REPRODUCING THE REAL PRESSURE COEFFICIENT USING A COMPUTATIONAL FLUID DYNAMICS PROGRAM HOW CLOSE IS *CLOSE ENOUGH*?

Prasasto Satwiko*, Nick Locke**, Michael Donn*
School of Architecture, Victoria University of Wellington, Wellington
**Opus International Consultants, Lower-Hutt, Wellington*

SUMMARY

This paper reports on the calibration of a commercial computational fluid dynamics program, CFD-ACE, using real measurements. The main purpose of the calibration was to develop an understanding of the modelling capabilities of CFD-ACE and its applicability to complex situations. A real building, the Texas Tech Experimental Building, was used as a case study. The calibration involved eight different scenarios, comprising combinations of two and three-dimensional modeling, standard k-ε and RNG k-ε turbulence models, and uniform or atmospheric boundary layer flows. The results compare pressure coefficient (C_p) data from the full-scale building, wind tunnel experimental results and computational fluid dynamics simulations. A general evaluation of the results found that there was no single ideal combination that can perfectly simulate the real phenomenon; the CFD-ACE program producing different results for each case. However, the combination of 3D modeling with a standard k-ε turbulence model and an atmospheric boundary layer appeared to give the best results. Comparison with the field data showed that the CFD-ACE program could reproduce the real conditions quite well. The deviation from the field data was much smaller than the reference CFD experiment and wind tunnel experimental results. The CFD-ACE program performed well as a tool for studying the wind flow around the Texas Tech Experimental Building.

INTRODUCTION

The aim of this experiment was to calibrate the CFD-ACE program against data from the Texas Tech Experimental Building[1,2]. The intention was to study the deviation of CFD-ACE results from the real conditions. By understanding the deviations, it was hoped that users could be more confident when using the CFD-ACE program to simulate other cases. In the process, a general question was raised about the match between real data and the predictions of CFD software. Given that there will be differences between any two measurements whether real or simulated, it can be expected that there will be differences between the CFD data and the Texas Tech data. The general question addressed in the experiment was what scale of difference between real and simulated data is acceptable. How close is *close enough*?

The answer to this question has far-reaching implication. CFD tools, like many other building environment simulation tools, are being used increasingly in consultancy. If consultants have the same difficulty in guaranteeing a match with reality then the buildings they construct to these CFD performance predictions will be seriously at risk of failure.

LIMITATIONS

Two limitations were imposed on the experiment:

- The calibration aimed to find the best performance of the CFD-ACE program in the hands of a person who understands fluid dynamics but not the equations used to represent it. No modification was made to the CFD codes.
- The experiment focused on the pressure coefficients of the windward and leeward faces of the Texas Tech. Experimental Building as there are many published research reports on this data readily available.

PROCEDURE

The experiment used the following procedure:

- Literature study. This included a study of basic architectural aerodynamics, basic computational fluid dynamics and reports on experiments using the Texas Tech. Experimental Building.
- Preliminary trial and error. This had two purposes: to adapt to the CFD program and to quickly approach its best performance. This included understanding the pre-processor (grid generator), main processor (flow solver) and post-processor (data presenter). Trial and error experiments combined recommendations found

in references (books, journals), the CFD-ACE manuals and discussion with experts.

- Selecting criteria.
- Main experiment. The main experiments were simulations run under different scenarios, using combinations of 2D/3D, standard k-ε/RNG k-ε turbulence models, and uniform/atmospheric boundary layer flows.
- Analysis.

REFERENCES

Many published reports discuss research projects that use field data from the Texas Tech Experimental Building. Some of them used the field data to verify wind tunnel experiments (eg, Surry[3], Cochran, et al.[4]) while others such as

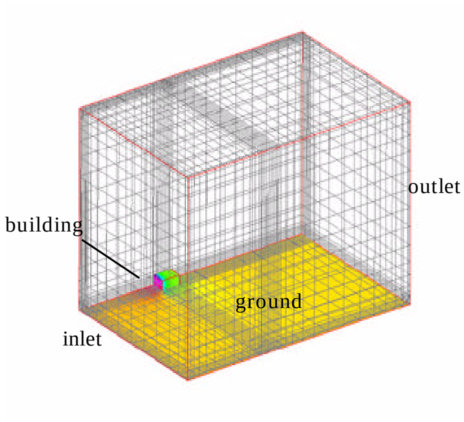


Figure 1 Three-dimensional grid

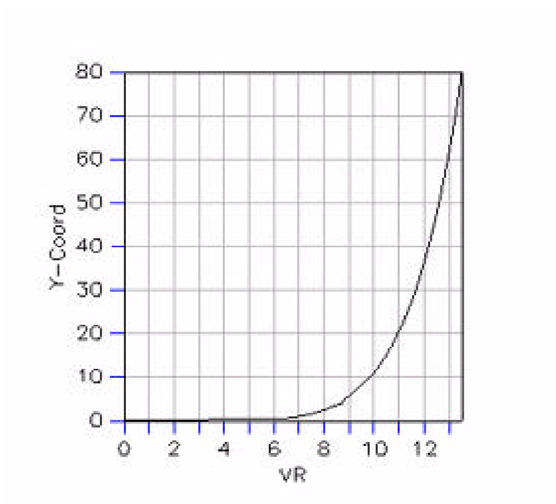


Figure 2 Atmospheric boundary layer wind profile

Selvam[5] used the field data to verify his wind tunnel and computational fluid dynamic simulation. This experiment used four sets of data for comparison: the field data, Selvam's wind tunnel, Selvam's computational fluid dynamics, and Opus Central Laboratories' wind tunnel (1:25 scale model). The field data (the actual conditions of the full-scale building) were used as the main reference. The other three references were used for comparison

PRELIMINARY TRIAL AND ERRORS

Preliminary experiments were focused on:

- How to design the grid
- How to achieve fast and good convergence
- How to approach the field data as close as possible

The preliminary study found that non-uniform grids were better than uniform grids. They allowed lower grid numbers without necessarily losing the important data because finer grid resolution could be allocated at critical places such as the surfaces of the building or where the flow changes most rapidly.

The main problem with using a non-uniform grid was the occurrence of bad cells caused by extreme aspect ratios, which may lead to convergence problems. According to CFD-ACE user support, the program can handle this problem and proved to

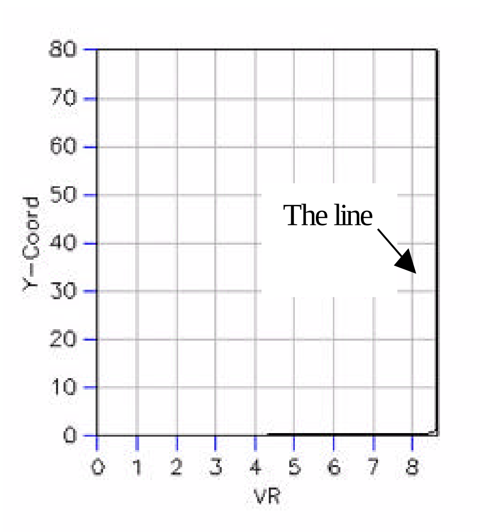


Figure 3 Uniform wind flow profile

be true, as experiments containing bad cells did not have any convergence difficulties. Non-uniform grids required less memory space and calculation time.

When the wind direction was perpendicular to the building (either to the long or short sides), the simulation could take advantage of the building symmetry. It was possible to model just one half of the building which saved memory and calculation time. Preliminary experiments showed that whole and half model simulations produced almost identical results. The half model simulation defined the cutting plane as a *symmetry* boundary condition.

To simulate the wind flow around the building a *virtual wind tunnel* was generated and the Texas Tech Building placed inside it. Unlike a real wind tunnel, which has limited dimensions, the virtual wind tunnel does not. It can be built to any size so that even a full-scale model can be put inside. The boundary of the flow domain can be defined in such a way that the wind can flow in and out, simulating the real world where there are no boundaries. This can be done by defining the boundary conditions either as outlets or as symmetry planes. Preliminary experiments found that the latter option was better. Defining the boundaries (north, low and high sides) as outlets resulted in convergence problems, especially when the boundaries were too close to the model and roughness was applied to the ground and building surfaces. Symmetry boundary conditions provided better boundary simulations when the flow domain was relatively large, as there was no wall friction and so allowed the wind to flow freely.

Li[6] recommends that the inlet and outlet should be positioned at least six and twenty times the building height, respectively, from the building. Preliminary experiments showed that this rule of thumb for the minimum outlet distance could be used for symmetry boundaries as well. The minimum inlet distance also worked well for 3D models with atmospheric boundary layer flows. For 2D models with uniform and atmospheric boundary layer flows the inlets should be located thirty and twenty times the building height, from the building, respectively. For 3D models with uniform flow, the inlet should be located twelve times the building height upstream. Placing the inlet closer or farther than that distance caused, respectively, an increase or decrease in the C_p magnitudes. In the main experiment, to reduce the number of variables, the inlets were located a distance of ten times the building height upstream of the building.

The following parameters were used:

- Building dimension: full scale, 9.1m wide x 13.7m long x 4.0m high.
- Incoming mean wind speed[7]: 8.6 m/s at eaves' height (4.0 m)
- Roughness height[8]: ground ~ 0.024, building surfaces ~ 0.0001
- Air properties: temperature= 300^{o}K, density= 1.177 kg/m^3, kinematic viscosity= 1.57x10^{-5} m^2/s
- Gravity: -9.81 m/s^2
- Atmospheric boundary layer is defined by the power law equation:[9]

$$V_h = V_{bl}.(h/h_{bl})^{"}$$

Where,
V_h = wind speed at the height of a given point, m/s
V_{bl} = gradient speed, the wind speed at the height of boundary layer, m/s
h = the height of the given point, m
h_{bl} = gradient height, the height of the boundary layer, m
" = mean speed exponent

Table 1 Values of gradient height, power law exponents and roughness height[1]

Terrain category	Terrain description	Gradient height, h_{bl} (m)	Roughness height (m)	Mean speed exponent, "
1	Open sea, ice, tundra, desert	250	0.001	0.11
2	Open country with low scrub or scattered tress	300	0.03	0.15
3	Suburban areas, small towns, well wooded areas	400	0.3	0.25
4	Numerous tall buildings, city centres, well developed industrial areas	500	3	0.36

- Turbulent kinetic energy, k = 2.22 J/kg and turbulent dissipation, ε = 8.27 J/kg.s

For Texas Tech., the longitudinal turbulence intensity (TI) is around 19% at four metres above the ground[1]. For a wind speed of 8.6 m/s the longitudinal fluctuating component has an rms value, u'= 0.19x8.6 = 1.634 m/s. As the atmospheric boundary layer turbulence is non-isotropic, the transverse and vertical fluctuating velocities (rms values) can be approximated by v'= 0.68u' and w'= 0.45u', respectively. Thus, v'= 1.111 m/s and w'= 0.735 m/s.

$$k = 0.5\,(u'^2 + v'^2 + w'^2)$$
$$k = 0.5\,(1.634^2 + 1.111^2 + 0.735^2)$$
$$= 2.22 \text{ J/kg}$$

ε can be calculated from the following formula[2] :

$$\varepsilon = k^{1.5}/8h$$

where k= turbulent kinetic energy, J/kg
8 = constant ~ 0.005
h = height of the enclosure, m

thus $\varepsilon = 2.22^{1.5}/(0.005\text{x}80) = 8.27$ J/kgs

CRITERIA

Criteria are needed to compare and evaluate the results of experiments. For this experiment, the criteria were selected based on the data available and their significance to building ventilation. This led to the selection of the pressure coefficient, C_p, as the parameter.

The pressure coefficient, C_p, is a common (and a more convenient) way of presenting pressure data[3,4] and is calculated using the following formula:

$$C_p = (P - P_o)/0.5.D.V^2$$

where P = local pressure, Pa
P_o= reference pressure, Pa
D = air density, kg/m^3
V = mean approaching wind speed at eaves height, m/s

The reference pressure (P_o) in free flow is considered to be zero[5]. P is the local static pressure and $0.5.D.V^2$ is the dynamic pressure at the reference point (which in this case is at eaves height in the undisturbed upstream flow).

Using the C_p data, five criteria were defined:

- Average windward C_p difference from the field data
- Average leeward C_p difference from the field data
- Windward maximum C_p difference from the field data
- Leeward maximum C_p difference from the field data
- Maximum inlet/outlet difference between CFD results and the field data.

The first two criteria were used to measure the CFD program'S ability to correctly simulate building surface pressures by averaging the C_p differences (between CFD simulations and the field data) at given points on the windward and leeward building faces. The second two criteria measured the differences (between CFD simulations and field data) of the absolute values of the maximum C_p on the windward and leeward building faces. The last criterion found the inlet/outlet C_p difference and how it differed from the field data; this criterion was useful for studying whether the C_p differences gave the same effect.

No guide is available as to what is an acceptable deviation. The simple rule is to get the smallest possible deviation or make the CFD results as close as possible to the field data. This leaves the question - how close should it be? An ideal result would be to obtain exactly the same values as the field data. Currently, this is unattainable as computational fluid dynamics codes are still being developed and fine-tuned. The terms such as *agree well* and *very good agreement* seem to be much more realistic and have been used by many authors in qualitatively explaining the degree of similarity. As a consequence, there has to be an accepted degree of tolerance.

The range of tolerance can be derived from Selvam's report. He states that his CFD experiment has good agreement with the field data. Another CFD researcher, Paterson, supports this[6]. There are deviations of up to 7% for the average difference in windward C_ps between Selvam's CFD simulations calculation and the full-scale data[7]. In a different case study, another CFD researcher, Shao, accepts 20% tolerance as *good agreement* between his CFD C_p results and the field data[8].

Expecting a perfect match between CFD results and full scale field data is not only unrealistic but can also be misleading. Real wind flows around and inside buildings are turbulent and fluctuate randomly with time. Measurements of these airflows are taken over a small period of time and are typically recorded using statistical parameters that are used to characterise the flow. CFD programs, on the other hand, usually calculate the flow based on steady state conditions and thus do not produce solutions, which *vary with time*. Thus, a small degree of difference between CFD results and the field data may be expected.

To determine how much deviation can be tolerated it is useful to look at the effect of changes in the Cp values when they are used in subsequent calculations. An example of this is the ventilation rate of a building that can be related to the pressure difference between openings in the building using the equation given below. Differences in the ventilation rate and its effect on indoor comfort can be used to determine an acceptable tolerance.

$$Q = C_d.A.V.(C_{pi} - C_{po})^{0.5}$$ where

Q = ventilation rate, m^3/s
C_d = discharge coefficient[9]
A = total opening area, m
C_{pi} = pressure coefficient at inlet
C_{po} = pressure coefficient at outlet
V = reference velocity (measured at the eaves height), m/s

Assuming, $A = 1$ m^2, $C_d = 1$, the areas of inlets and outlets are the same, and $V = 8.6$ m/s, then the real building has a ventilation rate,

$$Q = (1)(1)(8.6)(0.85)^{0.5}$$
$$= 7.93 \text{ m}^3/\text{s } (28548 \text{ m}^3/\text{h})$$

Cp values from Selvam's wind tunnel experiment, Selvam's CFD simulation and the Opus wind tunnel experiment produced ventilation rates of 7.6

m^3/s (27360 m^3/h), 7.4 m^3/s (26640 m^3/h) and 6.34 m^3/s (22824 m^3/h), respectively. The closest value to full scale is Selvam's wind tunnel, which has a 0.33 m^3/s (1188 m^3/h) ventilation rate difference from the real building. This ventilation rate difference is the result of a 0.11 C_p difference. Depending on the area of the opening, this difference in ventilation rate may or may not be significant. Based on the Texas Tech. Building's volume of around 498 m^3, the real building, Selvam's wind tunnel model, Selvam's CFD model and Opus' model would have an air change per hour rate, ACH, (at full scale) of 57, 54.94, 53.49 and 45.83.

ISSUES

After the preliminary experiments, which dealt mostly with the basic CFD procedure (efficient grid, fast and good convergence, rough and quick approach to match the results with the field data) the next experiments (the main experiments) focused on the accuracy and precision of the results. Factors that effected the accuracy and precision of the CFD experiments appeared throughout the simulation process. Issues arose from the initial stages of the CFD modelling, such as the geometry definition, to the last stage, the calculation method. Examples of issues which arose during the geometry definition were the effects of grid resolution (coarse/ fine), grid pattern (uniform/ non-uniform), and the dimension (2D/ 3D) that effect accuracy of the results. Factors that effected the calculation process include the turbulent model (standard k-ε, RNG k-ε, etc.) and the convergence criteria.

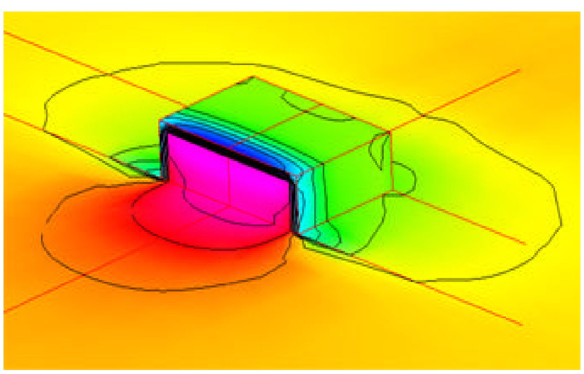


Figure 4 Pressure contour

In this experiment three issues were selected. These involved cases based on the combination of 2D/3D, k-ε/RNG turbulent model, and uniform/atmospheric boundary layer flows. Table 2 lists the combinations that were run.

The purpose of comparing 2D and 3D geometries was to see how well they simulated the wind flow. Two-dimensional modeling offered some advantages such as less memory space, preparation and calculation time when compared to three-dimensional modeling. The latter, however, gave a more realistic simulation of wind flows around buildings that are always three-dimensional.

The purpose of comparing k-ε and RNG (renormalisation group) k-ε turbulent model was to find the more precise model for a given case. The standard k-ε model is the most widely used model. It is robust, but its reliability for simulating wind flow around buildings is still arguable. Some researchers (such as Yuguo Li[10]) found it unreliable, while others state that it is reasonable for getting a rough idea of the wind flow. Paterson stated that in good CFD programs the k-ε turbulence model could produce good results. However, he recommended using the RNG k-ε turbulent model.

The purpose of comparing simulations with uniform and atmospheric boundary layer flows was to study how far the boundary layer affected the C_p. Preliminary experiments found that placing the inlet far away from the model resulted in C_p values that showed good agreement with the field data despite the uniform flow condition which was applied at the inlet. This is because the simulation of roughness over the ground generated a small boundary layer effect.

RESULTS AND DISCUSSIONS

Three variables were studied: C_p values on the windward and leeward faces of the building and the difference between the Cp at the inlet and outlet. All values are compared to the field data.

In general, there is no single ideal combination. It seems that what one needs to know will dictate the selection of the combination. One combination can simulate, for example, a good windward C_p but a bad leeward Cp. As an example, for natural ventilation one is probably most concerned with the inlet/outlet C_p difference as this is the value used to calculate indoor ventilation.

Table 2 indicates both the overall and individual performance of various CFD combinations. Care should be taken when considering the overall ranking as the table ignores the relevance of each of the variables (windward C_p difference, leeward C_p difference, etc.) by treating them all the same (no different weighting was applied). Each combination was given a value corresponding to its rank for a given variable, starting from eight (best) to one (worst). It is clearly seen that 3d-case3 is the best overall, followed by 3d-case4, then 3d-case1 and so on. 2d-case1 is the worst. However, on closer examination one sees that there is no perfect combination. For example, 3d-case3 is better overall than 3d-case4, although some variables (average windward Cp difference, windward max C_p difference) are better predicted by 3d-case4 than 3d-case3. It is easy to compare the performance of CFD-ACE for a specific variable as bigger ranking numbers indicate better performance.

Table 2 Experimental combinations and ranking of simulations

	2D				**3D**			
	Uniform flow		Atmospheric boundary layer		Uniform flow		Atmospheric boundary layer	
	k-ε	RNG	k-ε	RNG	k-ε	RNG	k-ε	RNG
	2d-case1	**2d-case2**	**2d-case3**	**2d-case4**	**3d-case1**	**3d-case2**	**3d-case3**	**3d-case4**
Average windward Cp difference	1	2	5	6	3	4	7	8
Average leeward Cp difference	2	4	1	3	8	6	7	5
Windward Max C_p difference	1	2	5	6	3	4	7	8
Leeward Max C_p difference	2	4	1	3	8	6	7	5
Max inlet/out-let C_p difference – field data (0.85)	1	2	3	4	5	6	8	7
Total	**7**	**14**	**15**	**22**	**27**	**26**	**36**	**33**

Note: Ranking : best (8) ←----→ worst (1)

Some other conclusions can be derived from comparisons of Table 2 as follows:

- In general 3D modeling is better than 2D modeling. (3d-case1 is better than 2d-case1; 3d-case2 is better than 2d-case2, etc.)
- Modeling the atmospheric boundary layer is better than the uniform flow (2d-case3 is better then 2d-case1; 3d-case3 is better than 3d-case1, etc.)
- The standard k-ε turbulence model is better than RNG k-ε turbulence model when used for three dimensional simulations (3d-case1 is better than 3d-case2; 3d-case3 is better than 3d-case4). However, for two dimensional simulations, the RNG k-ε turbulence model is better (2d-case2 is better than 2d-case1; 2d-case4 is better than 2d-case3)

The important question is whether one could use the 3d-case3 scenario without first testing a variety of other configurations. To answer this question it is useful to examine the absolute values of Cps produced by CFD-ACE. Figures 5 to 8 illustrate the comparison of cases with other experimental data (from wind tunnel and CFD simulations) and the field data. It can be seen in Figure 5 that the CFD-ACE, 3d-case3 scenario produces better results than the other experiments. Its average windward C_p differs from the field data by only 0.027. This is lower than other CFD programs which claim good agreement (Selvam's CFD = 0.067). Figure 8 shows that CFD-ACE, 3d-case3 inlet/outlet C_p differences differ from the field data by 0.003. This is again better than the other experimental results, which differ from the field data by between -0.07 and +0.13.

CONCLUSION

Despite the close ranking between 3d-case3 (standard k-e turbulence model) and 3d-case4 (RNG k-ε turbulence model), the standard k-ε model is considered to be better and should be able to be used confidently for other cases. Using a combination of three-dimensional modeling, the standard k-ε turbulent model and atmospheric boundary layer flow the CFD-ACE program simulates the real conditions.

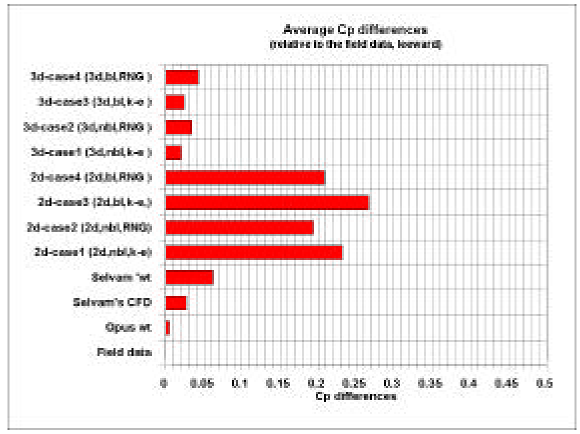
Returning to the general question raised in the introduction these results demonstrate that the answer to the question *How close is close enough?* is, always, problematic. This is because:

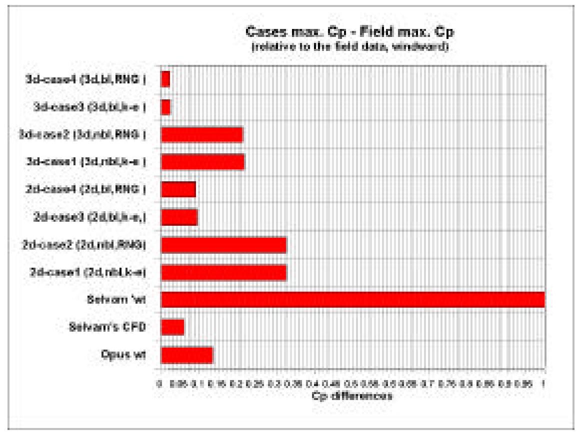
- Researchers very seldom address this issue in a manner that makes consultation of their published ‘authorities’ feasible;
- The criteria will depend on the circumstances: in a situation where the cost of failure is high the criterion will be more stringent. There is no absolute measure.

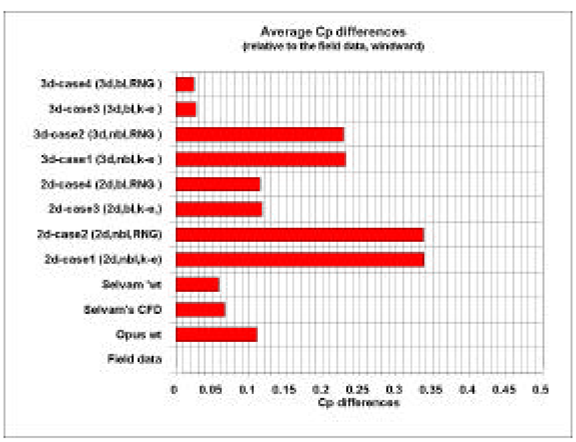
For the practitioners or the new user of simulation software like the CFD program evaluated here the lesson is simple: it is not enough to rely on the validation effort of program vendors. Each user must develop their own means of calibrating the programs they use. Each user should therefore be asking program vendors to provide with their programs the tools for routinely performing this calibration. Without these tools, it will always be possible to raise doubts about the reliability of the predictions of CFD and other building performance simulation software.

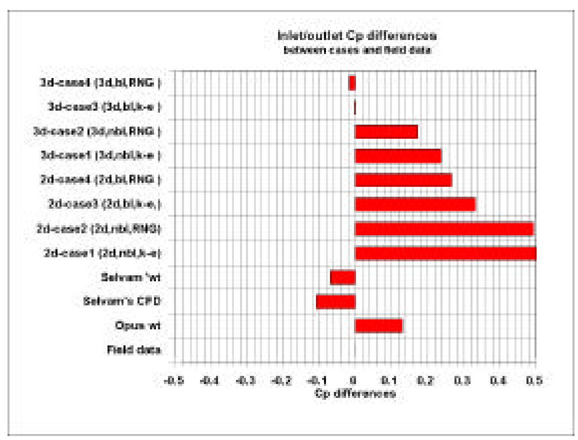
ACKNOWLEDGMENT

This research has been funded by the New Zealand Government through the Public Good Science Fund (PGSF) administered by the Foundation for Research Science and Technology.









REFERENCES

1 Levitan, M.L., Mehta, K.C., *Texas Tech Field Experiments for Wind Loads part I: Building and Pressure measuring System*, Journal of Wind Engineering and Industrial Aerodynamics, 41-44 (1992), pp. 1565-1576.

2 Levitan, M.L., Mehta, K.C., *Texas Tech Field Experiments for Wind Loads part II: Meteorological Instrumentation and Terrain Parameters*, Journal of Wind Engineering and Industrial Aerodynamics, 41-44 (1992), pp. 1577-1588.

3 Surry, D., 1991, *Pressure Measurements on the Texas Tech. Building: Wind Tunnel Measurements and Comparisons with Full Scale*, Journal of Wind Engineering and Industrial Aerodynamics, 38 (1991), pp. 235-247.

4 Cochran, L.S., Cermak, J.E., *Full- and Model - Scale Cladding Pressures on the Texas Tech University Experimental Building*, Journal of Wind Engineering and Industrial Aerodynamics, 41-44 (1992), pp. 1589-1600.

5 Selvam, R.P., *Computation of Pressure on Texas Tech Building*, Journal of Wind Engineering and Industrial Aerodynamics, 41-44 (1992), pp. 1619-1627.

6 through discussions by e-mails

7 Selvam, ibid.

8 Selvam, ibid.

9 Aynsley, R.M., Melbourne, W. and Vickery, B.J., 1977, *Architectural Aerodynamics*, London: Applied Science Publisher, Ltd.,p.89.

10 Aynsley, R.M., ibid.

11 Levitan, M.L., ibid.

12 Awbi, H.B., 1991, *Ventilation of Buildings*, London: Chapman and Hall, p.233.

13 Etheridge, D., Sandberg, M., 1996, *Building Ventilation: Theory and Measurement*, Chichester: John Wiley and Sons, p.20.

14 Aynsley, ibid. p.189.

15 Selvam, ibid.

16 discussion through the internet.

17 Selvam, R.P., *Computation of Pressure on Texas Tech Building*, Journal of Wind Engineering and Industrial Aerodynamics, 41-44 (1992), pp. 1624-1625.

18 Shao, L., Sharples, S., Ward, I.C., *Building Pressure Coefficient: Application of Three-dimensional CFD Methods to Prediction*, Building Service Engineering Research Technology, vol.13 (1992), no.2, pp.107-111.

19 see Fig.6.15 and Fig.6.16. in Aynsley, ibid.

20 Li, Y., *General Flow and Thermal Boundary Conditions in Indoor Air Flow Simulation*, Building and Environment, vol.29 (1994), no.3, pp.275-281.