SEMARAK ILMU PUBLISHING

CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

Numerical Investigation on The Effect of Grille Blockage Ratio on Air Flow Characteristics of Air Vents

Samin Enam[1], Muhammad Noor Afiq Witri Muhammad Yazid[1,*], Nor Azwadi Che Sidik[2]

[1] Department of Thermofluids, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia
[2] Malaysia-Japan International Institute of Technology (MJIIT), Universiti Teknologi Malaysia, Jalan Sultan Yahya Petra, Universiti Teknologi Malaysia, 54100 Kuala Lumpur, Federal Territory of Kuala Lumpur, Malaysia

ARTICLE INFO

Article history:
Received 24 july 2022
Received in revised form 25 August 2022
Accepted 25 September 2022
Available online 31 October 2022

Keywords:
Louver; Air Vent; Blockage Ratio; CFD

ABSTRACT

Vents with louvers are an important component in the indoor heating, ventilation, and air conditioning (HVAC) system in providing a degree of freedom to the occupants to direct the air flow and prevent foreign objects from entering or exiting the air ducts. As a result, whether designing air ventilation or any duct design that involves louvers, the influence of louver design cannot be ignored. For vent design, aside from louver angle, blockage ratio is an important factor in air distribution due to its effect on pressure drop and flow distribution to indoor space. The blockage ratio is often described as the ratio of the projected area of the structure in flow direction to the cross sectional area of the domain around the structure. The purpose of this study is to investigate the effect of varying blockage ratios on the air flow characteristics of air vents including velocity and pressure drop. The chosen air vent model for this study is the Proton Wira air vent. Results are obtained using computational fluid dynamics (CFD) analysis and by utilizing the free and easily available open source software via Code-Saturne. The parameters were set according to available literature. The study finds that the pressure drop increases with the increasing blockage ratio. The velocity drops almost 7% for blockage ratio greater than 3 and the pressure drop increases more than 4%.

1. Introduction

Louvers are a system of horizontal or vertical slats that are angled to admit air. They play a huge role in ensuring the air flow around enclosed spaces. Louver design allows for design flexibility and play a role in ensuring quality indoor air. And louver size will also determine the inflow rate [1]. It is well known that, rectangular air ducts used in HVAC systems in automotive vehicles allow the passenger to control air flow and prevent intrusion of foreign objects into car cabin or air duct. Due to louver’s compatibility with rectangular ducts, we see the popularity rise.

Yakubu and Sharples [2] proposed a quadratic relationship between pressure loss and airflow via a louvered system. The study found that at louver angles of 0 to 45 degrees, there is no resistance to airflow. The experiment's drawback is that it only tested smooth hardwood louvers with no

* *Corresponding author.*
E-mail address: mnafiqwitri@utm.my (Muhammad Noor Afiq Witri Muhammad)

https://doi.org/10.37934/cfdl.14.10.3241

inclination. Nakanishi *et al.,* [3] carried out simulation research that compared Yakubu with Sharple [2]. They determined that when the louver angle grows smaller, the pressure differential tends to diminish, and that by adjusting the louver angle by 20 degrees, the pressure drop could be limited to half. Later on, Chinchun *et al.,* [4] conducted a study that effectively uses a porous media model instead of an explicit model for CFD simulation of louvers in ventilated places, and then compares the two studies to show that the difference in velocity is less than 38% and the cost reduction is due to the simplified CFD model.

Faghani *et al.,* [5] found the association between Reynolds number, aspect ratio, and mixing capacity of rectangular cross-section jets based on simulation work. Quinn *et al.,* [6] did a numerical investigation of a turbulence-free square jet and discovered that under similar test conditions, the jet from a sharp-edged square slot spreads quicker than the jet from a circular or round slot. The turbulence properties match the behavior of mean streetwise velocity, and the elliptic treatment of the jet allows for the prediction of mean static pressure. Despite the fact that the aspect ratio was not specified, it is fair to presume that it was 1:1. At the same time, Berg *et al.,* [7] employed a two-equation turbulence model to simulate the turbulent flow field in free jets from rectangular slots. There is still a lot to learn about turbulence modeling. This flow is challenging from a measurement technical perspective, due to high turbulence intensities and high shear, while at the same time developing over a relatively short distance that makes it practical and manageable to work with on a typical lab scale [8]. The way for determining the degree of uncertainty is to run numerous simulations using various turbulence models and examine how modeling affects the outcome. The conventional *k-ε* turbulence model was used to reduce analytical complexity and limit calculative resources [9, 10].

Moreover, a simulation study by Pairan *et al.,* [11] concluded that the blockage ratio gives a significant effect to the spray penetration angle for each type of nozzles and wider the spray angle means a smaller droplet size is produced and more space to distribute the droplets. Although there has been a lot of computational and experimental research into the influence of aspect ratios and louver angles for rectangular ducts, the effect of grilles and louver areas, as well as Reynolds number, for this variable diameter is still mostly absent for tight places [13-15].

In conclusion, a study of the literature reveals that there is few research on the impact of blockage ratio on jet characteristics. The aim of this study is conduct numerical studies to investigate the impacts of grille blockage ratio on airflow characteristics for air vents. It should be noted that the airflow characteristics to be evaluated are in terms of pressure drop, and flow distribution. This study also attempts to validate the jet flow emanating from rectangular vent based on published data. The results obtained will describe the relationship between blockage ratio and air flow characteristics including pressure drop and air velocity. The current work is designed to provide a theoretical foundation for enhancing the aerodynamics of air vents in automotive vehicles.

2. Turbulence Modelling: *k-ε* Model

Standard *k-ε* turbulence models are an industry standard model for most commercial CFD solvers. It is almost ubiquitous in all CFD communities and is found in the bulk of nuclear engineering CFD literatures [16-18]. It is a two-equation model that includes both turbulence and dissipation kinetic energy:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = P - \rho\epsilon + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \rho L_k \tag{1}$$

$$\frac{\partial(\rho\epsilon)}{\partial t}+\frac{\partial(\rho u_j\epsilon)}{\partial x_j}=C_{\epsilon 1}f_1\frac{\epsilon}{k}P-C_{\epsilon 2}f_2\frac{\rho\epsilon^2}{k}+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial\epsilon}{\partial x_j}\right]+\rho L_\epsilon \quad (2)$$

where

$$P=\tau_{ij}\frac{\partial u_i}{\partial x_j} \quad (3)$$

$$\tau_{ij}=\mu_t\left(2S_{ij}-\frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right)-\frac{2}{3}\rho k\delta_{ij} \quad (4)$$

$$S_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \quad (5)$$

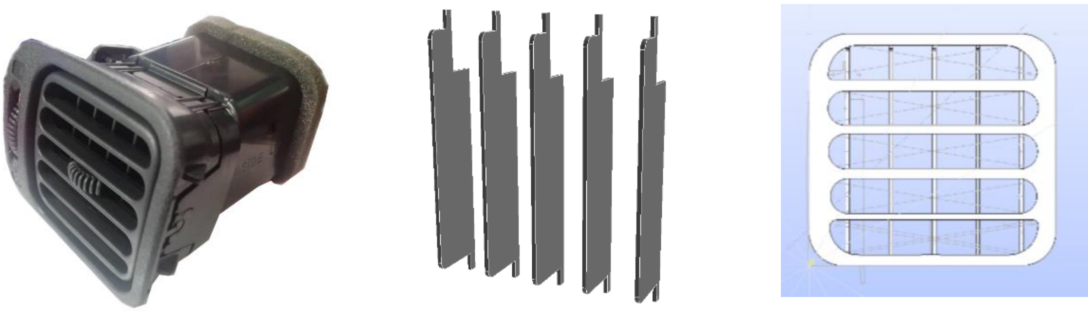
$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \quad (6)$$

$$C_\mu=0.09, C_{\epsilon 1}=1.44, C_{\epsilon 2}=1.92, C_{\epsilon 3}=1\ or\ 0, \sigma_k=1, \sigma_\epsilon=1.3 \quad (7)$$

3. Simulation Setup

3.1 3D Modelling of Air Vent

The vent with louver employed in this study is from Proton Wira, which can be seen in Figure 1 below. The hydraulic diameter of the vent is about 66 mm. Salome was used to create a CAD model of the complete geometry. Because this model is too intricate for CFD calculations, it has been simplified in several parts [19, 20]. This degree of simplification is not expected to have a significant impact on the flow. In addition, the louver is directed normal to vent frontal area, limiting the scope of this study to one directional flow only.



(a) Actual vent with louver (b) CAD model of louver (c) CAD model of vent with louver

Fig. 1. Proton Wira Air Vent

3.2 Computational Domain and Boundary Conditions

The geometry modelling was done using the geom module. The computational geometry matches to the study's reduced-scale geometry [12]. The computational geometry specifically includes the air vent with louver, a 10 cm opening for air ingress, and the enclosed box as shown in

Figure 2. Given their modest thickness of 0.01 m, the louvers are treated as zero-thickness walls in the models.

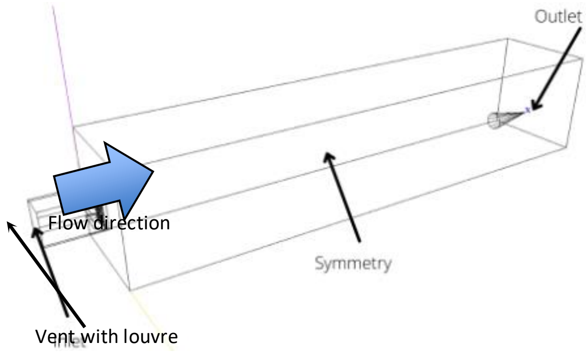


Fig. 2. Geometry Setup

A computational domain was defined with dimensions L_z, L_y and L_x in the lateral (z) spanwise (y), and streamwise (x) directions respectively. The computational domain was constructed using the following dimensions; extended 1000 mm in streamwise from the vent and 260 mm in lateral and spanwise. The inlet is specified about 100 mm in front of the air vent while the outlet is the boundary at the far end of the computational domain. The wall is adjacent to the inlet and the symmetry planes are adjacent to the wall and outlet, which are placed far from the expected jet flow development. The meshing was done using Salome mesh module with unstructured tetrahedral type.

For boundary conditions, four different inlet velocities are specified; 1.7 m/s, 2.57 m/s, 3.5 m/s and 5 m/s (equivalent to 0.027, 0.041, 0.056 and 0.08 m^3/s of flowrates), a range of airflow rates typically found in most sedan HVAC systems. Meanwhile, the value of turbulent intensity used is 0.05%. As from previous literature, we conclude that turbulent intensity of 0.01% did not show good agreement with experimental results.

3.3 Fluid Properties, Turbulence Model and Numerical Setup

The working fluid considered in this study is air at 293.15 K where the fluid properties are specified as in Table 1. Standard *k-ε* turbulence model with standard wall function have been used to resolve turbulence flow across the vent with louver. Hence, standard *k-ε* turbulence model is chosen in this study due to the proven accuracy that could be provided by this model [7]. Turbulent flow is often three-dimensional and time-varying, whereas laminar flow is typically two-dimensional and time-varying mainly during boundary layer transitions. In this study, the flow have been assumed to be unsteady to appropriately resolve the flow fluctuations near the louver. Hence, a sufficiently small time step have been chosen in accordance with courant number requirements of 1.

Table 1
Input Parameters for CFD Simulations

Parameter	Value
Temperature	293.15 K
Total pressure	101325 Pa
Density	1.2754 kg/m^3
Viscosity	0.00001625 Pa.s
Specific heat	1017.24 J/Kg/K

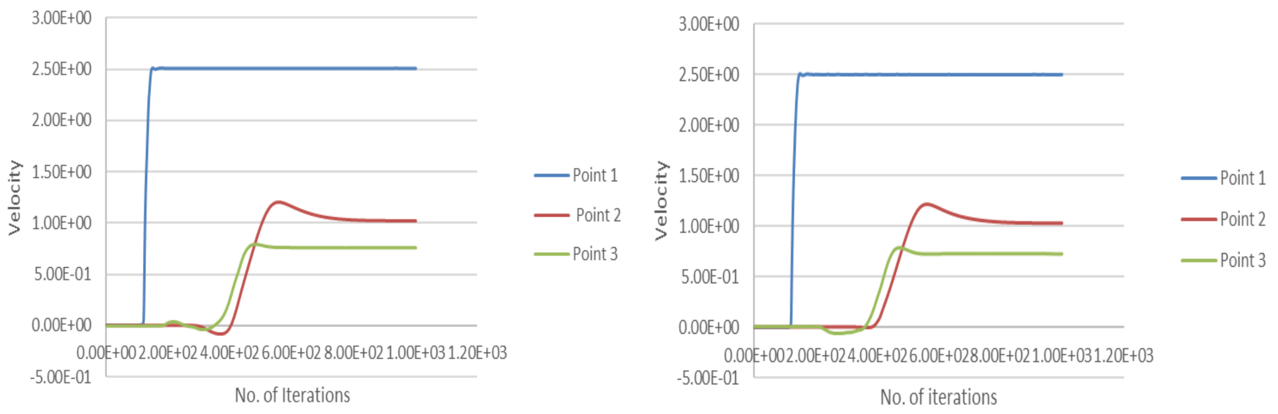
3.4 Verification Study

In order to ascertain whether the simulation is free from discretization error, four different meshes were constructed and tested at different time step, for fulfilling the courant number. Using different mesh sizing and minimum and maximum size of mesh, and changing the mesh type, various mesh was generated. The total mesh element and corresponding time step being tested are shown in Table 2.

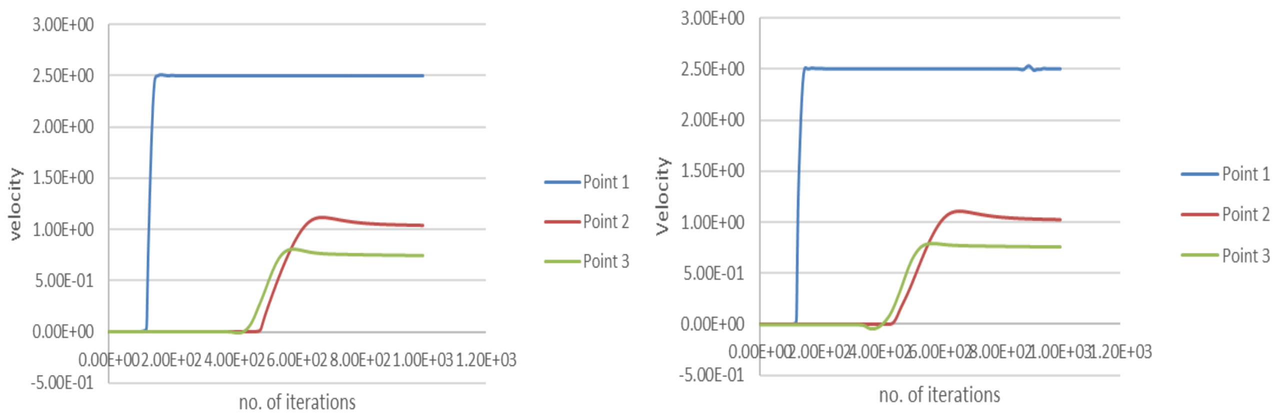
Table 2
Mesh count and time step

Mesh label	Mesh count	Time step (s)
M1	280,000	0.01
M2	320,000	0.05
M3	510,000	0.0025
M4	830,000	0.00125

In this study, the unsteady flow was simulated for more than 60 seconds in order to find the exact time the flow becomes steady. This steady flow when achieved is also termed as fully developed flow evaluated at three different points of the following coordinates; $x/D = 10$ and $y = 0$, $x/D = 20$ and $y = 0$, and $x/D = 20$ and $y = 2D$. In order to find the number of iterations required for flow to fully converge, the four meshes M1, M2, M3 and M4 were simulated for 1000 iterations at different time step values. The data for all four meshes were graphed against the velocity and number of iterations. It was found that for point 1, which is nearest to the inlet, the flow converges in 200 iterations (see Figure 3). But for coordinates farther away from the inlet, the number of iterations increases significantly. In addition, mesh 3 (M3) were not much different from mesh 4 (M4) especially for velocity data at point 2. Hence, mesh 3 is considered to be mesh independent.



(a) (b)



(c) (d)

Fig. 3. The results of mesh convergence (a) M1 (b) M2 (c) M3 (d) M4

4. Results and Discussion

For the validation study, the findings of Faghani *et al.,* [5] was taken into account because the Reynolds number range in the literature was found to be the same as the Reynolds number range in this study. The validation study is performed to ensure the jet flow from the vent without louver can be accurately captured using the chosen turbulence model and specified boundary conditions. As mentioned earlier, the model for validation was created using the dimensions according to Faghani *et al.,* [5]. In this case, the streamwise domain length is 100 times greater than the hydraulic diameter while the spanwise and lateral domain length is 20 times longer than hydraulic diameter. The results were compared for profile $x/D = 10$ as shown in Figure 4. The data shows good alignment with the data of the literature.

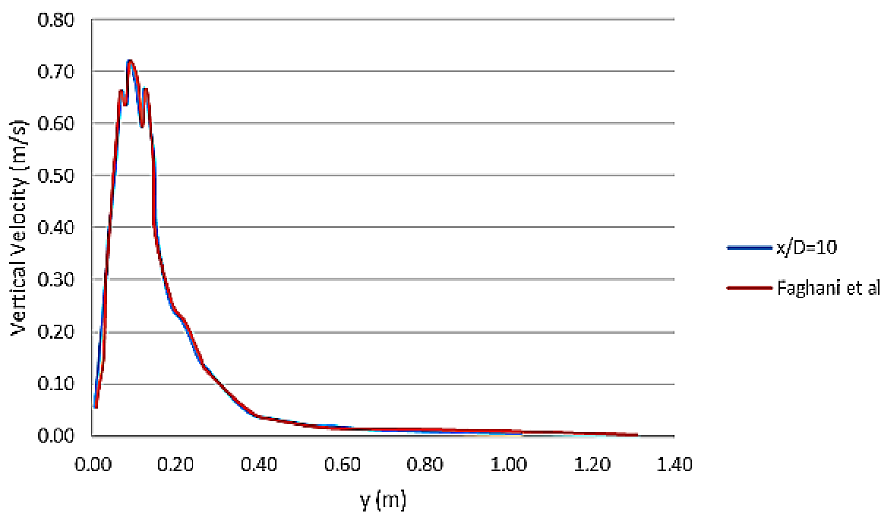
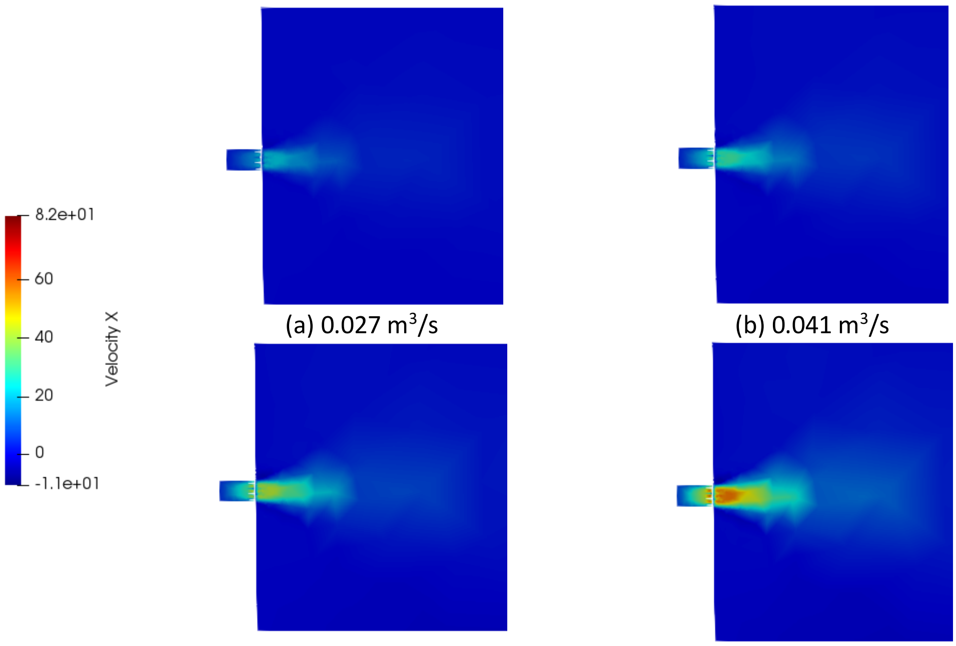


Fig. 4. Validation results

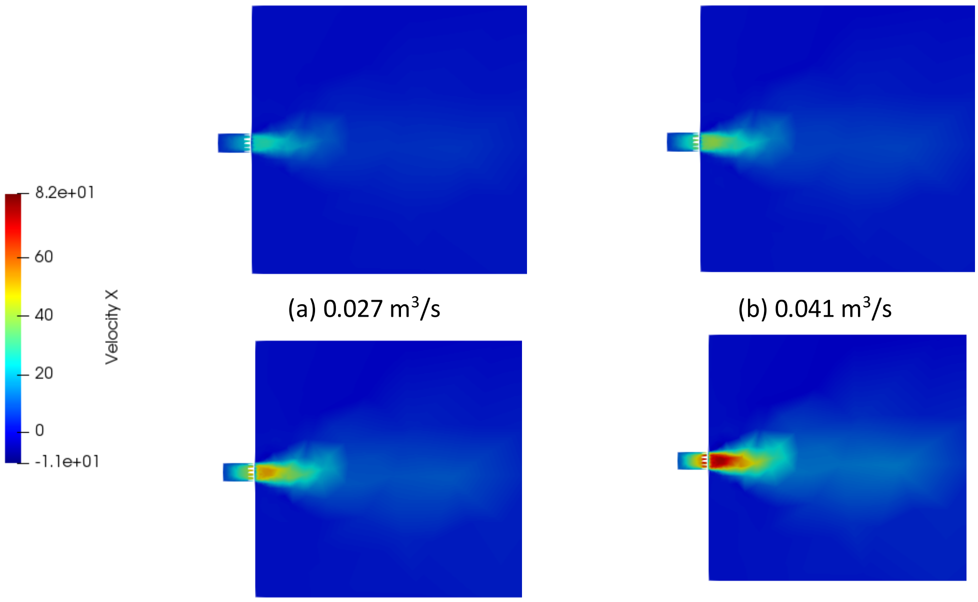
The blockage ratio is defined as the ratio of the louver's frontal area to the cross sectional area of the vent. In this study, different louver counts corresponds to different blockage ratio; 4, 5 and 6 louvers are equivalent to 2.25, 2.82, and 3.38 of blockage ratio, respectively. The velocity contours seen in the Figure 5, 6 and 7 below clearly show that the jet flow development due to blockage ratio of 6 louver vent, creates uneven velocity contour at the inlet. The jet development due to 5 Louver vent seems to be more well developed, and it travels a further distance compared to 4 louver and 6 louver vents. The velocity flow also seems to be more evenly distributed throughout the confined area. This can be held true for all flow rates. We can also observe clearly from the velocity contours that as the flow rate increases, the air also travels much farther. As can be seen for flow rate of 0.08 m^3/s, the air disperses through the vent to a much further distance as compared to 0.027 m^3/s. As for the blockage ratio, we can clearly see in Figure 6 that the air flows more evenly and much further for blockage ratio of 2.82. But as soon as the blockage increases more than 3, the flow disperses much less evenly and travels a smaller distance. This can be clearly seen in Figure 7 for the 6 louver vent. Hence, the velocity contours help to better visualize the effect of blockage ratio and flow rate on air flow characteristics. This visualization proves some key points including the fact that the 5 louver vent allows for smoother and faster air flow.



(a) 0.027 m^3/s (b) 0.041 m^3/s

(c) 0.056 m^3/s (d) 0.08 m^3/s

Fig. 5. Velocity contour for 4 louver vent at different flowrates



(a) 0.027 m^3/s (b) 0.041 m^3/s

(c) 0.056 m^3/s (d) 0.08 m^3/s

Fig. 6. Velocity contour for 5 louver vent at different flowrates

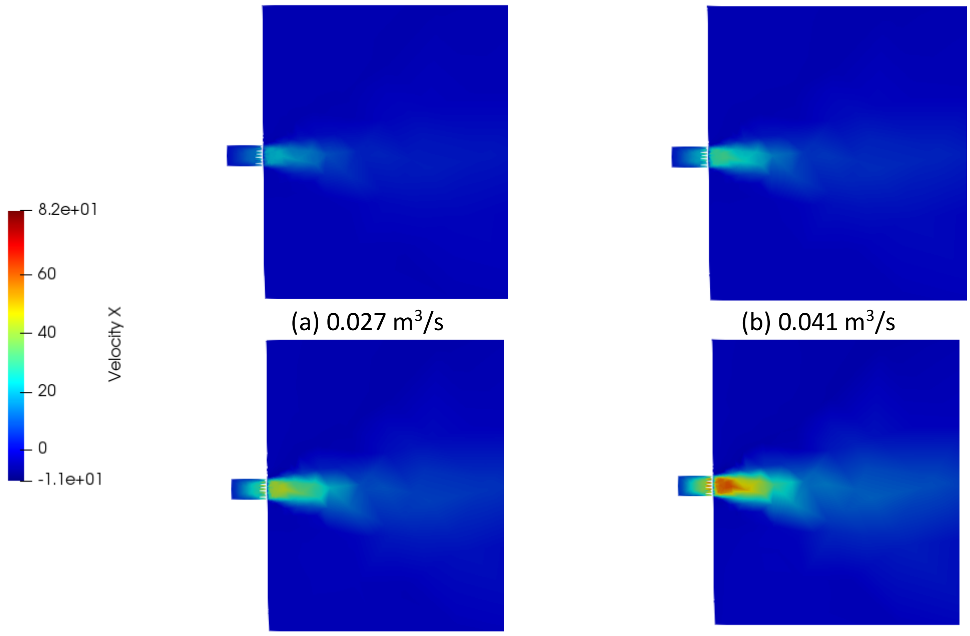


Fig. 7. Velocity contour for 6 louver vent at different flowrates

We can also see from Figures 5, 6 and 7 that the increasing blockage ratio also effects the jet flow of the air through the vent. From Figure 6, we can conclude that the 5 louver vent increases the volumetric flow at a given section of the jet and increases the width of the jet causing spread. But, as the blockage ratio increases to 3.38 in Figure 7, the volumetric flow decreases and the width of the jet also decreases causing spot air flow. These comparisons paint a very clear picture regarding the effect that blockage ratio can have on volumetric flow and jet development. The 5 louver vent hence can provide greater comfort when compared to 6 or 4 louver vents.

Pressure drop across the louver are affected by the flow rate and blockage ratio. The larger the flow rate is, the greater the pressure drop through a restriction is, as shown in Figure 8. Conversely, when the flow rate decreases, so does the pressure drop. Our results also implies that the vent with larger restriction or blockage ratio will require higher energy to operate the blower at the same flow rate attributed by the larger pressure drop. That means, the vent having 6 louvers will require far more energy to operate in comparison to the vent with 4 louvers. Typically, pressure drop corresponds with sound; if there is a significant pressure drop, more noise from the ventilation equipment will be heard. When the pressure drop is modest, the equipment is quieter.

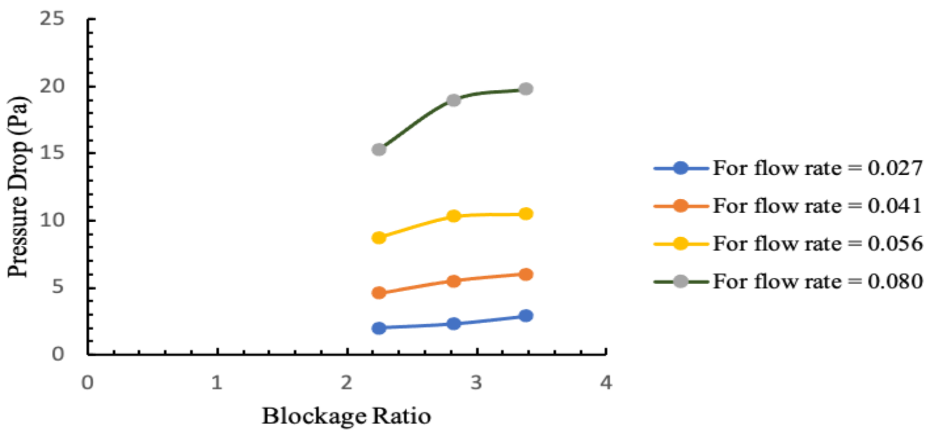


Fig. 8. Graph of pressure drop (Pa) against blockage ratio at different flow rates

From Figure 9, the maximum velocity for each blockage ratio increases, but when the blockage ratio changes from 5 louver vent to 6 louver vent, the maximum velocity takes a steep decline. This proves that the restriction caused by a huge blockage ratio, in this case 6 louvers, decreases the air speed in the vent. But, when the blockage ratio changes from 4 louvers to 5 louvers, the velocity does not decrease, but it increases. That further proves that the increasing blockage ratio will obstruct regular flow and cause hindrance in assuring comfort to passengers.

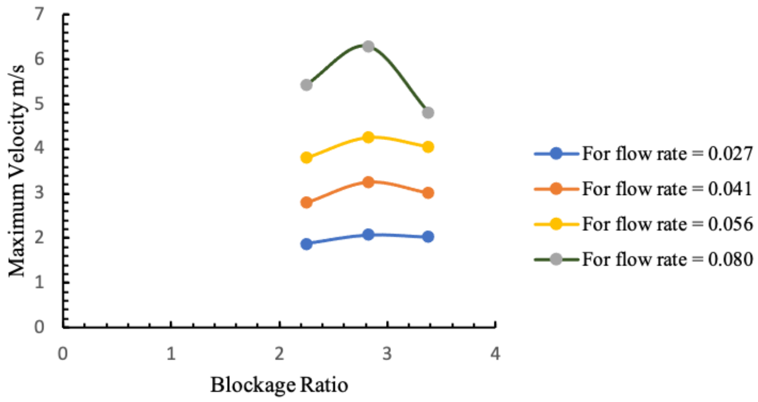


Fig. 9. Graph of maximum velocity against blockage ratio at varying flow rates

5. Conclusion

This study investigates how the size of the louver and the blockage induced by the louver impacts the air flow characteristics in any restricted environment, filling a gap in the current literature. HVAC design is a difficult endeavor that necessitates well-informed design decisions and the effective use of analysis tools throughout the design process. The advancements in computational design have enabled the integration of open source and easily accessible software to analyze the role of louvers in ventilation, but a framework for their integrated usage is still required. There are several studies in the literature that focus on louver angles and vent design, but ventilation and blockage analysis are understudied. By combining 3D design, CFD, and open source software analysis, this research produced a number of conclusions that will have a substantial impact on comfort and energy requirement. The study may be seen as an attempt to better understand air flow characteristics while also giving a step-by-step tutorial on how to undertake CFD studies utilizing open source, free, and easily accessible CFD software. The main conclusions of study can be as follows:

i. The study finds that the pressure drop increases with the increasing blockage ratio. At a blockage ratio greater than 3 *i.e.,* for 6 louver vent, the pressure drop increases significantly at the inlet. Compared to 5 louver vent, the pressure drop increases more than 4% for 6 louver vent.
ii. The velocity significantly drops for 6 louver vent, meaning the current 5 louver vent will provide thermal comfort as opposed to 6 louver or 4 louver vent.

References

[1] Kit, Jonathan Ho Siew, Chong Kok Hing, Basil T. Wong, Victor Bong Nee Shin, Lee Man Djun, and Christopher Jantai Anak Boniface. "Numerical Simulation of Alternative Smoke Control Approach in a High-Rise Building." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 80, no. 2 (2021): 1-12. https://doi.org/10.37934/arfmts.80.2.112

[2] Yakubu, G. S., and S. Sharples. "Airflow through modulated louvre systems." *Building Services Engineering Research and Technology* 12, no. 4 (1991): 151-155. https://doi.org/10.1177/014362449101200405

[3] Nakanishi, Toshikazu, Tamotsu Nakamura, Youichirou Watanabe, Katsumasa Handou, and Takahiro Kiwata. "Investigation of air flow passing through louvers." *Komatsu Tech. Rep.* 53 (2007): 1-9.

[4] Ooi, Chinchun, Pao-Hsiung Chiu, Venugopalan Raghavan, Stephen Wan, and Hee Joo Poh. "Porous media representation of louvers in building simulations for natural ventilation." *Journal of Building Performance Simulation* 12, no. 4 (2019): 494-503. https://doi.org/10.1080/19401493.2018.1510544

[5] Faghani, Ehsan, Reza Maddahian, Pedram Faghani, and Bijan Farhanieh. "Numerical investigation of turbulent free jet flows issuing from rectangular nozzles: the influence of small aspect ratio." *Archive of applied mechanics* 80, no. 7 (2010): 727-745. https://doi.org/10.1007/s00419-009-0340-z

[6] Quinn, Willie R., and J. Militzer. "Experimental and numerical study of a turbulent free square jet." *The Physics of fluids* 31, no. 5 (1988): 1017-1025. https://doi.org/10.1063/1.867007

[7] Berg, J. R., S. J. Ormiston, and H. M. Soliman. "Prediction of the flow structure in a turbulent rectangular free jet." *International communications in heat and mass transfer* 33, no. 5 (2006): 552-563. https://doi.org/10.1016/j.icheatmasstransfer.2006.02.007

[8] Yaacob, Mohd Rusdy, Rasmus Korslund Schlander, Preben Buchhave, and Clara M. Velte. "Experimental evaluation of kolmogorov's-5/3 and 2/3 power laws in the developing turbulent round jet." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 45, no. 1 (2018): 14-21.

[9] Archambeau, Frédéric, Namane Méchitoua, and Marc Sakiz. "Code Saturne: A finite volume code for the computation of turbulent incompressible flows-Industrial applications." *International Journal on Finite Volumes* 1, no. 1 (2004).

[10] Bhandari, D., and S. Singh. "Analysis of fully developed turbulent flow in a pipe using computational fluid Dynamics." *International Journal of Engineering Research and Technology* 1, no. 5 (2012): 1-8.

[11] Pairan, Mohamad Rasidi, Sharul Azmir Osman, Ahmad Nabil Md Nasir, Nur Hazirah Noh, Mohd Hizwan Mohd Hisham, Adjah Naqkiah Mazlan, Hanifah Jambari, and Muhamad Afzamiman Aripin. "The Blockage Ratio Effect to The Spray Performances." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 95, no. 1 (2022): 99-109. https://doi.org/10.37934/arfmts.95.1.99109

[12] Medina, Ricardo, Ashkan Motamedi, Murat Okcay, B. Uygar Oztekin, Gustavo Borel Menezes, and Arturo J. Pacheco-Vega. "On the implementation of open source CFD system to flow visualization in fluid mechanics." In *2012 ASEE Annual Conference & Exposition*, pp. 25-995. 2012.

[13] Sasongko, N., and M. Arif. "Open source computational fluid dynamic: challenges and its future." In *Conference on Open Source, Jakarta, Indonesia*. 2009.

[14] Saleem, Arslan, and Man-Hoe Kim. "CFD analysis on the air-side thermal-hydraulic performance of multi-louvered fin heat exchangers at low Reynolds numbers." *Energies* 10, no. 6 (2017): 823. https://doi.org/10.3390/en10060823

[15] Akamine, Yoshihiko, Takashi Kurabuchi, Masaaki Ohba, Tomoyuki Endo, and Motoyasu Kamata. "A CFD analysis of the air flow characteristics at an inflow opening." *International Journal of Ventilation* 2, no. 4 (2004): 431-437. https://doi.org/10.1080/14733315.2004.11683684

[16] Qing, Nelvin Kaw Chee, Nor Afzanizam Samiran, and Razlin Abd Rashid. "CFD Simulation analysis of Sub-Component in Municipal Solid Waste Gasification using Plasma Downdraft Technique." *Journal of Advanced Research in Numerical Heat Transfer* 8, no. 1 (2022): 36-43.

[17] Oberkampf, William L., and Timothy G. Trucano. "Verification and validation in computational fluid dynamics." *Progress in aerospace sciences* 38, no. 3 (2002): 209-272. https://doi.org/10.1016/S0376-0421(02)00005-2

[18] Gaioni, Valerio and Fernandez-Cosials, Mikel. (2020). "A Practical Verification and Validation Procedure for Computational Fluid Dynamics." (2020).

[19] Shukla, M., and Lakshminarasimha, N. "CFD Analysis of Airflow inside a Car Compartment. " *International Journal of Technical Innovation in Modern Engineering & Science* 4, no. 6 (2018).

[20] Rameshkumar, A., S. Jayabal, and P. Thirumal. "CFD analysis of air flow and temperature distribution in an air conditioned car." *International Refereed Journal of Engineering and Science* 2, no. 4 (2013): 1-6.