Validation of CFD Simulation for Wind Velocity Measurement in High-rise Apartment Buildings in Surabaya

Fristatesa Andriani, Asri Dinapradipta, FX. Teddy Badai Samodra
Departement of Architecture,
Institut Teknologi Sepuluh Nopember
e-mail: fristatesa.andriani@gmail.com

***Abstract*—The double-loaded circulation system makes the performance of cross ventilation in apartment buildings to be ineffective. Therefore, the alternative of the ventilation system is necessary to be examined in order to obtain an adequate velocity comfort. To determine the appropriate alternatives, we need to discover the suitable software. In this study, a validation of a simulation method using Computational Fluid Dynamics (CFD) is carried out to determine whether the software is capable to provide useful analysis for high-rise apartments. Based on the results, the validation of the correlation value (R^2) between the measurement and simulation is 0.67. It shows that the influence between variables is sufficient.**

***Keywords*— High-rise Apartment, Wind Velocity, Field Measurement, CFD Simulation.**

I. INTRODUCTION

Population density has caused building density, which has an impact on the lack of land for residential development, thus changing the pattern of development from horizontal housing to vertical housing or high rise apartments. Climate conditions in the tropics have characteristics; high air temperatures, relatively high humidity, and low wind speeds that make environmental conditions uncomfortable [1]. Relatively low wind velocity in Surabaya together with the apartment's circulation system i.e. double loaded makes ineffective ventilation system inside the apartment buildings.

The simulation program used is Computational Fluid Dynamics (CFD) which can produce output in the form of a pattern of airflow movement and its speed. Computational fluid dynamic (CFD) has become a useful tool for designers and researchers in the study of indoor and outdoor environments conditions in building design. Parameters such as air velocity and relative humidity solved by CFD are very important for designing an acceptable indoor comfort environment. The CFD technique has been applied with great success in building design and the advantages of analyzing [2] ventilation performance; [3]. [4] have shown that numerical solutions to flaw problems can be obtained quickly and well.

In this study, experimental simulations were carried out in the form of CFD simulations to validate the

relationship between field measurements and CFD simulations in high-rise buildings in the tropical climate.

II. METHOD

The purpose of this study is to simulate CFD and field measurements in high-rise apartment buildings in Surabaya. The selection of research methods must also be adapted to the needs of the study. To find out the wind velocity in apartment units, a research method is needed that has a focus on seeing the influence of variables and can manipulate the variables. Based on these needs, the experimental research method was chosen as the method in this study. Based on [5] experimental research can use computer assistance or modeling as an experiment to achieve the objectives in the study. In this study, digital simulations chose as the main challenge to make modifications to the actual building, also technical difficulties, and require a long time and high costs. This difficulty can be overcome by using digital simulations.

A. Field measurements

The object of the research is an apartment with a building height of 30 floors, divided into five podiums and 25 levels of apartment units. This apartment has a courtyard 2U type, which is divide into tower A and tower B. Field measurements are carried out on the 20th floor, in 4 apartment units in tower A explain in floor plan Figure 1b. The field measurement uses kanomax aerodynamics shown in Figure 1b, where the device records 24 hours of airflow in the room at 10 minutes' intervals. This measurement position is carried out on the 20th floor, as shown in Figure 2.

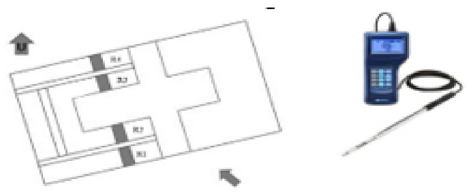


Figure 1. Floor plan and Kanomax aerodynamic.

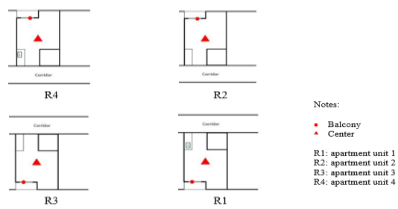


Figure 2. Field measurements position.

Wind velocity required for the simulation obtained by averaging annual wind speed data from the local meteorological office (BMKG Surabaya) and through a calculation using a power law or

$$Vz = VzG\left(\frac{z}{zG}\right)\alpha \qquad (1)$$

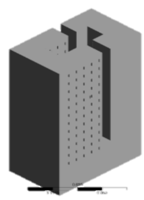
Where Vz is the wind velocity at a certain height (z), and VzG is the wind velocity at the reference height (G). According to the Surabaya City Statistics Agency (2014), the city of Surabaya is located at 7 ° 21'LS - 7 ° 21'LS and 112 ° 36'BT - 112 ° 54'BT. The area is a lowland with an altitude between 3-6 m above sea level, except the southern area with an altitude of 25-50 m above sea level. Surabaya City wind conditions show the highest wind speed in October, reaching 3.9 m / s. While the lowest wind speed occurs in March, April, June at 2.7 m / s with the wind direction and speed conditions that most often occur in the city of Surabaya in the last five years. Based on the image, the most wind direction moves west and east with a speed range of 10-12.5 m / s.

Wind velocity data from BMKG is assumed to be observed at the height of 10 m in an open country area, while the apartment model is located at the height of 80-100 m in a suburban area as mentioned in Table 2. Therefore, to obtain a proper wind velocity at the model location and height, a series of calculation using the Eq.(1) is required. Dir at 11.00 and 14.00 have the same value, which is 990%. The same percentage occurs at 12.00, 13.00, and 15.00, where the percentage of Dir is between 100-120%. Every hour there is a percentage of wind speed between 7 - 9 knots, with the highest speed at 12.00 and 15.00. Furthermore, the highest reference occurs at 12.00, 13.00, and 15.00 at the speed of 4.12 - 4.63 m / s. Results obtained from the NOAA wind data of Juanda Airport Wind data are wind speeds in the research building environment in Table 1 where the wind speed in the research object environment reaches 5.52 m / s at 12.00 and 15.00.

Tabel 1.
Wind Velocity Data from BMKG

Time	Dir	Knot	m/s	Freestream	m/s
00.00	990	5	2.57	4.28	2.77
01.00	150	8	4.12	6.85	4.43
02.00	990	8	4.12	6.85	4.43
03.00	990	9	4.63	7.71	4.99
04.00	990	8	4.12	6.85	4.43
05.00	990	9	4.63	7.71	4.99
06.00	110	11	5.66	9.43	6.10
07.00	100	13	6.69	11.14	7.20
08.00	100	13	6.69	11.14	7.20
09.00	60	15	7.72	12.85	8.31
10.00	990	5	2.57	4.28	2.77
11.00	990	7	3.60	6.00	4.29
12.00	110	9	4.63	7.71	5.52
13.00	120	8	4.12	6.85	4.91
14.00	990	7	3.60	6.00	4.29
15.00	100	9	4.63	7.71	5.52
16.00	120	6	3.09	5.14	3.33
17.00	150	5	2.57	4.28	2.77
21.00	220	3	1.54	2.57	1.66
22.00	270	2	1.03	1.71	1.11
23.00	990	3	1.54	2.57	1.66

Tabel 2.
Comparison of Two Validation Models

	Base Case	Verification Model
		
Unit dimension	12 m²	12 m²
Floor to floor	3 m	3 m
Podium	5 fl	5 fl
Apartemen	25 fl	25 fl
The building height	102 m	102 m
Courdyard	2U	2U
Tower	A & B	A & B

Wind flow prediction in this study was carried out by simulation using the Computational Fluid Dynamic (CFD) method. CFD is one method for research related to flow movements such as wind in a built environment. The development of technology related to CFD makes it possible to simulate wind flow in a user environment. One of the software used for CFD analysis in ANSYS Fluent. This software can analyze and calculate the prediction of wind flow on an object such as a building. The use of CFD as a simulation tool does not have a 100% accuracy level because the form of the model undergoes simplification. So it is necessary to verify the results of the simulation with field measurements to find out how much deviation (deviation) in the simulation results. There are three stages in simulating CFD, namely Processing, Solving, and Post Processing.

1) Processing: Geometrical Model and Meshing

The Processing stage is also called the modeling stage. Two main objects need to be considered modeling, namely building objects and domains in Figure 3 of the building. Domains in the CFD model can refer to as Boundary Layers. Boundary Layer is a representative form of ABL (Atmospheric Boundary Layer), which is the condition of the wind flow environment around a building object. In a domain that does not need to form an object that is the same as environmental conditions, this condition has become part of the input terrain roughness of the Boundary Layer. The form of domain Boundary

Layer suggested by Revuz et al. [6]; Elshaer [7]; and Fahmi et al. [8] In the ANSYS 14.0 application, modeling can be done using the geometry features of ANSYS or can use modeling software such as GAMBIT and AutoCAD 3D. Meshing is an import stage model in FLUENT ANSYS to determine the quality level of geometry. At this stage, the choice of the type of meshing made, the quality and method used.

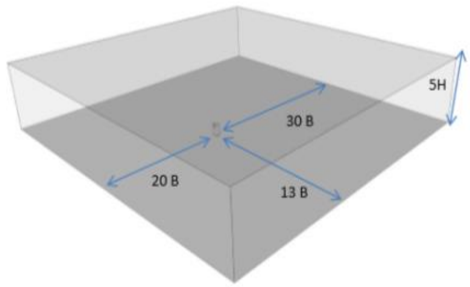


Figure. 3. Domain Setting.

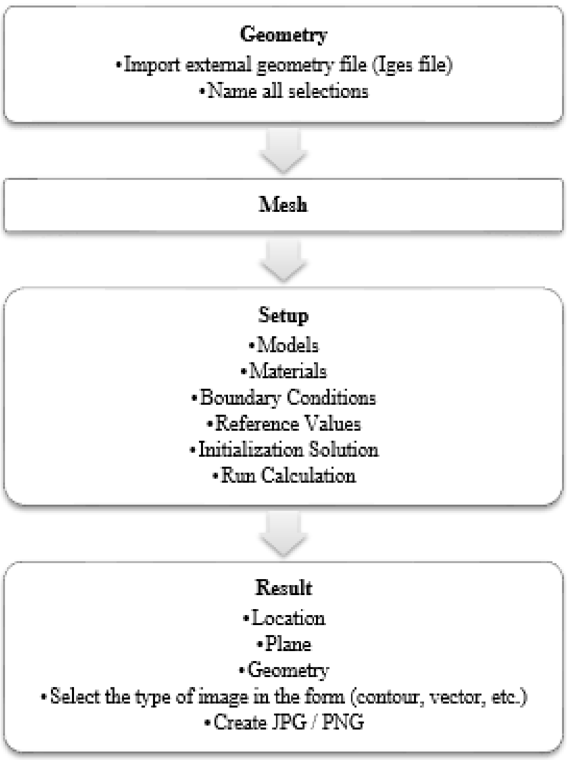


Figure 3. Simulation steps.

2) Solving: Running Calculation

In the stage of solving, selecting the type of calculation will be used. The type of calculation is chosen based on in general, a case of the object model analysis, which is already in 1 part or 1 body condition (for modeling from 3D Autocad), for Gambit modeling. Material settings, water names, and solids from the fieldset, how much radiation and thermal characteristics of the material. In the case of interior objects, in general, use the calculation type viscosity with K-ε epsilon RNG model because it is more stable [9]. Determination of position and type of inlet, outlet as outflow and input value of variable speed at the inlet. It is important to namely repetition of calculations so that convergent data obtained.

3) Post Processing: Result

This stage is the result of the calculation. The value of the variable generated from the calculation is the value of wind speed and air pressure. Analysis of the calculation results done in 2D or 3D mode. ANSYS has features that can produce visualization forms such as contour, streamline, wind direction, and graphics.

III. RESULT AND DISCUSSION

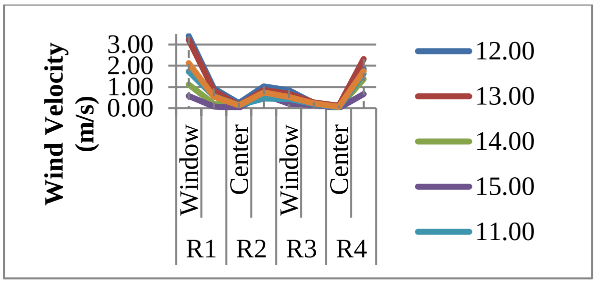


Figure 4. Result of field measurements.

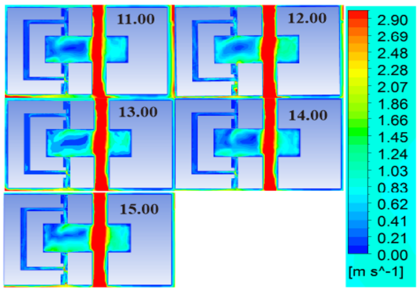


Figure 5. Wind velocity results from the CFD simulation.

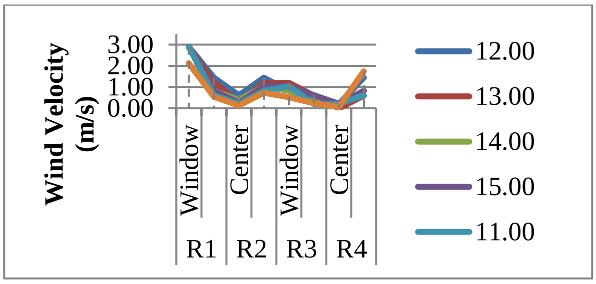


Figure 6. Results of CFD simulation.

In this study, 5 hours chosen with the hottest outdoor setting in 1 week of field measurement, which is from 11:00 to 15:00. The results of indoor wind measurements seen in Figure 4 and Figure 5 wherein the graph shows the fluctuations in wind movements from 11.00 where the wind flow increases, and after that, it gradually decreases at 3:00 p.m. Wind speed conditions in each unit have the same pattern. The maximum speed in the window area is 3.22-3.40 m / s, while for the middle area the maximum speed is 0.83-0.94 m / s. Figure 6 shows a graph of the results of wind speed measurements at the hottest hour of the day, where the window area produces greater wind flow with an average of 0.53-2.12 m / s compared to the center area of the unit with an average of 0.07-0.54 m / s. The natural resistance mechanism that occurs in buildings is crossing. Wind flow originated from outside the building, entered into the unit through the window and went out through the door to the corridor.

Based on verification, the simulation results have a higher value than the field measurements. The results of the verification on the five hours is showing in Figure 7. The higher value of the result can be caused by the simplified verification model in many forms, compared to the field measurement model that contains elements that inhibit wind flow, such as the presence of selling, in addition to the balcony and openings verification model in the unit only in the verification model, for units - the surrounding unit was abolishing. In conditions on the balcony field and openings in other units can inhibit the flow of the wind.

Based on the results of the average field measurements and simulations, explained in the bar chart -, the results show that wind velocity in the window area of units R1, and R4 have a greater value than other units, which are models of research verification. Wind speed in windows that do not face the courtyard has a greater average value, above 1.50 m / s because it gets wind flow directly from outside the building while the window facing the courtyard where the wind flow is low, below 1.00 m /s. Based on the results of verification, the value of R^2 from the measurement and simulation is 0.67, which is explaining in Figure 8. Classification of regression values, as mentioned [10] shows the classification of regression values between simulation results. Based on Table 3, the regression value is 0.67 and indicates that the simulation results and field measurements are quite good.

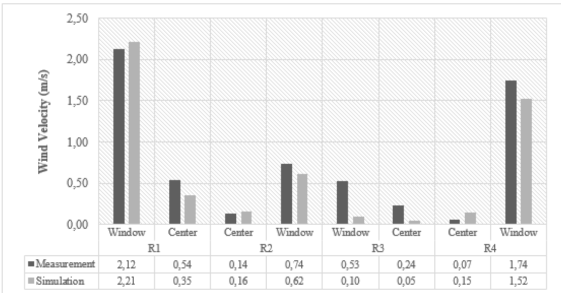


Figure 7. Wind velocity results from the simulation.

Tabel 3.
Classification of Regretion Values

Regression Value	Relations between Variables
$R^2 < 0.3$	The relationship or influence between variables is very weak
$0.3 < R^2 < 0.5$	Relations or influence between variables is weak
$0.5 < R^2 < 0.7$	The relationship or influence between variables is sufficient
$R^2 > 0.7$	The relationship or influence between variables is very strong

The velocity value for physiological cooling on the gradient is the average speed ratio of the room with comfort speed (Velocity comfort) with the following information:

1) V / Vc = 1, the V value of the room meets the needs
2) V / Vc >> 1, the condition of the room does not require wind speed for cooling
3) V / Vc << 1, wind speed cannot meet the speed requirement for physiological cooling.
following;
4) Quadrant A; speed values for physiological cooling are met, but cannot meet the required ACH standard values.
5) Quadrant B; ACH value and speed for physiological cooling are fulfilled. This condition is optimal.
6) Quadrant C; cannot fulfill ACH values and wind speed for physiological cooling.
7) Quadrant D; ACH values are fulfilled, but not at wind speeds for cooling.

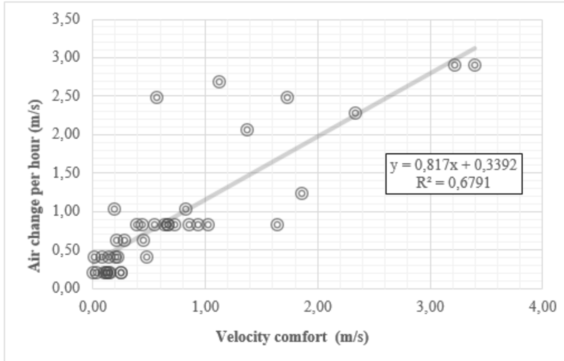


Figure 8. Comparison of regression result between the field measurement and CFD simulation.

Based on the results of verification, the value of R^2 from the measurement and simulation is 0.67, which is explained in Figure 8. It shows the classification of regression values between simulation results. Based on Table 3, the regression value is 0.67 and indicates that the simulation results and field measurements are quite good. [10] states that the comfort limit for equatorial conditions ranges from 19 ° C TE-26 ° C TE with the following division: temperature 26 ° C TE: Generally residents have started sweating. Temperature 26 ° C TE-30 ° C TE: Residents' resistance and workability begin to decline. Temperature 33.5 ° C TE-35.5 ° C TE: Environmental conditions begin to be uncomfortable. Temperature 35 ° C TE-36 ° C TE: Environmental conditions are no longer possible.

IV. CONCLUSION

This experiment shows the CFD simulation software is capable of providing an adequate result for analyzing wind velocity in a high-rise building. The current experiment has done to the building with 30 fl or equal to 102 m. Besides, the software does not only provide the value of wind velocity but also the contour area, which shows the pressure and the velocity of the wind. Base on the study, the wind velocity windows area is higher compared to the center area of the building.

Based on the results of validation, the value of R^2 from the measurement and simulation is 0.67. This value shows the influence between variables is sufficient.

Nevertheless, the strong influence should have a greater value than 7. The current results are affected by the reduction of the building elements due to the limitation of the CFD simulation software.

The current software validation is used to support the study on finding the passive cooling alternative. The next step is to apply the current method in the improvement model. In addition to the wind velocity, velocity comfort also determines by the temperature of the building. This CFD simulation, however, does not provide the temperature. Therefore, the next study requires other software such as Design-Builder.

REFERENCES

[1] Y. Rikugun Sanbō Honbu Rikuchi Sokuryōbu. and 陸軍参謀本部陸地測量部., *Ventilasi Solar Chimney sebagai Alternatif Desain Passive Cooling di Iklim Tropis Lembab*, vol. 3, no. 1. Dainihon Teikoku Rikuchi Sokuryōbu, 1923.

[2] Shuzo Murakami and Shinsuke Kato, "New Scales for Ventilation Efficiency and Calculation Method by Menas of 3-Dimensional Numerical Simulation for Turbulent Flow: Study on Evaluation of Ventilation Efficiency in Room," 空気調和・衛生工学会 論文集, vol. 11, no. 32, pp. 91–102, Oct. 1986.

[3] S. Murakami, "The role and application of ventilation effectiveness in design," *proceeding Int. Symp. room air Convect. Vent. Eff.*

[4] K. . Papakonstantinou, C. . Kiranoudis, and N. . Markatos, "Numerical simulation of air flow field in single-sided ventilated buildings," *Energy Build.*, vol. 33, no. 1, pp. 41–48, Nov. 2000.

[5] Linda Groat and DAvid Wang, "Architectural Research Methods."

[6] J. Revuz, D. M. Hargreaves, and J. S. Owen, "On the domain size for the steady-state CFD modelling of a tall building," *Wind Struct.*, vol. 15, no. 4, pp. 313–329, Jul. 2012.

[7] A. Elshaer, G. Bitsuamlak, and A. El Damatty, "Enhancing wind performance of tall buildings using corner aerodynamic optimization," *Eng. Struct.*, vol. 136, pp. 133–148, Apr. 2017.

[8] M. R. Fahmi, I. Defiana, and I. G. N. Antaryama, "Cross Ventilation in High-Rise Apartment Building: Effect of Ventilation Shaft Aperture Configuration on Air Velocity and Air Flow Distribution," *IPTEK J. Proc. Ser.*, vol. 4, no. 1, p. 65, Jan. 2018.

[9] F. Muhsin, W. F. M. Yusoff, M. F. Mohamed, and A. R. Sapian, "CFD modeling of natural ventilation in a void connected to the living units of multi-storey housing for thermal comfort," *Energy Build.*, vol. 144, pp. 1–16, Jun. 2017.

[10] D. S. Moore, W. L. Notz, and M. A. Flinger, *The Basic Practice of Statistics*. 2013.