THE 17th ICAS CONGRESS

THE DANIEL & FLORENCE GUGGENHEIM MEMORIAL LECTURE

CFD AND TURBULENCE

Marten T. Landahl
Massachusetts Institute of Technology*
and
The Aeronautical Research Institute of Sweden**

Abstract

Despite over one hundred years of research efforts and the emergence of the powerful CFD tool, our ability to accurately predict turbulent flows is still seriously limited. The lecture addresses the underlying difficulties and the prospects for future substantial progress in this field. Subjects to be taken up and discussed are: laminar-turbulent transition, the physics of turbulence, turbulence modelling, numerical simulation, experiments vs CFD, coherent structures in boundary layers and free shear layers, turbulence control, and the impact of future generations of supercomputers.

1. Introduction

The present day aerodynamicist, working at the forefront of aerospace engineering, is faced with a multitude of difficult tasks. In order to predict the forces, moments and loads on the flight vehicle he must be able to determine the flow around a complicated configuration over a wide range of flight conditions. The designer of the propulsive unit, on the other hand, needs to know the (fundamentally unsteady) internal flow through the compressor and turbine stages and also that in the burners incorporating complicated combustion processes in the separated flow behind the flame holders. For the external flow the aerodynamicist has at his disposal theoretical methods well tested for laminar flows such as the use of potential flow theory away from the viscously dominated regions in combination with boundary layer theory inside them. With these and with access to a supercomputer the aerodynamicist can generate useful results very efficiently and rapidly.

However, the flows of interest in aerospace engineering are always turbulent, at least in some parts of the flow. Only under extreme conditions can a boundary layer be kept laminar for Reynolds numbers in the range of millions typical for a flight vehicle. The flow in the strong vortices that separate from highly swept wing leading edges at large angles of attack is turbulent, as is generally the case in extended separated flow regions. The internal flow in the combustion chamber is purposely made turbulent so as to improve the mixing of the fuel and the oxidizer. The jet exhaust contains strong turbulent fluctuations which constitute the source of the intense noise that is generated.

Every student of fluid mechanics learns early that turbulent flows are in principle impossible to predict, the best one can hope for is to be able to determine average quantities of skin friction, heat transfer rate and the like with the aid of some semi-empirical model in which one essentially treats the flow as it were laminar, but with momentum and heat transport enhanced due to the turbulence. The student is also taught that the basic reasons for the difficulty with turbulence is the nonlinearity of the governing equations of motion, and that the flow is unsteady and random in nature, which makes the application of linear superposition and the buildup of the flow from simple and orderly basic solutions, methods so loved by the old masters (and the tools preferred by the teachers!), impossible to use. The student will finally be left with the idea that the supercomputers, whether existing ones or the even more powerful ones to expect in the future, will eventually allow predictions to be made directly from the Navier-Stokes equations without the need for modeling.

However, as a somewhat deeper study readily shows, although nonlinearity is indeed at the root of the problem, it is not only the failure of the standard theoretical tools that presents an insurmountable problem for turbulence prediction since it precludes an analytical treatment, but also the exceedingly complicated nature of the flow which makes a detailed prediction impractical and impossible, except for very simple flows at low Reynolds numbers, even allowing for a very optimistic evolution of future generations of supercomputers (see Chapman[1]). The necessity to incorporate very small scales in the calculations carries with it the requirement of very large computer memory, and the associated very small time steps leads to prohibitively long computation times for flows of the high Reynolds numbers of interest.

---

* Professor of Aeronautics and Astronautics
** Consultant

My lecture will discuss the fundamental difficulties of turbulent flow computations and whether and how these may be circumvented by new approaches. Perhaps the most intriguing new possibility that CFD has opened up is that of reaching a deeper understanding of the basic nature of turbulence through the detailed studies of turbulence flows that CFD allows. The clarification of cause-and-effect relationships of how turbulence is created and maintained would mean a giant step forward in the field of basic turbulence research and may lead to the construction of better and more efficient turbulence models for numerical prediction as well as give new ideas on how turbulence could be controlled. A substantial portion of the lecture will be devoted to this issue since it is here that CFD offers the hope of true scientific progress. In this connection one may also ask to what extent CFD has changed the traditional roles of analysis and experiments, a question that will be touched upon.

2. CFD, a powerful tool

During the last ten years CFD has developed into a powerful tool for the aerodynamicist allowing him to determine the aerodynamic characteristics of even fairly complicated configurations over large ranges of flight conditions within good engineering accuracy. Major driving forces for the rapid development of CDF has been the need for aerodynamic prediction methods for the transonic speed range, in which nonlinear effects and shock waves appear, and the interest in flows containing strong separated vortices which show up at large angles of attack for highly swept wings. A common early CFD approach has been the use of the inviscid (Euler) equations in combination with boundary layer corrections determined from codes employing some suitable turbulence model[2]. Surprisingly, the Euler solutions appear to work reasonably well also for flows with separated leading edge vortices[3] and give results in qualitative agreement with Navier-Stokes. However, they are not so successful when extensive regions of separated flow appear, their failure usually attributed to the shortcomings of the turbulence model used.

The standard numerical approach for computation of steady three-dimensional flow is to use finite difference techniques to represent spatial derivatives of the flow quantities (usually written in conservation forms) together with time stepping (in pseudo- or real time) until the solution converges to its steady final value. This may require a total computational time ten to twenty times the typical turnover time for the largest scale of eddy motion present in the flow. The time step in turn has to be chosen small enough to ensure numerical stability which has the consequence that the total computational time is set by the smallest scale of motion that needs to be represented.

The determination of the finite difference grid is usually the most labor intensive part of this process, requiring substantial skill and ingenuity on the part of the numerical analyst[4]. Considerable efforts have been devoted to the grid generation problem in order to design grids that account properly for edges, wing-fuselage joints, shock waves and other such regions where locally large changes of flow properties can be expected and therefore a finer resolution of the grid would be needed. Schemes in which finer resolution grids are embedded in coarser grids have been worked out in so called multigrid methods[5]. An ultimate goal is to make the mesh generation process automatic[9] as well as to allow direct interactive processing from a modern workstation.

With the latest supercomputers, such as the CRAY 2, the size of the internal memory no longer sets the practical upper limit of the number of grid points, and hence the spatial resolution, to be used. However, since the number of numerical operations per time step increases approximately as the cube of the number of grid points, the practical limit is instead set by the time required for convergence of the calculation.

Codes based on the Navier-Stokes equations have also been developed and are now becoming generally available and in use, both in industry and in the universities. They are not excessively more expensive to run than the Euler codes, perhaps requiring a factor of 2-3 more execution time for a given number of grid points, but they will of course require a fine grid resolution in boundary layer regions. In fact, Navier-Stokes calculations have become a fairly common ingredient in graduate theses in our department at MIT, also on the Masters level.

For separated flows and free shear flows the choice of the computational grid involves additional difficulties. Since the numerical computation will have to be carried out in a box of limited size one is faced with the fundamental difficulty of choosing a size of this box large enough to allow for the satisfaction of boundary conditions at infinity and yet a grid of small enough mesh size to give sufficient spatial resolution. The limited streamwise size of the computational box introduces the additional difficulty of how to handle inflow and outflow boundary conditions. In free shear flows the appropriate inflow conditions are that the velocities at the inflow boundary are specified, whereas at the outflow one the pressure is prescribed. For a jet or a free shear layer the width of the flow that needs to be treated will grow in the downstream direction making it impossible to find a computational box of finite dimension that will encompass the whole flow region of interest. A common way to handle these difficulties is to assume the flow to be periodic in the streamwise direction with a streamwise dimension chosen much larger than the largest characteristic scale of motion to be represented. The upstream boundary conditions are then incorporated simply by feeding the downstream solution into the upstream boundary. The downstream growth of the flow cross-stream dimension is determined by replacing the streamwise change by a temporal development.

3. Space-time resolution needed for turbulence simulation

One problem in all numerical flow calculations, whether for a steady flow or in direct simulation of turbulence, is how finely one needs to choose the spatial resolution in order to represent derivatives with sufficient accuracy and yet coarse enough to avoid excessive computation time. The spatial resolution needed will be determined from the geometry of the problem and from an estimate of the highest spatial changes that would arise in the flow, for example, due to gradients of turbulent stresses in the boundary layer. The resolution becomes particularly demanding for direct turbulence simulation at high Reynolds numbers for which the viscosity determines the smallest scales of eddy motion to be represented.

The size of the smallest dynamically significant eddy that would be found in a turbulent flow may be estimated from the Kolmogoroff scaling. This gives that the ratio of the macroscopic (largest) length scale L to the microscopic (smallest) length ℓ (the Kolmogoroff scale) is proportional to $R^{3/4}$, where R is the Reynolds number based on L and the root-mean-square fluctuating velocity. From this estimate one finds that the number of points N to resolve the smallest flow structures is proportional to $R^{9/4}$. The number of floating point operations required to update the solution per time step is of the order of $N^3 \log N$. The ratio the macroscopic to the microscopic time scales is proportional to $\sqrt{R}$ so that the number of time steps required to describe the flow during a characteristic period of the physically significant event is proportional to $\sqrt{R}$ with the proportionality factor being of the order of ten^8. Thus, the total number of steps needed for one realization is of the order of $R^{11/4} \log R$. Typical computational times required for one realization of a simple turbulent flow on the best available supercomputers is of the order of a minute for R equal to 100 but several hours for R equal to 500. With today's supercomputers one reaches the practical time limit for Reynolds numbers in the range of thousands. The range of tenths of thousand is beyond the present possibility but may perhaps be reachable with a future generation of massively parallel computers. To simulate turbulent flows around realistic three-dimensional flight vehicles is unlikely ever to become possible.

Recent work by Henshaw et al.[10] indicates, however, that the Kolmogoroff scaling may in fact give a too conservative estimate of the spatial resolution needed. They show that the smallest scale is inversely proportional to the square root of the Reynolds number based on the kinematic viscosity and the maximum of the velocity gradients. Unfortunately, an estimate of the magnitude of the largest velocity gradients to appear in the flow is only possible for two-dimensional turbulence.

4. The physics of turbulent flows, what has been learned from CFD?

Transition

Flow instability and transition to turbulence has been an area which has continued to attract great interest among fluid dynamicists during the last century and has been a continuous source of inspiration for applied mathematicians and physicists since the pioneering works of Rayleigh, Orr and Sommerfeld. The hope has been that through

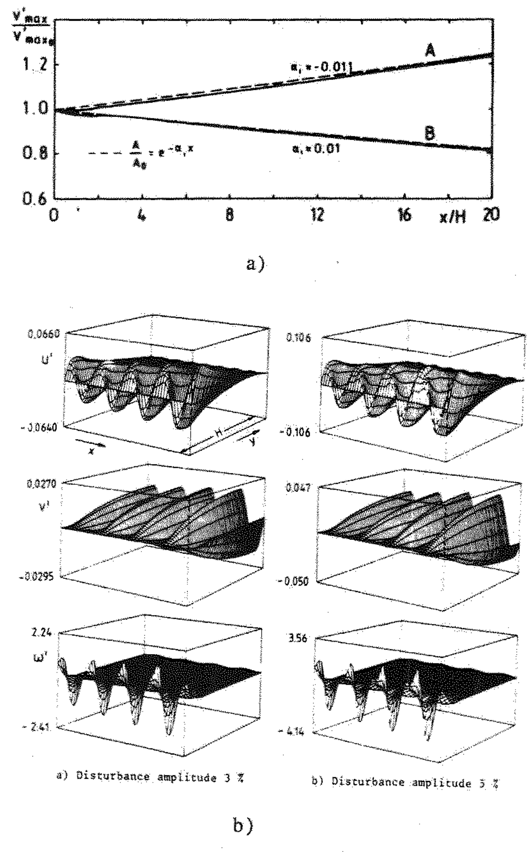


Fig. 1 a) Comparison between the numerical calculations and linear theory for the spatial growth rate of a two-dimensional instabiliy wave in a channel flow. b) Instantaneous streamwise, vertical and vorticity perturbations in the lower half of the channel for two different wave disturbance amplitude levels. (from Fasel and Bestek[10]) a deeper understanding of the transition process one would be able to learn more about the processes creating and maintaining the fluctuations in a fully developed turbulent flow. In the last decade CFD has been found a very successful and valuable tool for this field of research. The simulation of Fasel and Bestek[10] of two-dimensional instability waves in a channel flow (Figure 1) demonstrated, through comparisons with the linear theory, the accuracy of the numerical method used and also showed that for a two-dimensional wave nonlinearity becomes important first for fairly high wave amplitudes. Patera and Orzag [11] were the first ones to continue such calculations beyond the two-dimensional phase and demonstrated the eventual appearance of small-scale breakdown of the flow is due to a three-dimensional secondary instability[18] of the finite amplitude two-dimensional Tollmien-Schlichting instability wave (Figure 2, from Orzag and Patera[18]).

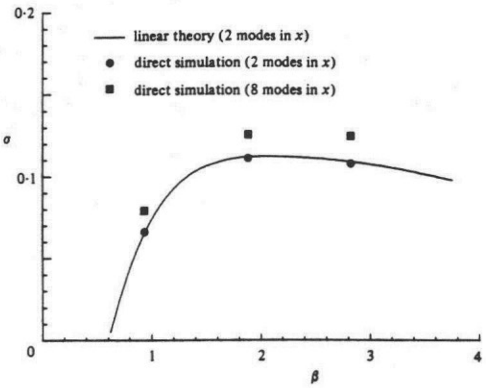


Fig. 2 The growth rate σ of three-dimensional disturbances riding on a two-dimensional neutrally stable wave of α=1.25 at R=4000 as function of spanwise wave number β. (from Orzag and Patera[18]).

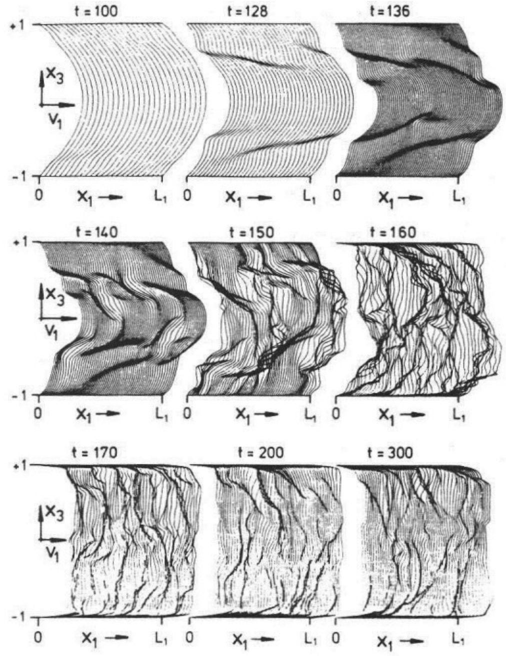


Fig. 3 Evolution of the unsteady streamwise velocity component in the plane of the peak. (from Gilbert[12])

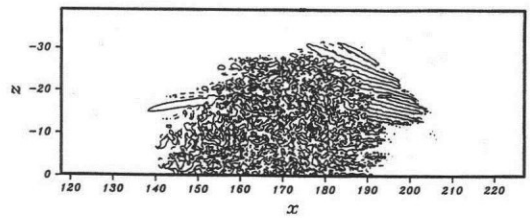


Fig. 4 Contours of normal velocity at center plane of a turbulent spot in a channel for nondimensional time t=258 (from Henningson et al.[14])

Gilbert[12] (Figure 3) also has simulated the evolution of an instability wave with three-dimensional effects taken into account with the breakdown into small scales incorporated (see also Dang and Deschamps[13]). The development of a turbulent transition spot has been studied both in a plane Poiseuille flow and in a boundary layer[14]. The results demonstrate that the spots have marked differences in the two cases, the channel flow spot showing the appearance of waves near its leading edges which are also seen in experiments (Figure 4). The simulations by Breuer and Landahl[15] of the evolution of an initial three-dimensional disturbance in the boundary layer, together with the associated experiments, points to the role that algebraic instability[16] may play in the evolution mechanism.

Calculations of instability and transition have also been carried out for free shear flows by Metcalfe et al.[17]. They found that two-dimensional disturbances grow initially forming an unstable wave train, as expected from instability theory, and then rolled up into a row of spanwise vortices. With increasing amplitudes the vortex cores break up into small-scale eddies and subharmonic instability leading to vortex pairing sets in. Three-dimensional disturbances were found to suffer from secondary instability leading to the formation of streamwise vortices (Figure 6).

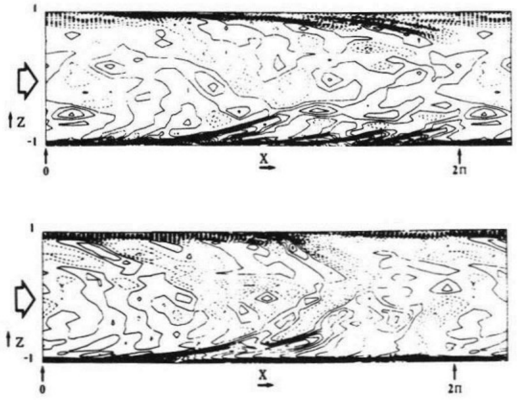
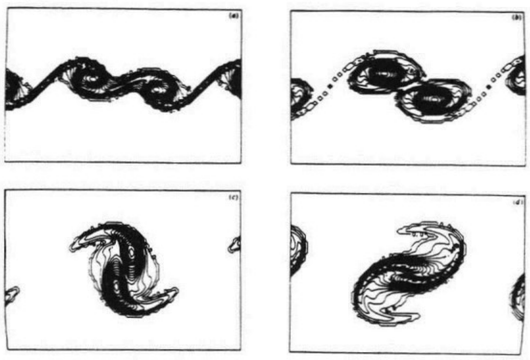
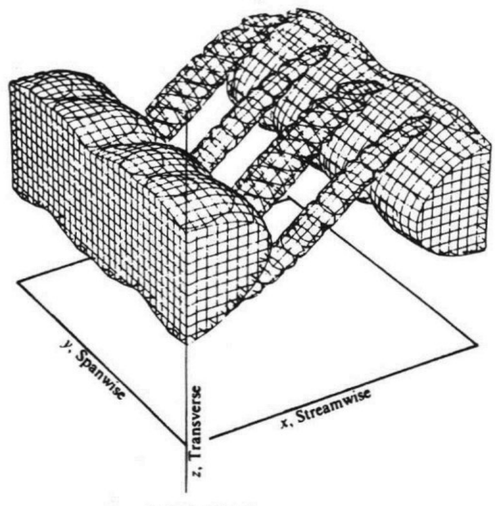


Fig. 5 Contours of constant spanwise vorticity during the late stages of transition of a channel flow at R=3000..

a) Results from direct simulation
b) Large-eddy simulation (from Dang and Deschamps[13])



a)



$|\omega_x|+|\omega_y|+|\omega_z|$

b)

Fig. 6 a) Spanwise vorticity contours for a free shear layer during vortex pairing (For times and other parameter for the calculation, see Figure 4. in the original paper by Metcalfe et al.[17]). b) Three-dimensional plot of 50% level of the sum of the absolute values of all three vorticity components for the shear layer (for parameters for the run, see Figure 10 in the original paper by Metcalfe et al.[17])

Structure of fully developed turbulence

I. Homogeneous and isotropic turbulence

The numerical simulations of Deissler[19] of isotropic turbulence in a box using a spectral method with 32^3 terms showed that the nonlinear energy transfer takes place between widely separated wave number bands and is highly intermittent with the indication that this occurs due to breakup ("bursting") of large eddies. He also found that homogeneous turbulence is chaotic in that it has a positive Liapounov constant, therefore being sensitive to the initial conditions, and shows a Poincare section without a specific pattern.

II. Shear flow turbulence

Deissler's[19] calculations of turbulence in a homogeneous mean shear showed that shear is very effective in producing small-scale structures. He found that the mixing, which is taken place most efficiently by the small-scale structures, is small everywhere, except in localized regions where it is intense and intermittent. This effect of a mean shear may be understood in a qualitative manner with the aid of a very simple model. Consider a fluctuating velocity field, u_i (i=1,2,3) which is carried passively along by the mean shear flow, U(y) ($y=x_2$). Then, adopting Taylor's frozen field hypothesis,

$$u_i = \omega_i(\xi,y,z)$$

where $\xi = x - U(y)t$.

From this one finds for the y-gradients,

$$\partial u_i/\partial y = -tU'(y)\partial\omega_i/\partial\xi - \partial\omega_i/\partial y$$

The first term will thus give rise to y-gradients which increase linearly with time (a closer examination shows that this comes about because of vortex stretching). Thus, the passive transport by the mean shear will lead to the formation of local regions of shear which will intensify with time and produce thin internal shear layers. The scales of the eddies created when such layers break up due to local instabilities will be set by the thickness of the layers, thus small scales are created from the breakup of old larger-scale eddies.

The highest values of mean shear is found near solid walls. Intense small-scale turbulence is therefore to be expected in the wall region of turbulent boundary layers, which indeed both experiments and numerical simulations show to be the region of highest small-scale activity. Because of their high engineering interest wall-bounded turbulent shear flows have received the major attention by the CFD community in later years.

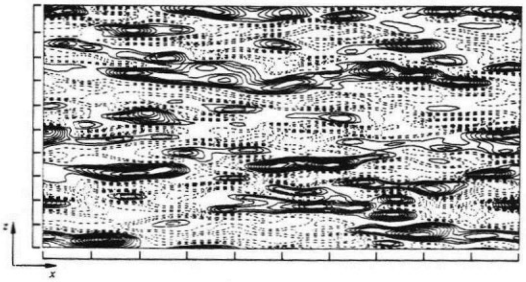


Fig. 7 Contours of constant streamwise velocity for a turbulent channel flow in the plane $y^+ = 6.14$ (from Moin and Kim[20])

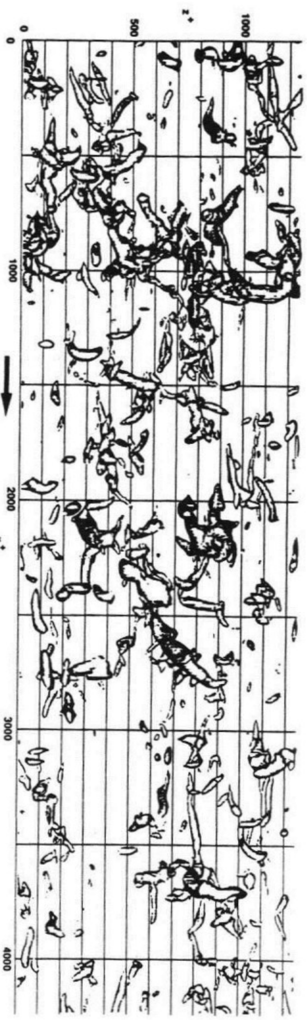
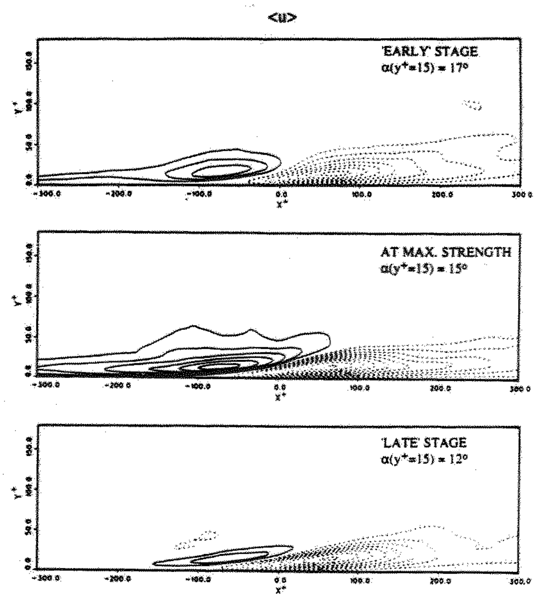


Fig. 8 Top-view of instantaneous three dimensional low-pressure structures in numerically-simulated turbulent boundary layer. Isobaric surfaces computed for $p' = -4.2\rho u_\tau$. (From Robinson[32])

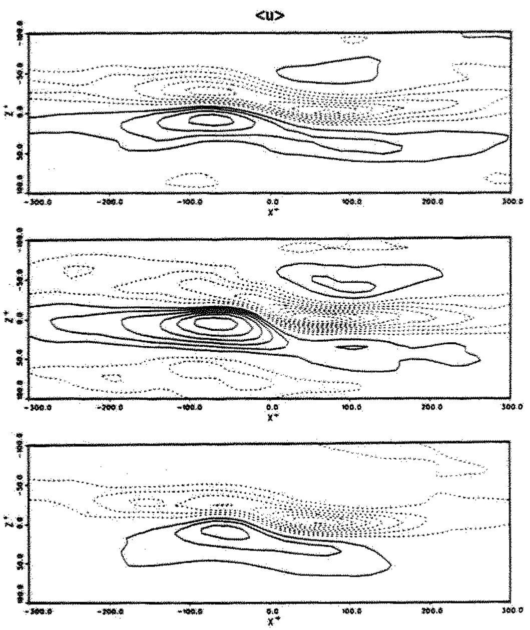
Because of the small-scale motion that needs to be incorporated and the long computation times that become the consequence, the simulation of wall-bounded shear flow turbulence (channel and boundary layer flows) has absorbed a substantial portion of the capacity of the world's supercomputers available to fluid dynamicists in the last few years. The extensive computations for channel flow turbulence carried out by Moin and Kim[20] together with the boundary layer computations by Spalart[21] have furnished valuable databases for fundamental studies. In these the simulated results may be treated as experimental data to which one may apply, e.g., statistical methods like conditional sampling, in order to search for recurring patterns ("coherent structures"). NASA has organized summer workshops on the studies based on these data bases [22,23]. The streamwise velocity field generated by Moin and Kim[20] reveals the presence in the near - wall region of highly elongated streamwise streaks of alternating high and low-speed flows (see Figure 7). A simple inviscid model employed by Landahl[24] indicates that the streaky structure is the consequence of algebraic instability[16] and that the turbulent stresses result from the development and streamwise growth of the streaks.

An example of the kind of interesting results that can only be found from numerical simulations is shown in Figure 8. By looking at regions of low pressure, Robinson[32] was able to identify the cores of the instantaneous strong vortices that appear in the flow. A comparison with Figure 7 makes it clear that the streaks seen near the wall do not consist of streamwise vortices, as they are often incorrectly termed in the literature.

Results from conditional sampling of the simulated channel flow by Alfredsson et al.[25] employing the so-called VISA method, in which the condition for sampling is taken to be when the variance of the streamwise fluctuations (determined for a selected streamwise averaging distance) exceeds the mean by a selected threshold, are shown in Figure 9. The VISA sampling method tends to sort out structures involving rapid streamwise changes. As seen, a characteristic flow feature that emerges is an inclined shear layer through which the streamwise velocity decreases rapidly in the streamwise direction from a value higher than the mean to one that is lower. It is remarkable that experiments on turbulent channel flow (Johansson et al.[26]) employing the VITA method (same as the VISA but averaging over time instead of space) show the same kind of inclined shear layer.



a)



b)

Fig. 9 a) VISA-educed conditional averages of the streamwise velocity in the xy-midplane of the structure at three stages of development (for parameters see original paper of Alfredsson et al.[25]) b) Conditional averages obtained with a modified VISA-method preserving spanwise asymmetry. Contours of the fluctuating streamwise velocity in the plane $y^+=15$ at three stages of development. (from Alfredsson et al.[25])

5. Statistical closure models and turbulence modeling

In statistical formulation of the turbulence problem, the well known stumbling block is the problem of closure; the number of equations for the statistical moments that one can derive from averaging the equations of motion is always one less than the number of moments sought. It is therefore necessary to adopt some hypothesis for how the higher moments are related to the lower-order ones in order to close the set of equations. The search for successful statistical closure models has occupied many theoreticians in the past including some physicists famous from other fields (e.g., Heisenberg). CFD offers in practice the only realistic means for detailed testing of such models. As an example we reproduce in Figure 10 comparisons made by Orlandi[27] of the nonlinear energy transfer term T(k) as function of the wave number, k, calculated by the Eddy-Damped Quasi-Normal Markovianized theory (EDQNM) of Orzag[29] with that obtained from a direct simulation. Apparently, the EDQNM closure model represents this important term with good accuracy.

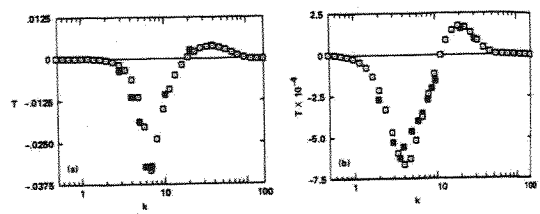


Fig. 10 Energy transfer wave number spectrum : (a) t=0.54, (b) t= 4.34, □ EDQNM ■ Direct simulation (from Orlandi [27])

In the simplest statistical treatment of the equations of motion one takes their ensemble average which produces equations for the mean velocity distribution involving the average transport of momentum by the velocity fluctuations, the so-called Reynolds stresses. For their solutions one therefore needs to model the Reynolds stresses in terms of the mean velocity distribution. Since in most engineering applications one is primarily interested in only averaged quantities, the use of the averaged equations together with turbulence modeling would be the preferred route to follow in most cases. The turbulence models may involve algebraic expressions or differential equations, or both, containing empirical constants. Turbulence modeling has been one of the major preoccupations of the researchers taking their leads from the early simple suggestions of Boussinesq (use of a turbulent viscosity) and Prandtl (mixing-length hypothesis).

The number of proposed turbulence models has naturally expanded over the years and thereby also the number of empirical constants they incorporate. The most commonly used models in industrial codes are the algebraic ones being based on extensions of Prandtl's mixing-length idea, and the k-ε one, which relates the dissipation ε to the kinetic energy k by model differential equations. To calibrate such models it is necessary to make detailed comparisons with experiments. This has led to the initiation of a series of turbulence modeling "olympiads" in the last two decades in which the participants are asked to use their preferred models on a number of selected standard test cases for which extensive sets of measurements have been carried out (but not available to the participants ahead of time!). Although some of the methods do indeed come out better than the others for some of the flows, so far there has not been singled out any clear winner for all test cases. The latest and most ambitious effort along this line has been a recent meeting in Lausanne[28] organized under the auspices of the European organization ERCOFTAC in which the experimental test base is an extensive set of measurements of the turbulent flow in a S-shaped channel.

One idea for turbulence modeling first introduced by meteorologists (Smagorinsky[30]) is to model only the smallest scales, so called subgrid-scale modelling, on the philosophy that most of the turbulence production and transport is carried by the large-scale, essentially inviscid eddies, whereas the smallest scales primarily take care of the dissipation. In the large eddy simulation (LES) method the large-scale eddies then can be calculated without direct modelling and one should be able to arrive at good results for the turbulent transport and hence for the averaged quantities. In recent extensions of the LES method one applies a filtering technique to the equations of motion to sort out the proper equations for the subgrid-scale motion[31]. An interesting example on how direct simulation and LES results compare is shown in Figure 5. Clearly, the LES results are in good qualitative agreement with those obtained from direct simulation. In particular, both methods show the appearance of intense shear layers near the walls during this late stage of transition. It is interesting that such shear layers are also found in fully developed turbulent flows near walls (see Figure 9).

6. The changing roles of analysis and experiments

The emergence of CFD as a major tool in turbulence research has naturally raised the question of what roles mathematical analysis and experiments will play in this field in the future in a situation where direct numerical computation is becoming increasingly efficient and economical. Clearly, analytical methods for prediction of turbulent flows, also those based on sophisticated statistical closure hypotheses, have not as yet lived up to the high hopes one once might have placed on them. For certain simple flows at low Reynolds numbers, numerical simulation might even replace experiments, especially for the study of flow quantities that are difficult or impossible to measure, such as instantaneous pressures in the interior of the flow.

However, both analysis and experiments are likely to remain important ingredients in basic turbulence research for the forseeable future. Experiments will of course always constitute the ultimate test of any theoretical prediction, especially at high Reynolds numbers for which direct numerical simulation will be out of the question for quite some time to come. For the low Reynolds numbers, experiments and simulation can go hand in hand both in verifying the predictions and in the use of these for controlling the experimental situation, for example in making sure that the probes used in the experiments are small enough to resolve the scales of interest and as an aid in the interpretation of the experimental results.

Analysis will still have an important role to play in turbulence prediction and research, even though the hopes of ever devising analytical prediction methods from first principles are highly unlikely ever to be fulfilled. To improve the numerical methods and judge their applicability and accuracy, considerable mathematical analysis is needed. There is still need for improved turbulence modeling, possibly with use made of new knowledge of coherent structures and their role in the creation of turbulent stresses. Application of analytical models to the numerical simulation results may help to unravel important basic turbulence mechanisms.

One important and intriguing goal for such research combining the tools of experiments, numerical simulations and analysis is to devise methods for controlling turbulence. In attached boundary layers the turbulence leads to enhanced skin friction; ways to reduce this without an accompanying extra drag penalty could lead to substantial savings in fuel. The turbulent fluctuations also constitute an undesirable noise source which one would like to reduce in the interest of passenger comfort. On the other hand, turbulence helps prevent separation, and an efficient technique for generating the "right" kind of turbulence may make possible the design of airfoils with increased maximum lift and hence allow lowered landing speeds. Turbulence is also desirable and necessary to achieve good mixing in burners.

7. Conclusions

The short review presented here on the application of numerical techniques to turbulence research and flow field predictions should give a picture of the very rapid and substantial progress that the emergence of the CFD tool has made possible in this very difficult area of fluid mechanics. The important new advances in the understanding of the physics of turbulence that have come out of this are particularly remarkable and are likely to continue thanks to the extensive databases that are becoming available from the numerical simulations and which may be treated as a rich store of experimental data. However, true future progress in our understanding of and ability to predict turbulent flows will require a sustained attack employing all the three main tools of the fluid mechanician; analysis, experiments, and numerical simulation. In the last one, we have seen a very rapid progress in the recent years and it is likely to become ever more important as more powerful hardware becomes available and new improved and more efficient codes are developed.

In this review, the new insight gained in the turbulence transition mechanism, in particular the later stage in which breakdown to smaller scales sets in, was given particular attention. Fully developed turbulence is in some sense controlled by the smallest scales and one may therefore hope that such insight will help in the design of better turbulence models, incorporating also the new knowledge on coherent structures such as the streaks near solid walls and the rolled up vortices in free shear flows. Better understanding of the turbulence production mechanism that could be gained through the combined application of numerical simulation, analytical methods, and laboratory experiments should also make possible the construction of more efficient computational methods based on the idea of large-eddy simulation. Such progress would be essential for reaching the ultimate goal of such research, namely to be able to understand the high-Reynolds number turbulence in the flow situations of interest to the aerospace engineer and to have the methods for predicting them accurately.

References

[1] Chapman, D.R. "Computational aerodynamics development and outlook", *AIAA J.*, Vol 17, p. 1293, 1979.

[2] Rizzi, A., Drougge, G. and Muller, B," Navier-Stokes and Euler Solutions for Vortex Flow over a Delta Wing", in *Proceedings, IUTAM Symposium Transsonicum III*, p. 305, Springer-Verlag, 1989.

[3] Murman, E.M., Rizzi, A., and Powell, K.G., "High Resolution Solutions of the Euler Equations for Vortex Flows",, in *Progress and Supercomputing in Computational Fluid Dynamics*, Birkhauser-Boston, p. 93, 1985.

[4] Eisemann, P.R. "Grid Generation for Fluid Mechanics Computations". *Ann. Rev.Fluid Mech.*,17, p.487, 1985.

[5] Eriksson, L.-E. "Generation of Boundary-Conforming Grids Around Wing-Body Configurations Using Transfinite Interpolation". *AIAA Journal*,20, p. 1313, 1982.

[6] Fuchs, L. and Zhao, H.-S. "Solution of three-dimensional viscous incompressible flow by a multigrid method." *Int.J.Numerical Methods in Fluids.*,4, p.539, 1984.

[7] Rizzi, A, Petersson, A. & Oppelstrup, J. "Exploring Interactive Mesh Generation". in Computer Graphics and Flow Visualization in CFD, VKI Lecture Notes, Brussels, 1989

[8] Canuto,c., Hussaini, M.Y., Quarteroni, A. and Zang, T.A. Spectral Methods in Fluid Dynamics, Springer, 1987.

[9] Henshaw, W.D., Kreiss, H.O. & Reyna, L.G. "On the Smallest Scale for the Incompressible Navier-Stokes Equations". in Theoretical and Computational Fluid Dynamics, p. 65, Springer Verlag, 1989.

[10] Fasel, H. and Bestek, H. "Investigation of Nonlinear, Spatial Disturbance Amplification in Plane Poiseuille Flow." in *Laminar-Turbulent Transition*, p.173, eds. R.Eppler and H.Fasel, Springer, 1979.

[11] Patera, A. and Orzag, S. "Subcritical transition to turbulence in plane shear flows. in *Transition and Turbulence*, ed. R.E. Meyer, p.127, Academic Press, 1981.

[12] Gilbert, N. "Numerische Simulation der Transition von der laminaren in die turbulente Kanalstromung. Forschungsbericht DFVLR-FB 88-55, Deutsche Forschungs- und Versuchsanstalt fur Luft- und Raumfarhrt, Gottingen, Germany. (Doctoral Dissertation at the Universitat Karlsruhe),1988.

[13] Dang, K. and Deschamps V. "Numerical Simulation of Transitional Channel flow," in Numerical Methods in Laminar and Turbulent Flow eds C.Taylor, W.G.Habashi and M.M Hafez, Pineridge Press, 1987, p. 423.

[14] Henningson, D.S., Spalart, P. and Kim, J. " Numerical simulations of turbulent spots in plane Poiseuille and boundary layer flows". Phys. Fluids 30, p.2914.

[15] Breuer, K. & Landahl, M. "The Development of a Localized Disturbance in a Boundary Layer". J. Fluid Mech. (to appear)

[16] Landahl, M.T. " A note on an algebraic instability of inviscid parallel shear flows." J. Fluid Mech.98, 243-251., 1980.

[17] Metcalfe, R.W., Orzag, S.A., Brachet, M.E., Menon, S., Riley, J. "Secondary instability of a temporally growing mixing layer". J. Fluid Mech., Vol.184, p.207, 1987

[18] Orzag, S.A. and Patera, A.T. "Secondary instability of wall-bounded shear flows" *Journal of Fluid Mechanics*,Vol. 128, p.347, 1983.

[19] Deissler, R.G. "On the Nature of Navier-Stokes Turbulence" NASA Technical Memorandum 101983, May, 1989.

[20] Moin, P.and Kim, J., "Numerical investigation of turbulent channel flow", *Journal of Fluid Mechanics*, Vol 118, p. 341, 1982.

[21] Spalart, P.R. "Direct simulation of a turbulent boundary layer up to R_{eq}= 1410", *Journal of Fluid Mechanics*, Vol. 187, p. 61, 1988

[22] "Studying Turbulence Using Numerical Simulation Databases "-I Proceedings of the 1987 Summer Program, Center for Turbulence Research, NASA Ames Research Center and Stanford University, Report CTR-S87, 1987.

[23] "Studying Turbulence Using Numerical Simulation Databases "-II Proceedings of the 1988 Summer Program, Center for Turbulence Research, NASA Ames Research Center and Stanford University, Report CTR-S88, 1988.

[24] Landahl, M.T. "On sublayer streaks", *Journal of Fluid Mechanics*, Vol.212, p.593, 1990.

[25] Alfredsson, P.H., Johansson, A.V. and Kim, J. "Turbulence production near walls: the role of structures with spanwise asymmetry", in *Proceedings of the Summer Program 1988, Center for Turbulence Research*,p. 131, December 1988

[26] Johansson, A.V., Alfredsson, P.H. and Eckelmann, H. "On the evolution of shear-layer structures in near-wall turbulence". In *Advances in Turbulence*, p. 383, *Proc. First European Turbulence Conference*, Lyon, 1986.

[27] Orlandi, P. "Validation of EDQNM for subgrid and supergrid models." in *Proceedings of the Summer Program 1987, Center for Turbulence Research*,p. 63, Report CTR-S88, December 1987.

[28] *Proceedings of the First ERCOFTAC Workshop*, Lausanne, March 1990, Cambridge University Press (to appear).

[29] Orzag, S.A. "Analytical theories of turbulence" *J. Fluid Mechanics*, 40, p.363., 1970.

[30] Smagorinsky,J. "General circulation experiments with the primitive equations. I. The basic experiment." *Mon. Weather Rev.* 91 p.99, 1963.

[31] Bardina, J., Ferziger, JH. and Reynolds, W.C.. "Improved turbulence models baased on large eddy simulation of homogeneous, incompressible, turbulent flows. Report TF-19, Dept. Mech. Eng., Stanford Univ. 1983.

[32] Robinson, S.K. "A review of vortex structures and associated coherent motions in turbulent boundary layers" In Proceedings of *Second IUTAM Symposium on Structure of Turbulence and Drag Reduction*, Federal Institute of Technology, Zurich, Switzerland, July 25-28, 1989.