特 集 1
研究解説
UDC 532.517.4 : 62

Turbulence Models for Practical Applications Part IV
——Examples of Model Applications for 2D Separated Flows——
乱流モデルの工学への適用 その 4
——2次元剝離流への適用例——

Wolfgang RODI* and Shuzo MURAKAMI*
W.ロ デ ィ・村 上 周 三

Examples are presented of calculations performed with the relatively simple turbulence models surveyed in the first half of the paper[1] (Part I, II). The examples cover a wide range of engineering and environmental flows and are subdivided into two-dimensional thin-shear-layer flows, two-dimensional separated flows and three-dimensional flows. In Part IV, calculated results of two-dimensional separated flows are presented and the quality of the calculated results is assessed by comparison with measurements. The paper focusses on applications of the k-ε turbulence model and its variants, but, when available, results obtained with other models are included for comparison.

1. INTRODUCTION

In Part IV, calculation examples are presented of two-dimensional (plane and axisymmetric) flows which are not of the boundary-layer type because they contain separation regions without predominant flow direction. Most calculations of separated flows have been carried out with the standard k-ε model. The wall function approach has been employed even though it is well known that this is not very accurate for recirculating flows, particularly not near separation and reattachment points. However, integration of the equations through the viscous sublayer (with the use of a low-Reynolds-number version) was until recently not possible because of the excessive computer storage and time required in the case of a separated flow calculation. Even outside the viscous sublayer, the numerical grid could often not be chosen so fine as to guarantee grid-independent results. The calculations of separated flows, where the grid lines are usually not aligned with the streamlines, are particularly prone to the effect of numerical diffusion influenced by the use of upwind differencing schemes for the convective terms in the equations. Therefore, some of the test calculations presented may have been affected by numerical diffusion and not all the discrepancies with experiments may be blamed on the turbulence model.

*Visiting Scientist (University of Karlsruhe)
**Center for Development of Instrumentation Technology, Institute of Industrial Science, University of Tokyo

2. FLOW OVER STEPS AND OBSTACLES

Plane channel flow with a sudden expansion (backward-facing step) was used as a test case for the Stanford Conference on Complex Turbulent Flows[2] (the expansion ratio was 1.5). This flow was calculated for the conference with a number of different models. Calculations with both the standard k-ε model and with stress-equation models yielded reattachment lengths of the order of 20% shorter than the measured one, associated with rather poor predictions of the velocity profile in the separated flow region and with significantly too high shear stress in the mixing layer springing off the step edge. Attempts to improve the predictions by using an algebraic stress model and/or a modified ε-equation according to relation (30) were successful only for subregions of the flow. All these calculations used wall functions to bridge the viscous sublayer. Recently, Cordes[3] employed the two-layer turbulence model described in Sec. 3 of part II to a newly investigated backward-facing step test case[4], expansion ratio of 1.125) resolving the viscous sublayer. The resulting streamlines and velocity as well as shear stress profiles are shown in Fig. 1, where they are compared with Cordes[3] calculations obtained with the standard k-ε model employing wall functions and with the measurements[4]. The two-layer model can be seen to predict the reattachment length in much better agreement with the experiments and also produces a small second corner eddy, which is absent in the k-ε model predictions but has frequently been observed in experiments. Also, the velocity and shear stress distributions improve and are in generally good accord with the data in the separation region. In the recovery region, both models predict a somewhat too slow return to developed channel flow.

In the nearest axisymmetric equivalent to the backward-facing step problem, the flow in a pipe whose diameter increases suddenly, the standard k-ε model appears to work satisfactorily. This can be seen from Fig. 2 where Ha Minh and Chassaings'[5] velocity profiles as measured and as calculated with the k-ε model are compared for the case of an expansion ratio $R_1/R_0=2$. The calculations were started at the expansion cross section ($x=0$) with the measured profile also given in Fig. 2. The k-ε model predicts the profile development fairly well in both the recirculation and the recovery region. The same authors' one-equation model calculations (solving only the k-equation and prescribing the length scale L algebraically) are also included in Fig. 2. They agree fairly well with the measurements in the recirculation region but do not so well in the redevelopment region. The one-equation model calculations of Bernard et al.[6] with their supposedly general length-scale formu-

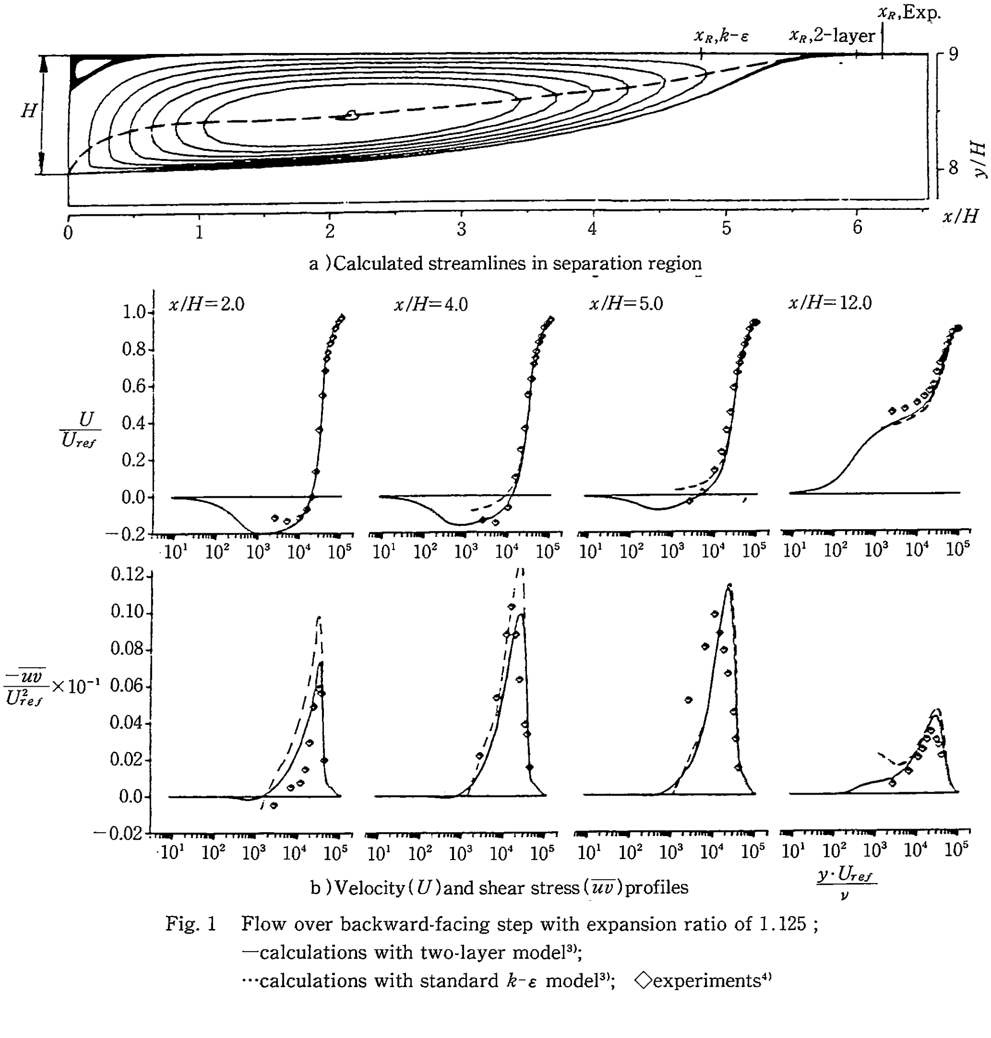


Fig. 1 Flow over backward-facing step with expansion ratio of 1.125 ;
—calculations with two-layer model[3];
···calculations with standard k-ε model[3]; ◇experiments[4]

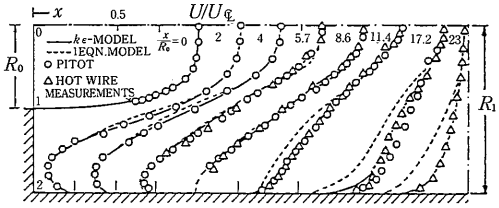
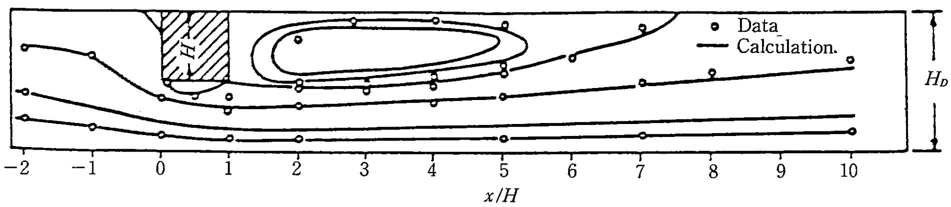


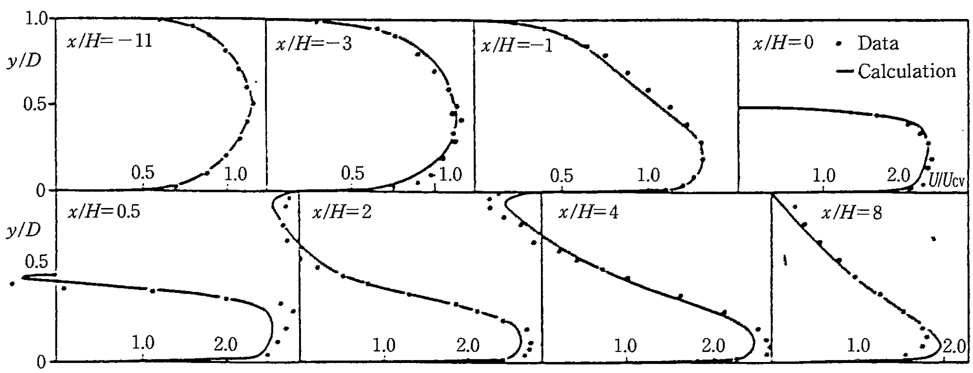
Fig. 2 Velocity profiles in a sudden pipe expansion with $R_1/R_0 = 2$[5)]

lae (for a desription see also[7)]) are, on the other hand, less satisfactory in the recirculation region for this flow. It appears that two-equation models determining the length scale from a transport equation are definitely superior for recirculating flow situations.

Durst and Rastogi[8)] applied the standard k-ε model to calculate the flow in a plane channel obstructed by a square block. The resulting streamlines are compared with the same authors' measurements in Fig. 3. The experiments indicate two small recirculation zones in front and on top of the block and a relatively large one behind the obstacle. All three zones are reproduced by the calculation. Fig. 3 shows the velocity profiles in the region near the obstruction and it can be seen that the disturbance of the initially developed channel profile upstream of the block is well predicted; the sizes of the recirculation regions on top and behind the block are also reasonably well predicted, but the agreement for the velocity profiles in these regions is not entirely satisfactory. The redevelopment of the profiles towards a fully developed profile (not shown here) is further predicted too slow. In a more recent study, Durst and Rastogi[9)] calculated situations with smaller blockage ratio H/H_D (defined in Fig. 3) in which case the opposing wall looses its influence on the flow around the obstacle. For these situations, Durst and Rastogi obtained somewhat too small separation regions with the standard k-ε model. They achieved reasonable agreement with experiments for these cases, including the situation of a very thin obstacle (fence) when they introduced the Richardson-number curvature correction (28) of Launder et al.[10)] discussed in part I of the paper. They also found that rather small mesh sizes were necessary in the vicinity of the obstacle in order to obtain accurate numerical solutions. Recently, Benodekar, et al.[11)] published similar calculations of the flow over a square obstacle and a fence without significant influence of an opposing wall. They did not employ the upwind differencing scheme prone to



a) Streamlines

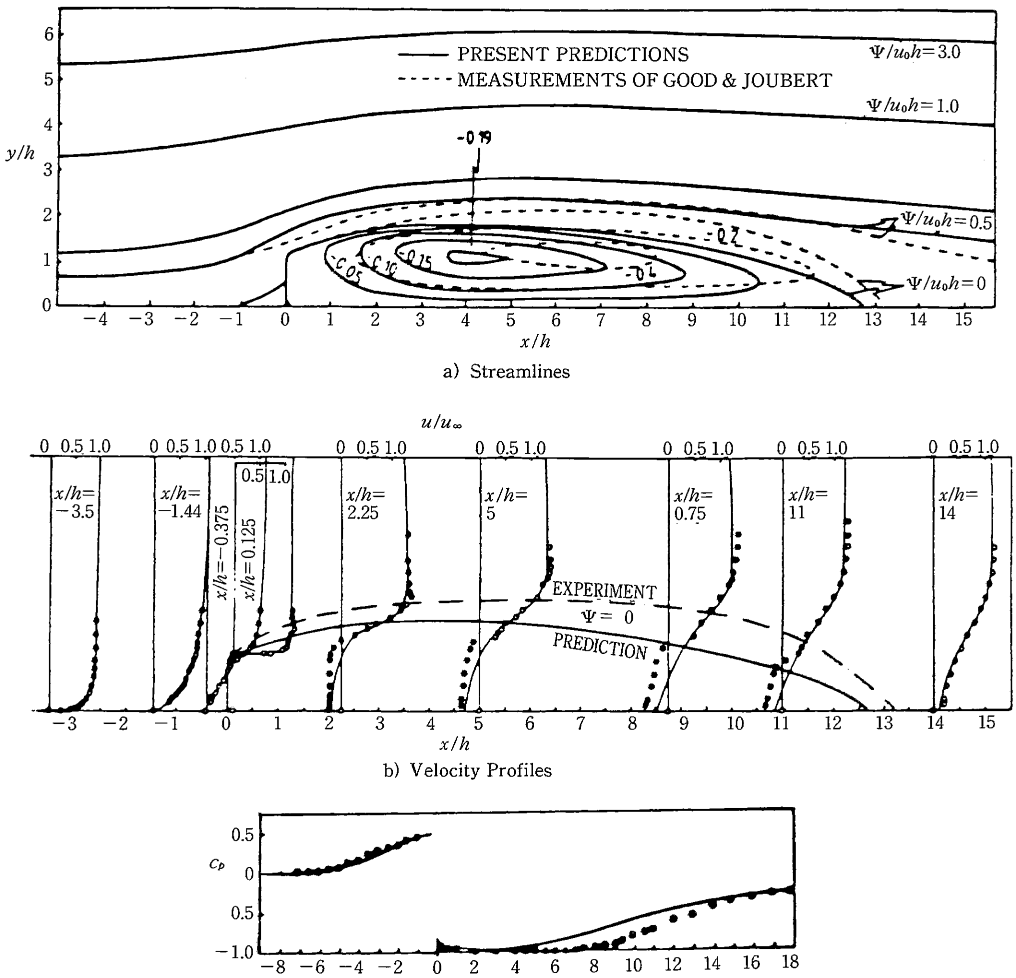


b) Velocity profiles in recirculation region

Fig. 3 Flow over a square obstacle placed in a plane channel[8)] numerical diffusion but a higher-order bounded skew upwind differencing scheme which is virtually free of numerical diffusion. Accordingly, they consider their numerical solutions as accurate and grid-independent. They further employed a modified version of the k-ε model, which combines the curvature correction to the c_μ-coefficient (46) with the modification (29) for the ε-equation. With this model, they obtained a velocity field for the flow over a square obstacle which is in fairly close agreement with the measurements in the near-field of the obstacle. As in the case of Durst and Rastogi's[8] results, the recovery (in this case towards a developed boundary layer profile) is predicted too slow. For the situation of the flow over a fence, Benodekar et al.[11] reported underprediction of the recirculation length by about 15% when the standard k-ε model was used. Replacement of this by the modified k-ε model improved the results so that the predicted recirculation length was now within 4% of the measured one. However, the resulting width of the separation zone is still somewhat underpredicted, as can be seen from the streamlines and vertical velocity profiles shown in Figs. 4a and 4b, respectively. The distribution of the wall static pressure displayed in Fig. 4c agrees well with the experimental data only in the upstream region but discrepancies exist downstream of the fence.

3. FLOW IN ENCLOSURES

Murakami et al.[13] used both the standard k-ε



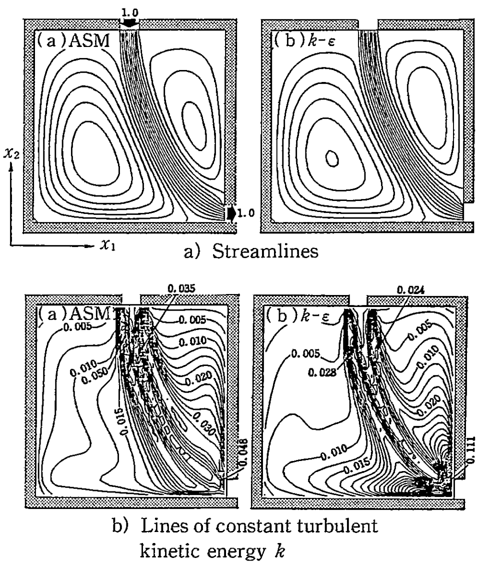
a) Streamlines

b) Velocity Profiles

c) Wall static pressure distribution

Fig. 4 Flow over fence, from 11) ; —calculations[11], ··● experiments[12] model and an algebraic stress extension to calculate the 2D air flow in a square room, with the inflow in the middle of the ceiling and the outflow through a slot in one of the side walls located right at the bottom. The geometry is illustrated in Fig. 5a which also exhibits the calculated streamlines, clearly showing the establishment of a curved jet-type main flow between inlet and outlet and of recirculation zones in the room. The differences in the streamlines between the 2 model predictions are relatively small. However, Fig. 5b showing the lines of constant turbulent kinetic energy k indicates a significantly different behaviour in k near the outlet with its stagnation regions. In these regions there appears to be excessive kinetic energy production with the k-ε model which is absent in the algebraic stress model predictions.

Hutchinson et al.[14] calculated the flow in the axisymmetric model combustion chamber sketched in Fig. 6 with the standard k-ε model. An experimental situation without combustion was simulated where all the fluid entered the combustion chamber through the annular inlet (i.e. no center jet was present) with a swirl component superposed. In this case, a complex flow is set up with several recirculation zones. The jet wants to entrain fluid which has to come from downstream and this leads to reverse flow near the combustion chamber walls and to a toroidal recirculation eddy in the corner. On the other hand, the reverse flow and the associated recirculation in the center region near the inlet are caused by the swirl which sets up a radial pressure gradient to balance centrifugal forces and leads to a low pressure region at the center, drawing fluid into this region from downstream. As can be seen from Fig. 6, all these complex phenomena are reproduced fairly well by the calculation. However, as the flow development is largely pressure-dominated, this may not so much be a merit of the turbulence model.



b) Lines of constant turbulent kinetic energy k

Fig. 5　Prediction of air flow in square room with standard k-ε and algebraic stress model[13]

4. UNCONFINED SEPARATED FLOW

The calculation of separated flows in the nearwake of bluff bodies is very demanding of both the turbulence model and the numerical scheme. In most flows of this type, the separation region is fairly short compared with the height of the obstacle so that the streamlines are strongly curved. As a consequence, streamline-curvature effects on the turbulence model are likely to be important, and the numerical diffusivity introduced by an upwind differencing scheme is likely to be substantial unless exceedingly fine grids are used. Also, the phenomenon of vortex shedding may be present, which is difficult if not impossible to cope with a steady calculation method. Leschziner and Rodi[15] calculated the flow in the wake of a disk surrounded by an annular jet, as sketched in Fig. 7. They investigated the performance of various numerical schemes and variants of the k-ε model and found that the hybrid central/upwind differencing

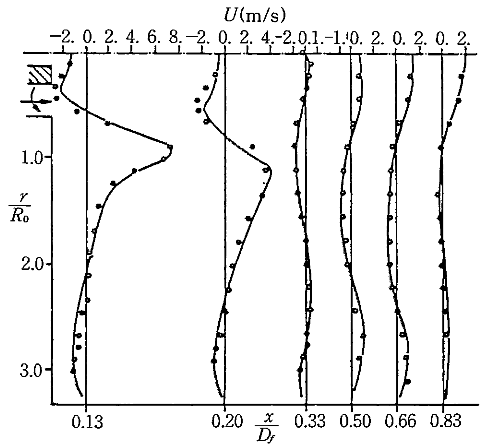
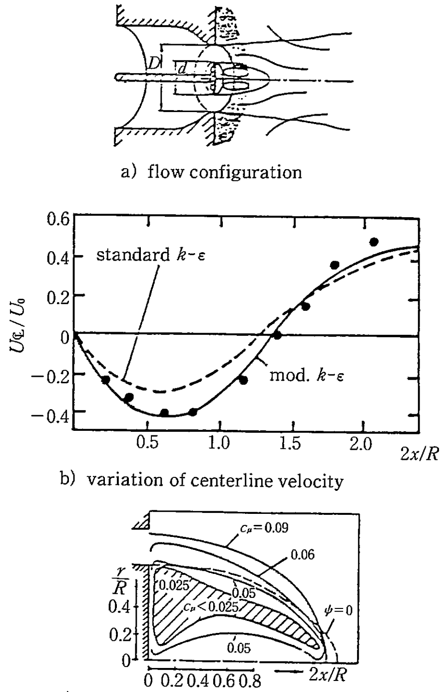


Fig. 6　Velocity profiles in model combustion chamber (from 14))

scheme is not suitable for unconfined recirculating flows (unless exceedingly fine grids are used). Accurate numerical solutions could be obtained, however, with two higher-order schemes investigated. Fig. 7b compares the axial variation of the center-line velocity calculated with a higher-order scheme and the standard k-ε model with the measurements of Durao[16]. The length of the recirculation zone is predicted somewhat too short, and the maximum negative velocity is not large enough. This indicates that the standard k-ε model overpredicts the eddy viscosity level in this flow. Leschziner and Rodi also carried out calculations with the curvature correction (46) to the c_μ-coefficient in the model and separately with a modified ε-equation employing the ε-production term (29). Both modifications yielded very similar results, which are included in Fig. 7b. The agreement with the experimental data is now fairly good. The c_μ-distribution resulting from the correction (46) is shown in Fig. 7c. The correction leads to a significant reduction in c_μ around the separation streamline, i. e. the curved shear layer bordering the recirculation zone. Within this zone, c_μ would tend to zero or even negative values, whence a (arbitrary) lower limit of $c_\mu = 0.025$ was imposed in the calculations. It should be mentioned that a combined use of modifications (29) and (46), which appears to work well for flow over surfacemounted obstacles[11], led to an overcorrection in this case.

Majumdar and Rodi[17] attempted to calculate the flow around a circular cylinder by solving the steady-state Navier-Stokes equations and simulating the Reynolds stresses in these with the standard k-ε model. They obtained a separation zone considerably larger than the measured one and a base pressure which is markedly too high so that the predicted drag coefficient is too small. Majumdar and Rodi concluded that the relatively poor predictions are due to the use of a steady-state model with symmetry conditions imposed at the centre plane. This model does not account for any vortex shedding occuring in reality and hence also not for any interaction of the shed vortices across the centre plane, which increases significantly the effective shear stress. In fact, the calculations correspond more to a situation with a splitter plate located at the centre plane behind the cylinder inhibiting the interaction of the periodic vortex motion. From these results it can be concluded that a steady calculation scheme is not really suitable for separated flow situations with pronounced vortex shedding. Such flows should be simulated with an unsteady method resolving the periodic vortex-shedding motion.



a) flow configuration

b) variation of centerline velocity

c) contours of c_μ for curvature correction

Fig. 7 Annular jet with wake behind disk —, ···calculations[15], ●data[16]

5. CONCLUSION REMARKS

The standard k-ε model is the most widely tested model for separated flows. The application examples have shown that the accuracy of the results is not as good as in the case of shear layers, but that it is often sufficient for practical purposes. Confined axisymmetric flows are predicted well by the model, and the same applies to confined plane flows in which the influence of an opposing wall was felt strongly (i. e. large blockage ratios). When the opposing wall is moved further away, the model tends to underpredict the size of the recirculation zone; the same is true also for most unconfined separated flows. Various modifications to the k-ε model have been suggested to improve its performance under such conditions, but none of them have been tested sufficiently to allow a clear recommendation. The curvature modifications (28) and (46) appear to work quite well for the flows for which they have been tested. When vortex shedding is a dominant feature in separated flow, the use of the k-ε model in a steady calculation procedure significantly underpredicts the effective momentum exchange, leading to poor predictions with the separation region predicted too long. Very few calculations of flows with larger separation regions have been reported with simpler models in which the turbulent length scale is prescribed empirically or calculated with some supposedly general algebraic relation. As a general empirical length scale prescription seems not feasible and the algebraic relations proposed did not prove sufficiently universal, the simpler models are judged not very suitable for such flows. They may however be used as near-wall submodels in a two-layer model. Reynolds-stress equation models have so far been tested little for separated flows. From the few applications reported, there is at present no strong evidence that these models are clearly superior to the k-ε model.

ACKNOWLEDGEMENTS

This paper was initiated while the first author (WR) was a guest at the Institute of Industrial Science of the University of Tokyo; he should like to express his deep gratitude for the overwhelming hospitality received during his stay at the Institute.

(Manuscript received, October 24, 1989)

REFERENCES

1) Rodi.W. and Murakami, S., "Turbulence Models for Practical Applications Part I ~ Part III" SEISAN-KENKYU Vol. 41, No. 8, 1989. 8, Vol. 41, No. 9, 1989, Vol. 41, No. 12, 1989. 12.
2) Kline, S.J., Cantwell, B. and Lilley, G.M., Eds., Proc. 1980-81 AFOSRHTTM Stanford Conference on Complex Turbulent Flows, 1982.
3) Driver, D.M. and Seegmiller, H.L., "Features of a Reattaching Turbulent Shear Layer in Divergent Channel Flow", AIAA Journal, Vol. 23, pp. 163-171, 1985.
4) Cordes, J., Private communication, 1988.
5) Ha Minh, H., and Chassaing, P., "Perturbations of Turbulent Pipe Flow", in Turbulent Shear Flows I, Springer Verlag, New York, 1979.
6) Bernard, J.P., Denoyelle, J.F., Huffenus, J.P. and Peube, J.L., "La Simulation de la Turbulence pour le Calcul des Ecoulements de Convection Mixte", Rept. ALSTHOM Technique des Fluides, Grenoble, 1974.
7) Rodi, W., Turbulence Models and Their Application in Hydraulics, Book publication of the Int. Association for Hydraulic Research, Delft, The Netherlands, 1980.
8) Durst, F. and Rastogi, A.K., "Theoretical and Experimental Investigation of Turbulent Flows with Separation", in Turbulent Shear Flows, Springer Verlag, Heidelberg, 1979
9) Durst, F. and Rastogi, A.K., "Turbulent Flow over Two-Dimensional Fences", in Turbulent Shear Flows II, Springer Verlag, Heidelberg, 1980.
10) Launder, B.E., Priddin, C.H. and Sharma, B.R., "The Calculation of Turbulent Boundary Layers on Spinning and Curved Surfaces", ASME J. of Fluid Eng., Vol. 99, 1977, pp. 321-239.
11) Benodekar, R.W., Goddard, A.J.H., Gosman, A.D. and Issa, R.I., "Numerical Prediction of Turbulent Flow over Surface-Mounted Ribs", AIAA J., 23, 1985, pp. 359-366.
12) Good, M.C. and Joubert, P.N., "The From Drag of Two-Dimensional Bluff Plates Immersed in the Turbulent Boundary Layers", J. Fluid Mech., 31, 1968, pp. 547-582.
13) Murakami, S., Kato, S., and Kondo, Y., "Simulation of Air Flow with and without Buoyancy by Means of Algebraic Second Moment", SEISAN-KENKYU, Vol. 41, No. 1, pp. 11-20.
14) Hutchinson, P., Khalil, E.E., Whitelaw, J.H. and Wrigley, G., "The Calculation of Furnace Flow Properties and their Experimental Verification", ASME J. Heat Trans., Vol. 81, 1976, pp. 276-282.
15) Leschziner, M.A. and Rodi, W., "Calculation of Annual and Twin Parallel Jets Using Various Discretization Schemes and Turbulence-Model Variations", ASME J. Fluid Eng., Vol. 103, 1981, pp. 352-360.
16) Durao, D.F.G., "The Application of Laser Anemometry to Free Jets and Flames with and without Recirculation", Ph. D. thesis, University of London, 1976.
17) Majumdar, S. and Rodi, W., "Numerical Calculations of Turbulent Flow Past Circular Cylinders", in Proc. 3rd Symposium on Numerical and Physical Aspects in Aerodynamic Flows, Long Beach, California, January 1985.