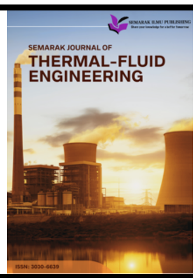
SEMARAK ILMU PUBLISHING

Semarak Journal of Thermal-Fluid Engineering

Journal homepage:
https://semarakilmu.my/index.php/sjotfe/index
ISSN: 3030-6639



Analysis of Louvre Angles in Cross Ventilation Systems for Enhanced Airflow and Indoor Cooling

Muhammad Amirul Amir Roshezam[1], Ishkrizat Taib[1,*], Zainal Ariff Zainal Abidin[1], Ridwan Abdurrahman[2]

[1] Department of Mechanical Engineering, Faculty of Mechanical Engineering and Manufacturing, Universiti Tun Hussein Onn Malaysia, 86400 Batu Pahat, Johor, Malaysia
[2] Department of Mechanical Engineering, Universitas Riau, Pekanbaru, 28293, Indonesia

ARTICLE INFO

Article history:
Received 29 November 2024
Received in revised form 12 December 2024
Accepted 19 December 2024
Available online 31 December 2024

Keywords:
Ventilation; building; air flow; angle; velocity

ABSTRACT

Cross ventilation is one of the most common strategies for enhancing indoor comfort and decreasing pollutants' concentrations. It can also be used as an indirect cooling tool to lower cooling requirements for buildings. Nevertheless, inadequate distribution of airflow in buildings and incorrect orientation of the louvres result in localized thermal discomfort and inadequate ventilation. This paper looks at the airflow patterns of a SolidWorks designed ventilation system and examines the influence of various louvre positions and slat angles of 15° and 30° through numerical modeling. The simulations were performed using RNG k-ε and SST k-ω turbulence models with the wind inlet velocity of 0.3 m/s which is the average wind velocity in Malaysia. The findings indicate that a 15° slat angle provides the best airflow distribution because the small angle allows a smooth entrance of air and little turbulence. On the other hand, a 30° slat angle increases velocity and pressure at the inlet and leads to turbulence and issues for indoor climate. The SST k-ω model provided higher velocity predictions than the RNG k-ε model, suggesting that it is better for capturing the detailed flow characteristics. Therefore, this study was able to show the effects of louvre angles on airflow patterns and found the 15° slat angle to be the most effective for ventilation with low turbulence.

1. Introduction

Normal cross ventilation is a widely used repair technique for comfortable indoor air and pollutant reduction [1-5]. It can also be used as a passive refrigeration system to mitigate building cooling demands [6]. However, the configuration of facade openings such as window panels significantly affects the efficiency of natural ventilation [7]. In addition, practical issues must be discussed, for example rain avoidance, burglarization, and excessive solar radiation. Architectural loops can be an efficient solution because they can decrease (especially when they are opaque) the entrance of direct solar radiation and prevent rainfall.

* *Corresponding author.*
E-mail address: iszat@uthm.edu.my

https://doi.org/10.37934/sjotfe.3.1.3140a

Several studies have explored unilateral or cross ventilation via louvre structure architecture. Some studies conducted cross-ventilation simulations in isolated buildings fitted with loops, wind tunnel tests, and computer fluid dynamics (CFDs) [8-11]. The opening position may therefore have a significant impact on indoor airflow, air shifts, and pollutant dispersion [12,13]. The air flow rate was lower in the building, with the lower portion of the facade having openings in the middle or top of the facade than in the building. There are few problems that are faced in buildings where air in the building does not flow properly and makes a few areas in the building become hot. Based on Jeong *et al.,* [14] the weakness is when the level of louvre ventilation is not place properly and make the air flow not going through smoothly. Another problem is usually when the angle of slat is not suitable and the air that goes through the ventilation is not properly entered.

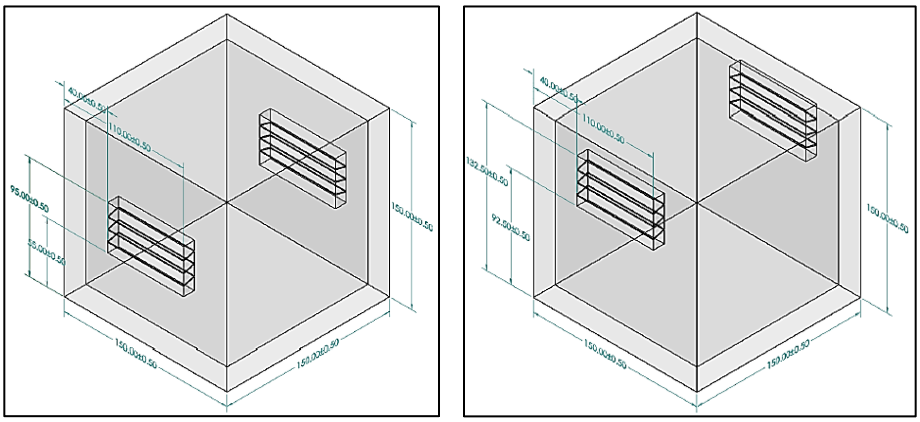
In this study, air ventilation was created using SolidWorks and imported to Ansys fluent 19.2 to determine the flow inside the air ventilation. The cube size of the ventilation with different place of louvres is set to determine the flow inside the ventilation. Another aim is to determine the best angle of slat with 15° and 30° angles that produce good air flow inside the air ventilation. Based on the study of Hazem *et al.,* [15] the angle of slat influences the airflow pattern inside the air ventilation. Next, this study also compared the best place of louvre and good angle of the slat because it can make the air inside the ventilation sufficient to supply the surrounding air.

Then, the simulation of air ventilation was run in Ansys fluent with RNG k-ε and SST k-ω model with the same velocity to determine the different results of the velocity and contour of the velocity inside the air ventilation. The velocity used is the minimum velocity 0.3 m/s in Malaysia. The simulation was performed in 1000 iterations or until the calculation converged. The result is an analysis based on the velocity contour and can be compared with each angle of the slat. This study used an air ventilation model with different place of louvres and angle that used is 15 and 30° to determine the flow inside the ventilation. The simulation was performed using computational fluid dynamics (CFD) to determine the best result.

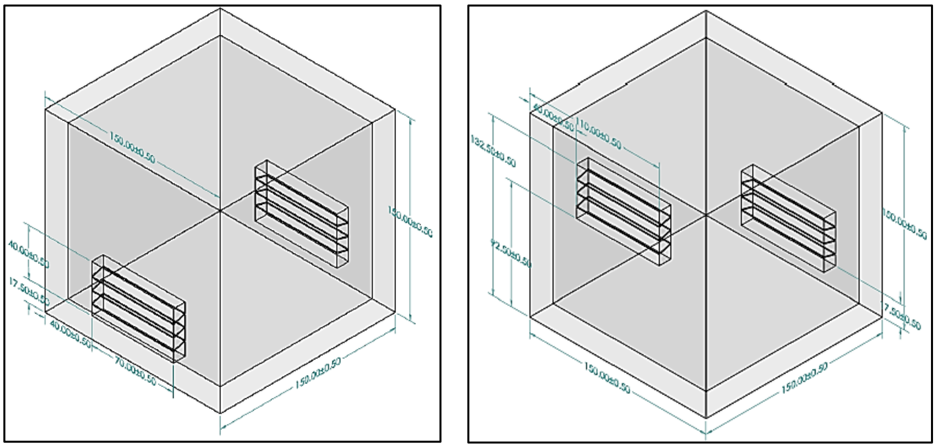
2. Methodology

2.1 Geometry of the Model

Figure 1 shows the model of air ventilation with different place of louvres. The model is based on a previous study [11]. For these studies, the angle of slat that was considered was 15° and 30° to determine the flow characteristic in the air ventilation. The geometry model was run with RNG k-ε and SST k-ω to compare the airflow characteristics based on the velocity contour. The size of this air ventilation unit is 150 mm x 150 mm x 150 mm with a cube shape.



(a) (b)



(c) (d)

Fig. 1. Model of air ventilation with different position of louvres (a) Louvres at centre (b) Louvres at upper (c) Louvres at down (d) Louvres at upper and down

2.2 Meshing of the Model

After the model has been generated, an unstructured grid is exposed across the flow domain. The grid comprises cells of various shapes, but usually it is triangular or rectangular for (2D) and tetrahedrons or hexahedron for (3D) is being used [16]. The grid type and density depend on the fluid domain geometry. Body sizing is performed in the air-ventilation fluid domain because the flow is on the fluid domain. Figure 2 shows the meshing of the model with a body size of 0.0062 mm and a number of nodes of 10011.

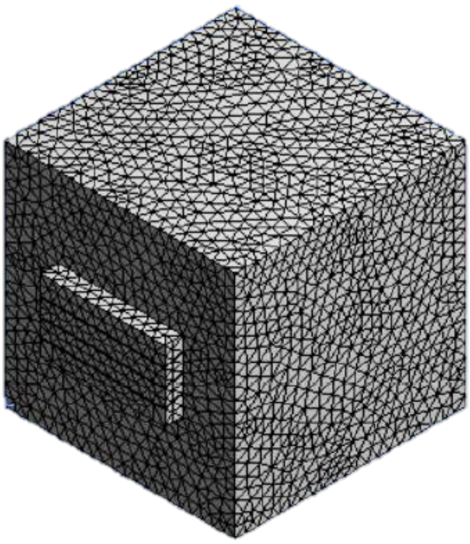


Fig. 2. Meshed model of air ventilation

2.3 Governing Equations

All of the equations used in CFD are based on the following principle, and they can be expressed in mathematical equation terms. Generally, they are changed in partial differential equation or call as Navier-Strokes Equation [17].

Continuity:

$$\frac{\partial p}{\partial t}+\frac{\partial (pu)}{\partial x}+\frac{\partial (pv)}{\partial y}+\frac{\partial (pw)}{\partial z}=0 \quad (1)$$

X- momentum:

$$\frac{d(pu)}{dt}+\frac{d(pu^2)}{dx}+\frac{d(puv)}{dy}+\frac{d(puw)}{dz}=-\frac{\partial p}{\partial x}+\frac{1}{Re_r}\left(\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{xz}}{\partial z}\right) \quad (2)$$

Y- momentum:

$$\frac{d(pv)}{dt}+\frac{d(puv)}{dx}+\frac{d(pv^2)}{dy}+\frac{d(pvw)}{dz}=-\frac{\partial p}{\partial y}+\frac{1}{Re_r}\left(\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{yz}}{\partial z}\right) \quad (3)$$

Z-momentum:

$$\frac{d(pw)}{dt}+\frac{d(puw)}{dx}+\frac{d(pvw)}{dy}+\frac{d(pw^2)}{dz}=-\frac{\partial p}{\partial y}+\frac{1}{Re_r}\left(\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}\right) \quad (4)$$

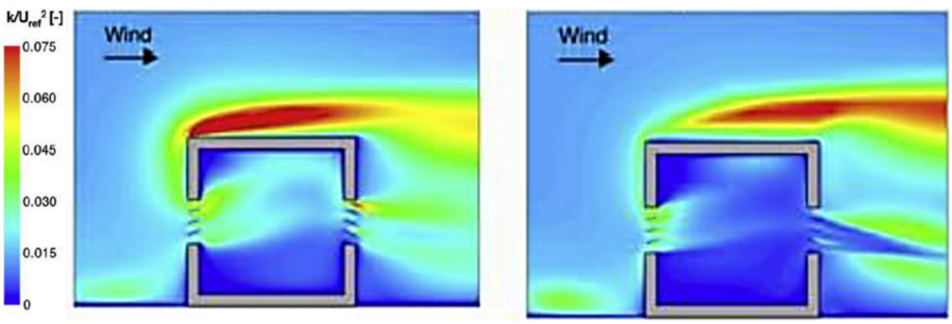
2.4 Parameter and Boundary Condition

From the geometry, each boundary condition is specified, and all parameters are inputted. In this study, the flow is considered as going through air ventilation at different angles of slat. The velocity used for the inlet was 0.3 m/s based on the average velocity in Malaysia [18]. For the outlet, zero pressures were set because no parameter was subtracted from the outlet. The fluid domain will determine the air ventilation velocity inside it and be compared with another angle of slat. A different model of with RNG k-ε and SST k-ω is run in this study to determine the flow pattern inside the air ventilation with the different place of louvres.

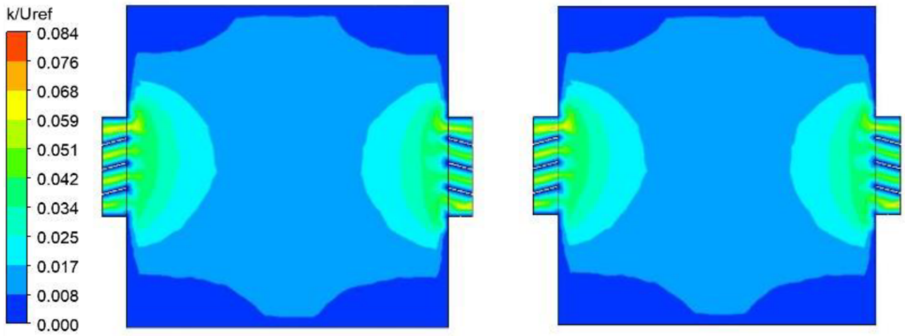
3. Results

3.1 Verification Louvre Angles in Cross Ventilation Systems

Based on the previous study by Kosutova *et al.,* [11] a verification method was performed to make sure the method was true or less error obtained from the verification. The verification results are shown in Figure 3. The model that had verified is RNG k-ε and SST k-ω to figure out the result from the simulation is correct. Based on the verification method, the error obtained was 10.71%, and the method was used for all simulation e of 15˚.



(a) (b)

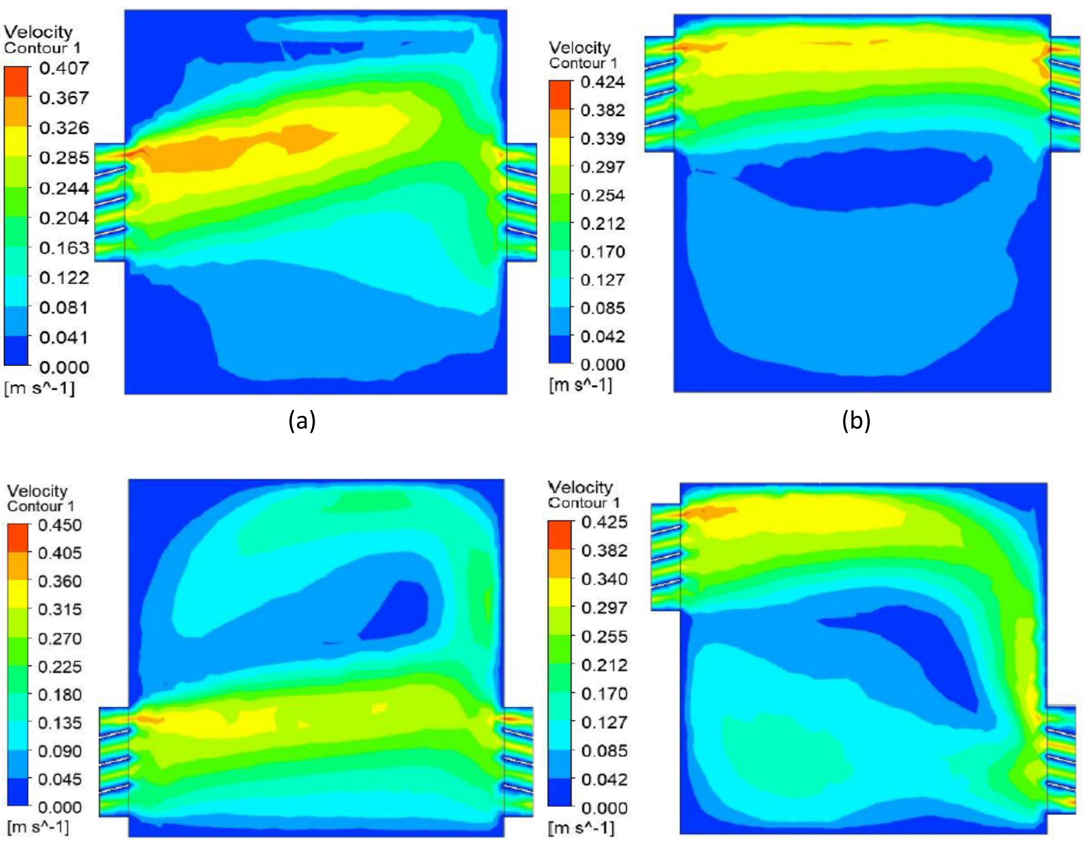


(c) (d)

Fig. 3. Verification results using different turbulence model (a) RNG k-ε (b) SST k-ω for previous study (c) RNG k-ε (d) SST k-ω for current study

3.2 Velocity Distribution Inside the Air Ventilation with the Different Position of Louvres
3.2.1 Velocity for slat angle 15°

Figure 4 shows the results of the velocity distribution inside the air ventilation with the different position of louvres. The velocity applied at the inlet was 0.3 m/s with an average wind in Malaysia. Based on the obtained results, the highest velocity inside the air ventilation was 0.45 m/s with the louvres at the centre and the lowest velocity was 0.407 m/s for the centre louvres. Based on the velocity contour obtained, the up and down louvres showed the wind that entry the louvres flow to most areas in the air ventilation.



(c) (d)

Fig. 4. Velocity distribution for RNG k-ε with 15 degrees of slat for different position of louvres (a) Centre (b) Upper (c) Lower (d) Upper and lower

Figure 5 shows the velocity contour inside the air ventilation at a slat angle of 15° with the RNG k-ω model. Based on the results, the lowest velocity obtains for SST k-ω is 0.402 m/s for centre louvres and the highest velocity was 0.477 m/s for down louvres. Based on contour, the down louvre with slat angle 15 degree obtains good air flow inside the air vent, because the wind flowed through most of the area inside the vent.

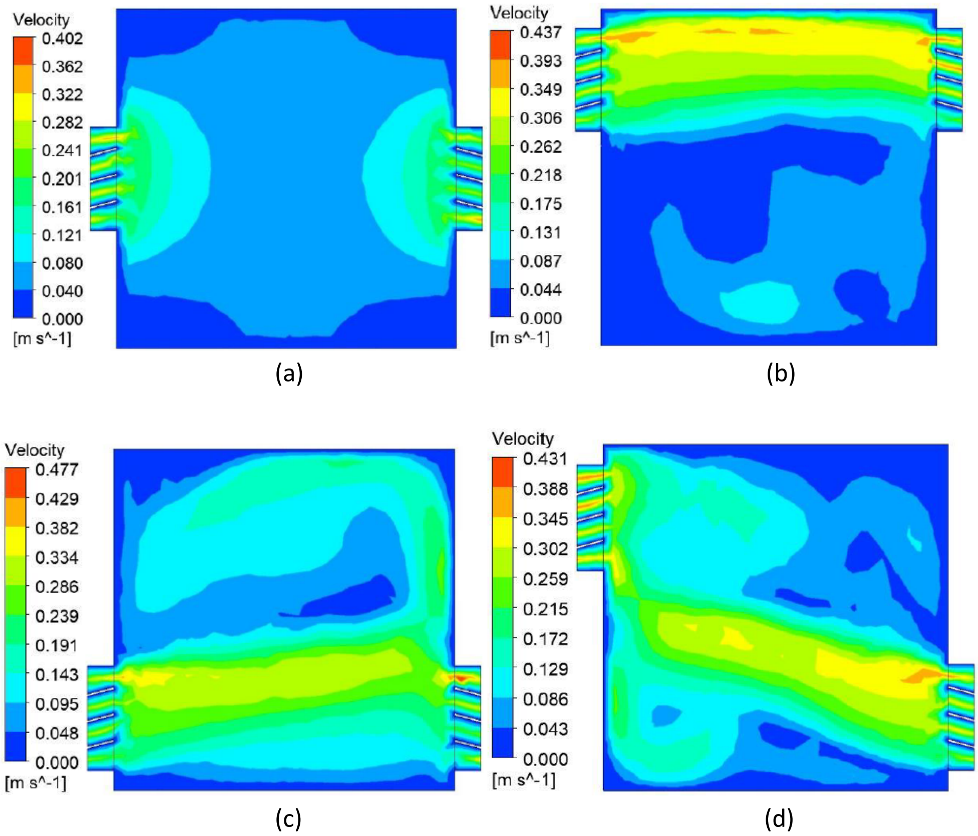


Fig. 5. Velocity distribution for SST k-ω with the slat angle 15 degree for different position of louvres (a) Centre (b) Upper (c) Lower (d) Upper and lower

3.2.2 Velocity for slat angle 30°

Figure 6 shows the velocity contour for slat angle 30 degree with RNG k-ε model. Based on the result obtain, the up louvres obtain the highest velocity of 0.563 m/s with a slat angle of 30°. These high velocities are obtained because the entry area becomes smaller and makes the velocity entry increase inside the air ventilation. The lowest velocity obtained was 0.532 m/s for the centre louvres because of the area inside the ventilation is wide than up louvre that the top wall was near the ventilation.

Figure 7 shows the velocity contour of the slat angle 30 for different place of louvres. Based on the results, the highest velocity was 0.569 m/s for up louvres because of the wind entry from the louvre is hit the top wall, which increased the velocity inside the ventilation. The lowest velocity obtained is for centre place of louvres is 0.547 m/s because the wind entry is far from any wall. When the wind hits any wall or something, it usually increases the velocity inside the object.

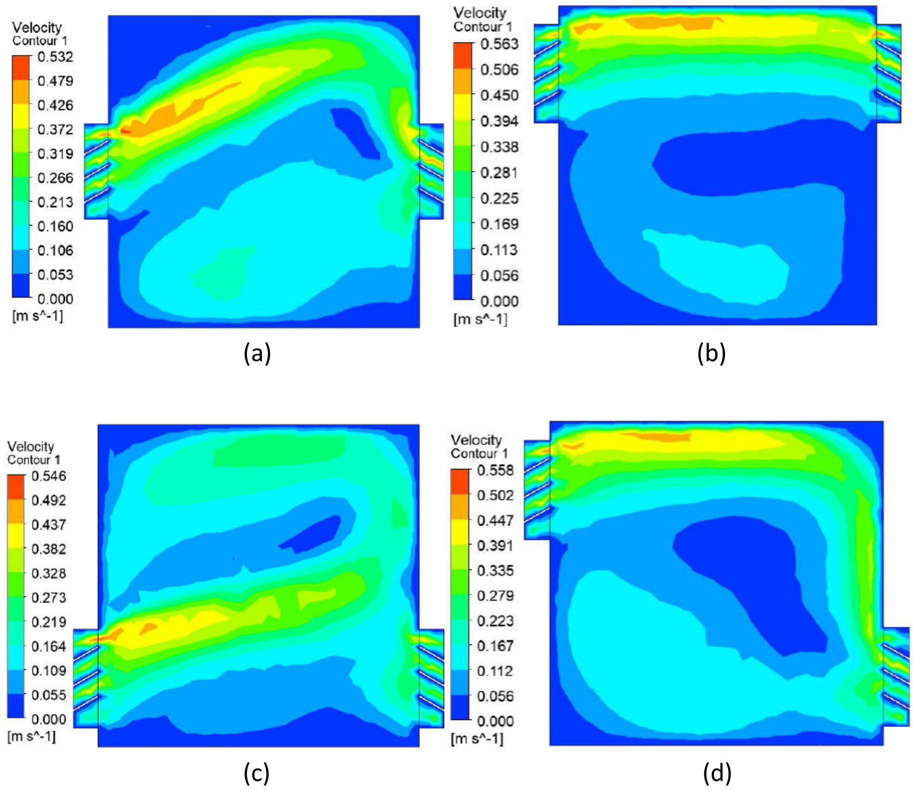


Fig. 6. Velocity distribution for RNG k-ε with 30° of slat for different position of louvres (a) Centre (b) Upper (c) Lower (d) Upper and lower

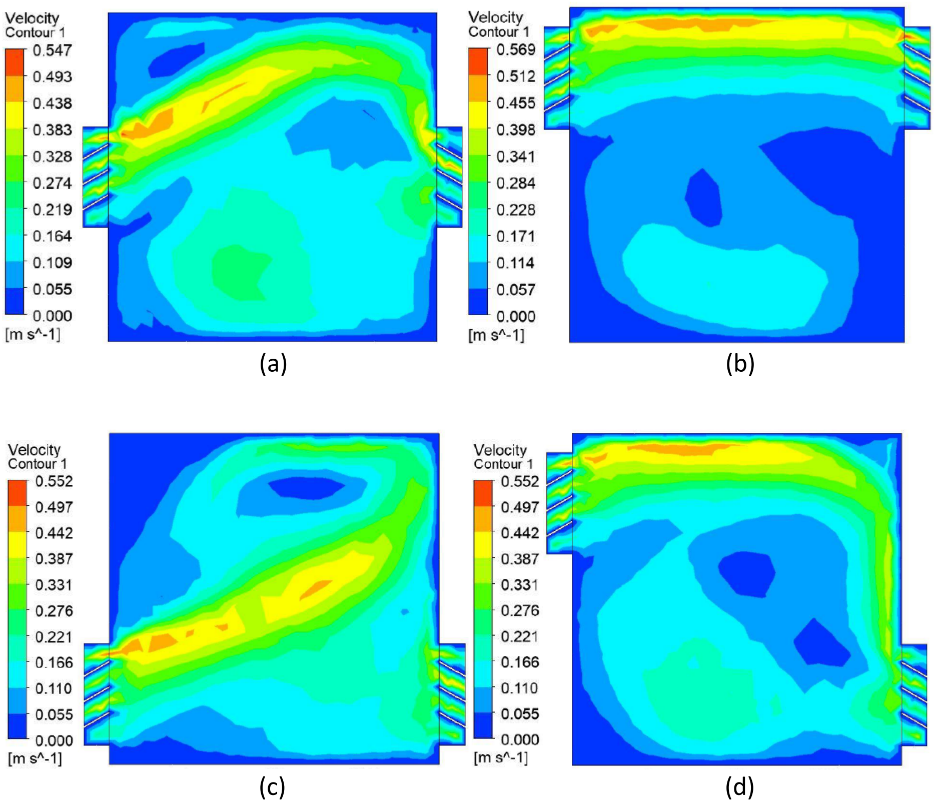


Fig. 7. Velocity distribution for SST k-ω with 30° of slat for different position of louvres (a) Centre (b) Upper (c) Lower (d) Upper and lower

From the obtained results, the angle that was used was 15° and 30° of slat affected the wind flow in the air ventilation. A correct orientation of the slats will enhance the airflow within the ventilation, but the velocity of wind around should also be taken into consideration. From the velocity contour for both 15 and 30 degrees of slat, it is clear that 15 degrees of slat is good for air flow inside the air ventilation with the louvres place at down but it gives a low velocity inside the ventilation. If the value angle of slat is raised, the velocity value inside the air ventilation will also rise, but the surrounding wind hinders the surrounding wind from entering the ventilation since the angle hinders the air from entering the ventilation.

Subsequently, position of louvres does affect the air flow pattern within the air ventilation [19,20]. Thus, it is clear that good airflow can be achieved from the suitable place of louvres and suitable angle slat. According to the findings, up and down louvres created satisfactory airflow within the air ventilation, and satisfactory airflow will make the building create satisfactory airflow. For another place of louvres, it obtains not enough air flow, and some of the wind entry into the ventilation does not go to some area inside the ventilation. According to the results, SST k-ω generated higher velocity value than RNG k-ε. This may be so because of the formula used in the calculation of the index. Based on the contour, for an angle of slat of 30°, good airflow occurs inside the airflow because most of the area inside the airflow is full with airflow. However, the high angle of the slat makes it difficult for air to enter the air ventilation.

4. Conclusions

Therefore, the main aim of this study was to identify the airflow pattern with various position of louvres, to identify the airflow pattern with slat angle of 15 and 30 degree of louvres, and to compare the airflow pattern with various position of louvre and various angle of slat. Moreover, in this study Ansys 19.2 software was applied for 3D steady state and four different types of configurations with configuration “centre”, “up”, “down” and “up down” with two different slat angle of louvre which is 15 and 30°. Further, for a slat angle of 15° RNG k-epsilon, using the results, the maximum velocity of the air ventilation is 0.45 m/s for the louvres at the middle and the minimum velocity of 0.407 m/s for the middle louvres. Subsequently, for a 15° slat angle SST k-omega, according to the outcome, the lowest velocity achieved for RNG k-ω is 0.402 m/s for centre louvres and the highest velocity is 0.477 m/s for down louvres. In addition, for a slat angle of 30° RNG k-epsilon, based on the result obtained, the up louvres get the highest velocity of 0.563 m/s at the slat angle of 30°. Next, for a slat angle of 30° SST k-omega, according to the results, the maximum velocity is 0.569m/s for up louvres due to the wind entering from the louvre hits the top wall and enhances the internal velocity of the ventilation. Other than that, on the velocity contour obtained for both 15 and 30 degrees of slat, it is found that 15 degrees of slat provides good air flow inside the air ventilation with the louvers placed at down but provides a slightly low velocity inside the ventilation. Last of all, the comparison contour of different place of louvre with RNG k-ε and SST k-ω model. Based on the obtained results, SST k-ω produced a higher value of velocity than RNG k-ε.

Acknowledgement
This research was supported by Ministry of Higher Education of Malaysia (MOHE) through Fundamental Research Grant Scheme (FRGS) (FRGS/1/2024/TK10/UTHM/02/6) and Universiti Tun Hussein Onn Malaysia (UTHM) through MDR (vot Q686).

References

[1] Cheng, Zheng, Lingling Li, and William P. Bahnfleth. "Natural ventilation potential for gymnasia–Case study of ventilation and comfort in a multisport facility in northeastern United States." *Building and Environment* 108 (2016): 85-98. https://doi.org/10.1016/j.buildenv.2016.08.019

[2] Costanzo, Vincenzo, and Michael Donn. "Thermal and visual comfort assessment of natural ventilated office buildings in Europe and North America." *Energy and Buildings* 140 (2017): 210-223. https://doi.org/10.1016/j.enbuild.2017.02.003

[3] Mohammadi, Murtaza, and John Calautit. "Impact of ventilation strategy on the transmission of outdoor pollutants into indoor environment using CFD." *Sustainability* 13, no. 18 (2021): 10343. https://doi.org/10.3390/su131810343

[4] Anaebonam, C. Kenneth, Jude Barnaby, and Callistus Okafor. "Exploring the use of natural ventilation strategies to create a healthy indoor environment." *International Journal of Innovative Environment Studies Research* 11, no. 3 (2023): 109-117.

[5] Ahmed, Tariq, Prashant Kumar, and Laetitia Mottet. "Natural ventilation in warm climates: The challenges of thermal comfort, heatwave resilience and indoor air quality." *Renewable and sustainable energy reviews* 138 (2021): 110669. https://doi.org/10.1016/j.rser.2020.110669

[6] Belleri, Annamaria, Marta Avantaggiato, and Roberto Lollini. "Ventilative cooling in shopping centers' retrofit: the Mercado del Val case study." *Energy Procedia* 111 (2017): 669-677. https://doi.org/10.1016/j.egypro.2017.03.229

[7] Tao, Yao, Haihua Zhang, Lili Zhang, Guomin Zhang, Jiyuan Tu, and Long Shi. "Ventilation performance of a naturally ventilated double-skin façade in buildings." *Renewable Energy* 167 (2021): 184-198. https://doi.org/10.1016/j.renene.2020.11.073

[8] Tablada, Abel, Jan Carmeliet, Martine Baelmans, and Dirk Saelens. "Exterior louvers as a passive cooling strategy in a residential building." In *26th conference on passive and low energy architecture, Quebec City, Canada*. 2009.

[9] Wu, Yu-Chou, An-Shik Yang, Li-Yu Tseng, and Chin-Lung Liu. "Myth of ecological architecture designs: Comparison between design concept and computational analysis results of natural-ventilation for Tjibaou Cultural Center in New Caledonia." *Energy and Buildings* 43, no. 10 (2011): 2788-2797. https://doi.org/10.1016/j.enbuild.2011.06.035

[10] Ramponi, Rubina, and Bert Blocken. "CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters." *Building and Environment* 53 (2012): 34-48. https://doi.org/10.1016/j.buildenv.2012.01.004

[11] Kosutova, Katarina, Twan van Hooff, Christina Vanderwel, Bert Blocken, and Jan Hensen. "Cross-ventilation in a generic isolated building equipped with louvers: Wind-tunnel experiments and CFD simulations." *Building and Environment* 154 (2019): 263-280. https://doi.org/10.1016/j.buildenv.2019.03.019

[12] Kasim, N. F. M., S. A. Zaki, M. S. M. Ali, N. Ikegaya, and A. A. Razak. "Computational study on the influence of different opening position on wind-induced natural ventilation in urban building of cubical array." *Procedia Engineering* 169 (2016): 256-263. https://doi.org/10.1016/j.proeng.2016.10.031

[13] Bazdidi-Tehrani, Farzad, Shahin Masoumi-Verki, and Payam Gholamalipour. "Impact of opening shape on airflow and pollutant dispersion in a wind-driven cross-ventilated model building: Large eddy simulation." *Sustainable Cities and Society* 61 (2020): 102196. https://doi.org/10.1016/j.scs.2020.102196

[14] Jeong, Seok-Hwan, Hyuk-Min Kwon, Se-Jin Ahn, and Jeong-Hoon Yang. "A study on the improvement of ventilation rate using air-flow inducing local exhaust ventilation system." *Journal of Asian Architecture and Building Engineering* 15, no. 1 (2016): 119-126. https://doi.org/10.3130/jaabe.15.119

[15] Hazem, Ahmed, Merzaka Ameghchouche, and Cherif Bougriou. "A numerical analysis of the air ventilation management and assessment of the behavior of double skin facades." *Energy and Buildings* 102 (2015): 225-236. https://doi.org/10.1016/j.enbuild.2015.05.057

[16] Pilkington, Theo C., Bruce Loftis, Thomas Palmer, and Thomas F. Budinger. *High-Performance Computing in Biomedical Research*. Boca Raton: CRC Press, 1993. https://doi.org/10.1201/9781003068136

[17] Nyende, Jacob, Isaac Enyogoi, John Mango, and Henry Kasumba. "Numerical Simulation of a Two-Dimensional Groundwater Pollute Transport Problem Using Incompressible Steady-State Navier-Stokes Equations and Diffusion-Convection Equations." *Modelling and Simulation in Engineering* 2022, no. 1 (2022): 7419502. https://doi.org/10.1155/2022/7419502

[18] Yau, Yat Huang, Keng H. Chuah, and M. T. Siew. "The Study on Thermal Environment and Airflow Pattern in an UFAD System Under a Cooling Mode." *Arabian Journal for Science and Engineering* 45, no. 2 (2020): 891-908. https://doi.org/10.1007/s13369-019-04184-z

[19] Yazarlou, Tayyebeh, and Esmaeil Barzkar. "Louver and window position effect on cross-ventilation in a generic isolated building: A CFD approach." *Indoor and Built Environment* 31, no. 6 (2022): 1511-1529. https://doi.org/10.1177/1420326X211061685

[20] Tai, Vin Cent, Joseph Wu Kai-Seun, Prasath Reuben Mathew, Lip Kean Moey, Xinwei Cheng, and David Baglee. "Investigation of varying louver angles and positions on cross ventilation in a generic isolated building using CFD simulation." *Journal of Wind Engineering and Industrial Aerodynamics* 229 (2022): 105172. https://doi.org/10.1016/j.jweia.2022.105172