NUMERICAL SCHEMES FOR STABILISATION, A CASE STUDY OF THE FLOW AROUND A CUBIC BUILDING

I G B Wijaya Kusuma

ABSTRACT

This study compares the performance of several numerical schemes to stabilise the iteration procedure while using a commercial package CFX, based on finite volume method. We propose the use of low value of $k$ and $ε$ combined with multi-blocks grids in this study since it proves to reduce the over-estimation of the production term of turbulent kinetic energy while using a standard $k-ε$ turbulence model. The use of the Van Leer scheme is suggested when applied coarse grids, since it will control the energy transport by diffusion. In general flow problems with high Reynolds number, this scheme also minimizes the truncation and round-off errors. In conclusion, a combination of low $k-ε$ values and Van Leer schemes reduces the over-estimation of $k-ε$ turbulence models and the number of iterations, which indicating the reduction of CPU time. Therefore, we note that using smaller $k-ε$ values does not only improve the turbulent viscosity correction but also improves the accuracy of the results, therefore, it can be applied in order to improve the performance of commercial package.

Keywords: Numerical schemes. Turbulence models, Building

INTRODUCTION

The steady convection-diffusion equation can be derived from the transport equation as follows,

$$\text{div}(\rho \mathbf{u}) = \text{div}(\Gamma \text{grad} \phi) + S_\phi$$

(1)

This equation represents the flux balance in a control volume. The left-hand side gives the net convective flux and the right-hand side contains the net diffusive flux and the generation or destruction of the property $\phi$ within the control volume [Versteeg and Malalasekera, 1995].

The main problem in the discretization of the convective terms is the calculation of the value of transported property $\phi$ at control volume faces and its convective flux across the boundaries. The diffusion process affects the distribution of the property along its gradients in all directions, whereas convection spreads influence only in the flow direction. This crucial difference appears in an exacting upper limit to the

*Department of Mechanical Engineering, Faculty of Engineering, University of Udayana, Bali Jayastra, Bali, Bali 80361, Indonesia
grid size, that is dependent on the relative strength of convection and diffusion, for
stabilisation of convection-diffusion calculations with central difference schemes [Versteeg and Malalasekera, 1995].

The central difference scheme has been used to represent the diffusion terms which appear on the right-hand side of the above equation and it stems logical to try linear interpolation to compute the cell face values for the convective terms on the left-hand side of the equation [Versteeg and Malalasekera, 1995].

The central difference scheme was consistent expressions to evaluate convective and diffusive fluxes at the control volume faces. If the local cell Peclet number is greater than 2 (such as in this model), the conservative approximation of the central difference scheme leads to physically impossible solutions since the scheme does not recognize the direction of the flow or the strength of convection relative to diffusion. Therefore, this scheme is only stable and accurate when the local cell Peclet number is lower than 2, i.e., if the velocity is small (hence in diffusion-dominated low Reynolds number flows) and/or if the grid spacing is small. For high Peclet number, the central finite volume schemes may suffer instabilities, therefore, upwind schemes or artificial viscosities are often introduced for stabilization.

Parria [2] derived the Van Leer split-flux vectors for moving curvilinear co-
ordinate systems, and successfully applied it to a fixed wing. Ferrand and Aubert [3]
applied the Van Leer scheme for inviscid transonic flow, but also presented an
alternative hybrid scheme to restore the same problem. Their new approach is based on
Van Leer's flux vector splitting and is called the mixed Van Leer method since it
conserves the advantages of the central schemes at low Mach numbers and the
advantages of Van Leer schemes elsewhere. It seems that Van Leer methods are
specialized suitable for transonic flows, but this scheme can also be applied for other
problems.

Wilkes and Thomson [4] presented a higher-order upwind difference scheme which
was robust and could be used without adding excessive under-relaxation or an
especially good initial approximation. Kawamura et al. [5] presented a new higher-order
upwind scheme for incompressible Navier-Stokes flow. The stability of the first-order
upwind scheme is very good but has a strong dissipative effect to the molecular viscosity.
The second-order upwind scheme has worse stability properties since it caused
undesirable propagation of errors. They developed a new upwind scheme which had
third-order accuracy. They mention that it has a local dissipative effect, but the global
effect is much smaller than the second-order. Li and Radman [6] suggested a new
generalised formulation for four-point discretisation of non-uniform grids.
They mention that the central difference scheme, the QUICK scheme, and the second-
order upwind scheme fail in this formulation. A second-order hybrid scheme was also
presented for non-uniform grids. The unbounded behaviour of the generalised
formulation was examined. A flux-corrected transport algorithm was then applied to the
above four schemes on a uniform grid. They noted that incorporation of flux-corrected
transport into the high-order schemes greatly improves the solution accuracy.

Choi and Yoo [7] presented numerical approaches by using both finite elements and
finite volume methods for the Navier-Stokes equation. They proposed hybrid numerical
methods which give accurate results and are free from the checkerboard-type of pressure distribution. A dual adaptation scheme was developed for evaluation of the viscous terms.

In the present study, a comparison of several numerical schemes to stabilize the central difference scheme of finite volume methods is performed, while using a commercial package CFX. The QUICK difference scheme does not produce a good stability, therefore, it is not used for this comparison study. A higher upstream scheme was considered to establish its convergence and accuracy, since it has been successful to solve non-symmetric linear systems. The Van Leer scheme is also introduced since it reduces oscillations.

The upwind scheme has a major drawback when the flow is not aligned with the grid lines. It should be noted that in high Reynolds number flows, the false diffusion can be large and give physically incorrect results. Therefore, grid refinement must be considered to eliminate the false diffusion. The higher upstream difference scheme is based on the backward difference formula of the upwind scheme, but it involves more neighbouring points to reduce discretization errors.

PHYSICAL DESCRIPTION

The hybrid difference scheme is based on a combination of central and upwind difference schemes. The central difference scheme is employed for small Pecler number \( Pe < 2 \) and the upwind scheme is employed for large Pecler number \( Pe > 2 \). The hybrid difference scheme uses piecewise formulae based on the local Pecler number to evaluate the net flux through each control volume. This scheme produces physically realistic solutions and is highly stable when compared with the higher upwind scheme. The disadvantage is that the accuracy in terms of Taylor series truncation error is only first-order [Versteeg and Malalasekera, 1995].

Although Wilkes and Thomson [6] noted that in the hybrid scheme it was not necessary to use excessive under-relaxation nor an especially good initial approximation, in the present study the successive over-relaxation point iteration has been used. Therefore, all schemes have the same numerical procedure with a relaxation parameter for each cycle. The solution algorithm is a variant of the SIMPLE scheme of Patankar, in which velocities are obtained by solving the momentum conservation equations using the pressure field and then the pressure field is corrected by using the imbalances in the mass conservation equations. A cubic building has been used as an initial test for three-dimensional problems. The flow around the building is modelled by using the Navier-Stokes equations and the k-\( \varepsilon \) turbulence model. Rectangular and body-fitted grid systems have been used. A uniform grid system has been applied at the building surfaces, first and second blocks, but a non-uniform grid system has been used at the third block and outer region. The staggered grid thus used avoids the evaluation of boundary condition for pressures [Patankar, 1980] and also provides much more accurate predictions [Selvam and Paterson, 1991]. The momentum, 4 and \( \varepsilon \) turbulence model equations are solved by successive over relaxation point iteration with an under-relaxation parameter of 0.7 for each cycle.
The boundary conditions for the inlet velocity are fixed at the initial power-law velocity profile, where \( U_* / u_* = (z/\epsilon)^{0.25} \), \( \nu = \nu = 0 \). The turbulent intensity was evaluated to be 6.2% according to Davenport's terrain roughness classification number \( s \), for a suburban terrain [Wieringa, 1992]. The grid swed has \( 112 \times 92 \times 99 \) nodes for the \( x, y \) and \( z \) directions, respectively.

On the truncated walls and building surfaces, the wall treatment is a combination of logarithmic and no-slip boundary conditions. The no-slip boundary condition \( u = v = w = 0 \) is the appropriate condition for the velocity components at solid walls. The implementation of wall boundary conditions in turbulent flows starts with the evaluation of \( y^\prime \), where a near-wall flow is taken to be laminar if \( y^\prime < 11.65 \). If the value of \( y^\prime > 11.65 \), the first node (from solid walls) is considered to be in the logarithmic-law region of a turbulent boundary layer. The relationship can be described as follows

\[
\frac{u^+}{
u^+} = \frac{1}{k} \ln \left( \frac{u^+}{\nu^+} \right)
\]

and

\[
T^+ = C_T \left[ u^+ + \frac{f (\sigma_T^+)}{\sigma_T} \right]^{0.75} \left[ 1 + \frac{0.28}{(\sigma_T^+)^{0.75}} \right] + 10^{-6} \frac{(\sigma_T^+)^{0.25}}{\sigma_T}
\]

where \( C_T \) is von Karman's constant (0.41), \( E \) is an integration constant which for smooth walls with constant shear stress has a value of 1.793, \( \sigma_T^+ \) is the laminar Prandtl number (0.707), and \( \sigma_T^+ \) is the turbulent Prandtl number (\( \approx 0.9 \)). A rectangular grid is applied in the simulation, as shown in Figure 1.

To minimize undesirable re-coupling effects, the computational domain has to be sufficiently wide, high and long. The Reynolds number of the main flow, based on the velocity at the building height, is about 2.3x10^5. The wind flow around a cubic building model has been numerically simulated with the small \( k-\varepsilon \) model approach by using different grid systems. Using the commercial package CFX, the computed pressures are compared with experimental, wind tunnel and numerical results.

RESULTS AND DISCUSSION

Non-dimensional pressure distributions along building surface for the different numerical schemes can be seen in Figure 2. From the results, it is clear that the pressure distribution for all models is similar. We note that the higher upwind difference scheme is actually suitable for high Reynolds number flows and also appears to eliminate the false diffusion. It is suggested that by using multi-blocks grid, the spatial oscillations or "wigglers" in the flow variables are reduced, therefore the truncation error and false diffusion are eliminated.
The hybrid difference scheme produced slightly higher results at the windward corner, but at the up side, the pressure distribution is a bit lower than the others. It can be seen that both Van Leer and higher upwind difference schemes produce similar pressure distribution everywhere.

Non-dimensional turbulent kinetic energy as a comparison between turbulent kinetic energy and its reference velocity along building surface is shown in Figure 3. It can be seen that the hybrid scheme produces slightly greater turbulent kinetic energy at the windward side. At the leeward side, the hybrid difference scheme also produces slightly higher turbulent kinetic energy. It appears that the hybrid scheme produces a slightly greater transport of turbulent kinetic energy by diffusion than the Van Leer or upwind difference scheme.

The rate of energy dissipation for all models along building surface can be seen in Figure 4. Similarly to the turbulent kinetic energy, the hybrid scheme produces higher values at the windward and leeward sides of the building. This confirms that the hybrid scheme produces a slightly greater transport of turbulent kinetic and its dissipation rate by diffusion than the other schemes. Since this transport can be large for high Reynolds number flows and/or very coarse grids, a hybrid difference scheme should not be used for a complex geometry problem where coarser grids may occur.

The use of the Van Leer scheme is suggested here for the following reasons. First, the turbulent kinetic energy and its dissipation rate seem to be the mean value of higher upwind and hybrid difference schemes. This indicates that in coarse cases, energy transport by diffusion will not increase too much. Second, the Van Leer scheme was derived for transonic or low Mach number flows. However, in general problems with high Reynolds number, the truncation and round-off errors can be minimized.

The main objective of this study is to compare the performance of several numerical schemes to stabilise the iteration procedure. Relative residuals for all models can be seen in Figure 5. It can be noted that the relative residuals of the hybrid scheme are larger than the other methods. After 30 iterations, the hybrid scheme seems to converge to a value just below 0.02. Its relative residual then decreases and reaches the same value of the other methods after 70 iterations.

Relative residuals for convergence as presented in this section are around 0.01. Zhcu and Stathopoulos [10], who used a two layers method, have greater tolerance value of 0.02. The number of iterations in our simulation is also smaller, indicating that the present simulation also had a reduced CPU time. Therefore, we note again that using smaller values does not only improve the turbulent viscosity correction but also improves the accuracy of the results.

The velocity field showing the reattachment length at the leeward side of the building is presented in Figures 6, 7 and 8. The reattachment length is an important parameter to describe the effects of shear rate on the flow characteristic. Therefore, many researchers correlated their results using this parameter. The effects of shear stress dominate the flow in the re-circulation zone, which reduces the ability to transfer energy by convection because of the local heat transfer coefficient becomes very low at this zone. At the reattachment point, the convection heat transfer is the lowest, but it tends to increase thereafter.
From the velocity results seen in Figures 6, 7 and 8, it can be seen that the non-dimensional reattachment length of a cubic building for a Reynolds number of $2.3 \times 10^3$ is 1.67 at the leeward side, 0.32 at the windward side and 0.55 at top of building, where $H$ is the height of a cubic building. According to Franck [13] who used the Large Eddy Simulation (LES) method, these reattachment lengths are comparable. The criterion of the reattachment length at the leeward side, windward side and top of building is that the shear stress on the wall is zero. Comparison of the reattachment length to other published results can be seen in Table 1, and can be seen that this study is fairly good for using or computation method.

The hybrid difference scheme produces an interesting result in that the reattachment length is about 1.42 (see Figure 7), shorter than the other methods. This indicates that the hybrid difference scheme increases convective effects in the flow direction, especially at the e-circulation zone. In problems where fluid flow piles a significant role, convection effects must be considered. Since diffusion always occurs alongside convection in nature, their combined effects should be examined and the hybrid difference scheme should not be used for computation method using convection effects.

CONCLUSIONS

The use of the Van Leer scheme is suggested here for the following reasons. First, the turbulent kinetic energy and its dissipation rate seem to be the mean value of higher upwind and hybrid difference schemes. This indicates that in case of coarse grids, energy transport by diffusion will not increase too much. Second, the Van Leer scheme was derived for transonic or high subsonic flows. However, in general problems with high Reynolds number, the truncation and round-off errors can be minimized.

The main objective of this study is to compare the performance of several numerical schemes to stabilize the iteration procedure. From the relative residuals for all models it can be noted that the relative residuals of the hybrid scheme are larger than the other methods. The use of low $k$-$\varepsilon$ values reduces the over-estimation of $k$-$\varepsilon$ turbulence models. Relative residuals for convergence-as presented in this section- are lower than Zhou and Stathopoulos [10], who used a two-layers method. The number of iterations in our simulation is also smaller, indicating that the present simulation also had a reduced CPU time. Therefore, we note that using smaller $k$-$\varepsilon$ values does not only improve the turbulent viscosity correction but also improves the accuracy of the results.

ACKNOWLEDGEMENT

The author would like to gratefully appreciate to his professional colleagues who have contributed an important input of this paper.

REFERENCES

LIST OF FIGURES

Figure 1 (a) Grid arrangement in the present study outer region (two equations of turbulence model, k-ε)

Figure 1 (b) A two-layers method
Figure 2 Non-dimensional pressure distribution for different numerical schemes.

Figure 3 Turbulent kinetic energy for different numerical schemes.
Figure 4: Turbulent energy dissipation for different numerical schemes.

Figure 5: Relative residues for three schemes.
Figure 6 Velocity vector of higher upwind difference scheme model

Figure 7 Velocity vector of hybrid difference scheme model
Figure 8 Velocity vector of Van Leer difference scheme model

Figure 9 Reattachment lengths where shear stress is zero at the wall

Table 1 Comparison of different numerical calculations of the separation zones

<table>
<thead>
<tr>
<th></th>
<th>1/</th>
<th>2/</th>
<th>3/</th>
<th>4/</th>
<th>5/</th>
<th>6/</th>
</tr>
</thead>
<tbody>
<tr>
<td>x0/H</td>
<td>0.31</td>
<td>0.35</td>
<td>0.9</td>
<td>0.37</td>
<td>0.32</td>
<td>0.32</td>
</tr>
<tr>
<td>xH/H</td>
<td>0.9</td>
<td>0.75</td>
<td>0.78</td>
<td>0.55</td>
<td>0.57</td>
<td></td>
</tr>
<tr>
<td>xH/H</td>
<td>1.61</td>
<td>1.68</td>
<td>1.75</td>
<td>1.65</td>
<td>1.67</td>
<td>1.42</td>
</tr>
<tr>
<td>Re(10^7)</td>
<td>1.25</td>
<td>-</td>
<td>-</td>
<td>4.8</td>
<td>2.3</td>
<td>2.3</td>
</tr>
</tbody>
</table>

1/ Larsson, Martinuzzi & Tropea [12] (experimental)
2/ Frank [13] (experimental)
3/ Werne & Wengle [14] (numerical)
4/ Frank [11] (numerical, LES)
5/ Present study (numerical, standard k-ε)
6/ Present study (numerical, hybrid difference scheme)