ASSESSMENT OF TECHNIQUES FOR ESTIMATING INDOOR VENTILATION QUALITY BY USING CFD

AHMAD IKMAL HAKIM BIN MOHD HILMI

This report is submitted
in fulfilment of the requirement for the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

JUNE 2021

DECLARATION

I declare that this project report entitled "Assessment of techniques for estimating indoor ventilation quality by using CFD" is the result of my own work except as cited in the references

Signature : ....

Name : AHMAD IKMAL HAKIM BIN MOHD HILMI

Date : 27 / 7 / 21

APPROVAL

I hereby declare that I have read this project report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering (Thermal Fluid).

Signature : ....

Name of Supervisor : CHENG SEE YUAN

Date : 25/9/21

DEDICATION

This study is truthfully dedicated to my beloved parents who have always been my source of inspiration and gave me strength whom I thought of giving up, who continually provide their moral and financial support.

To my lecturers, family and friends who encouraged me into being determined in everything I do.

And finally, I dedicated this study to the Almighty God for guiding me until I have finished my project and also for providing me the greatest strength and motivation.

ABSTRACT

This study is done to determine the indoor ventilation quality in both theoretical and simulation techniques. Besides, in this study, there are two types of Computational Fluid Dynamics (CFD) that have been applied. This is because of variations of results that need to be present as expected. The study is of how full flow field method and domain decomposition method impact the flow rate throughout the building. By comparing the result to the experimental result would make sure the CFD simulation is in correct matter. The finding is evaluated in comparison to domain decomposition method for the full flow field. Without the computational domain, the method of domain decomposition can be simulated. This approach reduced significantly the simulation time, which results only in the interior building's internal flow. Based on the results, the volume flow rate, Q obtained by CFD are not so close to the theoretical results. The percentage of differences between CFD and theoretical values is 47.30% and 49.23%.

ABSTRAK

Kajian ini dilaksanakan adalah untuk menentukan kualiti pengudaraan dalaman bangunan dari segi teori dan simulasi. Selain itu, terdapat dua cara yang diaplikasikan daripada pengiraan dinamik bendalir ini. Hal ini untuk mendapatkan keputusan yang pelbagai seperti yang dianggarkan. Kajian ini juga adalah untuk mengetahui bagaimana cara medan aliran penuh dan penguraian domain dapat meninggalkan kesan kepada kadar aliran di dalam sebuah bangunan. Dengan membandingkan keputusan yang diperoleh dengan keputusan eksperimen dapat membantu menjadikan simulasi lebih baik. Keputusan yang diperoleh hendaklah dibandingkan antara kedua dua cara simulasi tersebut. Teknik penguraian domain dapat dijalankan tanpa kehadiran domain pengiraan. Cara ini juga dapat mengurangkan masa simulasi kerana hanya bahagian dalam bangunan yang terlibat. Berdasarkan keputusan kajian, kadar aliran volumetrik, Q daripada CFD tidak begitu hampir dengan keputusan daripada teori. Peratusan perbezaan antara CFD dan teori ialah 47.3% dan 49.23%.

ACKNOWLEDGEMENT

The progress of my study is all thanks to the guidance and assistance given by many individuals and groups. I extremely appreciate the help from them all along to complete my project. First and foremost, I would like to express my sincere gratitude and acknowledgement to my Projek Sarjana Muda (PSM) supervisor, Dr Cheng See Yuan from Universiti Teknikal Malaysia Melaka (UTeM). Without his guidance and knowledge that were given to me, I would be facing a lot of problems during the process of my research progress including the advanced knowledge and other report writing skills. He has also guided me to achieve the success of my PSM study in many aspects such as motivation and tips. Furthermore, I would also like to thank my Projek Sarjana Muda (PSM) panels, Dr Yusmady bin Mohamed Arifin and Prof Madya Dr Tee Boon Tuan for assisting me to accomplish this PSM report. They have been providing me a lot of additional knowledge and advice regarding my research.

CONTENT

LIST OF FIGURES

LIST OF ABBREVIATIONS

IAQ	Indoor Air Quality
SBS	Sick Building Syndrome
CFD	Computational Fluid Dynamics
RANS	Reynolds-Averaged Navier-Stokes
PIV	Particle Image Velocimetry
LES	Large Eddy Simulation
Sk-ε	Standard k-ε
Rk-ε	Realizable k-ε
RNG k-ε	Renormalization Group k-ε
Sk-ω	Standard k-ω
SST k-ω	Shear Stress Transport k-ω
RSM	Reynolds Stress Model
ABL	Atmospheric Boundary Layer
UDF	User Defined Function

LIST OF SYMBOLS

Q	=	volume flow rate (m^3/s)
A	=	area of smaller opening (m^2)
V	=	outdoor wind speed (m/s)
Δp	=	Pressure difference
C_d	=	discharge coefficient.
V_r	=	reference wind speed (m/s)
C_p	=	wind pressure coefficient at given position
ρ	=	density of air [kg/m^3]
P_w	=	pressure due to wind [Pa]
U_z	=	mean wind velocity at a specific high [m/s]
z	=	Height coordinate
$U*_{ABL}$	=	Atmospheric boundary layer friction velocity
Z_O	=	Aerodynamics roughness length = 0.025mm
k	=	Von Karman constant (0.42)
$\mathcal{E}$	=	Turbulence dissipation rate
I_u	=	Stream wise turbulence intensity
a	=	parameter range between 0.5 to 1.5
k	=	Turbulence kinetic energy
$C\mu$	=	Empirical constant (0.09)
ω	=	Specific dissipation rate

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Building ventilation can be defined as the ability of the ventilation to control the air flow and increase the ventilation quality inside a building or space. Building ventilation is crucial to humans because basically everyone is living or doing their daily activities in a specific building. For example, house, office, and school. Therefore, a good Indoor Air Quality (IAQ) is important to avoid the consequences on human health such as Sick Building Syndrome (SBS) (Abdull, 2013). Presently, many buildings would depend on mechanical ventilation rather than natural ventilation. However, to reduce their annual heating period, natural ventilation has more advantages to reduce the energy consumption for the ventilation. Natural ventilation should be increased because it provides better health and environmental concerns and used about 15% lesser cost compare to air- conditioned equivalent. (Cheung & Liu, 2011).

Good indoor ventilation quality can be defined as an environment that has ideal ventilation which is the implementation and supply of clean air, controlled contaminants and the humidity, and convenient temperature level. There are several ways for estimating indoor ventilation quality. One of them is by evaluating air distribution systems. Air distribution involves airflow and transport of heat and airborne pollutants. It is mathematically defined by a set of partial differential equations which is Navier-Stokes equations. However, these can only be solved for basic and optimum conditions. For complex geometry and/or complex boundary conditions, numerical methods can be used to calculate these equations or solve their modelling versions, given the initial and boundary conditions that is Computational Fluid Dynamics (CFD).

CFD solves the governing equations of fluid dynamics on a computer. It includes both spatial and temporal field variable solutions. For example, velocity and temperature. Besides, CFD can estimate the dispersal of contaminants in a building that is the computational domain. The CFD may also be used to forecast indoor air quality, thermal comfort, fire and smoke spread, and wind movement around buildings. The gravitational force effect may be important in the airflow in buildings. In the case of CFD, ventilation parameters may be modified depends on the situation.

For example, physical variables, supply openings and building geometry. There are two types of CFD simulations including full flow field and domain decomposition method.

The full flow field assumed that the construction within a computational field is in the domain inlet of the computation, able to view the flow in and out of the building. The airflow begins in the computer domain and flows into the building which is located within a range from the computer domain. The decomposition of the domain is analysed separately from the outside and inside. The simulation is first carried out on the outside of the building, and the boundary condition is set at the inlet of the computational domain.

1.2 PROBLEM STATEMENT

It is useful to see the velocities profile between the two methods that the airflow for the full flow field and the domain decomposition method can be distinct. The Ansys software offers various turbulence models to get the best settings by referring to previous researches. The differences between CFD simulation and theoretical equation could be outlined in the first estimation as follows in which that simulation is an outcome of creativity and a theory that occurs as a result of the application of a theory.

1.3 OBJECTIVE

The objectives of this project are as follow:

1 To measure the air distribution systems in an indoor environment to estimate the indoor ventilation quality.
2 To investigate the indoor air flow properties through CFD and non CFD simulation approaches by full flow field method, domain decomposition method and theoretical method.
3 To analyse the airflow properties in an enclosed space such as velocity distribution, air flow rate and temperature distribution for good indoor ventilation quality.

1.4 SCOPE OF PROJECT

The scopes of this project are:

1. This research focuses on the indoor ventilation quality which means only indoor environment or building is simulated by CFD modelling.
2. This research focuses on indoor ventilation quality and not indoor air quality. This narrows the scope of indoor ventilation quality properties to air flow properties such as air flow distribution and air velocity distribution.
3. Karava's Building model with the scale of 0.1m x 0.1m x 0.08m with ratio down to 1:200 is used.
4. The computational domain of 5L of the building model is being used while the downstream is at 10L of the building model.
5. Simulation of the full flow field and domain decomposition method and compared with the theoretical result.
6. The theoretical equations used are obtained from the previous researches such as volume flow rate equation.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter is emphasizing journals or any related articles to the CFD simulation of building. The previous work or knowledge done by many people could facilitate this study.

2.2 NATURAL VENTILATION

Natural ventilation is the deliberate passive movement of outside air through intended openings such as windows, louvers and doors into the building. Natural ventilation does not need mechanical applications to operate. Instead, it relies solely on passive physical phenomena such as the pressure of the wind or the impact of the stack. Natural ventilation openings can be adjusted accordingly. Adjustable openings can be operated automatically and manually or even both at the same time. Natural ventilation is a sustainable, energy-efficient and clean technology that is well accepted by occupants. It may be used to supply fresh air for humans. Besides, it is applied to cool buildings in acceptable situations. We could determine the applications of its techniques and also their effectiveness by the current external conditions and also the building design. Thus, its performance truly depends on the design process and needs to be designed together with the building (Heiselberg, 2004).

Air can be blown by wind through windows on the upwind side of the building. Air is then sucked out on the downwind side and the roof. The air could rise and withdraw through the ceiling and enter through lower windows. This is because of the temperature differences between the indoor air and outdoor air.

2.3 WIND

The air will flow based on the pressure which is from high pressure to low pressure. This effect is based on the principle of Bernoulli which uses differences in pressure to move the air. Natural wind pressure usually varies from 0.004 to 0.14 inches of water column (Bhatia, A., 2014).

The volume flow rate Q can be calculated using the formula below:

$$Q = C_d A \sqrt{\frac{2\Delta p}{\rho}} \qquad Equation\ 2.1$$

Where Cd is the discharge coefficient, A is the area, ρ is the density and Δp is the pressure difference which can be calculated by:

$$\Delta p = 0.5\rho C_p V_r \qquad Equation\ 2.2$$

Where Vr is the reference wind speed and Cp is the pressure coefficient at the opening.

2.4 AIRFLOW

Airflow can be defined as the air movement from one place to another. The presence of air is the major factor of airflow. Just like fluid, the particles of air flow naturally from areas with high pressure to lower pressure areas. Airflow can be applied mechanically or passively. Air is able to move through the building by the differential pressure of indoors and outdoors, this can be created through natural force (wind-induced pressure difference). Figure 2.1 shows the airflow through the building with 0° of wind. Cases for air flow to occur are a difference of pressure between two places and a continuous way or opening connecting the points (Straube, 2008). C_d depends on angle of incidence of wind such as for 0°equal to 1, 45° equal to 0.4 and 90° equal to 0.8 (Bhatia, A., 2014).

The rate of wind driven airflow through the opening in the building can be calculated by using:

$$Q = C_d AV \qquad Equation\ 2.3$$

where,

1. Q = Volumetric flow rate through the opening (m^3/s)
2. A = Free area of inlet opening (m^2)
3. V = Air velocity leaving the opening, (m/s)
4. Cd = Discharge coefficient

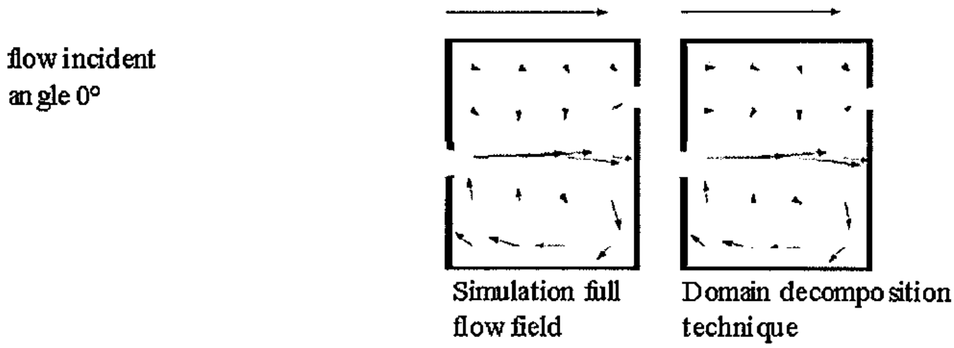

Figure 2.1: The flow rate of air through the building at full flow field and domain decomposition technique. (Kurabuchi, 2009)

2.5 AIRFLOW PRESSURE

The air pressure can be defined as pressure that the air exerts on any surface in contact with it, arising from the collisions of the air molecules with the surface (Deland & Kessler, 2019). The air pressure at a given location is called a force exerted in all directions on the based on the weight of the air above it (Tiwari, 2018). When the wind hits the building wall, high pressure is formed at the upwind face and there is a low pressure region behind the building which is called the downwind façade (Charisi, Thiis & Aurlien, 2019). Besides, it is also the force that balances the weight of the mercury column in the Torricellian barometer (Deland & Kessler, 2019). Full- scale and wind tunnel experiment is the best way to get the most accurate reading of wind pressure coefficient on current technology. The wind pressure that flows over the building can be defined as the wind pressure coefficient Cp (Charisi, Waszczuk & Thiis., 2017). The pressure coefficient can be expressed as:

$$P_w = \frac{1}{2}\rho {U^2}_z C_p \qquad \textit{Equation 2.4}$$

where,

1. Pw = pressure due to wind [Pa]

2. ρ = density of air [kg/m^3]

3. Cp = wind pressure coefficient at given position

4. Uz = mean wind velocity at a specific high [m/s]

Based on American Standard, wind pressure can be calculated by using

$$P_w = 0.00256 \text{ x } U_z^2 \qquad \textit{Equation } 2.5$$

The constant 0.00256 reflects an air mass density of 0.0024 slugs/ft3 (1.2255 kg/m3) and constants required to convert mph to ft/s. Basic wind speed V is expressed in terms of fastest-mile speed (mph) at 10 m above ground in flat open country. The basic wind speed has an averaging time of 3600/V sec and an annual probability of occurrence of 0.02 (Das, N. K., 1985)

2.6 COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics (CFD) can be defined as the proven field of fluid mechanics in which fluid flow problems are solved and analyzed using computational methods and numerical algorithms (Udoewa & Kumar, 2009-2011). Besides, CFD also means a method that applies physics, applied mathematics and computer science in order to simulate and analyze the fluid flow (Thabet & Thabit, 2018). Figure 2.2 shows the overall process of CFD. CFD used the Navier-Stokes equation as the governing equation. After the simulation, the simulation results will be compared to the experimental result (Zuo, 1981).

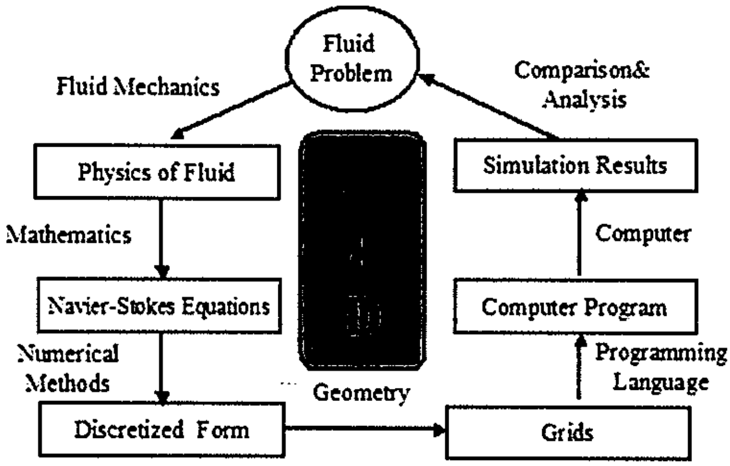

Figure 2.2: The overall process of CFD (Zuo, 1981)

2.6.1 GEOMETRY OF BUILDING MODEL

Based on Karava's research, the building model is assumed to be a 4 or 5 stories building. The full scaled dimension is estimated at 20m x 20m with 16m height. However, it has lowered the scale to a ratio of 1:200 which resulted to 10 x 10 x 8cm of flat roof building model. The model is used for the cross- ventilation as well as the atmospheric Boundary Layer Wind Tunnel (Karava, 2008). Furthermore, the model is used with different opening positions to test the experimental result (Particle Image Velocimetry (PIV)). The opening area is also known by wall porosity which can be calculated by dividing the opening area over the wall area. Different wall porosity also one of the variables of this experiment (Karava & Stathopoulos, 2012). Figure 2.3 shows the different opening positions of the building model. For complex urban conditions, model validation must be applied when measurements are usually not available. This is done for simpler configurations (Blocken, 2011). The measurement data are generally available in other simpler cases. So, it is noticed that steady RANS is the most suitable method while LES is considered to be not preferable (Tominaga, 2005). Figure 2.4 shows part of geometry and computational grid for CFD study of pedestrian-level wind conditions in the Shinjuku Sub-central area in Tokyo, Japan while figure 2.5 states the part of geometry and computational grid for CFD study of pedestrian-level wind conditions around the Amsterdam "ArenA" football stadium in the Netherlands (Blocken & Persoon, 2009).

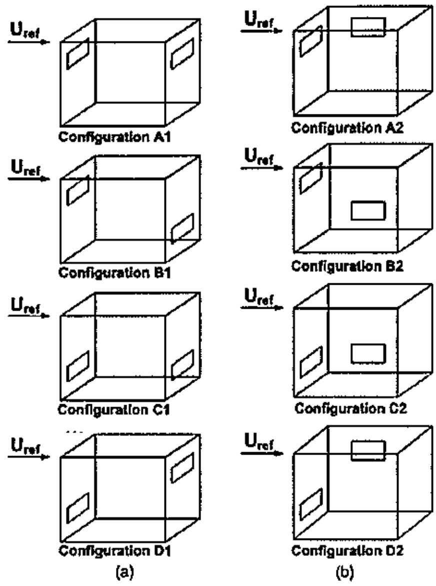

Figure 2.3: Different opening considered for the studying the effect of wall porosity and opening location on internal pressure (Karava & Stathopoulos, 2012).

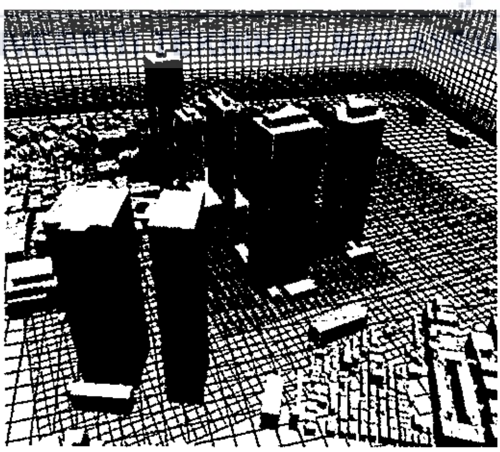

Figure 2.4: Part of geometry and computational grid for CFD study of pedestrian-level wind conditions in the Shinjuku Sub-central area in Tokyo, Japan (Tominaga, 2005).

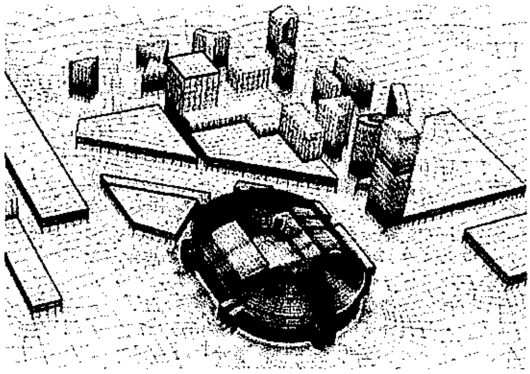

Figure 2.5: Part of geometry and computational grid for CFD study of pedestrian-level wind conditions around the Amsterdam "ArenA" football stadium in the Netherlands (Blocken & Persoon, 2009).

2.6.2 COMPUTATIONAL DOMAIN

The computational domain able to represent the geometrical and use for the boundary condition imposition. The domain must cover up all the physical features of the model (Peles, 2014). For the building model, the computational domain for calculation of external and internal flow is a cuboid shape region that is covered up the building with 50cm tall, 150cm long and 100cm wide. The building is centered 50cm downwind from the entrance (Meroney, 2009). The computational domain must be sufficiently large to capture all the flow around the building. To reduce the simulation time, the full sizing of the model is cut into half using the YZ plane due to both sides are symmetrical (Yuan & Yang, 2020). Based on Yuan C. S. (2007), the suggested domain size was 5L (L= Length of the building) upstream and 10L downstream, 5L away from each side, and 5L from the roof.

2.6.3 MESHING

The structured mesh is being used as the model can be easily sliced into an evenly distributed shape. The number of elements is approximately at 297,724 cells. Changing the number of divisions will create finer cells (Yuan & Yang, 2020). Figure 2.6 shows meshing with structured cell (Yuan & Yang, 2020). The computational grid was fully based on van Hooff and Blocken. In order to control the cells in the surroundings of the building model, a maximum stretching ratio of 1.2 is applied whereby the transition from the small thickness of the walls (2 mm) to the larger scales in the domain happens (Blocken & Ramponi, 2012).

Based on Ramponi and Blocken (2012) the grid would have 575,247 hexahedral cells. The cell size may increase gradually when moving far away from the building because this able to reduce the simulation time and the details far away from the building are not that important compare to the one near the model. The number of cells can be increased by changing the coarse grid to the fine grid (Blocken & Ramponi, 2012).

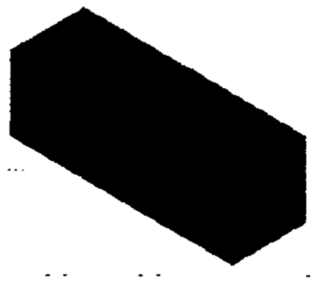

Figure 2.6: Meshing with structured cells (Yuan & Yang, 2020)

2.6.4 TURBULENCE MODELS

The aim of using the turbulence model is to predict the time-averaged field such as velocity, pressure, and temperature without measuring the full turbulent flow as in Reynolds- averaged Navier- Strokes Models (RANS) and Large Eddy Simulation (LES). RANS contain few different models such as Standard *k*-ε (*Sk*- ε), Realizable *k*-ε (*Rk*- ε), Renormalization Group *k*-ε (*RNG k*- ε), Standard *k*-ω (*Sk*- ω), Shear Stress Transport *k*- ω (*SST k*-ω), and Reynolds Stress Model (RSM). The indoor stream wise wind speed obtained with the other k-ε models is up to 6 times higher (at x/D = 0.75) than the one obtained using the SST k-ω model which is the reference case. The turbulence models are compared to the PIV experimental result and found out that the *SST k-ω* model which is the reference case and *RNG k*-ε model gives the best performance out of the others 6 models (Blocken & Ramponi, 2012).