Progress in Engineering Application and Technology Vol. 2 No. 1 (2021) 453-463
© Universiti Tun Hussein Onn Malaysia Publisher's Office

PEAT

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/peat
e-ISSN : 2773-5303

Study Usage of Computational Fluid Dynamics (CFD) for Indoor Simulation

Nurul Filzati Jamaludin[1], **Fatimah Mohamed Yusop**[1]*, **Nurdalila Saji**[1]

[1]Department of Civil Engineering Technology, Faculty of Engineering Technology, UTHM Kampus Pagoh, Johor, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/peat.2021.02.01.045
Received 13 January 2021; Accepted 01 March 2021; Available online 25 June 2021

Abstract: Computational fluid dynamic (CFD) is an important tool that capable of solving fluid flow problems and estimating physical fluid flow and heat transfer. By using CFD, air flow can be simulated to accurately design spaces that provide the required flow and velocity. The wrong choice of a numerical method and a gridding technique in the solution of CFD problems it will give effect to get the accurate results. Thus, the aim of this study to investigate the application of CFD for indoor simulation and to identify the suitable turbulence model for indoor simulation by the lower percentage difference. The studies used information from the previous research that used CFD simulation. The result shows the most turbulence model that be used is realizable k-ε with the percentage difference less than 10.00 %. So that, realizable k-ε is the suitable turbulence model for indoor simulation with lowest percentage difference, occupy less computational resources and minimize computing time. Realizable k-ε model is recommended and the solution accuracy is higher than the others. In addition, Ansys Fluent is the most widely used code in indoor simulation because it is more user-friendly due to its user-defined functionality

Keywords: Indoor, Computational Fluid Dynamic, Turbulence Model, Velocity, Temperature

1. Introduction

Conventional heating, ventilation and air conditioning (HVAC) technologies are used to maintain a uniform indoor climate in line with design requirements. However, some inhabitants are more susceptible to local air flow, temperature and relative humidity due to human tastes as well as differences in clothing and activity levels. As a result, thermal irritation occurs for a minority of occupants, particularly though the thermal environment complies with the standards' guidelines. In addition, sign of sick of building syndrome have resulted in greater attention to inhaled air quality depending on human desires and expectations. A conceptual change from the design of standardized indoor environments to the design of independently managed environments is desirable, even in terms of energy usage in buildings. Several recent studies have shown that individually managed conditions

*Corresponding author: fatimahy@uthm.edu.my

combined with a wider range of thermostat settings for the indoor context result in greater thermal comfort and allow energy savings in buildings.

CFD is a computer-based mathematical simulation tool capable of solving fluid flow problems and estimating physical fluid flow and heat transfer [1]. CFD is used intensively as a tool to determine the interior atmosphere of the building and its relationship with the building envelope, as well as to assess the outdoor environment around the building. CFD also can be used to evaluate wind loading on houses, bridges, and street canyons to forecast future wind flows. CFD was used to test possible natural ventilation, mixed-mode ventilation and HVAC systems in houses, which typically require the measurement of air temperature, velocity and relative humidity, among other parameters [2]. The CFD research is more useful than the experimental method since certain similar interactions between variables in post-processing can be found here. Within houses, internal air flow has also been modelled, also with obstructions to the flow such as furniture and people. With little computing work it is easy to generate a detailed picture of the airflow.

1.1 Problem Statement

There are problems related to CFD simulation, among them is the error of applying CFD simulation in a building or system. Many indoor problems associated with the industry involve the understanding and analysis of air flow. Typical examples include ventilation, climate, air quality and thermal comfort. The wrong choice of a numerical method and a gridding technique in the solution of CFD problems. So that, it cannot get the accurate results. Both experimental work and numerical models can provide the necessary information for solution of any particular problem. This study is more on indoor because the outdoor wind condition is extremely unpredictable, airflow infiltration is unstable with a wide variety of fluctuation frequencies. In temperature also have difference between outdoors and indoors, it can be very significant specifically in winter or summer. Thus, the aim of this study to investigate the application of CFD for indoor simulation. And to identify the suitable turbulence model for indoor simulation by the lower percentage difference. The studies used information from the previous research that used CFD simulation.

1.2 Objectives

The objective of this study are:

- To investigate the application of CFD for indoor simulation
- To identify the suitable turbulence model for in indoor simulation

1.3 Scope of Study

The scope of this study are:

- Gathering information from the previous research that used Computational Fluid Dynamic (CFD) for indoor system.
- Comparison of CFD code software, turbulence model and percentage difference between CFD data with experiment data.

1.4 Turbulence Modelling

The development and use of a mathematical model for forecasting the result of turbulence. Despite decades of study, there is no theoretical hypothesis to model the evolution of these turbulence flows. In order to realistically model a given problem, it is important to identify the inlet turbulence intensity. Here are a few examples of typical estimates of the incoming turbulence intensity:

i. High turbulence (between 5.00 % and 20.00 %): cases of high velocity flow inside complex geometries. Examples: heat exchangers, flow in spinning equipment such as fans, motors, etc.

ii. Medium-turbulence (between 1.00 % and 5.00 %): flow in not-so-complex geometries or low speed flow. Examples: wide pipe flow, airflow flow, etc.

iii. Low turbulence (well below 1.00 %): situations of still or extremely viscous fluids, very high efficiency wind tunnels. Examples: outward flow through vehicles, submarines, aeroplanes, etc.

There are many turbulence models such as standard k-epsilon, renormalization-group (RNG) k-epsilon, realizable k-epsilon, standard k-omega, shear-stress transport (SST) k-omega, Reynolds stress model (RSM) and large eddy simulation (LES) models. There is a trade-off between accuracy and computational costs for each of these models as shown in Figure 1.

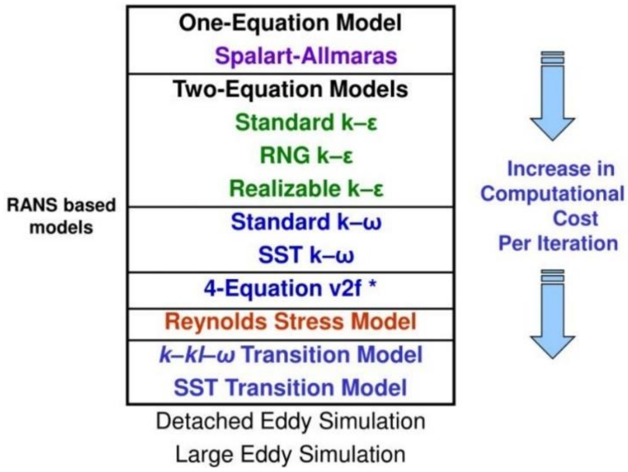

Figure 1: Turbulence model overview

Reynolds-Averaged Navier Stokes (RANS) has equations that are resolved for the time-average flow activity and the frequency of turbulent variations. Based on Figure 1, Spalart-Allmaras is a low-cost RANS model that solves a transport equation for a modified eddy viscosity. Represents a comparatively recent class of one-equation models where it is not appropriate to measure the length scale of the local shear layer thickness. Then, the k-epsilon model has three types, which are standard k-epsilon, RNG k-epsilon and realizable k-epsilon. The higher cost of computation between three of them is the realizable k-epsilon. It accurately calculates the spread rate of both flat and circular jets. It is also likely to have superior results compared to the standard k-epsilon model for rotational flows, boundary layers under high adverse pressure gradients, separation and recirculation.

Large Eddy Simulation (LES) has definitely the ability to produce more accurate and reliable results than the Reynolds-Averaged Navier-Stokes (RANS) method simulations. Fortunately, LES has a higher simulation complexity and a much higher computational cost. LES is at least theoretically more accurate and consistent than RANS since more of the flow is resolved. However, the higher simulation complexity makes it easier to ruin an LES than a RANS simulation. LES could provide a direct prediction of the turbulence intensity, but for the fully formed flow, the prediction accuracy of the average flow variables was not enhanced, although LES could provide advantages for the unfulfilled turbulent flow.

2. Materials and Methods

The methodology process was explained in more detail on the methods for carrying out the analysis on variety of main points of the process flow such as the tools, procedures and necessary equipment for collecting and evaluating the result. In addition, these preliminary studies would analyse the data from the result of previous research on tools analyse air flow in indoor system. Figure 2 shows a research methodology in which the study flow was considered in this project.

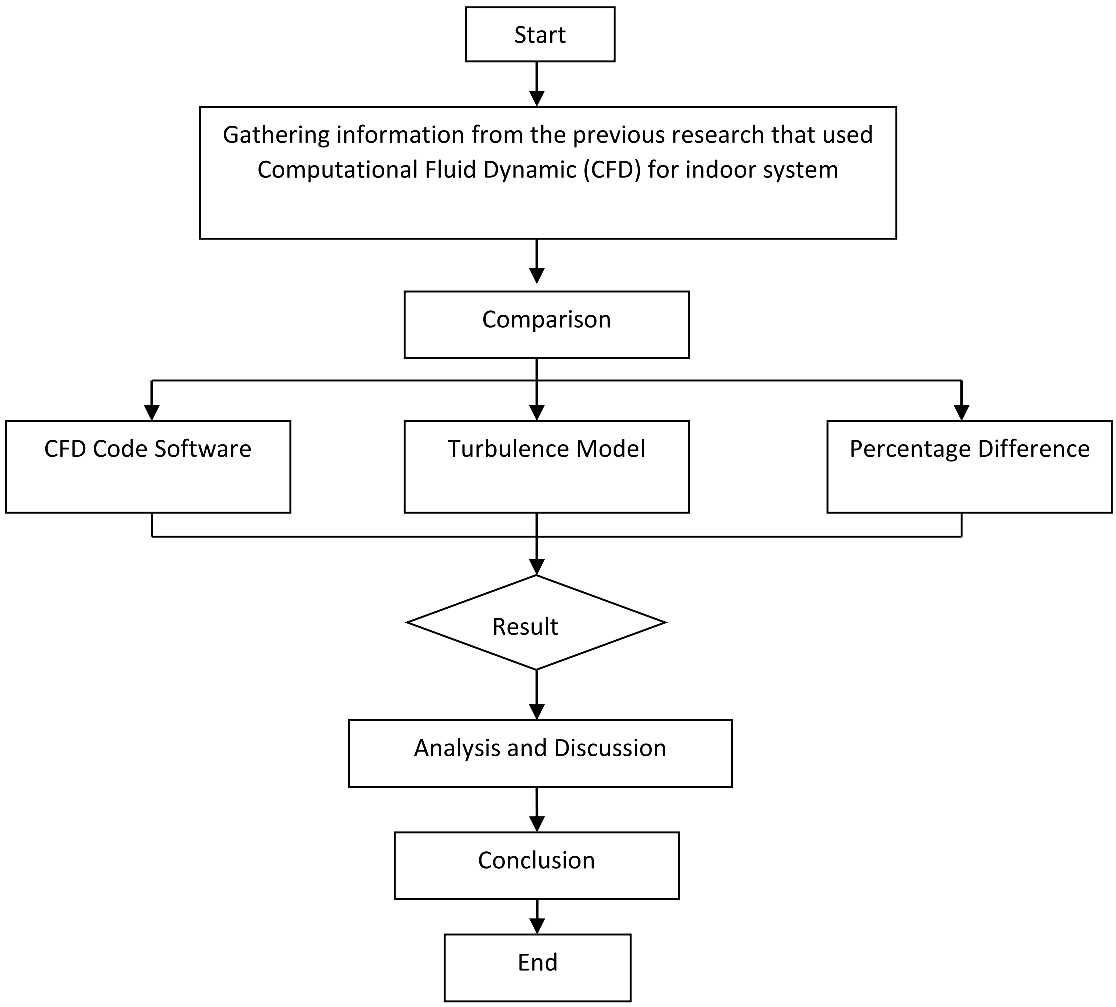

Figure 2: The flowchart of methodology

2.1 Gathering information from previous research

There are 40 previous research was investigated within the scope of the review of this study. The research was from any sources that have related to the indoor system. Besides, all the research have similar method which is using CFD simulation as well as the topic discussed.

2.2 Comparison

The CFD code and turbulence model was different for the previous research and had a similar result which is to get the data simulation of air flow. However, the choice of CFD code and turbulence model was belongs to the researchers on how they collect data and the computer simulation for analysis the result. All the research had a different percentage in the difference between data simulation and experiment data. At the same time, the comparison was made to gain the suitable turbulence model.

3. Results and Discussion

It will discuss more about the condition of air flow which is correlated to the air flow content in the building or the system. Apart from this, the result and data analysis between 40 research were compared through their methodology and result from data. But only 25 of 40 research had a difference percentage between simulation data and experiment data because they do not compare with the experiment data.

3.1 Summary of selected articles

Table 1 shows the selected research was brief in summary about the topic content. It is one of the methods used to provide a concentrated overview of the completed research study presented in a community academic journal. In addition, this research summary gives prospective readers a short, clear commentary and offers insight into the subject of articles, CFD code, turbulence model and percentage difference.

Table 1: Summary of the articles that has been summarized

Articles	CFD Code	Turbulence Model	Percentage Difference (%)	
Lin. S, Tee. B. T, Tan. C. F. (2015) [3]	Fluent	RNG k-ε	16.7%	>10
Malek. N. A, Khairuddin. M. H. (2015) [4]	Fluent	Standard k-ε	13.11%	>10
Sun. Z, Wang. S. (2009) [5]	TRNSYS	RNG k-ε	20%	>10
Cheng. L. Y. (2014) [6]	Fluent	Standard k-ε	< 20%	>10
Gao. C (2011) [7]	Airpak	RNG k-ε	4.8% - 25.4%	>10
Chiang. W, Wang. C, Huang. J. (2011) [8]	Phoenics	Standard k-ε	0.99% - 1.05%	<10
Ng. J, Navarednam. S, Wong. K. Y, Chong. C. T. (2019) [9]	Ansys Fluent	RNG k-ε	3.45%	<10
Chung. L. P, Ahmad. M. H, Ossen. D. R, Hamid. M. (2014) [10]	Not specified	Standard k-ε	7.2% - 18%	>10
Gilani. S, Montazeri. H, Blocken. B (2015) [11]	Ansys Fluent	SST k- ω	4.2% - 11.9%	>10
Chen. Z, Xin. J, Liu. P. (2020) [12]	Ansys Fluent	Standard k-ε	10% - 40%	>10
Calautit. J. K, Hughes. B. R. (2014) [13]	Ansys Fluent	Standard k-ε	12%	>10
Zhang. Y, Kacira. M, An. L. (2016) [14]	Ansys Fluent	Rk-ε	7.5%	<10
Aryal. P, Leephakpreeda. T (2015) [15]	Not specified	Not specified	<10%	<10
Bhowmick. S. (2019) [16]	Ansys Fluent	Standard k-ε	3.7%	<10
D'Agostino. D, Congedo. P. M. (2014) [17]	Fluent	Standard k-ε	1.5%	<10
Cheung. J. O. P, Liu C. H. (2010) [18]	Ansys Fluent	Standard k-ε	10% - 15%	>10
Tong. X, Hong. S. W, Zhao. L. (2018) [19]	Ansys Fluent	RNG k-ε	10% - 14%	>10
Muhsin. F, Yusoff. W. F, Mohamed. M. F, Sapian. A. R. (2017) [20]	Ansys CFX	Standard k-ε	<10%	<10
Amoresa. C. P, Mazarrón. F. R, Cañas. I, Sáez. P. V. (2019) [21]	STAR CMM	AKN, Rkε, Skε Low, Skε Two, v2-f	5% - 10%	<10
Shan. X, Xu. W, Lee. Y.K, Lu. W. Z. (2018) [22]	Not specified	RNG k-ε	10.2%	<10
Maryanczyk. A. F, Schnotale. J, Radon. J, Was. K. (2014) [23]	Ansys Fluent	Rk-ε	0.62%	<10

Turcanu. F. E, Popovici. C. G, Verdes. M, Ciocan. V, Hudisteanu. S. V. (2020) [24]	Ansys Fluent	Standard k-ε, k-ω	<10%	<10
Blázquez. J. L. F, Maestre. I. R, Gallero. F. J. G, Gomez. P. A. (2017) [25]	Ansys Fluent	Standard k-ε	3.3% - 9.3%	<10
Song. J, Meng. X. (2014) [26]	Ansys Fluent	Standard k-ε	7.4%	<10
Hong. B, Qin. H, Jiang. R, Xu. M, Niu. J. (2018) [27]	Not specified	Standard k-ε	26.21%	>10

3.2 Comparison of CFD Code Software

CFD analysis software programmes are practically infinite. CFD programmes are commonly used in the aerospace, industrial and shipbuilding industries. Figure 3 indicates that of all the publications analysed in this report, 55.00 % of the studies used ANSYS Fluent software (including the early version, called Fluent). Other applications widely used include STAR-CCM+, Airpak, PHOENICS, CFX, and TRNSYS. The study compared ANSYS Fluent and CFX, which are currently the most common, to predict indoor air parameter distribution. The study observed that the two programmes achieved nearly the same results with similar computing effort and stressed that ANSYS Fluent was more user-friendly due to its user-defined functions.

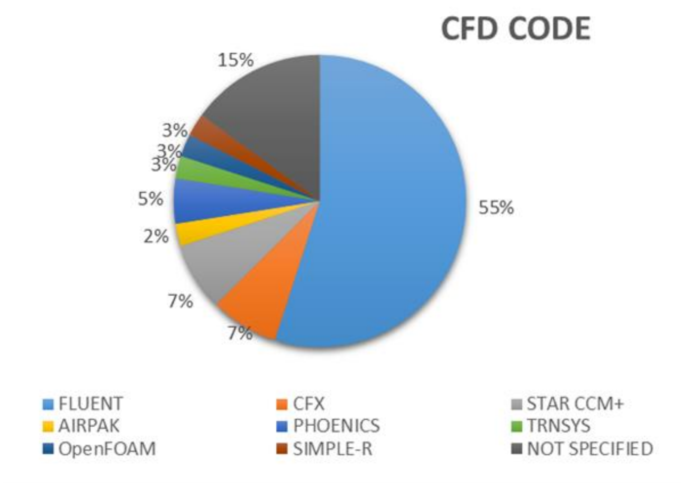

Figure 3: CFD code distribution of the research

3.3 Comparison of Turbulence Model

Primarily, indoor studies use RANS methods. Three of the most widely used RANS turbulence models are the standard k-ε turbulence model (SKE)(44.00 %), renormalization group k-ε turbulence model (RNG)(19.00 %) and realizable k-ε turbulence model (RKE)(9.00 %). Shear stress transport k-ω (SST) was used in one study, v2f turbulence model in two studies and standard k-omega (k-ω) turbulence model in one study. Figure 4 present the other of the turbulence models used in studies of indoor systems.

The k-ε model is the most commonly used and proven turbulence model for applications of any simulation which is explains its success. It is developed primarily for flat shear layers and recirculating flows. It is typically useful for free-shearing layer flows with relatively small pressure gradients as well as in confined streams where the Reynolds shear stress is most important. It can also be defined as the simplest turbulence model for which only initial and/or boundary conditions are needed.

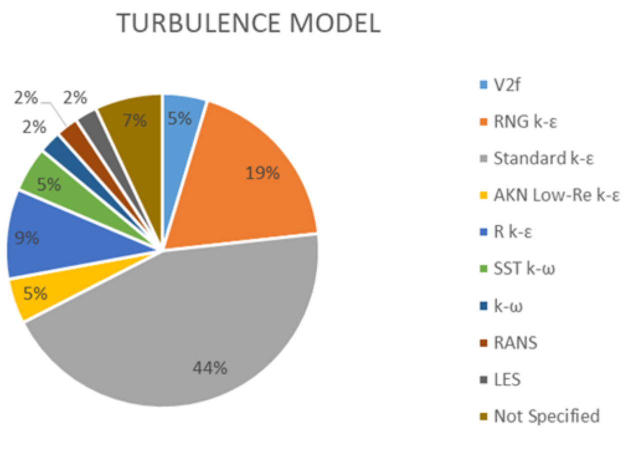

Figure 4: The turbulence model distribution of the research

3.4 Comparison of Percentage Difference

A comparison of the CFD result with the experimental results was needed for the reporting of the CFD analysis. Based on Figure 4, the most researcher preferred to use standard k-ε but the most accurate is realizable k-ε due to less computing time and the lowest percentage difference. In Figure 5 indicated the percentage of realizable k-ε model used with percentage difference less than 10.00 % is 100.00 %. The percentage of standard k-ε model used with percentage difference less than 10.00 % is 53.00 % and for percentage difference more than 10.00 % is 47.00 %. Standard k-ε model and realizable k-ε models have been shown to produce good results for various applications and to provide a good balance between accuracy and computational effort. But in order to satisfying the need for accuracy, the need for engineering simulation has been to occupy less computational resources and to minimize computing time. Realizable k-ε model is recommended and the solution accuracy is higher than the others. In nearly any measure of comparison, the realizable k-ε indicates a superior capacity to capture the mean flow of complex structures. It also the higher in computational cost per iteration and has a low -speed flow.

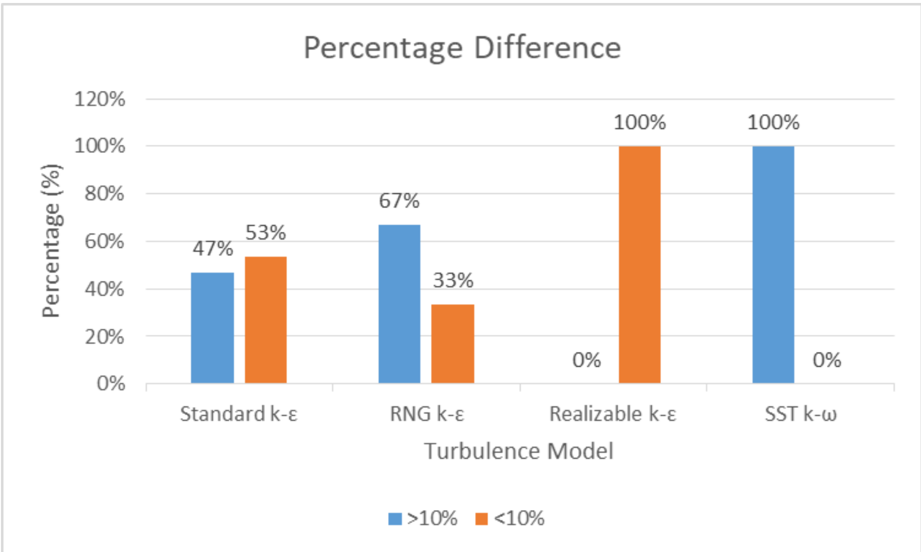

Figure 5: The percentage difference in less than 10.00 % and more than 10.00 % for each model

3.5 Discussion

It will discuss more about the condition of simulation which is correlated to the air flow content in the building or system. Regarding the results, all the 40 research were using the same methodology which is CFD simulation but using the different code and turbulence model. CFD simulation is the important tools in application for indoor simulation. Based on all previous studies, it stated the data from CFD simulation is the better agreement and acceptable than the experimental. According to the result of comparison, there are most of studied used the ANSYS Fluent software because it efficient and flexible workflows. It also provide a wide range of turbulence models to capture the effects of turbulence accurately and effectively.

For the type of turbulence model, almost half percent (44.00 %) used the standard k-ε models. It is because standard k-ε models predicts well for isotropic (high Re) flows, basic flows, flat and radial jets (but not circular jets) and plumes. The most stable from the other k-ε model. Based on the comparing percentage difference between CFD data and experimental, it can conclude which one is suitable turbulence model with lowest percentage for indoor simulation. It show the realizable k-ε is the accurate turbulence model for indoor simulation because it had a lowest percentage difference, occupy less computational resources and minimize computing time compare to standard k-ε model. So that, realizable k-ε model is recommended and the solution accuracy and computational cost is higher than standard k-ε model.

4. Conclusion

In conclusion, through of the comparison made between forty previous research were responded to two objectives of this study which are to investigate the application of CFD for indoor simulation and identify the accurate turbulence model for indoor simulation. The result shows that majority of all researchers are using the Ansys Fluent as a CFD code software. Even it achieved nearly the same result computing effort and stressed with one another software, it still was more user friendly due to its user defined functions. The standard k-ε model is the most commonly used by the researcher but according to the result of comparing the percentage difference, it show Rk-ε is the accurate turbulence model for indoor simulation because it had a lowest percentage difference, occupy less computational resources and minimize computing time. Realizable k-ε model is recommended and the solution accuracy is higher than the others. This study had analysis the application of CFD for indoor simulation is a good option and it decreases dependency on experiments. It also can help in improved the air flow, ventilation and thermal comfort in buildings or systems.

4.1 Recommendation

Based on this study, there are recommendation need to be improved for the next study approach. First of all, the method and tool used must be described and clarified more in detail on the basis of its specification and the specification of the requirements or part of the set used. This is because researchers need to know the state of the device based on the parameter measured which has a strong correlation to the object.

Second, the researcher must mention the standard used either by national or international standards for their case study. The details on the part of the standard used must be given to ensure that the reader knows enough about what they are attempting to offer. In addition, the standard must be in the latest version, which is the acceptable standard based on part categorization in the system.

In addition, the limitation of the study should be clearly indicated in terms of the type of CFD, the parameter used and the percentage difference. In the other hand, it will direct and help prospective researchers to work and study in the future. The content of the analysis can be delivered quickly. Lastly, the contributions of this study are to help and guide the potential researcher for making the decision on choosing the turbulence model for simulate the air flow in indoor system.

Acknowledgement

The authors would like to thank the Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] [1] Versteeg, H. K., & Malalasekera, W. (1995). An introduction to computational fluid dynamics: The finite volume method. Harlow: Longman.

[2] Bangalee, M., Lin, S., & Miau, J. (2012). Wind driven natural ventilation through multiple windows of a building: A computational approach. Energy and Buildings. 45, 317-325. doi:10.1016/j.enbuild.2011.11.025

[3] Lin, S., Tee, B. T., & Tan, C. F. (2015). Indoor Airflow Simulation inside Lecture Room: A CFD Approach. IOP Conference Series: Materials Science and Engineering, 88, 012008. doi:10.1088/1757-899x/88/1/012008

[4] Malek, N. A., Khairuddin, M. H., & Rosli, M. F. (2015). Thermal comfort investigation on a naturally ventilated two- storey residential house in Malaysia. IOP Conference Series: Materials Science and Engineering, 88, 012014. doi:10.1088/1757-899x/88/1/012014

[5] Sun, Z., & Wang, S. (2009). A CFD-based test method for control of indoor environment and space ventilation. Building and Environment, 45(6), 1441-1447. doi:10.1016/j.buildenv.2009.12.007

[6] Cheng, L. Y. (2014). Air flow buoyancy surrounding buildings in Malaysia. Faculty of Engineering, University of Malaya. Retrieved from http://studentsrepo.um.edu.my/id/eprint/8342

[7] G. C. (2011). The study of natural ventilation in residential buildings. Department of Building Services Engineering. Retrieved from https://theses.lib.polyu.edu.hk/bitstream/200/6301/1/b24625516.pdf.

[8] Chiang, W., Wang, C., & Huang, J. (2011). Evaluation of cooling ceiling and mechanical ventilation systems on thermal comfort using CFD study in an office for subtropical region. Building and Environment, 48, 113-127. doi:10.1016/j.buildenv.2011.09.002

[9] Ng, J., Navarednam, S., Wong, K. Y., & Chong, C. T. (2019). Flow and dispersion simulation using Computational Fluid Dynamics: A Case Study for EduCity in Iskandar Malaysia. IOP Conference Series: Earth and Environmental Science, 268, 012129. doi:10.1088/1755-1315/268/1/012129

[10] Chung, L. P., Ahmad, M. H., Ossen, D. R., & Hamid, M. (2014). Effective Solar Chimney Cross Section Ventilation Performance in Malaysia Terraced House. Procedia - Social and Behavioral Sciences, 179, 276-289. doi:10.1016/j.sbspro.2015.02.431

[11] Gilani, S., Montazeri, H., & Blocken, B. (2015). CFD simulation of stratified indoor environment in displacement ventilation: Validation and sensitivity analysis. Building and Environment, 95, 299-313. doi:10.1016/j.buildenv.2015.09.010

[12] Chen, Z., Xin, J., & Liu, P. (2020). Air quality and thermal comfort analysis of kitchen environment with CFD simulation and experimental calibration. Building and Environment, 172, 106691. doi:10.1016/j.buildenv.2020.106691

[13] Calautit, J. K., & Hughes, B. R. (2014). Measurement and prediction of the indoor airflow in a room ventilated with a commercial wind tower. Energy and Buildings, 84, 367-377. doi:10.1016/j.enbuild.2014.08.015 J

[14] Zhang, Y., Kacira, M., & An, L. (2016). A CFD study on improving air flow uniformity in indoor plant factory system. Biosystems Engineering, 147, 193-205. doi:10.1016/j.biosystemseng.2016.04.012

[15] Aryal, P., & Leephakpreeda, T. (2015). CFD Analysis on Thermal Comfort and Energy Consumption Effected by Partitions in Air-Conditioned Building. Energy Procedia, 79, 183-188. doi:10.1016/j.egypro.2015.11.459

[16] Bhowmick, S. (2019). Assessing The Impact Of Passive Cooling On Thermal Comfort In Lig House Using Cfd. Journal of Thermal Engineering, 414-421. doi:10.18186/thermal.623212

[17] Dagostino, D., & Congedo, P. M. (2014). CFD modeling and moisture dynamics implications of ventilation scenarios in historical buildings. Building and Environment, 79, 181-193. doi:10.1016/j.buildenv.2014.05.007

[18] Cheung, J. O., & Liu, C. (2010). CFD simulations of natural ventilation behaviour in high-rise buildings in regular and staggered arrangements at various spacings. Energy and Buildings, 43(5), 1149-1158. doi:10.1016/j.enbuild.2010.11.024

[19] Tong, X., Hong, S., & Zhao, L. (2018). Using CFD simulations to develop an upward airflow displacement ventilation system for manure-belt layer houses to improve the indoor environment. Biosystems Engineering, 178, 294-308. doi:10.1016/j.biosystemseng.2018.08.006

[20] Muhsin, F., Yusoff, W. F., Mohamed, M. F., & Sapian, A. R. (2017). CFD modeling of natural ventilation in a void connected to the living units of multi-storey housing for thermal comfort. Energy and Buildings, 144, 1-16. doi:10.1016/j.enbuild.2017.03.035

[21] Porras-Amores, C., Mazarrón, F. R., Cañas, I., & Sáez, P. V. (2019). Natural ventilation analysis in an underground construction: CFD simulation and experimental validation. Tunnelling and Underground Space Technology, 90, 162-173. doi:10.1016/j.tust.2019.04.023

[22] Shan, X., Xu, W., Lee, Y., & Lu, W. (2018). Evaluation of thermal environment by coupling CFD analysis and wireless-sensor measurements of a full-scale room with cooling system. Sustainable Cities and Society, 45, 395-405. doi:10.1016/j.scs.2018.12.011

[23] Shan, X., Xu, W., Lee, Y., & Lu, W. (2018). Evaluation of thermal environment by coupling CFD analysis and wireless-sensor measurements of a full-scale room with cooling system. Sustainable Cities and Society, 45, 395-405. doi:10.1016/j.scs.2018.12.011

[24] Flaga-Maryanczyk, A., Schnotale, J., Radon, J., & Was, K. (2014). Experimental measurements and CFD simulation of a ground source heat exchanger operating at a cold climate for a passive house ventilation system. Energy and Buildings, 68, 562-570. doi:10.1016/j.enbuild.2013.09.008

[25] Blázquez, J. L., Maestre, I. R., Gallero, F. J., & Gómez, P. Á. (2017). A new practical CFD-based methodology to calculate the evaporation rate in indoor swimming pools. Energy and Buildings, 149, 133-141. doi:10.1016/j.enbuild.2017.05.023

[26] Song, J., & Meng, X. (2015). The Improvement of Ventilation Design in School Buildings Using CFD Simulation. Procedia Engineering, 121, 1475-1481. doi:10.1016/j.proeng.2015.09.073

[27] Hong, B., Qin, H., Jiang, R., Xu, M., & Niu, J. (2018). How Outdoor Trees Affect Indoor Particulate Matter Dispersion: CFD Simulations in a Naturally Ventilated Auditorium.

International Journal of Environmental Research and Public Health, 15(12), 2862. doi:10.3390/ijerph15122862