Prediction of the turbulent flow in a diffuser with commercial CFD codes

By Gianluca Iaccarino

1. Motivation & background

There have been a few attempts in the literature to compare the performance of commercial CFD codes; for instance, laminar and turbulent test cases have been proposed to several CFD code vendors by the Coordinating Group for Computational Fluid Dynamics of the Fluids Engineering Division of ASME (Freitas, 1995). A series of five benchmark problems were calculated, with all the mesh generation and simulations performed by the vendors themselves; only two of the problems required turbulent simulations. The first of these benchmarks is the flow around a square cylinder: the flow is unsteady and all of the codes predicted the measured Strouhal number reasonably well. However, poor accuracy was obtained in the details of the wake flow field. It was also noted that, depending on the code used (and assuming grid-converged results), the *same* k-ϵ model predicted very different results (from 2% to 16% accuracy in the Strouhal number, for example). The reasons for this difference include different grids, non-demonstrated grid convergence, different implementation of the models, and different boundary conditions. It must be pointed out that the predictions for this problem are strongly affected by the treatment of the stagnation point region: as reported in Durbin (1996), the k-ϵ models predict a *spurious* high level of turbulent kinetic energy near the stagnation point.

The other turbulent problem reported in Freitas (1995) was the three-dimensional, developing flow in a 180-degree bend. In this case all of the solutions reported were unsuccessful in predicting the measured data in the bend region. The resolved structure of the flow field was significantly affected by the choice of the turbulence model.

The uncertainties associated with different computational grids (*i*), boundary conditions definition (*ii*), convergence (*iii*), and numerical schemes (*iv*) do not allow specific conclusions to be drawn from the comparison other than that *further research into more advanced turbulence models for use in commercial CFD codes is required* (Freitas, 1995).

In order to complete a fair comparison between different CFD codes and to establish concrete conclusion on the state-of-the-art of commercial CFD codes, all of the differences (*i*-*iv*) must be fully addressed and, if possible, eliminated. In the present work an effort has been made to control all of these parameters. The codes available to us for comparison are CFX, Fluent, and Star-CD. The objective is to compare their predictive capabilities for the simulation of a turbulent separated flow. Several turbulent closures are available in these codes, ranging from k-ϵ-type models to full Reynolds stress closures. In addition, various near-wall treatment are available in the codes. Two models are selected: the k-ϵ Low-Reynolds model by Launder & Sharma (1974) and the $\overline{v^2} - f$ by Durbin (1995).

The test case is two-dimensional, turbulent flow in a diffuser. Due to the adverse pressure gradient, the flow is separated and a long recirculation bubble exists. This problem has been selected because a very reliable experimental database is available. In addition to laboratory data, a detailed large eddy simulation (LES) study was carried

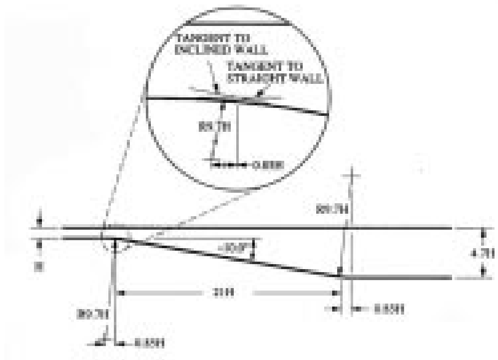

FIGURE 1. Asymmetric diffuser geometry.

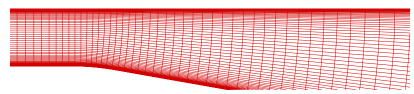

FIGURE 2. Computational grid.

out at the Center for Turbulence Research, and the resulting numerical database is also available for comparison.

2. Numerical method

The incompressible Navier-Stokes (NS) equations are solved, together with additional equations for turbulent quantities as are needed to compute an eddy viscosity. The discretized equations are solved in a segregated manner with the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm or its “consistent” variant, SIMPLEC (Vandoormaal, 1984), used for stability to achieve the pressure-velocity coupling. In the SIMPLE algorithm, the continuity equation is converted into a discrete Poisson equation for the pressure.

The solution procedure can be described as follows: the differential equations are linearized and solved implicitly in sequence, starting with the pressure equation, followed by the momentum equations, by the pressure correction equation, and finally by the equations for the scalars (turbulence variables). Within this loop, the linearized equations for each variable are integrated using a linear system solver.

Three codes have been used in this work: CFX, Fluent, and Star-CD. All of the codes allow the user to implement customized models through user defined subroutines.

In all of the simulations SIMPLEC has been used (SIMPLE for Star-CD) with QUICK (Leonard, 1979) space discretization for the mean flow and first order upwinding for the turbulence equations. The same under-relaxation factors were used in all the calculations.

Several turbulence models are available in these codes; most of them are derived from the standard k-ϵ model with different treatment of the wall region.

The Low-Reynolds model of Launder & Sharma (1974) and the $\overline{v^2} - f$ model (Durbin, 1995) are used in this work. The first model is available as a standard option in all the codes (even if slightly different damping function are employed in Star-CD); the $\overline{v^2} - f$ model has been implemented using the user defined subroutines in each of the codes. Such implementation ensures that exactly the same model is used in each of the codes.

3. Results

The two-dimensional diffuser considered here was a test-case for the 8th ERCOFTAC/IAHR/COST Workshop on Refined Turbulence modeling in Espoo, Finland, 17-18

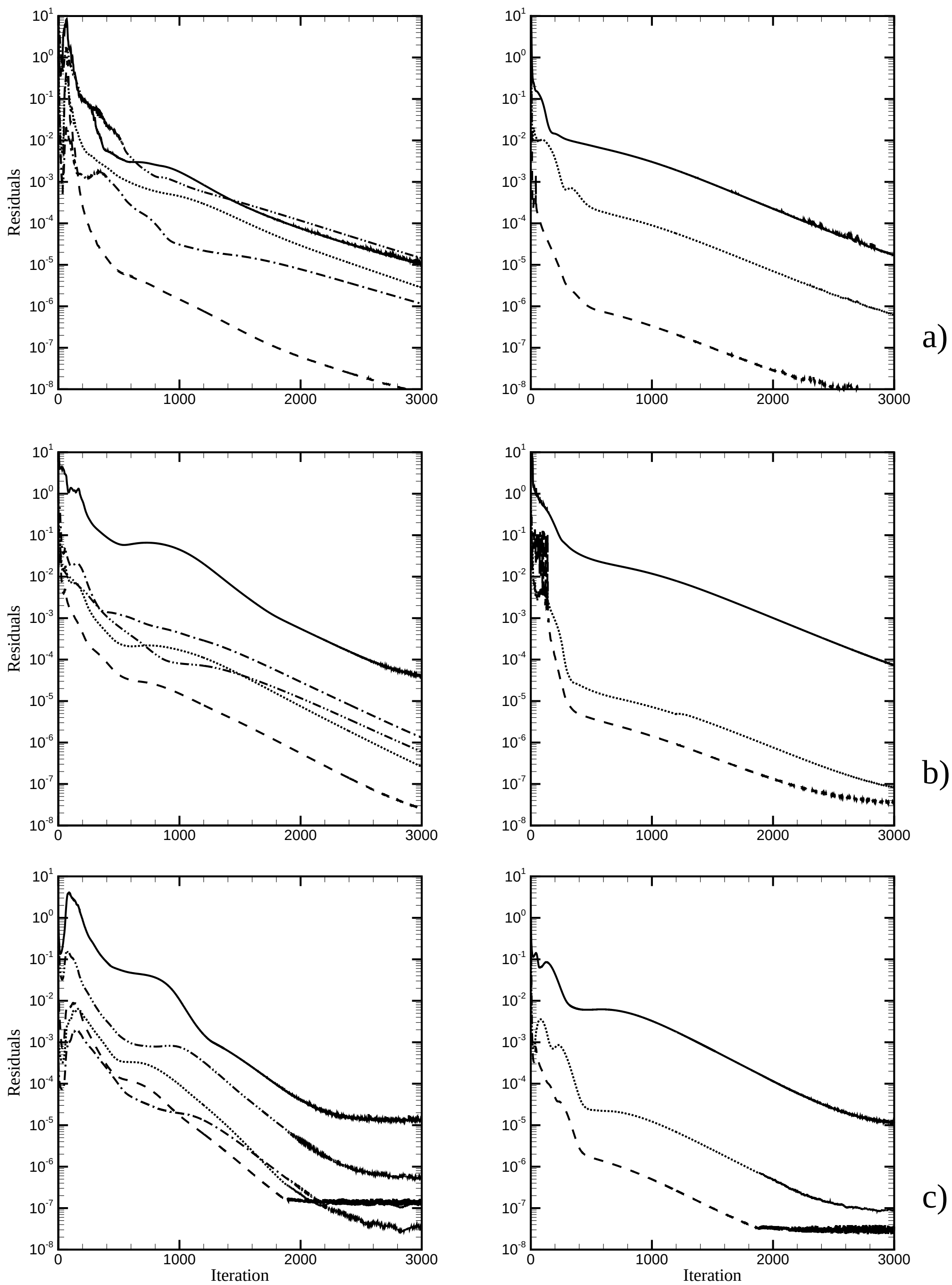

FIGURE 3. Convergence history (L_∞ Norm). Left column: $\overline{v^2} - f$ model; right column: Low-Reynolds k-ϵ model. a) CFX v4.3; b) Fluent v5.3; c) Star-CD v3.1. - Symbols: ——— x-momentum, ········ k, – – – – ϵ, —·— v^2, —··— f

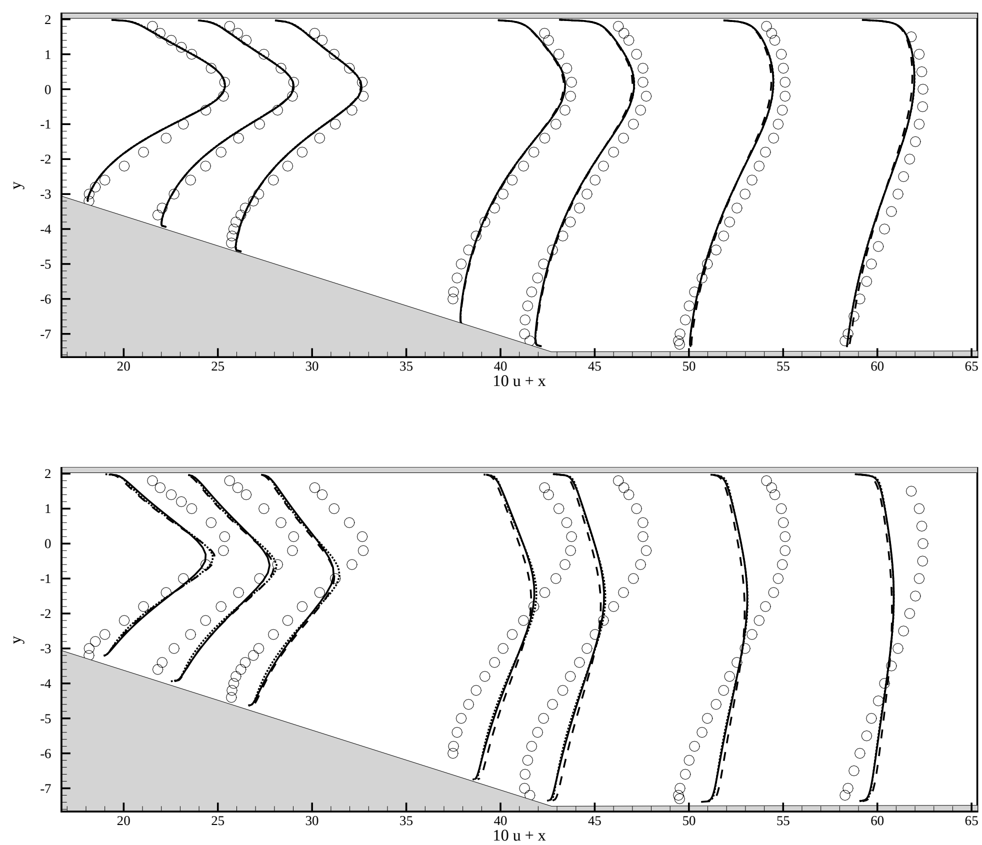

FIGURE 4. Streamwise velocity profiles. Top: $\overline{v^2} - f$ model; bottom: Low-Reynolds k-ϵ model - Symbols: ∘ Experiments, ——— CFX, – – – – Fluent, ········ Star-CD

June 1999. The geometry is presented in Fig. 1 and the inlet conditions are specified as fully-developed channel flow at Re=20,000, based on the centerline velocity and the channel height. Separate channel flow computations were carried out using each code and each turbulence model, and the resulting profiles were used as inlet conditions for the simulation of the diffuser.

An experimental database is available from Obi (1993) and Buice & Eaton (1997); the data include mean and fluctuating velocities at various stations in the diffuser and skin friction on both walls. The data can be obtained directly from the following Website (www.aero.hut.fi/Ercoftac/ws8/case8_2).

A structured grid made up of 124x65 points in the streamwise and wall normal direction, respectively, was used. A detail of the computational grid in the region close to the connection between the channel and the diffuser is included in Fig. 2.

The same grid was also used in the LES study by Kaltenback *et al.* (1999). In this study a very detailed comparison between numerical and experimental data was carried out, and the agreement was very good. The simulations provided insights into the physics of the separation process and provided a verification of the suitability of this problem as a CFD test case.

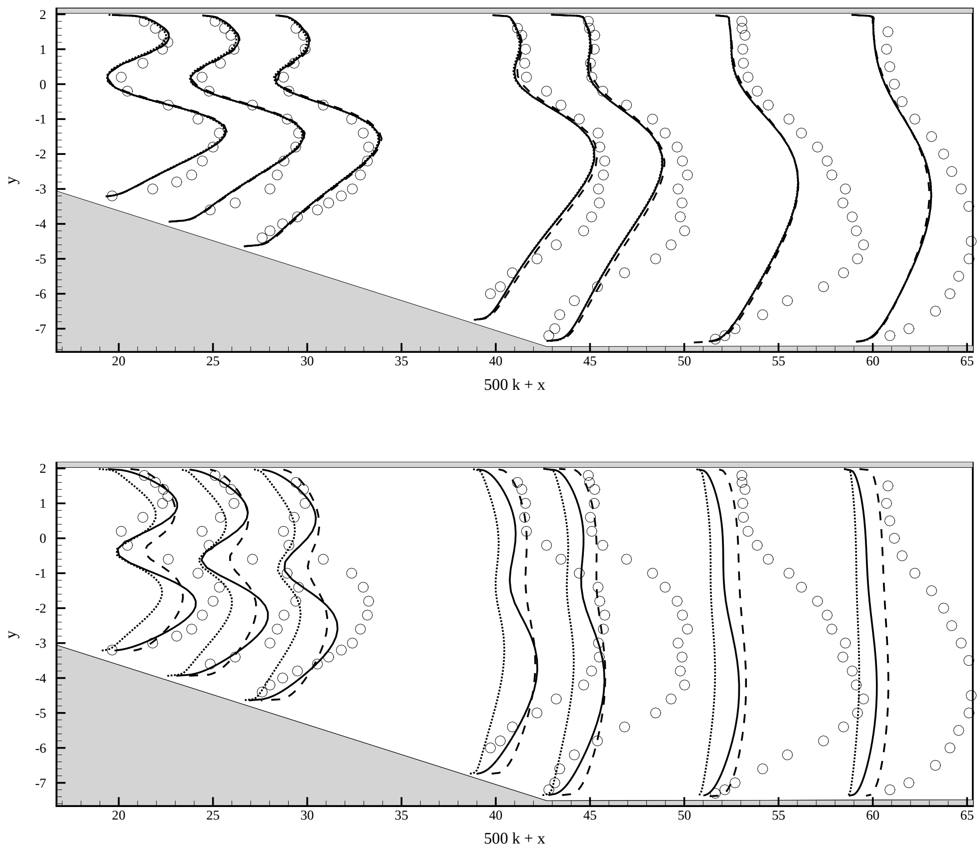

FIGURE 5. Turbulent kinetic energy profiles. Top: $\overline{v^2} - f$ model; bottom: Low-Reynolds k-ϵ model - Symbols: ∘ Experiments, ——— CFX, – – – – Fluent, ········ Star-CD

In Fig. 3, convergence histories for the all simulations are presented. The residuals have been normalized by their values at the first iteration. The convergence level reached after 3,000 iterations is comparable in all the cases, although slightly lower residuals are obtained using the $\overline{v^2} - f$ in both Fluent and CFX, and vice-versa in Star-CD.

A comparison between the computations and the experimental data on mean velocity is reported in Fig. 4 at several stations in the diffuser. The $\overline{v^2} - f$ results are consistently in very good agreement with the measurements for the mean velocity; in particular, the separation zone is captured (even if the maximum intensity of the recirculating velocity is underestimated). The predictions using the k-ϵ model are in poor agreement with the data because the model fails to respond correctly to the adverse pressure gradient and misses the separation completely. The comparisons reported in Fig. 5 confirm the quality of the $\overline{v^2} - f$ predictions as compared to the k-ϵ; in this plot the turbulent kinetic energy is presented.

The peak of the turbulent intensity is very well predicted by the $\overline{v^2} - f$ model in the diffuser, but in the recovery region (after the reattachment), the model underestimates the level of kinetic energy. This is consistent with the $\overline{v^2} - f$ calculations shown in Durbin (1995), the LES results reported in Kaltenback *et al.* (1999), and with the recent

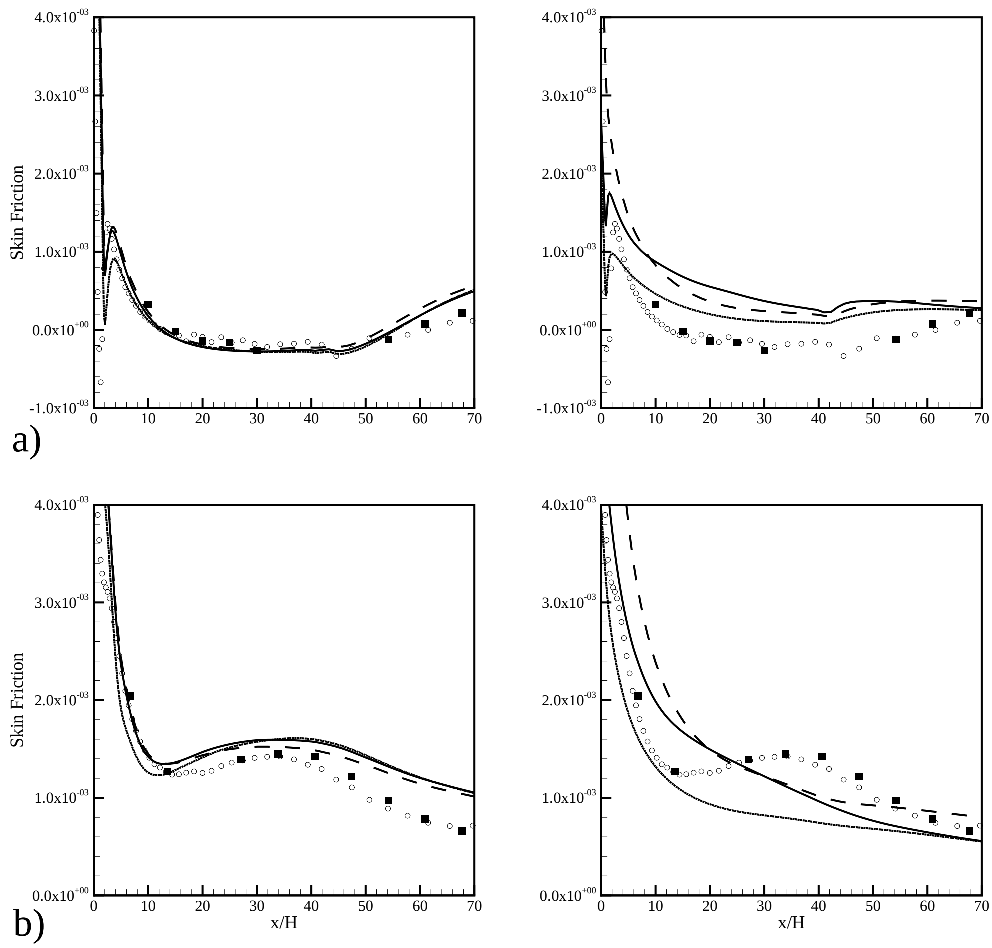

FIGURE 6. Skin friction distribution. Left column: $\overline{v^2}-f$ model; right column: Low-Reynolds k-ϵ model. a) Lower wall; b) Upper wall - Symbols: ∘ LES, ■ Experiments, —— CFX, – – – – Fluent, ········ Star-CD

computations presented in Apsley & Leshziner (2000) using quadratic and cubic non-linear k-ϵ models. One possible reason for this disagreement is the presence of strong three-dimensional effects after the flow reattachment.

The results using the k-ϵ model fail to capture the asymmetric development of the turbulent kinetic energy and underestimate its magnitude in the diffuser. In addition, the three codes show some differences when nominally the same k-ϵ model is invoked. The disparities are in the mean velocity and especially in the turbulence intensity. The very good agreement obtained by using the $\overline{v^2}-f$ suggests that the differences are not related to the numerical technique used, but to the implementation of the turbulence model itself. For example, different discretization or approximation of the source terms in the k-ϵ equations could lead to the mentioned differences.

Finally, in Fig. 8 the skin friction coefficients on the lower and upper wall are reported. The separation bubble on the curved wall is indicated by a negative friction from $x/H \approx 7$

to $x/H \approx 30$. The $\overline{v^2} - f$ model predicts a bubble in very close agreement with the experiments. The k-ϵ model fails to predict any separation (as already noted); in addition, the three codes predict quite different friction levels when the k-ϵ closure is employed.

4. Conclusions

A comparison between three CFD commercial codes has been reported for the turbulent flow in a planar asymmetric diffuser. Two turbulence models have been used: the first is the Low-Reynolds k-ϵ model (with Launder and Sharma damping functions) which is available as a standard feature in the codes; the second is the $\overline{v^2} - f$ model, which has been implemented through user defined routines.

The same grid and the same spatial discretization have been used for all the simulations. In addition, a similar iterative procedure based on the SIMPLE technique has been used. In terms of convergence behavior, all of the codes reach the steady state approximately in the same number of iterations, regardless of the turbulence model used. The accuracy of the calculations as compared to the experimental and LES data is very good using the $\overline{v^2} - f$ model. The length of the recirculation region is captured within 6% and the friction levels on both walls agree reasonably well with the data. The negative velocity in the separation bubble is slightly underestimated. The results using the k-ϵ model do not show any recirculation. The flow is fully attached, and this leads to a severe underprediction of the maximum velocity in the diffuser.

An effort has been made to control all of the aspects of the simulations so that exactly the same results were expected using different codes; in particular, the implementation of the $\overline{v^2} - f$ turbulence model has been carried out consistently. $\overline{v^2} - f$ results do indeed show an almost perfect agreement: CFX and Star-CD predict almost exactly the same results, with Fluent being slightly more dissipative. The results using the k-ϵ model, on the other hand, show a strong sensitivity to the code used. The damping functions used in CFX and Fluent are exactly the ones proposed by Launder and Sharma, but the results are different, especially in terms of turbulent quantities and friction coefficients. This may be due to differences in implementation details which are not specified in the user manuals. In general, the differences between the k-ϵ results are much larger than those obtained using $\overline{v^2} - f$, suggesting that they are less due to details of the numerical procedure used in the code than to the definition of the turbulence model itself.

Today, one of the challenges in using commercial CFD codes is to choose between several physical/numerical models available. The cross comparison presented in this work proves that the basic numerical techniques are reliable and deliver the expected performance in terms of accuracy and convergence. On the other hand, the selection of the correct physical model is crucial for the success of the simulations. Using one of the available turbulence models, the results were not accurate and, in addition, are not reproducible between the codes.

REFERENCES

Apsley, D. D. & Leshziner, M. A. 2000 Advanced Turbulence Modeling of Separated Flow in a Diffuser. *Flow, Turbulence and Combustion.* **63**, 81-112.

Buice, C. U. & Eaton, J. K. 1997 Experimental investigation of flow through an asymmetric plane diffuser. *Report No.TSD-107*, Thermosciences Division, Department of Mechanical Engineering, Stanford University, Stanford, CA, USA.

CFX 4 1999 *User Guide.* AEA Technologies Engineering Software.

Durbin, P. A. 1995 Separated Flow Computations with the $k-\epsilon-v^2$ Model. *AIAA J.* **33**, 659-664.

Durbin, P. A. 1996 On the k-ϵ Stagnation Point Anomaly. *Int. J. Heat and Fluid Flow.* **17**, 89-90.

Fluent 5 1998 *User Guide.* Fluent Inc..

Freitas, C. J. 1995 Perspective: Selected Benchmarks From Commercial CFD Codes. *J. Fluids Eng.* **117**, 210-218.

Kaltenback, H. J., Fatica, M., Mittal, R., Lund, T. S. & Moin P. 1999 Study of the Flow in a Planar Asymmetric Diffuser Using Large Eddy Simulations. *J. Fluid Mech.* **390**, 151-185.

Launder, B. E. & Sharma, A. 1974 Application of the Energy-Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disk. *Letters in Heat and Mass Transfer.* **1**, 131-138.

Leonard, B. P. 1979 A Stable and Accurate Convective Modeling Procedure Based on Quadratic Upstream Interpolation. *Computer Methods in Applied Mechanics and Engineering.* **19**, 59-98.

Obi, S., Aoki, K. & Masuda, S. 1993 Experimental and Computational Study of Turbulent Separating Flow in an Asymmetric Plane Diffuser. *Ninth Symp. on Turbulent Shear Flows*, Kyoto, Japan, August 16-19, p. 305.

Speziale, C. G., Abid, R., Anderson, E. C. 1990 A critical evaluation of two-equation models for near wall turbulence. *AIAA Paper 90-1481.*

Star-CD 1998 Manuals Version 3.1. *Computational Dynamics, LTD.*

Vandoormaal, J. P., & Raithby, G. D. 1984 Enhancements of the SIMPLE Method for Predicting Incompressible Fluid Flows. *Numer. Heat Transfer.* **7**, 147-163.