Journal of Construction in Developing Countries, Vol. 14(2), 1–20, 2009

Validation of the Computational Fluid Dynamics (CFD) Method for Predicting Wind Flow Around a High-Rise Building (HRB) in an Urban Boundary Layer Condition

Abdul Razak Sapian

Abstract: Virtual Experiments or, more specifically, the Computational Fluid Dynamics (CFD) method have been used in numerous airflow studies. Most CFD code providers claim that their software can simulate and model airflow around a building. However, the literature shows little or no evidence of the implementation of this software for this purpose. Therefore, this study attempts to validate and justify the reliability of this software, by investigating the wind flow around a high-rise building. The main goal of this paper is to determine if the CFD method can be used to study wind flow around a high-rise building, with a focus on the effect natural cross ventilation within a building to predict the indoor air velocity, for human thermal comfort purposes. A software called FloVent from Flomeric Inc. UK was used in the experiments. The data obtained from the simulation are compared with wind tunnel data. The result of the analysis shows that the deviation between the CFD and wind tunnel data is less than 15% on average. This result indicates that the CFD can be used as an alternative method for investigating wind flow around high-rise buildings in an urban boundary layer condition.

Keywords: CFD, High-rise residential building, Urban wind, Natural cross ventilation

INTRODUCTION

The utilisation of pressure differences due to wind has wide applications, especially for the natural ventilation of residential, school, and industrial buildings during the summer in temperate and hot humid climates (Aynsley et al., 1977). Empirical methods that take wind effects into account are used to predict natural ventilation. The pressure distribution over the external envelope of the building must be estimated in these analyses.

As a result of the building shape and the influence of the surrounding topography, there are a few methods that can be used to predict the dynamic pressure induced by wind over the building envelope. These methods are:

Department of Architecture, Kulliyyah of Architecture & Environmental Design, International Islamic University Malaysia, Jalan Gombak, 53100 Kuala Lumpur, MALAYSIA.
*Corresponding author: arazaks@iiu.edu.my

i. Standards and guides
ii. Generalised algorithms
iii. Wind tunnel experiments
iv. Computational Fluid Dynamic (CFD) experiments

Among these methods, most researchers suggest that the best is the wind tunnel method (Wang, 1996; Pitts and Ward, 1983). They contend that the wind tunnel test is a basic and accurate means for studying airflow within a building. It is capable of simulating a reduced scale wind that nearly matches real wind flow.

The wind tunnel method utilises a physical model and its surroundings. The physical model and surroundings are constructed and placed in the wind tunnel where they are subjected to a controlled wind flow. Pressure sensor taps are installed at various points on the building envelope, corresponding to ventilation openings.

However, wind tunnel applications for buildings in Malaysia are quite new and expensive, and there are very few for high-rise experiments. Therefore, the CFD approach is the best alternative. There are numerous benefits derived from using the CFD to solve problems of predicting the flow and condition of air (MacLeod and Waskett, 1996). One benefit is that the CFD can reduce or eliminate the need for physical modelling. Alternative designs can be quickly and easily investigated to instil confidence in the proposed design. A CFD as applied to the built environment could be used to reasonably model the airflow. However, most of these refer to the airflow inside a building (Malsiah, 2001; Wang, 1996; Kevin and Waskett, 1996).

CFD Modelling

CFD is a modelling technique that entails representing a fluid flow problem by mathematical equations. The mathematical equations are based on the fundamental laws of physics. For building applications, the parameters of interest include velocity, pressure, temperature, turbulence intensity, and possibly the concentrations of smoke and contaminates. Therefore, the set of equations in the CFD are related to those variables.

In general, the equations involved are those for the conservation of momentum, which are sometimes referred to as the Navier-Stokes equations, the conservation of mass, and the transport equations for turbulent velocity and its scale (Awbi, 1991). In solving these equations, a computer calculation technique is used. This process is also known as Computational Fluid Dynamics.

COMPUTATIONAL FLUID DYNAMICS (CFD)

CFD has been used in construction modelling for almost 30 years. This method was first applied to buildings in the 1970's by Nielsen (Jones and Whittle, 1992). Since then it has developed into an invaluable tool for predicting fluid dynamics. The very rapid development of computers and the increased user-friendly CFD software has led to cost-effective building analysis using CFD (Whittle, 1996).

CFD is commonly used in building design, particularly for fluid flow simulations or modelling. These models can be divided into two categories, depending on if the analysis is of the inside or the outside the building (MacLeod and Waskett, 1996).

1. Internal models
    - Room air distribution
    - Pressure regime analysis
    - Contamination analysis
    - Airflow within a duct
    - Analysis of temperature profiles
    - etc.

2. External models the flue
    - Examination of the behaviour of emissions
    - Investigation of louver positions
    - Analysis of wind pressure
    - etc.

THE EXPERIMENT TO TEST THE RELIABILITY AND VALIDITY OF CFD SOFTWARE

FloVent has been used in many airflow studies. However, most of them indicate the airflow inside a building (Malsiah, 2001; Wang, 1996; Kevin and Waskett, 1996). The FloVent manufacturers claim that the software can simulate and model airflow around a building. However, a literature search does not show evidence of its use for this purpose. Therefore, this investigation attempts to validate and justify the reliability and validity of this software.

According to Bakaran and Stathopoulos (1992), the validation of the computed results is an integral part of numerical simulation. Consequently, a series of pilot studies was conducted.

Objective

The objectives of this testing are two fold:

1. To validate the possibilities of this software in studying wind flow around HRB specifically, as well as other building typologies in general. Thus, the data obtained from the simulation are acceptable in terms of its validity and confidence.

2. To determine the appropriate computational procedures and conditions of HRB and other building typologies.

Method

According to Baskaran and Stathopoulos (1992), results are sometimes validated using full-scale measurements, and are more often validated using wind tunnel data. Since full-scale measurement data are often unavailable, and it is time consuming to create an ABL generator for existing wind tunnel facilities, the Abdul Majid (1996) ABL wind tunnel data of urban wind profiles is often the most feasible to use.

The model chosen was a simple "flat-faced" rectangular 50 m high (12 storey) office building. Figure 1 shows the Abdul Majid (1996) wind tunnel simulation model.

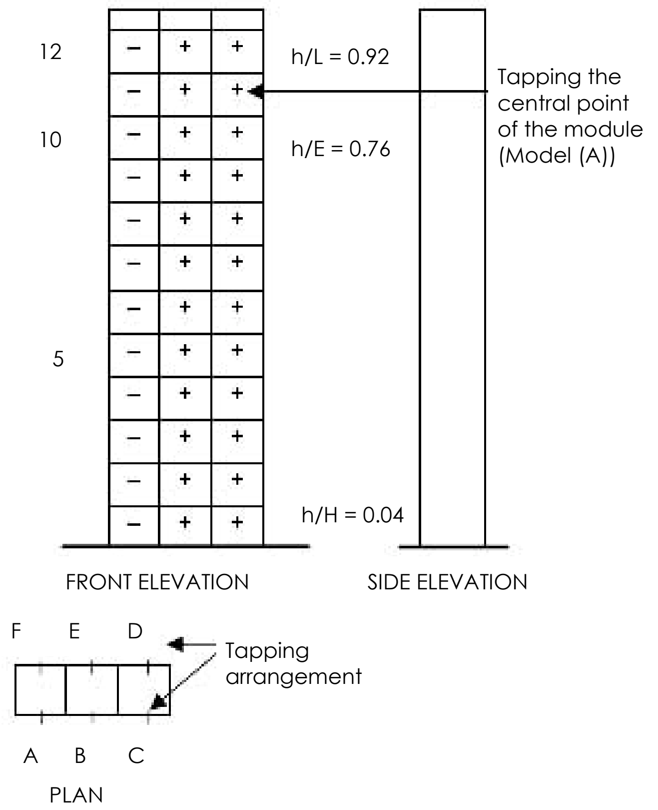


Figure 1. Plan and Elevation of the Simulation Model (12 storey or 50 m building)
Source: Abdul Majid, (1996)

Model

A base model of 50 m HRB at isolated building conditions was used. The size of the building was 78 m × 10 m × 50 m. Figure 2 shows a schematic of the model. The building was placed inside an overall domain solution size of 312 m × 450 m × 240 m high. The position of the building inside the overall domain solution was 120 m from x-plane, 250 m from z-plane, and 0 m from y-plane. Figure 3 shows the overall domain solution and the position of the model inside the overall domain solution. Figure 3 also can be considered to be the overall design set-up.

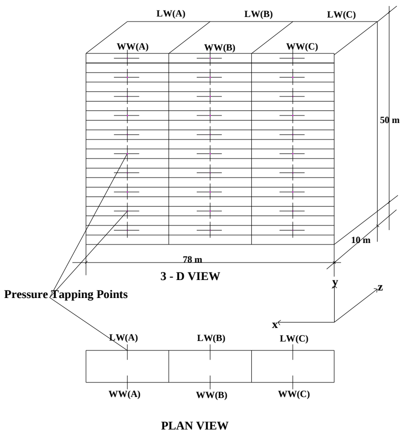


Figure 2. Simulation Model Shows the Pressure Tapping Points and the Dimensions

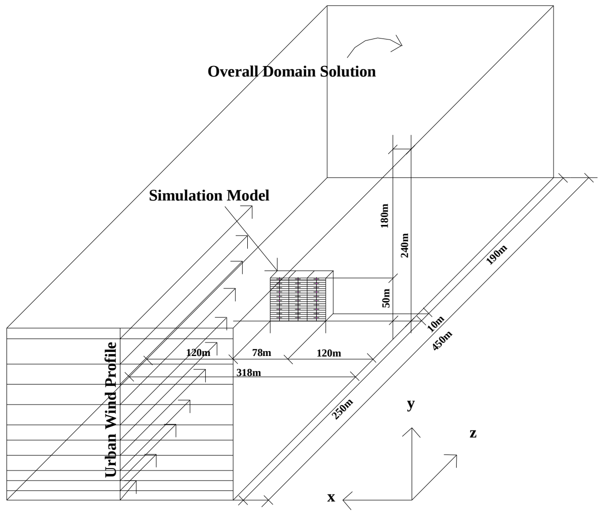


Figure 3. Overall Simulation Model Schematic

ABL Generator and Theoretical Wind Profile

The ABL generator was downloaded from the FloVent website. The ABL generator provided by FloVent uses the Log Law model to create the required theoretical wind profile. Since the Kuala Lumpur city centre contains both high and low-rise buildings, the roughness length (Z_o) equivalent to 2.0 as proposed by ASCE (1999) was used.

Table 1 shows the ASCE (1999) atmospheric boundary layer characteristics for different terrain roughness.

Log Law model:

$$V_z = V_{ref}\,[\log(Z / Z_o) / \log(Z_{ref} / Z_o)]$$

Where:

V_z = the mean wind speed at height Z
Vref = the mean wind speed at some reference height Z_{ref}
Z_{ref} = the reference height
Z = the height for which the wind speed V_z is computed
Z_o = the roughness length or log layer constant

Table 1. Atmospheric Boundary Layer (ABL) Characteristics for Different Terrain Roughness

Class	Terrain Description	Z_o (m)	α	I_u(%)	Exp.	Z_g (m)
1	Open sea, fetch at least 5 km	0.0002	0.10	9.2	D	215
2	Mud flats, snow; no vegetation, no obstacles	0.005	0.13	13.2		
3	Open flat terrain; grass, few isolated obstacles	0.03	0.15	17.2	C	275
4	Low crops; occasional large obstacles, x'/h > 20	0.10	0.18	27.1		
5	High crops; scattered obstacles, residential suburban, 15 < x'/h < 20	0.25	0.22	27.1	B	370
6	Parkland, bushes; numerous obstacles, x'/h ~ 10	0.5	0.29	33.4		
7	Regular large obstacle coverage (dense spacing of low buildings, forest)	1.0 – 2.0	0.33	43.4	A	460
8	City centre with high and low-rise buildings	≥ 2.0	0.40 ~ 0.67	-		

Source: ASCE (1999)

Reference Wind Speed

Table 2 and Figure 4 show the summary of surface wind speed for 33 years (1969 – 2002). This data was recorded at the Subang meteorological station by the Malaysia Meteorological Service Department.

Table 2. Summary of the Mean Surface Wind Speed (Monthly) for 33 Years (1969–2002)

1969 – 2002 (33 years)	J	F	M	A	M	J	J	A	S	O	N	D	AN
Mean surface wind speed (m/s)	1.1	1.2	1.3	1.2	1.3	1.3	1.4	1.5	1.4	1.4	1.3	1.2	1.3

Source: Malaysia Meteorological Services Department.

The data show that the annual mean surface wind speed was 1.3 ms^{-1}. However, the data were recorded at 19.2 m (height of anemometer head above the ground level). In this study, the international standard reference height for mean wind at 10 m above ground was used. Therefore, the wind speed must be corrected to a wind speed that referred to the international standard reference height. This can be done using the log law equation and by selecting the appropriate ABL characteristic from Table 2. As a result, the mean wind speed of the Kuala Lumpur urban terrain can be predicted using the same log law equation.

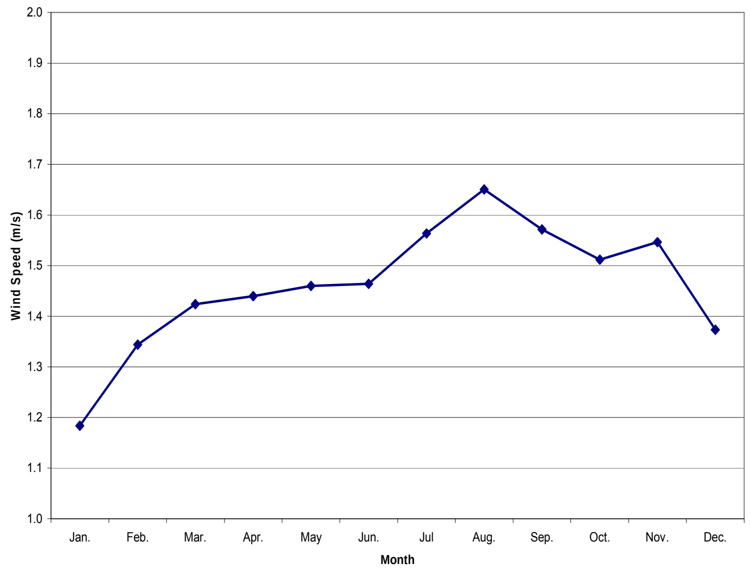


Figure 4. Summary of the Mean Surface Wind Speed (Monthly) for 33 Years (1969–2002)
Source: Malaysia Meteorological Services Department.

Log Law model:

$$V_z = Vref\ [\log (Z / Z_o) / \log (Z_{ref} / Z_o)]$$

Where:

V_z = the mean wind speed at height Z (Gradient wind)

V_{ref} = the mean wind speed at some reference height Z_{ref}

Z_{ref} = the reference height

Z = the height for which the wind speed V_z is computed (Gradient height)

Z_o = the roughness length or log layer Constant

According to the description given by ASCE (1999) and shown in Table 2, Subang meteorological station ABL characteristics can be described as open terrain (grass and a few isolated obstacles). Therefore, its mean wind speed exponent (α) can be assumed as 0.15, the roughness length (Z_o) was 0.03, and the gradient height (Z) was 275 m. Hence, the mean wind speed at the 10 m reference height was:

$$V_z = V_{ref}\ [\log (Z / Z_o) / \log (Z_{ref} / Z_o)]$$

$$V_{19.2} = 1.3\ [\log (275 / 0.03) / \log (19.2 / 0.03)]$$
$$= 1.83\ ms^{-1}$$

Therefore,

$$V_{ref} = V_z / [\log (Z / Z_o) / \log (Z_{ref} / Z_o)]$$

$$V_{10} = 1.83 / [\log (275 / 0.03) / \log (10 / 0.03)]$$
$$= 1.17\ ms^{-1}$$

Thus, the corrected mean wind speed for Subang at the 10 m reference height was approximately 1.17 ms^{-1} using the same log law equation, the gradient wind for Kuala Lumpur can be obtained. However, the Kuala Lumpur ABL characteristic must first be determined. Referring to Table 2, the Kuala Lumpur terrain condition can be described as a city centre with high and low-rise buildings. Observation showed that although Kuala Lumpur was approximately 20 km from Subang, it has a denser area compared with Subang. Therefore, its mean wind speed exponent (α) can be assumed to be 0.40, with a gradient height of 460 m, and a roughness length (Z_o) of 2.0 (ASCE, 1999). Hence, the gradient wind for Kuala Lumpur is:

$$V_z = V_{ref} [\log (Z / Z_o) / \log (Z_{ref} / Z_o)]$$

$$V_{z(KL)} = 1.17 [\log (460 / 2) / \log (10/ 2)]$$
$$= 3.94 \text{ ms}^{-1}$$

This gradient wind can then be used to obtain the 10 m reference wind for the Kuala Lumpur urban area.

$$V_z = V_{ref} [\log (Z / Z_o) / \log (Z_{ref} / Z_o)]$$

$$V_{10(KL)} = 3.94 / [\log (460 / 2) / \log (10/ 2)]$$
$$= 1.17 \text{ ms}^{-1}$$
$$\approx 1.0 \text{ ms}^{-1}$$

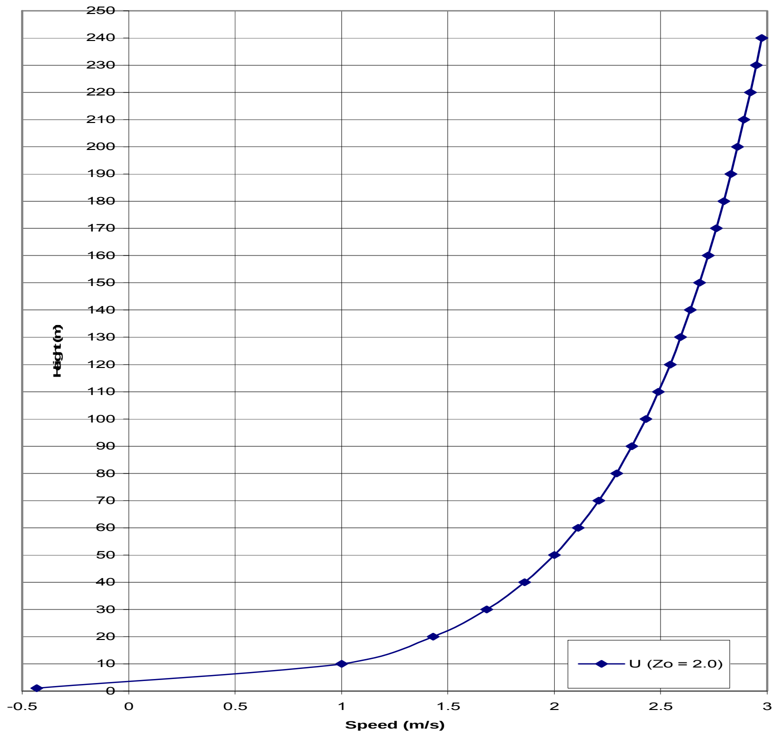


Figure 5. Mean Wind Speed Profile Using the Log Law Model (Z_o = 2.0, reference height = 10 m and reference speed = 1.0 ms^{-1})

The corrected Subang mean wind speed at the 10 m reference height was used to determine the Kuala Lumpur mean wind speed. It was predicted that the Kuala Lumpur mean wind speed at the 10 m reference height was approximately 1.0 ms^{-1}. This value was within the ranges obtained from the case study done by Abdul Razak et al., (2001). Therefore, the mean surface wind speed of 1.0 ms^{-1} at the 10 m reference height was used in this study to estimate the vertical pressure distribution in the HRB. Figure 5 shows the urban wind profile generated by the FloVent software using a log law model with a 1.0 ms^{-1} wind speed at the reference height.

Pressure Tapping Point

The simulation converted the pressure into Cp values as the final result. Therefore, monitoring points were placed at the front and rear of the building surface near the openings. These monitoring points acted as virtual pressure tapping points that were commonly used in wind tunnel testing. Figure 2 shows the model with the tapping point locations. The total tapping points included 60 numbers, with 30 numbers placed at the front and 30 numbers placed at the rear.

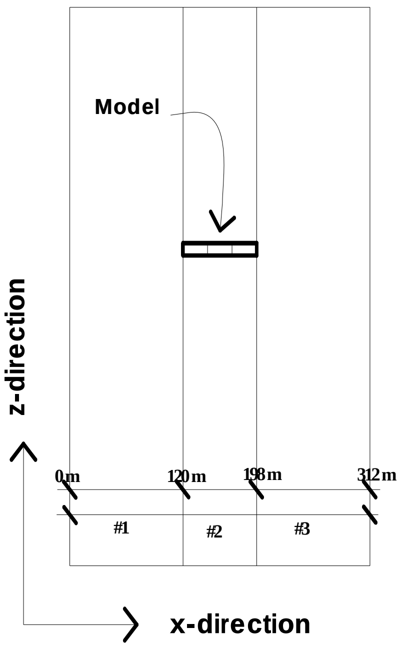
Grid System

A Cartesian-type grid was used for this simulation. The system grid was defined in the x, y, and z directions. The x-direction system grid was defined by three grid constraints (x, y, and z).

In CFD applications, the computational grid cells define the solution domain. The number and size of the cells represent the level of resolution that the calculation can achieve. Cells vary in size and are generally defined as increasing, decreasing, or uniform.

Smaller and uniform grid cells are normally defined in areas where large solution gradient variables are evident. Failure to provide enough mesh in these areas will result in the supply jet or boundary layer flow being insufficiently resolved. This result can also be described as an unrealistic local situation. For economic purposes (in terms of computing time), it is usual to either decrease or increase the grid from the model or object, in the spatial sense, in an area removed from those of importance.

Consequently, for this experiment, the total number of cells produced from this system grid was 36 numbers (X-direction) × 40 numbers (y-direction) × 48 numbers (z-direction). This gave a total of 69,120 cells. Therefore, the numerical solution for this model was carried out in 69,120 control volumes.



Grid Constraint #1:

Start location: 0 m
End location: 120 m
No. of grid: 7 numbers
Type of grid distribution: decreasing

Grid Constraint #2:

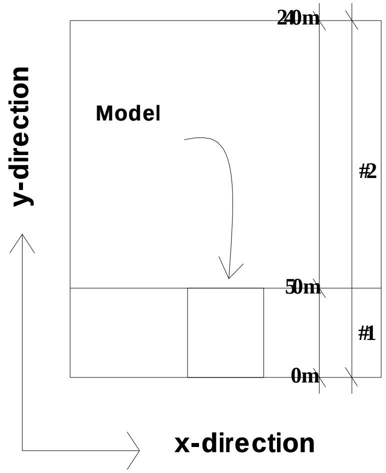
Start location: 120 m
End location: 198 m
No. of grid: 22 numbers
Type of grid distribution: uniform

Grid Constraint #3:

Start location: 198 m
End location: 312 m
No. of grid: 7 numbers
Type of grid distribution: increasing

Total grid at x-direction: 36 numbers

Figure 6. Grid System at x-direction



Grid Constraint #1:

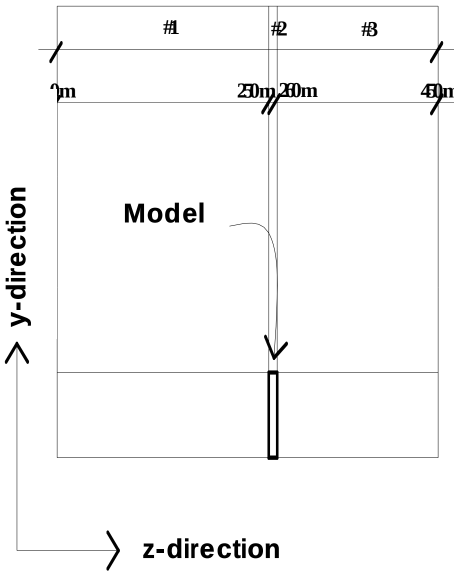
Start location: 0 m
End location: 50 m
No. of grid: 30 numbers
Type of grid distribution: uniform

Grid Constrain #2:

Start location: 50 m
End location: 240 m
No. of grid: 10 numbers
Type of grid distribution: increasing

Total grid at y-direction: 40 numbers

Figure 7. Grid System at y-direction



Grid Constraint #1:

Start location: 0 m
End location: 250 m
No. of grid: 23 numbers
Type of grid distribution: decreasing

Grid Constrain #2:

Start location: 250 m
End location: 260 m
No. of grid: 6 numbers
Type of grid distribution: uniform

Grid Constrain #3:

Start location: 260 m
End location: 450 m
No. of grid: 19 numbers
Type of grid distribution: increasing

Total grid at z-direction: 48 numbers

Figure 8. Grid System at z-direction

Simulation Condition

The simulation conditions used in this model were steady-state, revised k-ε turbulence, and negative Y-direction gravity. The overall domain solution used was:

1. Global system setting of datum pressure at 1 atm
2. Ambient temperature is 32°C
3. External temperature is 32°C
4. Turbulent kinetic energy (*k*) is 0.1 J kg^{-1}
5. Turbulence dissipation rate (ε) is 0.1 W kg^{-1}
6. Heat transfer coefficient is 10 W / (m^2 K^{-1})

The solution control for this simulation was set as described below. The final value was obtained after several try and error iterations until the simulation attained convergence.

1. The overall solution control was set using a segregated conjugate residual with 500 numbers for the outer iteration. The fan relaxation used was 1.0.

2. Variable solution control was pressure with 50 numbers of inner iteration.

3. Additional solver control was also set as pressure with a 1.0 linear relaxation.

The input values described above were set as the simulation conditions under which the solution was converged. It took approximately 8 h to solve the iteration using a very basic personal computer.

RESULTS

Table 3 summarises the comparative analysis of the average pressure coefficients (Cp), showing the relative differences and relative percentage (%) differences between the CFD and the wind tunnel (WT) at the windward (WW) and leeward (LW) pressure tapping points.

The results show that the average Cp differences at the windward point ranged from 0.02 to 0.05. The average differences of the leeward Cp values range from 0.004 to 0.03. The average relative percentage (%) differences at the windward point range from 3.86% to 9.55%. The leeward average relative percentage (%) differences range from 1.17% to 14.84%.

Figure 9 shows a graphical summary of the average relative differences and average percentage (%) relative differences of the pressure coefficients between the CFD and wind tunnel at the windward and leeward pressure tapping points. These values are considered small and are in good agreement with results from Shao (1992), Selvam (1992), Malsiah (2001), Nugroho et al., (2007), and Agung (2007). Figure 10 plots the vertical pressure distribution of CFD data superimposed with Abdul Majid (1996) wind tunnel data. It can be concluded that these profiles are similar. The windward vertical pressure distribution profiles show that the maximum Cp value is at the 2/3 of the building height, or a height ratio of 0.8 to 0.9. The Cp value decreases going up or down the building, but increases again at the height ratio of 0.1 to 0.2. After this point the Cp values decrease.

The leeward Cp values are quite consistent. The Cp values range from –0.2 to –0.45. Due to the effect of the high-speed winds that pass over the roof of the building, the suction zone at the top of the building is much higher compared to the lower part. This is in line with the Abdul Majid (1996) findings.

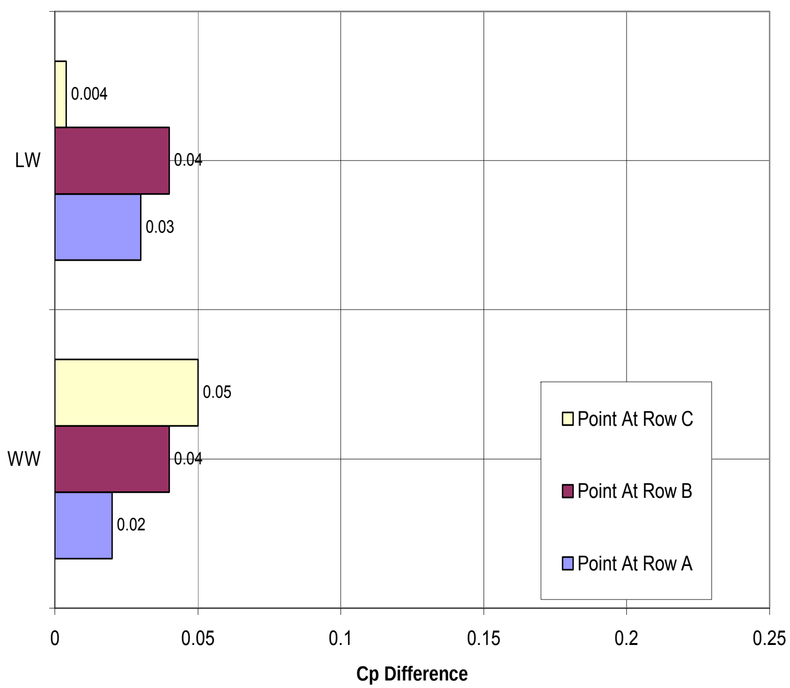
Table 3. Summary of the Average Relative Differences and Average Relative Differences Percentage (%) of Pressure Coefficients between the CFD and Wind Tunnel at the Windward and Leeward Pressure Tapping Points

WINDWARD									
H/h	Cp A CFD	Cp A WT	CFD – WT	Cp B CFD	Cp B T	CFD – WT	Cp C CFD	Cp C WT	CFD – WT
0.98	0.56	0.59	–0.03	0.50	0.61	–0.11	0.55	0.55	0.00
0.88	0.83	0.61	0.22	0.82	0.64	0.18	0.83	0.57	0.26
0.78	0.79	0.61	0.18	0.80	0.60	0.20	0.79	0.59	0.20
0.68	0.69	0.59	0.10	0.72	0.60	0.12	0.69	0.60	0.09
0.58	0.60	0.52	0.08	0.63	0.56	0.07	0.60	0.53	0.07
0.48	0.52	0.56	–0.04	0.56	0.56	0.00	0.52	0.49	0.03
0.38	0.46	0.53	–0.07	0.51	0.50	0.01	0.45	0.46	–0.01
0.28	0.41	0.50	–0.09	0.48	0.51	–0.03	0.41	0.42	–0.01
0.18	0.37	0.43	–0.06	0.45	0.46	–0.01	0.37	0.40	–0.03
0.08	0.36	0.44	–0.08	0.45	0.52	–0.07	0.36	0.44	–0.08
Average	0.56	0.54	0.02	0.59	0.56	0.04	0.56	0.51	0.05
Av Diff. (CFD – WT)		0.02			0.04			0.05	
Rel % Diff. CFD		3.86			6.08			9.55	

Continued on next page

Table 3. (continued)

LEEWARD									
H/h	Cp A CFD	Cp A WT	CFD – WT	Cp B CFD	Cp B WT	CFD – WT	Cp C CFD	Cp C WT	CFD – WT
0.98	–0.46	–0.35	–0.11	–0.44	–0.34	–0.10	–0.46	–0.36	–0.10
0.88	–0.43	–0.39	–0.04	–0.41	–0.36	–0.05	–0.44	–0.36	–0.08
0.78	–0.40	–0.38	–0.02	–0.37	–0.39	0.02	–0.41	–0.38	–0.03
0.68	–0.37	–0.38	0.01	–0.32	–0.36	0.04	–0.38	–0.38	0.00
0.58	–0.35	–0.37	0.02	–0.28	–0.38	0.10	–0.35	–0.35	0.00
0.48	–0.33	–0.38	0.05	–0.25	–0.34	0.09	–0.33	–0.36	0.03
0.38	–0.31	–0.37	0.06	–0.23	–0.29	0.06	–0.31	–0.33	0.02
0.28	–0.30	–0.37	0.07	–0.22	–0.32	0.10	–0.30	–0.37	0.07
0.18	–0.28	–0.39	0.11	–0.20	–0.29	0.09	–0.29	–0.35	0.06
0.08	–0.28	–0.39	0.11	–0.20	–0.29	0.09	–0.28	–0.35	0.07
Average	–0.35	–0.38	0.03	–0.29	–0.34	0.04	–0.35	–0.36	0.004
Av Diff. (CFD – WT)		0.03			0.04			0.00	
Rel % Diff. CFD		–7.38			–14.84			–1.17	



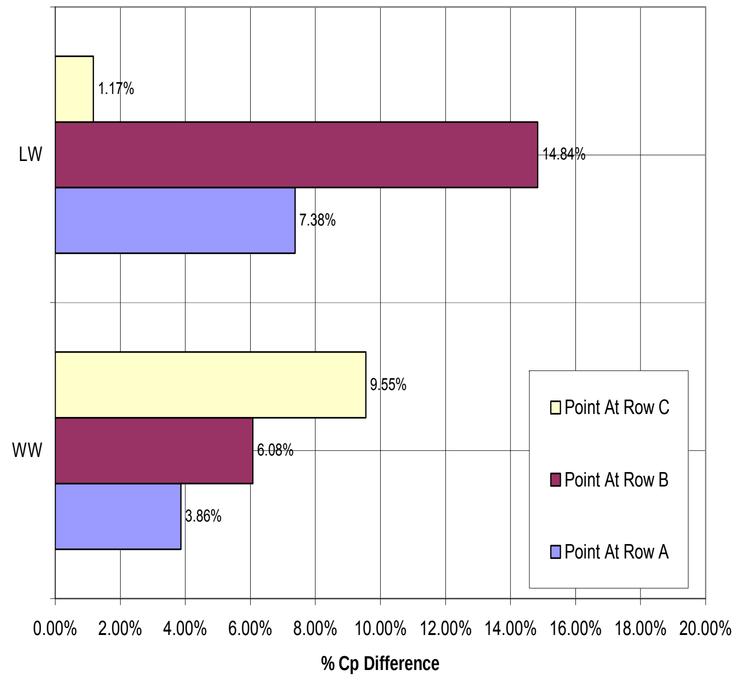


Figure 9. Summary of the Average Relative Differences and Average Percentage (%) Relative Differences of Pressure Coefficients between the CFD and Wind Tunnel at the Windward and Leeward Pressure Tapping Points

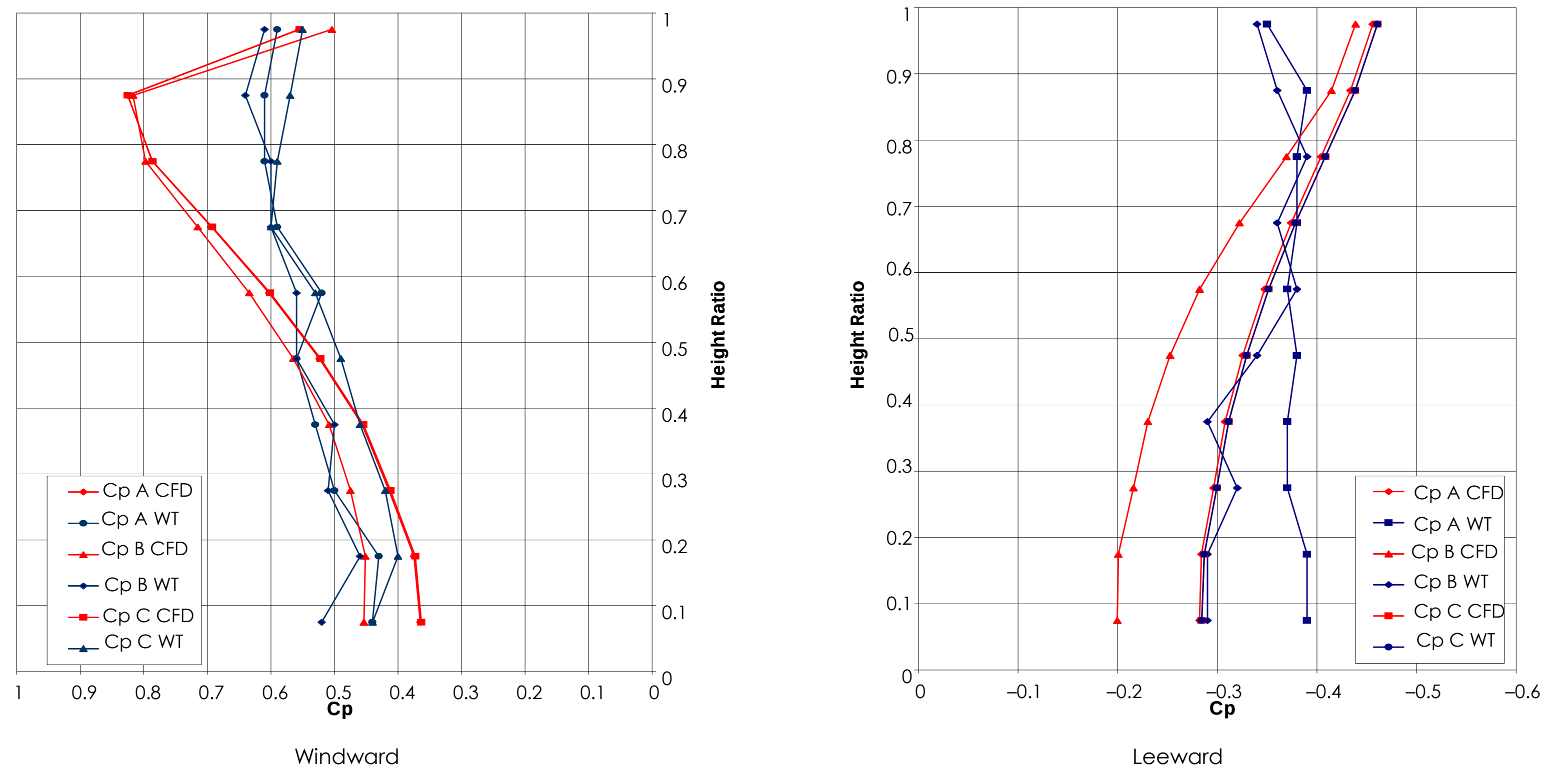


Figure 10. Vertical Distribution of CFD Windward and Leeward Pressure Coefficients (Cp) Superimposed with the Abdul Majid (1996) Wind Tunnel Windward and Leeward Pressure Coefficients (Cp).

CONCLUSION

From this study, it can be concluded that:

1. CFD software, and in particular FloVent, is capable of simulating the airflow around buildings and producing the pressure distribution values caused by the wind pressure.

2. The results obtained from the simulation show a deviation of less than 15% compared with wind tunnel results. Shelvam (1992) allowed up to a 7% deviation for the average windward point between his CFD calculation and the experimental data. In a separate case study, another CFD expert, Shao et al. (1992), accepts a 20% tolerance for a good agreement between his CFD Codes' Cp results and the experiment data. It is unrealistic and misleading to expect a complete match between CFD results and experiment data. Airflows around and inside buildings are turbulent and vary with time. Therefore, any measurements of airflow variables are generally noted as average values. CFD programs, on the other hand, tend to calculate the flow based on a particular set of steady state conditions (Satwiko et al., 1998). Thus, a certain degree of deviation (between CFD results and experiment data) can be tolerated. Therefore, these CFD results are found to be reliable and acceptable.

3. The above results indicate that the computational procedures and conditions applied in this effort are valid. Hence, the same experimental procedures and conditions can be adopted for use in similar objects to be tested.

In general, this testing procedure gives some indication that CFD is a reliable alternative for investigating the wind flow around a building. It can be used to efficiently simulate a virtual wind tunnel as well as real experimental data.

REFERENCES

Abdul Majid I. (1996). ***Wind-Driven Natural Ventilation in High-Rise Office Buildings with Special Reference to The Hot-Humid Climate of Malaysia.*** *PhD diss.*,

Abdul Razak, S., Haliza, M. and Mohd Hamdan A. (2001). Evaluation of Thermal Comfort Performance in Low Cost Flats: Case Study of Sri Perak Flats, Kuala Lumpur. Proceeding on Development of Passive Solar Design & Technology in Tropical Climates. Japanese Society for the Promotion of Science-Vice Chancellor Committee (JSPS_VCC) Seminar. School of Housing Building & Planning, Universiti Sains Malaysia, Penang. 12 November 2001.

American Society of Civil Engineers (1999). Wind Tunnel Studies of Buildings and Structures. ***ASCE Manuals and Reports on Engineering Practice***, No. 67 (1999), USA.

Awbi, H.B. (1991). Ventilation of Buildings, (2nd ed.). London: Spon Press.

Aynsley, R.M., Melbourne, W. and Vickery, B.J. (1977). ***Architectural Aerodynamics.*** London: Applied Science Publishers Ltd.

Baskaran, A. and Stathopoulos, T. (1992). Influence of Computational Parameters on Evaluation of Wind Effects on The Building Envelope. ***Building and Environment***, 27(1): 39—49.

Jones, P.J. and Whittle, G.E. (1992). Computational Fluid Dynamics for Building Air Flow Prediction – Current Status and Capabilities. ***Building and Environment***, 27(3): 321—338.

MacLeod, K.G. and Waskett, P.R. (1996). Computational Fluid Dynamic (CFD) – A Timely Concept? CIBSE/ASHRAE Joint National Conference, 29th September – 1st October 1996,Harrogate, UK.

Malaysia Meteorological Services Department, (1969–2002). Malaysia Meteorological Service, Climatological Section, Kuala Lumpur.

Malsiah, H. (2001). ***Keberkesanan Pengudaraan Semulajadi pada Rekabentuk Bilik Darjah Sekolah Piawai di Malaysia.*** Mast. diss., Universiti Teknologi Malaysia.

Nugroho, A.M., Ahmad, M.H. and Ossen, D.R. (2007). Preliminary Study of Thermal Comfort in Malaysia's Single Storey Terraced Houses. ***Journal of Asian Architecture and Building Engineering***, 6(1): 175—182.

Nugroho, A.M. (2007). Solar Chimney Geometry for Stack Ventilation In Malaysia Terrace House. PhD diss., Universiti Teknologi Malaysia.

Pitts, A.C. and Ward, I.C. (1983). ***A Review of the Prediction and Investigation of Air Movement in Buildings.***University of Sheffield.

Satwiko, P., Locke, N. and Donn, M. (1998). Reproducing The Real Pressure Coeficient Using A Computational Fluid Dynamic Program: How Close is Close Enough? Proceedings of the 32th Annual Conference of the Australia and New Zealand Architectural Science Association. 15 – 17 July 1998, University of Wellington, New Zealand.

Selvam, R.P. (1992). Computation of Pressure on Texas Tech Building. ***Journal of Wind Engineering and Industrial Aerodynamics***, 43(3): 1619—1627.

Shao, L., Sharples, S. and Ward, I.C. (1992). Building Pressure Coefficients: Application of Three-dimensional CFD Methods to Prediction. *Building Service Engineering Research Technology*, 13(2): 107—111.

Wang, Z. (1996). *Controlling Indoor Climate: Passive Cooling of Residential Buildings in Hot-Humid Climates in China.* Report TABK-96/1008, Department of Building Science Sweden: Lund University.

Whittle, G. (1996). Flow Simulation Using Computational Fluid Dynamics – A Cost Effective Tool Flow Simulation. Air *Infiltration Review*, 18(1): 5—7.