REPORT 94-24

Finite volume computation
of 2D incompressible turbulent flows
in general coordinates on staggered grids

M. Zijlema, A. Segal and P. Wesseling

ISSN 0922-5641

Reports of the Faculty of Technical Mathematics and Informatics no. 94-24

Delft 1994
Abstract

A brief review of some publications on computation of incompressible turbulent flow in complex geometries is given. A 2D finite volume method for the calculation of turbulent flow in general curvilinear coordinates is described. This method is based on a staggered grid arrangement and the contravariant flux components are chosen as primitive variables. Turbulence is modeled by the standard $k$-$\varepsilon$ model with wall functions. Applications of this method to a variety of flows are presented. The results of computations are compared with experimental data and other numerical solutions and are found to be satisfactory.

1 Introduction

Solving incompressible turbulent flows using a boundary-fitted coordinate system is an important technique in CFD. It permits an easy and accurate implementation of boundary conditions and makes the computation of flows in complex geometries possible. For this purpose two different approaches can be adopted. The first approach is a complete transformation of the Navier-Stokes equations to general coordinates. Such a coordinate-invariant formulation contains many geometric quantities and thus leads to more work and storage. Moreover, the equations involve Christoffel symbols which may lead to inaccuracies on non-smooth grids. Another approach is a partial transformation, in which only the independent variables are transformed while retaining the velocity components as Cartesian components. As a result, the governing equations have a strong conservation form which has a much simpler structure and thus simplifies programming of codes.

The choice of the two approaches mentioned above depends on a grid arrangement: staggered versus colocated. On staggered grids one may choose Cartesian velocity components as unknowns in the Navier-Stokes equations, because of simplicity. But these velocity components, in general, are not perpendicular to grid lines. Therefore, this approach may lead to unstable discretization. Hence, the choice of grid-oriented velocity components as dependent variables is preferable. On colocated grids Cartesian velocity components as primitive variables in the momentum equations are satisfactory.

In order to compute turbulent flows a turbulence model has to be chosen. Many investigators adopt isotropic eddy-viscosity formulations, of which a $k$-$\varepsilon$ model is the
most prominent. This model consists of two transport equations for the turbulent kinetic energy $k$ and its dissipation rate $\varepsilon$. Both equations contain a quantity $P_k$ called the production rate of turbulent energy. In the coordinate-invariant approach this quantity contains Christoffel symbols (in 2D eight different Christoffel symbols occur). Hence, it seems attractive to implement the $k-\varepsilon$ model with the Cartesian approach.

In the light of the above observations, the colocated/Cartesian approach for solving the Reynolds-averaged Navier-Stokes equations with the $k-\varepsilon$ model has become very popular and has been widely used over the last 15 years [Habib and Whitelaw, 1982], [Rhie and Chow, 1983], [Perić, 1985], [Chen and Patel, 1989], [Deng, 1989], [Piquet and Queutey, 1990], [Lien and Leschziner, 1991], [Cho and Fletcher, 1991], [Melaaen, 1991], [Coelho and Pereira, 1992], [Zhu and Rodi, 1992], [Xu et al., 1993], [Rolfes et al., 1993] and [Issa and Oliveira, 1994].

Within the colocated approach the pseudo-compressibility method can also be used. This has been done e.g. in [Kwak et al., 1986] and [V. Michelassi and F. Martelli, 1990]. The former used an algebraic eddy-viscosity model while the latter used the $k-\varepsilon$ model. The main disadvantage of this method is that for time-dependent flow problems it is difficult to obtain accurate mass conservation.

Although the colocated approach is very popular, an important disadvantage of this approach is that there are special measures required to obtain a stable discretization (for example the Rhie and Chow interpolation to avoid decoupling between pressure and velocity).

Staggered grid arrangement has important advantages for incompressible flow computations, because this arrangement avoids non-physical pressure oscillations. Some publications discretizing the Reynolds-averaged Navier-Stokes equations for staggered grids with Cartesian velocity components as primitive variables including the $k-\varepsilon$ model in general coordinates are [Rapley, 1985], [Braaten and Shyy, 1986], [Yung et al., 1989] and [Chen et al., 1990]. It should be recognized that this approach may give rise to unstable discretizations. Following Chen et al. (1990), the more common way to avoid this, is to choose the coordinate system such that the angle between the velocity components and the grid lines is not too large. However, for special domains it is very difficult to achieve this, especially in 3D.

Discretizations for staggered grids with grid-oriented velocity components as unknowns with the $k-\varepsilon$ model are presented in [Pope, 1978], [Rastogi, 1984], [Demirdzic et al., 1987], [Stern et al., 1988], [Majumdar and Rodi, 1989] and [Koshizuka and Oka, 1991]. Stern et al. (1988) have formulated the governing equations in vector notation which has the advantage that Christoffel symbols do not occur explicitly. Rastogi (1984) and Majumdar et al. (1989) use contravariant velocity components to solve the Navier-Stokes equations, while Pope (1978), Demirdzic et al. (1987) and Koshizuka et al. (1991) employ contravariant physical velocity components as unknowns.

In [Mynett et al., 1991], [Wesseling et al., 1992] and [Segal et al., 1992] a coordinate-invariant discretization of the incompressible Navier-Stokes equations on a general staggered grid has been developed. Standard tensor notation has been used. As a consequence, the formulation contains Christoffel symbols. In spite of that, discretization accuracy can be maintained if certain rules concerning the
approximation of the geometric quantities are followed and the contravariant flux components are chosen as primitive variables, and the grid is not too non-smooth. Good results are obtained for 2D laminar flows on fairly smooth grids.

In this paper a finite volume method for turbulent flow predictions on staggered grids in complex geometries employing the contravariant flux components as dependent variables is presented. Turbulence is modeled by the standard high-Re k-ε model with wall functions. The discretization incorporates central differences for the momentum equations and hybrid central/upwind differences for the turbulence equations. Accurate approximation of production of turbulent energy (defined in (2.13)) is obtained. The method is applied to some complex 2D turbulent recirculating flows, namely driven cavity flow, flow over a backward facing step, constricted tube flow and flow through an U-bend.

2 Equations governing incompressible turbulent flows in general coordinates

In order to formulate the equations governing turbulent flow in general coordinates, tensor notation will be used; for an introduction see [Aris, 1962].

The physical domain with curved boundaries Ω is transformed to a rectangle G with the mapping

\[ T : x = x(\xi), \quad x \in \Omega, \xi \in G \] (2.1)

Here, \( x \) are Cartesian coordinates and \( \xi \) are boundary conforming curvilinear coordinates. The mapping is assumed to be regular, i.e. the Jacobian of the transformation does not vanish. Covariant base vectors \( a_\alpha(\xi) \), contravariant base vectors \( a^{\alpha}(\xi) \), and the covariant and contravariant metric tensors \( g_{\alpha\beta} \) and \( g^{\alpha\beta} \) are defined as

\[ a_\alpha(\xi) = \frac{\partial x}{\partial \xi^\alpha}, \quad a^{\alpha}(\xi) = \frac{\partial \xi^\alpha}{\partial x}, \quad g_{\alpha\beta} = a_\alpha(\xi) \cdot a_\beta(\xi), \quad g^{\alpha\beta} = a^{\alpha}(\xi) \cdot a^{\beta}(\xi) \] (2.2)

The square root of the determinant of the covariant metric tensor, denoted by \( \sqrt{g} \), equals the Jacobian of the transformation. The following formulae for covariant derivatives of tensors of rank zero, one and two, respectively, are used in this paper:

\[ \phi_\alpha = \frac{\partial \phi}{\partial \xi^\alpha} \] (2.3)

\[ U_\gamma^\alpha = \frac{\partial U^\alpha}{\partial \xi^\gamma} + \{ \alpha \} U^\gamma \] (2.4)

\[ U^\alpha_\gamma = \frac{1}{\sqrt{g}} \frac{\partial \sqrt{g} U^\alpha}{\partial \xi^\gamma} \] (2.5)

\[ T^{\alpha\beta}_\gamma = \frac{1}{\sqrt{g}} \frac{\partial \sqrt{g} T^{\alpha\beta}}{\partial \xi^\gamma} + \{ \alpha \} T^{\gamma\beta} \] (2.6)

where

\[ \{ \alpha \}_{\gamma\beta} = a^{(\alpha)} \cdot \frac{\partial a_(\beta)}{\partial \xi^\gamma} = \frac{\partial \xi^\alpha}{\partial x^\gamma} \cdot \frac{\partial^2 x^\delta}{\partial x^\gamma \partial \xi^\beta} = \{ \alpha \}_{\beta\gamma} \] (2.7)
The Christoffel symbol of the second kind. The summation convention holds for Greek indices.

Turbulent flow is governed by the continuity equation and the Reynolds-averaged Navier-Stokes equations. The Reynolds stresses are related to the mean rates of strain through the isotropic eddy-viscosity \( \nu_t \), which is calculated by the standard high-Re k-\( \varepsilon \) model [Lauder and Spalding, 1974]. The tensor formulation of these equations is given by

\[
\frac{\partial U^\alpha}{\partial t} + (U^\alpha U^\beta)_{,\beta} + (g^{\alpha \beta} p)_{,\beta} - ((\nu + \nu_t)(g^{\alpha \gamma} U^\beta_{,\gamma} + g^{\beta \gamma} U^\alpha_{,\gamma}))_{,\beta} = F^\alpha
\]

\[
\nu_t = c_\mu \frac{k^2}{\varepsilon}
\]

\[
\frac{\partial k}{\partial t} + (U^\alpha k)_{,\alpha} - \left( \frac{\nu_t}{\sigma_k} g^{\alpha \beta} k_{,\beta} \right)_{,\alpha} = P_k - \varepsilon
\]

\[
\frac{\partial \varepsilon}{\partial t} + (U^\alpha \varepsilon)_{,\alpha} - \left( \frac{\nu_t}{\sigma_\varepsilon} g^{\alpha \beta} \varepsilon_{,\beta} \right)_{,\alpha} = \frac{\varepsilon}{k} (c_{\varepsilon 1} P_k - c_{\varepsilon 2} \varepsilon)
\]

where \( U^\alpha \) is the contravariant mean velocity component, \( p \) is the pressure, \( \nu \) is the kinematic viscosity, \( F^\alpha \) is an external force per unit volume, \( k \) is the turbulent kinetic energy, \( \varepsilon \) is the turbulent energy dissipation rate and \( P_k \) is the production of turbulent energy, given by

\[
P_k = \nu_t g_{\alpha \beta} (g^{\alpha \gamma} U^\beta_{,\gamma} + g^{\beta \gamma} U^\alpha_{,\gamma}) U^\delta_{,\delta}
\]

Finally, \( c_\mu, c_{\varepsilon 1}, c_{\varepsilon 2}, \sigma_k \) and \( \sigma_\varepsilon \) are dimensionless constants which, respectively, are taken to be 0.09, 1.44, 1.92, 1.0 and 1.3. The values of these constants are recommended in [Lauder and Spalding, 1974].

Specification of the boundary conditions is straightforward, except in near-wall regions where wall functions are adopted to avoid integration through the viscous sublayer and to obtain log-layer solutions [Lauder and Spalding, 1974]. These wall functions are given by:

\[
\tau_w = \begin{cases} 
\nu \frac{U^\prime \cdot t}{Y^+}, & Y^+ < 11.3 \\
\frac{c_\mu^{1/4} \sqrt{k \ln(EY^+)}}{\ln(EY^+)} u \cdot t, & Y^+ \geq 11.3
\end{cases}
\]

where

\[
Y^+ = \frac{c_\mu^{1/4} \sqrt{k}}{\nu}
\]

and

\[
P_k = \tau_w \frac{u \cdot t}{Y^+}
\]

\[
\varepsilon = \begin{cases} 
c_\mu^{3/4} k^{3/2} Y^+, & Y^+ < 11.3 \\
c_\mu^{3/4} k^{3/2} \ln(EY^+)/\sigma Y, & Y^+ \geq 11.3
\end{cases}
\]
Here, $\tau_w$ is the wall shear stress, $Y$ is the distance perpendicular to the wall, $\kappa$ is the Von Kármán constant ($\approx 0.4$) and $E$ is a roughness parameter, approximately equal to 9.0 for a smooth wall. The quantity $u \cdot t$ denotes the tangential velocity along the wall and is given by

$$u \cdot t = \frac{g_{10} U^\alpha}{\sqrt{g_{11}}}, \quad t = 1 \text{ or } 2$$

(2.18)

Finally, the flux of turbulent energy through the wall is set to zero and the value of $\varepsilon$ at the first grid point away from the wall is determined from

$$\varepsilon = \frac{\epsilon^3}{\kappa Y}$$

(2.19)

3 Discretization of the governing equations

3.1 Invariant finite volume discretization

In discretizing the governing equations (2.8)-(2.12), the following requirements should be met for accuracy reasons:

1. The geometric identity $\int \epsilon^{(a)} d\Gamma_\alpha = 0$ should be satisfied after discretization.

2. When representing a constant velocity field $u$ in terms of its contravariant components $U^\alpha$, and recomputing $u$ from $U^\alpha$, the original vector field $u$ should be recovered exactly.

The first requirement imposes rules on the approximation of geometric quantities. The second requirement can be met if the flux components $V^\alpha = \sqrt{g} U^\alpha$ are chosen as unknowns. More details can be found in [Mynett et al., 1991], [Wesseling et al., 1992] and [Segal et al., 1992].

A finite volume method is used to discretize the governing equations on a staggered grid in the computational rectangle $G$. In $G$ we choose a uniform grid, choosing the mapping $x = x(\xi)$ such that the mesh-size $\delta x^\alpha = 1$. Figure 3.1 shows the locations of the points for the velocities $U^\alpha$ and pressure $p$ in the grid. The turbulence

![Figure 3.1: Arrangement of the unknowns for a staggered grid](image)

quantities $k$ and $\varepsilon$ are evaluated at pressure points. For brevity the momentum and $k-\varepsilon$ equations are written in the following form:
\[ T_{\alpha\beta} = F^\alpha - \frac{\partial U^\alpha}{\partial t} \]  
(3.1)

where

\[ T^{\alpha\beta} = U^\alpha U^\beta + g^{\alpha\beta} \rho - (\nu + \nu_i)(\gamma^\alpha U^\beta_{\gamma
\gamma}) \]  
(3.2)

and

\[ Q^\alpha = S_\phi - \frac{\partial \phi}{\partial t} \]  
(3.3)

where

\[ Q^\alpha = U^\alpha \phi - \frac{\nu_i}{\sigma_\phi} g^{\alpha\beta} \phi_{\beta} \]  
(3.4)

Here, \( \phi = k \) or \( \varepsilon \) and \( S_\phi \) represents the nonlinear source terms. For convenience we introduce the local cell coordinates given by Figure 3.2.

\[
\begin{array}{c|c|c|c|c|c|c|c|c|c}
 & 2 & + & + & + \\
\hline
1 & 0 & + & 0 & 0 & 0 & + \\
\hline
0 & 0 & + & + & + & + \\
\hline
-1 & -2 & + & + & + \\
\hline
-2 & -1 & 0 & 0 & 1 & 2
\end{array}
\]

Figure 3.2: Local cell coordinates

Discretization of the continuity equation is obtained by integration over a finite volume \( \Omega \) with center \((0,0)\), using (2.5):

\[
\int_\Omega U^\alpha d\Omega = \int_\Omega \frac{\partial \sqrt{g} U^\alpha}{\partial \xi^\alpha} d\xi^1 d\xi^2 \approx V^1_{(1,0)} + V^2_{(0,1)} = 0
\]  
(3.5)

The momentum equation (3.1) is discretized in space as follows, taking for example a \( U^1 \)-cell with center at \((1,0)\), using (2.6):

\[
\int_\Omega T^{1\beta}_{\gamma\delta} d\Omega = \int_\Omega \frac{\partial \sqrt{g} T^{1\beta}_{\gamma\delta}}{\partial \xi^\gamma} d\xi^1 d\xi^2 + \int_\Omega \{ \gamma_{\beta} \} T^{\gamma\delta} \sqrt{g} d\xi^1 d\xi^2 \\
\approx \sqrt{g} T^{11}_{(1,0)} + \sqrt{g} T^{12}_{(1,1)} + \sqrt{g} \{ \gamma_{\beta} \} T^{\gamma\delta}_{(1,0)}
\]  
(3.6)

Integration of the momentum equation with \( \alpha = 2 \) over a \( U^2 \)-cell with center \((0,1)\) is done similarly.

Using (2.5), the transport equation (3.3) is integrated over a pressure cell with center \((0,0)\) which yields

\[
\int_\Omega Q^\alpha_{d\Omega} \approx \sqrt{g} Q^1_{(1,0)} + \sqrt{g} Q^2_{(0,1)}
\]  
(3.7)
The right-hand sides of (3.1) and (3.3) are integrated using midpoint rules:

\[ \int_{\Omega} (F^{\alpha} - \frac{\partial U^{\alpha}}{\partial t}) d\Omega \approx \sqrt{g}_{(k,l)}(F^{\alpha} - \frac{\partial U^{\alpha}}{\partial t})_{(k,l)} \]  

(3.8)


and

\[ \int_{\Omega} (S_{\phi} - \frac{\partial \phi}{\partial t}) d\Omega \approx \sqrt{g}_{(0,0)}(S_{\phi} - \frac{\partial \phi}{\partial t})_{(0,0)} \]  

(3.9)

with \((k,l) = (1,0)\) if \(\alpha = 1\) or \((k,l) = (0,1)\) if \(\alpha = 2\).

The discretization is completed by substituting (3.2) in (3.6) and (3.4) in (3.7). Furthermore, \(U^{\alpha}\) is replaced by \(V^{\alpha}/\sqrt{g}\). The cell-face fluxes containing cell-face values (convection) and derivatives (diffusion) have to be approximated. Central differences will be employed, except for the convection of turbulence equations in which a hybrid central/upwind scheme [Spalding, 1972] will be adopted. The reason for this is as follows. The \(k\)-\(\varepsilon\) equations are highly nonlinear and coupled. As a consequence, the computational task is complicated, because numerical experiments have shown that the convergence of iterative methods, to solve this coupled nonlinear problem, is adversely affected by negative values of \(k\) and \(\varepsilon\) which occur when convection is approximated by a non-positive (e.g. central) scheme. Using the hybrid scheme, the approximation of the face value \(\phi\) at point \((1,0)\), for example, is given by

\[ \phi_{(1,0)} = \{ 1 - s(Pe_{(1,0)}^{1}) \} \phi_{C} + s(Pe_{(1,0)}^{1}) \phi_{U} \]  

(3.10)

where \(\phi_{C}\) and \(\phi_{U}\) are given by

\[ \phi_{C} = \frac{1}{2}(\phi_{(0,0)} + \phi_{(2,0)}) \]  

(3.11)

\[ \phi_{U} = \frac{1}{2}\{ 1 + \text{sign}(V_{(1,0)}^{1}) \} \phi_{(0,0)} + \frac{1}{2}\{ 1 - \text{sign}(V_{(1,0)}^{1}) \} \phi_{(2,0)} \]  

(3.12)

Furthermore, the mesh-Péclet number \(Pe_{(k,l)}^{\alpha}\) is defined by

\[ Pe_{(k,l)}^{\alpha} = \frac{\sigma_{\phi}V_{(k,l)}^{\alpha}}{2\nu_{t}\sqrt{g}_{(k,l)}g_{(k,l)}}, \quad \text{no summation over} \ \alpha \]  

(3.13)

If \(|Pe_{(k,l)}^{\alpha}| > 1\), a first order upwind scheme is used, otherwise a switch to central differences is applied. The switching function \(s(Pe)\) is defined as

\[ s(Pe) = \begin{cases} 
0, & |Pe| \leq 1.0 \\
1, & |Pe| > 1.0 
\end{cases} \]  

(3.14)

Non-orthogonal coordinates introduce mixed derivatives in diffusion terms, which make the scheme non-positive even when upwind discretization is applied. Many authors (e.g. Rhie and Chow (1983), Demirdzic et al. (1987) and Melaen (1991)) treat these derivatives in an explicit manner, i.e. calculating them from values obtained in the previous iteration, but we found that our iterative solver allows to treat the mixed derivatives implicitly. Nonetheless, in some circumstances (highly non-orthogonal grid or large non-alignment of grid lines and streamlines) this scheme
may produce numerical instability. In such cases certain precautions will have to be taken; for a discussion see [Zijlema, 1983].

The discretization of the production of turbulent energy (2.13) is carried out by substituting (2.4) in (2.13) and with central differencing. Again, $U^\alpha$ is replaced by $V^\alpha/\sqrt{g}$. In spite of the presence of Christoffel symbols numerical experiments have shown that this discretization gives good results on reasonable smooth grids.

The discretization of the $V^1$-momentum equation results in the 19-point stencil presented in Figure 3.3. The $V^2$ stencil is obtained by rotation over 90°. The total number of variables linked together in the transport equation is 9.

Implementation of boundary conditions for the momentum and transport equations is discussed in [Segal et al., 1992] and [Zijlema, 1993]. Also discussion about the implementation of periodic as well as antiperiodic boundary conditions in our code can be found in [Segal et al., 1993].

### 3.2 Time discretization and solution method

The spatial discretization yields systems of ordinary differential equations of the following form:

$$DV = O$$

$$\frac{dV}{dt} + N(V) + GP = F$$

$$\frac{d\phi}{dt} + T(V)\phi = B + S_\phi$$

where $V$, $P$ and $\phi$ denote algebraic vectors containing the velocity, pressure and scalar unknowns, respectively. Furthermore, $D$ and $G$ are the discretized divergence and gradient operators, $N$ and $T$ represent the discretization of convection and diffusion terms, $F$ contains the discretized source term and boundary values, $B$ represents a right-hand side term arising from the boundary conditions and $S_\phi$ contains the source term which is a function of $V$ and $\phi$. Time discretization takes place with the implicit Euler method. Linearization of nonlinear terms is carried out with the standard Newton method:

$$N(V^{n+1}) \approx N(V^n) + \frac{\partial N}{\partial V}(V^{n+1} - V^n)$$
\[ S_{\phi}^{n+1} \approx S_{\phi}^{n} + \frac{\partial S_{\phi}^{n}}{\partial \phi} (\phi^{n+1} - \phi^{n}) \]  

(3.19)

For both the \( k \) and \( \epsilon \) equations the inequalities \( S_{\phi}^{n} \geq 0 \) and \( \frac{\partial S_{\phi}^{n}}{\partial \phi} < 0 \) hold, which preserves positivity of the solutions. For a derivation see [Zijlma, 1993]. The resulting systems of linear equations are solved by a Krylov subspace method of GMRES type [Saad and Schultz, 1986] with preconditioning. This method is very suitable for non-symmetric matrices, has a relatively good rate of convergence, and is vectorizable to a satisfactory degree. For more details we refer to [Vuik, 1993]. Most authors, like Rapley (1985), Demirdzic et al. (1987), Majumdar and Rodi (1989), Melaaen (1991) and Coelho and Pereira (1992) adopted more slowly convergent iterative methods for solving the momentum and turbulence equations, of which the line Gauss-Seidel method and the strongly implicit method of Stone (1968) are the most prominent. Both iterative methods are only partly vectorizable.

The overall solution algorithm can be summarized as follows: first, the continuity equation and the momentum equations are solved. To ensure a divergence-free velocity field a second order pressure-correction method as described by Van Kan (1986) is used. Details can be found e.g. in [Segal et al., 1992]. Finally, the equations for \( k \) and \( \epsilon \) are solved in a decoupled way. Time-stepping is repeated until a stationary solution is obtained.

4 Results

In this section applications of the present method to 2D turbulent flows, both geometrically and physically complex, are described. Only flow situations that are well documented in the literature will be investigated. These include flows in a driven cavity, over a backward facing step, through a constricted tube and through an U-bend.

4.1 Driven cavity flow

The calculation of a driven cavity flow shown in Figure 4.1 is compared with the experiments of Baines and Knapp (1965). Furthermore, experimental data for the velocity along the vertical plane in the middle of the cavity are taken from [Rodi, 1984].
At walls, the wall functions are applied. At the moving wall the experimental surface shear stress $\tau_s$ is given. In order to reduce the turbulence length scale near the moving wall, the following boundary conditions for $k$ and $\varepsilon$ are used:

$$k = \frac{u_{rs}^2}{\sqrt{\sigma_\mu}}, \quad \varepsilon = \frac{u_{rs}^3}{\kappa y} \quad (4.1)$$

where $u_{rs} \equiv \sqrt{\tau_s / \rho}$ is the surface friction velocity and $y$ is the distance perpendicular to the moving wall. The Reynolds number based on the surface friction velocity and the depth $h$ of the cavity is approximately 6,000.

Figure 4.2 shows the 80x50 mesh in which the grid cells are highly stretched in order to resolve the steep velocity gradients near the walls.

The predicted normalized velocity profile along the vertical plane in the middle of the cavity is shown and compared with the experimental data in Figure 4.3. It is seen that the calculation is in very good agreement with the measurements. Hence, it may be concluded that this type of flows are well described by the standard $k-\varepsilon$
model. Furthermore, our general coordinate discretization is accurate on the non-uniform grid of Figure 4.2. Convergence of the preconditioned GMRES method was not adversely affected by the high (up to 10) mesh aspect ratios which occur.

4.2 Turbulent flow over a backward facing step

Turbulent flow over a backward facing step is a widely used benchmark problem to evaluate the performance of turbulence models in the prediction of separated flows. It is well-known that the standard \(k-\varepsilon\) model with wall functions underpredicts the reattachment length in the backward facing step by an amount in the order of 20-25%. This has been widely discussed since the 1980-81 AFOSR-HTTM Stanford Conference on Complex Turbulent Flows, see e.g. [Demirdzic, 1982], [Nallasamy, 1987] and [Thangam and Speziale, 1991].

In this section we present some results. For the computations the flow configuration of [Kim et al., 1980] (which was one of the test cases in the aforementioned conference) has been selected. In this case the ratio of step height to outlet channel height is 1:3 and the Reynolds number based on the step height and \(U_0\) is 44,000. The domain is sketched in Figure 4.4.

![Figure 4.4: Geometry of the backward facing step flow](image)

Inlet profiles for velocities and turbulence quantities are specified at 4 step heights upstream of the step corner. These profiles are obtained from a preliminary calculation of a fully developed turbulent flow in a straight channel for the given Reynolds number. Wall functions are used at the upper and lower walls and outflow conditions (i.e. normal stress and tangential velocity are zero and zero normal gradient conditions for turbulence quantities) are applied 30 step heights downstream of the step.

Figure 4.5 shows a part of the non-orthogonal 40x94 mesh employed. Overall, the mesh is smooth except near the step. In fact, the backward facing step problem is suited for a multiblock approach in which an orthogonal coordinate system can be employed. However, our purpose is to test our general coordinates discretization. Inaccuracies will occur with the present grid, which will be shown later. An initial calculation with a 20x45 grid indicates little change in the reattachment length compared to the result from the 40x94 grid. Hence, further grid refinement seems unnecessary.

In Figure 4.6 the computed streamlines are shown. The calculated reattachment
Figure 4.5: A part of the 40x94 grid for the backward facing step flow

Figure 4.6: Predicted streamlines for flow over the backward facing step

length is $x_r/H = 5.3$, which gives 24\% underprediction of the experimental reattachment length of $x_r/H \approx 7$. This result resembles the results found earlier at the aforementioned conference very well. Plots of calculated isobars, turbulence intensity (defined as $\sqrt{\epsilon}/U_0$) and turbulence length scale (defined as $c_\mu k^{3/2}/\epsilon H$) are shown in Figure 4.7.

Figure 4.7: Predicted features of turbulent flow over the backward facing step: isobars, turbulence intensity and length scales, respectively

Profiles of the normalized streamwise velocity are given in Figures 4.8, 4.9 and 4.10 and are compared with the experimental data. As can be seen, there is reasonably good agreement between the computations and the experimental results and
Figure 4.8: Streamwise velocity profile at $z/H = 5.33$

Figure 4.9: Streamwise velocity profile at $z/H = 8.00$

discrepancies are mainly due to the underprediction of the reattachment length. Furthermore, profiles of the normalized Reynolds shear stress are presented in Figures 4.11, 4.12 and 4.13. Deficiencies occur near the recirculation zone, where the shear stress peaks are underpredicted, whereas in the recovery region far downstream of the reattachment the shear stress is reasonably well predicted. The results and their deficiencies are typical of $k$-$\varepsilon$ modeling. The results obtained with the present method are in very good agreement with the computations of Demirdzic (1982).
Figure 4.10: Streamwise velocity profile at $z/H = 13.33$

Figure 4.11: Shear stress profile at $z/H = 7.66$
Figure 4.12: Shear stress profile at $x/H = 10.30$

Figure 4.13: Shear stress profile at $x/H = 15.67$
From Figure 4.14 it is seen that small wiggles occur in the streamlines due to the non-smoothness of the grid near the step. Although undesirable these wiggles seem to have a very small influence on the solutions, and thus are not too dramatic. As mentioned before, a multiblock approach combined with Cartesian grids is a natural approach to avoid this problem. This can be demonstrated with a computation of laminar flow over a backward facing step with $Re = 300$. It is concluded from Figures 4.15 and 4.16 that better results are obtained with the multiblock method.

Figure 4.15: Inaccurate streamlines for $Re = 300$ with a non-smooth grid

Figure 4.16: Accurate streamlines for $Re = 300$ with an orthogonal grid computed with multiblock

For more details on the multiblock method, we refer to [Brakkee and Wilders, 1994]. A way to improve the single block results would be by further smoothing of the discontinuity in the grid.
4.3 Turbulent flow through a constricted tube

The method is applied to an even more challenging turbulent flow case: turbulent flow through a tube with a sinusoidal constriction, as shown in Figure 4.17. This flow has been studied experimentally by Deshpande and Giddens (1980). Earlier numerical studies related to this flow type have been carried out by Rastogi (1984) and Melaan (1991). The former used general orthogonal coordinates with a staggered grid arrangement, while the latter employed general non-orthogonal coordinates on a collocated grid.

Although the flow is axisymmetric, it was found that there is not much difference between the flow through a planar constricted duct and through a circular constricted pipe. Therefore, we consider the calculation of the flow in 2D plane constriction.

The height of the duct is 50.8 mm and the Reynolds number based on that height and the average inlet velocity is 15,000. The height and the base length of the constriction are \( \frac{1}{3}R_0 \) and \( 4R_0 \), respectively, where \( R_0 \) is the half height of the duct. Due to symmetry, only the lower half of the domain needs to be considered. The inlet profiles for the velocity and turbulence quantities were specified at the plane \( x = -4R_0 \). Following Deshpande et al. (1980), at the inlet a fully-developed power-law profile \((n \approx 6.4)\) is assumed for the streamwise velocity:

\[
 u_{in} = u_0(1 + \frac{y}{R_0})^{1/6.4} \tag{4.2}
\]

where \( u_0 \) is the centerline velocity. For the turbulence quantities \( k \) and \( \varepsilon \), the following inlet profiles are assumed:

\[
 k_{in} = 1.5I_T u_{in}^2, \quad \varepsilon_{in} = \frac{c_{\mu}}{l} \frac{3/2}{3/2} \tag{4.3}
\]

Here, \( I_T \) is the turbulence intensity, taken to be 3% and \( l \) is the mixing length given by:

\[
 l = \min(\kappa y, 0.1R_0) \tag{4.4}
\]

In addition, wall functions, symmetry and outstream conditions are imposed in the usual way.

Grid dependence tests have been performed with four grids, consisting of 50x20, 75x30, 100x60 and 150x100 cells, respectively. Figure 4.18 shows the 50x20 grid. The test focussed primarily on the separation and reattachment lengths. The computed separation and reattachment lengths are presented, together with the measurements.
Figure 4.18: A typical grid for the constricted tube flow (50x20 cells)

<table>
<thead>
<tr>
<th></th>
<th>Separation $x_s/R_0$</th>
<th>Reattachment $x_r/R_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>0.5</td>
<td>4.0</td>
</tr>
<tr>
<td>Rastogi, 41 x 21</td>
<td>1.2</td>
<td>2.4</td>
</tr>
<tr>
<td>Melaaen, 52 x 22</td>
<td>0.89</td>
<td>3.02</td>
</tr>
<tr>
<td>Present, 50 x 20</td>
<td>1.06</td>
<td>3.07</td>
</tr>
<tr>
<td>Present, 75 x 30</td>
<td>0.93</td>
<td>3.10</td>
</tr>
<tr>
<td>Present, 100 x 60</td>
<td>0.64</td>
<td>3.36</td>
</tr>
<tr>
<td>Present, 150 x 100</td>
<td>0.56</td>
<td>3.50</td>
</tr>
</tbody>
</table>

Table 4.1: Variation of separation and reattachment lengths with different grids

and earlier calculations of Rastogi and Melaaen in Table 4.1. From this Table it is seen that calculated separation length is overpredicted, while the reattachment length is underpredicted. Furthermore, the agreement between the present results and the results of Melaaen corresponding to grid resolution of the same order can be said to be satisfactory. It should be noted that Rastogi used the hybrid scheme and Melaaen used the power law scheme of Patankar (1980) for solving both the momentum and turbulence equations. Moreover, Melaaen also employed a second-order upwind scheme [Peric, 1985] for solving the momentum equations while the power law scheme is still used for solving turbulence equations. In that case, on the 52x22 grid the calculated separation and reattachment lengths are $x_s/R_0 = 0.56$ and $x_r/R_0 = 4.07$, respectively, as has been reported in [Melaaen, 1991].

The predicted lengths of separation and reattachment decrease and increase, respectively, with grid refinement approaching the experimental lengths. Although the reattachment and separation lengths still show some variation between two finest grids, further mesh refinement was not performed because of prohibitive demands for computing time and to avoid that the first $y^+$ point is too close to the wall. Hence, all subsequent computations were done with a 150x100 grid. Figure 4.19 presents typical features of this flow, including streamlines and contours of isobars and turbulence quantities.

Figures 4.20, 4.21 and 4.22 show the streamwise velocity predictions at different stations. The calculation is seen to yield good agreement with the measurements, except for the station $x/R_0 = 2.0$, although the profiles shapes are the same. This
Figure 4.19: Typical flow pattern for contricted tube: streamlines, isobars, turbulence intensity and length scales, respectively

Figure 4.20: Streamwise velocity profile at $x/R_0 = 0.0$
Figure 4.21: Streamwise velocity profile at $z/R_0 = 2.0$

Figure 4.22: Streamwise velocity profile at $z/R_0 = 11.0$
is not surprising, because it is well-known that the $k$-$\varepsilon$ model does not scale well in and around the recirculation zone. Furthermore, both the present method and the methods of Rastogi and Melaan produce very similar velocity profiles.

Figure 4.23 shows a comparison of the predicted wall static pressure with the measurements. In the acceleration region, where a steep decline of the pressure occurs, the pressure is well predicted. In the deceleration part, however, the present method gives an excessive pressure recovery. The same conclusion is drawn by both Rastogi and Melaan.

Finally, the profile of the centerline turbulence intensity is compared with measurements in Figure 4.24. The comparison indicates that the predicted turbulence intensity increases too early, and the measured one increases after separation occurs.
whereas the computed one decreases there. Moreover, the peak of the predicted turbulence intensity is too low. Again, it is believed that the errors lie in the \( k-\varepsilon \) model. Although the agreement between the predicted and measured turbulence intensity is not satisfactory, there is a qualitative accordance with the computed results of Rastogi.

### 4.4 Turbulent flow through an U-bend

This section is concerned with the application of the present method to the prediction of flow through a sharp two-dimensional U-bend as shown in Figure 4.25. Such a flow occurs for example in cooling passages within gas-turbine blades. Earlier numerical studies were made by e.g. Choi et al. (1989) and Bo and Iacovides (1993). We compare our results with numerical results from Bo et al. and experimental data taken from [Bo and Iacovides, 1993].

The bend geometry considered here has a curvature ratio \( R_c/D = 0.65 \), which causes separation. Here \( D \) is the width of the duct. The Reynolds number based on the duct width and the centerline inflow velocity \( U_c \) is 100,000. The inflow boundary is located at three duct widths upstream of the bend, whereas the outflow is specified at eight duct widths downstream of the bend. At smooth walls, the wall functions are applied. The streamwise velocity at inflow is described according to the power-law profile with \( n = 7 \), whereas \( k \) and \( \varepsilon \) are obtained from the formula (4.3). Here, the turbulence intensity is taken to be 6\% (in accordance with the experiment) and \( l = \min(\kappa y, 0.5 D) \).

First, computations were performed to examine the grid dependence of the solutions. To this end, three grids of 50x30, 90x50 and 150x90 cells were used. Figures 4.26 gives the 50x30 grid. The grid is non-orthogonal. Test results are shown in Figure 4.27 and 4.28 for the streamwise velocity and the turbulence intensity at 1\( D \) downstream of the bend. It can be seen that unlike the turbulence intensity the velocity is already grid independent on the 90x50 grid. Further grid refinement was not pursued, because it is believed that it gives smaller differences than those provoked by the \( k-\varepsilon \) model. Hence, the calculations for the 150x90 grid are considered.

Contour plots of streamlines, isobars, turbulence intensity and length scales are shown in Figures 4.29, 4.30, 4.31 and 4.32. The streamlines show the expected
Figure 4.26: A typical grid, 50x30 cells

Figure 4.27: Grid refinement test for U-bend (velocity profiles)

Figure 4.28: Grid refinement test for U-bend (turbulence intensity profiles)
Figure 4.29: Streamlines for flow through U-bend

Figure 4.30: Isobars for flow through U-bend

Figure 4.31: Turbulence intensity $\sqrt{k/\nu}$ for flow through U-bend

behaviour and also indicate that the length of the recirculation zone is about one duct width. The isobars shows also the qualitatively correct behaviour: high pressure in the impingement region, low pressure in the recirculation region and pressure recovery after separation. Also expected is the build up of turbulence intensity from the walls, especially in the impingement region.

The results of the computations are compared with the predictions obtained by Bo et al. (1993) and the measurements taken from [Bo and Iacovides, 1993]. Bo et al. performed 3D calculations on orthogonal grids. Present results will be compared with their results at the symmetry plane. They used the standard $k-\epsilon$ model, whereas a $k-l$ model is used in the near-wall regions instead of wall functions. Furthermore, they employed two convection schemes: the hybrid central/upwind scheme and a blended scheme, called LODA [Zhou and Leschziner, 1988], in which the convective transport is approximated by a combination of the well-known QUICK and first
order upwind schemes. Blending of these two schemes is determined by a so-called blending factor which is generally unknown. Hence, an iterative procedure is needed in order to determine the blending factor. Another disadvantage of this blending technique is that several measures are introduced to improve numerical stability and to prevent negative values of \( k \) and \( \varepsilon \) [Bo et al., 1993]. Therefore, one may expect that this blending technique is very costly.

In Figures 4.33, 4.34, 4.35 and 4.36 the streamwise velocity and the turbulence intensity profiles are presented at 1\( D \) and 3\( D \) downstream of the bend. "LODA" means that the LODA scheme is employed on both momentum and turbulence equations on a 133x67x35 grid, whereas "HYBRID" denotes the application of the hybrid scheme on both momentum and turbulence equations on a 177x77x35 grid. With respect to our computations, the following conclusions may be drawn. At 1\( D \) downstream of the bend, the velocity is reasonably well predicted, although the recirculation length is underpredicted. There is no qualitative accordance between the prediction and measurements of the streamwise velocity at 3\( D \) downstream of the bend. Furthermore, at 1\( D \) downstream of the bend, the turbulence intensity near the inner side of the duct is predicted very well, while the turbulence level near the outer wall is overestimated. Finally, at 3\( D \) downstream of the bend, the
turbulence intensity is somewhat lower than the measured one.

Overall, the standard $k$-$\varepsilon$ model does not produce accurate solutions, because this model is not capable of predicting effects of curvature on the turbulence. But there is qualitative agreement between computation and experiment, and satisfactory agreement between the HYBRID scheme of Bo et al. and the present computations.

5 Conclusions

An invariant finite volume discretization in general coordinates of the Reynolds-averaged Navier-Stokes equations with the standard $k$-$\varepsilon$ model on staggered grids has been presented. Central differencing is used for convection of momentum, whereas
the hybrid central/upwind scheme is employed for convection of \( k \) and \( \varepsilon \), to prevent negative values of \( k \) and \( \varepsilon \). The diffusion terms are approximated with central differences. The mixed derivatives in these terms are treated implicitly. The present method was applied to several 2D turbulent flows in arbitrarily shaped domains. The agreement between the results of the computations and the experimental data or the predictions obtained by other methods was found to be satisfactory, within the limitations of the standard \( k-\varepsilon \) model employed. The calculation of the backward facing step flow has demonstrated that the present discretization is not accurate enough on very non-smooth grids. It turns out that a multiblock method is more suitable to avoid such inaccuracies.
References


