Penerbit Akademia Baru

CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

CFD HVAC Study of Modular Badminton Hall

Open Access

Mohd Azmi Ismail[1,*], Mohd Sabri Che Jamil[2]

[1] School of Mechanical Engineering, Universiti Sains Malaysia, 14300 Nibong Tebal, Pulau Pinang, Malaysia
[2] Silara Akses Sdn Bhd, Bandar Sri Damansara, 52200 Kuala Lumpur, Malaysia

ARTICLE INFO

Article history:
Received 21 May 2020
Received in revised form 20 July 2020
Accepted 25 July 2020
Available online 31 July 2020

ABSTRACT

Modular badminton hall is proposed to Badminton Association Malaysia in 2017. This hall has several advantages including; easy to transport and relocate, and fast installation and dismantling. The air ventilation is one of the important aspects in designing the modular badminton hall. The low room temperature and the air velocity are important characteristics, and they must to take into account during design and calculation. Therefore, the present work employed CFD ANSYS Fluent to study the effect of exhaust fan numbers and exhaust fan arrangements on the room temperature and the air velocity. The heat source are contributed by humans, and the solar radiation effect are included in the computational. Five designs with various number of exhaust fans and arrangements are studied. According to the result, Design 5 produces the lowest average room temperature, 26.52°C. Although the average air velocity of Design 5 is the highest; around 0.083m/s, it still acceptable as it has the maximum air velocity spot lower than 0.1m/s. In the present work, Design 5 shows the best performances and the best air ventilation design for the Modular Badminton Hall in Malaysia.

Keywords:
CFD; HVAC; badminton modular hall

1. Introduction

In Malaysia, badminton is the second most popular sport, after soccer [1]. Many of the world class badminton players are from Malaysia; including Lee Chong Wei, Misbun Sidek, Rashid Sidek etc. Thus, the federal government of Malaysia spent the millions of Ringgit Malaysia on badminton facilities, including badminton hall. Two years ago, Badminton Association Malaysia (BAM) received a proposal to develop modular badminton hall (MBH). The advantages of the MBH are i. easy to transport and relocate; ii. fast installation and dismantling; iii. minimal ground work; and iv. wide variety configurations [2].

The heating, ventilation and air conditioning (HVAC) is an important aspect in badminton hall design. The temperature, the relative humidity (RH), the air change per hour (ACH) and the air velocity are important characteristics, to provide comfortable and conducive environment for

* *Corresponding author.*
E-mail address: azmi_meche@usm.my (Mohd Azmi Ismail)

https://doi.org/10.37934/cfdl.12.7.9099

badminton players and spectators. Therefore, the present simulation predicts the air temperature and the air velocity inside MBH, at the given heat and ACH. According to Sport Hall, Design and Layout [3], the room temperature, the ACH and the air velocity should be at 18˚C, 1.5 and 0.1m/s, respectively. However, in Malaysia, to achieve the room temperature of 18˚C, it consumes very high electricity cost. Therefore, the present work only focuses on reducing room temperature by forced convection, in order to obtain the low temperature as possible, without air conditioning system. Sometime, relative humidity is important to ensure comfortability of the badminton player. Some method was used to control air relative humidity. Thus, Pravinth *et al.,* [4,5] and Khalid *et al.,* [6] study to control and amend air relative humidity with water nozzle arrangements and desiccant materials.

Currently, many researchers employ CFD to predict the room temperature and the air velocity inside the badminton hall. Parikh [7] employs CFD to predict the flow behavior inside badminton hall. He used k-epsilon turbulent model to predict the air velocity. He claimed that the acceptable temperature for badminton hall between 20 ˚C and 25˚C. At the ACH of 3.27, and the volume flow rate of 24.2m^3/s, the author obtained the maximum air velocity for badminton hall 49m x 49m x 12.15m was 0.13m/s, slightly higher than the allowable air velocity (0.1m/s). Thus, the author recommended some opening must be close off. Moreover, the author not consider solar radiation in his study. In 2016, Darshakkumar *et al.,* [8] conducted the CFD simulation study on the badminton hall in Vadodara, India. They set ACH of 3; which was 1.5 higher than the minimum ASHRAE standard. Similar to Parikh [7], the solar radiation was not includes in the simulation study. According to the CFD result, the maximum air velocity was 0.18m/s, which is not acceptable for playing condition. Therefore, the author recommended that the ACH in that badminton hall must be around 1.5.

In the other application, the CFD also used to predict the temperature and the air velocity behaviors in sport hall. Qian and Yang [9] employ CFD Icepak to predict and design the air temperature and the air velocity inside gymnasium competition hall. According to their finding, the combined air distributors have higher energy efficiency than the single air distributor. They claimed that the CFD produces accurate and valuable guideline in future design. Stathopoulou and Assimakopoulos predict the indoor environment condition using CFD Phoenix [10]. The simulation results have been validated with experimental results, in order to enhance reliability of the CFD results. The simulation results revealed that the spectators at upper seat will experience discomfort condition, due to the high temperature and the high CO_2 concentration, especially at hot weather and during match.

More than that, the CFD also used to examine the indoor air quality. Isele *et al.,* [11] employing the CFD to investigate the indoor air quality of the sport hall. The natural air ventilation; the warm contaminated air rises towards ceiling, and the cool fresh air flows downward towards ground floor, have been simulated by the authors. Their results shows that the distinct interface area have higher CO_2 concentration than the ceiling and floor areas. Other than these, the CFD also widely used in the electronic cooling application [12], the commercial aircraft design [13,14] and the sport equipment [15]. In the present study, the CFD is used to predict the air temperature and the air velocity in the MBH at the given heat, for the various number exhaust fans and its arrangements. The solar radiation is take into account in the present simulation study.

The development of modular hall as badminton court is introduced in the present work. In fact, until now, no modular hall is employed as badminton hall, especially in Malaysia. Therefore, the study of modular hall for badminton must be comprehensive, and the first step is by using CFD. An objective of the study is to investigate the effect of numbers and arrangements of exhaust fans on air velocity profile and temperature distribution, which is take into account of solar radiation.

2. Methodology

The model and dimension, the boundary condition and the computational setup are discussed in the methodology section.

2.1 Geometric Modelling and Dimensions

The present work developed 5 MBH models in order to investigate the air circulation and the air temperature inside MBH, in order to reduce the air temperature and the air velocity. The length, width and height of all MBH are 27.2m, 25m and 12.9m, respectively. The geometric modelling has been drawn using GAMBIT pre-processor. The 2.65 million hexagonal mesh elements were used in the present simulation. Table 1 show the numbers of exhaust fans for all model.

Table 1
The numbers of exhaust fans

Design	Exhaust fan number
1	18
2	18
3	26
4	18
5	26

2.2 Boundary Conditions

Afterwards, the CFD Fluent 17.2 version was used to predict the temperature and air velocity inside the MBH. The ambient temperature of the present simulation was 25˚C. The ground floor, roof and side walls were set as wall. The pressure inlets were set above wall, and it has the height of 10cm. The value of pressure inlet and temperature were 0Pa and 25˚C, respectively. The pressure outlets was employed to represent exhaust fans. The area of all of exhaust fans were 0.041m^2, and each of them has capability to suck out 500cfm of the air inside MBH. All the exhaust fans were located on the side walls.

For simulation Design 4 and Design 5 only, they have 10cm air gap between the roof and the side walls. The air gap was set as pressure outlet with 0Pa. The purpose of the air gap was to remove radiation heat from the roof. The material of roof was set as steel, and the other wall were set as concrete.

For heat boundary condition, the spectator or human was set as heat flux, and they released heat by natural convection. In the present work, the total heat released by human was 2870W. The simulation model take into account the heat radiation. The Direct solar irradiation and the diffuse solar irradiation were set as 1250W/m^2 and 800W/m^2, respectively.

2.3 Computational Setup

The incompressible ideal gas was employed in the present work, in order to activate natural convection. In spatial discretization section, standard formulation was selected in pressure equation. Second order upwind was used in momentum and energy equations. 1000 iteration has been set and the simulation converged at iteration of 963. Figure 1 illustrates the geometric configuration and boundary condition setup of the present simulation.

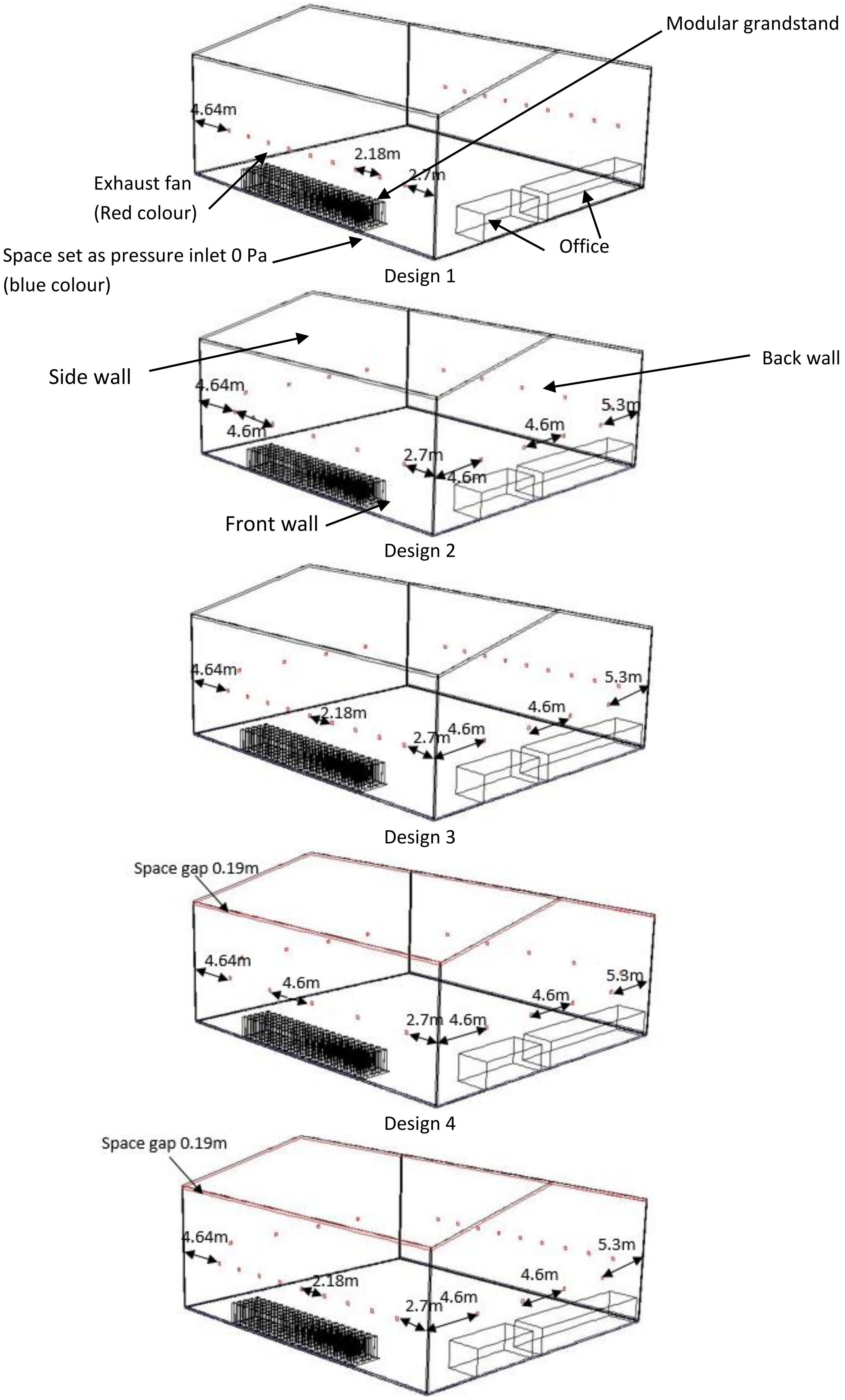

Fig. 1. The geometric configuration and boundary condition setup

2.4 Grid Information and Grid Independent Test

The present work employs 3.5 million hexagonal meshing. The structural meshing was developed using GAMBIT pre-processor. It cost 4 days for one case to converge. It take long time for one case to converge due to parallel simulation is not allowed for solar radiation application. Then, the grid independence test was conducted before examining the velocity and temperature distribution inside badminton modular hall. Three different numbers of meshes were used: 1500000 (coarse mesh), 3500000 (fine mesh) and 4800000 (very fine mesh). According to the result, the average temperature inside modular hall for Design 1 and coarse mesh is 26.02°C. The fine mesh and very fine mesh predicting the average temperature inside modular hall are 27.87°C and 27.44°C, respectively. Therefore, the fine mesh is used in the subsequent design simulation, as the fine mesh has a temperature almost same with very fine mesh, in which temperature of fine mesh is 1.5% higher than very fine mesh.

3. Results and Discussion

Figure 2 illustrates the wall temperature contour of MBH. In general, the wall temperature for all models, are less than 30˚C. The figures also show that the highest temperature contour occurs on the roof of MBH, for all models. The solar radiation is a potential root cause of the abovementioned phenomenon. The maximum solar radiation happens at noon, and at this time, roof has the highest view factor exposed to the sun. The figure also shows that the wall temperature decreases as it locates away from the MBH roof. It happens due to the inlet of cool air ventilations are located at floor level. According to the figure, the lowest wall temperature is produced by Design 5. It means the exhaust fan arrangements and the number of exhaust fans have a huge potential to reduce the wall temperature.

The air velocity contours at height 1m above floor surface are illustrated Figure 3. According to the figure, all the design produce the air velocity less than 0.1m/s. The air velocity of 0.1m/s is the maximum air velocity allowed by sport England standard [3]. According to the figure, the highest average air velocity are produced by Design 4 and Design 5.

Figure 4 illustrates the air temperature contour inside the MBH. According to the figure, Design 5 wall shows the lowest air temperature inside the MBH, however, Design 1 shows the highest air temperature. Design 4 produces lower the air temperature than Design 3, even though, Design 3 has higher the number of exhaust fan than Design 4. The figure also illustrates that the ground level has lower temperature than the upper level. It is expected that the heat source in the present work comes from roof, due to solar radiation especially at the noon. In addition, the heat released by spectators move upwards to the roof, due to the buoyancy force, then exit to outside through the exhaust fans.

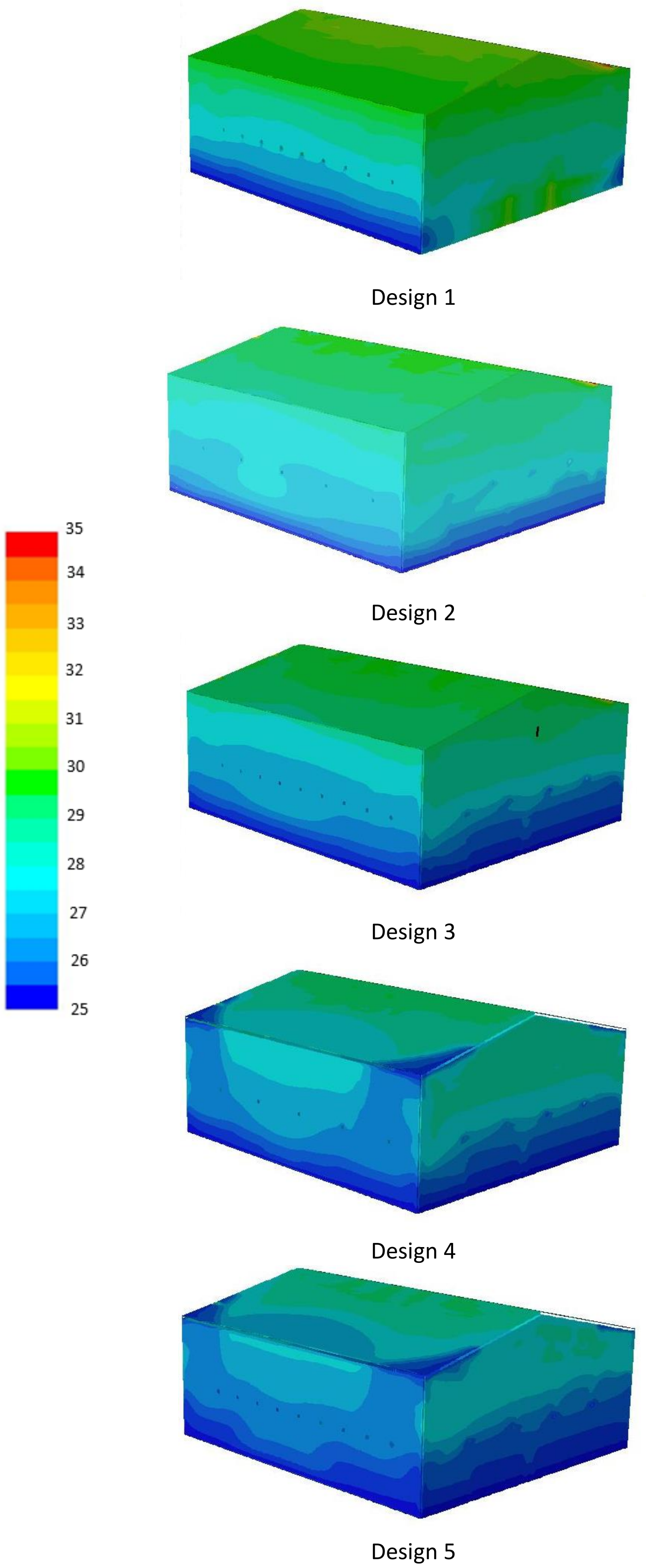

Fig. 2. The wall temperaure contour of MBH (°C)

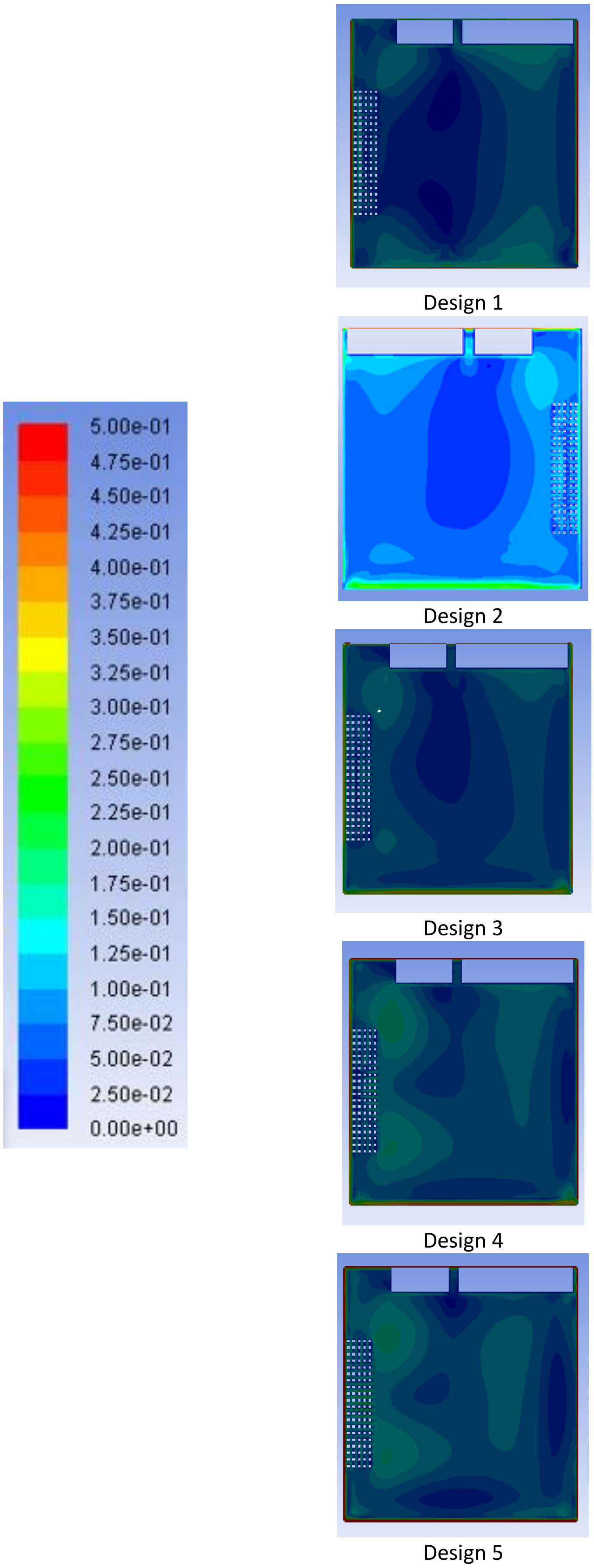

Fig. 3. The air velocity contour at height of 1m of floor surface (m/s)

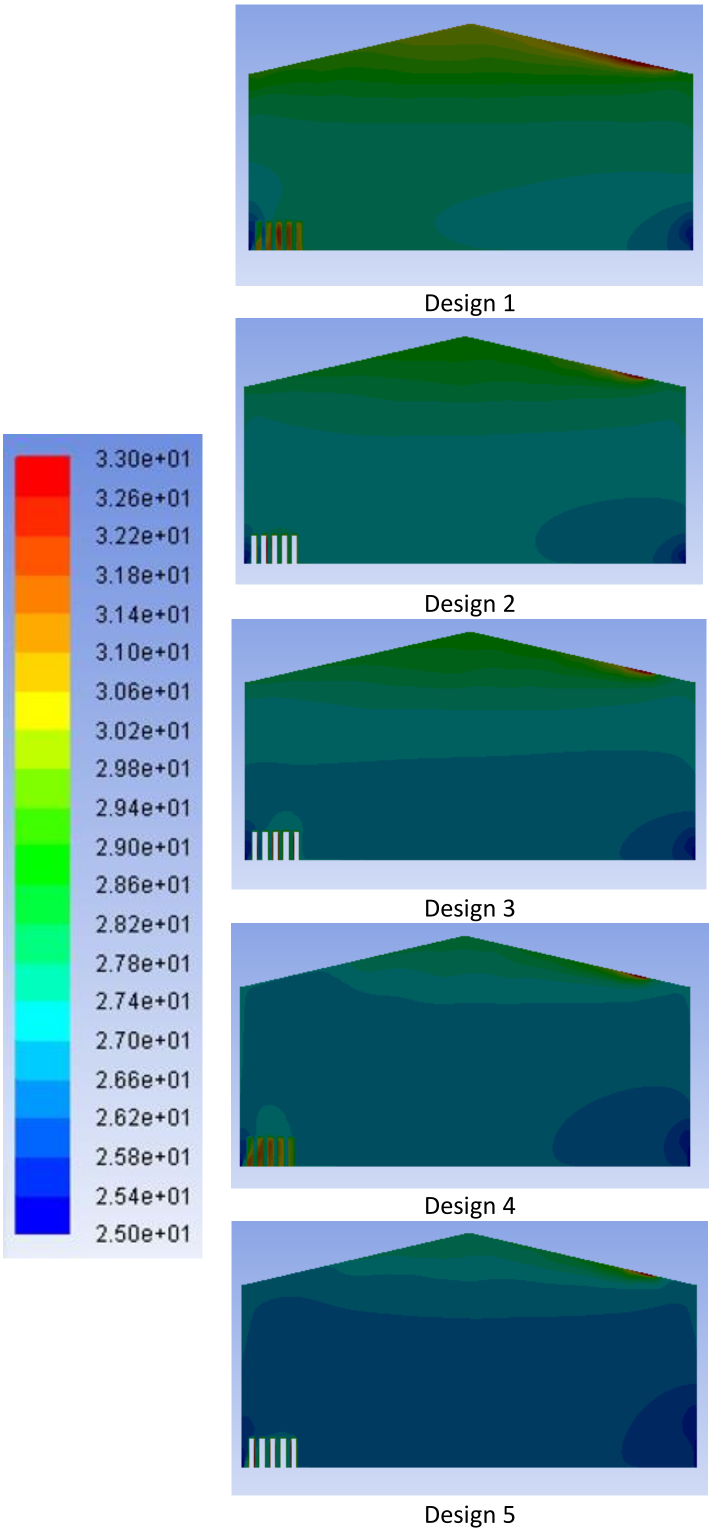

Fig. 4. Temperature contour in side view (°C)

Table 2 summaries the average air velocity in Figure 3. Design 2 is predicted to produces the lowest air velocity at height of 1m, followed by Design 1, Design 3, Design 4 and Design 5. In addition, Design 2 has the lowest maximum air velocity spot than the other designs. Meanwhile, Model 1, Model 2, Model 3 and Model 4 have similar maximum air velocity spot. A potential reason is, Design 2 have lower number exhaust fan; 18 exhaust fans, and they are located all MBH wall sides. Although

Design 1 also used 18 exhaust fans, however, they are located only at front and back walls, as illustarted in Figure 1.

Table 2
Summary air velocity in plane y=1.0m

Design	Average air velocity (m/s)	Max air velocity in court area (m/s)
1	0.075	0.1
2	0.061	0.075
3	0.076	0.1
4	0.098	0.1
5	0.099	0.1

The summary of MBH simulation study is shown in Table 3. The lowest wall temperature is produced by Design 5, and the highest hotspot wall temperature is presented by Design 1. For the average air velocity aspect, Design 2 show the best performance, which is the lowest average air velocity. It is followed by Design 3, Design 1, Design 4 and Design 5. The average air temperature shows similar pattern to hotspot wall temperature. The lowest average air temperature is 26.52˚C and it is produced by Design 5. The highest average air temperature is produced by Design 1.

Table 3
Summary of overall performance

Design	Hotpsot wall temperature (°C)	Average air velocity (m/s)	Average air temperature (°C)	ACH
1	29.85	0.0675	27.87	2
2	28.94	0.0603	27.38	2
3	28.61	0.0652	27.17	2.88
4	28.31	0.0806	26.80	3.02
5	28.07	0.0863	26.52	4.36

At the beginning, the study only focuses on ACH of 2. Since at ACH of 2, or 18 exhaust fans, shows low the maximum air velocity in MBH, the study increases the number of exhaust fan up to 26 exhaust fans. Therefore, Design 3 and Design 5 have higher ACH than Design 1 and Design 2. As the empty space gap between the wall and the roof was applied in Design 5 and Design 4, they are functioned as the secondary natural exhaust fan and they help in remove heat from the roof. Thus, Design 4 shows higher ACH and the average air temperature than Design 3, even though, Design 4 has lower number exhaust fans than Design 3. Design 5 show the highest ACH and the lowest air average temperature as it has the highest number of exhaust fans, and owns the secondary natural exhaust fan between the wall and the roof.

Although Design 1 and Design 2 share the same number of exhaust fans, Design 2 show better performance; lower the wall temperature and the average air velocity, than Design 1. Design 1 and Design 2 has same number of exhaust fan, however, the exhaust fan arrangement of Design 2 is different with Design 1. For Design 1, the exhaust fans only located at front and back walls. Thus, most of the air movement only focuses on front and back walls. However, the exhaust fans of Design 2 are located on all walls of MBH. Thus, air movement flows uniformly inside MBH. Therefore, Design 2 has lower air velocity and air temperature than Design 1.

4. Conclusions

The prediction of the air velocity and the air temperature inside MBH are discussed. The heat of spectators and solar radiation are take into account in the simulation. The effect of the exhaust fans and its arrangement, are investigated and discussed in this paper. The result indicates that the local air velocity for all design is or less than 0.1m/s. The highest hotspot area happen in MBH roof. Design 5 produces the lowest average air temperature; 26.52˚C, followed by Design 4, Design 3, Design 2 and Design 1.

Acknowledgement
The authors acknowledge funding from the School of Mechanical Engineering, Universiti Sains Malaysia and Ministry of Higher Education Malaysia's Fundamental Research Grant Scheme (FRGS Grant 203.PMEKANIK.6071411)

References

[1] Jahi, Tajuddin Md, Haziq Ikhwan Zawawi, and Norazah Abd Rahman. "Effect of skirt angle and feathers formation on shuttlecock aerodynamics performance." *Jurnal Teknologi* 76, no. 8 (2015): 95-99. https://doi.org/10.11113/jt.v76.5631
[2] M. S. Che Jamil. *Modular Badminton Hall Proposal*. Kuala Lumpur, 2017.
[3] C. Robin Wilson, A. S&P and D. Deloitte. *Sport Hall Design and Layout, Updated and Combined Guidance*. London, 2015.
[4] Balthazar, Pravinth, Mohd Azmi Ismail, A. S. B. A. Soberi, R. Muhammad Iftishah, and M. Hussin. "Experimental study of the effect of piccolo tube pipe on the air-conditioning experimental rig." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 53, no. 1 (2019): 95-105.
[5] Balthazar, Pravinth, Mohd Azmi Ismail, Andyqa Abdul Wahab, Mohammad Nazmi Nasir, Muhammad Iftishah Ramdan, and Hussin Bin Mamat. "Effect of Arrangement and Number of Water Mist Spray Nozzles on Air Humidity." In *Intelligent Manufacturing and Energy Sustainability*, pp. 123-132. Springer, Singapore, 2020. https://doi.org/10.1007/978-981-15-1616-0_12
[6] Kahlib, Nor Amyra Zulianey, Farah Diana Mohd Daud, Maizirwan Mel, and Ahmad Zahirani Ahmad. "Synthesis and characterization of silica ceramic membranes via sol-dip coating." *Journal of Advanced Research in Materials Science* 39, no. 1 (2017): 1-7.
[7] K. P. Nirav, "Ventilation system validation for badminton court." *in Research and Innovations in Science, Engineering & Technology*, 2017.
[8] D. Shah, G. Darshan and P. Devendra. "Determining the Flow Rates Required and Temperature of Inlet Cold Stream to Provide Thermal Comfort in Badminton Court of Sama Sports Complex Using CFD." *International Journal of Advance Research in Engineering, Science & Technology* 3, no. 2 (2016): 95-110.
[9] Q. Feng and Y. Li, "The Numerical Simulation and Study on Gymnasium Indoor Thermal Environment of Airpak," *in International Conference on Energy, Power and Electrical Engineering*, (2016) 24-29.
[10] Stathopoulou, O. I., and V. D. Assimakopoulos. "Numerical study of the indoor environmental conditions of a large athletic hall using the CFD code PHOENICS." *Environmental Modeling & Assessment* 13, no. 3 (2008): 449-458. https://doi.org/10.1007/s10666-007-9107-5
[11] Isele, André, Gerrit Höfker, and Malcolm Cook. "Numerical Study on the Carbon Dioxide Distribution in a Naturally Ventilated Space." In *Proceedings of Building Simulation, 12th Conference of International Building Performance Simulation Association, Sydney*, pp. 14-16. 2011.
[12] Azmi, Husin, Mohd Zulkifly Abdullah, and Mohd A. Ismail. "Numerical and experimental investigations of speaker-driven synthetic jet actuator for electronics cooling applications." *Heat Transfer Research* 50, no. 14 (2019): 1369-1381. https://doi.org/10.1615/HeatTransRes.2019026060
[13] Ismail, Mohd Azmi, and Jian Wang. "Effect of nozzle rotation angles and sizes on thermal characteristic of swirl anti-icing." *Journal of Mechanical Science and Technology* 32, no. 9 (2018): 4485-4493. https://doi.org/10.1007/s12206-018-0845-x
[14] Khai, Lee Chern, Mohd Azmi Ismail, Qummare Azam, and Nurul Musfirah Mazlan. "Experimental study on aerodynamic performance of nacelle lip-skin bias flow." *Journal of Mechanical Science and Technology* 34 (2020): 1613-1621. https://doi.org/10.1007/s12206-020-0323-0
[15] Hart, J. H., J. R. Potts, and D. J. James. "Comparison of turbulence modelling approaches in simulation of a feather shuttle: a porous conical bluff body." *Sports Engineering* 21, no. 4 (2018): 465-478. https://doi.org/10.1007/s12283-018-0293-y