4.3

Preliminary Results of CFD Simulations for the Scenario of a Recent Field Study in an Urbanized Domain

Ronald M. Cionco[1], Alan Huber[2], and Wei Tang[3]

[1]Retired, Las Cruces, NM
[2]NOAA/ARL/Atmospheric Sciences Modeling Division
in partnership with US EPA National Exposure Research Laboratory, RTP, NC
[3]Lockheed-Martin Operations Support, RTP, NC
in support of US EPA Environmental Modeling and Visualization Laboratory

1. INTRODUCTION

In our progression of research for a better understanding of wind flow in an urbanized domain (i.e. proceeding from physical modeling results to atmospheric field measurements and now on to our present study) ultra-high resolution computations with a mature computational fluid dynamics (CFD) code have been completed based upon a recent field study. A method for running a single CFD model specifically for six different wind directions is demonstrated to show how this model can simulate the full range of the wind speeds and directions at field measurement locations surrounding a single building within a cluster of six other adjacent buildings. More information on the field experiments, including the wind sensors, may be found in Cionco et al (2006).

2. THE BUILDING MODEL

A digital model of the main study building (front and center) and six surrounding buildings (all buildings generally less than 10 m high) was made to match engineering drawings of the buildings supported by field verifications of key building dimensions. Figures 1 and 2 show overviews of the buildings with poles marking the four 10-m towers (SW,S,N,NE), three 3-m tripods (SE Tripod, NE Tripod, Reattach), and a 5-m roof mast (Roof). The faces of the three front buildings are oriented parallel with the South to North direction (left to right). The view looking perpendicular to these building faces is toward the West direction.

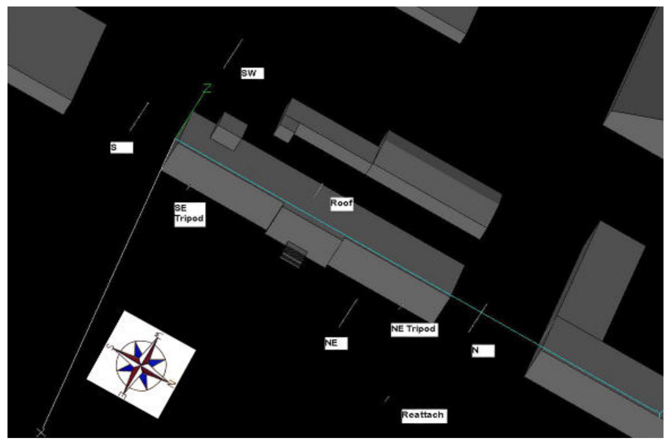


Fig. 1. Location of wind sensors.

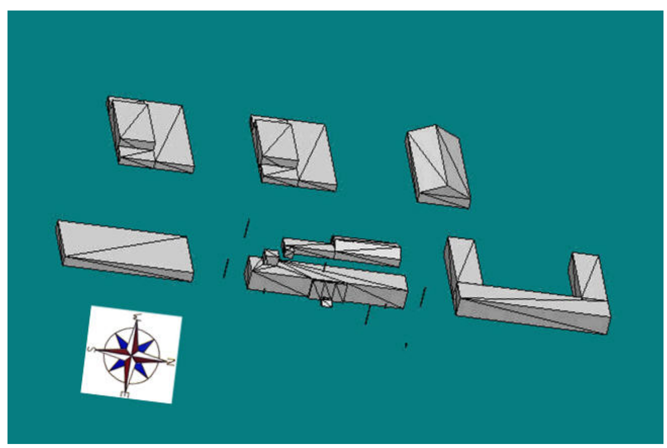


Fig. 2. Overview of buildings and sensor locations.

3. CFD MODEL SET UP

Special applications using the FLUENT (2006) CFD software code were developed and applied to support this study. FLUENT is codes to solve the governing equations for the conservation of mass, momentum, energy, and scalars such as a pollutant. The study domain is divided into discrete control volume cells using a computational grid mesh. Unstructured meshing (hexahedral dominate) supports variable volume cell sizes throughout the domain. This allows for better computational efficiencies by being able to concentrate the grid mesh in areas where finer mesh is most critical in resolving complex flows.

The software has options for either steady or unsteady (time-varying) solutions. For this study to date we are applying steady-state flow solutions

to demonstrate how they may be used to simulate time-varying field measurements of wind speed and wind direction.

Solutions require a selection of boundary conditions and a model for turbulence. These CFD simulations used a realizable k-e turbulence model that was applied with the solution for the steady-state RANS (Reynolds-Averaged Navier-Stokes) equations.

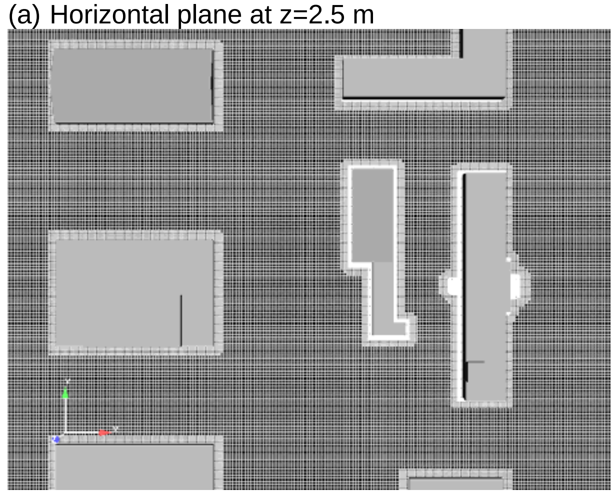
A horizontal model domain of 400 m by 400 m was set surrounding the 7 building cluster. A vertical domain of 200 m was found to be of sufficient depth for simulation of the influences surrounding the building cluster with all buildings less than 10 m high. Cell size is 0.25 m surrounding the faces on the main building and

0.5 m surrounding other building faces. Below 20 m the maximum, cell size is 1 m, filling most of the domain and ground surface. Above 20 m, the cell size grows from 2 m, to 4 m and finally 8 m. The total cell count is 4.5 million. Figure 3 presents overview examples showing the mesh.

The setup of the CFD boundary layer was developed to determine a good match to field measurement for the SW Tower. The SW Tower field measurements are upwind of the main study building. However, based on examination of its wind profile and the subsequent CFD simulations the SW Tower may be influenced some by the upwind building. The total mass flow and surface roughness (z_0) was adjusted in the CFD boundary layer over a flat surface. An initial analysis of the 3 level wind profiles examine a best log-linear fit plus ratios of wind speed at 2.5 m and 5 m to the 10 m wind speed. The initial goal was to match the average wind speed for the SW Tower: z = 2.5, 5, 10 m; wind speed = 4.944, 5.672, and 6.9586 m/s. An initial estimate of z_0 = 0.089 m was found to be deficient.. Then a value of z_0 = 0.3 m along with an increase in the mass flow rate was found to provide a better match to the shape of the wind profile. Figure 4 presents these profiles.

The single set up with the wind profile using z_0 = 0.3 m matching the average SW Tower wind profile was applied as the inlet boundary for all applications. The CFD simulated winds were normalized by the value at the SW Tower 10-m location and rescaled as appropriate to match the field wind speed at the SW Tower 10-m location.

(a) Horizontal plane at z=2.5 m



(b) Vertical plane looking Northward

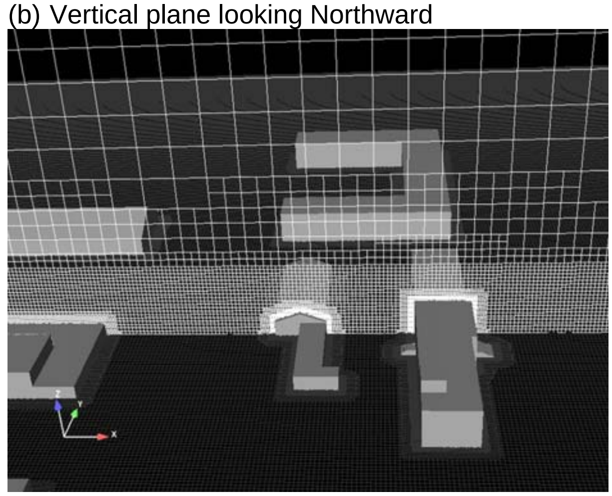


Fig. 3. Overview of computational mesh.

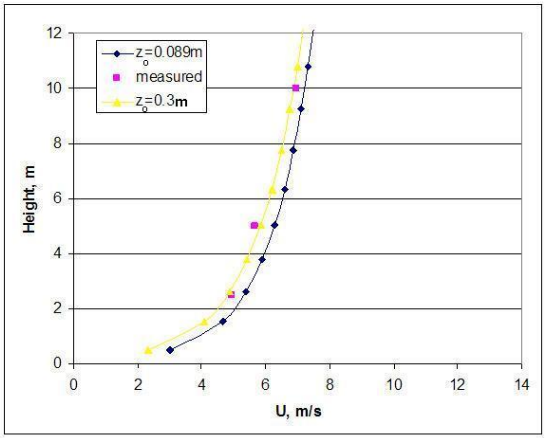


Fig. 4. Mean velocity: measured and CFD for two z_0 Values.

4. METHODS USED

For this study, the only field measurements that were seen by the CFD modelers prior to completing the CFD simulations were those collected specifically at the 10-m level of the upwind 10-m tower – the Reference Tower. We, therefore, consider this to be a “blind test” of the CFD modeling methods.

The high frequency wind measurements were smoothed to their 1-minute running averages for a five hour period. See Figure 5 below for examination of the range of wind speed and directions during this period.

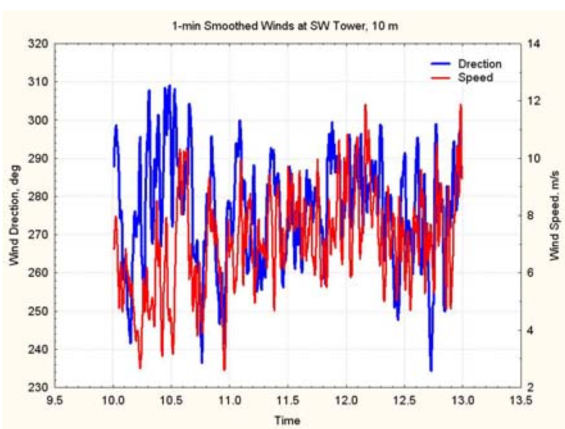


Fig. 5. Wind direction and speed with 1-minute smoothing.

The running average (smoothed) 1-minute (measured) directions where categorized into 10-degree wind direction bins, analyzed, and compared with a steady-state constant direction CFD model simulations, such that the

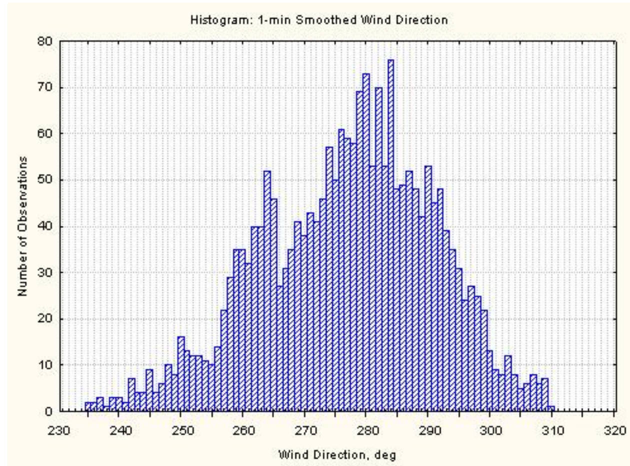


Fig. 6. Histogram for 1-minute smoothing.

model inlet boundary wind directions were set at 250, 260, 270, 280, 290, and 300 degrees. Figure 6 shows a histogram of the measured wind directions.

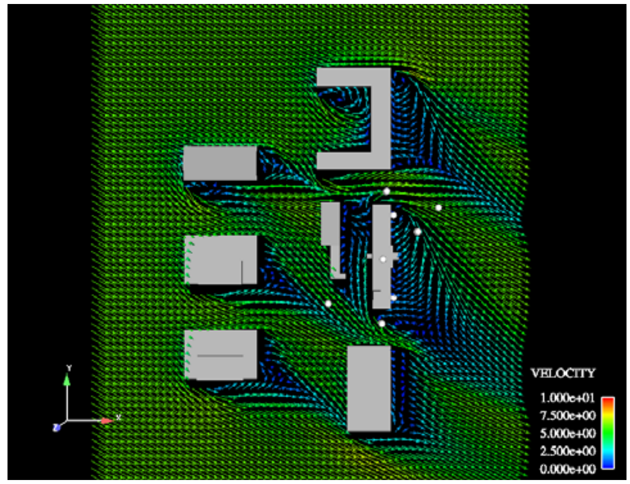
During this study, wind fields were generated for each 10 degree increment between 250 to 300 degrees to cover the range of field conditions. There were only six CFD simulations each having the same inlet boundary conditions. The only difference among the six CFD simulations was the building orientation. A 1-minute running average (smoothed) wind direction was used to provide low frequency wind direction fluctuations that may be represented by a series of CFD steady-state fixed wind direction simulations. Keep in mind that the time series wind direction variation is not something that is directly simulated by a steady-state constant direction CFD model. However, this method provides an efficient way to simulate an approximation to a time-varying field situation.

Results will be shown as derived from the present model study demonstrating a method using steady-state CFD model simulations to model real time varying winds within a building cluster knowing only an upwind 10-m wind speed and direction not significantly influenced by the buildings.

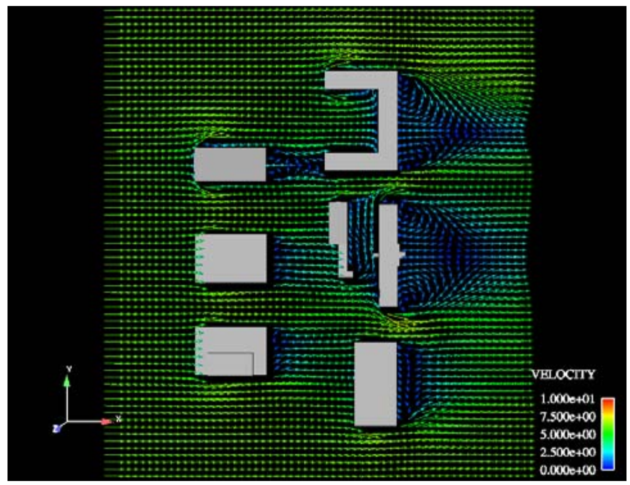
4. SIMULATIONS

The six case simulations were completed using a small 16 processor computing cluster. Each case took 48 hours wall clock time to finish using 16 processors. Recently, access to a larger computing cluster system has demonstrated performance on a similar type problem scaled well to 128 processors. Therefore, it seems practical that the present example case studies each could be rerun within 8 hours wall clock time. In any case, the quality of software and hardware is continuing to expand and become increasingly more practical to support fine-scale simulations within urbanized domains. In part a goal of this study is to determine and evaluate best methods for practical applications.

One critical feature of airflow in an urbanized domain is the separation and reattachment zones near building surfaces. The case studies were examined to identify these zones. These zones were small or nonexistent near the leading building faces and the building roofs. However, significant zones with recirculation were identified from the downwind (leeward) building faces. For example, Figure 7 shows the recirculation zone on a vertical slice of wind vectors. The horizontal extent of the recirculation zones on the leeward side of all the buildings can be identified by examining a horizontal plane as show for example by Figure 8. The white dots in Figure 8 identify the location of the wind sensors (see also Figure 1).



(a) 300 degrees



(b) 270 degrees

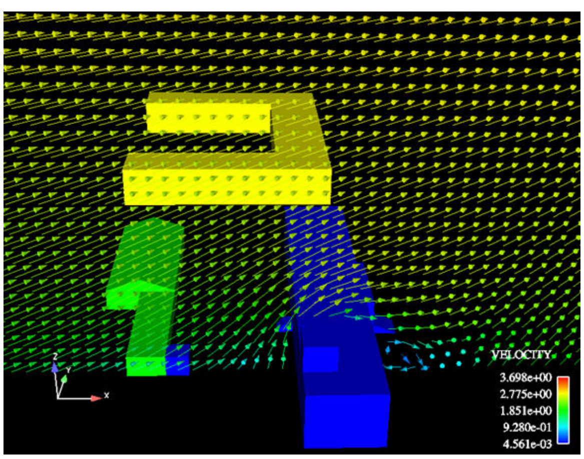
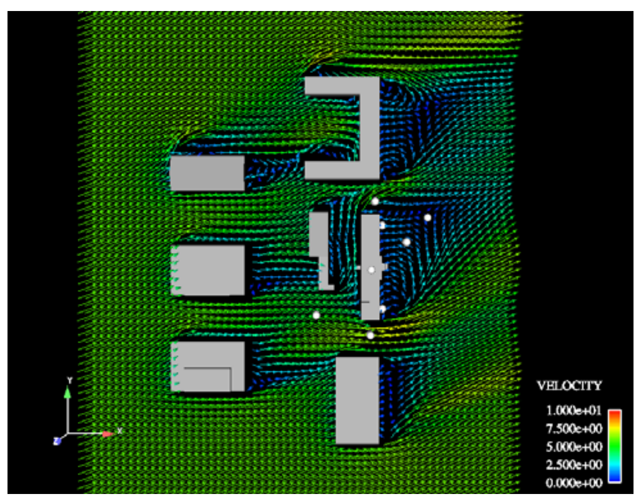


Fig. 7. Example steady-state velocity vectors on a vertical plane passing through the center of the main building for angle = 270 degrees (scale 0 to 3.6 m/s).



(c) 250 degrees

Fig. 8. Steady-state velocity vectors for angle= 250, 270 and 300 degrees (scale 0 to 10 m/s).

5. RESULTS

In that only one inlet wind speed profile was necessary, the single wind speed profile was developed to represent the wind profile upwind of the building cluster and to match the normalized measurements from the Reference Tower.

The steady-state, constant direction CFD model simulations were run for a constant wind speed. For comparison to the time series wind measurements, CFD simulation wind velocity for any specific location, VCFD(x, y, z), was normalized by its value at the 10 m level (VCFDREF10) of the Reference Tower. Then comparisons between CFD velocity and the field measurements were made by taking the location specific ratios, VCFD(x, y, z)/ VCFDREF10, and multiplying by the Reference SW Tower's 10 m measured value. This can produce a time series of CFD wind speed and direction that is comparable with the field measurement.

The discrete CFD model values for wind direction and wind speed have been plotted and compared with identically scale plots of the 1-minute averaged wind measurements. Figure 9 presents comparisons for wind direction. Figure 10 presents comparisons for wind speed. Winds for the S Tower are not shown but can be simply described as mating the SW Tower but slightly increased in magnitude likely due to flow convergence through the passageway between its neighboring buildings.

Results for Wind Direction

For the NE Tower, winds match for elevation (10 m) above building roof height while for the lower elevations (2.5 m, 5 m) the winds are influenced by the building induced circulations. The N Tower is between two neighboring buildings and upwind of the building induced circulation zones. There are slight affects on the wind direction. The Roof wind directions are only slightly affected. While the effects at the N Tower and Roof are slight the trends and differences among the elevations are similar for both model and measurements. Each of the Tripods has distinctly different. For Reattach the more Southerly winds have a greater influence due to a more extended circulation pattern in the main building wake flow. Both corner Tripods show that winds near the leeward building face flow outward from the center zone toward each building corner. Overall the model and measurements are on average very comparable. There are naturally more random variations of wind direction among measurements in the circulation zone than for the CFD model.

Results for Wind Speed

The wind speed is significantly reduced in all building zones below the roof level and is accelerated for the one location above the roof. The model and measurements for wind speed are comparable. Modeling the wind speed within the whole domain using only the measured 10-m wind from the SW Tower should be considered a difficult challenge that has been met using the methods developed for this study. There are naturally more random variations of wind speed among measurements in the circulation zone than for the CFD model.

6. SUMMARY

Results from the present model study demonstrate a method using steady-state CFD model simulations to model real time varying winds. This was a "blind test" in that the modelers only saw the measured winds for the SW Tower 10-m level. In general, the size and strength of the average circulation zones appear to be well modeled. There are naturally more random variations of wind direction and wind speed among measurements in the circulation zone than for the CFD model.

In the future there is an interest for directly simulating time-varying winds using unsteady RANS and fine-scale "large eddy simulation" CFD methods to examine fluctuations that cannot be simulated using the steady-state methods as applied in the present study. There is an interest in extending both the present steady-state methods and future direct time-varying simulation to model pollutant transport and dispersion within building clusters.

There are applications where the generally good performance demonstrated by the present study methods can be used either as a prognostic tool for determining winds within a cluster of buildings or as a source of data to support the development of empirical-based faster response models.

Acknowledgements – The research presented here performed by Alan Huber was performed under the Memorandum of Understanding between the U.S. Environmental Protection Agency (EPA) and the U.S. Department of Commerce's National Oceanic and Atmospheric Administration (NOAA) and under agreement number DW13921548. The modeling was completed with base support from the US EPA's Environmental Modeling and Visualization Laboratory.

REFERENCES

Cionco, Ronald M., Gail T. Vaucher, Sean D'Arcy, and Manuel Bustillos, 2006: Near-building turbulent intensities, fluxes and vortices. Proceedings of the Sixth Symposium of the Urban Environment, Atlanta, GA, Jan 30 – Feb 2, 2006. http://ams.confex.com/ams/Annual2006/techprogram

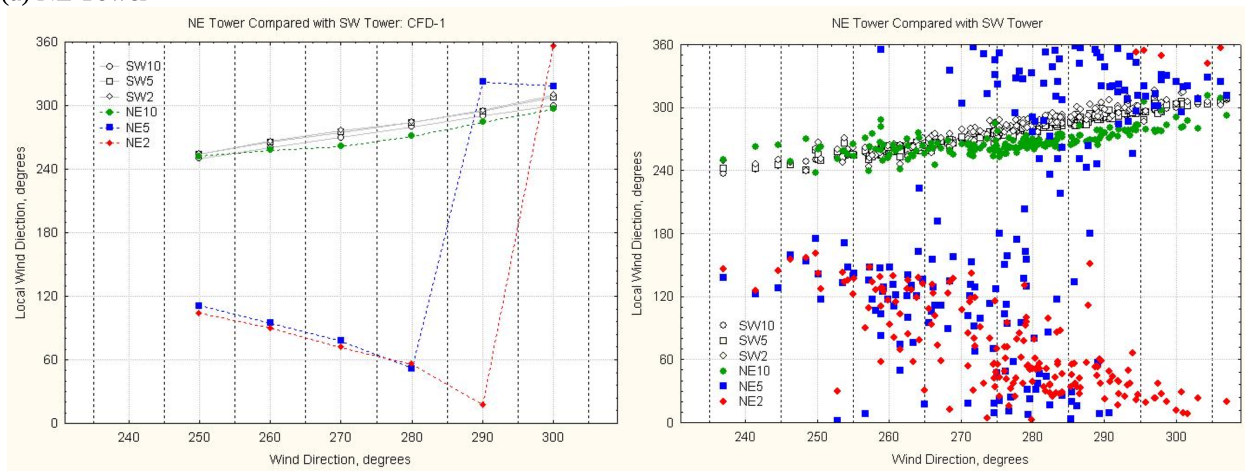
Fluent, Inc, 2006: FLUENT 6 User's Guide. Fluent Inc, Lebanon, NH.

Huber, A.H., Tang, W., Flowe, A., Bell, B., Kuehlert, K., and Schwarz, W., 2004: Development and applications of CFD simulations in support of air quality studies involving buildings. 13th Joint Conference on the Applications of Air Pollution Meteorology with the Air & Waste Management Association, (CD-ROM) Vancouver, British Columbia, Canada, August 23-27, 2004.

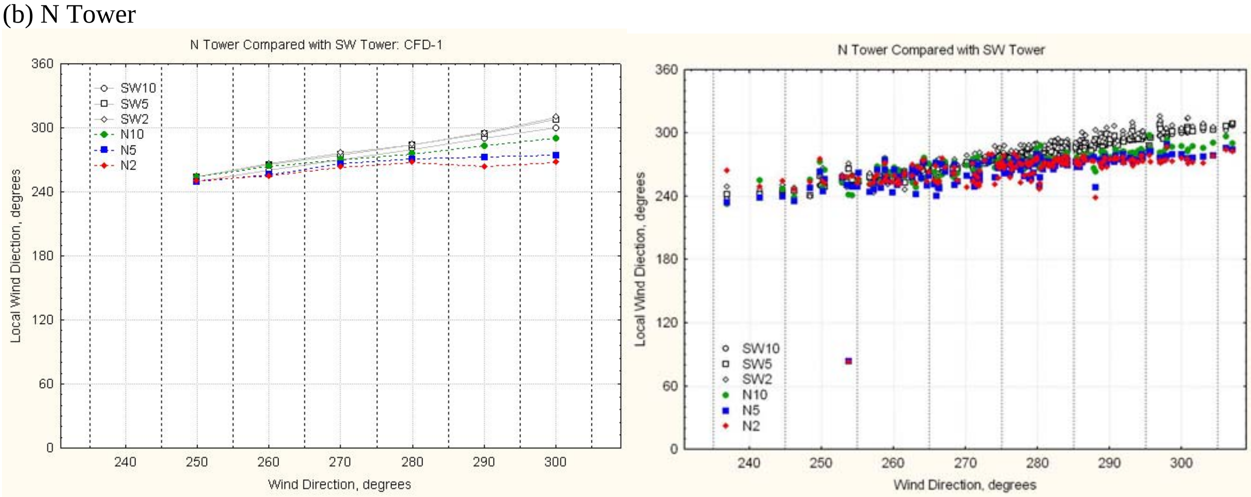
Huber A., M. Freeman, R. Spencer, W. Schwarz, B. Bell, and K. Kuehlert, 2006. Development and Application of CFD Simulations Supporting Urban Air Quality and Homeland Security. Proceedings of the Sixth Symposium on the Urban Environment, Atlanta, GA, Jan 30 – Feb 2, 2006. http://ams.confex.com/ams/Annual2006/techprogram

Tang, W., A. Huber, B. Bell, W. Schwarz, and K. Kuehlert, 2006. Application of CFD Simulations for Short-Range Dispersion over the Open Fields of Project Prairie Grass: Neutral Atmospheric Boundary Layer (in Journal review)

(a) NE Tower



(b) N Tower



(c) Roof

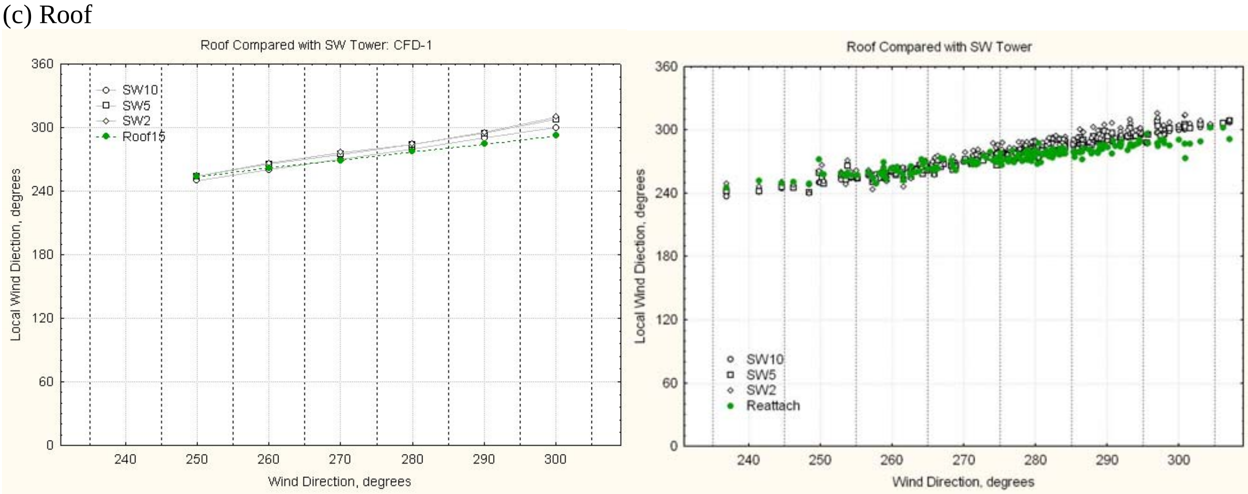


Fig. 9. Left Side: CFD simulated wind directions versus fixed inlet boundary wind direction.
Right Side: Wind direction of measured 1-minute smoothed wind speed versus the10-m SW Tower wind direction.
(Continued on next page)

(d) Tripods

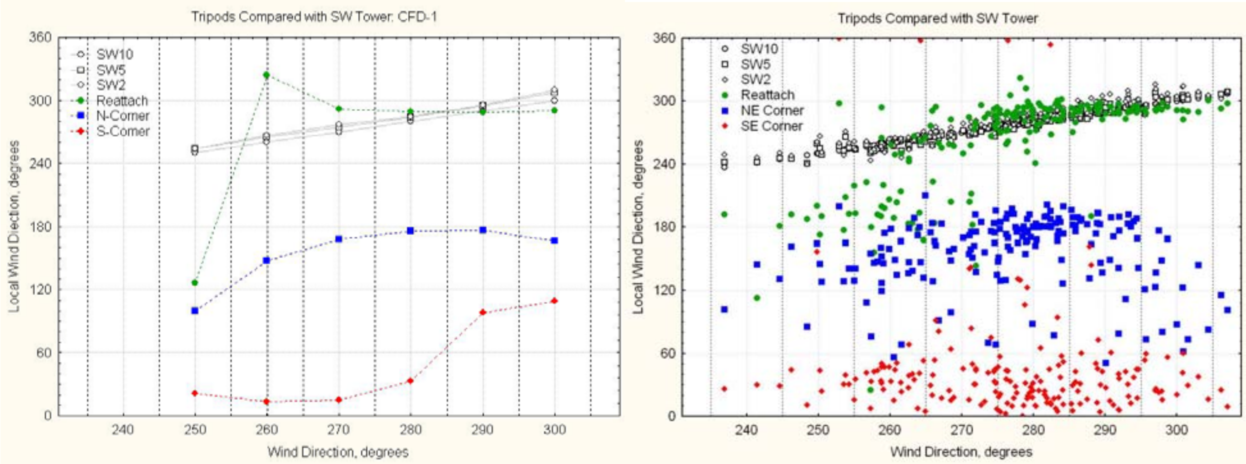
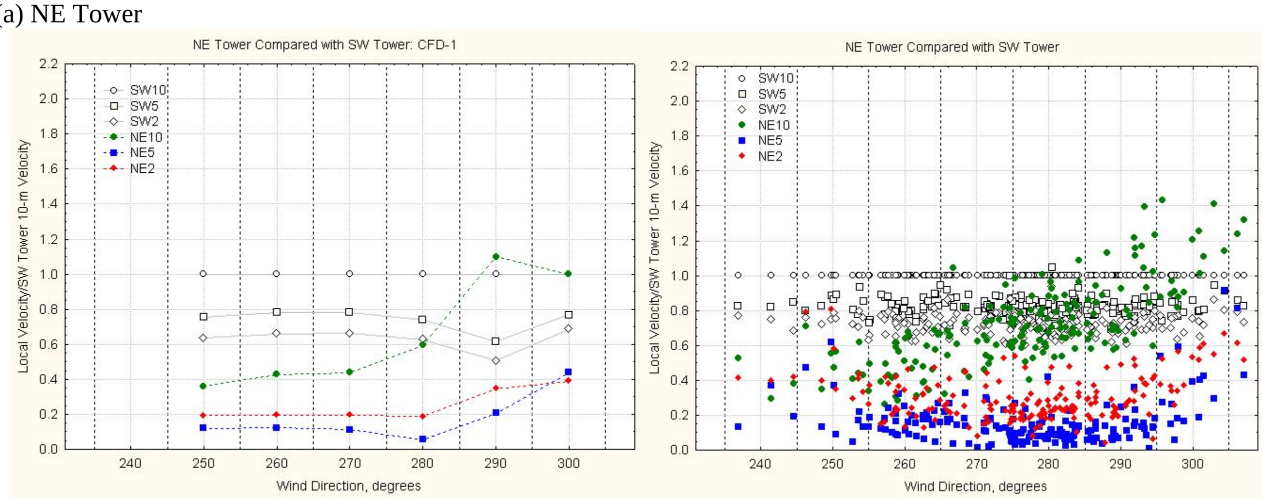


Fig. 9. Left Side: CFD simulated wind directions versus fixed inlet boundary wind direction.
Right Side: Wind direction of measured 1-minute smoothed wind speed versus the10-m SW Tower wind direction.

(a) NE Tower



(b) N Tower

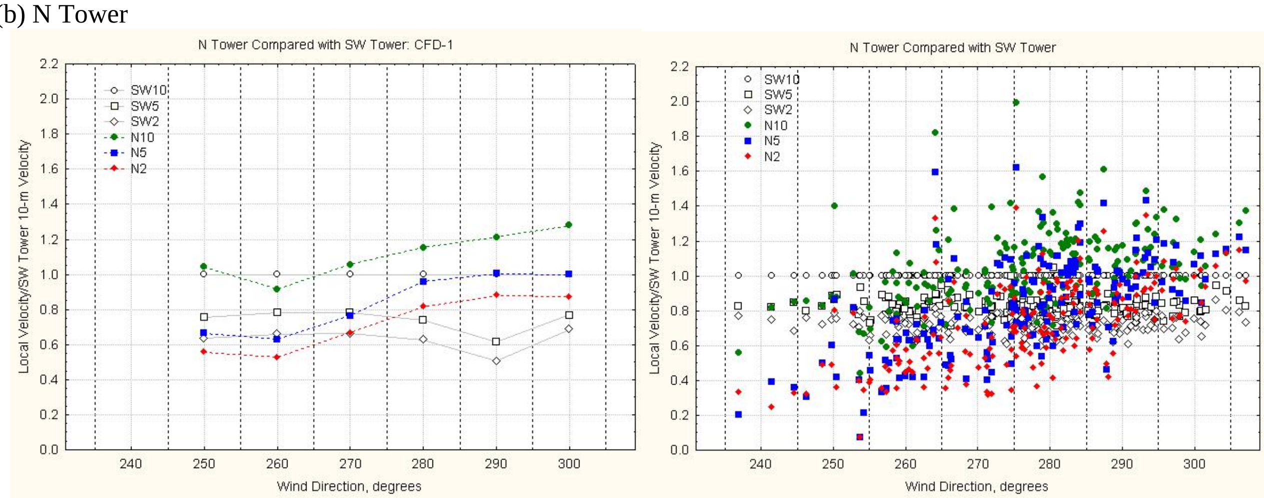
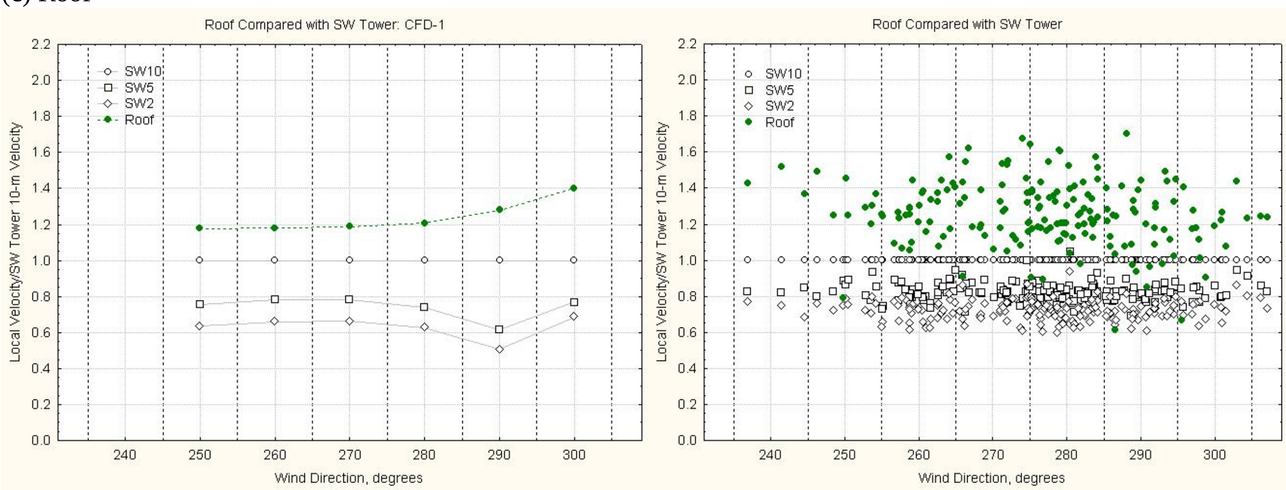


Fig. 10. Left Side: Ratio of CFD simulated wind speed to 10-m SW Tower wind speed versus fixed boundary inlet wind direction. . Right Side: Ratio of measured 1-minute smoothed wind speed to 10-m SW Tower wind speed versus the 10-m SW Tower wind direction.

(Continued on next page)

(c) Roof



(d) Tripods

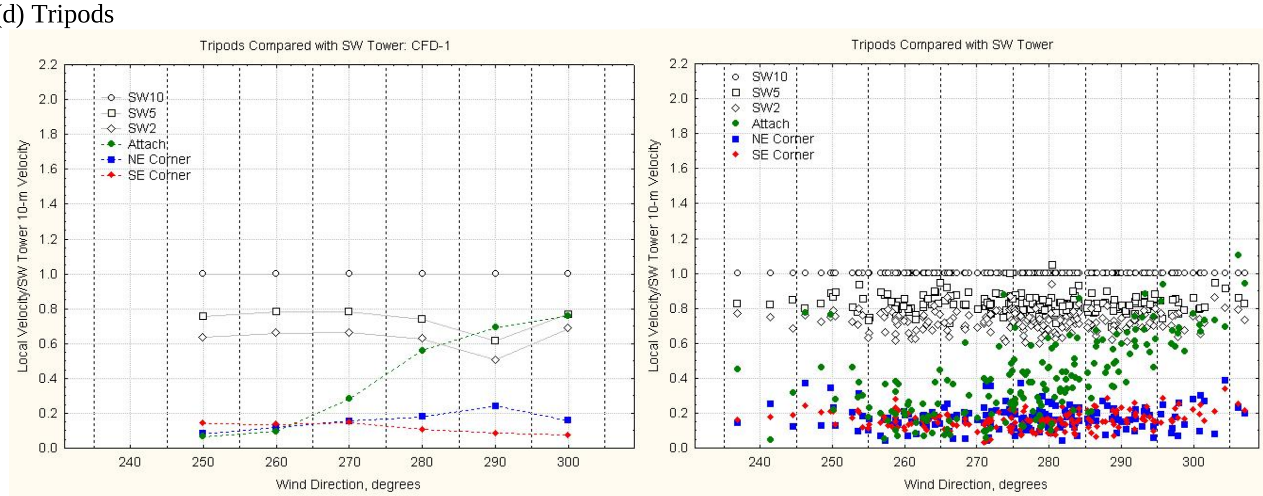


Fig. 10. Left Side: Ratio of CFD simulated wind speed to 10-m SW Tower wind speed versus fixed boundary inlet wind direction. . Right Side: Ratio of measured 1-minute smoothed wind speed to 10-m SW Tower wind speed versus the 10-m SW Tower wind direction.