Evaluation of FLUENT for predicting concentrations on buildings

David Banks[1], Robert N. Meroney[2], Ronald L. Petersen[1] and John J. Carter[1].

1. Cermak Peterka Petersen, Inc., 1415 Blue Spruce Drive, Fort Collins, CO 80524
2. Colorado State University, Fluid Mechanics and Wind Engineering Program, Department of Civil Engineering, Fort Collins, CO 80523

Paper # 70223

ABSTRACT

Predictions of concentrations on a simple rectangular building due to exhaust from a short stack on the buildings roof are obtained using flow simulations performed by FLUENT, a Computational Fluid Dynamics (CFD) computer code. These CFD predictions are compared to a database of wind tunnel data for various wind speeds and exhaust parameters. Separate CFD simulations were simultaneously performed at Cermak Peterka Petersen (CPP) and at Colorado State University (CSU) so that the influence of operator decisions could be investigated. While generally capturing trends in the plume dispersion qualitatively, the CFD predicted concentrations were sometimes incorrect by orders of magnitude.

INTRODUCTION

At the most recent worldwide Computational Wind Engineering (CWE) conference, keynote speaker Ted Stathopoulos summarized the ability of CFD modelling to supplant wind tunnel testing in wind engineering as follows[1]:

> “ In spite of some interesting and visually impressive results produced with CWE, the numerical wind tunnel is still virtual rather than real. Its potential however, is extremely high and its progress should be monitored carefully. Many more parallel studies – numerical and experimental – will be necessary in order to increase the present level of confidence in the computational results. Practitioners should be warned about the uncertainties of the numerical wind tunnel results and urged to exercise caution in their utilization.”

It is perhaps the tremendous potential alluded to above that has lead some CFD practitioners to present results of such numerical studies of flows over complex external building complexes in trade journals[2] and other promotional material[3]. Yet validation examples which would bolster the credibility of such studies, necessary in light of the caution advises above, are seldom provided. Without evidence that a CFD technique can reproduce experimental results in a similar situation, it is hard to have much confidence in the results. Furthermore, it is difficult to detect an error in a CFD prediction unless the true answer is known, since the CFD solution will always be internally consistent. A properly converged solution will not present any glaring inconsistencies (such as those evident when experimental equipment fails to operate properly), even if the flow patterns it provides are incorrect.

It has been suggested that a CFD code intended for use in the analysis and design of a microenvironment such as the flow around buildings demonstrate its ability to reproduce results for a set of base case configurations[4]. This paper presents an initial effort in this direction by simulating the dispersion of pollutants from a small stack on the roof of a simple rectangular building. An extensive series of wind tunnel experiments were conducted on this model[5], and a few selected cases for a single wind direction are examined in this paper. Note that while the original study presented the results as full-scale values, these CFD runs are intended to simulate the wind tunnel tests. This is not a comparison of wind tunnel and CFD with full scale, but rather an attempt to have the CFD code reproduce the wind tunnel results.

Demonstrating that a code can reproduce the correct concentration measurements is only half of the battle, however. As several studies have shown, the decisions made by the CFD code operator (as to the mesh, the numerical scheme, the boundary conditions, the turbulence model, to name some crucial issues) can significantly influence the outcome of the simulation[6]. A full validation involves the coupled ability of the user and the CFD code to conduct the simulation[7]. This requires a certain familiarity on the part of the operator with the situation being modeled, since it is not uncommon to achieve the correct results (in this case, accurate concentrations) for the wrong reasons. For example, a crude CFD solution (with the large numerical errors and a coarse grid) may provide better agreement with the data than a refined CFD solution in a certain situation[8]. It would be unwise, however, to depend upon such simulation inaccuracies to achieve good results in a different simulation situation.

The investigate the influence of operator decisions, Robert Meroney and David Banks did the modelling separately, both using FLUENT. Neither operator had access to the results until after the simulations have been completed.

WIND-TUNNEL DATABASE

A single 1 ft tall, 2 ft. wide, 1 ft long rectangular building was placed in the CPP environmental boundary layer wind tunnel test section, which is 12 ft wide and 8 ft tall. The upstream fetch, roughly 50 ft in length, was covered in surface roughness to allow a representative simulation of the atmospheric boundary layer to develop[9]. The resulting velocity and turbulence profiles are depicted in Figure 1, and correspond to a suburban exposure.

Tracer gas was released from stacks with inside diameters varying between 11 and 17 mm. Flow velocities from the stacks varied from 0.4 m/s to nearly 6 m/s. A range of stack heights between 0 and 3.6 inches was tested for three different wind directions: 0°, 45°, and 90°. The velocity at the reference height of 1m was 2 m/s in all cases. The effects of different sizes of architectural screens surrounding the stacks were also investigated. The wind tunnel experiments are described in detail by Carter[5].

For this stage of the data base validation process, only the a few of the no-screen cases for the 90° wind direction have been compared with CFD results. Table 1 summarizes the flow rates and stack heights simulated.

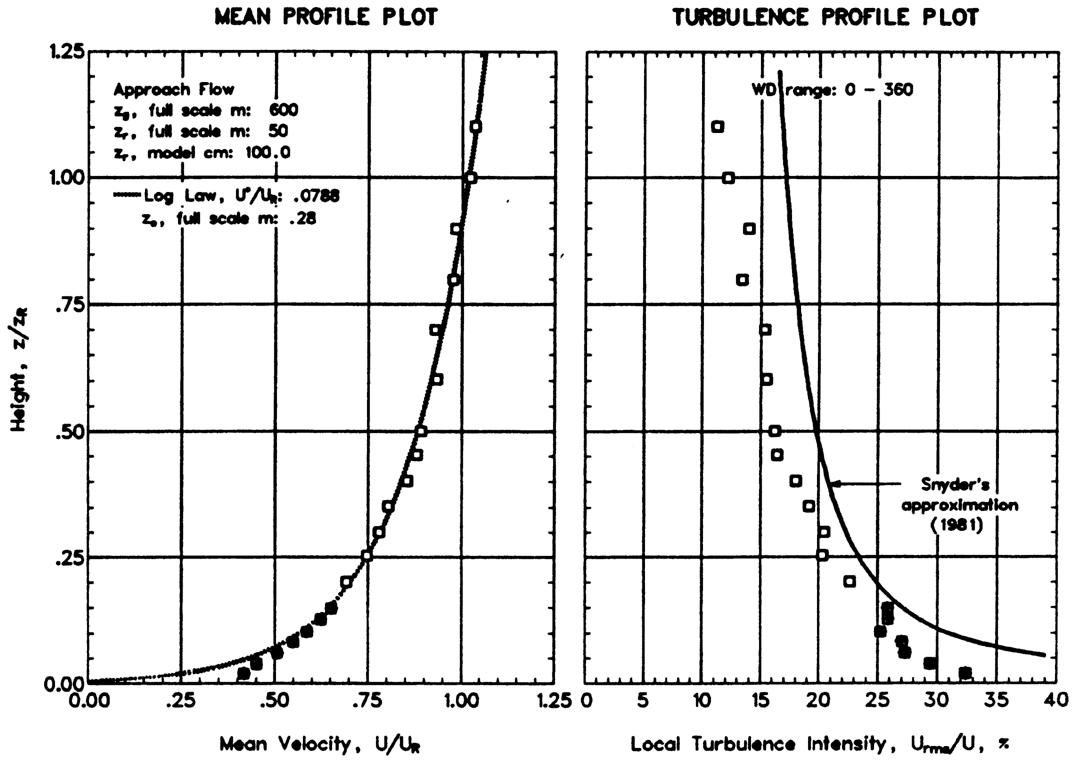


Figure 1: Velocity and turbulence intensity profiles at test section for wind tunnel tests. Z_R = 1m.

	Model scale			Corresponding full scale information at 1:50		
Case Number	Stack Height (mm)	Flow Velocity (m/s)	Stack Inside Diameter (mm)	Stack Height (ft)	Flow velocity at Stack height (mph)	Q: volume flow rate (cfm)
815	30	1.32	7.14	5	5.5	500
819	90	1.32	7.14	15	5.5	500
823	30	5.37	11.1	5	5.5	5000
827	90	5.37	11.1	15	5.5	5000
839	30	0.44	7.14	5	16.5	500
841	60	0.44	7.14	10	16.5	500
847	90	1.79	11.1	5	16.5	5000

Table 1: Wind tunnel test cases compared to CFD simulations in this report

CFD SETUP

Perhaps the most vexing problem currently facing the CWE practitioner is the selection of the turbulence model. Large Eddy Simulation (LES) is generally acknowledged to provide the most accurate answers, but requires considerably more computational resources than the various Reynolds Averaged Navier Stokes schemes[10]. The most common RANS turbulence model in use today, the k-ε model, is known to seriously miscalculate the nature of separated flow, and as a result, modifications to the basic k-ε scheme have been attempted[11]. To date, all tests in this validation exercise have been performed using a RANS turbulence model developed by Wilcox[12], the standard k-ω model, as implemented in FLUENT.

In order to create a suitably turbulent boundary layer near the 1:50 scale model, the wind tunnel experiments included large roughness elements in the test sections. Both the CSU simulations and the CPP simulations decided independently to include these elements, as shown in Figure 2.

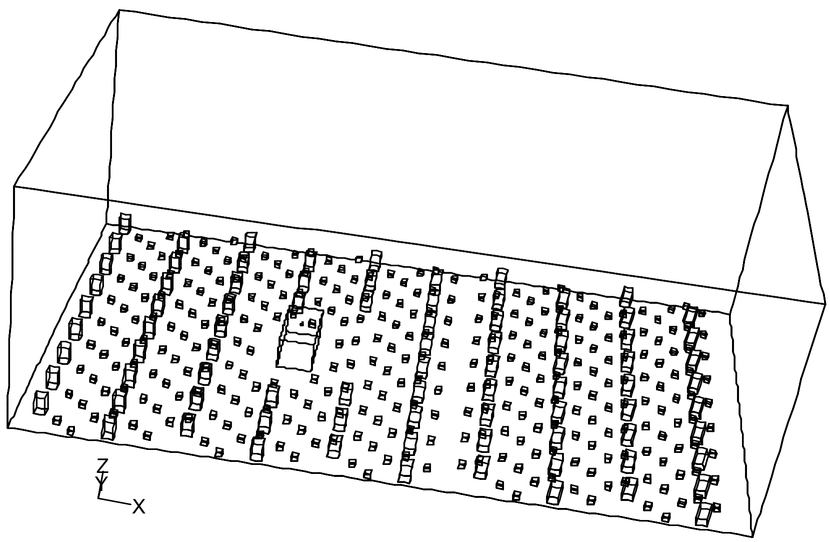


Figure 2a: Computational domain for CPP CFD simulation. Flow passes from left to right. Side walls and ceiling are solid boundaries.

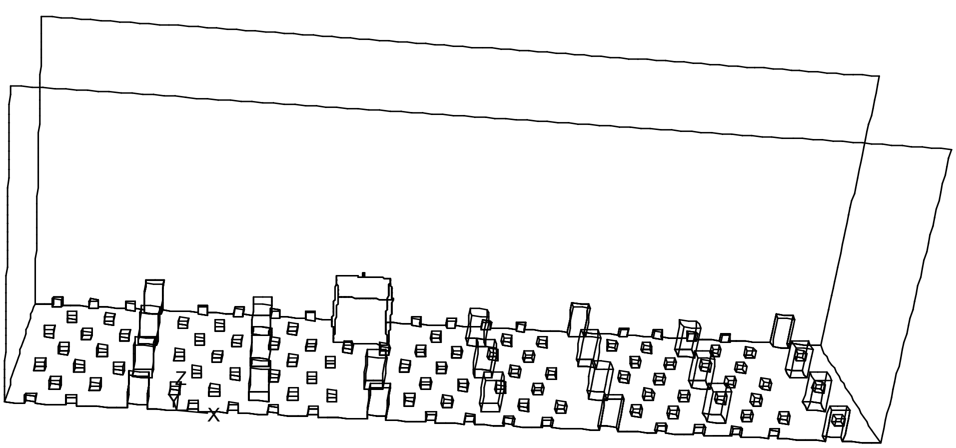


Figure 2b: Computational domain for CSU CFD simulation. Flow passes from left to right. Side walls and ceiling are mirror or symmetry boundaries.

The computational domain for the CSU simulations is significantly smaller. This was achieved through the use of symmetry boundary conditions along the sides and top, including a symmetry plane passing down the middle of the tunnel and through the building and stack. As both simulations were limited to fewer than 10^6 cells by computational resources, this permitted a much greater concentration of cells near the building. The CSU simulation features 4600 cells on the half building; the CPP simulation features 3400 on the entire building. Both simulations made use of 500,000 – 600,000 cells in total.

The CPP simulation imposes the profiles of Figure 1 on the test section inlet. However, this turbulence profile is not in equilibrium with the roughness present in the simulated test section. This indicates that the CFD model is not accurately portraying the manner in which the roughness generates and dissipates turbulence, since this roughness does match that used in the wind tunnel tests. The reasons for this are not clear at this time. It is possible that the coarse grids used around the roughness elements do not provide adequate resolution for proper turbulence development near the floor. This problem has also been reported in other validation studies[13].

One solution is to develop the inlet boundary conditions by separately simulating the flow over a similar upstream fetch of roughness, and this was done for the CSU solutions. The results are shown in Figure 3. While this process achieves velocity and turbulence profiles which are in equilibrium with the test section, the turbulence intensity drops off too rapidly with distance above the floor.

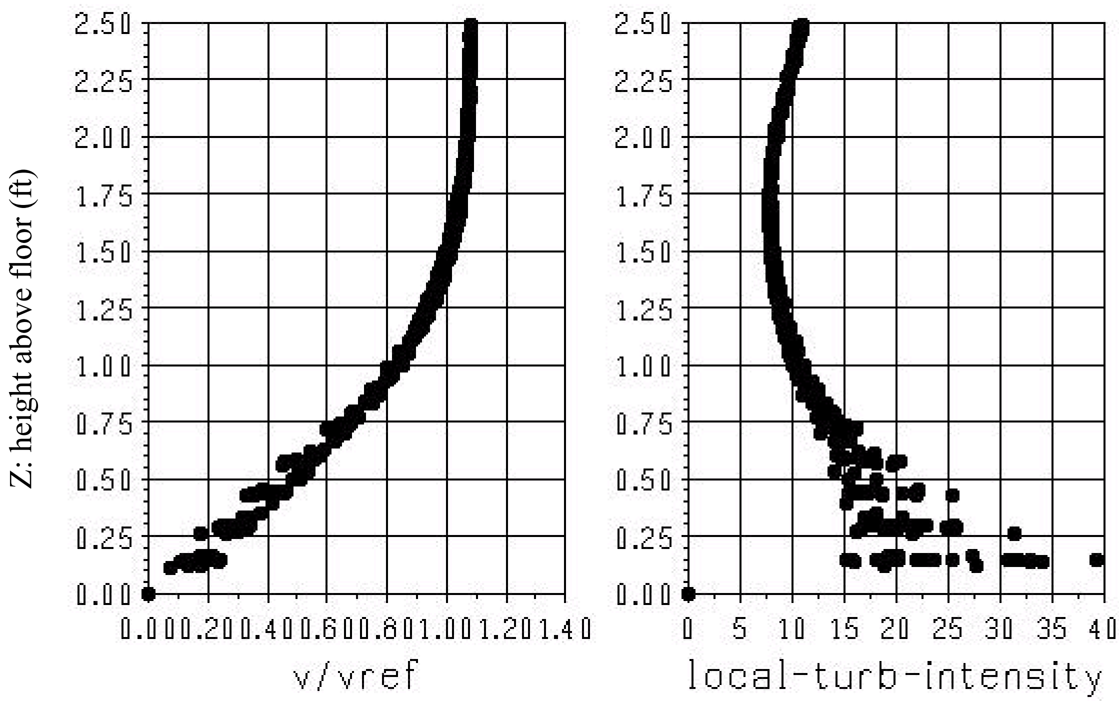
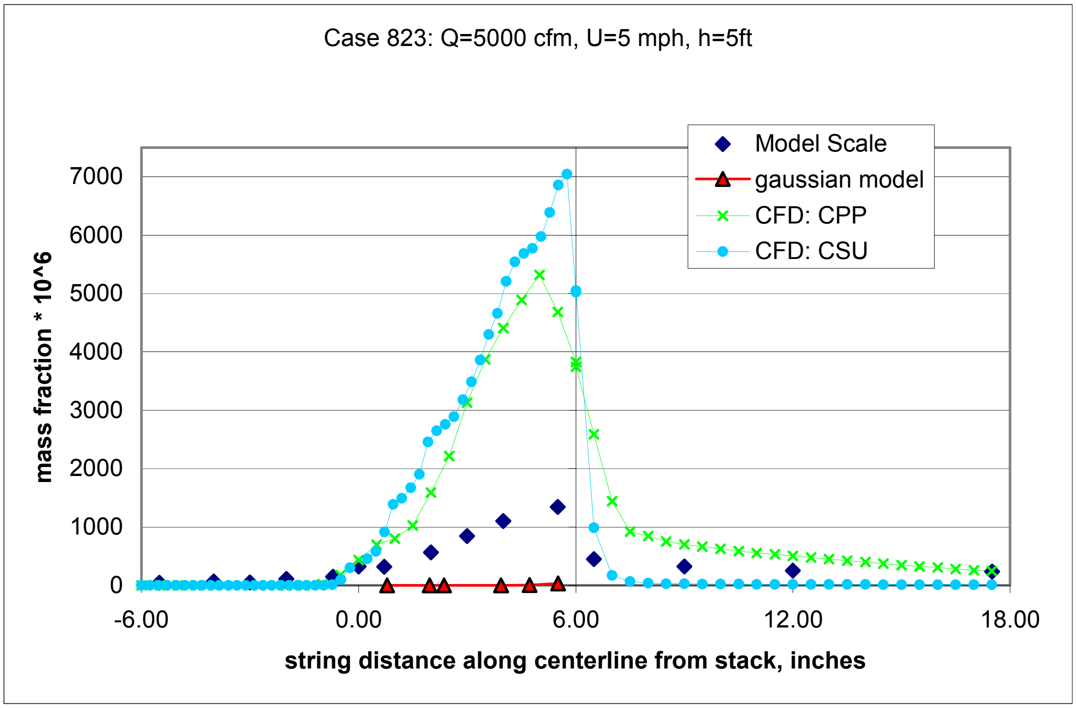


Figure 3: CSU velocity inlet boundary conditions generated using a 50-foot long tunnel with roughness elements and trip matching the experimental configuration. Local turbulence intensity is based on the local velocity. The velocity profile is normalized by V_{ref} = 2m/s.

RESULTS

A comparison between the wind tunnel data and the two simulations is provided in Figure 4.



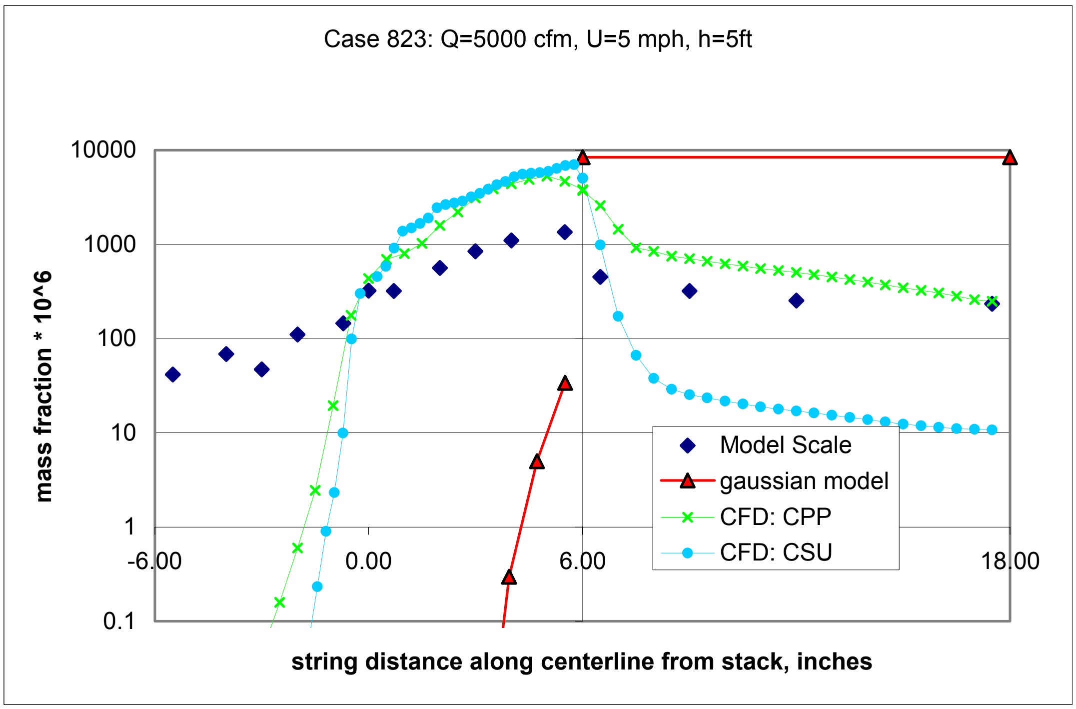


Figure 4: Concentrations along building centerline for case 823.

The data are taken along the centerline of the building. The point 0.00 is the location of the stack. The front or leading edge of the roof is at –6 inches, and the corner of the roof and rear wall is at +6 inches. The base of the rear wall is at 18 inches from the stack (string distance).

The CPP solution matches the data slightly more closely than the CSU solution, in spite of the use of a coarser grid and less carefully constructed inlet boundary conditions. A logarithmic contour plot of mass fraction is shown in Figure 5 for the CSU simulation of case 823. Without experimental validation, there would be no way of knowing that this reasonable looking plume is producing concentrations 5 time too high on the roof and 10 times too low on the rear wall.

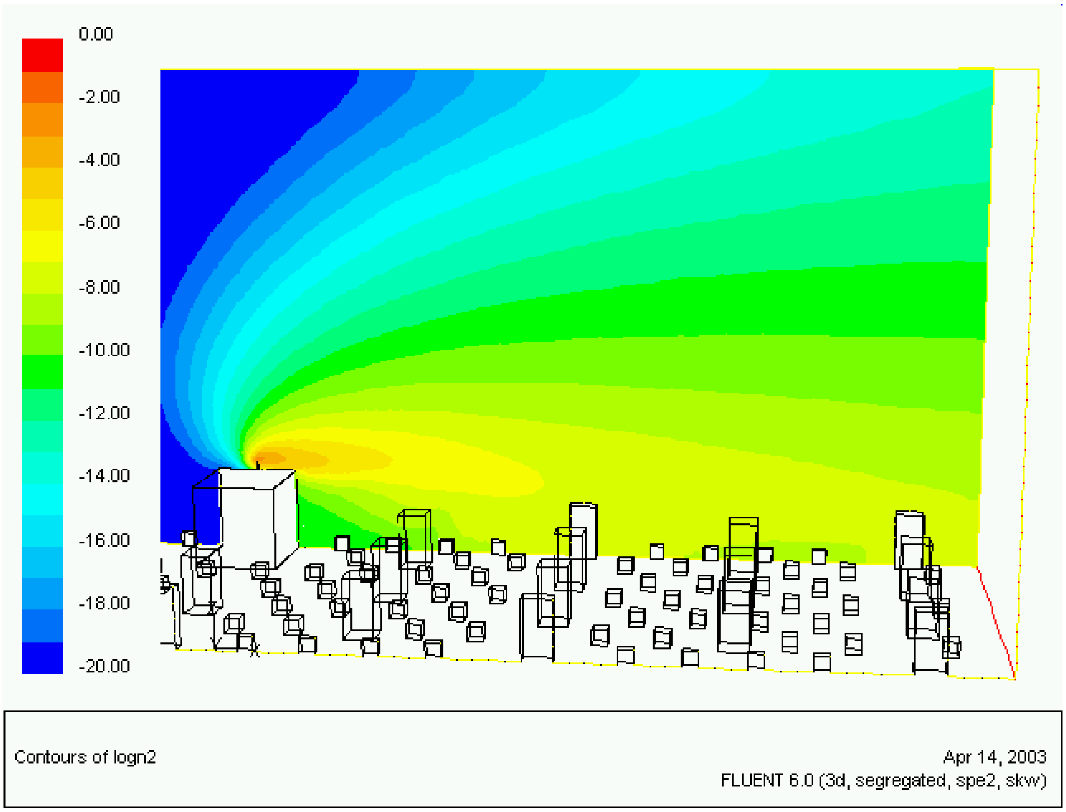
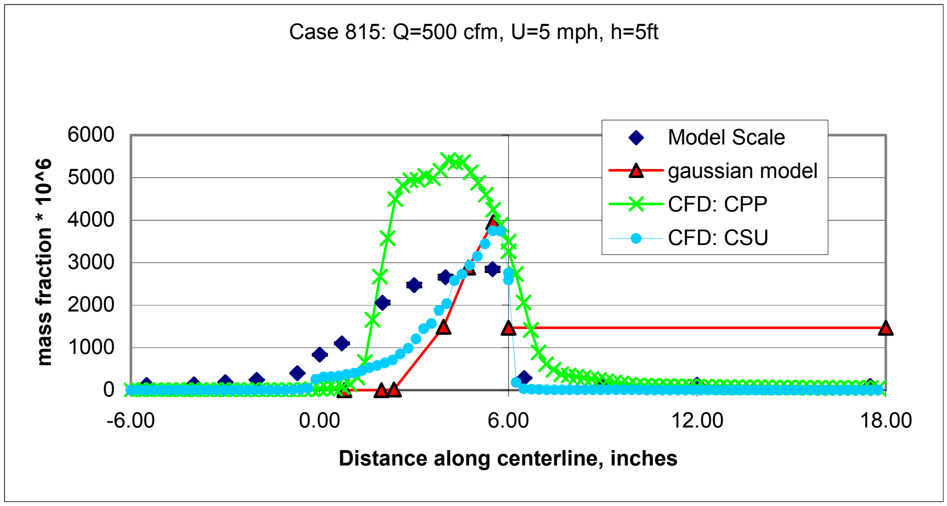


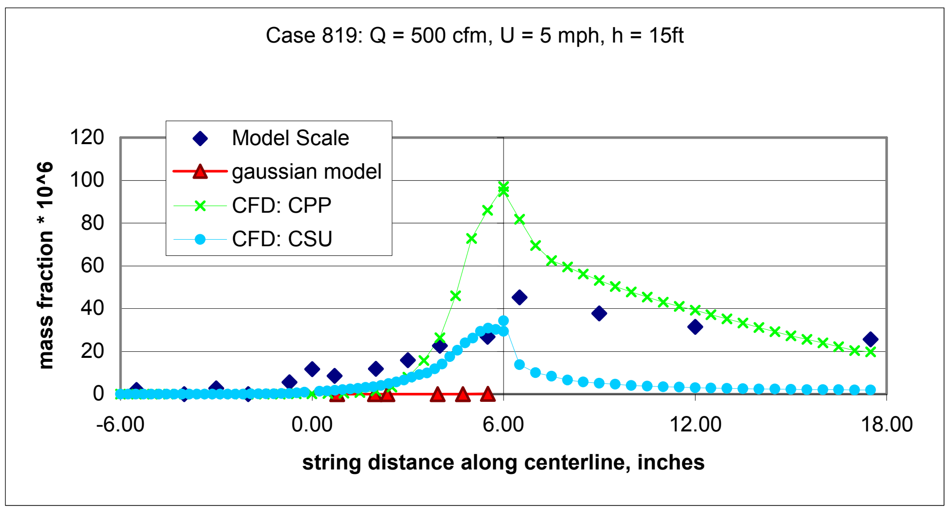
Figure 5: Contour plots of the natural log of the mass fraction for case 823, CSU simulation.

Both the CPP and CSU simulations overpredict the peak concentration downstream of the stack on the roof and near the top of the rear wall. This is true for any case in which there is a substantial mass fraction (> 0.1%) on the roof, as seen in Figure 6. The CFD simulation also fails to predict the presence of any stack gas on the upstream side of the roof. It should be noted, however, that for mass fractions below 10^{-4}, the wind tunnel instrumentation is approaching the limits of its resolution.

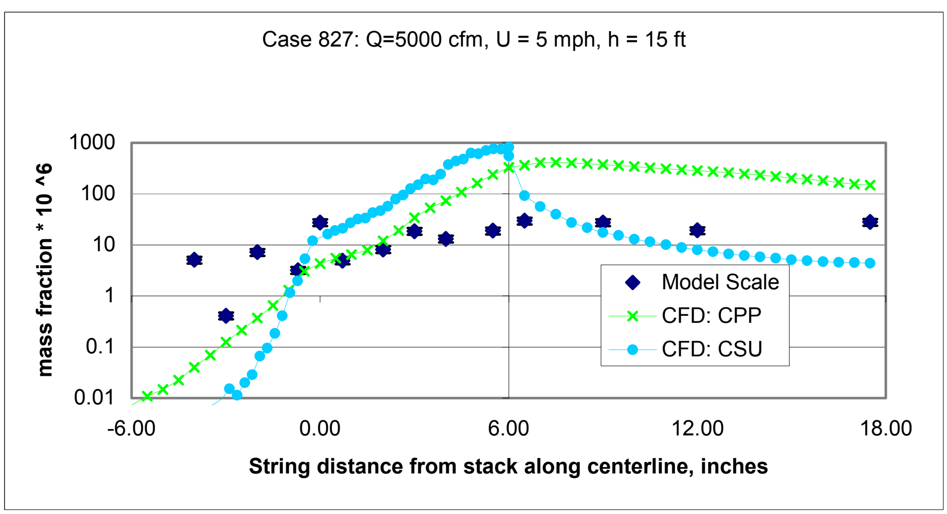
For comparison, Figures 4 and 6 also show concentrations estimated using the simple Gaussian plume model of SCREEN3. For the roof, this assumes that the roof is ground level. A single value is predicted for the recirculation region behind the building.



a)

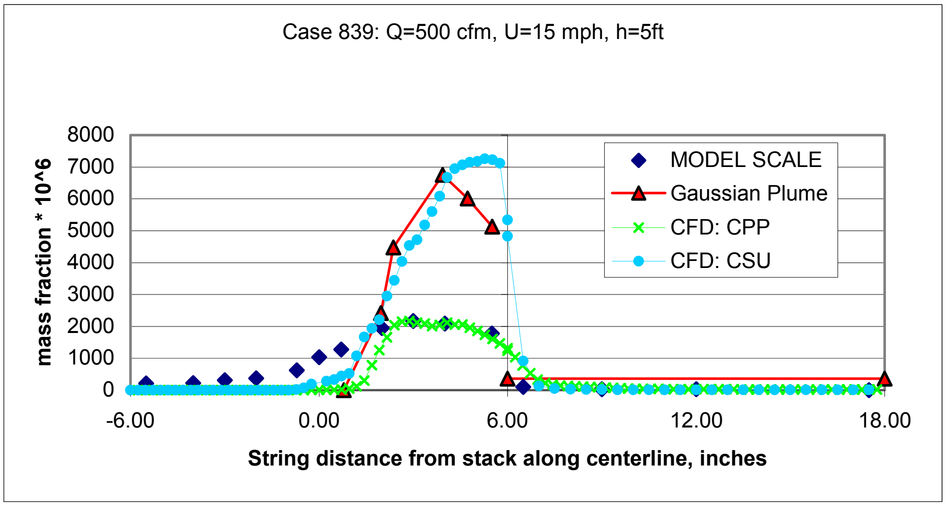


b)

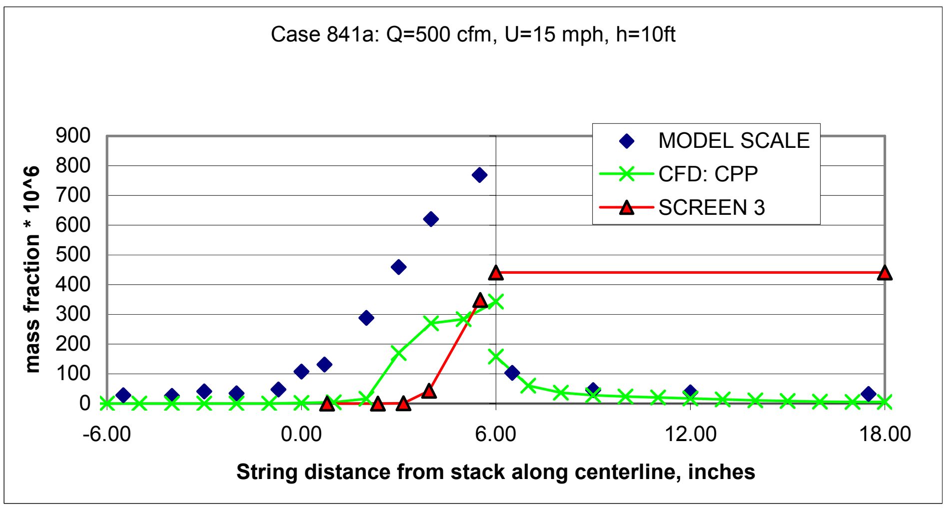


c)

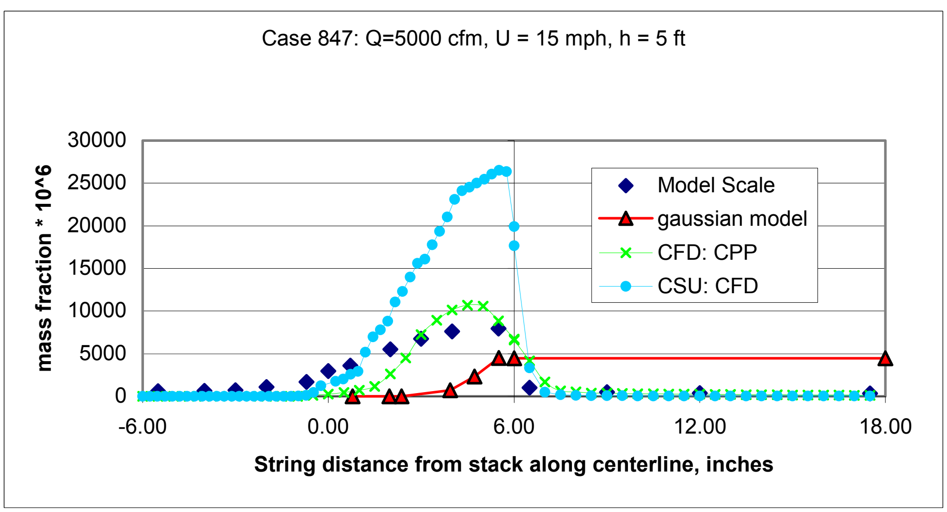
Figure 6: Plots of concentration vs. centerline position, comparing experimental results with CFD predictions.



d)



e)



f)

Figure 6: Plots of concentration vs. centerline position, comparing experimental results with CFD predictions. (continued)

CONCLUSIONS

In order for CFD simulations to be used as a design tool, the code and the manner in which the code is used need to be validated against experimental data for a similar situation. This process has been undertaken at CPP and CSU. At present, the simulations are not able to provide reliable estimates of concentrations on the building. This is not surprising, given then many previous investigations have found that concentrations in the vicinity of bluff bodies such as buildings are consistently overpredicted when using RANS turbulence models[14].

Future work

The comparison of CFD results to wind tunnel data for other wind directions and for locations downwind of the building is planned for the near future. These tests should include an assessment of the degree to which the present solutions are grid independent.

Tests using other RANS turbulence models are also planned. However, it is likely that if steady state solutions derived using RANS models are ever to provide accurate concentration measurements, it will be through the use of a quasi-steady weighting technique, such as that suggested by Quinn et al[15]. Large eddy simulation, being inherently unsteady, could circumvent the need for a quasi steady analysis, but the relative computational requirements need to be assessed.

REFERENCES

1 Stathopoulos, T. "The Numerical Wind Tunnel for Industrial Aerodynamics: Real or Virtual in the New Millenium?" *Wind and Structures* **2002***, 5*, 193-208.

2 Improving the Safety Standard on an Offshore Platform. In *Fluent News*, 2001; pp 10.

3 CAES Home Page. http://www.bckpc.com/caes.asp (accessed February 2003)

4 Huber, A. H.; Freeman, m.; Rida, S.; Kuerlert, K.; Bish, E. "Development and Applications of CFD Support of Air quality Studies of Roadway and Building Microenvironments." 94th A&WMA Annual conference: Orlando, FL, 2001.

5 Carter, J. J. M. Sc. Thesis, "The Influence of Architectural Screens on Exhaust Dilution." Colorado State University: Fort Collins, Colorado, 1997.

6 Castro, I. P.; Graham, J. M. R. "Numerical wind engineering: the way ahead?" Proc. Instn. Civ. Engrs Structs & Bldgs 1999, 134, 275-277.

7 Chen, Q.; Srebric, J. “How to verify, Validate, and Report Indoor Environment Modeling CFD Analyses”; ASHRAE RP-1133: Atlanta, Georgia, 2001

8 Hall, R. C. (Ed.) “Evaluation of Modelling Uncertainty: CFD Modelling of Near-Field Atmospheric Dispersion”; European Commission Directorate-General XII: Science, Research and Development: Surry, United Kingdom, 1997.

9 Simiu, E.; Scanlan, R. “Wind Effects on Structures”; 2nd ed.; John Wiley and Sons, Inc., 1986.

10 Murakami, S. “Current Status and Future Trends in Computational Wind Engineering.” *Journal of Wind Engineering and Industrial Aerodynamics* **1997***, 67-68*, 3-34.

11 Tsuchiya, M.; Murakami, S.; Mochida, A.; Kondo, K.; Ishida, Y. “Development of a new k-ε model for flow and pressure fields around bluff body.” *Journal of Wind Engineering and Industrial Aerodynamics* **1997***, 67&68*, 169-182.

12 Wilcox, D. C. *Basic Fluid Mechanics*; DCW industries: La Canada, California, 1997.

13 Richards, P. J.; Dunn, A. D.; Parker, S. “A 6m cube in an atmospheric boundary layer flow: Part 2: Computational solutions.” *Wind and Structures* **2002***, 5*, 177-192.

14 Meroney, R. N.; Leitl, B. M.; Rafailidis, S.; Schatzmann, M. “Wind-tunnel and Numerical Modeling of Flow and Dispersion about Several Building Shapes.” *International workshop on CFD for wind climate in cities*: Hayama, Japan, 1998.

15 Quinn, A. D.; Wilson, M.; Reynolds, A. M.; Couling, S. B.; Hoxey, R. P. Modelling the dispersion of a tracer gas in the wake of an isolated low-rise building. *Wind and Structures* **2001***, 4*, 31-44.

KEY WORDS

Dispersion Modeling, wind tunnel, model validation, CFD