13.2 MEAN FLOW PATTERN THROUGH A SIMPLE MOCK URBAN ARRAY -WATER CHANNEL EXPERIMENTS AND MODELING

Hansheng Pan[1], Xiangyi Li[1], Michael J. Brown[2], Jong-Jin Baik[3], Seung-Bu Park[3], and Marko Princevac[1*]

1. Univerisity of California at Riverside, Riverside, California
2. Los Alamos National Laboratory, Los Alamos, New Mexico
3. Seoul National University, Seoul, Korea

1. INTRODUCTION

Urban boundary layer has specific characteristic influenced by the larger roughness elements such as corps, utilities and buildings (Britter and Hanna, 2003; Rotach, 1999). At street scale, the dispersion is controlled by turbulent mixing and mean flow transport through the network of streets (Belcher, 2005). Since pollutants, such as vehicle emissions are in close proximity to people in urban area, the understanding of the influence of street architecture on the wind flow, also on the dispersion of pollutants can help us study the urban air quality.

Most of past researches, including wind tunnel experiments studied by Kastner-Klein (2004) and Macdonald (2002) and field experiments reported by Dobre (2005) and Eliasson (2006) focused on individual buildings or on the urban canopy as a whole. Coceal (2006) and Milliez (2007) reported several numerical methods applied to simulate the flow in street canopy, which need to be verified by experimental results. The objective of this research is to study the flows through relatively simple arrays of buildings. Results will also be used to develop urban flow parameterizations in the fast response dispersion models.

Urban boundary layer simulations were performed in the water channel at the University of California, Riverside (UCR), in the Laboratory for Environmental Flow Modeling (LEFM). A re-circulating water channel was utilized to create controlled flow field to model wind flows through simple mock urban environments. In particular, flows through arrays of cubic mock buildings were studied. The experimental results and the comparison with numerical model results are given.

2. EXPERIMENTAL SETUP

The water channel at the University of California, Riverside is a custom-designed re-circulating water channel with a test section that is 1.5 m long, 1 m wide and 0.5 m deep. The axial pump (Carry Manufacturing, Inc., 20 HP, 8" in diameter) drives the flow from the settling tanks and produces a maximum mean velocity of 0.5 m s^{-1}. A variable frequency controller allows flow control with a resolution of 1/100 Hz (from 0 to 60 Hz) which corresponds to the mean velocity change resolution of only 0.08 mm s^{-1}. Two 0.5 m thick flow conditioners, in the form of honeycombs, are placed at the entrance to the channel to prevent flow recirculation and reduce the turbulence level. The perforated screens are used for adjusting the flow to the desired logarithmic profile. The channel flow is steady and becomes fully developed before reaching the test section. Figure 1 gives the schematic and photo of the water channel.

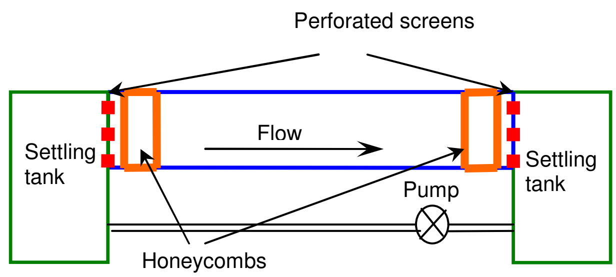


Fig. 1a. Water channel schematic

* *Corresponding author address:* Marko Princevac, Univ. of California at Riverside. Dept. of Mechanical Engineering, Riverside, CA 92521; email: marko@engr.ucr.edu

Fig. 1b. Water channel

For velocity measurements TSI's Particle Image Velocimetry (PIV) system was used. The PIV system includes a 400 mJ Nd-YAG laser (Big Sky Laser Technologies, Inc), a LASERPULSE Synchronizer (TSI Inc.), and a PowerView Plus 2M camera with CameraLink frame grabber (TSI Inc.). Particles used to seed the channel were Pliolite Ultra 100 (Eliokem). Insight 3G (TSI Inc.) software was utilized for data collection and TecPlot (Tecplot, Inc.) was used for the velocity vector field visualizations. Detailed velocity and concentration fields can be measured in horizontal plane. (see schematics on Figure 2 for corresponding camera/laser setups).

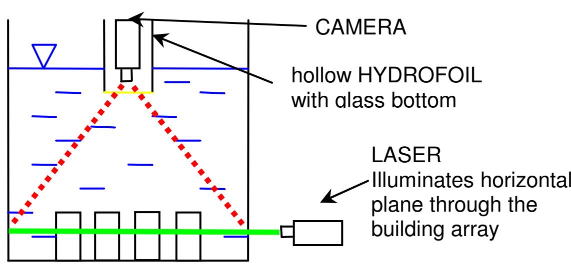


Fig. 2. Schematic of the laser-camera configuration for the PIV measurements in horizontal plane

Building configurations simulating simple urban patterns were accomplished using highly polished acrylic models to minimize effects of refraction and attenuation of the laser sheet utilized for the PIV measurements. By utilizing the PIV system, resultant flow fields were measured over a period of up to 5 minutes.

3. EXPERIMENTAL RESULTS

Initial experiments were conducted without any buildings in the channel to verify the logarithmic velocity profile and a sufficient level of turbulence in the background flow. Experiments related to flows through regular arrays of cubes will be presented here.

An array of 3 x 3 cubes was placed perpendicular to the flow in the channel test section. The cubes have H = 5 cm long edges and the distance between cubes ("street width") was S = 5 cm. The same condition was applied to the experiments for an array of 5 x 5 cubes.

3.1 *Flow through a 3 x 3 Array of Cubes*

Schematic of the measured flow pattern is given in Figure 3. As fluid enters the array it is deflected sideways after the first building row. It flows back into the array behind the second building row.

The detailed measured flow field in the horizontal plane, for the uniform height array, is given in Figure 4. The presented flow field is measured at 1/3H (1.6 cm) from the ground. The measurements were also performed at 1/2H and 2/3H from the ground level. The inflow/outflow pattern on both sides of the array is the same independently on the measurement height.

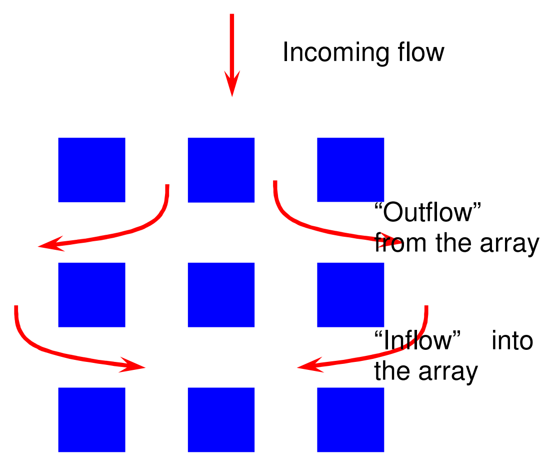
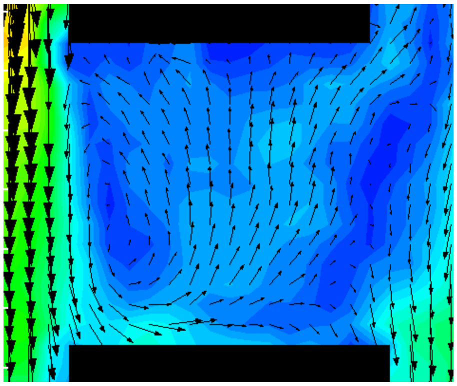


Fig. 3. Schematic of the measured flow pattern in the horizontal plane through a 3 x 3 building array.

a)



b)

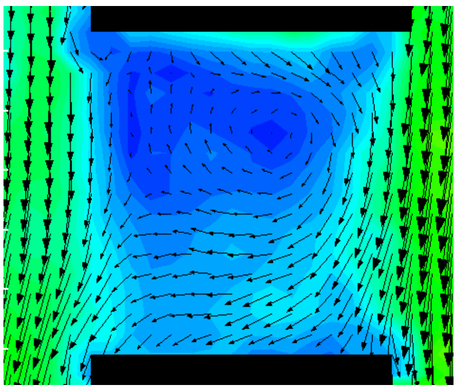
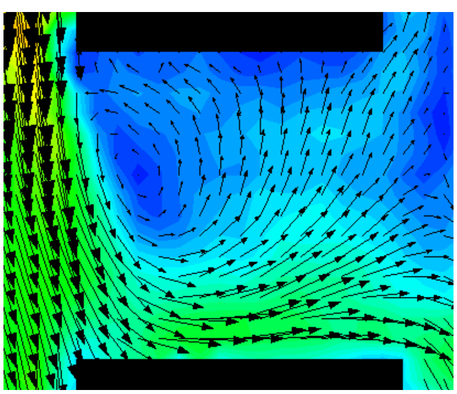


Fig. 4. a) Outflow and b) Inflow in a 3 x 3 building array with uniform height (refer to schematic in Fig. 3).

Figure 5 presents the flow pattern measured at the same locations and under the same incoming flow conditions as the one given in Figure 4 with the only difference being the tall central building. Note how the tall building causes even more intense “side flows” from and into the array.

a)



b)

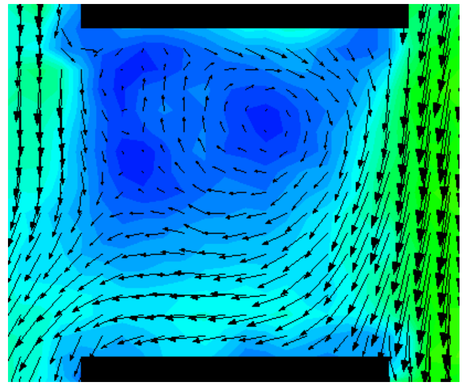


Fig. 5. a) Outflow and b) Inflow in a 3 x 3 building array with the middle building of double height (refer to schematic in Fig. 3).

3.2 *Flow through a 5 x 5 Array of Cubes*

Schematic of the measured flow pattern in a 5 x 5 building array is given in Figure 6. The fluid enters the array after each row of building except the first one. The detailed measured flow field is given by Figure 7.

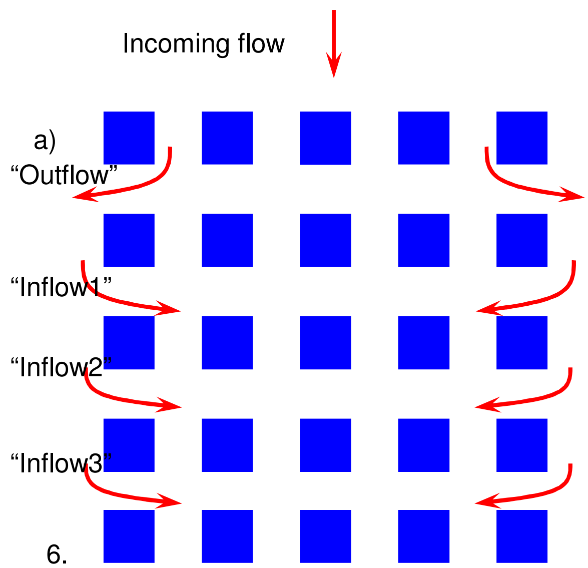


Fig. 6. Schematic of the measured flow pattern in the horizontal plane through a 5 x 5 building array.

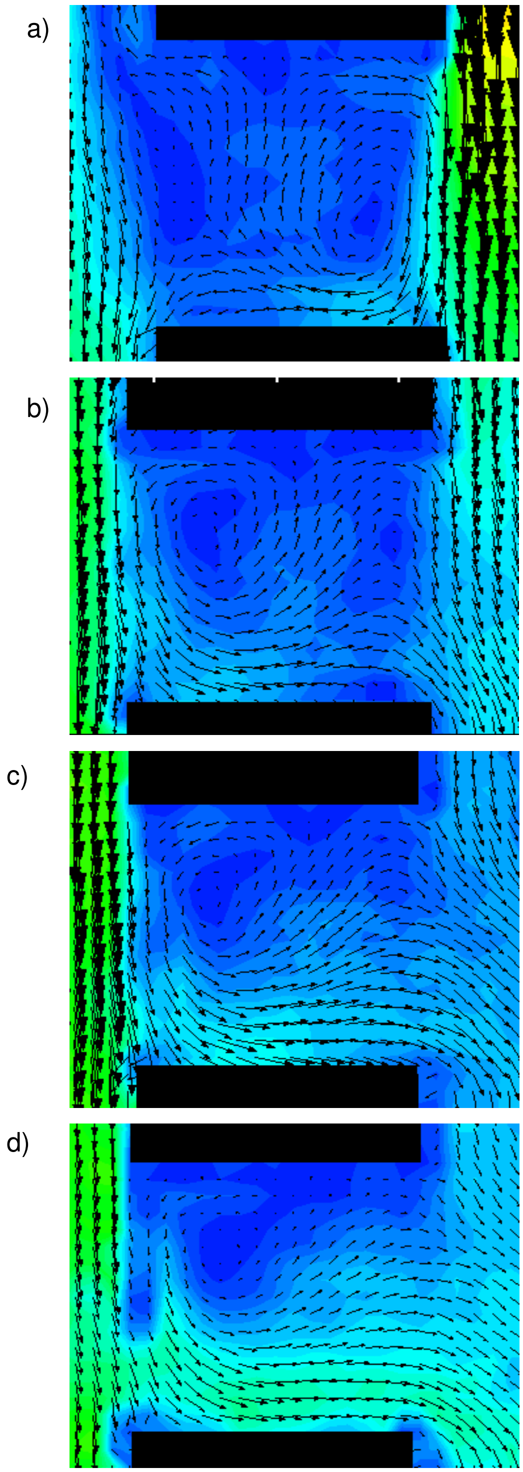


Fig. 7. a) Outflow, b) Inflow1, c) Inflow2 and d) Inflow3 in a 5 x 5 building array with uniform height (refer to schematic in Fig. 6).

4. NUMERICAL MODEL RESULTS

The CFD model with the *k*–ε turbulence closure scheme (Baik *et. al.*, 2003) was deployed to the same building configuration as the laboratory experiments. Assuming neutral atmosphere, logarithmic velocity profile with a roughness length of 0.05 m is used as the incoming wind. Reference velocity was 10 m s^{-1} at the 10 m height above the ground level. Cubical building height is also 10 m. Zero shear condition is used at the domain boundaries. Model results for the flow through the uniform height array and flow pattern through the array with tall central building are given in Figures 8 and 9, respectively. The model reproduced the measured flow patterns. The model was run at very high *Re* ~ 10^7 eliminating any possibility that the observed pattern is some low *Re* artifact.

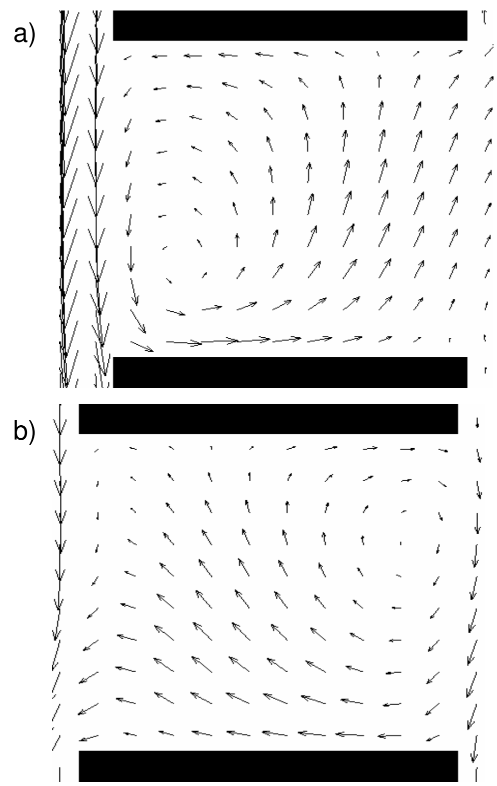


Fig. 8. a) Outflow and b) Inflow in a 3 x 3 building array with uniform height produced by a CFD model compared with Figure 4a and 4b.

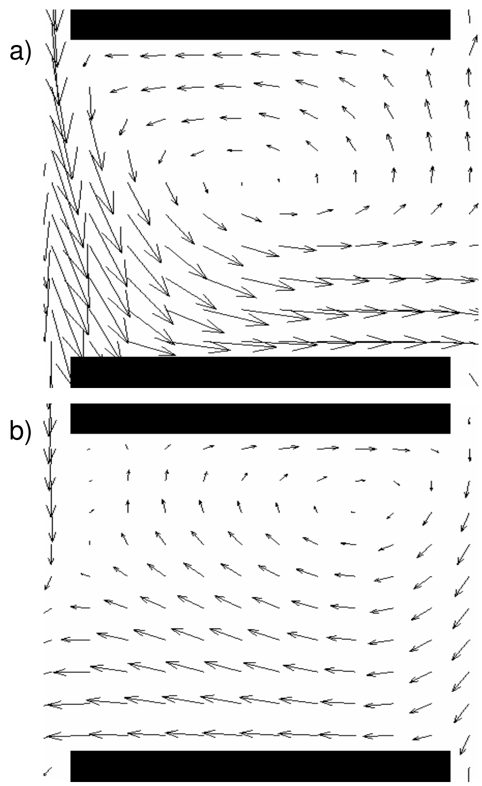


Fig. 9. a) Outflow and b) Inflow in a 3 x 3 building array with the middle building of double height produced by a CFD model compared with Figure 5a and 5b.

5. SUMMARY

Urban boundary layer simulations were performed in the water channel at the University of California, Riverside (UCR), in the Laboratory for Environmental Flow Modeling (LEFM). A re-circulating water channel was utilized to create controlled flow field to model wind flows through simple mock urban environments. In this communication the flows through the arrays of both 3 x 3 and 5 x 5 cubical mock buildings are reported. A new flow pattern within the array was observed in horizontal plane. This flow pattern shows regions of the mean outflow from the array and the mean inflow into the array (Figures 4, 7 and 8). For the case of the tall central building this “sideway” flow is significantly enhanced (Figures 5 and 9). This is the first time that such detailed measurements through a modeled urban setup are available. This pattern may be responsible for the observed and well documented (Belcher, 2005) initial plume spread within the array of obstacles.

ACKNOWLEDGMENTS

This work is supported by the Los Alamos National Laboratory, contract number 38510-001-06.

REFERENCES

Baik, J.-J., Kim, J.-J., Fernando, H. J. S., 2003: A CFD model for simulating urban flow and dispersion. J. Appl. Meteorol., 42, 1636-1648.

Belcher, S.E., 2005: Mixing and Transport in Urban Areas, *Phil. Trans. R. Soc. A*, 363, 2945-2968,

Britter, R.E. and Hanna S.R. 2003: Flow and Dispersion in Urban Areas. *Annu. Rev. Fluid Mech*. 35, 469-496.

Coceal, O., Thomas, T.G., Castro, I.P. and Belcher, S.E., 2006: Mean Flow and Turbulence over groups of urban-like cubical obstacles, *Bound. Lay. Meteo.,* 121, 491-519.

Dobre, A., Arnold, S.J., Smalley, R.J., et al., 2005: Flow field measurements in the proximity of an urban intersection in London, UK. *Atmos Environ.* 39, 4647-4657.

Eliasson, I., Offerle, B., Grimmond, C.S.B., et al., 2006: Wind fields and turbulence statistics in an urban street canyon. *Atmos Environ.*40, 1-16.

Kastner-Klein, P.E. and Rotach, M.W. 2004: Mean flow and turbulence characteristics in an urban roughness sublayer. *Bound.Lay. Meteo.* 111, 55-84.

Macdonald, R., Carter, S. and Slawson, P., 2002: Physical modeling of urban roughness using arrays of regular roughness elements, *Water, Air, & Soil Poll.*, 2, 541-554.

Milliez. M and Carissimo B., 2007: Numerical simulations of pollutant dispersion in an idealized urban area, for different meteorological conditions, *Bound. Lay. Meteo.*122, 321-342.

Rotach, M.W., 1999: On the Influence of the Urban Roughness Sublayer on Turbulence and Dispersion. *Atmos Environ*. 33. 4001-4008.