Thermal Performance Analysis of Courtyards in a Hot Humid Climate using Computational Fluid Dynamics CFD Method

Abdulbasit Almhafdy*, Norhati Ibrahim, Sabarinah Sh Ahmad, Josmin Yahya

Faculty of Architecture, Planning and Surveying, Universiti Teknologi MARA, Shah Alam, Malaysia

Abstract

Computational Fluid Dynamics (CFD) has been applied in many researches in order to investigate the ventilation in buildings. However, researches on natural ventilation inside courtyard using CFD techniques are rare. Courtyard aspect ratio and cantilevered roof are selected for investigation using CFD in IES<VE> software. Wind and air temperature was obtained and thus, the thermal comfort is evaluated by the Predicted Mean Vote (PMV) index. Results show that aspect ratio and cantilevered roof plays a significant effect on the wind speed and, consequently, the thermal comfort. The (1:2) plan aspect ratio performs better than the (1:1) plan aspect ratio.

Keywords: Courtyard; CFD; thermal performance; PMV

1. Introduction

Computational fluid dynamics (CFD) is a simulation method that usually used to study buoyancy driven natural ventilation within the indoor or the outdoor areas. The CFD techniques are becoming an alternative and common method to investigate natural ventilation in buildings due to its informative results and low labor and equipment costs (Murakami, 2006; Murakami, Zeng, & Hayashi, 1999).

However, CFD applications for buildings in Malaysia are quite new (Sapian, 2009), especially for semi outdoor area such as courtyard (Almhafdy, Ibrahim, Ahmad, & Yahya, 2013b). Therefore, this paper
attempts to use CFD approach in different design variants of courtyard in Malaysian hospital namely aspect ratio and cantilevered roof as they have been highlighted by (Almhafdy, Ibrahim, Ahmad, & Yahya, 2013a).

2. Literature Review

Because of the complex thermal airflow phenomenon happened in the courtyard inside building in hot humid climate, several researchers including (Berkovic, Yezioro, & Bitan, 2012), (Rajapaksha, Nagai, & Okumiya, 2003) and (Berkovic et al., 2012) made efforts to discover a relatively accurate method to simulate ventilation for these particular spaces.

2.1. CFD application

In the last few years the improvement of the Computational Fluid Dynamics (CFD) has made it possible to get a better understanding and to simulate the air flow which occurs in ventilated rooms and buildings. Only in recent years the three-dimensional geometrical models using a fine grid have made the models more accurate, due to the theory of turbulence and increasing computational power.

Many researchers have studied and applied various CFD models including (Franke et al., 2004) for wind engineering, (Sapian, 2009) for high rise buildings and (Li, 2007) for type of opening. All have reported that the accuracy of CFD models depend on the grid size, the boundary conditions, and the type of approximations used.

Examples of CFD research is done by Livermore & Woods (2006) who has been studied the application of stack ventilation as secondary ventilation for multiple story building. The secondary ventilation within the lower floor is shown to increase with the ratio of the size of the openings on the lower to the upper floor and also the height of the stack. Results indicate that the lower need the large inlet area to preserve the high amount in the ventilation through the upper floor.

Another study by D’Agostino & Congedo (2014) to investigate the adequacy of natural ventilation in a historical building located in the South of Italy. The aim was to find an approach that could be able to provide a suitable microclimate for the conservation in that building.

Based on the model outputs, closing two windows was found acceptable. In particular, the model allowed establishing which some of windows can produce damaging airflows in delicate areas and other windows should not be open as high airflow patterns and microclimatic variations can be generated.

Gan (2010) has introduced the computational domain used for simulation buoyancy-driven natural ventilation in a vertical tunnel with opening for inlet and outlet. The research has shown that using the domain which has larger physical size than the object of multi-story ventilation will give more accurate ventilation rate inside the tunnel. The tunnels are not only small size but also have the size of the building courtyard.

2.2. CFD in courtyard

Many studies around the world have provided some strategies to improve the thermal performance inside courtyards using CFD modeling. To mention some of them, Chiang & Anh (2012) has studied the effect of natural enhanced by buoyancy force ventilation inside courtyard in high-rise residential building using CFD software called PHOENICS/FLAIR. Gradient of air temperature and airflow velocity in the courtyard is observed for all situations. The gradient is non-linear and is particularly high near the top of courtyard.
Another investigation on the natural ventilation within courtyard and another urban forms were done by Taleghani et al., (2014). The courtyard form was analysed for the hottest day so far in the temperate climate of the Netherlands (19th June 2000 with the maximum 33°C air temperature). The results establish that duration of direct sun and mean radiant temperature (MRT) play a significant effect in thermal comfort. The courtyard (compare to the other cases) offers the most comfortable microclimate compared to the other studied urban forms.

3. Methodology

A commercial computational fluid dynamic (CFD) software MicroFlo IES<VE> 2013 is used for simulation of airflow and heat transfer through the U-shape courtyards.

MicroFlo is a Computational Fluid Dynamics (CFD) application in IES<VE> software that is concerned with the numerical simulation of fluid flow and heat transfer processes. The objective of CFD applied to buildings is to provide the designer with a tool that enables them to gain greater understanding of the likely air flow and heat transfer processes occurring within and around building spaces given specified boundary conditions which may include the effects of climate.

Due to the time constraints as well as the complexity processes of CFD simulation, three models are selected for CFD process as shown in figure 1.

3.1. Description of the building

Three types of U-shape courtyards are considered in this CFD simulation. The square (1:1) and the rectangle (1:2) are selected by varying the length and width. And the third model is rectangle (1:2) with cantilevered roof that covered 60% of the top opening area of the courtyard. According to Almhafdy et al., (2013a) most common of the courtyards in Malaysia are ether square or rectangle. So the changes are also based on the area that is proposed in these design variants as well.

Fig. 1. The three selected models for CFD simulation
On the other hand, figure 1 shows that only indoor areas that directly attached to the courtyard (highlighted with brown colour) are included in obtaining the results of the simulation for CFD analysis. The indoor areas that not directly attached to the courtyard (highlighted with green colour) are excluded in the results. This is will insure that the indoor areas that included in the analysis have the same area and volume in all 3 models. Therefore, as seen in figure 1, the indoor areas of the 3 sides that directly attached to the courtyard are 2000m² (800m² for 2 blocks and 400m² for the third block) with three floors, however, as the areas of the courtyards are varied, the length and width of the attached buildings of courtyards are varied as well.

3.2. Simulation process

Due to the complexity of CFD simulation processes, step by step explanations are provided in the following sections.

- **ApacheSim Thermal Application**
  The surface temperatures for CFD boundary conditions of the models are exported by ApacheSim. ApacheSim application is a dynamic thermal simulation program based on first-principles mathematical modeling of the heat transfer processes occurring within and around a building. This application provides an environment for the detailed evaluation of building and system designs, allowing them to be optimized with regard to comfort criteria and energy use. The simulation is driven by real weather data and may cover any period from a day to a year. The time-evolution of the building’s thermal conditions is traced at intervals as small as one minute. The simulation requirements including APlocate for weather data of Kuala Lumpur, Suncest solar radiation and MacroFlo for the natural ventilation inside the courtyard are linked to the ApacheSim and the simulation period is set for the whole year.

- **Surface Boundary Conditions**
  The initial surface temperatures are set using the boundary conditions from Apache results using Vista application. By the selecting the space (courtyard) that need to be simulated in CFD, the boundary conditions can be imported using Import boundary data. On the other hand, figure 2 shows that the desirable wind that its temperature is less than 29°C is primary blowing from the north direction which is considered of 37.5% of the hours in the whole year (Tang. & Chin, 2013). Therefore, the CFD is used only for north direction and all the three models of the U-shape courtyard are oriented to the north.

- **Grid Settings**
  The default grid spacing and merge tolerance are defined. The merge tolerance enables grid lines that are separated by a distance less than the tolerance to be merged into a single grid line to minimize superfluous gridding. 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. The grid cells are defined for each model and they are: square (1:1) with 766656 cells, rectangle (1:2) with 263340 cells and rectangle (1:2) that have cantilevered roof with 255360 cells.
- Turbulence Model
  There are two types of turbulence models: k-e and Constant effective viscosity. However, in this paper, method k-e is selected as it is the most generally accepted and widely used. The k-e model calculates turbulent viscosity for each grid cell throughout the calculation domain by solving two additional partial differential equations, one for turbulence kinetic energy and the other for its rate of dissipation.

![Fig. 2. The wind rose diagram shows the direction of the wind that is less than 29°C. Source: Tang. & Chin, (2013)](image)

- MicroFlo Monitor
  As shown in figure 3, The graphical display monitor will graphically display the residual errors of all iteration for the selected variables as well as for defined the cells. All the information of the CFD setting are displayed in this monitor like cells, outer iteration, turbulence model, boundary conditions file and the estimation time for the simulation process.

![Fig. 3. MicroFlo monitor for CFD simulation processes](image)
MicroFlo Viewer

The MicroFlo viewer application can be used to display your CFD results. As seen in figure 4, the results of this paper are extracted using this application in terms of top view for the three typical environmental indicators namely air temperature, air velocity and Predicted Mean Vote (PMV). The comfort in courtyards at pedestrian level, 1.5 m above ground level, (representing the breathing zone height of a standing person) was evaluated by the modified Predicted Mean Vote (PMV). PMV was initially devised to quantify thermal comfort indoors, but later has been extended to outdoor environment (Berkovic et al., 2012).

![Image of MicroFlo viewer application for CFD results](image)

Fig. 4. The MicroFlo viewer application for CFD results

4. Results and Discussion

The results of this experiment are presented in figures 5, 6 and 7. Three environmental indicators are obtained in order to evaluate the thermal performance inside the U-shape of the three U-shapes of courtyards. Environmental indicators namely air temperature, air velocity and PMV. Moreover, a comparison between the three models in terms of CFD results are also explained in the following sections.

4.1. Courtyard with aspect ratio of (1:1)

The U-shape of courtyard with aspect ratio (1:1) is a square and the results of the CFD simulation are presented in figure 5. The results presented are obtained in the level of 1.5 m above the ground floor. The result shows that the location of the side opening to the north that made the courtyard U-shape has effect on the airflow inside the courtyard. For the temperature, it can be seen clearly that from the opening side is less than the enclosed side. For instance, within the opening side of the U-shape of the courtyard that face the north, the air temperature is recorded with 32.4°C while towered the opposite side of the courtyard, which is enclosed, the air temperature is recorded from 32.8°C to 33.4°C.

Moreover, the interpretation of the temperature performance can be shown in the CFD result of the velocity performance in figure 5. The figure shows the velocity gradients from (0.01m/s) to (2 m/s) from the opening side to the other side of the courtyard within the plan. It can be seen clearly that the openings of the U-shape of the courtyard, either from side or even form the top, are playing significant effect in the wind flow as well as the temperature. Based on the result of air temperature and air velocity, the PMV
was calculated and, as seen in figure 5, the results are compatible with the velocity and temperature. The PMV results are gradients from 1.4 to 2.8. The figure of PMV results recorded with 1.4 within the area that closes to the opening side. While the rest area of the courtyard, the PMV results recorded with 1.6 to 2.8.

Overall, the opening through the north side in the U-shape courtyard shows remarkable effect on the thermal performance inside the courtyard.

Fig. 5. Results of CFD simulation in U-shape courtyard with plan aspect ratio of (1:1)

4.2. Courtyard with aspect ratio of (1:2)

The U-shape of courtyard with aspect ratio (1:2) is a rectangle shape and the results of the CFD simulation are presented in figure 6. As done before, the results illustrated are acquired in the level of 1.5 m above the ground floor. As seen in the (1:1) courtyard, the temperature can be seen clearly that from the opening side is less than the enclosed side, however, the (1:2) aspect ratio of courtyard shows slightly better performance than the (1:1). For instants, within the opening side of the U-shape of the courtyard that face the north, the air temperature is recorded with 32.6°C while towered the opposite side of the courtyard, which is enclosed, the air temperature is recorded from 32.4°C to 32°C.

Moreover, the interpretation of the temperature performance can be shown in the CFD result of the velocity performance in figure 6. The figure shows the velocity gradients from (0.01 m/s) to (2 m/s) from the opening side to the other side of the courtyard within the plan. The difference between (1:2) and (1:1) courtyards is that half of the (1:2) area that face the opening side have better air velocity from 1.34 to 1.9 m/s. as can be seen in figure 6 for air velocity, only limited area within the (1:2) courtyard result with less than 0.45 m/s.

It can be seen clearly that the openings of the U-shape of the courtyard, either from side or even form the top, are playing significant effect in the wind flow as well as the temperature. The PMV was calculated and, as seen in figure 6, the results are attuned with the velocity and temperature. The PMV results are gradients from 1.2 to 2.2 which is better than the PMV result of the (1:1) courtyard. The figure of PMV results recorded with 2 within the area that closes to the opening side. While the rest area of the courtyard, the PMV results recorded with 1.2 to 1.6.

Overall, as (1:2) courtyard provide shading area more than the (1:1) courtyard, the shading area within the courtyard play significant effect on the thermal performance more than the opening area inside the courtyards.
4.3. Courtyard with aspect ratio of (1:2) with cantilevered roof

The U-shape of courtyard with aspect ratio (1:2) is a rectangle shape covered by cantilevered roof. The area of the cantilevered roof ratio was 60% compare to overall top opening of the courtyard. The results of the CFD simulation are presented in figure 7. Following the same procedures in previous steps, the results illustrated are obtained in the level of 1.5 m above the ground floor.

As seen in the (1:2) courtyard with cantilevered roof, the temperature can be seen clearly that from the opening side is more than the enclosed side, however, this courtyard shows slightly better performance than the previous cases. For instants, within the opening side of the U-shape of the courtyard that face the north, the air temperature is recorded with 32.2°C while towered the opposite side of the courtyard, which is enclosed, the air temperature is recorded from 32.6°C to 31.4°C.
Moreover, the explanation of the temperature performance can be shown in the CFD results of the air velocity performance in figure 7. The figure shows the velocity gradients from (0.01 m/s) to (2 m/s) from the opening side to the other side of the courtyard within the plan. Results show no differences between (1:2) and (1:2) with cantilevered roof. It can be seen slightly decrease in the (1:2) courtyard with cantilevered roof especially in the opening side that face the north direction. It can be seen clearly that the shading area of the U-shape of the courtyard that provided by any media such as cantilevered roof are playing significant effect in the wind flow as well as the temperature.

The PMV was calculated and, as seen in figure 7, the results are attuned with the velocity and temperature and moreover, the shading area shows a clear effect on the level of comfort of the PMV index. The PMV results are gradients from 0.8 to 1.4 which is better than the PMV result of the (1:1) courtyard as well as the (1:2) courtyard without cantilevered roof. The figure of PMV results recorded with 1.4 to 1.0 within the area that closes to the opening side. While the rest area of the courtyard, the PMV results recorded with 0.8. Overall, as (1:2) courtyard with cantilevered roof provide shading area more than the (1:2) courtyard without cantilevered roof, the shading area within the courtyard play important effect on the thermal performance more than the opening area inside the courtyards.

5. Conclusion

The design variants of courtyard such as plan aspect ratio and cantilevered roofs (as shading devices) have been investigated in this experiment. MicroFlo application in IES<VE> software has been used to perform the CFD simulation. The effect of buoyancy-driven ventilation is evaluated by allowing ventilation through opening side of the courtyard that oriented to the north. The CFD study was conducted during 28th of March as this month was reported the hottest month in the whole year, in Malaysia.

Three environmental indicators were observed namely air temperature (°C), air velocity (m/s) and the thermal comfort which is evaluated by the Predicted Mean Vote (PMV) index.

Results show that the aspect ratio and cantilevered roof plays a significant effect on the wind speed and, consequently, the thermal comfort. The U-shape of courtyard with aspect ratio of 1:2 (which is considered as a rectangle) performs better than the U-shape of courtyard with aspect ratio of 1:1 (which is considered as a square). In addition, increasing the shading area using cantilevered roof has shown a remarkable improvement for the thermal comfort.

In a nutshell, the results have indicated that manipulation and treatment of courtyard plan aspect ratio and shading area will increase the potential of improving its natural ventilation and thus, the thermal performance.

Acknowledgements

This research is made possible with the financial support provided by the Research Management Institute (RMI), Universiti Teknologi MARA, Malaysia under its Research Intensive Faculty (RIF) funding program. We therefore thank RMI as well as the Faculty of Architecture, Planning and Surveying for this research opportunity.
References


Taleghani, Mohammad, Kleerekoper, Laura, Tenpierik, Martin, & van den Dobbelsteen, Andy. (2014). Outdoor thermal comfort within five different urban forms in the Netherlands. *Building and Environment*(0). doi: http://dx.doi.org/10.1016/j.buildenv.2014.03.014