INVESTIGATION OF AIRFLOW AND HEAT TRANSFER AROUND A CUBIC BUILDING

I G B Wijaya Kusuma*

ABSTRACT

Due to rapid urban development accompanied by the increase of artificial heat release in urban and rural areas, the investigation of heat transfer and airflow around a cluster of buildings was studied by using a standard k-ε turbulence model combined with multi-block grids and adopting low fluctuating velocities. A combination of low fluctuating velocity components and Van Leer schemes reduces the over-estimation of the production term of the turbulent kinetic energy in standard k-ε turbulence models and the number of iterations, reducing the CPU time. It is shown that the pressure distribution at the windward corner of a cubic building is minimised by using this model.

INTRODUCTION

The flow patterns around building were studied by many researchers both experimentally and numerically, more than two decades. In the last two decades, many numerical algorithms were introduced in order to validate the results, not only by introducing new turbulence models but also by developing improved schemes for numerical stability during computation. The validity of results for a cubic building, therefore, could be compared either with experimental or numerical results.

An experimental investigation of the flow around surface-mounted cubes in uniform, irrotational and sheared turbulent flow was described by Castro and Robins [Castro and Robins, 1977] and became a benchmark to validate numerical models. The practical importance of bluff-body aerodynamics has increased over the past few decades, with an enormous increase in the literature concerning laboratory simulations, full-scale measurements, and more recently, numerical calculations and theoretical predictions of a wide variety of bluff-body flows.

The flows around full-scale or model buildings were investigated much less thoroughly, despite the obvious practical implications regarding, say, effluent dispersion or wind loads on buildings. There were little experimental data available even for the simple case of surface-mounted bodies in a uniform upstream flow. A uniform upstream flow meant one in which the mean velocity is uniform and the turbulence intensity is very low, less than 0.5 %, except in the thin boundary layer which must exist on the surface even if the body is mounted on a false floor [Castro and Robins, 1977].

Wind tunnel simulation of the atmospheric boundary layer was generated and evaluated by some researchers. The variations of the pressure field on the cube were discussed in relation to the incident flow-field parameters. The velocity profile at the model position and the local surface roughness, largely determine the pressure on the windward side. This is specific to each building scheme and cannot be estimated from general tests of a building shape [Hunt, 1982].

Computational fluid dynamic methods based on Cartesian or cylindrical co-ordinate systems have certain limitations for irregular geometries. In practice, the boundary geometries can be complex and often irregular. The governing equations in body-fitted coordinates are much more complex than the Cartesian ones, especially the momentum equations. The structured grid method uses a curvilinear body-conforming mesh [Stathopoulus and Zhou, 1993]. This approach is very suitable for simulating high Reynolds number viscous flow fields, therefore most of the current CFD results for the Navier-Stokes equations employ structured grid systems.

The common strategy to use a body-fitted grid for a complex body is to employ multiple zone and multiple grid methods, such as the multi-blocks method. The multi-blocks method was also shown to be a very powerful approach for complex configurations. The procedure to generate the grid requires multi-blocks zoning of the flow field.

The main problem in the discretization of the convective terms is the calculation of the value of the transported flux at control volume faces and its convective flux across the boundaries. The diffusion process affects the distribution of a transported quantity 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 grid size, which is dependent on the relative strength of convection and

* Dr. Ir. I Gusti Bagus Wijaya Kusuma, Department of Mechanical Engineering, Faculty of Engeenering, University of Udayana

diffusion, for stabilisation of convection-diffusion calculations with central difference schemes [Versteeg, and Malalasekera, 1995].

Parpia [Parpia, 1998] derived the Van Leer split-flux vectors for moving curvilinear co-ordinate systems, and successfully applied it to a fixed wing. Ferrand and Aubert [Ferrand and Aubert, 1996] applied the Van Leer scheme for inviscid transonic flow, but also presented an alternative hybrid scheme to resolve the same problem. Their approach was based on Van Leer's flux vector splitting and was 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 special and suitable for transonic flows, but this scheme can also be applied for other problems.

Wilkes and Thomson [Wilkes and Thomson, 1983] presented a higher-order upwind difference scheme that was robust and could be used without adding excessive under-relaxation or an especially good initial approximation. Kawamura *et al* [Kawamura, Takami and Kuwahara, 1985] presented a new higher-order upwind scheme for incompressible Navier-Stokes flow. The stability of the firstorder upwind scheme is very good but has a strong diffusive 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 that has third-order accuracy. They mentioned that it had a local diffusive effect, but the global effect was much smaller than the second-order.

Li and Rudman [Li and Rudman, 1995] suggested a new generalised formulation for four-point discretization schemes of non-uniform grids. They mentioned that the central difference scheme, the QUICK scheme, and the second-order upwind scheme fall into 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 [Choi and Yoo, 1994] presented numerical approaches by using both finite element and finite volume methods for the Navier-Stokes equation. They proposed hybrid numerical methods that 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.

Most commercial packages follow the description above, since they have limitation during computational process. But unfortunately, some methods sometimes do not suitable to adopt directly and perform inappropriate results. Based on that reality, a comparison of several numerical schemes is carried out in the present study, to assess the stabilisation of finite volume methods during computational process. The main objective is to increase the compressibility of air in the separation regions, especially for high Reynolds numbers, in order to reduce the over-estimation of standard k-ε models.

PHYSICAL DESCRIPTION

A cubic building uses as an initial test for three-dimensional problems of a cluster of buildings. A typical grid arrangement for a cubic building can be seen in Figure 1. The flow around the building is modelled by using the Navier-Stokes equations and the k-ε turbulence model. Rectangular and body-fitted grid systems are used in this simulation.

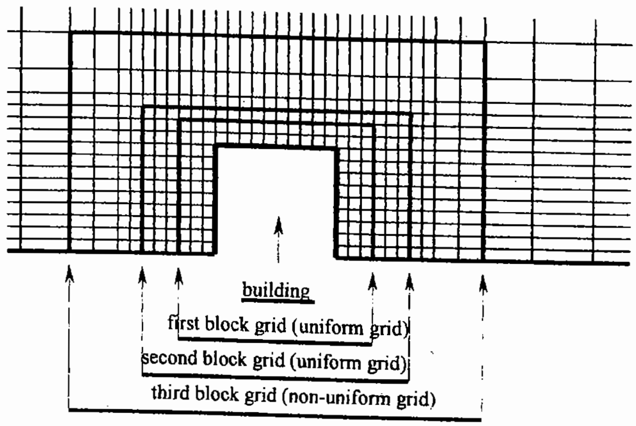

Figure 1. Grid arrangement in the present study

A uniform grid system applies at the building surfaces, first and second blocks, but a non-uniform grid system uses at the third block and outer region. The staggered grid thus used avoids the evaluation of boundary condition for pressures and also provides much more accurate predictions [Selvam and Paterson, 1991]. The wind flow around a cubic building has been numerically simulated with the standard k-ε turbulence model. The model is divided into four blocks, the wind flow in the third and fourth blocks is computed with the standard k-ε model, and the flow in the second and first block or near the building surfaces is also simulated with the standard k-ε model. The commercial package CFX was used, and the computed pressures compared with experimental, wind tunnel and numerical results.

Although Wilkes and Thomson [Wilkes and Thomson, 1983] 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 is used. Therefore, all schemes have the same numerical procedure with a relaxation parameter for each cycle. The momentum, k and ε turbulence model equations are solved by successive over-relaxation point iteration with an under-relaxation parameter of 0.7 for each cycle. The solution algorithm is a variant of the SIMPLE scheme of Patankar [Patankar and Spalding, 1972], 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.

The boundary conditions for the inlet velocity are fixed at the initial power-law velocity profile $u/u_g = (z/z_g)^{0.25}$, where u and u_g are mean velocities at height z and at a reference point at height z_g respectively, with $v = w = 0$. The turbulent intensity was evaluated to be 6.2% according to Davenport's terrain roughness classification number 4, for a suburban terrain [Wieringa, 1992]. The inlet flow profile can be seen in Figure 2. The grid used has 112 x 92 x 59 nodes for the x, y and z directions, respectively.

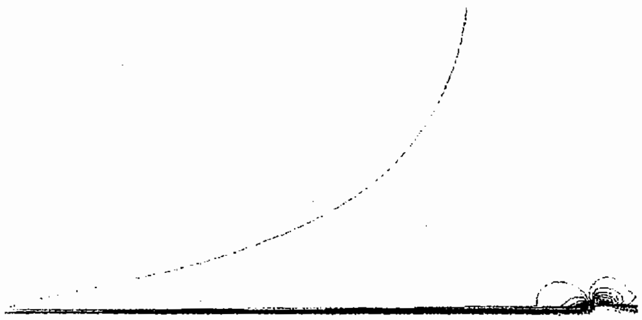

Figure 2. Flow profile in the present study

On the truncated walls and building surfaces, the wall treatment is a combination of logarithmic and no-slip boundary conditions. The implementation of wall boundary conditions in turbulent flows starts with the evaluation of y^+, where a near-wall flow is taken to be laminar if $y^+ \leq 11.63$. If the value of y^+ is greater than 11.63, 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

$$u^+ = \frac{1}{k}\ln\left(Ey^+\right) \qquad (1)$$

and

$$T^+ = \sigma_{T,l}\left(u^+ + \left[9.24\left[\left(\frac{\sigma_{T,l}}{\sigma_{T,t}}\right)^{0.75} - 1\right]\times \left\{1 + 0.28\exp\left[-0.007\left(\frac{\sigma_{T,l}}{\sigma_{T,t}}\right)\right]\right\}\right]\right) \qquad (2)$$

where von Karman's constant, k = 0.4187, E is an integration constant which for smooth walls with constant shear stress has a value of 9.793, the laminar Prandtl number $\sigma_{T,l} = 0.707$ and the turbulent Prandtl number $\sigma_{T,t} = 0.9$.

To minimise 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 2.3 x 10^5.

RESULTS AND DISCUSSION

Murakami *et al* [Murakami *et al,* 1990; Murakami, 1990; Murakami *et al,* 1996] found that the k-ε turbulence model over-estimates the k in the topside of the windward slope. Zhou and Stathopoulus [Zhou and Stathopoulus, 1997] reported that the reverse flow behind the building, the reverse flow in front of the building near the ground and the high velocity near the windward corner found in experiments have been successfully reproduced by both k-ε and two-layers methods. However, the separation at the above regions cannot be predicted by the usual k-ε model since there is an over prediction of the mixing length scale near walls, especially at the windward corner.

Based on the fact above, low turbulence values of both k-ε have been used in the present study. The low values of both k-ε reduce the overestimation near the wall since they produce less turbulence production. The block method and the grid arrangement used 'in this presentation also reduce the truncation error during computational process. A combination of low values of k-ε, block method and the grid arrangement, not only reduce the overestimation near the windward side of building and reduce the number of iteration, but also improve the accuracy of the results. The computed pressure coefficients along the centre line of the obstacle are plotted in Figure 3, and agree well with Castro and Robins' results [Castro and Robins, 1977]. An exception occurs where at the windward side near the ground produces a bit lower but a bit higher at the leeward side, compared to that of Castro and Robins [Castro and Robins, 1977].

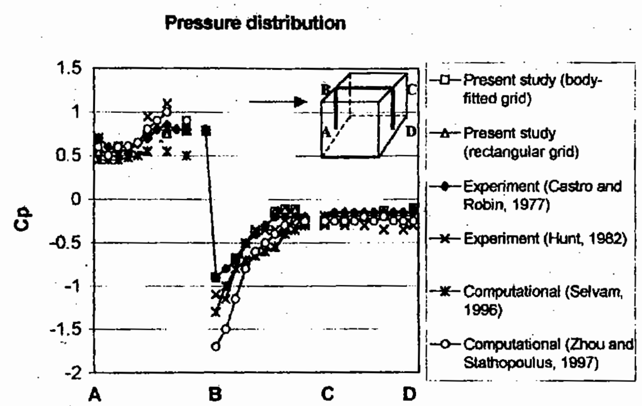

Figure 3. Pressure distribution around a cubic obstacle

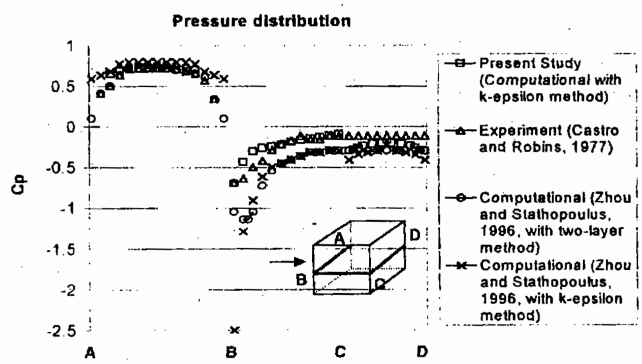

Figure 4. Pressure distribution at a half-building height

As described above, the numerical procedures for the three-dimensional model are correlated to Selvam [Selvam, 1996]. In the present study, it can be seen that pressure distributions at the windward side given in Selvam are lower than the present results. But, at the windward corner, the pressure of Selvam is higher than the present results.

Since Selvam produced a larger pressure distribution at this point, we suggest that the turbulence values used in his simulation are relatively high, as dictated that around 2.5 at the windward side, 4.5 at the top and 0.41 at the leeward side. The maximum of non-dimensional turbulent kinetic energy in the present study is 0.27 at the windward side, around 0.1 at the top and 0.05 at the leeward side. It seems that Selvam used very high turbulence values and/or the thickness sub-layer at the windward corner was not sufficiently refined, or $y^+ \leq 11.63$.

For another comparison relates to present study, Zhou and Stathopoulus [Zhou and Stathopoulus, 1996] applied a two-layer method, combining the k-ε model in fully turbulent regions with a one equation model in the near wall, or inner region. The layer was divided into two sections, inner and external regions. The wind flow in all external regions is computed with the standard k-ε model, but the flow near the building surfaces is simulated with the one-equation model.

Although their pressure distributions at the windward side agree very well with the experimental results of Castro and Robins [Castro and Robins, 1977], but the pressure distribution at the windward corners is very high, far away from our numerical and the experimental results.

At the leeward side, all numerical results produce the same pressure distribution. From the comparison study, it noted that although the two-layer method was used for the simulation, the boundary condition for turbulence values in the external region should have a reduced value. Therefore, one-equation models in the inner region should be associated with lower turbulence values in the external region. However, Zhou and Stathopoulus [Zhou and Stathopoulus, 1996; Zhou and Stathopoulus, 1997] demonstrated that the two-layer model provides a better result at the windward side, but not at the windward corner. For multi-layer models, we suggest that lower turbulence values of k-ε should be assigned in the external region.

For further comparison, the relative residue for convergence is presented in Figure 5. From that figure, it can be seen that the relative residue of present study is around 0.01. Zhou and Stathopoulus [Zhou and Stathopoulus, 1996], who used a two-layer method, have a greater tolerance value of 0.02. The number of iterations in our simulation is also smaller, indicating that the present simulation also reduces CPU time. Therefore, we note again that using smaller k-ε value not only improve the turbulent viscosity correction but also improves the accuracy of the results.

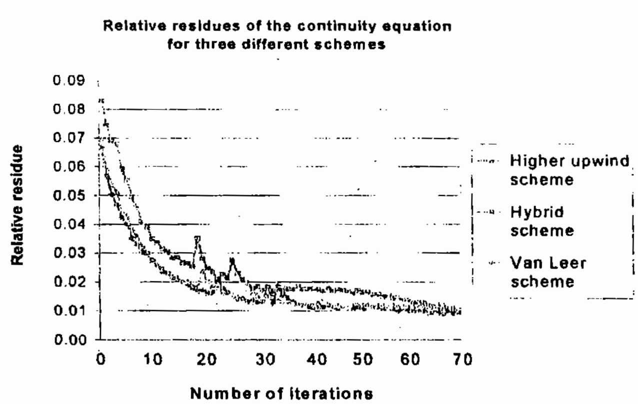

Figure 5. Relative residues for three schemes

In the recirculation zone, the effects of shear stress dominate the flow, which reduces the ability to transfer energy by convection, and the local heat transfer coefficient becomes very low at this zone. At the reattachment point the shear stress is zero, then convection heat transfer becomes the lowest, but tends to increase thereafter. This reattachment length however is a parameter to understand the effect of shear stress and convective energy on the flow. The reattachment length is about 1.67 at the leeward side and 0.32 at the windward side, for the flow that has a Reynolds number of 2.3×10^5.

In the present simulation, it can be seen that the reattachment length is 1.67 at the leeward side and 0.32 at the windward side of the building. These reattachment lengths are comparable to Frank [Frank, 1989] who used experimental and Frank [Frank, 1996], who used the Large Eddy Simulation (LES) method. The reattachment length of present simulation is better than the result of Larousse *et al* [Larousse A., R. Martinuzzi and C. Tropea, 1991] who used experimental, especially at topside of building, and much better than Werner and Wengle [Werner and Wengle, 1991] who use LES model on their computational. Comparison of the reattachment length to other published results can be seen in Table 1 and Figure 6. Based on the results, we suggest that the use of two-equation method such as k-ε is fairly good and comparable to LES or experimental, and this method also can be used to improve the use of standard k-ε model on computational.

Table 1. Comparison of the reattachment length to other published results

	/1/	/2/	/3/	/4/	/5/
xf/H	0.31	0.33	0.9	0.37	0.32
xt/H	0.9	---	0.75	0.78	0.55
xr/H	1.65	1.68	1.75	1.65	1.67
Re(10^5)	1	2.5	-	4.8	2.3

/1/ Larouse, Martinuzzi & Tropea [21] (experimental)
/2/ Frank [22] (experimental)
/3/ Werner & Wengle [23] (numerical)
/4/ Frank [20] (numerical, LES)
/5/ Present study (numerical, refined of standard k-ε)

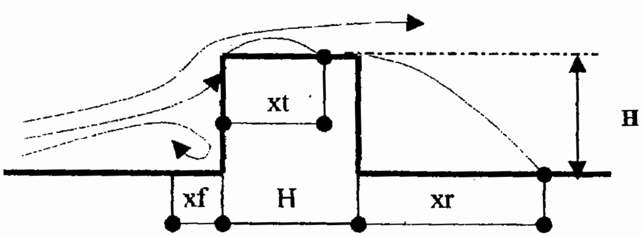

Figure 6. reattachment length where shear strees is zero at the wall

CONCLUSIONS

Based on the results of pressure distribution around building surface and number of iteration, it can be concluded that the use of low k-ε values or low fluctuating velocity components reduces the overestimation of k-ε turbulence models. This leads to an increased pressure at the windward side but reduces it at the leeward side. The higher the turbulence values, the greater the pressure distribution at the windward corner. Application of a two-layers method, i.e. the combination of the k-ε model in fully turbulent regions with a one-equation model in the near wall region, should be followed by the use of appropriate boundary conditions at the inner region. Although pressure distributions at the windward side agree very well with experimental results, the pressure distribution at the windward corner is very high. Therefore, one-equation models in the inner region should be associated with lower turbulence values in the external region. In order to reduce the over-prediction of eddy viscosity, lower turbulence values of k and ε should be used. For multi-layer models, we suggest that lower turbulence values of k-ε should be assigned in the external region. The model used here also reduced the number of iterations, indicating a reduction of CPU time. Therefore, we note that using smaller k-ε value not only improve the turbulent viscosity correction but also improves the accuracy of the results.

REFERENCES

Castro, I.P. and A.G. Robins, "The flow around a surface mounted cube in uniform and turbulent stream", Journal of Fluid Mechanics, 79, 307-335, (1977).

Hunt A., "Wind tunnel measurements of surface pressures on cubic building models at several scales", Journal of Wind Engineering and Industrial Aerodynamics, 10, 137-163, (1982).

Stathopoulus T. and Y.S. Zhou, "Computation of wind pressure on L-shaped buildings", Journal of Engineering Mechanics, 119, 1526-1541, (1993).

Versteeg H.K. and W. Malalasekera, W, "An Introduction to Computational Fluid Dynamics": The Finite Volume Method, Longman Scientific & Technical, England, (1995).

Parpia I.H., "Van Leer flux vector splitting in moving coordinates", AIAA Journal, 26, 113-115, (1998).

Ferrand P. and S. Aubert, "New mixed Van Leer flux splitting for transonic viscous flow", AIAA Journal, 34, 190-193, (1996).

Wilkes N.S and C.P. Thomson, "Evaluation of higher-order upwind differencing for elliptic flow problems", Numerical Methods in Laminar and Turbulent Flow, Pineridge Press, Swansea, 248-257, (1983).

Kawamura T., H. Takami H and K. Kuwahara, "New higher-order upwind scheme for incompressible Navier-Stokes equations", Lecture Notes in Physics, Springer-Verlag, Berlin, (1985).

Li Y. and M. Rudman, "Assessment of higher-order upwind schemes incorporating FCT for convection-dominated problems, Numerical Heat Transfer", Part B: Fundamentals, 27, 1-21, (1995).

Choi H.G. and J.Y. Yoo, "Streamline upwind scheme for the segregated formulation of the Navier-Stokes equation", Numerical Heat Transfer, Part B: "Fundamentals, 25, 145-161, (1994).

Selvam R.P. and D.A. Paterson, "Computation of pressures on and velocities near the Texas Tech. Building using staggered and non-staggered grids", 9th ASCE Structures Congress, ASCE, New York, (1991).

Patankar S.V. and D.B. Spalding, "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows", Int. J. Heat and Mass Transfer, 15,1787, (1972).

Wieringa J., "Updating the Davenport roughness classification", J. Wind Engineering and Industrial Aerodynamics, 41-44, 357-368 (1992).

Murakami S., A. Mochida A and Y. Hayashi, "Examining the k-ε model by means of a wind tunnel test and large-eddy simulation of the turbulence structure around a cube", J. Wind Engineering and Industrial Aerodynamics, 35, 87-100, (1990).

Murakami S., "Computational wind engineering", J. Wind Engineering and Industrial Aerodynamics, 36, 517-538, (1990).

Murakami S., A. Mochida, R. Ooka, S. Kato and S. Iizuka, "Numerical prediction of flow around a building with various turbulence models": Comparison of k-ε, EVM, ASM, DSM, and LES with wind tunnel test, ASHRAE Transactions, 102, 741-753, (1996).

Zhou Y.S. and T. Stathopoulus, "A new technique for the numerical simulation of wind flow around buildings", J. Wind Engineering and Industrial Aerodynamics, 72, 137-147, (1997).

Selvam R.P., "Numerical simulation of flow and pressure around a building", ASHRAE Transaction, 102, 765-772, (1996).

Zhou Y.S. and T. Stathopoulus, "Application of two-layer methods for the evaluation of wind effects on a cubic building", ASHRAE Transaction, 102, 754-764, (1996).

Frank W., "Three-dimensional numerical calculation of the turbulent flow around a sharp-edged body by means of large-eddy simulation", J. Wind Engineering and Industrial Aerodynamics, 65, 415-424,(1996).

Larousse A., R. Martinuzzi and C. Tropea, "Flow around surface-mounted, three-dimensional obstacles", 8th Symposium on Turbulent Shear Flows, TU-Munich/ Germany, 14-4-1 - 14-4-6, (1991).

Frank W., "Building aerodynamics", in Yang WJ (Ed.), Handbook of Flow Visualization, Hemisphere Publishing Corporation, New York, (1989).

Werner H. and H. Wengle, "Large eddy-simulation of turbulent How over and around a cube in a plate channel", 8th Symposium on Turbulent Shear Flows, TU-Munich/Germany, 19-4-1 - 19-4-6, (1991).