BBAA VI International Colloquium on:
Bluff Bodies Aerodynamics & Applications
Milano, Italy, July, 20-24 2008

NUMERICAL COMPUTATION OF AIR FLOW AND POLLUTION DISPERSION AROUND BUILDINGS

Mohammad Omidyeganeh and Jalal Abedi
Department of Chemical and Petroleum Engineering, University of Calgary, Calgary, Alberta, Canada, T2N1N4 Phone: (403) 220-5594
e-mails: momidyeg@ucalgary.ca, jabedi@ucalgary.ca

EXTENDED ABSTRACT

Air pollutants disposed from or near a building often have higher than the allowable concentration of various contaminants. When the pollutants are past the acceptable levels the impact is adverse. It necessitates some action to ensure the safety those who may come into contact with the pollutants. If the levels are high enough urgent action must be taken which often includes the evacuation of the premises.

In this study we have developed a finite volume code to simulate air flow at the atmospheric boundary layer to predict pollution dispersion from a variety of sources. The flow field around a single building placed in a surface boundary layer is fully turbulent and very complex. There is separation and recirculation on each surface of the building and when this problem is extended to an arbitrary configuration of a building understanding the conditions of pollutant dispersion is clearly compounded. To address this work will be to apply the solution concept of Large Eddy Simulation (LES) to turbulent flow over and around buildings with sharp edges and corners. Due to the large-scale unsteady air motions around bluff bodies, LES can be expected to be more accurate and reliable than the RANS models which are commonly used to calculate mean flow and the effects of turbulence on mean flow properties. In LES, the contribution of the large, energy carrying scales to momentum is computed exactly. The outcome is that only the effect of the smallest scales of the turbulence is modeled.

The purpose of our work is to apply the solution concept of LES to turbulent flow over and around buildings with sharp edges and corners. An implicit finite volume method is used to solve the incompressible Navier-Stokes equations on staggered grids. Convection terms in momentum equations are approximated by TVD (Total Variation Diminishing) with UMIST limiter function. In this study a central difference scheme is used for diffusion terms and Adams-Bashforth scheme for time marching. The PISO algorithm (Pressure Implicit with Splitting of Operators) makes up the main body of the code. It involves two corrector steps which enable the robust convergence behavior in addition to less computational effort than conventional SIMPLE, SIMPLER or SIMPLEC algorithms. The finite volume solver uses the top-hat filter. The grid filtering is applied implicitly through the discretisation.

Filtering the Navier-Stokes and the continuity equations gives the governing equations

$$\frac{\partial(\rho\bar{u}_i)}{\partial t}+\frac{\partial}{\partial x_j}\left(\rho\overline{u_i}\,\overline{u_j}\right)=-\frac{\partial\bar{p}}{\partial x_i}+\mu\frac{\partial^2\overline{u_i}}{\partial {x_j}^2}-\frac{\partial\tau_{ij}}{\partial x_j} \tag{1}$$

$$\frac{\partial \overline{u_j}}{\partial x_j} = 0 \tag{2}$$

A Smagorinsky-Lilly SGS model was applied in LES which assumes that the smallest turbulent eddies are almost isotropic. Therefore, the SGS turbulence model becomes

$$\tau_{ij} = -2\mu_{SGS}\bar{S}_{ij} + \frac{1}{3}\tau_{ii}\delta_{ij} = -\mu_{SGS}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) + \frac{1}{3}\tau_{ii}\delta_{ij} \tag{3}$$

The SGS viscosity is evaluated as

$$\mu_{SGS} = \rho(C_{SGS}\Delta)^2 \left|\bar{S}\right| = \rho(C_{SGS}\Delta)^2 \sqrt{2\bar{S}_{ij}\bar{S}_{ij}} \tag{4}$$

The code has the capability to use different values of C_{SGS} between 0.01 and 0.25. In this study based on simulation results $C_{SGS} = 0.1$ is the most appropriate value to use.

Inflow boundary conditions are very challenging since the inlet flow properties are convected downstream. This coupled with and inaccurate specification of the inflow boundary condition can strongly affect simulation quality. In the finite volume code two different methods can be used. The simplest method is to specify a mean streamwise velocity component to obey the power-law expressed as $u(z) \propto z^{1/4}$ in order to represent the flow of atmospheric boundary layer conditions. Other velocity components are assumed to be zero and then superimpose Gaussian random perturbations with the correct turbulence intensity. The second method is the extension of the computational domain further upstream and the use of a turbulence free inflow.

The familiar fully developed condition is used for the outflow boundary condition. Side faces and upper face boundaries are assumed to obey free slip boundary conditions. Boundary conditions at horizontal and vertical walls were specified by assuming that at the grid points (P) closest to the wall – these are the instantaneous velocity components tangential to the wall (u_p, v_p) - are in phase with the instantaneous wall shear stress components (τ_{ub}, τ_{vb}). The instantaneous velocity distribution is assumed to follow the linear law of the wall

$$\left|\tau_{ub}\right| = \frac{2\mu\left|u_p\right|}{\Delta Z} \qquad \text{for } \left|u_p\right| \le \frac{\mu}{2\rho\Delta Z} A^{\frac{2}{1-B}} \tag{5a}$$

$$\left|\tau_{ub}\right| = \rho\left[\frac{1-B}{2} A^{\frac{1+B}{1-B}}\left(\frac{\mu}{\rho\Delta Z}\right)^{1+B} + \frac{1+B}{A}\left(\frac{\mu}{\rho\Delta Z}\right)^{B}\left|u_p\right|\right]^{\frac{2}{1+B}} \qquad \text{for } \left|u_p\right| > \frac{\mu}{2\rho\Delta Z} A^{\frac{2}{1-B}} \tag{5b}$$

The code consists of two separate parts. First part is to solve Navier-Stokes equations to get to the statistically steady state condition. The second part facilitates our ability to solve the mass equation in addition to the Navier-Stokes equations to find the pollution dispersion.

$$\frac{\partial C}{\partial t} + \frac{\partial(u_j C)}{\partial x_j} = D_{AB}\left[\frac{\partial^2 C}{\partial {x_j}^2}\right] + \dot{N} \tag{6}$$

A cubical building model, with a height of H_b=*1m* is mounted on a plate with length of *17*H_b and width of *8*H_b. The center of cube is at the coordinate *(5.5*H_b*,4*H_b*)*. The height of computational domain is *4*H_b. Air properties are used in the modeling and the Reynolds number is 100000. This allows us to compare our results with experimental results obtained by Murakami and Mochida (1988) and Martinuzzi et al. (1991). Figure1 shows the structured grids around the cube generated by the code.

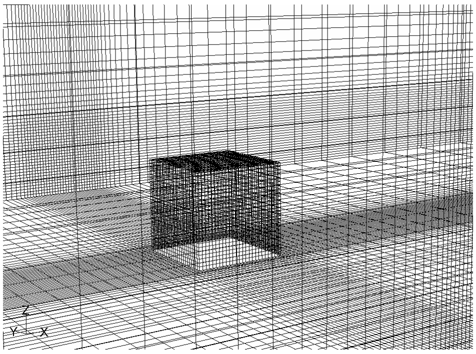

Figure 1 Grids around the cube

The effect of the grid resolution on the flow is tested for three different grid systems. The basic system has the finest grid by $0.04H_b$ spacing next to the wall with grid stretching of *1.15*. The second grid system tested has finer spacing ($0.02H_b$) next to the wall so that we can examine the grid spacing effect. Finally we explored effect of decreased stretching (1.05) which facilitates our ability to examine the stretching ratio effect. These three systems have 74×63×43, 109×98×73, and 122×95×57 grid points respectively. The results have been finalized as of yet.

.

The code is running by dimensionless time steps equal to 0.005 until get to the statistically steady state condition. The following results are from the first system of gridding that just mentioned and considers the laminar power-law inflow. These results provide information about the turbulence generation and development in domain and time. More results and discussion on the work will be provided at a later date. Figures 2 and 3 show velocity vectors around and behind the building in different times at the central vertical plane.

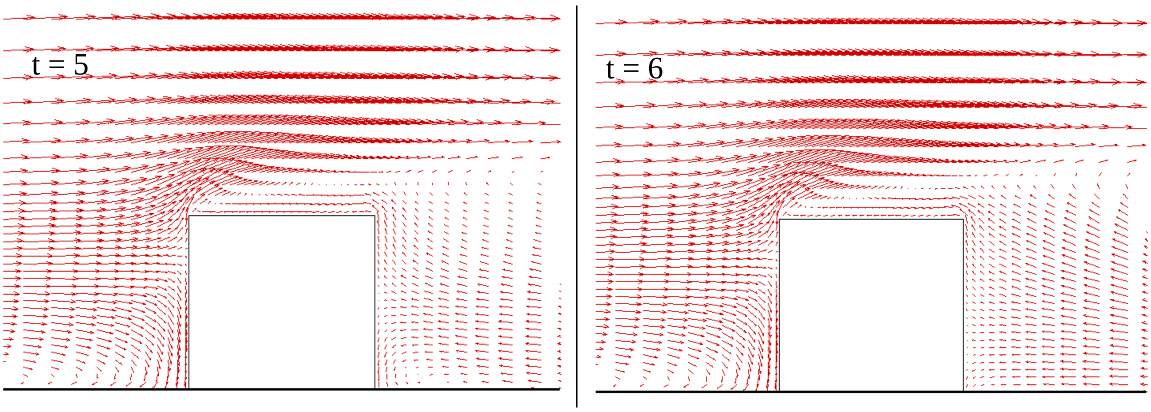

Figure 2 Velocity vectors around building at the central vertical plane at two different times

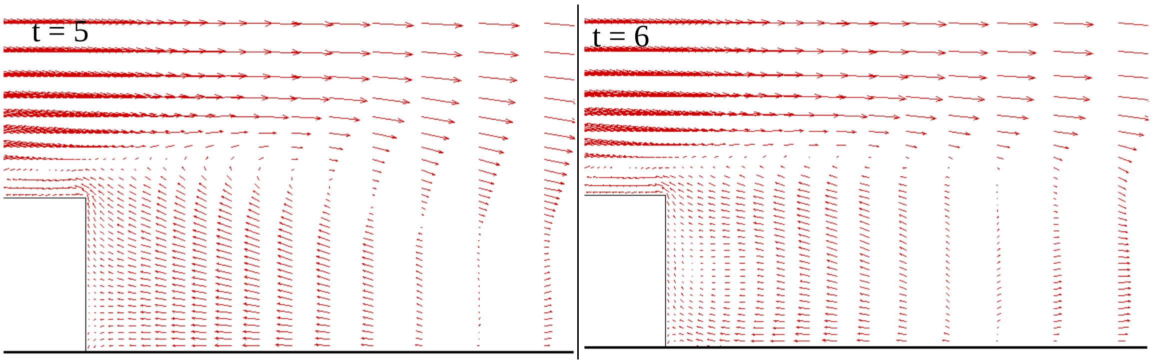

Figure 3 Velocity vectors behind building at the central vertical plane at two different times

The turbulence generated is apparent when the vortex shedding is shown in Figure4 the velocity vectors around the building at horizontal plane through center of cube at four different time steps are shown.

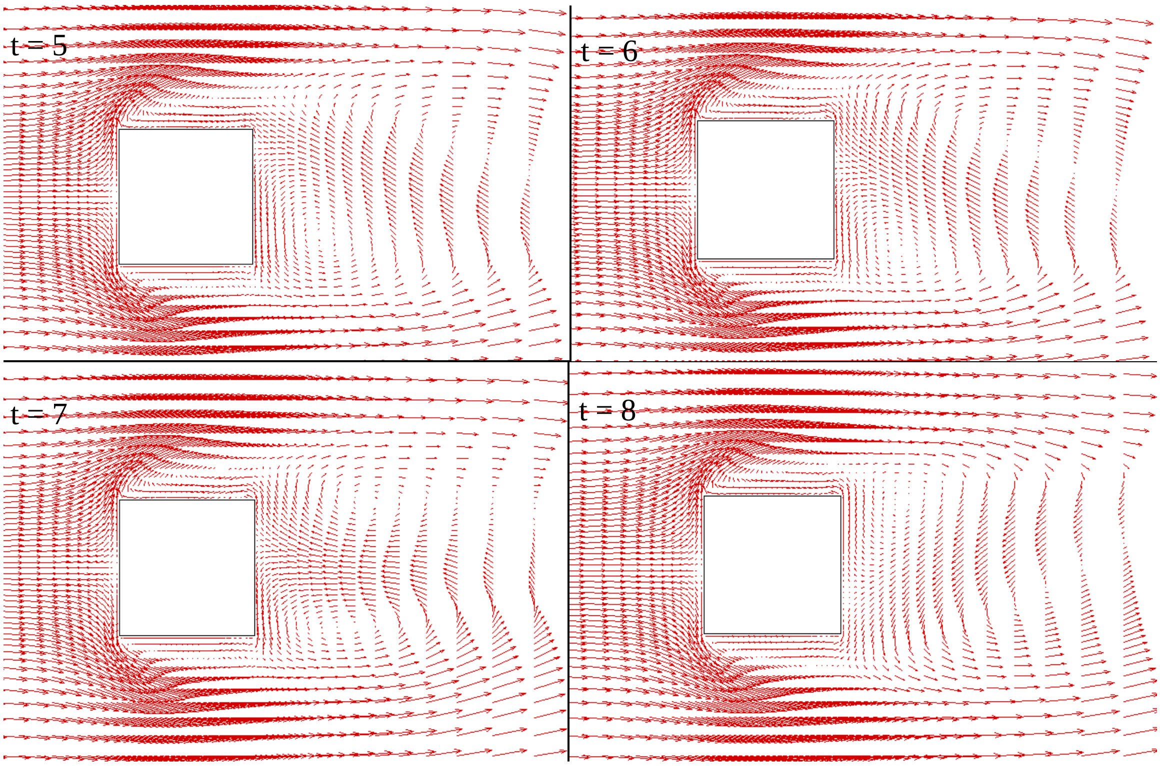

Figure 4 Velocity vectors around building at the central horizontal plane at four different times