INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS, VOL. 7, 277–289 (1987)

FLOW SIMULATION IN CHANNELS WITH DISTORTED GEOMETRY USING A SPECTRAL CODE WITH CO-ORDINATE TRANSFORMATION

T. R. FODEMSKI

Thermo-Fluids Engineering Research Centre. The City University, Northampton Square, London EC1V 0HB, U.K.

P. R. VOKE

Turbulence Unit, Queen Mary College, University of London, London, U.K.

AND

M. W. COLLINS

Thermo-Fluids Engineering Research Centre, The City University, Northampton Square, London, EC1V 0HB, U.K.

SUMMARY

Turbulence is essentially four-dimensional in character, and requires the corresponding treatment of the well-known Navier–Stokes equations. However, this has only been possible over the past twenty years and then by using the largest computers available. Interest is now turning from the initial, mainly smooth channel, simulations to geometries of eventual engineering significance. This paper reports a new code using the spectral methods of Orszag, but also incorporating a novel generalized co-ordinate transformation approach.

Initial predictions for smooth channels agree well with published data. For distorted geometries, the initial velocity field has considerable influence on the success of the simulations. This is accommodated by gradual (step) changes towards the required distortion, so that the initial velocity field for the 'new' geometry is the final field from the previous step. Examples are given of different two-dimensional channel geometries achieved, and these include the successful prediction of recirculating flows.

KEY WORDS Large eddie simulation Spectral methods Computer simulation of flows Distorted geometries Co-ordinate transformation

INTRODUCTION

Turbulent fluid flow and heat transfer are of great importance in engineering, but at the same time their character is very complex. Here we discuss several features of this complexity, namely the basic equations, the multi-dimensional nature and the range of scales.

With the measurement of quantities such as velocity fluctuations it is understandable that turbulence is often viewed as having randomness in a statistical sense. In fact, however, it is being increasingly recognized (for example, by Spalding[1]) that laminar and turbulent flows are essentially identical, with the same ruling Navier–Stokes and energy equations. It is worth recalling that Osborne Reynolds[2] accepted this identity around a century ago, when trying to reconcile the different friction factor behaviours for laminar and turbulent flows. However, the basic equations are non-linear, and to treat a fluid flow require simultaneous satisfaction of momentum and

0271–2091/87/030277–13$06.50

Received 6 June 1986

continuity principles. As a consequence, even for the more tractable laminar flows (often genuinely two-dimensional in nature) treatments of any generality (for example, that of Collins[3]) needed good digital computers.

In addition, turbulent flows are essentially always four-dimensional in character. Physically, they may be viewed as the motion of time-dependent three-dimensional eddies of various sizes. These range from that of the confining duct geometry to that where dissipation to heat by fluid friction takes place. Reynolds's original averaging procedure[2] does not in fact immediately require the loss of treatment of this eddy motion, but that is how it is usually applied. Mathematical modelling is used for the additional Reynolds stress terms in the momentum equations (for example, by Launder and Spalding[4] and geometrical symmetry and mean flow steadiness are used to reduce the number of dimensions treated. However, we point out that in a two-dimensional turbulent flow defined in such a way, the fluid motion is still four-dimensional.*

The final factor is that of the range of turbulence scales. For transition and low turbulent flows, the dissipation or Kolmogoroff length scale is not too far separated from the large scales, and all motions are resolvable (full or direct simulation—DS). For higher turbulence, only the higher scales can be resolved (large eddy simulation—LES) and the subgrid scale (SGS) must be modelled. Such modelling is of a fairly universal nature. Since the initial work of Smagorinsky,[6] a number of simulations have been performed in confined ducts, usually for plane geometries.

Two main methods are apparent, spectral and finite-difference, exemplified by Moin and Kim[7] and Grötzbach and Schumann,[8] respectively. A finite difference approach has the advantage of potentially accommodating abrupt geometry changes. However, for a given number of points treated, spectral methods are more accurate, and the method itself mirrors the turbulence spectrum.

The motive for this particular work was to write a simulation code applicable to distorted geometries.† Immediate applications include heat transfer enhancement using surface roughness elements, and recirculating flow downstream of a backward-facing step or a sudden circular expansion. However, a code accommodating general, and if possible non-orthogonal, boundaries and grid distortions was recognized as being very desirable. The possibility then arose of using the basic equations in generalized co-ordinate form. Although this would be computationally expensive, it was discovered that a compact form could be used,[10] which avoided the necessity of computing the affine connection coefficients (or Christoffel symbols of the second kind). This was substantially due to the use of the vorticity rather than the stress form of the equations. A careful assessment of computational requirements[11] showed that for two-dimensional surface distortions the method would not be too computationally demanding. Also the approach itself would be ultimately applicable to three-dimensional distortions.

DESCRIPTION OF CODE AND METHOD OF GRID GENERATION

The spectral approach to fluid dynamic simulation is well documented,[12–14] and our system of codes (FDS) is described elsewhere.[15] Here we concentrate on those aspects that are important for presentation of our results.

The compact form of the covariant Navier–Stokes equations[10] is used, so that we work in terms of the covariant (or the contravariant) components of the velocity vector, rather than the Cartesian components. In the compact form the equations are formulated without the use of the affine

* If such were needed, recent experimental confirmation is provided by holographic interferometry results for a two-dimensional heated plane channel. The unsteadiness and spanwise effects are clearly evident.[5]

† To which interest is now passing.[9]

connection coefficients. The computation method involves only the use of metric coefficients for the geometry being treated; though the transformation matrix elements are retained to allow graphic output of the flow field in Cartesian co-ordinates, they are not used in the simulation itself.

The geometries we treat are two-dimensional, the channel having a distorted lower wall and a plane upper wall. The Cartesian co-ordinates of the lower wall are input to the code, together with a set of required gradients for the co-ordinate lines leaving the wall. These are used to generate the co-ordinate lines from the lower wall to the upper wall, using very fine (2000-point) finite differences. The gradient is decreased in a controlled manner to zero, so that the co-ordinate lines hit the upper wall at right angles. It is also possible to specify that the co-ordinate lines should be orthogonal to the lower surface, and we frequently do this. The walls are always surfaces for which z is constant in the transformed space.

This method allows interactive control of the co-ordinate spacing, and, to a lesser extent, control of the orthogonality of the co-ordinates. After the Cartesian co-ordinates of all the mesh points have been fixed, spectral methods are used to compute the transformation matrix elements and the six independent coefficients of the metric tensor at every point. These are computed once in each simulation and stored for subsequent calculations within the simulation code.

At present the transformation quantities (matrices and metric coefficients) are computed using the same spectral resolution as is used in the subsequent simulation, though it would not be difficult (or too computationally expensive) to use a higher resolution; it has been suggested that this may well be desirable.

The method as described is similar in principle to the multi-surface method of Eiseman.[16] It has been used to generate three-dimensional transformations, but these are not used in our simulations at present since they require much greater storage for the metric coefficients. The method may also be extended to cope with distorted upper surfaces. Checks for co-ordinate crossing are incorporated.

The subsequent simulations follow our methods for undistorted geometries[15] as closely as possible. The terms in the underlying equations are split into parts similar to those in the undistorted case, together with additional terms arising from the distorted geometry. The latter involve the metric coefficients, and are treated explicitly by Adams–Bashforth time stepping, like the non-linear term in the equations. The undistorted pressure and viscous terms are advanced by Crank–Nicolson time stepping, so that a value of the pressure at the advanced time may be obtained. Continuity is enforced for the velocity fields at the advanced time.

We find that the method is geometry-dependent unconditionally unstable for highly distorted geometries. This is almost certainly because in such cases the geometric part of the pressure gradient term is large and our treatment of the pressure term is therefore no longer a good approximation. Since an error at one time step in the computation of the pressure will affect the right hand side at the next step, some form of instability seems likely. An instability may also arise from the viscous term in highly distorted geometries. The unconditional form of the instability means that it is almost certainly not arising from the advection term in the equations. Time steps are chosen within the appropriate Courant limit.

The four fields are advanced in time in spectral space, that is, using Fourier expansion coefficients in the x (streamwise) and y (spanwise) directions. Chebyshev series are used in the z direction (between the walls). The wall boundary conditions are imposed using the spectral-tau method. The problem reduces to the solution of a sparse, broadly banded set of linear equations for every pair of Fourier coefficients. The techniques are well known and have been described by Gottlieb and Orszag.[12]

The non-linear and geometric terms are computed in configuration space, the fields being calculated at the collocation points by Fourier transformation (pseudo-spectral method). Aliasing

errors arising from this procedure are not eliminated because of the computational expense of full de-aliasing.

The CPU time for a simulation in a distorted geometry is less than twice that in an undistorted geometry, because of the economy of our solution method for the transformed equation. However, the advantage is not as great as this figure suggests, since the sparse matrix solutions arising from the coupled-equation approach do not vectorize well on the CRAY-1 vector computer.

EXPERIENCE WITH SIMULATION IN DISTORTED CHANNELS

The main interest in this part of the study using the code described above was the simulation of flow in plane channels, eventually with geometries of engineering significance. Up to now published papers have discussed simulation mainly in smooth channels, although work for different geometries (using the finite difference method[17–19] and different applications (air flow over mountains[20]) have already been reported. However, no report using a similar method and approach to our analysis and code has been found in the open literature. Therefore it is reasonable (and possibly helpful for others) to write more information and details about our experience and knowledge gained during this work.

Using the code described, flow simulations were successful in plane channels with small distortions, but not in those with large distortions—as shown in Figures 1 and 2. The velocity field in the channel shown in Figure 1, after unsuccessful simulation (about 30 steps long), is shown in Figure 3.

The sharpness of the distortion is also important, i.e. even for small but sharp distortions, simulations were not successful. Both the size and sharpness of the distortion are probably connected with the occurrence of an unconditional geometry-dependent instability in the code.

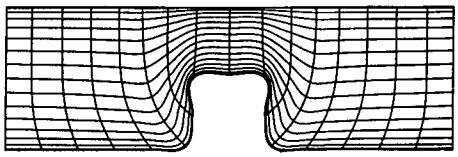

Figure 1. Plane channel with square rib, $A/H = 0{\cdot}5$; grid $16 \times 16 \times 17$

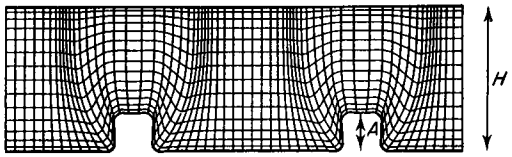

Figure 2. Plane channel with two square ribs, $A/H = 0{\cdot}25$; grid $64 \times 16 \times 17$

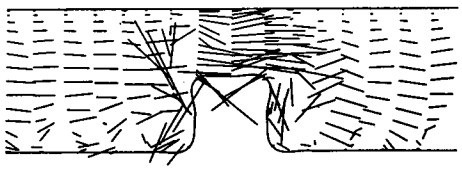

Figure 3. Velocity field of unsuccessful simulation

It is worth noting that we regard a successful simulation one which can be performed for at least $5\,H/u_\tau$ (this non-dimensional unit of time is known as a 'letot' since it is a rough measure of the large eddy turn-over time).

For a $16 \times 16 \times 17$ grid and a time step $t = 0{\cdot}0025$ this required about 1 h computing time on the CRAY-1S computer. Flow was simulated for 1800 steps or 4·5 letots—an increase of about 40 per cent compared with an undistorted channel (i.e. one not needing transformation of co-ordinates). It should be noted that once simulation was successful for that period it could probably be performed successfully for greater times: we never experienced simulation failure for times greater than 5 letots. However, it should be pointed out that only for one particular geometry was a simulation carried out for longer than 10 letots: this was because of limitations on computer time.

To find a practical limit for geometries for which our code could give successful simulations a series of channels was investigated with cosine-waves of different amplitude, or ratio A/H (for example Figure 4). This type of distortion can easily take full advantage of the grid generation method described above. The general conclusion from this exercise is that the initial velocity field has considerable influence on the success of the simulation. This initial velocity field is a direct consequence of the generated distribution of co-ordinates. In the case of the initial velocity field being too far from the probable velocity field, calculations do not converge to the latter. Using co-ordinates orthogonal to the walls throughout the whole channel, simulation can be successful for greater cosine-wave amplitude (or A/H). Undoubtedly, co-ordinates orthogonal to the walls result in an initial velocity field closer to reality, specifically for the direction of the fluid velocity close to the walls. Therefore when dealing with substantial distortion of the channel orthogonal co-ordinates had to be used.

Simulations successful for times around 4 to 5 letots result in velocity fields which we conclude are close to those in real flows. This closeness is a question which for plane channels can be verified relatively easily. However, for distorted channels it can be only positively judged when computer simulations are compared with experimental results—which unfortunately are sparse and available only for certain geometries. Therefore the problem remains (when using our code) of how close our results for resolved geometries are to those which are useful and/or have experimental data. To bring the former as close as possible to the latter, a special method was developed of step change in the channel geometry.

It takes full advantage of the fact that the velocity field after a simulation time of a few letots is converging to the required field. The opportunity therefore arises of using the velocity field produced at this stage as an initial velocity field for a flow simulation in a channel of somewhat greater distortion than the preceding one. Hence, a series of steps of geometry changes will result in the final, required geometry being increasingly approached. Our criterion for a successful simulation for any step remains the same: it should last about 5 letots. The extent of development of the geometry from a given initial geometry is chosen by trial to secure success on the next step. It is worth noting that in a few cases success of the simulation on the more complex geometry was achieved by returning to the simulation on the previous geometry and running for greater time, before submitting the velocity field so produced once again to the new geometry. Similarly, the guarantee of a successful flow simulation on the next step increases when the geometry step change decreases. These two practical ways of increasing simulation success, however, also increase the computing time required. Figures 4–6 show initial, typical interim and final channel geometries, during a successful evolution from a cosine-wave towards a perfect-rib. Although simulations for a sharp geometry of the type shown in Figure 1 were never achieved, it is important to note that successful simulation at each interim stage cannot be achieved otherwise.

The whole process of step changes of channel geometry introduction is explained in detail in Figures 7–9. They show the change of turbulent energy E_T with time (letots) at two Reynolds

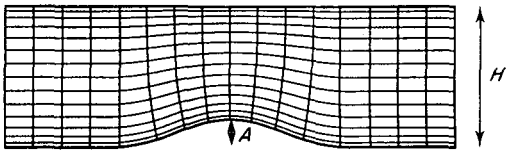

Figure 4. Plane channel with cosine-wave, $A/H = 0{\cdot}2$; grid $16 \times 16 \times 17$. Initial stage of geometry

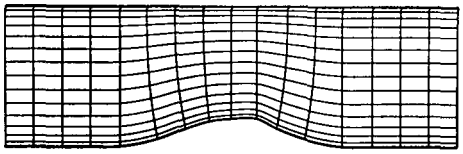

Figure 5. Interim stage of channel geometry

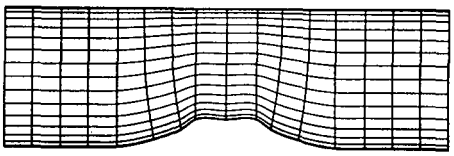

Figure 6. Final channel geometry in which simulation was successful

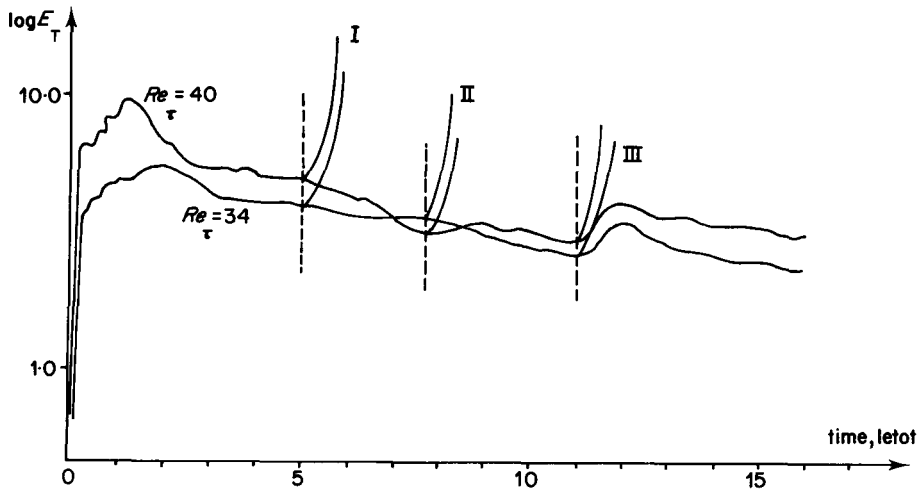

Figure 7. Unsuccessful introductions of step geometry changes at different points of simulation

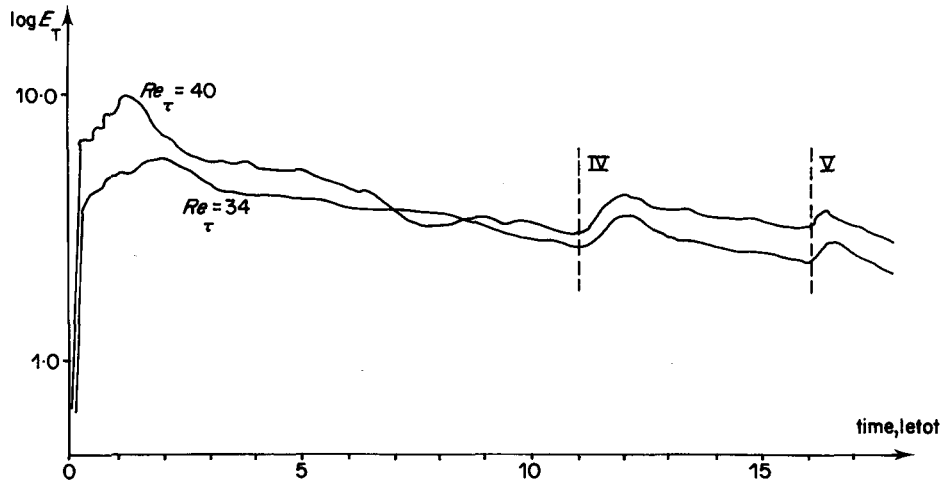

Figure 8. Successful introductions of step geometry changes at different points of simulation

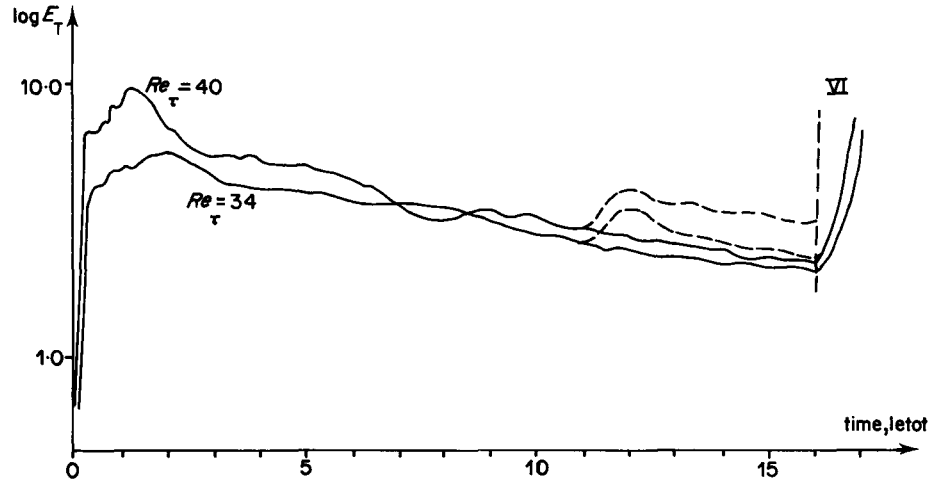

Figure 9. Unsuccessful introduction of too big a step geometry change after about 16 letots of simulation

numbers (34 and 40). The chosen range of Reynolds number is based on results obtained for a plane channel (see Figure 10, taken from Reference 15). That is, for $Re_\tau > 33$ there is clear evidence of transition to self-sustaining turbulence and for $Re_\tau > 40$ the result of a direct simulation begins to be unphysical and an SGS (subgrid scale) model must be used. It should be noted that calculation of E_T for a distorted channel was carried out in the same manner as for the plane channel. In the former case the distorted geometry acts as a turbulent promoter and gives an additional contribution to E_T when compared with the plane channel. Therefore, in this case, E_T always sharply increases when a simulation begins (see Figures 7–9). However, its qualitative value as a function of time gives the information required—specifically whether or not a statistically stable state of simulation was achieved. This may be defined as only a small fluctuation of E_T being present, with its mean value being constant.

Figure 7 shows the change of turbulent energy E_T during three unsuccessful attempts (I, II, III) of

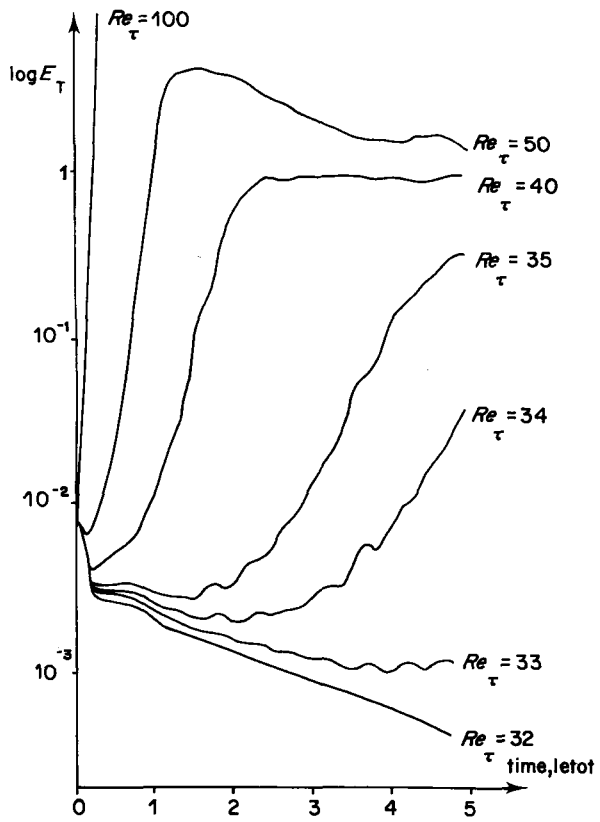

Figure 10. Turbulent kinetic energy E_T during direct simulation of flow in a plane channel near transition

step changes in channel geometry. The first attempt (I), i.e. introduction of the geometry shown in Figure 5 after about 5 letots of simulation in the geometry shown in Figure 4, results in a velocity field similar to that shown in Figure 3. The reason for this failure, in the light of further results, was that the step change of channel geometry was introduced too early, when the simulated velocity field had not reached a state close enough to the real velocity field. Note that the turbulent energy E_T during further simulation in the geometry shown in Figure 4 still decreases. Similar conclusions should be drawn from the second unsuccessful attempt (II), i.e. introduction of geometry shown in Figure 5, despite the fact that the new geometry was introduced at the moment when turbulent energy E_T, for $Re_\tau = 40$ shows a local minimum. Finally, the third unsuccessful attempt (III) in Figure 7, refers to the introduction of the geometry shown in Figure 6, after about 11 letots of simulation in the geometry of Figure 4. In this case, the reason for the failure in the light of further results, was that the step change of the channel geometry was too great.

Figure 8 shows successful introduction of two steps of channel geometry changes (IV and V). The introduction marked as IV, i.e. introduction of the geometry shown in Figure 5, was made after about 11 letots of simulation for the geometry in Figure 4. The next, marked as V, i.e. introduction of the geometry of Figure 6, was made after about 5 letots of simulation for the geometry of Figure 5, preceded, as mentioned, by 11 letots of simulation in a channel of the geometry shown in Figure 4. It is worth noting that the introduction of the channel geometry shown in Figure 6 was unsuccessful not only in the case marked as III in Figure 7. It was also not successful even after prolonged simulation to 16 letots in a channel of the geometry shown in Figure 4. This attempt, marked as VI, is shown in Figure 9.

The final conclusion from the whole exercise is in fact twofold. The step change of channel geometry method increases the limit of the size and sharpness of distortion in which simulation using our code can be performed successfully. It should be, however, applied gradually with sufficient length of simulation in the interim stages.

The method described might be useful in other codes if a similar problem were encountered. It allows simulations to be performed also in channels with ribs, or other distortions, which are not sharp (as that from Figures 1 or 2), but smoothed by the effect of deposition.

The presence of the channel distortion in real flow causes a recirculation region; therefore simulation should also produce such a region. Its size and positioning are influenced by three factors: the shape, the value of the ratio A/H and the Reynolds number.

With our code, at this stage of its development the use of these three factors is restricted for the reasons explained earlier. Additionally, the final shape of the channel distortion in which our code can be used successfully is such that practically there are no experimental data with which our simulation results can be compared. It is also thought that a $16 \times 16 \times 17$ grid is too coarse to achieve reasonable resolution and allow detailed comparison with other simulation or experimental data (if available). For these reasons we present here only qualitative information about recirculation regions in our simulations.

1. For the three geometries shown in Figures 4–6, recirculation regions are not steady, i.e. during simulation their lengths (the distances between the points A and B in Figures 11–13) and positioning (points A and B) change. The length of this region slightly but noticeably increases with increase of the distortion sharpness. It covers 0·2 to 0·4 of the length between distortions in channels with the geometries shown in Figures 4 and 5, and 0·25 to 0·45 of this length for the geometry shown in Figure 6.
2. The movement of point A has an oscillatory character. Its position repeats about every 300–500 steps of calculation, i.e. 0·75–1·25 letots. There is a tendency towards more frequent oscillations when the sharpness of the distortion increases.

It should be stressed that although oscillations of point A are present, the flow pattern and velocity field do not repeat in all details. This is undoubtedly the result of two facts: the nature of turbulent flows and also because of the continuous convergence of the simulating flow to the real one (note that turbulent energy E_T in Figures 7–9 still decreases). The removal of the influence of the latter could only be done by prolonged simulation, probably for at least 10 letots. However, since it would involve more expensive computing time, it was decided at this stage of the work to stop the simulation at that point and concentrate upon implementation of conclusions reached from work already performed in the future flow simulation (using the code discussed) in the channel geometry of practical interest.

CONCLUSIONS AND FUTURE WORK

The analysis and code reported in this paper have been successfully applied to give large eddy and direct simulations of turbulent flow in smooth plane channels, either smooth or with two-dimensional surface distortions. Grid and boundary generation are achieved by using a generalized form of the basic Navier–Stokes equations.

Results for smooth channels compare well with published data, and those for distorted geometries include satisfactory predictions of recirculating flows. The unconditional instability encountered, however, limits the sharpness of distortion treatable, and the code will, therefore, be used in parallel with a new three-dimensional time-marching finite difference code which satisfactorily accommodates abrupt boundary changes.

A comparison is planned with experimental data obtained using holographic interferometry.

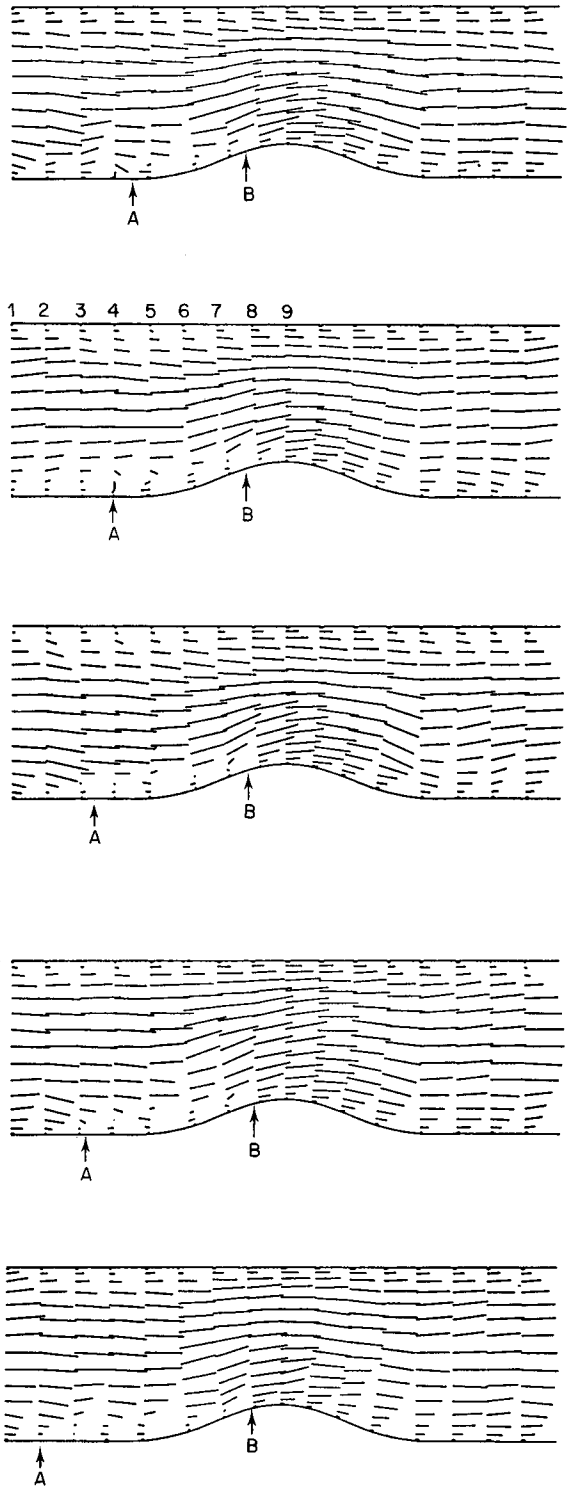

Figure 11. Velocity field in channel with geometry from Figure 4, with recirculation region (between points A and B)

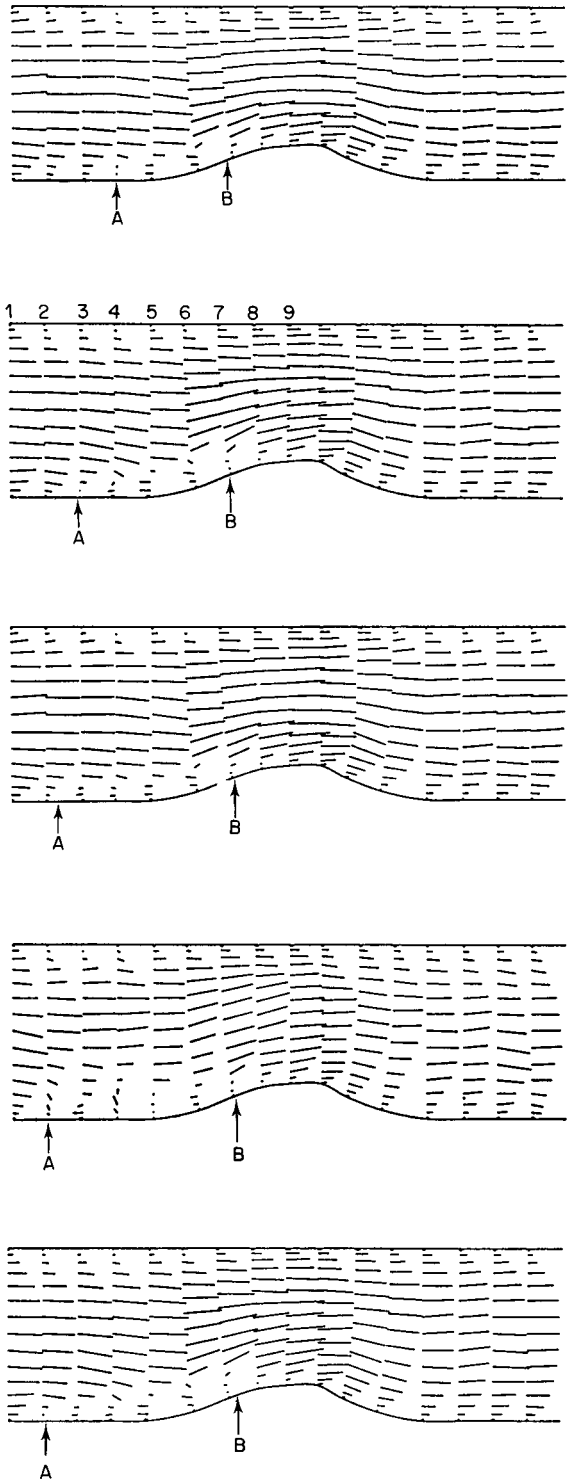

Figure 12. Velocity field in channel with geometry from Figure 5 with recirculation region (between points A and B)

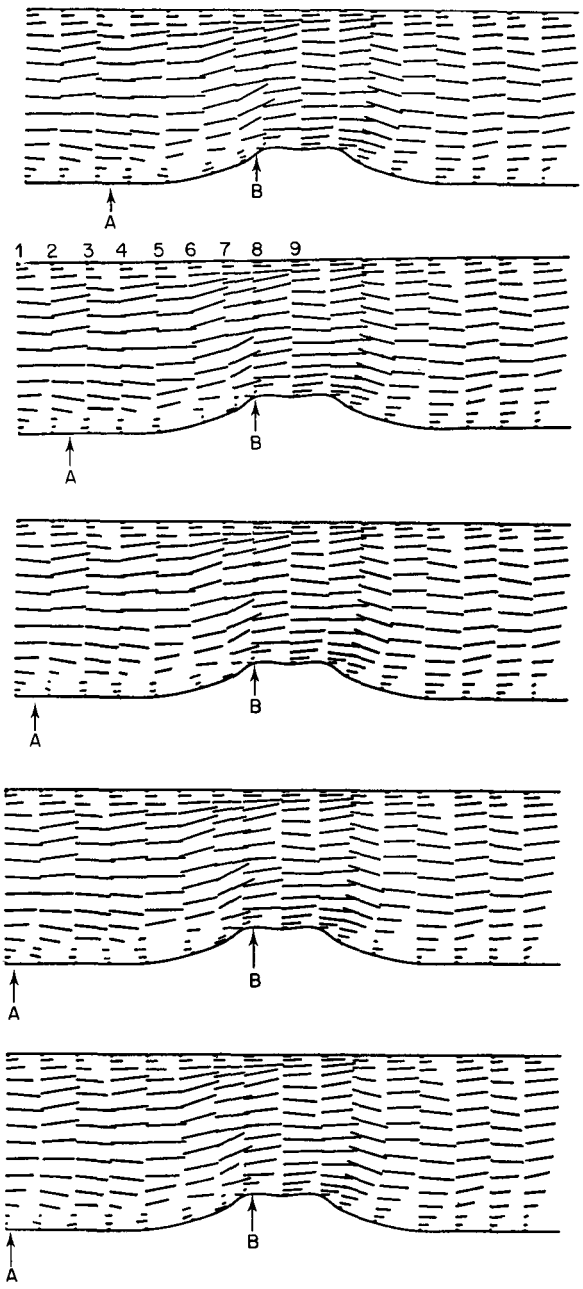

Figure 13. Velocity field in channel with geometry from Figure 6 with recirculation region (between points A and B)

The geometry is that of originally square individual roughness elements, but smoothed by the effect of deposition.

ACKNOWLEDGEMENTS

This work has been supported by two Cooperative Research Grants from the U.K. Science and Engineering Research Council (Grants GR/B41452 and CR/C90577), with the Atomic Energy Research Establishment, Harwell as the industrial sponsor.

Computational facilities were provided by the CRAY-1S computers at AERE Harwell and the University of London Computer Centre.

REFERENCES

1. D. B. Spalding, 'Turbulence models for heat transfer', *Proc. 6th Int. Heat Transfer Conf.*, Thorn Press, Toronto, 1978, Vol. 6, Keynote Papers, p. 33.
2. O. Reynolds, 'On the dynamical theory of incompressible viscous fluids and the determination of the criterion', *Phil. Trans. A.*, **186,** 123 (1895).
3. M. W. Collins, 'A comprehensive finite-difference treatment of various laminar flow convection problems in circular tubes' in C. Taylor, E. Hinton and D. R. J. Owen (eds), *Numerical Methods for Non-Linear Problems*, Pineridge Press, Swansea, 1980, p. 735.
4. B. E. Launder and D. B. Spalding, *Mathematical Models of Turbulence*, Academic Press, London and New York, 1972.
5. J. F. Lockett, M. W. Collins and J. R. Croft, 'Real-time holographic technique for the study of two-dimensional thermal boundary layers', (to be submitted to *Int. J. Heat Mass Transfer*), 1986.
6. J. Smagorinsky, 'General circulation experiments with the primitive equations', *Mon. Wea. Rev.*, **91** (3), 99 (1963).
7. P. Moin and J. Kim, 'Numerical investigation of turbulent channel flow', *J. Fluid Mech.*, **118,** 341 (1982).
8. G. Grötzbach and U. Schumann, 'Direct numerical simulation of turbulent velocity-pressure- and temperature-fields in channel flows', in F. Durst *et al.* (eds), *Proc. Symp. on Turbulent Shear Flows*, Penn. State University, 1977, p. 1411.
9. J. Kim, 'Final discussion' at the *Conference on Direct and Large Eddy Simulation of Turbulent Flows, Euromech 199*, Munich, 1985.
10. P. R. Voke and M. W. Collins, 'Forms of the generalised Navier–Stokes equations', *J. Eng. Maths*, **18,** 219 (1984).
11. P. R. Voke and M. W. Collins, 'Fluid dynamic simulations in general coordinates', in C. Taylor, J. A. Johnson and W. R. Smith (eds), *Numerical Methods in Laminar and Turbulent Flow*, Pineridge Press, Swansea, 1983, p. 1204.
12. D. Gottlieb and S. A. Orszag, *Numerical Analysis of Spectral Methods*, Society for Industrial and Applied Mathematics, Philadelphia, 1977.
13. S. A. Orszag, 'Spectral methods for problems in complex geometries', *J. Comput. Phys.*, **37,** 1980, p. 70.
14. P. Moin and J. Kim, 'On the numerical solution of time-dependent viscous incompressible fluid flows involving solid boundaries', *J. Comput. Phys.*, **35,** 381 (1980).
15. P. R. Voke and M. W. Collins, 'Large-eddy simulation of turbulent flow in plain channels', *Report AERE HTFS RS551*, 1984.
16. P. R. Eiseman, 'A multisurface method of coordinate generation', *J. Comp. Phys.*, **33,** 118 (1979).
17. T. Kobayashi, M. Kano and T. Ishihara, 'Prediction of turbulent flow in two-dimensional channel with turbulent promoters', *Bulletin of JSME*, **27,** 1893 (1984) and **28** 2940 and 2948 (1985).
18. N. S. Wilkes, 'Prediction of the turbulent fluid flow through a channel with rib-roughened surfaces', *AERE-R 9874*, 1980.
19. N. S. Wilkes, 'The prediction of heat transfer from surfaces roughened by transverse ribs', *AERE-R 10293*, 1981.
20. H. Volkert and U. Schumann, 'Development of an atmospheric mesoscale model—setting up the version in terrain following coordinates', *Proc. 6th GAMM Conf. on Num. Meth. in Fluid Mech.*, Göttingen, 25–27 September 1985.