ERCOFTAC WORKSHOP ON

DATA BASES AND TESTING OF CALCULATION METHODS FOR TURBULENT FLOWS

UNIVERSITY OF KARLSRUHE, GERMANY, 3-7 APRIL 1995

J.CH. BONNIN, T. BUCHAL, W. RODI, UNIVERSITY OF KARLSRUHE

1. INTRODUCTION

This workshop was an outcome of the research project «Data Validation and Comparison in Fluid Mechanics» financed by the CEC Programme SCIENCE and coordinated by the organisors of the workshop (for a progress report on the project see ERCOFTAC Bulletin 22, 1994). The aim of the project, in which 7 ERCOFTAC members participated, was to collect experimental and numerical data on turbulent flows, to check the data for their reliability and suitability for test cases, to set up test cases and perform calculations with various turbulence models and to create a data bank from which the data can be accessed. The project work has yielded data for over 70 flows, and 15 well documented test cases have been set up. It seemed beneficial to the European CFD community to offer some of these test cases to a broader circle of computors for testing their calculation methods. Hence, a workshop was organised towards the end of the project, and 5 test cases were issued and computors invited to submit their results to be presented and compared at the workshop. This is in line with 3 previous ERCOFTAC/IAHR Workshops on Refined Flow Modelling which have succeeded 14 IAHR workshops with the same title. At these, only one or at most two test cases were considered; with the availability of a variety of cases established in the SCIENCE project it seemed natural to associate the workshop at the end of the project with the ERCOFTAC/IAHR series. Also, the workshop seemed an ideal opportunity for invited experts to present reviews of the state of the art in turbulent flow computations to an interested audience.

The workshop was attended by 64 persons from 16 countries, with 34% coming from industry.

In the following, a brief report is given on the presentations at the workshop, the test cases issued, the results submitted and the conclusions that can be drawn.

2. WORKSHOP PROGRAMME

On the first day, an overview of the work carried out in the SCIENCE Project was given by W. Rodi. Thereafter, the 70 flow cases for which data have been collected and stored in a data bank were briefly introduced case by case by J. C. Bonnin and T. Buchal. On the third day, an introduction to the data bank created within the project at the University of Surrey, UK, was given by J. C. Bonnin and A. Ciani, including a demonstration on a workstation illustrating the access to and the contents of the data bank. On separate days, 5 review lectures were given by leading experts in the field of computational fluid dynamics:

- Developments in turbulence modelling for industrial applications by B.E. Launder.
- Second-moment closure for complex 3D flows - implementations and applications by M.A. Leschziner.
- Importance of turbulence model development and validation for industrial applications by D. Laurence.
- Importance of CFD code validation: An industrial view by M.V. Casey.
- Large-eddy simulation: Its role in improving and understanding prediction methods for turbulence by P.R. Voke.

The first two speakers, from the group at UMIST (UK) which has played a leading role in the development of the turbulence models used today in industry revealed some recent directions of the work carried out in their laboratory, mainly from the view point of applicability and suitability of turbulence models for engineering problems. The third and fourth speaker, who came both from industry (EDF and Sulzer, respectively) emphasized the importance of the validation of CFD codes and of the turbulence models involved. The last speaker from the University of Surrey presented interesting results of large-eddy simulations and discussed the suitability of this method as a predictive tool for industry in the near future.

A major portion of the workshop was devoted to the submitted calculations of the 5 test cases issued. Each test case was dealt with on a separate day. First the test case and the experiments on which it is based were described in some detail. Then, each contributor who had submitted calculation results had the opportunity to briefly present the main features of his calculation procedure. After this, the organisors, supported by a group from the Electricité de France, presented and compared the calculated results obtained by the various contributors followed by a detailed discussion. In separate lectures, calculations were presented and discussed for 3 additional test cases:

- Vortex-shedding flow past a square cylinder by W. Rodi and G. Bosch, using various versions of the k–ε turbulence model,
- Flow through a circular to rectangular transition duct by T. J. Craft and F.S. Lien using the latest versions of the Reynolds-stress model developed at UMIST,
- Boundary-layer flow in diverging and converging ducts by J.C. Bonnin , using again the k-ε model.

The workshop was concluded by a final discussion in which test cases for future workshops were preselected, and finally the opening of the SCIENCE project data bank at the University of Surrey on May 1, 1995, was announced; the data bank can be accessed through

http://fluindigo.mech.surrey.ac.uk

3. TEST CASES, SUBMISSIONS AND RESULTS

Following the well established procedure of previous ERCOFTAC/IAHR workshops, the prescription of all geometrical parameters and boundary conditions that should be adopted by the computors, the detailed experimental and in one case DNS data and the instructions for presenting results were distributed upon request to a large number of possible contributors: more than 110 all over the world. Internet facilities were used as much as possible: Almost all the data files were sent via electronic mail and the calculation results were submitted via FTP transfer to a workstation set up at the institute of the organisors. In the end, 36 computor groups met the deadline of February 28, 1995, and more than 500 MBytes of disk space were necessary to store the contributed result files, descriptions of numerical methods and grid representations. These also filled more than 600 pages of a booklet that was distributed to the participants at the workshop. Some of the groups submitted results obtained with various turbulence models, and the following table gives an overview on the submissions for each of the 5 test cases.

Case	Contributions	Groups	Turb. Models
1	16	8	12
2	56	20	23
3	13	8	11
4	8	7	6
5	6	5	3

In summary, the following turbulence models were used: High-Reynolds-number k-ε model, RNG modified version of it, several versions of low-Reynolds-number k-ε models, k-ω model, non-linear k-ε models, algebraic stress models (ASM), Reynolds-stress-equation (RSE) models and large-eddy simulations (LES); where applicable either with (standard or non-standard) wall functions or with a simpler formulation near the wall (two-layer models).

In the following, for each case the test case will be described briefly and the main calculation results summarised with a typical result presented. This summary is based on revised results submitted after the workshop.

Test Case 1: Couette flow with plane and wavy fixed wall

Three subcases were considered which are sketched in Fig. 1a. In cases A and B the walls were plane and the flow developed from channel flow between two fixed walls to Couette flow between a fixed and a moving wall. The difference between cases A and B is the Reynolds number. Experimental results due to Corenflos et al (1993) are available for Re 3D 3000 and 5000, and for the low Reynolds number DNS data are available from Kuroda et al (1993). In case C the upper fixed wall is wavy and periodic so that the flow is also periodic and the measurements have been carried out by Nakabayashi et al. (1991). Case A is characterised by virtually zero shear stress at the moving wall so that the mean velocity has a zero gradient at this wall and there is no shear layer and hence no turbulence production near this wall. Because of the low Reynolds number, wall functions could not be used and the viscous near-wall layer had to be resolved. All models predicted fairly well the development of the mean velocity from the symmetric developed channel flow profile to the strongly asymmetric developed Couette flow profile with zero gradient at the moving wall. The latter developed profile is also reproduced fairly well by all models (see Fig. 1b), the differences being no larger than those between experimental and DNS data. Near the upper wall where the shear production of turbulence takes place, the turbulent kinetic energy k is predicted too low by the Launder-Sharma (LS) low-Re k-ε model (a well known deficiency) and the near-wall peak is much better predicted by most RSE models and also by newer versions of the two-layer k-ε model. Near the lower wall, where no production takes place, there is little difference between the various predictions and they agree fairly well with the data.

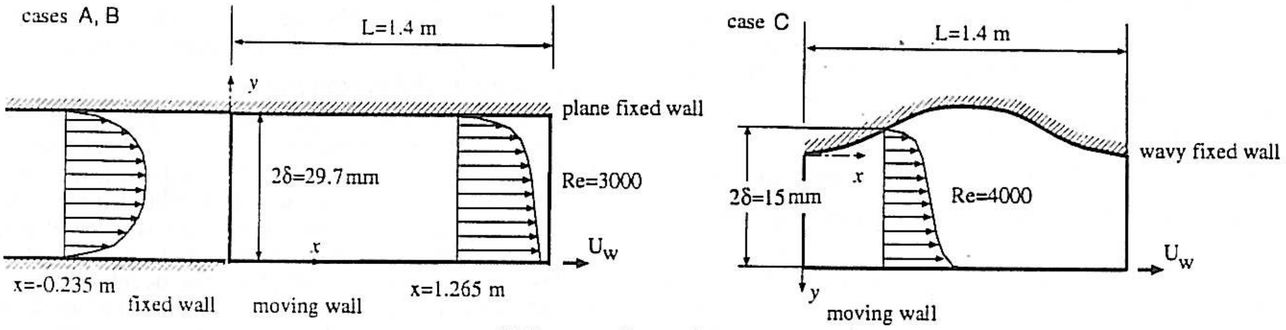
In case B with the higher Reynolds number (Re 3D 5000) a nearly antisymmetric velocity distribution develops in the Couette flow and there is a shear layer also near the bottom wall and turbulence production. In the developed Couette flow, the shear stress should be uniform, but this is not quite so in the experimental distribution which has a small peak near the lower wall. Hence the experimental flow was probably not quite developed and therefore a comparison of the calculations for developed flow with these data cannot be entirely conclusive. The level of calculated shear stress is generally higher than the measured one and so is the predicted k-level in the channel centre and this discrepancy may have to do with the fact that the experimental flow was not fully developed. The velocity distribution is again well predicted by all models. Most k-ε-based models yield a fairly uniform k-distribution while in the experiments there is a pronounced k-peak near the walls (again perhaps partly due to the lack of developed flow), but some k-ε and RSE models produce moderate near-wall peaks.

In case C with an upper fixed wavy wall only one section covering one wave length was simulated and compared with the measurements. Because of the periodicity of the flow, inflow conditions did not have to be specified. Again the velocity profiles are generally well predicted by all models and there are similar differences in the turbulence intensity between the Launder-Sharma k-ε models and RSE models as discussed above. However, it should be mentioned that LS model calculations submitted by different computors produced quite different distributions in the friction velocity along the wavy fixed wall, indicating a sensitivity to the details of the numerical treatment.

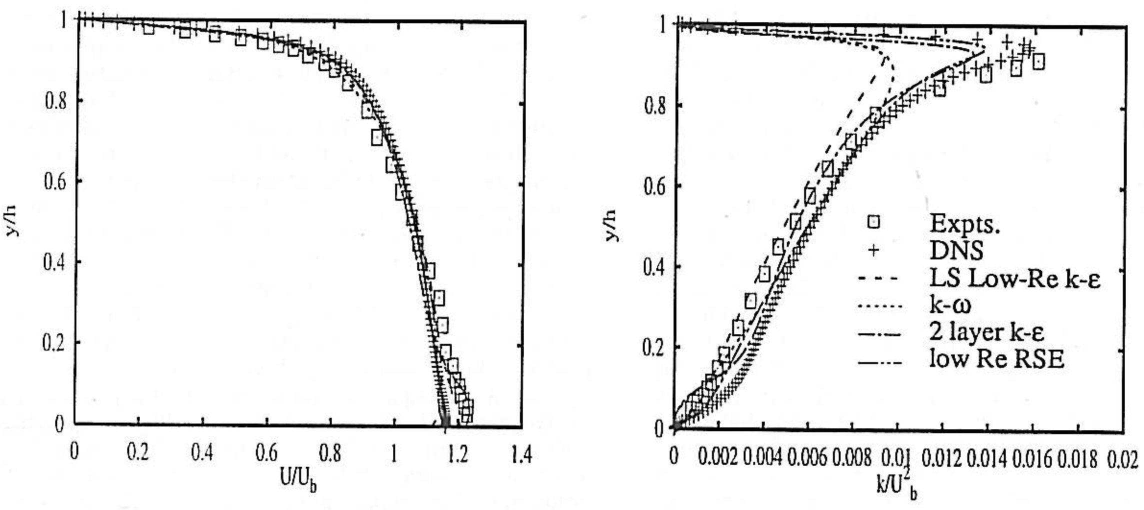
Test Case 2: 2D model hill flows

Subcase 2A concerns the flow over a single 2D hill (geometry see Fig. 2) placed in a channel and subcase 2B the flow over a series of consecutive hills with the same geometry. The opposite channel wall is 6 hill heights from the bottom wall and the oncoming flow was developed channel flow. The measurements are due to Almeida et al. (1993). A fairly large recirculation zone develops behind the hills. In case 2A with a single hill, calculations were started 3.6 hill heights upstream of the hill centre using the developed channel-flow measurements at this station as inflow conditions. Altogether 38 different results were submitted for this case. For 7 calculations, the predicted streamlines are shown and compared with the experimental ones in Fig. 2. Altogether 6 groups submitted results obtained with the standard k-ε model using standard wall functions. Of these, 5 are reasonably close concerning the streamlines, and the calculations of U. Delft shown in Fig. 2 are a typical example. In these calculations the separation zone is predicted too short by 25 to 40%. One calculation with nominally the same model (U. Chalmers) yielded a much smaller separation region (see Fig. 2) and the reasons for this could not be determined. When using the RNG version of the k-ε model, a longer, more realistic recirculation region was obtained. Also, resolving the viscosity-affected near-wall layer with either a low-Re version of the model or with a one-equation model in a two-layer approach yields longer recirculation lengths in fairly good agreement with the experiments. This, and the results obtained with some non-standard wall functions show clearly that the prediction of the separation zone depends strongly on the near-wall treatment. The 4 k- ω model calculations agree very well among each other concerning the streamlines and predict a separation region which is too thick and about 30% too long. Calculations obtained with standard RSE models using wall functions yield longer separation regions than the k-ε model, which are however still a little shorter than experimentally observed. The quality of the predictions of the velocity profile in the lee of the hill is of course closely related to that of the recirculation zone. All models underpredict the level of turbulent kinetic energy k and shear stress $\overline{uv}$ in the separated shear layer, and the RSE models are lowest. Further downstream after reattachment, where the observed k-and $\overline{uv}$ levels drop, most models pick up these quantities better; superior results are obtained here again when the viscous near-wall layer is resolved. In the ascending flow and on the hill crest, most models generate an unrealistic peak in the U-velocity near the hill wall and excessive k and $\overline{uv}$. The latter quantities (but not U) are predicted fairly realistically by the RSE models and altogether the k-ω model produces better results in this region than the k–ε model.

For case 2B with consecutive hills (placed 4.5 hill heights apart) measurements were carried out between the 7th and 8th hill (of a series of 10), and from the experimental paper it appeared that the flow was developed and periodic in this region. Hence, in the test case specification it was suggested to assume periodicity conditions. Such conditions were used in most of the calculations submitted (17 entries); however, a number of contributors raised doubts about the flow being developed and periodic after the 7th hill and performed non-periodic calculations (5 entries), either by calculating the flow over the full series of hills or by consecutive hill-by-hill solutions. The standard k-ε model with wall functions produced fairly realistic streamlines while models resolving the near-wall region and RSE models produced too long separation zones. These remarks apply both to the periodic and non-periodic calculations, the latter giving the better results. The velocity profiles are predicted better in the non-periodic calculations than in the periodic ones, the RSE model giving too large negativevelocities in the valley, as was to be expected from the streamlines. The U-velocity produced by the LES in the upper half of the channel exhibits strange high values.However, the non-periodic LES gives the most realistic k-distribution while the othernon-periodic calculations produce generally too low k-levels near the hill wall, the RNG and RSE models being considerably lower than the k-ε model results. Similar remarks apply to the shear-stress distribution.

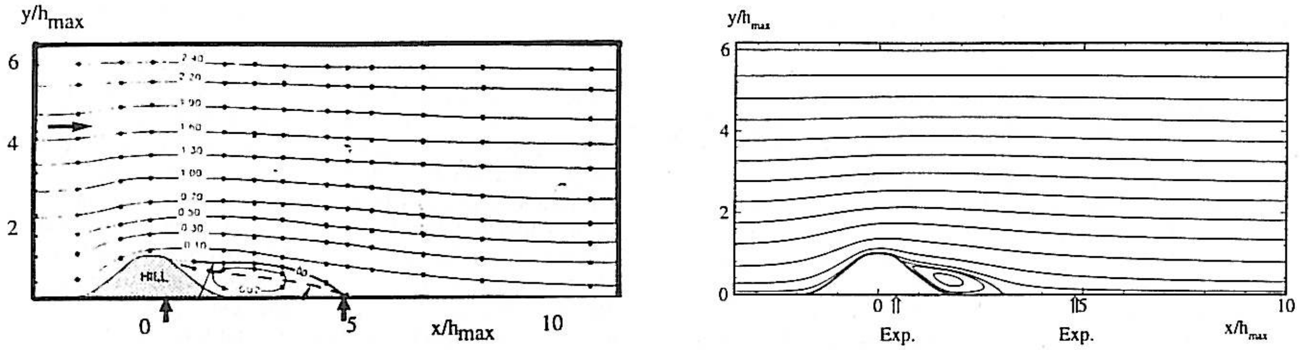


a) Flow configuration



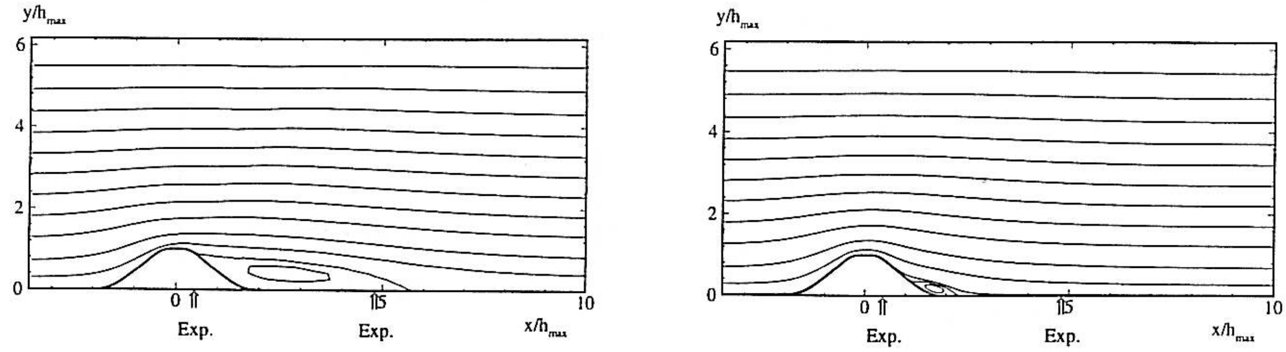
b) Velocity (left) and turbulent kinetic energy (right) profiles in developed Couette flow of case 1A

Figure 1: ***Test case 1: Couette flow with plane and wavy fixed wall***



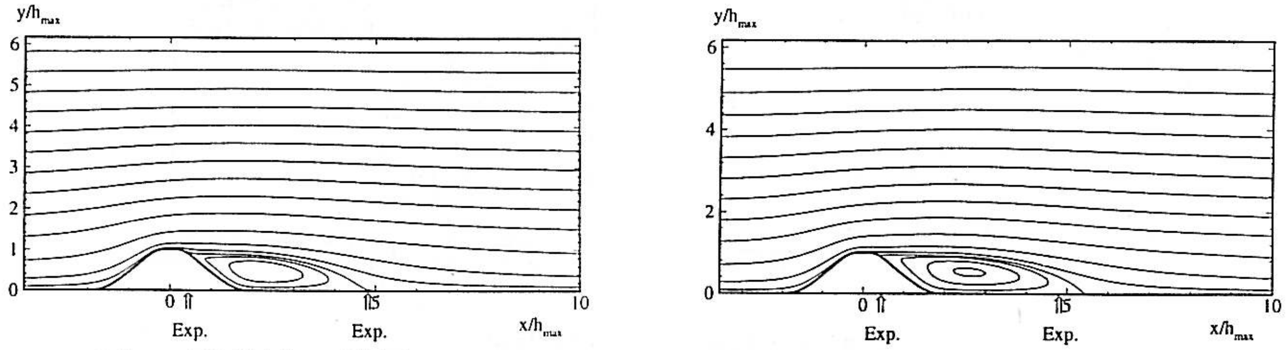
Experiments

k-ε U. Delft



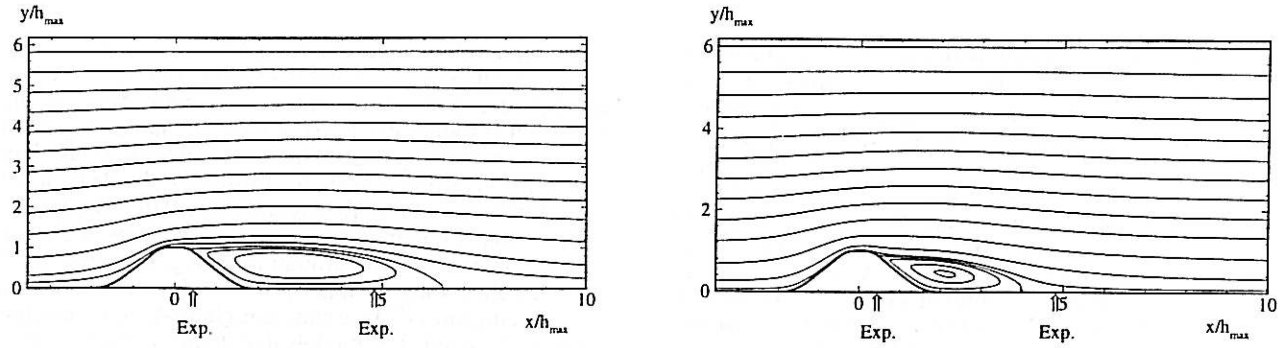
RNG UMIST

k-ε U. Chalmers



2-layer RMM-k-ε U. Florence

LS-low-Re-k-ε U. Chalmers



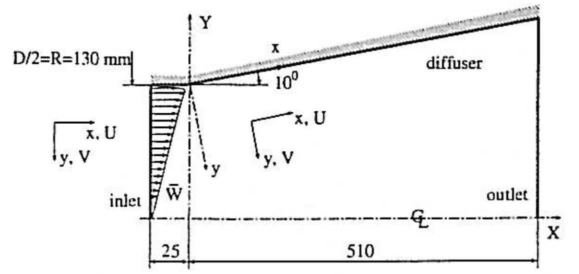
k-ω U. Florence

LRR-RSE U. Delft

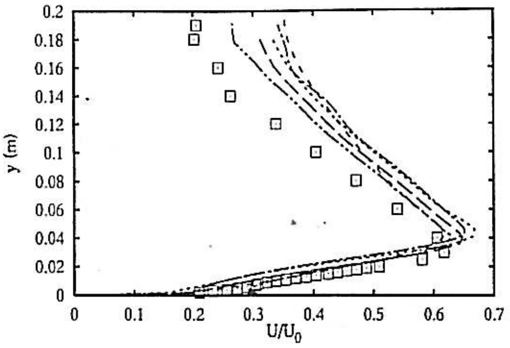
Figure 2: ***Test case 2: 2D model hill flows; streamlines for case 2A single hill***

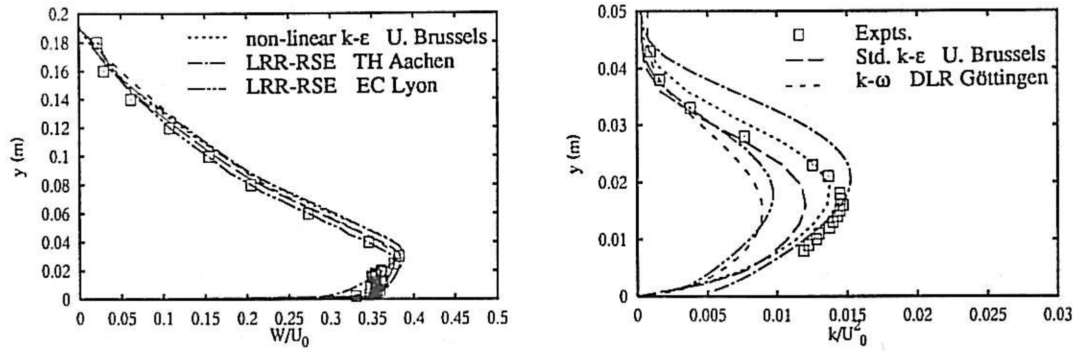
Test case 3: Swirling boundary layer in conical diffuser

This case concerns swirling flow in a 20° conical diffuser (see Fig. 3a) as measured by Clausen et al. (1993). At the entrance to the diffuser, the streamwise velocity was nearly uniform except in the boundary layer occupying 10% of the radius. Outside of the boundary layer the flow was in solid body rotation. The swirl was strong enough to prevent separation on the diffuser wall; it causes a reduction of the streamwise velocity on the axis but was not strong enough to cause flow reversal there. The calculations were started 0.2 R upstream of the diffuser entrance using measured profiles. In Fig. 3b, predicted U, W and k profiles are shown for some of the models and compared with the measurements at station x 3D 0.33m downstream of the diffuser entrance. Most models initially predict the U and W velocity profiles quite well, but towards the diffuser outlet, the reduction in U-velocity in the centre is underpredicted by most models. There are significant differences between the two RSE entries using nominally the same Launder, Reece and Rodi model. The Lyon calculations give the best lowering of U in the centre but this is strangely associated with smaller shear stresses and k-levels than in other models. As can be seen from the k-profiles in Fig.3b, there are large differences in the turbulence quantities predicted by the two groups with the same model. The best agreement concerning the turbulence quantities was obtained by U. Brussels with the standard k-ε model and with non-linear k-ε model versions. Altogether it seems that the standard k-ε model with wall functions gives results in this case which, apart from an insufficient reduction in the centre-line velocity, are reasonable for all quantities that are of practical interest. More complex models do not seem to yield a marked improvement in this test case which does not have a very strong swirl.



a) Flow configuration



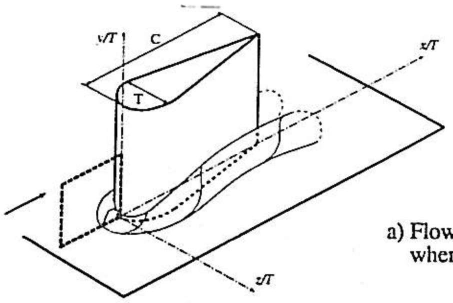


b) Profiles of U, W and k at x = 0.33 m

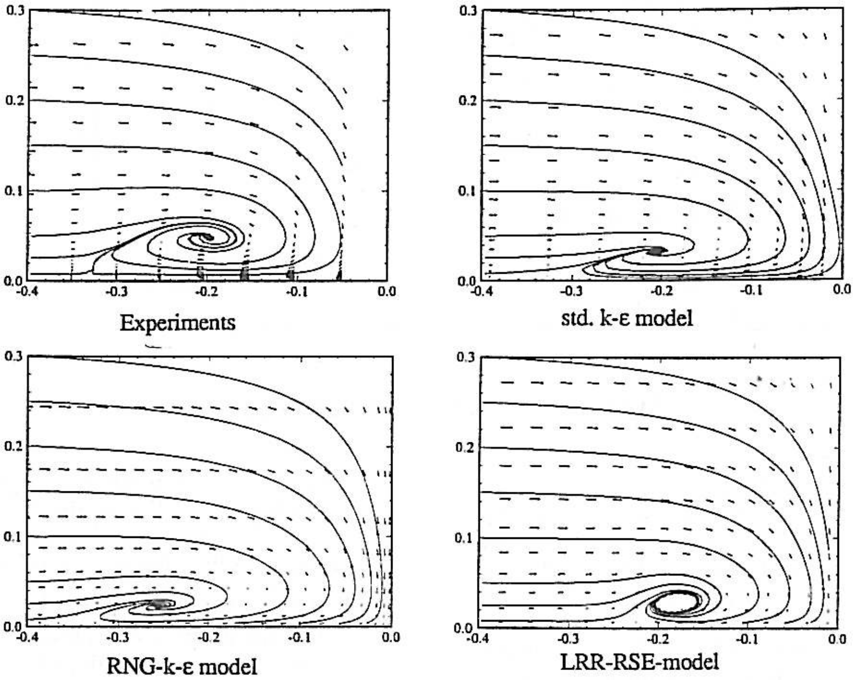
Figure 3: ***Test case 3: Swirling boundary layer in conical diffuser***

Test Case 4: Wing/body junction with separation

This case concerns the flow around a cylindrical airfoil mounted on a flat ground plate (see Fig. 4a). The experiments are due to Fleming et al. (1993) who measured the streamwise and secondary flow velocities as well as the Reynolds stresses in several vertical planes. The boundary layer on the flat plate separates when it approaches the leading edge of the wing and a horseshoe vortex is formed which sweeps around the wing in the junction corner (see Fig. 4a). For this three-dimensional case, 7 computors submitted results; 6 of them employed versions of the k-ε model and one the Launder, Reece and Rodi RSE model. 5 of the groups used wall functions and 2 the two-layer approach. In Fig. 4b, velocity vectors and streamlines constructed from these are given in the symmetry plane in front of the wing for 3 calculations (all using wall functions) together with the experimental results. It can be seen that the general flow picture is well captured by the various models but that the separation vortex is predicted too thin and, particularly with the RSE model, also too short. The experiment shows fairly high turbulent kinetic energy k in the vortex region with low values in the immediate stagnation region while most models have the maximum of k more towards the corner and produce excessive k in the stagnation region in front of the leading edge. This is a well known problem with the standard k-ε model, but in this case the RSE model also suffers from this, albeit to a lesser extent. The best k-distribution is obtained with the RNG version of the k-ε model. When sweeping around the wing, the strength of the horseshoe vortex is underpredicted by all models, the vortex being more diffuse and the higher k-levels being too close to the wing. The models that do not yield excessive k in the stagnation region (RNG version and to a lesser degree RSE) are best. In the plane immediately downstream of the trailing edge of the wing, size and strength of the vortex are also underpredicted. k now has two maxima, one towards the outer edge of the vortex and the other in the wake behind the trailing edge; this feature is picked up by all models and best reproduced by the RSE model, while the RNG model now gives too low k.



a) Flow configuration and plane where results are shown



Experiments

std. k-ε model

RNG-k-ε model

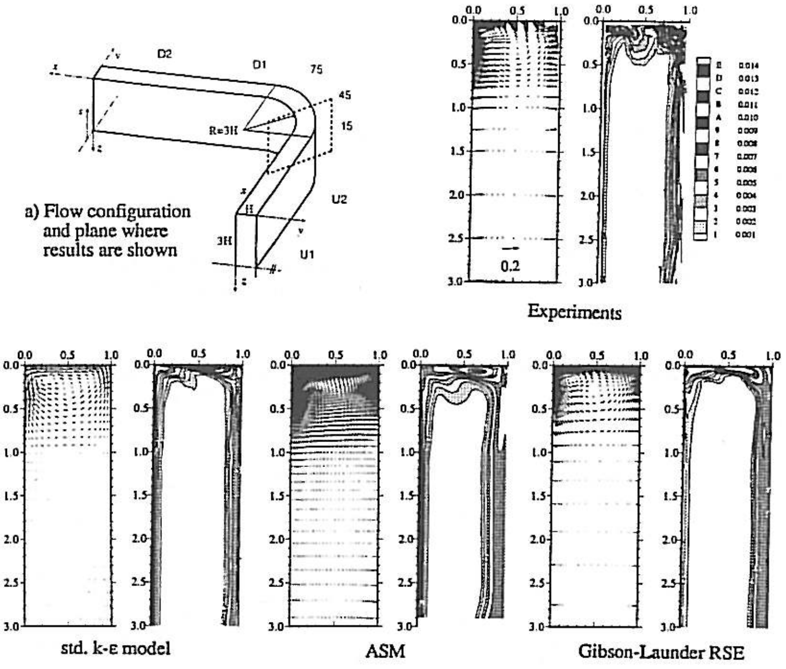
LRR-RSE-model

b) Secondary flow velocity vectors and streamlines in symmetry plane in front of wing

Figure 4: ***Test case 4: Wing/body junction with separation***

Test Case 5: Developing flow in a curved rectangular duct

This case concerns the flow through a 90° bend of a rectangular duct with aspect ratio 6 (duct height equals 6 x duct width H - see Fig. 5a). The radius of the inner wall is $R^i = 3H$ and the duct has a straight section of length 7.5 H upstream of the bend and of 25.5 H length downstream of the bend. Detailed measurements of the velocity components and Reynolds stresses are reported in Kim and Patel (1994). The calculations were started 4.5 H upstream of the bend, where detailed velocity and Reynolds-stress measurements are available. Originating from the wind tunnel contraction, secondary motions (2 counterrotating vortices) are present at this cross-section near the top and bottom wall. Calculations were carried out with various versions of the k-ε model, including non-linear and algebraic stress (ASM) variants and with the Gibson-Launder RSE model. Since the Reynolds number was high (U_0H/ν = 224000), all models employed wall functions. From the initial station to the beginning of the bend, the secondary flow vortices stemming from the wind-tunnel contraction are diffused more in the calculations than in the experiments, but least so with the ASM model. In the bend, near the top and bottom wall the boundary layer thickens on the inner wall and a complex secondary flow pattern develops, leading to complex contours of streamwise velocity and turbulence quantities. This is qualitatively quite well predicted by most models; it seems that the ASM and RSE models give the closest accord with the measurements (see Fig. 5b). Away from the top and bottom walls, two effects influence the development of the boundary layers in the bend: the inner wall boundary layer is first exposed to a favourable pressure gradient and then to an adverse one and has convex curvature; the outer wall boundary layer is first exposed to an adverse and then to a favourable pressure gradient and has concave curvature. Hence, a strong asymmetry develops, and k is reduced near the inner wall and increased near the outer one due to the effect of curvature. This is again best predicted by the RSE and ASM models while in fact the k-ε model leads to an unrealistic increase of k near the inner wall. At the end of the bend, there is a wide region of high kinetic energy on the outer wall whose extent is not well reproduced by the models, ut still best by the ASM model. The friction coefficient also develops a strong asymmetry between inner and outer wall as one moves through the bend, first giving higher values on the inner wall (due to the difference in pressure gradient) and at the end of the bend giving higher values on the outer wall (due to combined effects of differences in pressure gradient and curvature). This development is not predicted consistently well by any model and there are fairly large differences between the 3 calculations obtained with the standard k-ε model and wall functions. Altogether, the results for this test case are not very conclusive, whence it was decided to issue this case once more for the next workshop.



a) Flow configuration and plane where results are shown

Experiments

std. k-ε model

ASM

Gibson-Launder RSE

b) Secondary velocity vectors (left) and contours of turbulent kinetic energy (right) in bend at 45°

Figure 5: *Test case 5: curved duct flow*

CONCLUDING REMARKS

The workshop has shown that there is considerable interest, also in industry, in test calculation exercises and that developers, vendors and users of CFD codes are prepared to participate in such exercises. The workshop has shed light on the performance of a wide variety of turbulence models for calculating a number of flows of engineering relevance and, even though it was difficult to reach general conclusions, a number of useful observations could be made among which are the following: the more complex Reynolds-stress equation and also algebraic-stress models reproduced in some cases better the details of turbulence quantities, but for the test cases considered these models were not consistently better than simpler two-equation models as far as the mean quantities are concerned. In the single-hill flow, the prediction of the separation behaviour was improved when the near-wall region was resolved rather than treated by wall functions and also by moving to an RSE model, but this was not the case for the series of hills. The difficulty in reaching general conclusions was partly due to the fact that the results obtained with nominally the same turbulence model were sometimes quite different, the experiments were on closer look not exactly for the specified situations, and particularly for 3D flows the results were difficult to compare and assess. For future workshops, the experiments must be scrutinized even more, and efforts must be taken to avoid or at least explain major differences in results obtained by different computors with the same model. The next workshop (5th ERCOFTAC/IAHR Workshop on Refined Flow Modelling) will take place from April 24 to 26 at EDF in Chatou and will have test case 5 (curved duct) again but also a 2D wall jet, a natural convection boundary layer and natural convection flow in tall cavities. The authors should like to thank Dr. D. Laurence and his colleagues at EDF for their help in processing, presenting and interpreting a part of the results submitted. The workshop was financially supported by COMETT II through UETP ERCOFTAC, COST, the University of Karlsruhe and EDF.

The 600 page folder containing all the revised (after the workshop) calculation results and descriptions of the calculation methods can be purchased from:

Professor W. Rodi, Institute for Hydromechanics
University of Karlsruhe, Kaiserstr.12
76128 Karlsruhe, Germany
E-mail: Rodi @bau-verm.uni-karlsruhe.de

An order form can be obtained electronically from:

ftp://ftp-ifh.bau-verm.uni-karlsruhe.de/workshop/Proceedings/Order-form

The information contained in the folder (30 MByte) is also available online from:

ftp://ftp-ifh.bau-verm.uni-karlsruhe.de/workshop/Proceedings

REFERENCES

K. Corenflos, S. Rida, J.C. Monnier, P. Dupont, K. Dang Tran, M. Stanislas: Experimental and numerical study of a plane Couette-Poiseuille flow as a test case for turbulence modelling. Engineering Turbulence Modelling and Experiments 2, Rodi and Martelli Eds., Elsevier Publ., 499-508, 1993.

A. Kuroda, N. Kasagi and M. Hirata: Direct numerical simulation of turbulent plane Couette-Poiseuille flows: effect of mean shear on the near wall turbulence structures.Symp. on Turb. Shear Flows 9, Kyoto, Japan, 8-4-1, 1993.

K. Nakabayashi, O. Kitoh and H. Iwata: Turbulent Couette type flow with an alternating pressure gradient. Symp. on Turb. Shear Flows 8, Munich, poster I-13, 1991.

G.P. Almeida, D.F.G. Durao and M.V. Heitor: Wake flows behind two-dimensional model hills. Exp. Thermal and Fluid Science, 7, 87, 1993.

P.D. Clausen, S.G. Koh and D.H. Wood: Measurements of a swirling turbulent boundary layer developing in a conical diffuser. Exp. Thermal and Fluid Science, 6,39, 1993.

J.L. Fleming, R.L. Simpson, J.E. Cowling and W.J. Devenport: An experimental study of a turbulent wing-body junction and wake flow. Exp. in Fluids 14, 366, 1993.

W.J. Kim and V.C. Patel: Origin and decay of longitudinal vortices in developing flow in a curved rectangular duct. J. of Fluids Engineering, 116, 45, 1994.