J8.7 TESTING OF A NON-ITERATIVE CFD MODELING APPROACH FOR AN URBAN STREET CANYON

A. A. Gowardhan[1] and E. R. Pardyjak[2]
[1]Los Alamos National Laboratory, Los Alamos, NM, [2]University of Utah, Salt Lake City, UT

1. INTRODUCTION

The QUIC (Quick Urban & Industrial Complex) fast response dispersion modeling system produces high-resolution wind and concentration fields in cities. It consists of an urban wind model QUIC-URB, a Lagrangian dispersion model QUIC-PLUME, and a graphical user interface QUIC-GUI. Such models, which can quickly produce the required velocity and concentration field, have many applications. Some of the applications are as follows (Kastner-Klein, 2003).

1. Vulnerability assessments (where many simulations must be performed).
2. Training, table top exercises (where feedback or interaction is desired).
3. Emergency response.
4. Sensor siting & source inversion tools.

The 3D wind model, QUIC-URB, explicitly solves for the initial flow field around buildings using a set of empirical parameterizations and then conserves mass for this initial velocity field to obtain a final velocity field (Pardyjak and Brown, 2001).

The QUIC-PLUME dispersion model is Lagrangian model which tracks the movement of particles as they disperse through the air (Williams et. al, 2004). QUIC-PLUME uses the mean wind field computed by QUIC-URB and produces the turbulent dispersion of the airborne contaminant using random walk equations.

In this work, the Quasi-CFD (Q-CFD) solution replaces the standard Röckle parameterization for a *street canyon*. The term *street canyon* ideally refers to a relatively narrow street with buildings lined up continuously along both sides (Nicholson, 1975) as shown in Fig. 1.

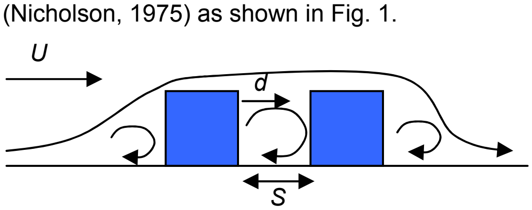

Fig. 1: Schematic of a 2D street canyon showing important flow features.

The original QUIC-URB parameterization for street canyon was based on Röckle's (1990) Dissertation work. The Röckle parameterization for skimming flow in the canyon was given by:
Vertical velocity:

$$\frac{w(x,y,z)}{U(H)} = -\left|\frac{1}{2}\left(1-\frac{d}{0.5S}\right)\right|\left(1-\frac{S-d}{0.5S}\right) \qquad (1)$$

Horizontal velocity:

$$\frac{u(x,y,z)}{U(H)} = -\frac{d}{(0.5S)}\left(\frac{S-d}{0.5S}\right) \qquad (2)$$

Where, *d* is the distance from leeward canyon wall, *S* is the stream wise building spacing and *U(H)* is the incident wind velocity normal to canyon at building height *z=H* as shown in Fig. 1. For non-normal upwind flows, the along-canyon component of wind is assumed to be unmodified (Kaplan and Dinar, 1996).

The original Röckle (1990) parameterization for a *street canyon* produced a canyon vortex that was too strong and symmetric and the lateral wall normal vortices produced at the canyon ends were weak. It was also observed that the Röckle parameterization was not able to produce the required channeling effects in the *street canyon* for non-perpendicular incident flows (Kastner-Klein, 2003).

In this work, a simplified Quasi-CFD technique has been developed which addresses all of these issues and still yields a rapid solution of the flow field. The new model is compared to the original model and experimental data.

2. INTRODUCTION TO 3D QUASI-CFD STREET CANYON MODEL

This model extends CFD techniques to a 3D street canyon between two buildings by solving the 3D Reynolds-Averaged Navier-Stokes equations for incompressible flow using a constant modeled turbulent kinematic viscosity (ν_T).

Real World building flow is a high Reynolds number phenomenon (i.e. turbulent flow) and the atmospheric turbulent processes need to be

* *Corresponding author address:* Akshay Gowardhan, Los Alamos National Laboratory, Los Alamos, NM-87545
e-mail: agowardhan@lanl.gov

modeled. However, detailed solutions as found in models based on the full solution of the Reynolds-Averaged Navier-Stokes (RANS) equations (e.g., *k-ε* model) and in Large Eddy Simulation (LES) models are too computationally expensive for fast response modeling.

It was hypothesized that the 3D RANS equations will give a solution sufficient for an initial wind field in QUIC-URB (to replace Eqn. 1 and 2 above) by using a constant value of turbulent kinematic viscosity. It was also hypothesized that the pressure Poisson equation could be incompletely solved, only for *single iteration,* making the algorithm much faster. The results that are shown below indicate that for many cases these hypotheses are valid for fast response modeling.

3. MODEL DESCRIPTION AND SOLUTION PROCEDURE

The partial differential equations governing the motion of air flows in a non-rotating coordinate system are given by Reynolds-Averaged Navier-Stokes equations for incompressible flow without body forces and may be written as:

Continuity Equation:

$$\frac{\partial \overline{U_i}}{\partial x_i} = 0 \tag{3}$$

Momentum Equation:

$$\frac{\partial \overline{U_i}}{\partial t} = -\frac{\partial(\overline{U_i}\,\overline{U_j})}{\partial x_j} - \frac{1}{\rho}\frac{\partial \overline{P}}{\partial x_i} - \frac{\partial(\overline{u'_i u'_j})}{\partial x_j} + \nu \frac{\partial^2 \overline{U_i}}{\partial x_j \partial x_j} \tag{4}$$

Where,

$\overline{U_i}$ = Mean velocity in the x_i direction

u'_i = Fluctuating velocity

$\overline{P}$ = Mean pressure

ρ = Average density

$\overline{u'_i u'_j}$ = Reynolds stresses

ν = Kinematic viscosity

Based on the presumption that there exists an analogy between the action of viscous stresses and Reynolds stresses on the mean flow, a simplified turbulence model with a constant modeled turbulent kinematic viscosity (ν_T) was used.

$$\tau_{ij} = -\frac{\partial(\overline{u'_i u'_j})}{\partial x_j} = \nu_T \frac{\partial^2 \overline{U_i}}{\partial x_j \partial x_j} \tag{5}$$

$$\frac{\partial \overline{U_i}}{\partial t} = -\underbrace{\frac{\partial(\overline{U_i}\,\overline{U_j})}{\partial x_j}}_{I} - \frac{1}{\rho}\frac{\partial \overline{P}}{\partial x_i} + \nu_T \underbrace{\frac{\partial^2 \overline{U_i}}{\partial x_j \partial x_j}}_{II} \tag{6}$$

To solve the Navier Stokes equations*,* a *pressure correction* algorithm using a central differencing scheme on a collocated mesh was used which is similar to the classical SIMPLE method (Patankar, 1980).

a. Variation of the residual in pressure Poisson solver

The pressure Poisson equation (Patankar, 1980) can be solved using an iterative *Jacobi* method (Ferziger, 2002). For complete convergence, this method requires a number of iterations proportional to the cube of the number of grid points in one direction. But, Fig. 2 shows that that the pressure Poisson equation, when solved by Jacobi method, converges more than 40% after 1st iteration and more than 90% by 10th iteration. So, as an initial estimate of the flow field the pressure Poisson equation may be solved for only *one iteration*.

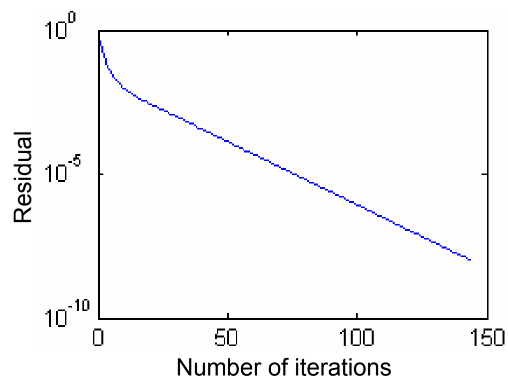

Fig. 2: Variation of the residual in pressure Poisson equation as a function of number of iterations.

b. Modeled turbulent kinematic viscosity

The density of air was taken as 1kg/m^3 and the modeled turbulent kinematic viscosity, as 1x 10^{-2} m^2/s. The value of the modelled turbulent kinematic was determined so as to make the system stable. Also, the modelled value of kinematic viscosity diffuesed a sufficient amount of momentum into the street canyon to produce results comparible to the wind tunnel data of Brown et al. (2001).

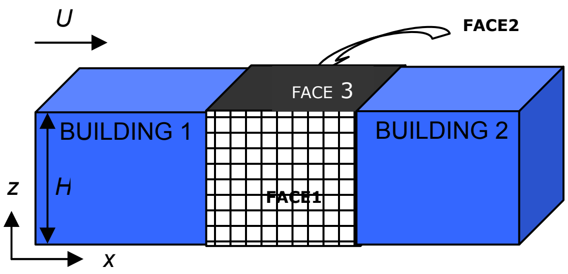

Fig. 3: Schematic diagram of the computational domain.

c. Initial and boundary conditions

The velocity and pressure fields throughout the domain were initialized with zeros. A velocity boundary condition was applied to FACE 1, FACE 2 (opposite to FACE 1) and FACE 3 (see Fig. 3), which was obtained from the initial velocity field produced by QUIC-URB. In QUIC-URB, an initial wind field is prescribed over the whole domain based on an incident flow which is developed by using the power law, log law or Macdonald/Cianco building Boundary Layer profile. The power law can be expressed as shown in Eq. 7.

$$u = u_{ref}\left(\frac{z}{z_{ref}}\right)^n \qquad (7)$$

Where u_{ref}, z_{ref}, and n are reference velocity, reference height and the power law index respectively which are input by the user. All other faces were given wall boundary condition i.e. no slip boundary condition ($\overline{U}_i = 0$).

d. Variation of fluxes in the computational domain

It was observed that the values of the advective and diffusive fluxes (terms I and II respectively in Eq. 6) in the domain depend largely on the incident wind angles. To investigate the importance of these terms, the Quasi-CFD model was solved by neglecting the advective fluxes and considering only diffusive fluxes *(o-)* as well as by considering both the fluxes *(*-)*.The following plots show the variation of fluxes along the x, y and z axes passing through the center of the street canyon and at an incident wind angle of 270 degrees.

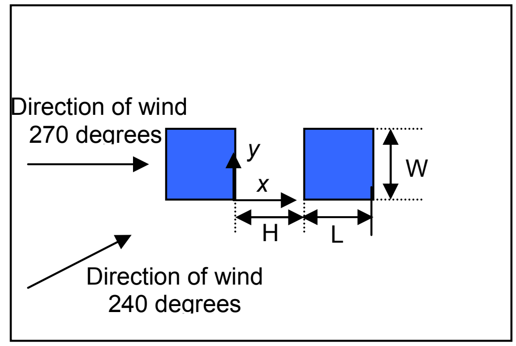

Fig. 4: Schematic showing the direction of wind with respect to the canyon geometry.

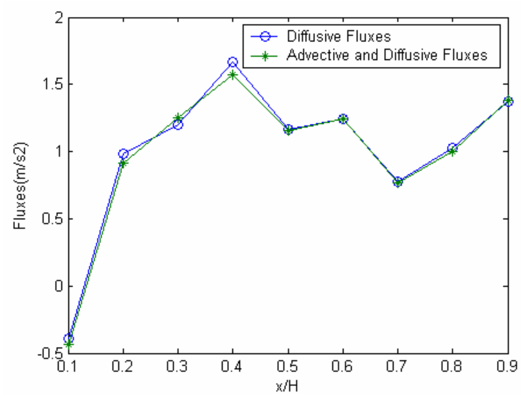

Fig. 5: Variation of fluxes in the x direction along the axis passing through the center of street canyon for an incident wind angle of 270 degrees.

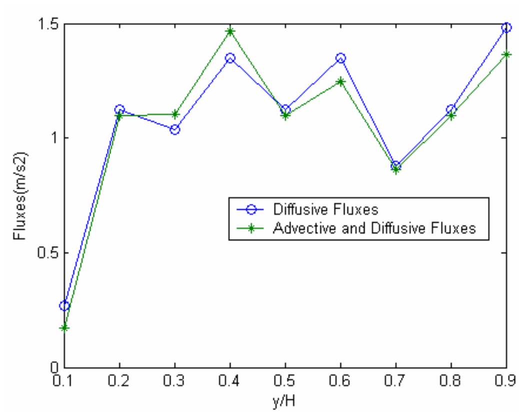

Fig. 6: Variation of fluxes along the axis in the y direction passing through the center of street canyon with an incident wind angle of 270 degrees.

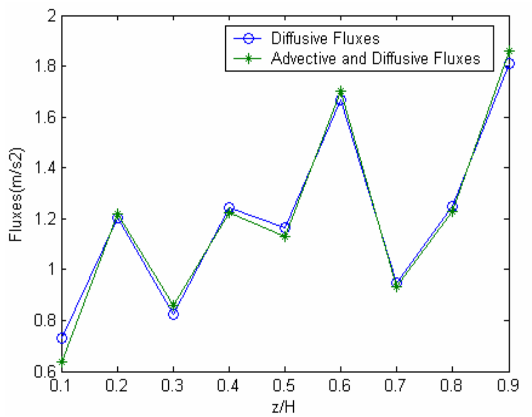

Fig. 7: Variation of fluxes along in the z direction along the axis passing through the center of street canyon for incident wind angle of 270 degrees.

Figures 5, 6 and 7 clearly show that the diffusive fluxes play a dominant role for an incident wind angle of 270 degrees. Therefore it can be said that for this particular case neglecting the advective fluxes can be justified.

Figures 8, 9 and 10 show the variation of fluxes along the x, y and z axis passing through the center of the street canyon for an incident wind angle of 240 degrees, for the cases, where (a) only the diffusive flux was considered and (b) where both the advective and diffusive fluxes were considered.

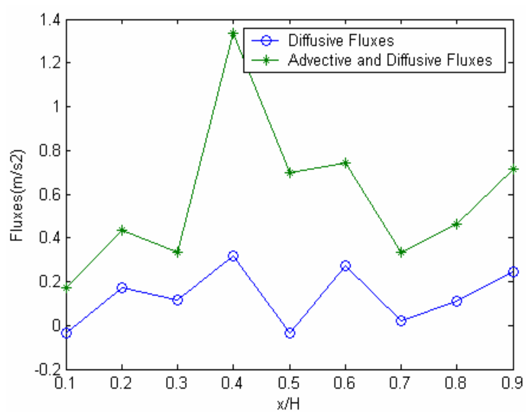

Fig. 8: Variation of fluxes in the x direction along the passing through the center of street canyon with for an incident wind angle of 240 degrees.

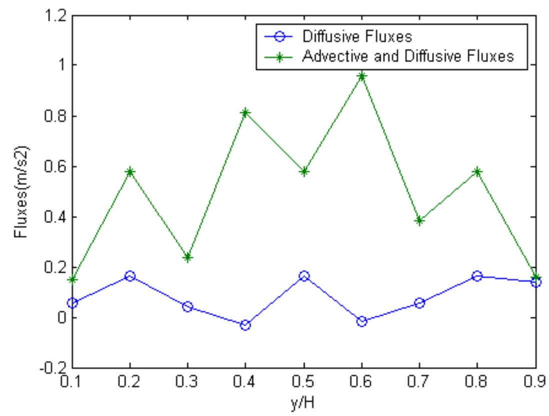

Fig. 9: Variation of fluxes in the y direction along the axis passing through the center of street canyon for an incident wind angle of 240 degrees.

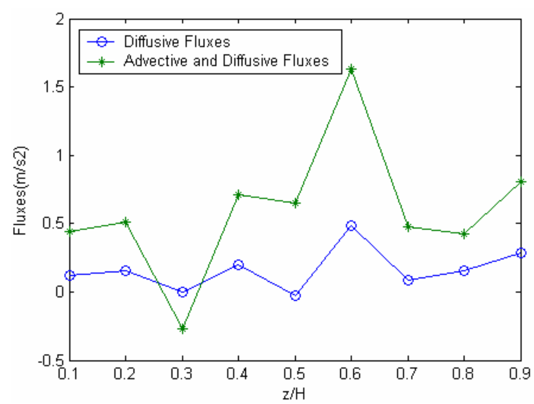

Fig. 10: Variation of fluxes in the z direction along the axis passing through the center of street canyon for an incident wind angle of 240 degrees.

Figures 8, 9 and 10 show that for an incident wind angle of 240 degrees, the advective fluxes also play an important role as the values of both the fluxes are comparable. For this case, when the Quasi-CFD model was solved by considering both the fluxes, the total flux has a considerably higher value compared to when it was solved by neglecting the advective fluxes.

As expected, the results indicate that the advective fluxes cannot be neglected for incident wind angles other than 270 degrees.

3. DESCRIPTION OF VALIDATION WORK

QUIC-URB results using the Q-CFD model and the original Rockle street canyon parameterizations for a cubical street canyon were compared with available data for two incident wind angles.

a. Validation of cross flow over an array of cubical buildings

The experimental data were obtained from the experiments carried out in the U.S. Environmental Protection Agency's (EPA) fluid modeling facility maintenance. The wind tunnel was 3.7 m wide 2.1 m high and 18.3 m long. The free stream air speed in the wind tunnel was 0.3-8 m/s. A pulsed wire anemometry (PWA) system was used to measure velocities (Brown et al. 2001).

The 3D array was 7 x 11 and consisted of cubes (0.15 x 0.15 x 0.15m) with a spacing of H giving S/H ratios of unit. According to the criteria of Oke (1987) the 3D arrays should be somewhere between the skimming and wake interference flow regimes. The length scale was equal to H and the reference velocity was 3 m/s at z =H, which produced a flow with the Reynolds number approximately equal to 30,000, which is well above the critical value required for Reynolds number independence (Brown et al., 2001).

b. Validation of off-angle flow over a street canyon between two cubical buildings

The lack of availability of experimental data for cases where the incident wind angle is some angle other than 270 degrees into a street canyon, led to the use of the commercial computational fluid dynamics package FLUENT 6.0.

As a part of this work, the FLUENT analysis was done on a simple 3D street canyon having two buildings as shown in Fig. 12 and the incident wind angle was chosen to be 240 degrees.

A two building street canyon as shown in Fig. 12was considered Here, *H* is the height of the building, S_y is the spanwise building spacing and S_x is the streamwise building spacing. The length of the building in the spanwise direction is *L* and its width is *W*.

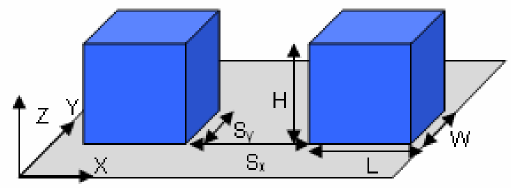

Fig. 12: Schematic of a simple 3D street canyon.

A mesh was created in the preprocessor GAMBIT. First, a *2 x 1.5 x 1.5 m* domain was generated with two identical cubical buildings with dimensions *0.2 x 0.2 x 0.2 m*. The southwest corner of the first building was placed at *x=0.5 m*, *y=0.5 m* and the second building was placed at *x=0.9 m* and *y=0.5 m* which produced a cubical street canyon as shown in Figure 13.

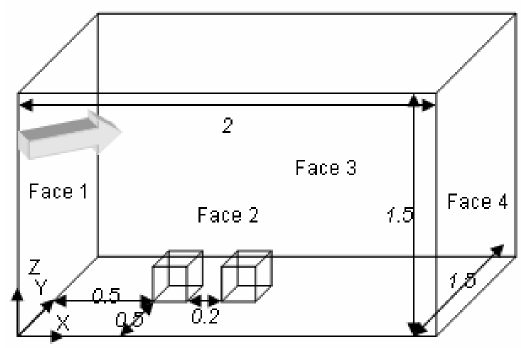

Fig. 13: Wire-frame representation of the mesh created in GAMBIT (Dimensions in meters).

Both of the buildings were subtracted from the domain to yield a single volume which was meshed by using *hex* elements and a spacing of *0.02 m* to get a fine mesh.

The different faces of the volume were given various boundary conditions. The top and bottom face of the domain and all the faces of the buildings were given a *wall* boundary condition. Face 1 and 2 of the domain were given a *velocity inlet* boundary condition and face 3 and 4 of the domain were given an *outflow* boundary condition. This mesh was exported to FLUENT.

The case was read into the Fluent and the air was selected as the fluid. The inlet faces were supplied with velocity components in the *X* and *Y* direction as 1m/s and 0.57 m/s respectively for the incident wind angle of 240 degrees.

The Standard K-ε turbulence model was chosen and at the inlet, the turbulence intensity was set to be 5% and the length scale was assumed to be 0.2 m (the height of the building). The inflow turbulence intensity is the most influential factor.

A high value (e.g. $\varepsilon / U_{in} = 0.1$, where ε is the turbulent dissipation rate) was not observed in the field measurements by Rotach (1995), but was present in the experiments described by Kastner-Klein (1999). Thus, for this analysis, the value of turbulent intensity was set to 5%.The problem was solved using FLUENT assuming incompressible 3D flow.

4. RESULTS AND DISCUSSION

a. Cross flow over an array of cubical buildings

The QUIC-URB results for cross flow over cubical buildings with different street canyon parameterizations were compared with the EPA wind tunnel data (Brown et. al, 2001). Since this model was very big, a representative model of 9 buildings (3x3) was used in QUIC-URB code as shown in Fig. 14.

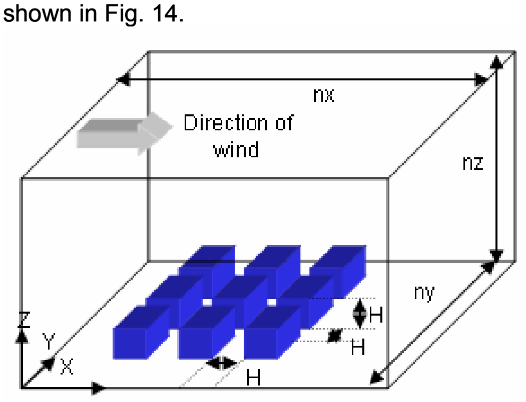

Fig. 14: Representative model of the building array used in QUIC-URB.

i. Qualitative results

Figure 15 and 16 are the vertical vector plots for a cubical street canyon (*W*=*H*=*L*) in the X-Z plane for the original Röckle model and the Q-CFD model. The above results are for the first street canyon along the central row in span-wise direction and the inflow wind is perpendicular to the building face. Figure 16 is a vertical slice vector plot of the experimental data (Brown et al., 2001).

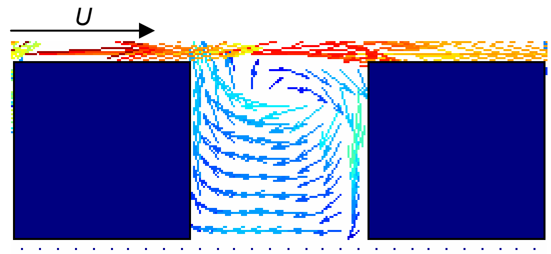

Fig. 15: Velocity vector plot for a vertical slice along the street canyon centerline showing the central canyon vortex for the original Röckle model (Inflow wind normal to the building face).

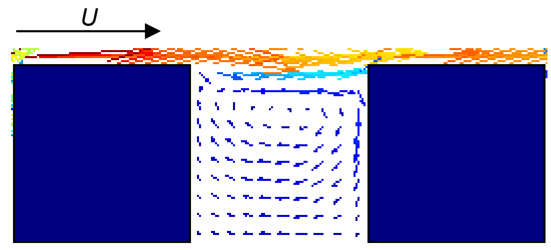

Fig. 16: Velocity vector plot for a vertical slice along the street canyon centerline showing the central canyon vortex for the Q-CFD model (Inflow wind normal to the building face).

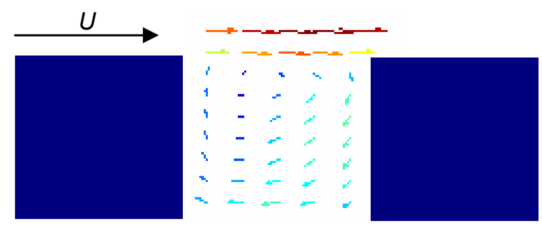

Fig. 17: Velocity vector plot for a vertical slice along the street canyon centerline showing the central canyon vortex for the experimental data (Brown et al., 2001).

As can be seen from the figure, the Quasi-CFD model produces a weaker central canyon vortex as compared to the original Röckle model which also compares better with the experimental data.

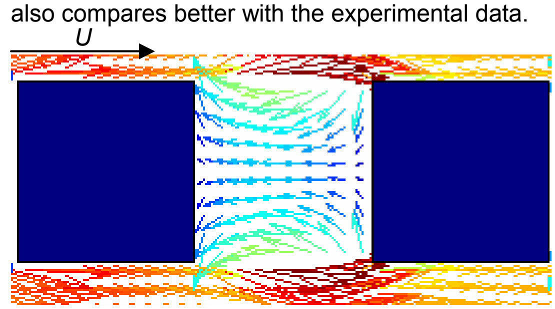

Fig. 18: Velocity vector plot for a horizontal slice along z=0.5H plane showing the wall normal vortices calculated using the original Röckle model (Inflow wind normal to the building face).

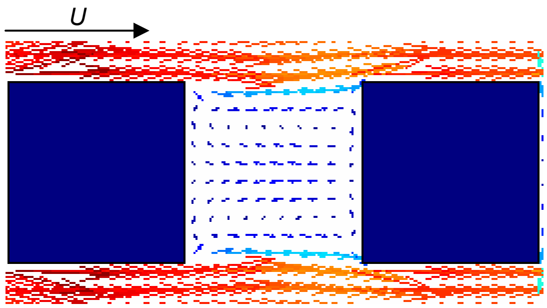

Fig. 19: Velocity vector plot for a horizontal slice along z=0.5H plane showing the wall normal vortices for the Quasi-CFD model (Inflow wind normal to the building face).

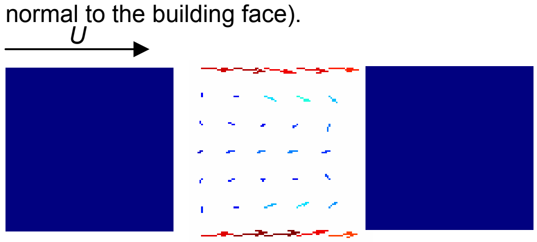

Fig. 20: Velocity vector plot for a horizontal slice along z=0.5H plane showing the wall normal vortices for the experimental data (Brown et al., 2001)

Figures 18, 19 and 20 show the velocity vectors in a horizontal slice (X-Y plane) at z=0.5H.These figures clearly show that the wall normal lateral vortices produced by the Q-CFD model are more well defined as compared to the original Röckle model and compare better to the experimental data.

ii. Direct comparison to experimental data

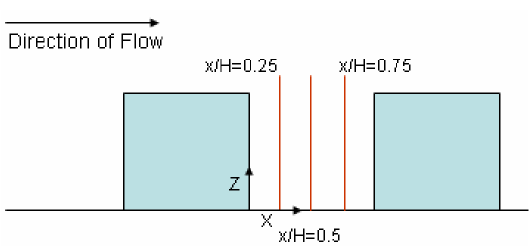

Fig. 21: Measurement locations in the first street canyon along the central row in the span-wise direction.

To test the Quasi-CFD urban street canyon model, QUIC-URB results were compared to data collected in the wind tunnel at the U.S. Environmental Protection Agency's Fluid Modeling Facility (Brown et al., 2001) described above for a 3D array of 7 x11.

The inflow winds were perpendicular to the building faces. QUIC-URB results were computed using the Q-CFD model and the original Röckle (1990) model. The grid resolution was set to 1 meter/grid.

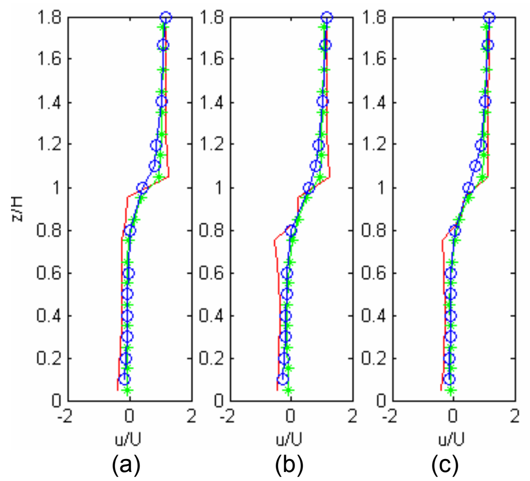

Fig. 22: Vertical profiles of normalized streamwise velocities along the street canyon centerline at (a) x=0.25H, (b) x=0.5H and (c) x=0.75H in the first canyon.(-o- EPA data, -- Röckle Model,-*-Q-CFD model)

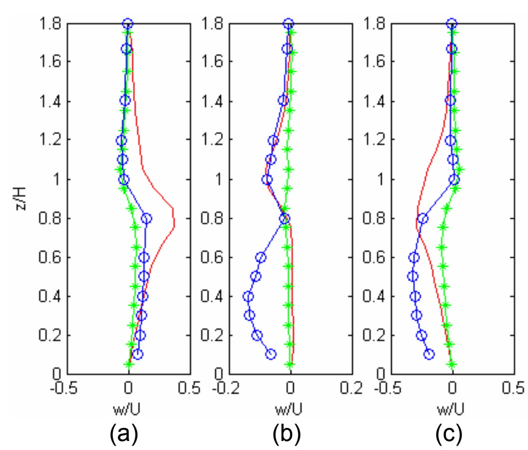

Fig. 23: Vertical profiles of normalized vertical velocities along the street canyon centerline at (a) x=0.25H, (b) x=0.5H and (c) x=0.75H in the first canyon.(-o- EPAl data, --Röckle Model,-*- Q-CFD model)

Figures 22 and 23 show the vertical profiles of normalized velocity *u/U(H)* and *w/U(H)* for QUIC-URB with the Q-CFD model (-*-), the original Röckle model (--) and the experimental measurements (-o-) in the first street canyon along the central row in the span-wise direction. The computed streamwise velocities using the Q-CFD urban street canyon model clearly match the experimental data better. The vertical velocities, however, are not much better. This is a partially a result of the sensitivity of the vertical velocities to spatial location in the canyon in conjunction with the magnitude of the velocities. For example, it can be seen in Figs.16 and 17 that the location of the center of the vortex of Q-CFD model is shifted slightly downstream compared to the experimental data.

Figures 24 and 25 show the lateral profiles of normalized velocity *u/U(H)* and *v/U(H)* for QUIC-URB with the Q-CFD model, the original Röckle model and the experimental measurements in the first street canyon of the 4th row of buildings.

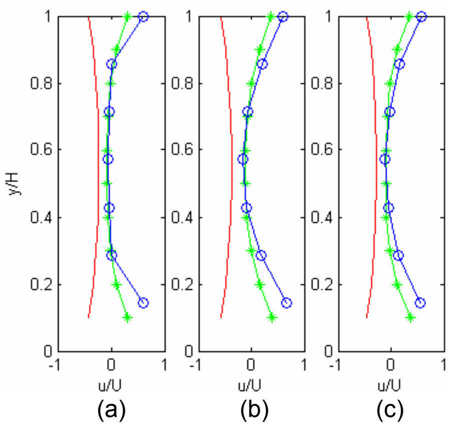

Fig. 24: Normalized streamwise velocities along the y direction at (a) x=0.25H, (b) x=0.5H and (c) x=0.75H at z=0.5H in the first canyon.(-o- EPA data, -- Röckle Model, -*- Q-CFD model).

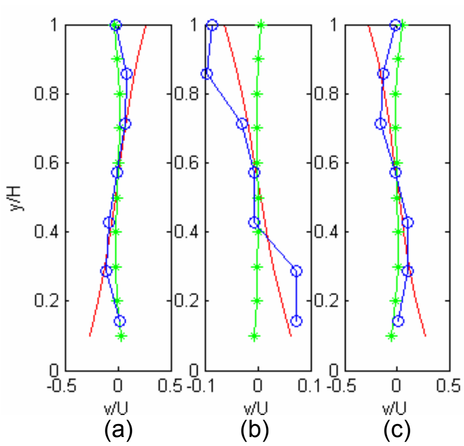

Fig. 25: Normalized spanwise velocities along the y direction at (a) x=0.25H, (b) x=0.5H and (c) x=0.75H at z=0.5H in the first canyon.(-o- EPA data, -- Röckle Model, -*- Q-CFD model).

Figures 24 and 25 clearly show that the wall normal lateral vortices produced by the Q- CFD model compares well with the wind tunnel data.

b. Flow over a simple street canyon between two cubical buildings with inflow angle of 240 degree.

i. Qualitative results

Figures 26 and 27 are horizontal velocity vector plots produced by the original Röckle model and the Q-CFD model respectively for an inflow wind angle of 240 degree. It is observed that the original Röckle model produces a poor channeling effect as compared to the Quasi- CFD model, which is an important phenomenon in areas having high density of buildings.

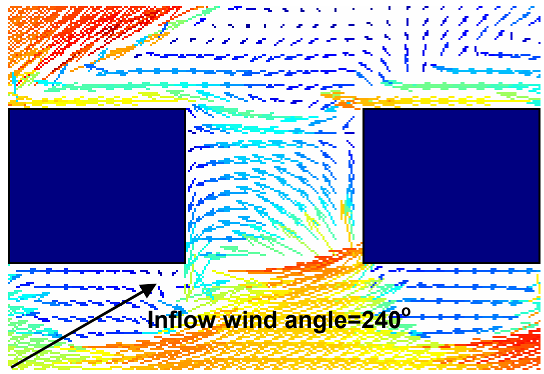

Fig. 26: Velocity vector plot for a horizontal slice along z=0.5H showing lack of channeling in the original Röckle model (Inflow wind angle=240 degrees).

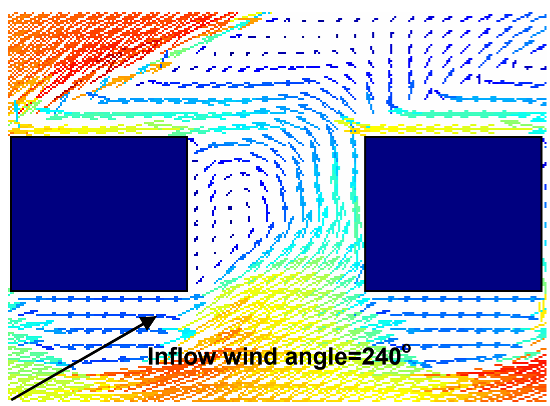

Fig. 27: Velocity vector plot for a horizontal slice along z=0.5H showing the channeling effect produced by the Q-CFD model (Inflow wind angle=240 degrees).

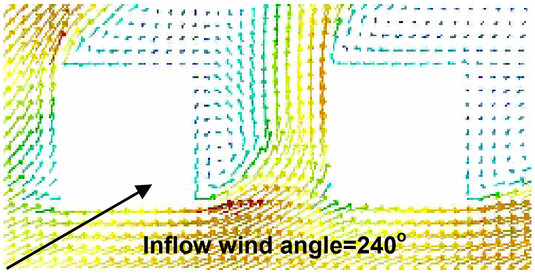

Fig. 28: Velocity vector plot for a horizontal slice along z=0.5H showing the channeling effect produced for a case run in FLUENT (Inflow wind angle=240 degrees)

Figure 28 is the vector plot of the wind field obtained form FLUENT for the same configuration. It is observed that the Q-CFD vector plot shows good resemblance to the FLUENT vector plot.

ii. Direct comparison to FLUENT calculations

The Quasi-CFD urban street canyon model was tested for an off-wind angle of 240 degrees. QUIC-URB results from the Q-CFD model and the original Röckle model were compared to data obtained from the case run in FLUENT for this particular configuration.

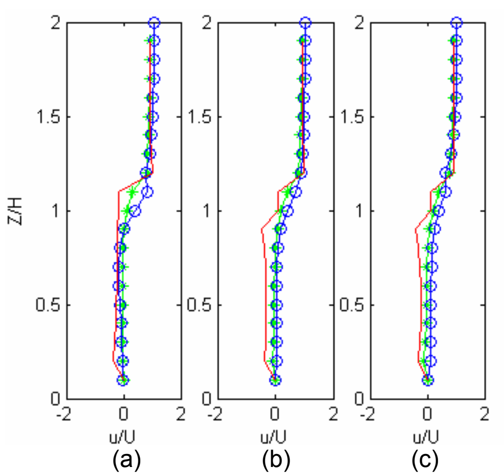

Fig. 29: Vertical profiles of normalized streamwise velocities along the street canyon centerline (y=0) at (a) x=0.25H, (b) x=0.5H and (c) x=0.75H (-o- FLUENT data, -- Röckle Model, -*- QNS model).

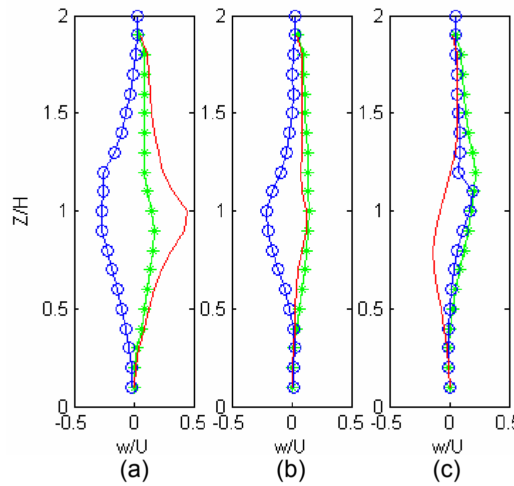

Fig. 30: Vertical profiles of normalized vertical velocities along the street canyon centerline (y=0) at (a) x=0.25H, (b) x=0.5H and (c) x=0.75H in the first canyon (-o- FLUENT data, -- Röckle Model, -*- QNS model)

Figures 29 and 30 indicates that the velocity field produced by the Quasi-CFD urban street canyon model (-*-) compares well with the FLUENT data (-o-).

5. ERROR COMPARISON

The velocity fields produced by both the Quasi-CFD urban street canyon model and the original Röckle model were compared to EPA wind tunnel data of a 3D 7 x 11 array of cubical buildings. Point by point comparison of streamwise velocities along in the vertical direction was done at the center of the street canyon.

Figure 31 show that the Q-CFD model reduced the relative error substantially when compared to the wind tunnel data.

The Q-CFD model also produces a symmetric central canyon vortex, where as it was observed that the central canyon vortex is shifted towards the leeward side of the street canyon. This deficiency can be removed by applying proper boundary condition to the Q-CFD model at the top of the street canyon. This can be achieved by introducing an appropriate rooftop parameterization.

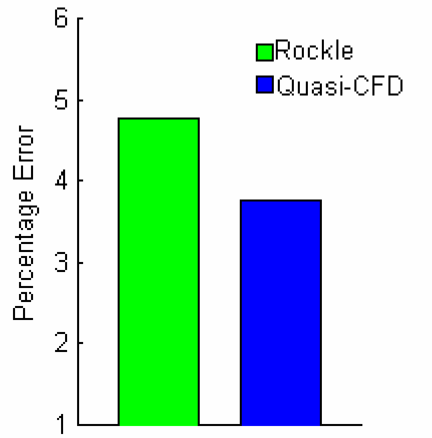

Fig. 31: Comparison of percentage errors for the Quasi-CFD and the original Röckle model for an urban street canyon.

6. COMPUTATIONAL EFFORT

Figure 32 shows a comparison of the computational effort required to solve for the velocity field for a domain size of 130 x 100 x 30 by using the Quasi-CFD and the original Röckle urban street canyon model. The domain had 6 street canyons, each having 10 x 10 x 10 grid cells.

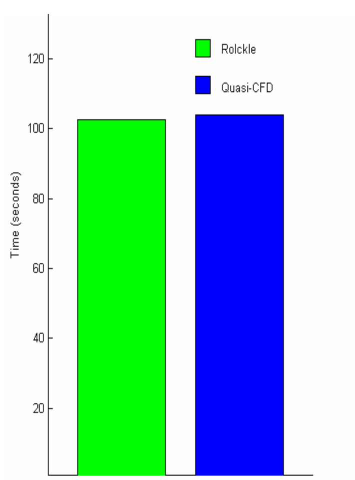

Fig. 32: Comparison of computational effort for the Quasi-CFD and the original Röckle model for urban street canyon.

It is observed that the Quasi-CFD model is only slightly more expensive (0.397%) than the Röckle (1990) model because the initial velocity field produced by the Q-CFD model was mass consistent and took fewer iterations to converge in QUIC-URB.

7. CONCLUSION

The Quasi-CFD model extends CFD techniques to a street canyon by partially solving the 3D Navier Stokes for turbulent flow. The pressure Poisson equation is not solved for complete convergence but only for *one iteration*. The value of turbulent kinematic viscosity was a constant specified to ensure a stable solution.

The convergence of the pressure Poisson equation was observed as a function of iteration and it was concluded that the Poisson equation converged more than 90% in first few iterations. Hence, it decided to solve the equation for only one iteration to produce results useful for a initial parameterization.

It was also observed that the advective fluxes can be neglected for a wind angle of 270 degrees, but for all other wind angles the advective fluxes play an important role. The Q-CFD model predicted the canyon vortex very well and produced strong wall normal vortices. The computational effort was only slightly increased and the model works well for different wind angles.

The Q-CFD model was validated for an incident wind angle of 240 degrees. Due to non-availability of wind tunnel data for such cases, a similar case was simulated using the commercial CFD package FLUENT. It was observed that the Q-CFD model predicts the magnitude and direction of the components of velocity very well as compared to the original model.

8. REFERENCES

Brown, M.J., R. Lawson, D. Decroix, R. Lee, 2001: Comparisons of centerline velocity measurements obtained around 2D and 3D building arrays in a wind tunnel, (Eds. D. Boyer and R. Rankin), *Proceedings of the 2001 International Symposium on Environmental Hydraulics*,Tempe, Arizona.

Ferziger, J.H., Peric, M. 2002: Computational Methods for Fluid Dynamics, Springer, New York.

Hotchkiss, R.S., Harlow, F.H., 1973: Air pollution in street canyons. EPA-R4-73-029.

Kaplan, H. and N. Dinar, 1996: A Lagrangian dispersion model for calculating concentration distribution within a built-up domain, *Atmospheric Environment*, **30**, 4197-4207.

Kastner-Klein, P., E.R. Pardyjak, A. Gowardhan, M.J. Brown, 2003: Evaluation of the fast response model QWIC-URB based on wind tunnel flow measurements in idealized street canyons. *11th International Conference on Wind Engineering*, Lubbock, TX June 2003.

Nicholson, S.E., 1975: A pollution model for street-level air. *Atmospheric Environment*, **9**, 19-31.

Oke, T.R., 1988. Street design and urban canopy layer climate. *Energy and Buildings*, **11**, 103-113.

Pardyjak, E.R. and M.J. Brown, 2001: Evaluation of a Fast-Response Urban Wind Model–Comparison to Single-Building Wind-Tunnel Data, (Eds. D. Boyer and R. Rankin), *Proceedings of the 2001 International Symposium on Environmental Hydraulics*,Tempe, Arizona.

Patankar, S.V., 1980: Numerical heat transfer and fluid flow, McGraw Hill, New York.

Röckle, R., 1990: Bestimmung der stomungsverhaltnisse im Bereich Komplexer Bebauugsstrukturen. Ph.D. thesis, *Vom Fachbereich Mechanik, der Technischen Hochschule Darmstadt*, Germany.

Rotach, M.W., 1995: Profiles of turbulence statistics in and above and urban street canyon *Atmospheric Environment*, **29**, 1473-1486.

Williams, M.D., M. Brown, D. Boswell, and B. Singh, 2004: Testing of the QUIC-PLUME model with wind-tunnel measurements for a high-rise building. *13th Conf. on Applic. of Air Poll. Meteor. with AWMA*, Vancouver, Canada, Amer. Meteor. Soc.