Computations of turbulent flows for industrial and environmental applications

3 May 2000

Prof.dr. J.C.R. Hunt, FRS

TUDelft

Delft University of Technology

Faculty of Design, Engineering and Production
Laboratory for Aero- and Hydrodynamics

Computations of turbulent flows for industrial and environmental applications

Intreerede

Uitgesproken ter gelegenheid van de aanvaarding van het ambt van deeltijd hoogleraar in de Faculteit Ontwerp, Constructie en Productie, Subfaculteit Werktuigbouwkunde, van de Technische Universiteit Delft

Op 3 Mei 2000 door

Prof.dr. J.C.R. Hunt, FRS

Mijnheer de Rector Magnificus,
Leden van het College van Bestuur,
Collegae hoogleraren en docenten,
Andere leden van de universitaire gemeenschap,
Zeer gewaardeerde toehoorders,
Dames en Heren.

“Rector Magnificus, members of the board of governors,
Fellow professors and other members of the university community
Esteemed listeners
Ladies and gentlemen

1 THE IMPORTANCE OF FLUID FLOWS AND THE GROWTH OF CFD

The main aims of sustainable economic development are to improve living standards, secure people's health and safety and expand people's capabilities and horizons, especially through education, travel and communications. For the development to be sustainable, it is also essential to preserve and enhance the environment. These aims can only be achieved by the widespread and deep application of science and technology and advances in the practice of engineering and medicine, as well as by the more widely recognized practice of economics, management and effective politics. Fluid mechanics, with its adjunct sciences of thermodynamics and chemistry, is a striking example of a field of science and technology that is making valuable contributions to economic and social development and the solution of environmental problems. The theme of this lecture is to show how its contributions are spreading more widely and becoming more valuable and creative by the use of appropriate computational methods.

The main reasons why fluid flows are important in industry, technology and the environment are that:

(a) in the manipulation of materials most processes partly or wholly involve fluids, for example metallurgy, food manufacture, chemicals, etc., and
(b) these activities generally involve the utilization of natural fluids in the atmosphere, ocean, rivers and ground water, for example for transport (ships, aircraft), combustion (which of course relies on air), dispersion of pollutants, wind and wave energy, and for cooling buildings.

An extensive review of the role of industrial and environmental applications of fluid mechanics was written by Hunt (1).

How important is it to apply the latest research to these fluid flow problems? Would standard methods not be sufficient? The answer is that the considerable resources put into research and development studies of fluid motion, together with those in other technologies (notably materials and computers), have been effective in improving products and in helping to create new markets. There have been notable technological breakthroughs:

1. In material processing, new extremely strong composite materials are made as a result of mixing liquid metals or the use of metallic sprays.
2. Engines of ships, planes and vehicles have more than doubled their fuel efficiencies and, in the case of space vehicles, the engines have become significantly more reliable.
3. The fluid dynamic drag of aircraft, trucks and ships have been reduced by factors of three or more by careful design (and new lighter materials).
4. The aerodynamic noise of aircraft and cars has likewise been reduced.
5. Weather forecasts for five days are now practical using the largest computers, running currently at about 10^{11} flops.
6. New bioengineering fluid flow devices have been produced for saving, prolonging and enhancing human life.

Even in industries where the design and the technology of fluid flow systems have changed little over the past 10 years there have also been considerable economic and environmental benefits simply from the wider understanding of fluid flow processes, the wider use of computational models and the faster and more reliable design of fluid flow systems; for example, in the design of mecanical pumps, filters (see Fig. 1) and mixers, and in engineering and aeronautical engineering, less time is now spent on experimental testing or wind tunnel tests. Indeed, some commercial aircraft have been designed using only computational methods. In environmental studies calculations can now reliably be made of pollution levels from proposed industrial/urban developments to within a required accuracy. These advances in computational fluid dynamics (CFD) have arisen from the development in basic understanding and modelling of fluid mechanical phenomena, especially turbulence, two-phase flows (for example gas-liquid mixtures), combustion and chemically reacting flows, and then their simulation in practical computational schemes. The improvements in numerical methods have been a vital part of this progress. The term CFD is used here in a more general and fututristic way, to include all practical computational methods and turbulent flows, which near extend to Large Eddy simulation methods codes for turbulence spectra etc.

Since the main practical applications of CFD are in assisting the competitive provision of goods and services, those practising, managing or commissioning CFD need to understand the capabilities and limitations of different methods. They should also know about what progress in practical CFD to expect in the next few years on the basis of current research work.

They should probably ask some of the following questions. What kinds of complex flow involving mixtures of fluids with different compositions will it be possible to compute, and with what accuracy? Will one single type of code be suitable for all applications? Will improved computational techniques and new techniques for measuring and contrasting flows lead to quite new technological possibilities? How is Europe addressing this challenge?

This lecture has benefitted from reviews by Rodi **(2)**, Launder **(3)** and Holmes et al.**(4)** from practitioners of CFD and the conclusions drawn from recent meetings on turbulence Hunt (5), Nieuwstadt (6). There seems to be a good degree of consensus, which I have attempted to reflect in this lecture. I have also aimed to use as little mathematics as possible!

2 MODELLING TURBULENT FLOWS

2.1 Current ideas about turbulence

The first question to ask about a flow is about the fluid or fluids involved, for example whether it simply involves a single liquid or gas, or a mixture, and whether thermodynamic or chemical processes are involved. Assuming there is a single homogeneous fluid moving at speeds much less than the speed of sound (compressibility and other complex fluids and processes will be touched on later), one should then ask whether the flow is dominated by inertial or by viscous forces, or, in engineering terms, whether it is an aero/ hydrodynamic flow or more like a lubrication type of flow.

To answer this question we need to know the value of the Reynolds number, Re, in the particular flow in question (Fig. 2a). This dimensionless quantity is defined by the formula

$$\mathrm{Re} = \frac{U_0 L}{\nu}$$

in terms of two quantities of the flow itself, an average velocity Uo, and an 'average' distance, or 'length scale', L, over which the velocity changes (for example the width of a jet or a pipe), and in terms of a relevant property of the fluid (which is approximately independent of the flow), namely the kinematic viscosity v (equal to the ratio of the viscosity to the density). The magnitude of *Re* in engineering flows varies from 10^{-2} in some lubrication problems, to 10^2 in typical power hydraulics, to 10^6 for airflow round an aircraft or small ship [see reference **(7)**].

As the Reynolds number increases, the nature of these flows changes so much that quite different approaches to their computation are necessary. The first most characteristic change is that of the smallest distances over which the velocity changes decrease (because the gradients in velocity increase as the relative effect of viscous stresses decreases), so that more detailed (and therefore more computationally expensive) calculations are necessary (for example the distance over which temperature at the front of a turbine blade decreases is of the order of $L(\mathrm{Re})^{-1/2}$ or typically one-hundredth of the thickness of the blade in operating conditions). The second characteristic change as Re increases is that the flow becomes unstable in the sense that the flow pattern changes and fluctuates. With further increases in *Re,* these perturbed motions themselves become unstable so that the velocity becomes progressively more chaotic, in the sense that it is unpredictable in its spatial variation and over time. For high enough, but finite, values of *Re,* this process is continuously replicated, leading to fully developed turbulence in some part of the flow, for example along an aircraft wing or, possibly, everywhere, as, for example, in a stirred tank.

The flows in which this transition to turbulence occur are generally shear flows, such as jets, wakes, pipe flows, shearing flows between boundaries, etc., because they persist for long enough for the initial instabilities to build up into turbulence as they travel downstream. The critical Reynolds number $\mathrm{Re}_{\mathrm{crit}}$ is quite low in 'free' shear flows where there is no boundary to suppress the fluctuations [for example $\mathrm{Re}_{\mathrm{crit}} \sim 10$ (to within a factor of three) for jets and wakes], while in flows near rigid boundaries $\mathrm{Re}_{\mathrm{crit}}$ is much greater (for example $\mathrm{Re}_{\mathrm{crit}} \sim 10^3$ for boundary layers and pipe/channel flows). When the Reynolds number is increased above its critical value, the nature of the turbulence changes; most notably, as in any other kind of motion, the length decreases over which the smallest scale variations occur, which in this case corresponds to the sizes of the smallest edies.

So what is a turbulent flow? It consists of a large set of eddy motions, such as vortices, which are only loosely correlated with each other and which range in size from those comparable to the mean flow itself, for example the width of the jet, to those at the smallest scales (as explained above). This ratio of sizes might be about 10^2 when $\mathrm{Re} \sim 10^4$ (for example in an engineering pipe flow) and rise to 10^3 or 10^4 for jet exhausts or atmospheric flows. Also the larger eddies persist for a substantial time as they are carried along the flow (for example for a distance equivalent to more than 40 diameters along a pipe), whereas those on the smallest scale are distorted and evolve quite rapidly. Even though there are laminar flows that are unsteady and may even be unpredictable (for example rising bubbles in oil), turbulence usually poses special engineering problems, and also creative possibilities, by changing the nature of the flow to one that is unsteady and random on a wide range of scales and frequencies.

Lewis Fry Richardson in 1922 described the essence of the dynamical interaction between these eddies in his famous parody of Swift's rhyme on fleas:

> Great whirls have little whirls
> that feed on their velocity and
> little whirls have lesser whirls
> and so on to viscosity
> (in the molecular sense).

Engineers need to understand the features of turbulent motion and allow for it in their calculations; but only to the extent that it has a significant effect for their particular problem. In some flows, or in particular local regions of a complex flow, the turbulence is merely a small disturbance on the mean motion-a kind of molecular motion, and has little engineering impact. However, this is not generally the case, even though turbulent fluctuations (whose r.m.s. value is denoted by u') are usually small compared to the mean velocity U, which is usually defined as the velocity averaged over time in a steady engineering process or over many cycles in an unsteady process such as an engine. In typical engineering flows, the r.m.s. magnitudes of the fluctuations u' are about 15 per cent of U, although they sometimes rise to about 80 per cent in flows behind baffles, buildings or other bluff obstacles. The reason for weak turbulence having a significant effect is that along the length of the flow or over its duration, just like the much smaller stresses caused by molecular viscosity, the effective 'Reynolds' stresses caused by the fluctuations change the mean flow, and hence determine the critical engineering parameters, such as pressure drop or surface drag. Because the *sizes* of turbulent eddies are so much larger, of the order of L, than those of molecular motion (in a gas the mean free path - 1 μm), the turbulence always significantly increases the rate of heat transfer, mixing, etc.

Practical methods of calculating these changes, from Boussinesq **(8)** onwards, have usually been based on drawing an analogy between the motions of eddies and of molecules in a gas (for example that eddying simply leads to a larger value of viscosity or of thermal diffusivity). From the pioneering atmospheric and laboratory studies of Taylor **(9)** and Prandtl **(10)** it was clear that this approach could only be applied to a limited range of flows. However, it has only been in the last twenty-five years that measurements and direct computations of turbulent flows have been detailed enough to enable students of turbulence to specify these limitations and, where it is relevant, to propose new concepts and better practical models.

The new ideas and the types of models used can best be understood from the answers to two basic questions:

1. Is turbulence a universal state of nature, with general laws of behaviour or equations of state governing its statistics, similar to those of the behaviour of gas molecules?
2. If not, can turbulent flow be classified into different *types* or subclasses, for example distorting flows (such as flows impinging on to an obstacle) or shear flows (in a pipe, jet, etc.)? In that case each flow of a given type should have certain similarities, whether in their statistical descriptions (or equations governing the variations of these statistics) or in the form of the eddy motions.

The answer to the first question might have been a qualified 'yes' forty years ago; for example, Landau and Lifshitz (11) stated that turbulent flows *tend* to some general state in certain ideal conditions. However, current fundamental and applied research in turbulence has shown that in quantitative terms the answer is 'no', even though all well-developed turbulent flows have certain qualitative statistical and physical properties **(5)**. Notably these are: (a) three-dimensional random motion on many length scales (for future reference let these three velocity components have root mean square values u', v', w', and let the specific kinetic energy be $K = \frac{1}{2}\rho u_0^2$ where $u_0^2 = u'^2 + v'^2 + w'^2$); (b) the energy spectrum E(k), of different sizes of eddy ~ k^{-1}, has a single maximum, corresponding to the dominant eddy motion with length scale L_x, (see Fig. 2b); (c) the eddy diffusivity D_e, that defines the diffusion of heat or matter, is finite and of the order of $u_o L_x$. The reason why the answer is 'no' is because these statistical properties can differ significantly between flows; notably there are differences in the values of the ratios of different r.m.s. components u′/v′ and in the forms of the spectrum E(k) [these affect the relations between the rate of viscous dissipation per unit mass e, and u_o and L_x as defined by the ratio $e/(u_0/L\ )$] and between u_o, L and the turbulent diffusivity De, viz. $De\ /\ (u_o\ L_x)$.

The reason why these statistical properties and differential equation models of turbulence statistics vary between different types of turbulent flow is essentially because turbulent eddies are large and not like gas molecules: firstly, because of their size, the eddy scales tend to be comparable with those of the mean flow scales L, and, secondly, because they have a significant 'memory' or correlation time-scale T_L which is determined by how long it takes for large eddies of scale Lx and velocity u_0 to interact with each other; it is found experimentally that T_L - L_x/u_0. In unconfined flows such as jets or wakes, L_x increases and u_0 decreases along the length of the turbulent flow and therefore the memory time T_L increases. In fact, it increases at the same rate as the time (T) for an eddy to travel along the flow, that is $T_L \sim T$. Therefore the statistics of the larger energetic turbulent eddies are always partly dependent on how the turbulence is initiated.

On the other hand, if the turbulence is confined, for example in a pipe, the statistics of the flow reach a steady state, so that L_x and u_0 do not vary along the flow. Then T_L is much less than T, and the initial state of the turbulence is forgotten **.(12)** However, in this case the larger eddies are limited by the size and shape of the pipe and therefore cannot be universal. Thus, either because of its growing 'memory' in unconfined flows or because of its distortion by particular boundaries, the large-scale turbulence cannot have a completely general form. Note that, although the current answer to the first question is 'no', it does not contradict Kolmogorov's **(13)** hypothesis that *certain* aspects of the small scales of turbulence have a universal statistical structure-a result that is of great practical value for models of chemical mixing and the propagation of waves through turbulence [see the reviews in **(14), (5)]**.

If turbulence does not reach a universal state by internal random motion, why should one expect, even in a particular 'type' of flow, any features in the largescale eddy motion to be the same or to be modelled by similar methods in different flows. As shown in Fig. 3, based on laboratory flow visualization studies and direct numerical simulations, the answer is that characteristic large-scale eddies-or 'coherent structures'--do indeed form in a distinct way in different 'types' of flow, such as those affected with mean shear (that is where $\partial U/\partial y \neq 0$) **(15),** with curvature in the mean streamlines, with rigid boundaries, or those affected by stable or unstable buoyancy forces **(12).** In each case it is found that the form of the eddies is broadly the same irrespective of how the turbulence is initiated or how it enters the flow region being considered.

> In other words, there is a tendency towards self-organization
> rather than statistical equilibrium.

In the case of shear flows the eddies are elongated vortices sloping in the direction of the shear, which in curved flows are parallel to the mean motion. It is therefore natural, as well as being theoretically justified [for example reference **(16)],** to expect that certain turbulence statistics, or the approximate 'turbulence model' equations used for their derivation, are similar for all flows within each of these 'types'.

2.2 Turbulence mechanisms that need to be 'modelled'

Having noted the need to estimate certain basic statistics of turbulence in order to calculate the practically important aspects of engineering flows one should go on to review the parts of the CFD models that provide these statistics. Use is made of the fact that turbulence has particular features in different types of flow to describe the key mechanisms that determine how the turbulence statistics vary in these flows (see Fig. 4).

2.2.1 Production

Where there are variations of the mean large-scale flow (with mean shear/strain components such as $\partial U/\partial y$ or $\partial V/\partial y$), each of the vortices in the eddies are stretched or compressed, leading to an increase in the variance of some velocity components (for example the transverse component v'^2 in an accelerating flow, where $\partial V/\partial y$ is negative) and a decrease in others, at a rate proportional to this variance and to the mean strain (for example $-v'^2 \partial V/\partial y$) (There are obvious analogies to the increase in the energy of elastic solids under strain.) In most engineering flows the largest rate of strain is caused by mean shear $\partial U/\partial y$, where the eddy vortices are stretched and rotated. The net effect is an increase in all the turbulence components, u′, v′, w′.

2.2.2 Dissipation

As turbulent eddies interact they may merge, leading to larger eddies, and hence an increasing 'memory time' T_L but they also tear and distort each other so as to generate a cascade of energy to smaller scales which, as explained already, determine the rate of dissipation by viscous stresses **(16)** viscosity. This is why the rate of dissipation ε can be modelled in terms of the large-scale turbulence and of variations in the mean flow that strain it, without needing to model the details of the small scale. However understanding these unsteady motions involving the life-cycle of small scale eddies **(19)** is necessary to estimate mixing, chemical reactions and two phase flow processes). The rate of dissipation ε is of the order of the ratio of $u\square_0$ to the length scale L_x but the numerical factor varies

(typically by a factor of two) between different types of flow. When turbulence is in a state of 'local equilibrium' the rates of production and dissipation are in balance, as occurs in turbulent boundary layers near a rigid surface.

2.2.3 'Diffusion' of turbulence

Intense eddy motion in one region of the flow diffuses outwards into other regions of the flow as a result of the self-induced motions of the vortices and their mutual interactions. This is why the thickness of wakes and jets increases along their axis. At the same time there is a mixing between fluid that is turbulent and the external fluid that may have a different temperature or concentration, which is not turbulent. This takes place at the randomly moving but very thin interface between rotational dissipative motions on one side and irrotational velocity fluctuations on the other. Most mixing occurs where the interface rolls up and engulfs the external flow. The result of these random motions is a net transport of turbulent energy outwards at a mean boundary velocity E_b of the order of u_0. This transport phenomenon is modelled as a diffusive process proportional to the mean gradient of u^2_0; except very close to the interface, this is a valid approximation inshear flow, for example at the outer edge of jets. But this is not necessarily a good approximation for transport in flows that are far from equilibrium and with rapid variations in length scale, such as near bluff obstacles where there is a strong production of turbulence by straining motion.

2.2.4 Action at a distance

In many practical flows there are adjacent regions where the mean flow and turbulence are quite different, for example when the wake of one row of turbine blades impinges on to the boundary layers of a downwind blade or where a sideways jet enters a larger volume (Figs 4 and 1). The vortices, say with length L_x in the adjacent regions of turbulence can directly induce eddy motions in the other region over a limited distance (of order L_x); this 'action at a distance' is generally reduced by the sheltering action of strong shear. The spreading of these externally induced motions have to be considered using the concepts and models of the previous self-induced energy transport. As a result of these mechanisms, the free stream turbulence outside the boundary layer on a turbine blade amplifies the turbulence within the layer after a certain streaming distance, it can also trigger the onset of turbulence. Either of these interactions tends to thicken the layer and change the profile U(y) causing a sharper gradient near the surface so as to increase the skin fraction. By contrast external turbulence has little effect on the pressure fluctuations in a wake or below roughness elements **(21).** Understanding and modelling these mechanisms is proving to be one of the greatest challenges for CFD **(22).**

2.2.5 Rate of change and advection of turbulent energy

When turbulence is not in a state of local equilibrium and the diffusion of turbulence is not balancing the difference between local production and local dissipation, it means that the turbulent energy locally is changing, either with time or spatially, as it is advected into or away from the local region by the mean flow. This occurs in turbulent wakes of obstacles and in most shear flows except very close to boundaries.

2.2.6 The effects of rigid walls and gas-to-liquid surfaces

Since on any rigid surface the velocity is zero relative to the surface, the velocity fluctuations are zero. However, because of the large variations of velocity over small distances (associated with high Reynolds number flow), there are significant velocity fluctuations very close to the surface; indeed, the magnitude of the energy of the parallel component (u^2) has its maximum value at a distance above a smooth surface of the order of 10^2 *L/Re* (or less than 1 mm for a large gas pipe) where the viscous stresses are still significant. These rapid variations in mean velocity and in the turbulence are associated with similarly rapid variations in the rates of production and dissipation relative to their local maxima. As flow visualization shows, the intensity of these fluctuations is caused by local 'bursting' instabilities of the eddy motion near the wall **(23)**. When Re is much greater than 10^4, each eddy interacting with the surface produces its own internal layer in which the lengths of the vortical structures may extend over 10 boundary layer depths **(5).**

Despite the rapid variation in the form and scale of turbulent eddy motion in this region, because the local dynamics are dominant, the 'diffusive' effects are relatively less significant. Aspects of these motions have still to be understood, especially, for example, how large eddies do or do not affect, through 'action at a distance' or shear sheltering the fluctuating contribution to surface shear stress and how surface roughness elements change the turbulence around them. At a gas-liquid surface, the mechanisms are different; there are no rapid variations in the rates of production and dissipation because the mean shear dU/dy is small. However, as also occurs near a rigid surface, the distortion to the turbulence arises from the 'blocking effect' acting on the larger scale motions, especially those normal to the surface.

2.2.7 Adjustment of anisotropy

There are usually significant differences in the strength of the three components of turbulence (u', v',w') and in the length scales in the different directions. These may originate from the initial or upstream state of the turbulence or the anisotropic production in a straining flow, or the blocking effect of the boundaries. In most flows, notably shear flows, pressure fluctuations tend to transfer energy from the most energetic to the least energetic components. However, because turbulent eddy motion is not like that of gas molecules, this is not a universal tendency, as vortex dynamics readily explains. Second-order models now reflect this non-universality **(24)**).

3 REQUIREMENTS OF CFD CODES

3.1 Output and other types of requirement

Any practical method of calculation should be designed so that its output is compatible with data available and so that the method is appropriate for the user in terms of its operational convenience and availability of necessary resources (see Table 1). It is also necessary to define the flow 'domain' or volume where the calculation is to be performed (see Fig. 5). This decision depends on other factors, such as knowing the flow entering the domain and the computational capacity available.

For many users of CFD codes the only output required is information about the broad features of the mean flow pattern and quantitative estimates of the variables, such as mean flow-rate, pressure drop or heat transfer rates. This approach supplements or replaces the more traditional engineering approximate calculations based on non-dimensional coefficients, for drag or heat transfer, or those based on equivalent one-dimensional integral equations which cannot account for variations across the flow. Well-designed and user friendly CFD codes enable the designer of engineering devices to examine the consequences of various designs on the fluid flow aspects of their performance. Figure 1 shows the results of simplified calculations of flow through a liquid filter with a central baffle plate. Showing how the flow *changed* as the location of the baffle plate changed led to a decision on the optimum design. Although only a basic level of output may be required (which is denoted as level 1), depending on the type of flow and the accuracy that is required, quite different levels of input data of complexity of the model are necessary (which are discussed below).

At the next 'level' (2) in terms of improved output for practical flow calculations either the mean flow has to be considerably more accurate (for example the drag of a wing calculated to within 1 per cent) or basic statistical features of the turbulence are required (such as the intensity of turbulence and its broad effects on combustion and mixing).

To provide the third 'level' (3) of output, calculations are required for more complex aspects of the turbulence, such as the variation in the spectrum of energy, E(k) (which is necessary to calculate unsteady loads on structures or the production of noise), or the form of the eddy structures of turbulence (for example because of their influences on bubbles and particles, in two-phase flows (**25**) and on the efficiency of combustion devices).

3.2 Input

Having decided on the required level of output, the next step in planning the use of CFD is determining whether sufficient input data are available to perform the necessary calculations. Even this is not a straightforward decision because it depends on the nature of the flow within the flow region to be calculated as it enters. In two common situations, the input is known; the first is where the entering flow is effectively non-turbulent (for example for an aircraft in flight) and the second occurs when the entering turbulence has a well-established form whose details are well known (for example flow from a straight pipe section entering a complex flow region (as in Fig. 1). More often, however, the input flow is only known rather approximately (as in Fig. 5). The practical reasons may be because of the lack of specification of other components in a design or because it may be difficult, or costly, to measure or possibly calculate it, such as the flow within the curved pipes entering an internal combustion engine. In these cases assumptions may have to be made using data for comparable flows.

For most calculations the input data are required at a comparable level of detail and accuracy as the output data. However, there are situations where it may be necessary to have the input data available in *greater* detail and at a *higher* level (for example the turbulence length scale) than is required for the output, simply because a higher level model has to be used to ensure the required degree of accuracy in the output (for example in the mean flow quantities).

How do the practitioners of CFD answer the basic question about the sufficiency of data? It is necessary in principle to know about the sensitivity of (a) the calculation method to type and detail of data input and (b) the given flow. For example, to calculate the mean velocity in the wake downwind of an obstacle (an aerofoil or a plate), even to an accuracy of 50 per cent, it is necessary to specify some details of the turbulence in the wake near the obstacle [for example see reference **(3)]**. The simplest level of model cannot make use of this input data and therefore has inherent errors. Recent 'blind' tests of CFD (level (**21**))codes have shown how for certain flows, especially those near sharp boundaries, the calculations are very sensitive to the specification of the input turbulence **(22).** A user of this code might then be advised to evaluate the sensitivity of the calculation and consider a range of input data before making any design decisions. However, such precautions tend to be regarded in practice as 'expensive luxuries'. In that case beware of disappointment in the results!

3.3 Turbulence models

It has been seen that, depending on the practical problem to be solved, different 'levels' of output data are required; the form of the input data depends on the output requirements and the turbulence model to be used (see Table 1).

First consider problems with a level 1 output. If the form of the mean velocity profile of a turbulent flow is changing slowly, that is U(y) varies with x (say over 30 boundary layer depths or pipe diameters), so that the forms of the eddy structure do not change significantly, it is quite usual in engineering and environmental work to base calculations for the mean profile *U(y)* and Reynolds stress (τ) on Prandtl's mixing length model. This is a simple equation relating τ to $|\, dU/dy \,|$ and to a length scale (of the order of the correlation length, L_x) that has to be *assumed* at every point (x, *y, z)* in the flow. The mixing length equation and its variants [for example the Spalart Almaras model (**26**) used in aeronautical boundary layers] all involve a mathematical relation between, certain properties of the mean velocity vector U, the position vector **x** and coefficients that have been derived for particular types of flow, for example shear flows near a wall. These coefficients may have been derived empirically, or by calculation using higher level models or numerical simulation. For modelling the flow very close to the wall, where stresses caused by molecular viscosity are significant, the kinematic viscosity υ enters the relation.

The mean flow is obtained by solving together the differential equations for the mean momentum (which involve both τ and U) and for continuity (for conservation of matter), together with the mixing length algebraic equation. The solution requires as input the mean velocity U and the boundary conditions on the walls, say at y = 0, have to be specified. If they are smooth the no-slip boundary condition applies; if they are rough and perhaps mobile (like a water surface) some wall 'boundary conditions' have to be derived empirically to specify the solution, either empirically (eg by specifying a roughness length) or by coupling the fluid flow problem to a model of the physics at the boundary.

As with any practical computational model, it tends then to be applied to more complex types of flow; it is generally found that inaccuracies grow the greater the difference in the way that the turbulence is distorted and adjusts compared with the original 'type' of flow. Thus if equations for a flat boundary layer are applied to flows over undulating walls or boundaries, or for gas flows over water waves (a problem for chemical engineers as much as oceanographers), as the undulations steepen the distortion of the flow increases and fluctuates, so that the errors using the mixing length increase. Another case where the mixing length approach cannot be applied is the common engineering problem of turbulent flow in non-circular pipes.

The next level of turbulence model most widely used in practice is similar to the mixing length model in that there is a relationship between the mean shear stress τ and the local gradients of the mean velocity (that is dU/dy in a shear flow), but the difference is that it explicitly models the dependence on the kinetic energy K and length scale L_x ($\sim K^{3/2}/\varepsilon$) of the turbulence. (Thus in a shear flow

$$\tau \simeq - C_\mu \,(K^2/\varepsilon)\, \partial U/\partial y \qquad (1)$$

where $C\mu$ is a coefficient.) The other important feature of the model is that it represents the way in which the turbulence usually develops at a different rate to that of changes in the mean velocity as a result of the different mechanisms affecting the turbulence and those of the mean flow (see Section 2.2). These differences in the 'rate processes' require the introduction of two new partial differential equations for the kinetic energy K and for the dissipation rate ε, which together effectively define the length scale of the turbulence, as explained in Section 2.1. The two equations for K and ε , first introduced by Kolmogorov **(14)**, were developed for widespread engineering practice in the 1970s by Launder, Spalding, Rodi and their colleagues at Imperial College [see references **(27** and **(28)]**. The equations contain terms that model most of the dominant processes. The mathematical forms of the terms and the coefficients involved are based respectively upon physical arguments and comparison with a range of experiments (particularly shear flows) (see Fig. 6). In principle this method requires details of the turbulence, υιz K and ε, for its data input which are not required for level 1 models. In many cases they are not available, and therefore have to be estimated based upon knowledge of similar entry flows. Since this model is often only required to provide level 1 output, some uncertainty and rough approximations about input data are allowable, given this level of user requirements. Mean flow patterns calculated using this model may well be broadly correct, even where there are indented boundaries and where there are a number of inflows and outflows into the flow region, such as an aeroengine combustor **(30).** However, calibration of the code with a similar flow is generally desirable. A notable success in the use of this model was the correctness of the flow pattern and path of the flame front that was calculated to simulate the events in Kings Cross underground station during the fatal fire in November 1987 **(31).**

Recent discussion between industry and the research community [5] have led to conclusions about the limitations on the use of the K-ε model in flows where the mean velocity gradients are changing in magnitude and directions, such as in three-dimensional shear flow over a curved surface (for example over swept wings) or where there is intense, anisotropic turbulence that significantly affects the mean flow (for example in certain wake flows or distorted jets; see Fig. 7) or in flows with strong swirl, especially where they impinge at a stagnation point, or where the turbulence intensity changes rapidly over distances much less than the scale of the large eddies (as in thermal convection) **(35).** Then it is preferable to use a more complex model involving fewer assumptions, but up to seven extra model equations for the separate components of the Reynolds stress $u_i\, u_j$ of the turbulence, together with an equation for the dissipation rate for ε.. The forms of the equations for models of this type first developed by Launder *et al.* **(36)** and by Lumley **(37)** continue to undergo developments, either to extend the applicability of the models (for example to flows with higher 'strain' rates, greater anisotropy or wider ranges of Reynolds number) or to improve the accuracy of the model for particular types of flow (for example those with stable density gradients, which have internal wave motion as well as turbulence).

These second-order models are less widely used in engineering and environmental flows than the K-ε models, because they require more detailed input data (or more assumptions) and are more sensitive to them. Also they require significantly more computational effort and can be more sensitive to the computational methods used in their solution. Probably their greatest contribution to the practice of turbulent modelling has been in their highly reduced and simplified form of ‚algebraic stress models', when the equations give rise to a useful relation between the different stress components (τ, u'^2 , v'^2, …, etc.) and the local mean velocity gradients. This is a marked improvement on the simple eddy viscosity form used in the K-ε model. Both types of level 2 model are only appropriate for the types of flow where the scale of the most energetic turbulent eddies are of the same order or smaller than the distance over which the mean velocity is changing (this is satisfied in a shear flow where $L_x < (|\, dU/dy| / |\, d^2U/d\, y^2\, |)$ Also it is necessary to neglect the effect of action at a distance of larger eddies, which is particularly important near boundaries or in the presence of large-scale 'free-stream' turbulence outside a shear layer [see Fig. 8 and reference **(40)].**

To illustrate how certain features of turbulence structure are common to similar types of flows, consider how the dissipation rate ε and the relevant integral length scale vary in slowly varying shear flows U away from and close to boundaries. This example also illustrates the problem in turbulence models of relying on local relationships. In a wide range of shear flows U(y) (away from boundaries) the dissipation rate is related to the mean velocity gradient by $\varepsilon \cong \beta_s (v'^3)/L_\varepsilon$, where $L_\varepsilon^{-1} \sim A_s <dU/dy>/v'$. Note that the expression < > denotes an average value over a distance of order L. (this averaging is only significant in shear flows where locally $dU/dy \simeq 0$). However, as shown in Section 2.2, at a distance y from a rigid boundary, the length scale of the velocity v normal to the boundary, defined as $L_x^{(v)}$ (which is approximately equal to L_ε), is also influenced by the blocking effect. In a wide range of shear flows near boundaries these two effects can be combined in a single formula (for very high Reynolds number flows, outside the surface viscous layer):

$$1/L_\varepsilon \simeq 1/L_x^{(v)} \simeq A_B/y + A_s <dU/dy>/v'$$

where < > denotes an average value. This effectively combines Prandtl's concept of a local model for turbulence near a wall and a spatially averaged model for shear layers, wakes, etc. **(12, 41).** A similar approach to modelling length scales in turbulence models has been introduced by Orszag *et al.* **(42)** based on a purely statistical physics concepts of turbulence.

The eddy structure in turbulence changes when it is driven by thermal convection or in strongly accelerationg flows. It produces a different value of the coefficient β_s and in the latter case very different values (at a distance y from the surface) of the ratio $\varepsilon y/(v'^3)$**(33).** Differential equation 'turbulence models' can usually account for quite rapid changes in the variation of the different velocity components across different flows; they

rely on the 'diffusion of turbulence' to model some mechanisms that may correspond more closely to those caused by 'action at a distance'.

The limitations of local models mean that a reasonable accuracy of level (2) output cannot in some circumstances be obtained by level (2) models. Turbulent flows round obstacles are a good example (see Fig. 8). This requires a higher level (3) model based on a few, more generally applicable, assumptions. For the physical reasons explained in Section 2.1, the only reasonably general assumptions about turbulence are those concerning the smallest scale motions. This is the basis of large eddy simulations (LES) in which finite size motions are directly calculated while the smallest scales are not calculated in detail but only approximately modelled.

However, if no assumptions are made about the statistics of the 'resolved' or largest scale motions greater than those of the assumed 'sub-grid scale motions', it means that their random space-time variations must be calculated directly and any required statistics must then be derived from the large periods of integration (say 1000 natural time-scales of the turbulence). These calculations or 'simulations' can be performed at different levels of accuracy depending on the relative size of the sub-grid motion to the resolved motion. However, generally this approach requires large computational resources and/or longer periods to perform the calculation. For moderate Reynolds numbers ($\lesssim 10^4$ for a pipe flow) it is possible (if 10^2 hours are allowed on a machine operating at 10^9 flops) to compute the flow with complete accuracy (provided the initial conditions are known) without assuming any sub-grid scale motion this is a direct numerical simulation (DNS).

A comparison is shown in Fig. 11between the numbers of grid points required for computing a significant engineering flow using LES and DNS. It is clear why DNS is not yet a practical option, even using the largest computer systems. Both LES and DNS models can also be used to calculate other statistical information about the eddy structure of the turbulence, for example the extreme values of fluctuations (of great importance in environmental models) or how the velocity is correlated over intervals of time and space, and its spectrum.

Reviewing models at different levels shows how more computation is required as the model level increases or, given the same computer capacity, how only relatively idealized turbulent flows can be computed with a high level of accuracy and detail (see Fig. 45).

3.4 Numerical methods

All the 'governing' partial differential equations (PDES) of CFD (momentum, continuity and turbulence models) assume that the statistical variables (such as moments of velocity, pressure, etc.) are continuous functions of space and time. For all but a few very simple situations, it is necessary to solve these equations approximately by defining the variables, say u(x), as a series of values (u_n) at a number of discrete volumes (or finite elements). Then (with finite difference methods) the derivatives in the PDEs can be re-expressed as differences between the nodal values at different points [for example (u_n - u_{n-1}), etc.]. This converts an insoluble calculus problem for deriving u(x) into the soluble problem of finding the variables u_n . in a large set of algebraic equations.

Increasing the number of points, and therefore reducing the distances between them [for example (x_n - x_{n-1})], in general brings the solution of the algebraic equations closer to the actual solution of the PDES. Nevertheless, however large the number of points used, which may exceed 10^7 in some large aerospace calculations, some errors always exist. Users of CFD need to be aware of them and what needs to be done to minimize them, or at least allow for them, in assessing computational results.

Firstly, even small numerical errors can lead to significant errors in the solution to the flow problem, especially where the flow is affected by regions with sharp changes in the shapes of boundaries [for example rounded blades in a stirred tank reactor **(46)]** or where the flow region and/or flow time are large enough that errors can build up. The errors are similar to those that are caused by an insufficiency or inaccuracy of input data, whether of the velocity entering the flow region or of the shape of the boundaries. They may even lead to chaotic and unpredictable solutions **(47).**

Secondly, different numerical methods (for example the way that derivatives are approximated) have their own particular types of error and also make different computational demands (see Fig. 12). Therefore, in assessing the performance of any CFD code it is essential to know which numerical methods have been applied and also the size and nature of the grid. The latter is becoming increasingly important as new methods are developed for distributing the grid points through the flow region. In some cases quite simple improvements in the grid point distribution in CFD codes have led to significantly greater accuracy for the same number of grid points. Can such improvements be predicted in advance or planned? Formal mathematical methods may facilitate the predictions in advance of the benefits of new numerical schemes (an approach practised more strongly in France). However, the usual approach is empirical, especially because the benefits are not generally applicable to all flows and are generally related to the particular model equations. [A notable example at the Meteorological Office was the 30 per cent reduction in the error of the forecast position of tropical cyclones obtained by improving the model and the numerical scheme **(48).]** An important new idea is to have part of the grid moving, for example with rotor blades **(46)** or with shock waves, or for coupling and oceanographical models in intense storm meteorology.

The third point to be aware of is that the comparative testing (or 'validation') of different codes against experimental data is only meaningful when the numerical method (including the precise form of discretization) is also specified; preferably it should be the same in both cases. **(5)**

4 DEVELOPMENTS IN CFD

4.1 Operational and resource questions about CFD codes

The wider and deeper application of CFD for practical engineering is following the same path as that of other branches of science and technology. First quite complex ideas and theories are converted into algorithms and robust computer codes. These need to have been thoroughly tested and quality assured, so that the user can follow all the intellectual and practical steps in producing the delivered code (Fig. 13a). In general such codes have to be produced so that they can be used by operators who only have a superficial understanding of all the ingredients and whose main job is to run the model for specific flow problems and then apply the results for particular purposes (see Fig. 13b). The latter step usually has several components: one is an assessment of the accuracy and general reliability of the model output for this particular problem (based on a previous similar calculation and a knowledge of similar flows), and then either an interpretation or communication of the result to those who wish to use the result. For this stage to be effective usually requires the fluid flow expert to have a good understanding of the technology and general requirements of the model user. Discussion with engineering and environmental practitioners of CFD show that their assessment, communication and application of the results is largely based on combining an understanding of the basic scientific principles with a case-by-case knowledge of how any particular CFD scheme works. The interpretation is seldom in practice influenced or helped by detailed knowledge of how the algorithms and code were constructed.

The intuitive insight of CFD users, of fluid flow designers and of those involved in practical environmental fluid flow problems (such as weather forecasts) is rapidly improving with developments in graphical presentation of computer results **(49)**. One possibility now being explored is the use of 'virtual reality' so that the user can visualize being in the fluid flow field, hurtling round the vortices or 'pausing' in a stagnation region, etc. The appropriate use of these visualization approaches is only just beginning **(38).**

It is now widely recognised that turbulent flows tend to fall into particular types. Therefore the users of CFD need to interpret the results for particular applications, and engineers specializing in a particular technology associated with certain types of flow should ensure that their CFD system has been developed and tested for these types. To become effective designers they need to become familiar with the use of the CFD system as a natural adjunct to their other techniques.

What level of resource, however, should be appropriate for this adjunct to engineering (and environmental) design? For those dealing with the development stage of major design projects a vital consideration is the time to run the calculation so as to maintain a programme of trials; for example, in a grid of 100 x 100 x 100 or 10^6 points a mainframe supercomputer is necessary (operating at, say, 10^{10} flops) to obtain the answer within two hours. For other kinds of project where the pressure of time is less, work stations (currently running at about 100 megaflops) are adequate to solve lesser problems in a few hours, or major computations in days.

For many organizations, from the largest to the smallest, the decision on the resources depends just as sensitively on the cost of staff to write the code and maintain it, or to run a commercially available code and use it effectively within the organization. In either case the staff level depends on the size of the code that is necessary. If the code has to be written 'in-house', a typical estimate is that one experienced programmer takes about one day to write 25 fully validated 'bug-free' lines of code, so that the key engineering design code of a large aerospace organization, which extends to about 10^7 lines (aerodynamics, structures, thermodynamics and electromagnetics), is worth about £10^7 just for the code-not counting the resources associated with research and testing. Maintaining and updating such a code requires about one experienced programmer per 20000 or 30000 lines of code. Even smaller consulting organizations have codes with 50 000 lines and the manpower resources costs can be calculated on a similar basis. The important point is that serious financial and human resource decisions need to be taken about the level of code that is to be used, in the light of all the requirements.

4.2 Progress in CFD codes for turbulent flows

One objective measure of the progress in CFD of turbulent flows over the past 25 years has been the increase in complexity of the 'frontier' fluid flow problems that have been studied in the comparisons of the codes. These are the problems where most innovative research and code development is focused at any given time.

At Stanford in 1968, the main emphasis was on calculating turbulent boundary layers on flat plates in pressure gradients, whereas in 1980, at the next international workshop, the emphasis had moved on to more complex shear flows, such as wall jets and corner flows, and to recirculating flows at the expansions of pipes. In the same period the codes had changed. In 1968 these were essentially variants of level (1) codes (with some embryonic level (2) codes with equations for K); in 1980 the emphasis had changed to level (2) codes.

At the ERCOFTAC (European Research Community of Flow Turbulence and Combustion) workshop at Lausanne, 1990 **(16),** the frontier problems changed again to the important aeronautical engineering problems of *accurate* calculations of three-dimensional turbulent boundary layers and boundary layers in a state of transition as a result of external turbulence. Rolls-Royce donated data from their test programme for this study.

The first problems were computed most successfully with level (2), 'second-order' equations, because all the separate Reynolds stresses could be calculated. However, the transition problem was most effectively modelled at level (3) by a large eddy simulation code of Voke **(50)** which does not involve assumptions about the large-scale eddies which change dramatically at transition. However, although statistical level (1) and level (2) models for fully developed turbulence cannot be used for flows in transition between laminar and turbulent flow, it has been found that special adaptations of level (2), (K-ε) models, are suitable for engineering calculations in a specified range of flows-a clear demonstration of the benefits of focusing modelling effort rather than attempting to develop all-encompassing models for a wide range of turbulent flows. Level (3) models (especially DNS, but also 'two-point' models such as the exact idealized RDT calculation) have given valuable insight into the limitations in level 1 and level (2) models and have led to improved approximation for particular classes of flow. In other words, a 'cascade' of modelling methods may be the most cost effective strategy. Combinations of these types of modelling are being introduced for certain industrial problems **(5)**

4.3 The new 'frontier' problems

The growing use of fluid flow technology and information about environmental flows not only necessitates more accurate calculations of the mean flow (U) and basic statistics of turbulence (u', v', ...), but increasingly more extensive (level 3) information about the flow. It should also be possible to use CFD more effectively to develop concepts and better fluid flow design solutions, which may well require more extensive design integration of fluid flow and other technologies, such as those listed in Section 1.

The object of some engineering designs is to produce less turbulence (for example noise production), while in others (for example internal combustion engines) more turbulence is required of the right type; for the fluid flow calculation in both design problems, the type of eddy structure is as important as the level of turbulence. It is only recently that objective methods for defining the eddy structure have been compared (and found to be less different than was thought at first). Now the basic fluid mechanics and the technology of this aspect of turbulence can progress more systematically **(51).** Also in specific classes of flow these structures can be modelled for practical purposes at a less complex level than large eddy simulation, for example using discrete vortex models.

The eddy structure is equally important for modelling combustion, particle motion and bubbles in turbulence. Vortices are efficient at concentrating and transporting bubbles, by sucking them in. In some cases solid particles are alse trapped, because if they are present when the vortex is formed they tend to diffuse outwards at a slower rate than that of the growth of the vortex. Vortical eddies can also enhance combustion. *Ad* hoc CFD models for these flows have been developed. There is no clear consensus yet as to whether LES, although the most general method, is the most practical method for computing flow problems that are sensitive to the eddy structure. The computing power and expertise required is considerable.

It is certainly true that some complex flows, including two phases, electromagnetic body forces, strong rotation, fluid/wave/solid interactions, etc., are being successfully modelled at levels (1) and (2). However, industrial engineers (specifically in the oil and ship-building industries) state that for these applications the codes have not yet been significantly well validated to give industry the confidence it is looking for. Even the methods of validation of codes and of their inter-comparison have not yet been generally agreed. Nevertheless, although experienced engineers are using CFD codes for complex problems, in some industries doubts prevail and the use of CFD is vestigial. Overcoming this problem is where the networks of research specialists and interested industries can contribute. This is one of the major roles in Europe of ERCOFTAC. [Spalding **(52)** set out in some detail the range of problems where modelling and experimental testing of models is needed; the task is certainly not complete.]

It is appropriate to end on the most challenging practical problem in turbulence; namely can we, if we so desire, suppress certain types of turbulence by 'active control'? Ffowcs Williams **(53)** has shown that it is possible in practice to suppress noise by introducing another sound source near by in anti-phase. It is not practical in terms of energy requirements to produce 'anti-turbulence', but recent direct numerical simulation and experimental studies **(54, 55)** have shown that it may be possible to *suppress* the growth of instabilities within a turbulent boundary layer flow by appropriately forcing the flow with active elements, such as by small vertical movements of

30 **Priddin**, C. **H.** Turbulent combustion modelling-a review. In *Advances in turbulence 3,* Proceedings of European Turbulence Conference, Stockholm, July 1990, pp. 279-300 (Springer-Verlag, Berlin).

31 **Fennell, D.** *Investigation into the King's Cross underground fire,* 1988 (Her Majesty's Stationary Office, London).

32 **Huang, P. G.** and **Leschziner, M. A**. Stabilization of recirculating flow computations performed with second-moment closure and third-order discretization. Proceedings of Fifth Symposium on *Turbulent shearflows,* Cornell, 1985, pp. 20.7-21.2.

33 **Boyle, D. R.** and **Golay, M. W.** Measurement of a recirculating two-dimensional flow and comparison to turbulence and model prediction. I: steady state case. *Trans. ASME, J. Fluids Engng,* 1982, **105,** 446-454.

34 **Leschziner, M. A.** Second-moment closure for complex flows. International Forum on *Mathematical modelling of processes in energy systems,* International Centre for Heat and Mass Transfer, Sarajevo, 1989.

35 **Kenjeres, S., Hanjalic, K.** 1999 Transient analysis of Rayleigh-Bernard convection with a RANS model int. J. Heat & Fluid Flow 1999, 20, 329-340.

36 **Launder, B. E., Reece, G. J.** and **Rodi, W.** Progress in the development of a Reynolds stress turbulence closure. *J. Fluid Mechanics,* 1975,68,537-566.

37 **Lumley, J. L.** Computational modelling of turbulent flows. *Adv. Appl. Mechanics,* 1978, 26,183-309.

38 **Uittenbogaard, R.** , (Reference on internal wave modelling on Strat flows)

39 **Murakami, S., Mochida, A**. and **Hayashi, Y.** Examining the K-ε model by means of a wind tunnel test and large eddy simulation of the turbulence structure around a cube. *J. Wind Engng and Ind. Aero.,* 1990, **35,** 87-100.

40 **Durbin, P. A.** A Reynolds stress model for near-wall turbulence. J. *Fluid Mechanics,* 1993, **249,** 465-498.

41 **Schlichting, H.** *Boundary layer theory,* 4th edition, 1960, pp. 480 and 592 (McGraw-Hill, New York).

42 **Orszag, S. A., Yakhot, V., Flannery, W.S., Boysan, F.**, **Choudhury, D., Maruzewski, J.** and **Patel, B.** Renormalization-group modeling and turbulence simulations. International Conference on *Nearwall turbulent-flows,* Tempe, Arizona (Eds R. M. C. So, G. Speziale and B. E. Launder, 1993, pp. 1031-1046 (Elsevier, Amsterdam).

43 **Bradshaw, P.** Variations on a theme of Prandtl. AGARD Conference Proceedings on *Turbulent shearflows,* 1971, p. C-1.

44 **Chapman, D. R.** A perspective on aerospace CFD. *Aerospace* America, January 1992.

45 **Nieuwstadt, F.T.M., Andren, A, Brown, R.A., Graf, J., Mason, P.J., Moeng, C.-H. and Schumann, U.** Large-eddy simulation of a neutrally stratified boundary layer: A comparison of four computer codes. Selected papers from the first ERCOFTAC Workshop on Direct and Large-eddy Simulation, (P.R. Voke, L. Kleiser and J.P. Chollet, eds.), Kluwer Academic Publishers (ISBN 0-7923-3106-0), 1994, 167-178.

46 **Luo, J. Y., Gosman, A. D., Issa, J.** C., **Middleton, J.** C. and **Fitzgeraid, M. K.** Full flow field computation of mixing in baffled vessels. *Trans. Inst. Chem. Engrs,* 1993, **71A,** 342-344.

47 **Iserles, A.** Dynamics of numerics. *Bull. IMA,* 1994, **30,** 106-115.

48 **Heming, J. T., Chan, J.C. L. and Radford, A.M**. A new scheme for the initialisation of tropical cyclones in the UK Meteorological Office global model. *Met. Applic.,* 1995, **2, 171 – 184**

49 **Hunt, J.C.R.** Advances in fluid mechanics and meteorology from visualisation of numerically and experimentally generated data.
Presented at ERCOFTAC Workshop on *Visualisation of numerically generated data,* Abingdon, June 1992. *ERCOFTAC Bull.,* 1993,16.

50 **Yang, Z.** and **Voke, P. R.** Large eddy simulation studies of bypass transition. In *Turbulence modelling and measurement,* Vol. 2 (Eds W. Rodi and F. Martelli), 1993 (Elsevier, Amsterdam).

51 **Bonnet, J. P.** and **Glauser,** M. N. Preface of *Eddy structure identification in free turbulent shear flows* (Eds J. P. Bonnet and M. N. Glauser), 1993, pp. xi-xiii (Kluwer, Dordrecht).

52 **Spalding, D. B.** The provisional scientific programme of ERCOF *TAC. ERCOFTAC Bull.,* 1989,2,5-7.

53 **Ffowcs Williams,** J. E. Anti-sound. *Proc. R. Soc. Land.,* 1984 **A395,** 63-88.

54 **Choi, H., Moin, P.** and **Kim, J.** Active turbulence control for dral reduction in wall-bounded flows. *J. Fluid Mechanics,* 1994, **262** 75-110.

55 **Fan, X., Hofmann, L.** and **Herbert, T.** Active flow control witl neural networks. AIAA Shear Flow Conference, Orlando, 1993.

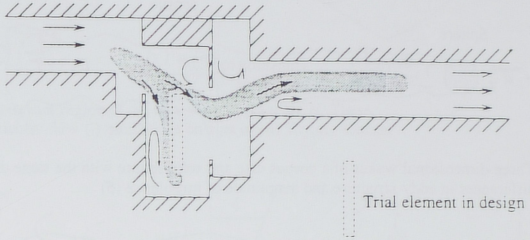
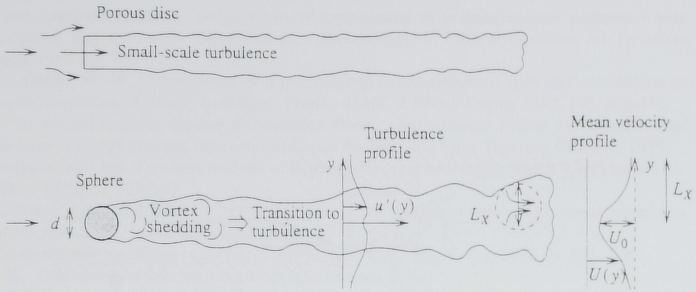
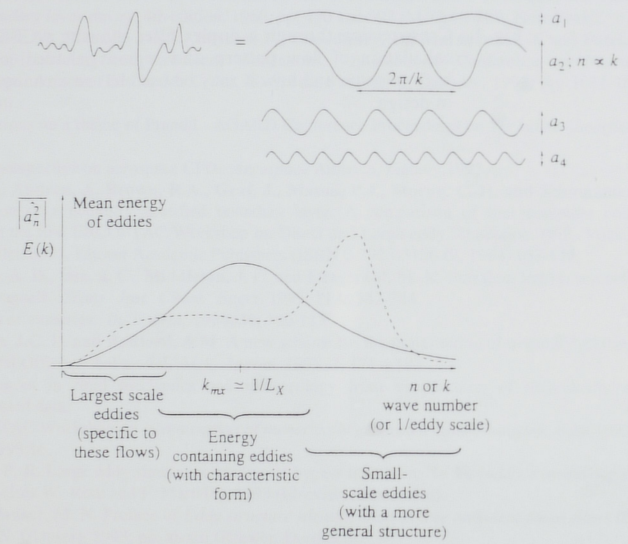


Fig. 1 Cross-section through a duplex filter showing a CFD calculation of flow pattern and regions (shaded) of high velocity, and how it may be used to test a change in design

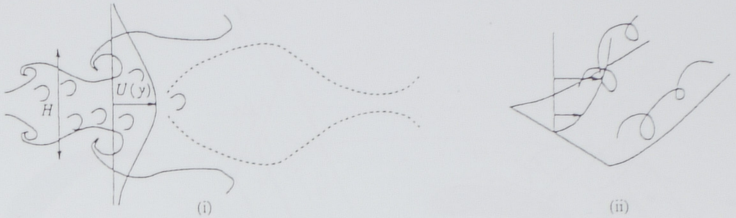


(a) Three-dimensional wakes of a porous disc and solid sphere with the same drag. Note the differences in eddy structure and magnitude of turbulence (5)

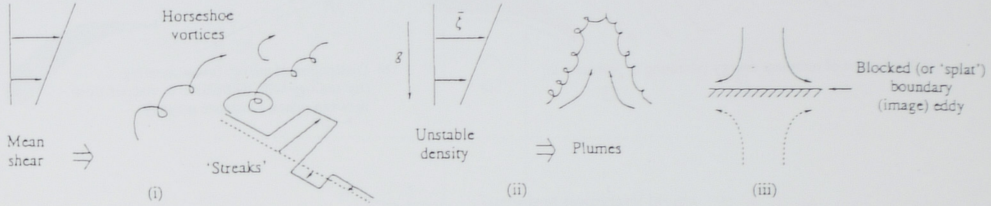


(b) Random turbulent signal represented by Fourier wave modes with random amplitudes a_1, a_2, ... and the connection with the energy spectra $E(k)$, for developing (dashed line) and fully developed (solid line) turbulence

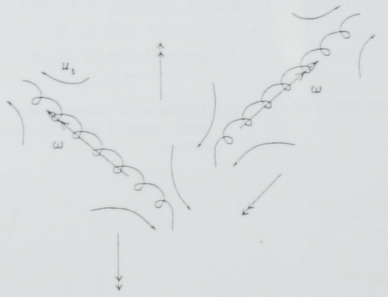
Fig. 2 Typical turbulent flow



(a) 'Large eddies' spanning the relevant region of the flow with length scale H. These are characteristic for the particular flow (i) oscillations on a jet or (ii) Taylor–Gortler vortices

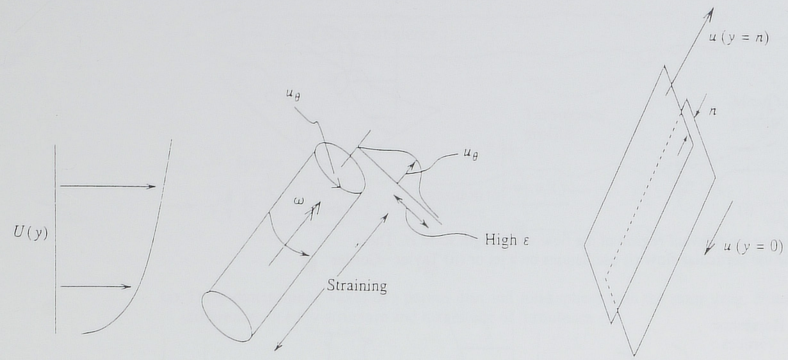


(b) 'Energetic' eddies with length scale L and having a general form characterized by the local straining motion, body force or boundary conditions. Examples are (i) shear ($L_x^{(v)-1} \sim (\partial U/\partial y)/v'$), (ii) buoyancy ($L_x^{(v)-1} \sim$ buoyancy flux$/(v')^3$, which is the Monin–Obukhov length), (iii) rigid boundary $L_x^{(v)} \propto y$



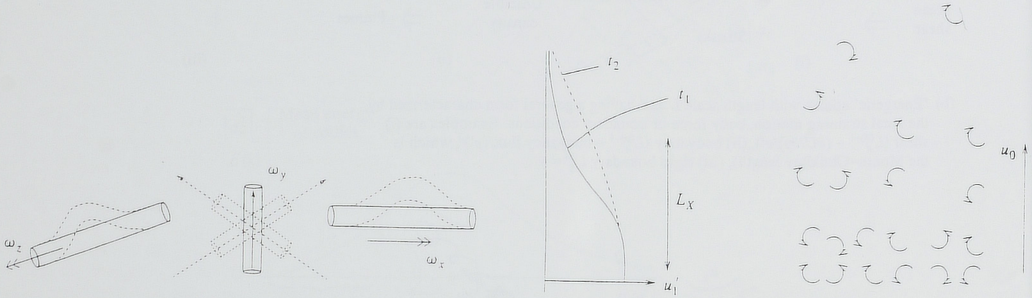
(c) 'Small'-scale eddies with their typical form of elongated vortices (ω) with spiral streamlines stretched by local straining motions u_s, which occur in flows at high Reynolds number. (Despite their non-isotropic form the eddies have approximately isotropic orientation, leading to isotropic second-order statistics)

Fig. 3 Eddy structure of turbulence at different scales and in different flows (12)



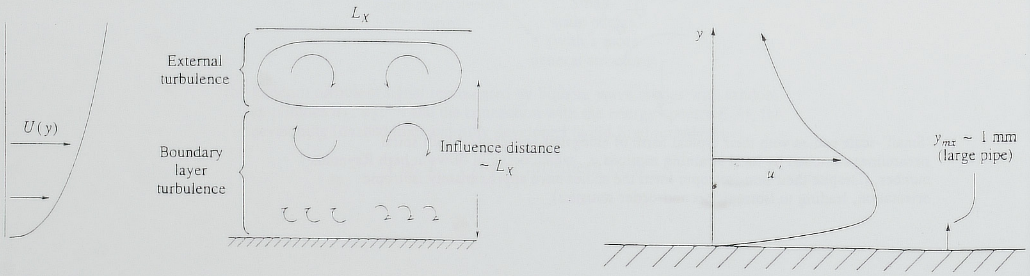
(a) Production of turbulent energy (straining of vorticity ω)

(b) Dissipation of energy (ε) by shearing (by analogy with dissipation in viscous flow between plates a distance n apart)



(c) Energy exchange between vortices ω_x, ω_y, ω_z in different directions, leading to adjustment of anisotropy

(d) 'Diffusion' of inhomogeneous turbulence (showing eddies spreading into a region of weak turbulence, in the x direction, with a speed of about u_0)



(e) Action at a distance

(f) Blocking at surfaces

Fig. 4 Schematic diagram of the main mechanisms that determine the eddy structure and statistics of turbulence

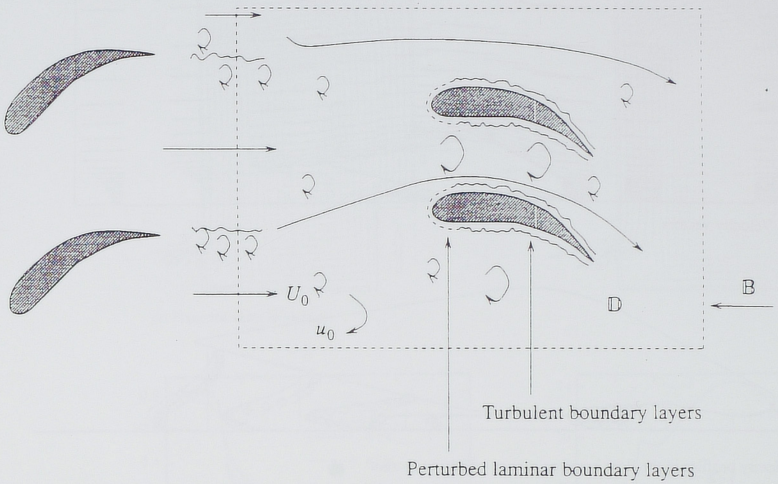
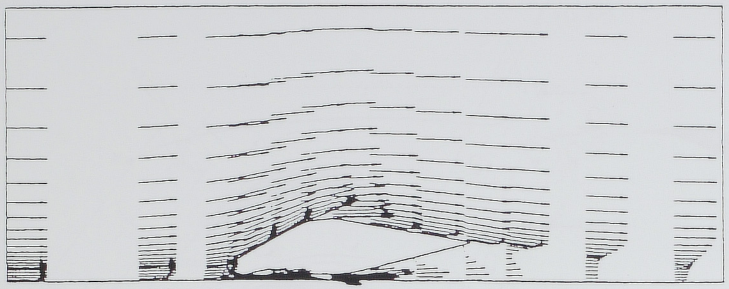
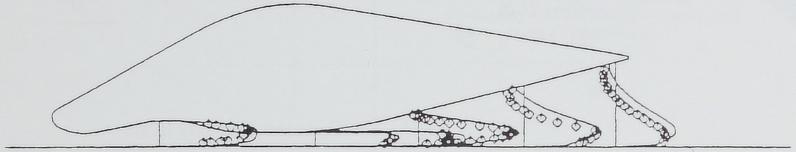


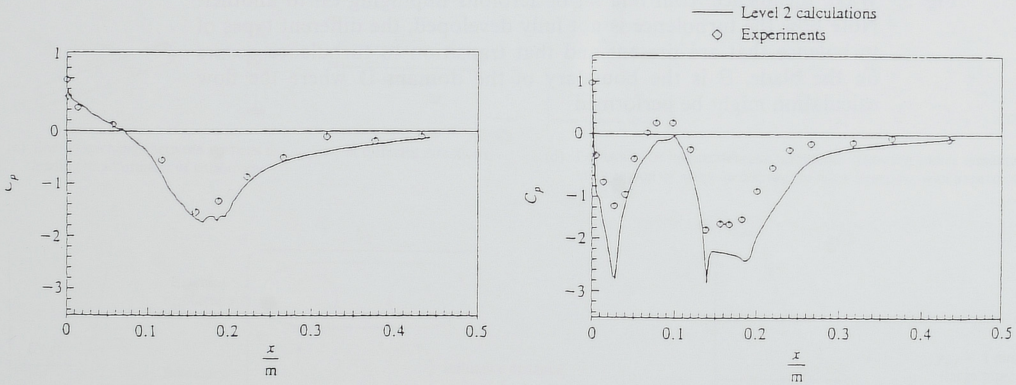
Fig. 5 Wake turbulence from one set of aerofoils impinging on to another. Note how the turbulence is not fully developed, the different types of turbulence that are present and that transition to turbulence occurs on the blade. 𝔹 is the boundary of the domain 𝔻 where the flow calculation might be performed



(a) Velocity vectors

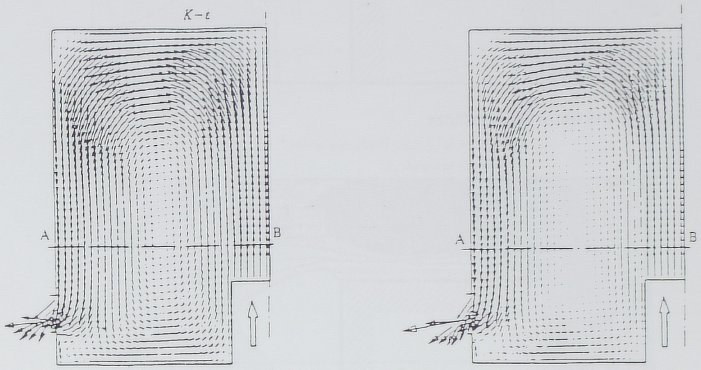


(b) Velocity profiles in the flow between car underbody and street

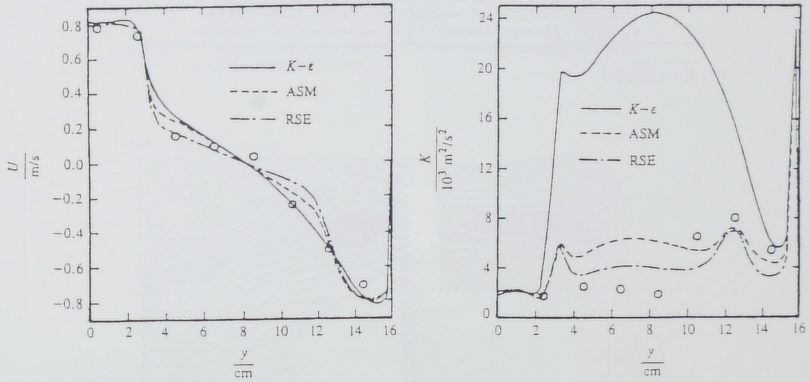


(c) Pressure distributions

Fig. 6 Flow around a two-dimensional car model K-ε [taken from reference (29)]

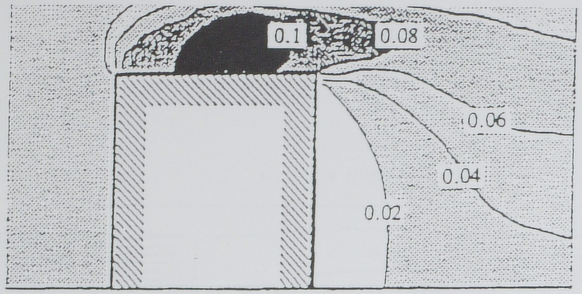


(a) Predicted velocity vectors

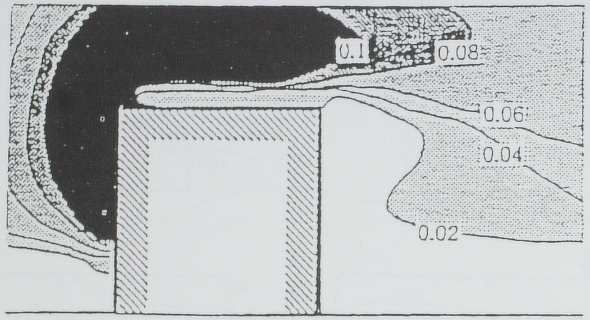


(b) U and K profiles at cross-section AB

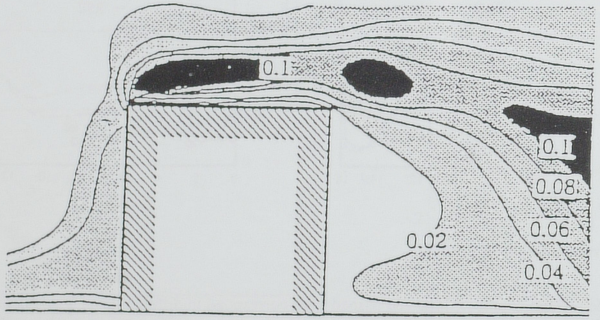
Fig. 7 Flow in plane plenum chamber; solid and dashed lines correspond to different level 2 calculations (32), circles show experiments (25) [taken from reference (34)]. ASM = algebraic stress model, RSE = Reynolds stress equation.



(a) Wind tunnel experiment



(b) $K-\varepsilon$ model



(c) LES

Fig. 8 Comparisons between (a) wind tunnel measurements of kinetic energy around a cube in a turbulent flow ($L/H \sim 1$) and two kinds of computation using (b) a level 2 statistical model (K-ε) and (c) a level 3 simulation model (large eddy simulation) (39)

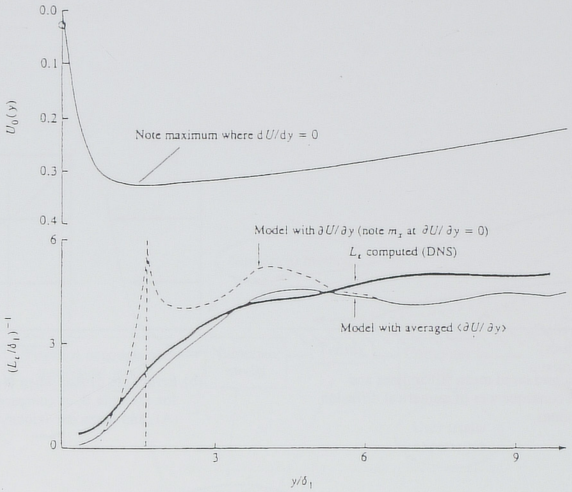


Fig. 9 Demonstration of limitation of turbulence models involving local gradients. Graph of dissipation length scale L_ε in an oscillating shear flow $U(y)$ near a wall at $y = 0$ computed by P. Spalart using direct numerical simulation compared with the model equation $L_\varepsilon^{-1} = A_B/y + A_s(1/v')\,[(dU/dy)$ or $\langle dU/dy \rangle]$. Note the improvement using the spatial averaging of $\partial U/\partial y$

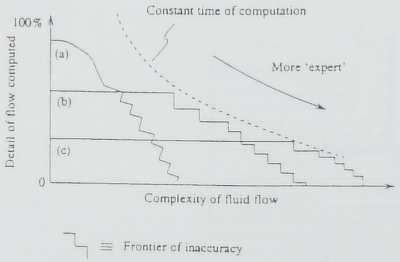


Fig. 10 Schematic graph showing the degree of detail of the computation as a function of the complexity of the turbulent flow for different levels of the computational model, for given computational capacity and run-time

(a) Indicates an advanced (level 3) simulation model (for example DNS) for an ideal flow obtained with a supercomputer (requiring many realizations and over 10–10^2 hours or computation if $Re \simeq 10^2$)

(b) Indicates an advanced (level 2) statistical model (for example second-order RST) for one typical engineering flow over 10 hours

(c) Indicates a simple statistical model (for example level 1) for many trials of complex engineering flows over 10 hours

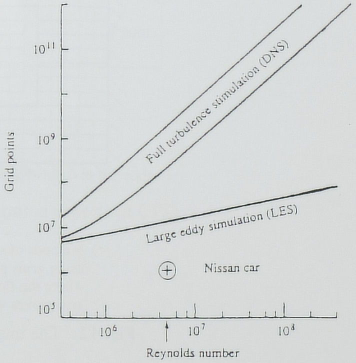
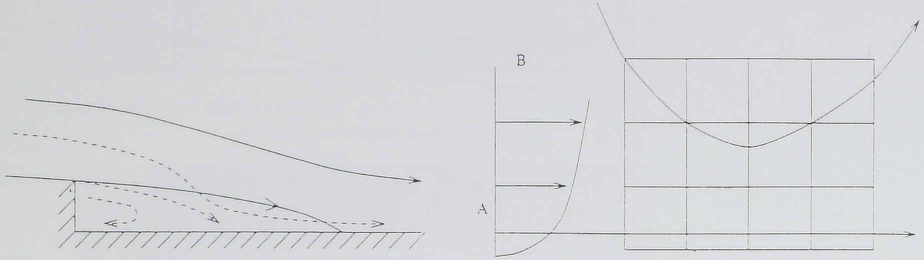
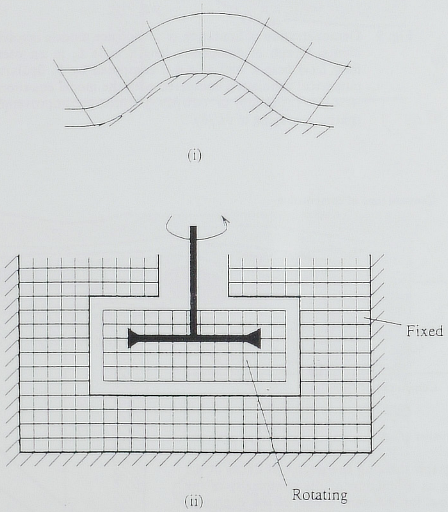


Fig. 11 Numbers of grid points required for different types of turbulence simulation (note the many floating point operations required for each grid point) [taken from reference (44)



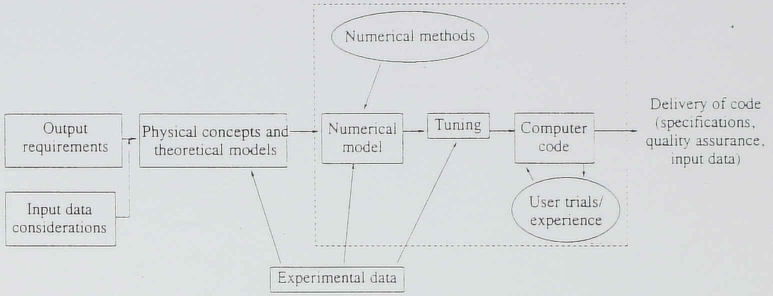
(a) Solid lines show measured mean streamlines and dashed lines the consequences of numerical diffusion on mean streamlines

(b) Errors are caused when the flow streamlines cross the grid, for example B, as compared to when they are nearly parallel (A), and when the velocity varies greatly across the grid

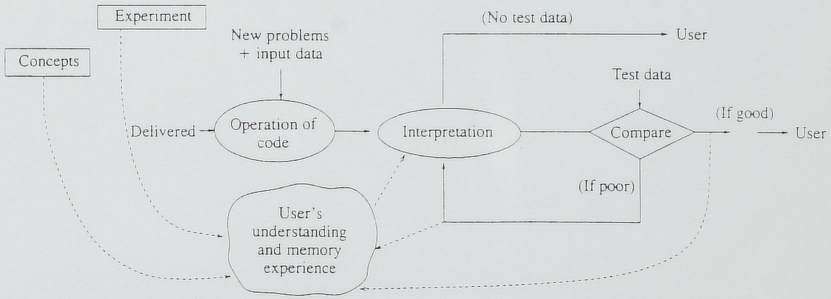


(c) Methods of overcoming problems (a) and (b) using grids parallel to the flow, with finer scales where the flow changes rapidly. In (ii) the inner grid moves with the rotor

Fig. 12 The importance of numerical methods in CFD



(a) Development (primarily within dashed line)



(b) The stages of operation, interpretation, testing and how the user 'adds value' to the code

Fig. 13 Schematic diagram of the organizational aspects of CFD codes