,[3] and the aileron hinge moments are problematic. Differences will be higher at flight Reynolds numbers, but it does not appear that a particular magnification occurred, even though Case 10 *is* sensitive to turbulence models, because of shock-induced separation.

We conclude that the principal effect of log-law changes is on skin friction, and offer Figure 3 as a rapid guide to estimate the effect at various Reynolds numbers.

Optimal grid distributions

We examine grid design, for the wall-normal direction in a turbulent boundary layer. Many writers correctly report the first y^+ value, but give little description of the rest of the grid. However, the stretching ratio is of high interest. A low first y^+ can be obtained through excessive stretching, and degrade the accuracy in the buffer and log layers. Fitting the interface between viscous and inviscid grid densities to the boundary-layer thickness δ consistently over the surface is equally challenging. Again, that takes more exploration of the flow fields than most users can devote time to.

We first seek the optimal grid distribution, in a slightly idealized situation, by estimating the truncation error. We assume that the turbulence model is numerically "friendly" so that the challenge is in the velocity profile and the shear stress. The basic equation relates the shear rate, the eddy viscosity, and the total stress:

$$\left(1 + \nu_t^+\right) \frac{\partial U^+}{\partial y^+} = \tau^+. \qquad (7)$$

Here we are overlooking artificial dissipation, and the only difficulty is to accurately compute $\partial U^+/\partial y^+$.
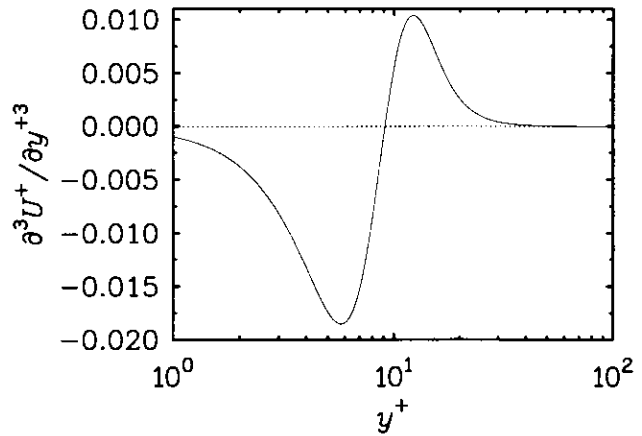

Fig. 5 Third derivative of velocity profile.

We first compute the formal truncation error. We assume second-order accuracy, so that the error is proportional to $\partial^3 U^+/\partial y^{+3}$. This quantity is shown in Figure 5 for the S-A profile and $\tau^+ = 1$ (zero pressure gradient). Naturally for large y^+ it tends to 0 like $1/y^{+3}$. Its integral is also 0, so that with a uniform grid spacing the errors would cancel to leading order after integration. If the grid spacing increases away from the wall, they don't. This figure makes the point that the region of y^+ around 10 to 15 is as critical as the very-near-wall region, say around 1.

In a search for the optimum grid, we look at the global error. The priority is to get an accurate velocity U^+, at some large Y^+; this controls the skin-friction coefficient C_f. Then the error scales with

$$\int_{y_1^+}^{Y^+} h^{+2} \left(\frac{\partial^3 U^+}{\partial y^{+3}}\right) dy^+ \qquad (8)$$

where h is the grid spacing. For now we assume the third derivative does not switch sign. This amounts to having the lower bound y_1^+ larger than about 9. The constraint is the number of points, that is,

$$\int_{y_1^+}^{Y^+} \frac{dy^+}{h^+}. \qquad (9)$$

By a Lagrange multiplier we see that the optimum grid has

$$h^+ \propto \left|\frac{\partial^3 U^+}{\partial y^{+3}}\right|^{-1/3}. \qquad (10)$$

In the log layer this leads to $h \propto y$, consistent with dimensional analysis. That means a constant stretching ratio $h_{j+1}/h_j = \sigma$, i.e., a geometric progression. This is a very common grid design. The relative error for U^+ is about 2% with $\sigma = 1.5$, which is sometimes used. That translates to 4% for the skin-friction coefficient, which is excessive. $\sigma = 1.3$ gives about 1% relative error, which is about low enough in view of the pure modeling errors. In many Navier-Stokes codes,

artificial-dissipation errors need to be added, and probably lead to ratios closer to 1.25 or 1.2.

If we now extend the geometric stretching with $\sigma = 1.3$ to the wall, we see that near $y^+ = 13$, at the second peak of the third derivative in Figure 5, h is about 4 times larger than at the wall. This suggests that a grid with a much more nearly uniform spacing of about 4 up to $y^+ = 20$, followed by geometric stretching, would be just as accurate as one with $h_1 = 1$ and geometric stretching (in fact, based on Figure 5, the errors would come closer to canceling). That type of grid would have only three fewer grid points, but it could be more helpful that its numerical stiffness would be reduced. Convergence could be noticeably improved.

Wall functions and grid requirements

The bulk of Aerodynamics CFD relies on simple turbulence models, for which the designers devoted attention to moderating the near-wall grid requirements,[40] following the lead of Baldwin and Barth.[7] Typical values for the first wall-unit normal spacing, Δy^+, are now around 2. This is lower than the guideline was at the time of algebraic models, namely about $\Delta y^+ = 5$. This does not seem related to the use of transport-equation models, but rather to higher accuracy expectations and/or to the higher levels of artificial dissipation in Navier-Stokes solutions, as opposed to boundary-layer solutions. Values such as 2 for Δy^+ are tolerable in terms of cost even at flight Reynolds numbers, being appreciably larger than those allowed by the k-ϵ model, not to mention Reynolds-stress models. As a result, wall functions are almost never used. When they are, it is in response to the needs of certain unstructured-grid algorithms, which have a low compatibility with extremely shallow cells,[41] or to cope with round-off errors if grids are generated in 32-bit mode.

Gridding with wall functions calls for the following comment. Let us suppose that the condition is imposed at $y^+ = 50$ without any error, in the sense that the velocity, shear rate and eddy viscosity have the same values as in a baseline solution with integration to the wall. A typical and sufficiently accurate grid distribution in the latter solution would have grid points at these y^+ values, rounded to integers: (...31, 36, 42, 50, 60, 72, 86, 94...). The consequence is that the wall-function grid points must be at (50, 60, 72, 86, 94...) if the same accuracy is to be obtained. The natural tendency, and probably the practice today in many CFD activities, is to set the points at (0, 50, 110, 182...). This cannot be adequate.

The consequence of this is that the grid savings associated with wall functions are *much* less interesting than the statement "the first point is at $y^+ = 50$" suggests. A wall function set at that height will only save 12 layers or so, for the price of its complexity and uncertain effect on stability.

Another issue is that, to be consistent with the accepted two-layer description of a *simple* turbulent boundary layer, the wall function would be set at a fixed y/δ, instead of a fixed y^+ as is often done (estimating u_τ for y^+ is easier than estimating δ). Here δ is the thickness of the boundary layer. Then, the benefit would grow with the Reynolds number. However, it is also accepted that the permissible value of y/δ decreases when the boundary layer is more complex, and we have no theory of the region of validity of the law of the wall in pressure gradients, to start with. This is not to imply that the accuracy of any turbulence model is guaranteed in such flows. However, integrating to the wall is simpler, and using wall functions requires more turbulence expertise than a typical CFD user has.

Detached-Eddy Simulation and Relatives

The DES motivation and formulation were published in 1997,[19] but not in a journal or at a mainstream conference, and its diffusion has been spotty. At least three independent efforts have addressed essentially the same charter, which can be summarized as follows. In practical geometries at practical Reynolds numbers, boundary layers must be treated by RANS, for cost reasons, but massive separation regions are better treated with LES, for accuracy reasons.

The DES formulation we have been using applies the S-A model over the whole field, with its input length scale (which is normally the distance to the wall) limited by the grid spacing; this turns the model into a plausible sub-grid-scale (SGS) model in regions where the grid spacing is smaller than the mixing length the RANS model would have.[1, 19, 42] The SGS version was calibrated in isotropic turbulence;[43] there is only one new constant, roughly equivalent to the Smagorinsky constant.

Mr. N. Geordiadis at NASA Glenn Research Center is simulating a supersonic mixing layer, growing from a splitter plate (personal communication, 2000). He treats the boundary layers by RANS with an algebraic model, and the mixing layer by LES with the Smagorinsky model. This is quite different from DES in the formulation, but not in the motivation. Most RANS models are calibrated to reproduce the growth of the mixing layer in steady mode, which could make the LES exercise appear superfluous, except that the models rapidly fail when compressibility becomes influential, or with streamline curvature. He also has vortex shedding from a blunt trailing edge, and shock reflections. In addition, the RANS of such a flow can be ambiguous in the sense that either a steady solution or an unsteady one with Kelvin-Helmholtz roll-up could be the "correct" one. It depends on the geometry and boundary conditions. Georgiadis finds that his simulations sustain three-dimensionality, and transition occurs quite differently from that in 2D runs.

These simulations need to be analyzed much more extensively.

Another independent effort is the Limited-Numerical-Scales (LNS) method of Batten, Goldberg and Chakravarthy.[45] After a fine bit of humor at the "pretentious acronyms" (making an acronym is a form of marketing, but it also serves a legitimate purpose, and often the alternative to an acronym is that the method is named after the author), they insist on the differences between LNS and DES, but mis-understandings are apparent. These are not new, and the unrefereed original DES paper must have left the door open for them.[19] A definition which we hope is more thorough will appear soon.[42] In particular, DES is definitely capable of reducing to DNS, at one end, and RANS at the other. It has now been applied as an LES model in a channel flow;[18] it is simply a matter of designing the needed grid. The channel results are imperfect, but not outright poor for an application without any adjustable parameters. Similar work with LNS is in progress.[45] LNS should not have any trouble progressing from 2D to 3D, but some issues such as gridding guidelines remain open.

The LNS and DES reasoning have much in common including, unfortunately, a sharp critique of a superficially similar proposal of Speziale in 1998[46] (specifically, the Kolmogorov length scale is not the correct one to use in the blending, and obtaining DNS as the fine-grid limit of LES is trivial which makes his blending function appear superfluous). However our understanding is that Prof. Fasel is in fact building on Speziale's proposal, at the present conference.[47] Both LNS and DES positions point out the deep ambiguities of unsteady RANS (which are by no means widely recognized[1]), as well as the desire to guarantee unsteadiness for some applications such as resonant cavities. Both arrive at a model that is non-zonal and is the smaller of a RANS model and an SGS model, locally. The definite differences are in the choice of the underlying RANS and SGS models, and these are not essential compared with the concept of blending RANS and SGS, and of doing it by using the weaker of the two models. For LNS to date, they are a k-ϵ model and the Smagorinsky model, both with cubic formulas for the Reynolds stresses. This seems less well balanced than DES as we have practiced it, with a one-equation model and linear constitutive relation. The Smagorinsky model is too crude to "justify" a cubic constitutive relation, and even the k-ϵ model, if it primarily controls the flow in thin shear layers, may not justify the cubic formula (possibly, the stagnation-point problems are the strongest motivation for the cubic feature). The creators of LNS have legitimate ambitions for it, and already offer it in a commercial CFD package (www.metacomptech.com/cfd++/)! The users should be warned that grid design is even more challenging in LNS than in RANS.

Arunajatesan, Sinha and Menon[48] also identify the need for a hybrid method and aim at a single model that adapts to LES and RANS roles. It is a one-equation k model for LES and a k-kl model for RANS. Again they take inspiration from Speziale, but not his implementation. They present successful isotropic-turbulence tests in LES mode and outline the plans for blending with RANS, but do not actually present hybrid simulations in this first paper, presented as a progress report. Their motivation is again the flow in cavities under aircraft.
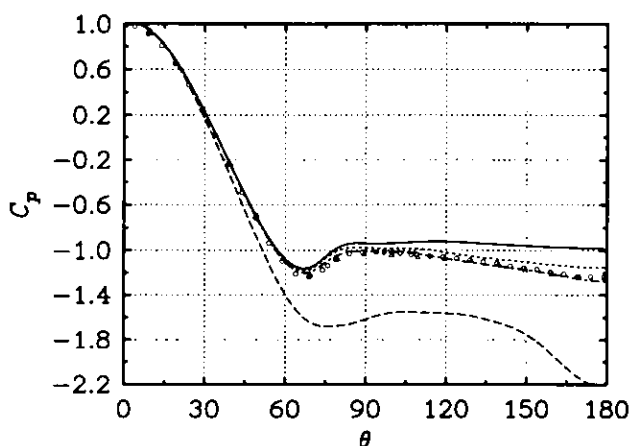

Fig. 6 Pressure coefficient on a circular cylinder.[42] —, coarse-grid DES; - - -, medium grid; - — -, fine grid; – – –, 2D unsteady RANS; o, experiments.

Studies that explicitly follow DES are the sphere work of Constantinescu and Squires[44] and the blunt-missile-base work of Forsythe, Hoffmann and Dieteker[15, 25] at this meeting. Both groups are rather satisfied; it seems that so far DES has never performed worse than RANS. However, some successes could be fortuitous; for instance, the S-A model should be corrected for compressibility in the supersonic mixing layers (as discussed above), and DES *could* have mimicked that correction by reducing the eddy viscosity. Our study of a thin airfoil up to 90° angle of attack[43] was also satisfying for DES and discouraging for unsteady RANS. However we recognize that in that case, DES relied entirely on RANS at low angles of attack, and almost entirely on LES at high angles.

Our circular-cylinder study[42] did more to mingle the two modes and truly exercise DES. It was quite successful, especially at subcritical Reynolds number with laminar separation, while not addressing the Reynolds-number range with the highest transition complexity. Figure 6 shows the pressure coefficients. The agreement with two experiments is impressive, as is the strong suggestion of grid convergence. Such convergence has been elusive in LES. Again, 2D URANS gives vortex shedding at the correct frequency, but the shedding is much too regular, and the drag is much too high. We admit that we are showing here the very best

result obtained from DES, but these simulations followed a systematic procedure for grid refinement (a factor of 2 in each direction between the coarse and fine grid), which precludes "fortunate" adjustments. Also, they were run on inexpensive Pentium personal computers with only 1 million grid points; therefore, success is not a reflection of massive computing expenses. Finally, these results use the S-A model in "trip-less mode" (the shear layer is laminar at separation and transitions by contact with the return flow); thus, the model is given no detailed information about the location and extent of transition. It is simply directed to transition before or after separation.
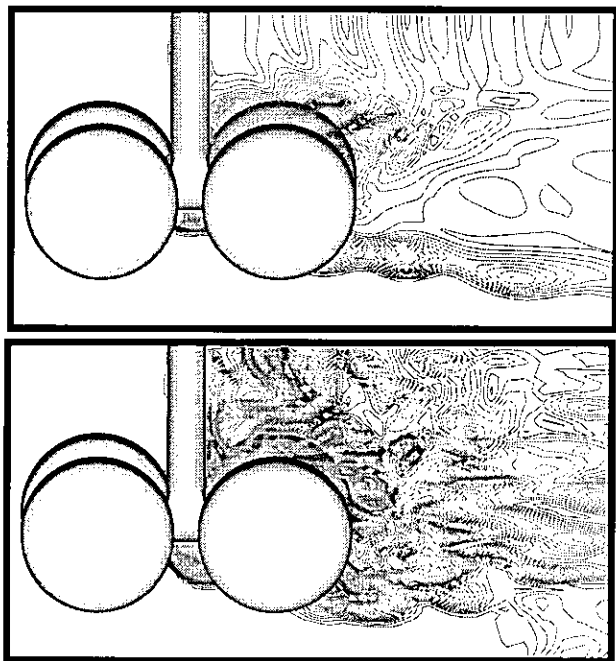
Fig. 7 Vorticity magnitude in the URANS (top) and DES (bottom) of a simplified landing-gear truck. Contours in the center-plane.

Ongoing DES investigations, all using the NTS code of the Strelets group, include subsonic and supersonic cavities, a raised airport runway, and a landing-gear truck (L. Hedges, personal communication, 2000, work funded by NASA, geometry courtesy of B. Lazos). The latter is shown in Figure 7, which illustrates the difference in the flow representation between unsteady RANS and DES. The DES has finer structures, even though the simulations used the same grid. The URANS has residual vortex shedding from the vertical post and the front axle, and more weakly from the wheels (that would be seen in other views). We will ask the question of whether the URANS unsteadiness maps to the low-pass filtered unsteadiness of the DES. The Strelets group is also exploring reductions in the numerical dissipation, compared with the high-order upwind schemes used until now for stability, and the adaptation of other underlying RANS models such as

SST.

As of today, DES appears to be robust, to be free of further adjustable constants, and to reward an increased numerical effort with a visibly enhanced description of the turbulence physics as well as improved quantitative comparisons.

Closing Remarks

In present-day CFD, more effort is devoted to address geometric complexity, with its numerical aspects of grid and algorithm, than to address physical flow complexity, in the form of turbulence. More effort is also *duplicated* by groups writing, maintaining or selling almost the same code. The community, including the code owners, and funding agencies have the rather justified impression that the improvement potential of the RANS turbulence treatment is low. In addition, serious investments in the form of complex turbulence models, or derivations through extensive algebra, or targeted and detailed experiments have unfortunately not born much obvious fruit. New understandings such as from coherent structures or nonlinear dynamics have hardly contributed to engineering (noise prediction could provide a counter-example). A surprise would be welcome in the RANS field.

These remarks apply to attached and mildly separated flows, which are inaccessible to full-domain LES in practice because of the huge ratio between the boundary-layer area and the footprint of the coherent structures an LES would need to resolve.[19] The logical step is then part-domain LES, whether called RANS/LES hybrid, DES, or LNS. These approaches will open new possibilities. Logically it should happen in the truck industry first. Larger strides in accuracy can be expected for massively separated flows than with mild or no separation, because the standards are much lower. Part-domain LES calls forth three remarks. First, such methods remain dependent on RANS models up to and including separation, and therefore give no reason to abandon RANS research. Second, the alternative of conducting time-accurate RANS calculations is found to be ambiguous, when the approach refuses to return a steady solution, and still fairly inaccurate in cases in which it does return an unsteady solution. Third, the objectives of CFD increasingly include unsteady information, for structural, noise or safety studies. In such cases, a perfect steady RANS solution would not be what the engineer needs.

References

[1] Spalart, P. R., "Strategies for turbulence modelling and simulations." To appear, *Int. J. Heat Fluid Flow*, 2000.

[2] Gatski, T., B., and Speziale, C. G., "On explicit algebraic stress models for complex turbulent flows." *J. Fluid Mech.* Vol. 254, 1993, 59.

[3] Spalart, P. R., and Allmaras, S. R., "A one-equation turbulence model for aerodynamic flows." *La Recherche Aérospatiale*, Vol. 1, 5-21, 1994.

[4] Birch, S. F., "One equation models revisited", AIAA-93-2903.

[5] Wilcox, D. C.. Turbulence modeling for CFD. 2nd Ed., 1998. DCW Ind., La Cañada, CA 91011. Note that the S-A pipe/channel subroutine provided with the book contains errors.

[6] Menter, F. R., "Two-equation eddy-viscosity turbulence models for engineering applications", *AIAA. J.* Vol. 32, No. 8, 1994, 57.

[7] Baldwin, B. S., and Barth, T. J., "A one-equation turbulence transport model for high Reynolds number wall-bounded flows", AIAA-91-0610.

[8] Durbin, P. A., "Separated flow computations using the $k - \epsilon - v^2$ model", *AIAA J.* Vol. 33, No. 4, 1995, 659.

[9] Perot, B., "Turbulence modeling using body force potentials", *Phys. Fluids* Vol. 11, No. 9, 1999, 2645.

[10] Kalitzin, G., "Application of the v^2-f model to aerospace configurations." CTR Ann. Res. Briefs 1999, Stanford, CA.

[11] Rumsey, C. L., Gatski, T. B., Ying, S. X., and Bertelrud, A., "Prediction of high-lift flows using turbulent closure models", AIAA-97-2260.

[12] Barakos, G., and Drikakis, D., "Investigation of nonlinear eddy-viscosity turbulence models in shock/boundary-layer interactions." *AIAA J.* Vol. 38, No. 3, 2000, 461.

[13] Ying, S. X., Spaid, F. W., McGinley, C. B., and Rumsey, C. L., "Investigation of confluent boundary layers in high-lift flows", *J. Aircraft*, Vol. 36, No. 3, 1999, 550.

[14] Forsythe, J. R., Hoffmann, K. A., and Damevin, H.-M., "An assessment of several turbulence models for supersonic compression ramp flow", AIAA-98-2648. Subm.,*J. Fluids Eng.*

[15] Forsythe, J. R., "Numerical computation of turbulent separated supersonic flowfields", PhD. Dissertation, Aerosp. Eng., Wichita State U., Spring 2000.

[16] Spalart, P. R. and Shur, M., "On the sensitization of simple turbulence models to rotation and curvature." *Aerospace Science and Technology* Vol. 1, 5, 1997, 297-302.

[17] Urbin, G., Knight, D., and Zheltovodov, A. A., "Compressible large eddy simulation using unstructured grid: supersonic turbulent boundary layer and compression corner," AIAA-99-0427.

[18] Nikitin, N. V., Nicoud, F., Wasistho, B., Squires, K. D., and Spalart, P. R., "An Approach to Wall Modeling in Large-Eddy Simulations." *Phys. Fluids* Vol. 12, No. 7, 2000.

[19] 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" (invited). First AFOSR International Conference on DNS/LES, Aug. 4-8, 1997, Ruston, Louisiana. In "Advances in DNS/LES", C. Liu and Z. Liu Eds., Greyden Press, Columbus, OH.

[20] Hunt, D., and Nixon, D., "A very large eddy simulation of an unsteady shock wave/turbulent boundary layer interaction", AIAA-95-2212.

[21] Dolling, D. S., and Murphy, M. T., "Unsteadiness in the separation shock wave structure in a supersonic compression ramp flow", *AIAA J.* Vol. 21, No. 12, 1983.

[22] Jovic, S., and Driver, D., "Reynolds number effects on the skin friction in separated flows behind a backward facing step." *Exps. Fluids* Vol. 18, 1995, 464.

[23] Catris, S., and Aupoix, B., "Density corrections for turbulence models", *Aerosp. Sci. Technol.* Vol. 4, 2000, 1.11.

[24] Shur, M., Strelets, M., Zaikov, L., Gulyaev, A., Kozlov, V., and Secundov, A., "Comparative numerical testing of one- and two-equation turbulence models for flows with separation and reattachment", AIAA-95-0863.

[25] Forsythe, J., Hoffmann, K., and Dieteker, J.-F., "Detached-eddy simulation of a supersonic axisymmetric base flow with an unstructured flow solver", AIAA-2000-2410.

[26] Jiang, F., "CFD predictions for control surface effectiveness", AIAA-2000-0510.

[27] Shur, M., Strelets, M., Travin, A., and Spalart, P. R., "Turbulence modeling in rotating and curved channels: assessing the Spalart-Shur term." *AIAA J.* Vol. 38, 5, 2000, 784-792.

[28] Chow, J. S., Zilliac, G. G., and Bradshaw, P., "Mean and turbulence measurements in the near field of a wingtip vortex", *AIAA J.* Vol. 35, No. 10, 1997, 1561.

[29] Holst, T. L., "Viscous transonic airfoil workshop. Compendium of results", AIAA-87-1460.

[30] Schlichting, H. Boundary-layer theory. 7th Ed. McGraw-Hill, New York, 1979.

[31] Spalart, P. R., "Topics in industrial viscous flow calculations." Colloquium: Transitional Boundary Layers in Aeronautics, Royal Netherlands Academy of Arts and Sciences, December 6-8, 1995. North-Holland.

[32] Wilcox, D. C., "Simulation of transition with a two-equation turbulence model", *AIAA J.* Vol 32, 2, 1994, 247.

[33] Coles, D. E. "The law of the wake in the turbulent boundary layer." *J. Fluid Mech.* Vol. 1, 1956, 191-226.

[34] Spalart, P. R., "Direct simulation of a turbulent boundary layer up to $R_\theta = 1410$." *J. Fluid Mech.* Vol. 187, 61, 1988.

[35] Zagarola, M. V. and Smits, A. J. "Mean-flow scaling of turbulent pipe flow." *J. Fluid Mech.* Vol. 373, 1998, 33-79

[36] Österlund, J. M., Johansson, A. E., Nagib, H. M. and Hites, M. H. "A note on the overlap region in turbulent boundary layers." *Phys. Fluids* Vol. 12, 1, 2000, 1-4.

[37] Bradshaw, P., Launder, B. E., and Lumley, J. L., "Collaborative testing of turbulence models", AIAA-91-0215.

[38] Grigson, C. W. B. "A planar friction algorithm and its use in analysing hull resistance." Roy. Inst. Naval Arch. paper W242, 1999.

[39] Cook, P. H., McDonald, M. A., and Firmin, M. C. P., "Aerofoil RAE 2822 – Pressure distributions, and boundary layer and wake measurements", AGARD-AR-138, 1979.

[40] Bardina, J. E., Huang, P. G. and Coakley, T. J., "Turbulence modeling validation, testing and development." NASA TM-110446, 1997.

[41] Frink, N. T., "Tetrahedral unstructured Navier-Stokes method for turbulent flows." *AIAA J.* Vol. 36, No. 11, 1998.

[42] Travin, A., Shur, M., Strelets, M., and Spalart, P. R., "Detached-eddy simulations past a circular cylinder." To appear, *Flow, Turb. and Comb.*, 2000.

[43] Shur, M., Spalart, P. R., Strelets, M., and Travin, A., "Detached-eddy simulation of an airfoil at high angle of attack." 4th Int. Symp. Eng. Turb. Modelling and Measurements, May 24-26 1999, Corsica. Eng. Turb. Mod. and Exp. 4, Rodi and Laurence Eds., Elsevier.

[44] Constantinescu, G., and Squires, K. D., "LES and DES investigations of turbulent flow over a sphere." AIAA-2000-0540.

[45] Batten, P., Goldberg, U., and Chakravarthy, S., "Sub-grid turbulence modeling for unsteady flow with acoustic resonance." AIAA-2000-0473.

[46] Speziale, C. G., "Turbulence modeling for time-dependent RANS and VLES: a review." *AIAA J.* Vol. 36, No. 2, 173, 1998.

[47] Wernz, S., and Fasel, H., "Control of separation using wall jets– Numerical investigations using LES and RANS." AIAA-2000-2317.

[48] Arunajatesan, S., Sinha, N., and Menon, S., "Towards hybrid LES-RANS computations of cavity flowfields." AIAA-2000-0401.