Int. J. Heat Mass Transfer. Vol. 27, No. 9, pp. 1485–1491, 1984
Printed in Great Britain

0017-9310/84 $3.00 + 0.00

Numerical computation of convective heat transfer in complex turbulent flows: time to abandon wall functions?

B. E. LAUNDER

Mechanical Engineering Department, University of Manchester Institute of Science and Technology, P.O. Box 88, Manchester M60 1QD, U.K.

(*Received* 8 *February* 1984)

Abstract—In applying finite-difference and other numerical approaches to the computation of elliptic or three-dimensional turbulent flows it is usual to adopt wall functions to bridge with a single cell the very thin region adjacent to the wall where viscosity modifies the turbulence structure. The paper suggests that such approaches are now unnecessary from the standpoint of computational economy and, in many cases, may be replaced to advantage by a fine-grid, near-wall treatment that extends the numerical computations through the buffer zone to the wall itself. Examples are given of three flows currently being explored at UMIST where the abandonment of the wall-function approach has led to a marked improvement in the prediction of convective heat transfer rates.

1. INTRODUCTION

MANY of Sir Owen's experimental studies [1, 2] still serve as a reminder that relatively few practically interesting problems in convective heat transfer can be adequately tackled with the two-dimensional (2-D), boundary-layer forms of the momentum and energy equations that are the subject of most textbook discussions. Flow recirculations or strong streamline curvature, often accompanied by important three-dimensional (3-D) effects, necessitate that the flow variables be treated as elliptic on at least one of the families of surfaces mapped by two of the coordinates adopted. Thus, in any numerical study, values of the dependent variables and the associated difference coefficients require storing at every node in such a surface. The numerical computation is thus inevitably a fairly heavy one.

When, as is usually the case, the flow is turbulent, quite apart from the problem of representing the physics of the transport processes, a further computational difficulty arises. Within a very thin zone near the bounding walls effective transport coefficients change by more than an order of magnitude as vigorous turbulent mixing gives way to purely molecular transport at the wall itself. To resolve this transitional layer properly with a numerical solution procedure requires a very fine mesh around those edges of the domain bounded by a wall. More often than not, however, the implied extra core required is not available to those wishing to make the computations—or, at least, not routinely so. Moreover, the near-wall mesh refinement will tend to slow down convergence rates quite seriously. For both reasons the vast majority of studies of convective heat transfer in elliptic flows have avoided the use of a fine near-wall grid by adopting 'wall functions' to bridge the whole of the near-wall sublayer where viscous effects on turbulence are significant [3–5]. In such approaches, the near-wall node is placed in a fully turbulent fluid. The so-called 'wall functions' are simply formulae which attempt to account, in overall fashion, for the effective conductances, sources and sinks of the region between the node and the wall, including all the viscosity affected sublayer. The 'wall functions' are largely based on experimental data for parallel flows in simple shear and their extension to forms suitable for separated flows [4, 5] while plausible, have never, to the writer's knowledge, been directly tested.

Of course, as larger and larger computer memories become available, some of the arguments for adopting wall functions are beginning to weaken. But, nevertheless, would a changeover to a fine-grid analysis produce benefits commensurate with the extra effort involved? This question is one that is currently being addressed in three separate numerical projects at UMIST on convective heat transport. The paper reports briefly on the results emerging so far from these studies. These indicate clear advantages in replacing the wall functions by a fine-grid resolution. In the course of this work a modified numerical solution of the near-wall region has been evolved [6] which accelerates convergence rates and reduces core requirements, in some cases, to a point where overall time and memory demands are comparable with wall-function approaches.

2. ALTERNATIVE NEAR-WALL TREATMENTS

Figure 1 refers to a region of flow in the immediate vicinity of a wall. The work has been based on a finite-volume discretization and the figure shows for each of the three approaches illustrated a single column of scalar nodes normal to the wall and their associated control volumes (analogous alternatives are possible within a conventional finite-difference or finite-element discretization). The wall-function approach indicated

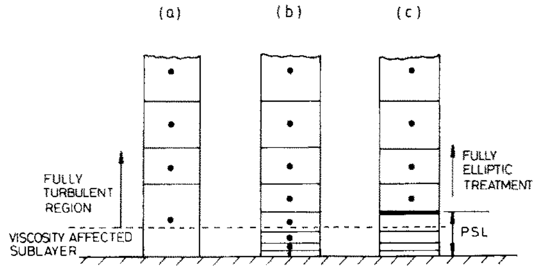

FIG. 1. Alternative near-wall treatments in turbulent elliptic flows.

in Fig. 1(a) adopts a large near-wall cell, possibly larger than those immediately adjacent to it. Although in principle wall functions could be developed to cope with the node falling inside the viscosity affected region,* the forms habitually adopted (based on generalizations of the semi-logarithmic velocity and temperature profiles and inferences therefrom) presume the near-wall node to be located in fully turbulent fluid. Extensive accounts of wall functions and their manner of application are given in refs. [3, 4]; a more recent formulation appears in ref. [7]. Here one can comment simply that they are economical to apply but that current versions seem reliable only where the existence of a semi-logarithmic near-wall region is assured. The work of Sir Owen Saunders on flow near spinning discs [1] or on the motion near a horizontal cylinder created partly by buoyancy and partly by rotation of the cylinder about its axis [2] gives striking examples of situations where these semi-logarithmic laws do not apply.

The usual alternative to wall functions is the elliptic fine-grid analysis illustrated in Fig. 1(b). Computations extend through the viscosity affected sublayer close enough to the wall to allow laminar flow boundary conditions to be applied. The number of nodes needed across the viscosity affected region depends on the turbulence model adopted. When a simple scheme like the mixing-length hypothesis is used in conjunction with, say, a cubic spline to represent the highly nonlinear variation in turbulent transport coefficients, seven or eight well placed nodes may suffice—at least for Prandtl numbers not too much greater than unity. With more elaborate approaches as many as 20–30 nodes may be required. At present, workers at NASA Ames making various computations of shock-boundary-layer interaction have been the only group to have made extensive use of this approach [8, 9]. Ironically, a very recent publication [10] reports the conversion of their effort to wall-function treatments, an indication that even at that institution computer resources are not yet limitless.

The third scheme [6] also adopts a fine-grid analysis except that over a thin layer adjacent to the wall—the parabolic sublayer (PSL)—the variation of static pressure is either neglected or, if the surface is highly curved, obtained by assuming radial equilibrium. This simplification, which removes the need for pressure nodes within the PSL, offers a very substantial core saving for 3-D semi-elliptic treatments [11] where the pressure alone is required to be stored over the whole flow domain. One also avoids the work of applying a pressure-correction to each node at every iteration. A further saving in computational effort is that within the PSL the velocity component normal to the surface is obtained by applying continuity to scalar cells rather than by solving the momentum equation normal to the wall. There is no unique connection between the thickness of the viscosity-affected layer and that of the region over which the PSL approximation is valid; it depends on the details of the flow. In about half the cases considered so far it has been possible to extend the PSL into fully turbulent fluid, while in the others a changeover to a conventional elliptic solution is made about half-way across the viscosity-affected region.

The principal computational results presented below have been obtained with the PSL approach, though where information is available from one or both of the alternative schemes this is given too.

3. SOME APPLICATIONS

In the following paragraphs, examples are reported that are drawn from current research projects on convective heat transfer in various tube geometries. The flows are very different one from another and for that reason a different numerical solving procedure has been adopted for each. All the schemes, however, are based on a finite-volume solution of the velocity components, pressure and energy; in addition, equations for the turbulent kinetic energy and its rate of dissipation are solved over most of the pipe cross-section. In all cases the conventional upwind difference approximation of convective transport has been replaced, in the first and third cases by the more accurate *quadratic* upwind interpolation [12, 13] and in the second by central differencing, the mesh being fine enough for the cell Peclet number to be less than two. In all cases the standard $k \sim \varepsilon$ Boussinesq viscosity model of turbulence [5, 14] is applied over the main flow and the turbulent Prandtl number for heat transport is taken as 0.9. Simpler viscosity models are used to span the low-Reynolds-number region, as is discussed in connection with the particular examples.

The first example relates to an abrupt expansion in a circular pipe, the downstream large tube being heated uniformly. Recent experimental data by Baughn *et al.* [15] have shown that for diametral expansions of 2:1 or greater the minimum Nusselt number occurs about one-step height downstream of the expansion due, presumably, to the presence of a small secondary recirculation near the base of the step. Computations by Mr C. Yap at UMIST using a 42 × 35 grid failed to discover any trace of this secondary circulation when using a wall-function boundary treatment. The

* Dr M. W. Collins (personal communication) has been exploring this approach.

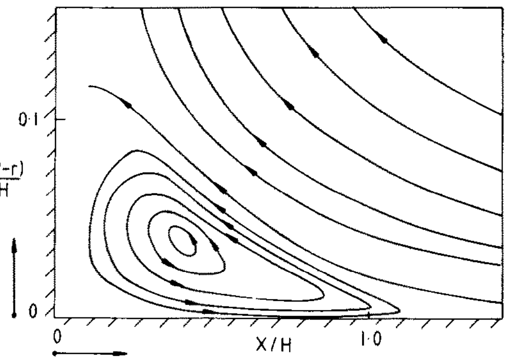

FIG. 2. Secondary recirculation in the corner region of an abrupt pipe enlargement. [Note different scales in streamwise (x) and radial (r) directions.] H, step height; R, pipe radius (Re = 33 000); radial expansion ratio 1.85 : 1.

switchover to a fine-grid, low-Reynolds-number analysis near the surface of the large pipe (with a 42 × 42 grid) has had a large effect on the predicted flow pattern in this nearly stagnant region. An appreciable secondary eddy is now formed, but while it reaches a full step height downstream it extends barely one-tenth of a step height normal to the wall, Fig. 2. It is arguably because of the thinness of this secondary eddy that the wall-function approach failed to detect its existence. Again, in contrast to the wall-function predictions, the computed Nusselt number near the step, shown in Fig. 3, rises as x approaches zero. This feature is qualitatively correct, though there are quantitative differences due at least partly to the fairly simple one-equation model [16] so far adopted for the transport processes in the low-Reynolds-number region. What is incontrovertible, however, is that despite the limitations of the turbulence model, the replacement of wall functions by a fine-grid treatment has allowed a more realistic modelling of the flow just downstream of the step.

The second example relates to flow in a spirally fluted tube: this is the name applied to tubes with numerous helical grooves or ridges around their boundary which impart a swirl to the fluid flowing through it. A family of tubes of this type evolved by Yampolsky [17] has been found to give a strong enhancement of Nusselt number without any corresponding rise in friction, a very

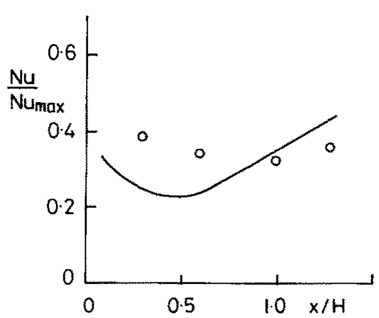

FIG. 3. Nusselt number behaviour in corner region: ——, fine grid computations; ○, experiment, Baughn *et al.* [15].

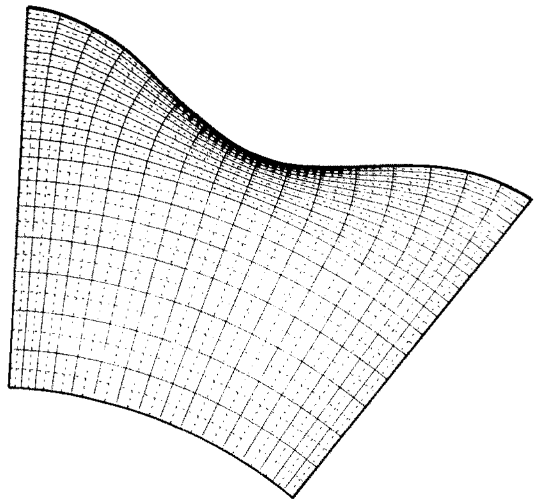

FIG. 4. Grid system adopted for analysis of flow in spirally fluted tubes.

attractive combination for heat-exchanger applications. Ms A. Barba at UMIST has been making computations of this flow with a view to optimizing the geometry. Barba *et al.* [18, 19] report laminar flow applications and show substantial heat-transfer augmentation, particularly for fluids of moderate Prandtl numbers. The computational mesh adopted for turbulent flow is shown in Fig. 4. The flow is solved over a sector of the cross-section covering one complete flute. The circumferential variations in the flow arising from the flutes actually penetrate less than half-way to the tube centre and so a 2-D mapping of the flow and thermal field is needed only for the outer region shown in Fig. 4; in the inner core region, the variables are solved just along a single radial string. Near the wall a curvilinear mesh is introduced to allow the grid to fit the undulations of the tube surface.

The initial computations in turbulent flow adopted the coarse-grid, wall-function approach but, as Fig. 5

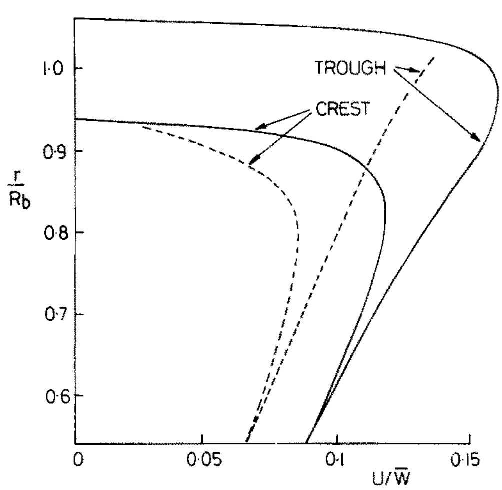

FIG. 5. Induced swirl velocities in flow through a spirally fluted tube: ——, fine grid; - - - - -, wall functions. Helix angle 15°; 10 flutes of sinusoidal profile amplitude 6% mean radius.

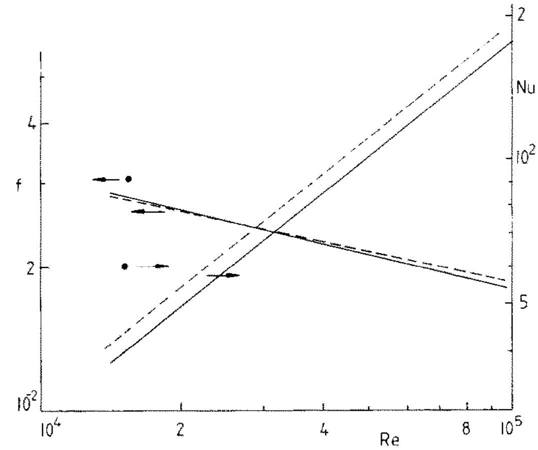

FIG. 6. Mean Nusselt number and friction factor in fully developed flow through spirally fluted tubes, $Pr = 0.7$: ——, smooth; -----, spirally fluted tube (specifications as Fig. 5); ●, spirally fluted tube, 30 flutes (other specifications as Fig. 5).

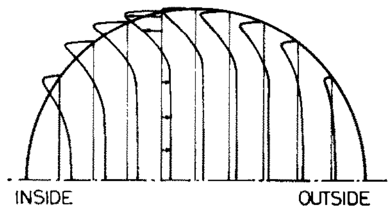

FIG. 7. Secondary velocity profiles at 25° in turbulent flow around a 90° bend: $Re = 43 \times 10^3$ (distance between adjacent data lines corresponds to 25% of bulk streamwise velocity).

shows, the maximum swirl velocity imparted to the fluid always occurred at the near-wall node—thus no reliance could be placed on its value. Accordingly, a fine-grid analysis was introduced instead in which the mixing length hypothesis (with Van Driest's [20] damping function applied to the mixing length) was used to cover the low-Reynolds-number region, being matched to the $k \sim \varepsilon$ Boussinesq viscosity model as soon as all the nodes on a circumferential row were located in fully turbulent fluid. Initially, a fully elliptic treatment was applied but later the application of PSL over the three nodes nearest the wall reduced overall computing time by at least a factor of two without significantly altering the resultant solution. The fine-grid solution gave levels of secondary flow some 25% higher than predicted by wall functions, Fig. 5. With these higher swirl velocities the computed heat-transfer behaviour displays a performance akin to that of the measurements. In Fig. 6 it is seen that with ten flutes with a 15° helix angle the Nusselt number lies some 10% above the smooth tube value at the same Reynolds number (the length scale in both parameters being based on the usual hydraulic diameter) while the friction factor is virtually coincident with the smooth-tube result. Yampolsky's [17] studies have mainly focused on 30 flutes and, for such a large number, securing convergent behaviour of the numerical scheme is proving difficult due to the increased importance of geometry-related sources and sinks in the difference equations. Nevertheless, the numerical result obtained very recently at a Reynolds number of 1.6×10^4 is highly encouraging for it shows the Nusselt number fully 50% larger than in a smooth pipe while the friction factor is only 10% above that for smooth-tube conditions.

The foregoing flow, in common with that in the vicinity of a spinning disc considered by Cobb and Saunders [1] and Richardson and Saunders [21], is one where the direction of the mean velocity vector undergoes an appreciable change in direction within the low-Reynolds-number region. For elliptic flows of this type, use of a fine-grid approach seems essential to reproducing tolerably correctly the near-wall behaviour. A further example is provided by the flow in a tube around a bend where, again, a strong secondary flow is created: fluid near the walls with low streamwise momentum is driven towards the inside of the bend by the centripetal pressure gradient. Mr H. Iacovides has been developing a computational procedure for such flows employing a 'partially parabolic' approach of the type first introduced by Pratap and Spalding [11]. The latter workers considered only bends of fairly mild curvature for which a wall-function approach apparently provided an adequate boundary condition. Iacovides' explorations have been focused on the tighter bends more typical of heat-exchanger practice, however, and here he has found that the peak secondary flow could not be properly resolved with such an approach. On changing over to a fine-grid treatment, it emerged that, for a mean bend radius of only 2.8 diameters, the maximum secondary flow occurred deep in the low-Reynolds-number region at a distance from the wall of less than 2% of the pipe radius, Fig. 7. In this particular flow the facility provided by PSL of allowing a fine near-wall grid for the velocity field without adding to the core demands of the pressure field (which alone is stored three-dimensionally) has been crucial in keeping numerical error small. The turbulence model employed in this study is the same as for the spirally fluted tube, that is, the mixing length hypothesis across the near-wall sublayer matched outside the low-Reynolds-number region to the standard $k \sim \varepsilon$ viscosity model. No experimental data appear to be available yet* with which to compare our predicted secondary velocities (Fig. 7). Enayet *et al.* [22], however, have recently reported streamwise velocity distributions in a 90° bend with a downstream tangent (bend radius of 2.8 pipe diameters) with which comparisons are drawn in Fig. 8. Of the stations at which data were obtained, the greatest distortion of the velocity profiles is seen at one diameter downstream of the bend, the station shown in the figure. Due to the problem of the refractive index of the Perspex tubing not matching that of the fluid, the laser beams were traversed along the non-parallel lines shown. The computational results succeed in

* An experiment by Professor J. A. C. Humphrey and his group at the University of California, Berkeley, is now in progress.

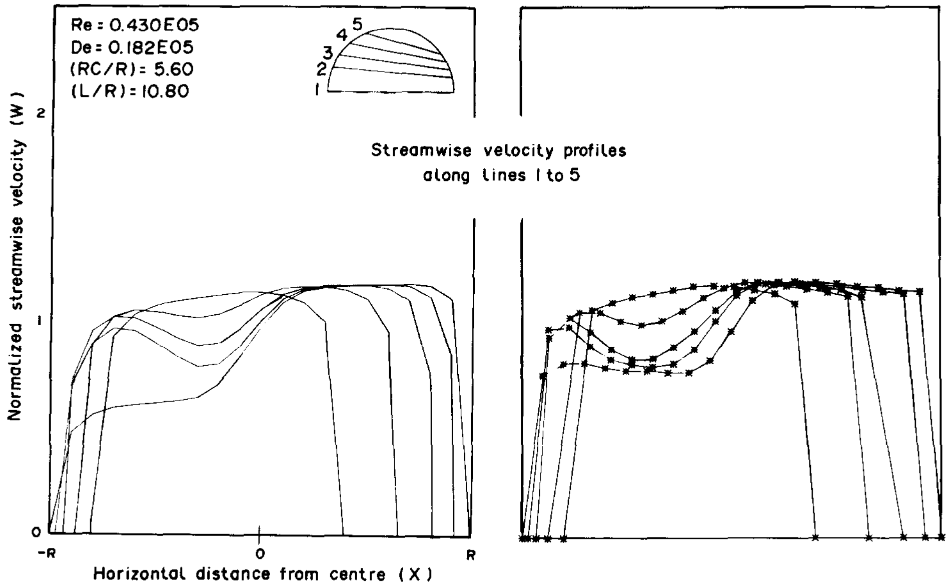

FIG. 8. Normalized streamwise velocity profiles one diameter downstream of 90° bend, $Re = 43 \times 10^3$: ——, computations; *—*, experiment, Enayet *et al.* [22].

mimicking with reasonably good accuracy the pattern of distortion of the streamwise velocity produced by the secondary motion.

At present, the best local pipe heat-transfer data of flow around bends seem to be those of Seban and McLaughlin [23]. The computations of this flow in Fig. 9 show, in common with the experiments, a markedly lower heat-transfer coefficient on the inside than the outside of the bend. This is mainly due to the accumulation of slow-moving fluid on the inside surface and the corresponding 'impingement' of high velocity fluid on the outside. Another agency is also at work, for it is well established that in a 2-D boundary layer, flow over a convex surface damps mixing while a concave surface leads to augmentation [24]. It is also known [24, 25] that unless special flow-dependent adaptations are made, the viscosity model does not reproduce sufficiently this sensitivity of real turbulence to streamline curvature so the overestimate of Nusselt number on the inside of the bend and its

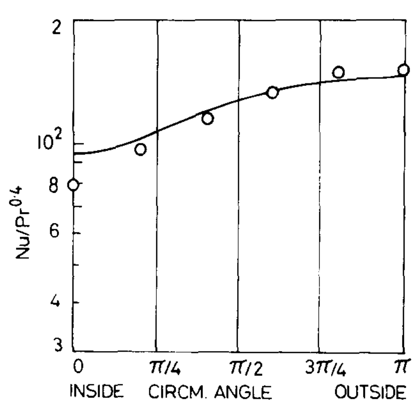

FIG. 9. Nusselt number variation from inside to outside of coiled tube; ——, computations; O, experiment [23].

underestimation on the outside is to be expected. In this study, as in those discussed earlier, one has a reasonable expectation that the accuracy of the predicted behaviour will be measurably and consistently improved by the replacement of the Boussinesq turbulent viscosity hypothesis by models derived from the exact transport equations for the Reynolds stresses and fluxes [26]. Developments in this direction are now underway in the projects discussed above.

4. CONCLUSIONS

The examples presented of computations of convective heat transfer in some complex strain fields indicate that, despite the simplicity of the low-Reynolds-number models so far employed, there are substantial benefits in some applications from adopting a fine grid near the wall in place of the conventional wall-function approach. The arguments for retaining wall functions on the grounds of savings in computer core and time have been weakened by the introduction of the PSL approach which neglects pressure variations across a very thin near-wall sublayer, a scheme which saves time and core compared with conventional elliptic treatments.

Acknowledgements—I wish to thank my research students Anna Barba, Hector Iacovides and Christopher Yap for making available extracts from their recent research results. Fuller accounts of their work are now being documented. The research on heat transfer in spirally fluted tubes and tube bends has been supported by the U.S. Office of Naval Research (Power Program).

I welcome the opportunity to thank Sir Owen for accepting me as a young lecturer in his department in 1964. The chance to work with one of the leading groups in computational heat transfer greatly affected the direction of my subsequent research for, until then, my work as a research assistant at MIT (an appointment, which incidentally, Sir Owen had also been instrumental in arranging) had been in experimental fluid mechanics.

REFERENCES

1. E. C. Cobb and O. A. Saunders, Heat transfer from a rotating disc, *Proc. R. Soc.* **236A,** 343 (1956).
2. J. T. Anderson and O. A. Saunders, Convection from an isolated heated horizontal cylinder rotating about its axis, *Proc. R. Soc.* **217A,** 555 (1953).
3. S. V. Patankar and D. B. Spalding, *Heat and Mass Transfer in Boundary Layers* (1st edn.). Morgan Grampian Press, London (1967).
4. A. D. Gosman, W. P. Pun, A. K. Runchal, D. B. Spalding and M. Wolfshtein, *Heat and Mass Transfer in Recirculating Flows.* Academic Press, London (1969).
5. B. E. Launder and D. B. Spalding, The numerical computation of turbulent flows, *Comp. Meth. Appl. Mech. Engng* **3,** 269 (1974).
6. H. Iacovides and B. E. Launder, *Trans. Am. Soc. Mech. Engrs, J. Fluids Engng* **106,** June (1984).
7. C. C. Chieng and B. E. Launder, On the calculation of turbulent heat transport downstream of an abrupt pipe expansion, *Num. Heat Transfer* **3,** 189 (1980).
8. J. R. Viegas and C. C. Horstman, Comparison of multi-equation turbulence models for several shock-separated boundary-layer interaction flows, AIAA Paper 78-1165, Seattle, Washington (1978).
9. J. R. Viegas and T. J. Coakley, Numerical investigation of turbulence models for shock-separated boundary-layer flows, AIAA Paper 77-44, AIAA 15th Aerospace Science Meeting, January (1977).
10. J. R. Viegas and M. W. Rubesin, Wall-function boundary conditions in the solution of the Navier–Stokes equations for complex compressible flows, AIAA Paper 83-1694, 16th Fluid and Plasma Dynamics Conf. (1983).
11. S. V. Pratap and D. B. Spalding, Numerical computations of the flow in curved ducts, *Aeronaut. Q.* **26,** 219 (1975).
12. B. P. Leonard, A stable and accurate convective modelling procedure based on quadratic upstream interpolation, *Comp. Meth. Appl. Mech. Engng* **19,** 59 (1979).
13. T. Y. Han, J. A. C. Humphrey and B. E. Launder, A comparison of hybrid and quadratic upwind differencing in high Reynolds number elliptic flows, *Comp. Meth. Appl. Mech. Engng* **29,** 81 (1981).
14. W. P. Jones and B. E. Launder, The prediction of laminarization with a 2-equation model of turbulence, *Int. J. Heat Mass Transfer* **15,** 301 (1972).
15. J. W. Baughn, M. Hoffman, B. E. Launder and R. Takahashi, Local heat transfer downstream of an abrupt expansion in a circular channel with constant wall heat flux, *Trans. Am. Soc. Mech. Engrs,* Series C, *J. Heat Transfer,* in press.
16. M. Wolfshtein, Some solutions of the plane turbulent impinging jet, *Trans. Am. Soc. Mech. Engrs,* Series D, *J. Basic Engng* **92,** 915 (1970).
17. J. Yampolsky, Spirally fluted tubing for augmented heat transfer, Gen. Atomic Rep. GA-A165442 (1979).
18. A. Barba, G. Bergeles, I. Demirdzic, A. D. Gosman and B. E. Launder, The computation of flow in a spirally fluted tube, *A.I.Ch.E. Symp. Ser. No. 225* **79,** 100 (1983).
19. A. Barba, G. Bergeles, A. D. Gosman and B. E. Launder, The prediction of convective heat transfer in viscous flow through spirally fluted tubes, ASME Paper 83-WA/HT-37 (1983).
20. E. R. Van Driest, *J. Aeronaut. Sci.* **23,** 1007 (1956).
21. P. D. Richardson and O. A. Saunders, Studies of flow and heat transfer associated with a rotating disc, *J. Mech. Engng Sci.* **5,** 336 (1963).
22. M. M. Enayet, M. M. Gibson, A. M. Taylor and M. Yianneskis, *Int. J. Heat Fluid Flow* **3,** 213 (1982).
23. R. A. Seban and E. F. McLaughlin, *Int. J. Heat Mass Transfer* **6,** 387 (1963).
24. P. Bradshaw, Effects of streamline curvature on turbulent flow, *Agardograph* No. 169 (1973).
25. B. E. Launder, C. H. Priddin and B. I. Sharma, The calculation of turbulent boundary layers on curved and spinning surfaces, *J. Fluids Engng* **99,** 231 (1977).
26. P. Bradshaw (editor), Turbulence, in *Topics in Applied Physics,* Vol. 12. Springer, Heidelberg (1976).

CALCUL NUMERIQUE DU TRANSFERT THERMIQUE PAR CONVECTION DANS LES ECOULEMENTS TURBULENTS COMPLEXES: EST-IL TEMPS D'ABANDONNER LES FONCTIONS DE PAROI?

Résumé—En appliquant la méthode des différences finies et d'autres approches numériques des écoulements turbulents ou elliptiques, il est usuel d'adopter des fonctions de paroi pour relier avec une seule cellule la très fine région adjacente à la paroi où la viscosité modifie la structure turbulente. On suggère que de telles approches ne sont pas nécessaires maintenant du point de vue de l'économie de temps de calcul et dans beaucoup de cas elles peuvent être remplacées avantageusement par un traitement à grille serrée, proche de la paroi qui étend les calculs numériques jusqu'à la paroi à travers la couche. Des exemples sont donnés pour trois écoulements fréquemment explorés à UMIST et où l'abandon de l'approche par une fonction de paroi a conduit à une amélioration marquée dans la prévision des flux thermiques convectifs.

NUMERISCHE BERECHNUNG DES KONVEKTIVEN WÄRMETRANSPORTS IN KOMPLEXEN TURBULENTEN STRÖMUNGEN: IST ES ZEIT ZUM VERZICHT AUF WANDFUNKTIONEN?

Zusammenfassung—Bei der Anwendung von finiten Differenzen und anderen numerischen Näherungen zur Berechnung von elliptischen oder dreidimensionalen turbulenten Strömungen ist es üblich, Wandfunktionen zu verwenden, um mit einer einzigen Zelle das sehr dünne Gebiet unmittelbar an der Wand zu überbrücken, indem die Viskosität die Turbulenzstruktur verändert. In der vorliegenden Arbeit wird gezeigt, daß solche Näherungen—jetzt vom Standpunkt der Rechenökonomie her gesehen—unnötig geworden sind und in vielen Fällen vorteilhaft durch ein feineres Netz an der Wand ersetzt werden können, womit die numerische Berechnung durch die Pufferzone bis direkt zur Wand hin ausgedehnt wird. Es werden Beispiele von drei Strömungen dargestellt, die zur Zeit an UMIST untersucht werden und bei denen der Verzicht auf die Wandfunktions-Näherung zu einer merklichen Verbesserung der Vorausberechnung des konvektiven Wärmetransports führte.

ЧИСЛЕННЫЙ РАСЧЕТ КОНВЕКТИВНОГО ТЕПЛОПЕРЕНОСА В ТУРБУЛЕНТНЫХ ПОТОКАХ СЛОЖНОЙ КОНФИГУРАЦИИ: НЕ ПОРА ЛИ ОТКАЗАТЬСЯ ОТ ЗАКОНОВ СТЕНКИ?

Аннотация—При использовании метода конечных разностей и других численных методов для расчета эллиптических или трехмерных турбулентных течений обычно применяются законы стенки на первом шаге в очень тонкой области, прилегающей к стенке, в которой вязкость влияет на структуру турбулентности. Показано, что для экономии времени расчета можно отказаться от этих методов и во многих случаях с успехом использовать метод мелкомасштабной сетки в пристеночной области, с помощью которого можно провести численные расчеты в буферной зоне вплоть до стенки. Даны примеры расчета трех течений, исследуемых в настоящее время в МИНТ, когда отказ от метода, включающего закон стенки, значительно улучшил точность расчета характеристик конвективного теплопереноса.