COMPUTATIONAL FLUID DYNAMIC SIMULATION: MAXIMISE THE PERCENTAGE OF AIR DISTRIBUTION INSIDE QUASI-FLOW HEAT EXCHANGER

S. S.Muhammad[1], A.Y. T. Al-Zubaydi[2]

[1]Faculty of Engineering and Information Technology, University of Technology Sydney, 15 Broadway, Ultimo NSW 2007, Australia.
[2]School of Mechanical and Mechatronic Engineering, University of Technology Sydney, 15 Broadway, Ultimo NSW 2007, Australia.

Article History: Received 12.11.2019; Revised 26.12.2019; Accepted 26.12.2019

ABSTRACT

Parallel plate quasi-flow air-to-air heat exchanger has been implemented in Heat Recovery Ventilation (HRV) system to enhance the operating efficiency even in small temperature difference. In the HRV system, heat energy from the exhaust air is recovered to pre-treat the fresh air coming into the air-conditioning systems. Therefore, HRV system can reduce the power consumption of Heating, Ventilation and Air Conditioning (HVAC) systems by enhancing the performance of the HRV system. Air velocity distribution over the plate of heat exchanger has a strong effect on the HRV system performance. The higher the percentage of area covered with air across the plate, the higher the heat transfer rate between two airstreams.

Inside the plate of the heat exchanger, duct inlet entrance cause air distribution to be non-uniform. Factor affecting air velocity distribution over the plate is investigated to reduce air velocity maldistribution. A simulation using ANSYS is conducted to investigate the effect of different louvres arrangements on the air velocity distribution across the plate of the heat exchanger. Results show that, with a smaller louvre orifice, Mode 1 performs best in term of air velocity distribution, and Turbulence Kinetic Energy (TKE) intensity due to turbulence flow generated at louvre arrangement and complement the effect of pressure drop across the plate. Mode 1 enhances the performance of the HRV system by increasing the percentage of air distribution across the plate with an approximation of 97%.

KEYWORDS*: Heating Recovery Ventilation, Parallel plate quasi-flow heat exchanger, Louvres arrangement, Percentage of air distribution*

*Corresponding Email: SitiSuhaida.BintiMuhammad@gmail.com

1.0 INTRODUCTION

Heating, Ventilation and Air Conditioning (HVAC) system, especially air conditioners, consume a significant amount of electrical power to condition the air quality inside the buildings (Al-Zubaydi & Hong, 2018). When the climate change effects increase, the demands of the HVAC system increase, and the consumption of energy to cooling or heating the air inside the buildings increases (Al-Zubaydi & Hong, 2018). Therefore, researchers developed ways to reduce the consumption of electrical power by increasing the efficiency of the HVAC systems. One leading measure is by implementing Indirect Evaporative Cooling (IEC) into the HVAC system as a standalone cooling system or an Energy Recovery Ventilation (ERV) system (Al-Zubaydi & Hong, 2018). In the HRV and IEC system, the heat energy from the exhaust air is recovered to pre-treat the fresh air coming into the air-conditioning systems. Although the temperature difference between fresh air and exhaust air is relatively small, parallel plate quasi-flow air-to-air heat exchanger plays a vital role in enhancing the overall system performance due to its efficiency in operating at small temperature difference (Al-Zubaydi, Hong, & Dartnall, 2016).

The air-to-air heat exchanger has been categorised into Heating Recovery Ventilators (HRVs) and Energy Recovery Ventilators (ERVs), in which HRVs function is to recovering only sensible heat while ERVs function is to recovering both sensible heat and latent heat (Al-Zubaydi & Hong, 2018). Results from many published studies show a significant impact on the efficiency of the heat exchanger by implementing the HRV system. One study conducted by (Fernández-Seara, Diz, Uhía, Dopazo, & Ferro, 2011) performed experiments on heat recovery system using polymer counterflow heat exchanger to preheat the fresh air (primary air). The experiments concluded that test parameters including temperature, relative humidity and airflow rate of the fresh air contributed to a variety set of heat exchanger's efficiency.

In previous research, the hexagonal-flat-fixed-plate quasi-flow air-to-air heat exchanger is used to investigate the relationship between HRVs and the HVAC system. Hexagonal-flat-plate quasiflow air-to-air heat exchanger (a combination of counterflow and crossflow arrangement on the plate) is used because quasi-flow heat exchanger has the best performance compared to counterflow, crossflow and concurrent flow heat exchanger and is easy to manufacture (Al-Zubaydi & Hong, 2018). The hexagonal shape with the apex angle at the entrance reduces the air turbulence over the plates of heat exchanger (Zhang, 2010).

The surface geometry of the heat exchanger plate has a strong effect on the sensible efficiency in the heat exchanger, mainly due to the turbulence flows generated (Vera & Quintero, 2015). Their study concluded that changing the plate surface geometry to increase the heat transfer rate and system efficiency, will increase the surface area and the air distribution over the plate area. The corrugated plate's surfaces provide ideal counter flow area over the plates compared to flat plate's surface and pinned plate's surface thus increases the efficiency of the heat exchanger (Al-Zubaydi, Hong, & Dartnall, 2016). The corrugated sinusoidal shows a significant increase in term of efficiency when compared to pinned plate's surface, and flat plate's surface, but the percentage of pressure drop across the corrugated surface is higher compared to other plate's surfaces (Al-Zubaydi, Hong, & Dartnall, 2016). Although the efficiency of the corrugated plate's surface is higher, considering the pressure drop generated due to turbulent flow is a necessary action. Thus, to reduce the pressure drop across the plate, flat plate's surface is chosen with modification at inlet entrance to increase the efficiency of the heat exchanger.

Other study conducted by (Chen, Yang, & Luo, 2016) performed experiments on energy recovery ventilation systems in four modes, including air-to-air aluminium heat exchanger in the crossflow arrangement. Their investigation focused on the effect of condensation in the heat exchanger channels with high Relative Humidity (RH) in the secondary air stream. They discovered that, due to the significant amount of latent heat, HRV system acted as an indirect evaporative cooling (IEC) system. HRV system and IEC system both utilise the heat energy from exhaust air (secondary air) to condition the fresh air (primary air) by utilising the air-to-air heat exchange between two air streams (Al-Zubaydi, Hong, & Dartnall, 2016). As one of the energy recovery ventilation and evaporative cooling system, the HRV system has been the main subject for future improvement not only to increase its efficiency but to reduce the energy consumption of the HVAC system.

According to the current studies, fluid distribution over the plate of heat exchanger has a strong effect on HRV system performance. However, only a few publications have reported this topic (Al-Zubaydi & Hong, 2019). Fluid distribution over an area is enhanced by spray nozzles and, to maximise the contact area between two air streams which are exhaust air (secondary air) and fresh air (primary air) over the hexagonal plates, the air stream passages (channel) design will be the main focus of this study. Two different modes of louvres arrangement will be analysed using Computational Fluid Dynamic, which is ANSYS Fluid Fluent to compare the evaluation indices results obtained to determine which mode of arrangement contribute to better performance of the heat exchanger thus minimising the fluid maldistribution. Evaluation indices are in terms of percentage of air distribution, pressure drop and Turbulence Kinetic Energy (TKE) intensity due to turbulence flow generated at the louvre arrangement, and the efficiency of the heat exchanger utilising a different mode of louvre arrangement. The hypothesis of this research is, the design that shows the maximum air distribution over the plate while performing best in other areas of investigation will contribute to higher heat transfer rate between two airstreams and higher efficiency of the heat exchanger.

2.0 PARALLEL PLATE HEAT EXCHANGER MODEL AND CFD MODELLING

This study is investigated using CFD simulation, which is ANSYS Fluid Fluent version 19.2. CFD simulation is a powerful engineering numerical simulation tool used for modelling the flow conditions by applying partial differential equations. With the increasing computer capacities and the number of researches apply the CFD for modelling different flow types, CFD modelling application continuously developing and emerging to ease users. Based on researches by (Al-Waked, Nasif, Morrison, & Behnia, 2013; Montazeri, Blocken, & Hensen, 2015; Pakari & Ghani, 2019; Yaïci, Ghorab & Entchev, 2013), 3D CFD prediction capacity and accuracy becoming more realistic to the experimental results. Therefore, to study the effect of louvres arrangement on the percentage of air distribution over the plate of the heat exchanger, ANSYS Fluid Fluent will be the software to run the simulation. The analytical results of the evaluation indices obtained from the simulation will be used to determine which mode contribute to a maximum percentage of air velocity distribution over the plate.

2.1 Physical Model

Figure 1 shows HRV's hexagonal-fixed-plate quasi-flow air-to-air heat exchanger. Material to fabricate the plate is plastic because the manufacturing processes of the plastic is least complicated and low-cost compared to Aluminium. Since louvres are position at the inlet duct, the study parameters will calculate all regions contacted by air. For a better analysis of the effect of louvres arrangement on the percentage of air distribution across the plate, the hexagonal plate heat exchanger is divided into three sections; inlet duct, plate and outlet duct. Air distribution across the plate is our main concern since this is where heat transfer occurs between exhaust air and fresh air.

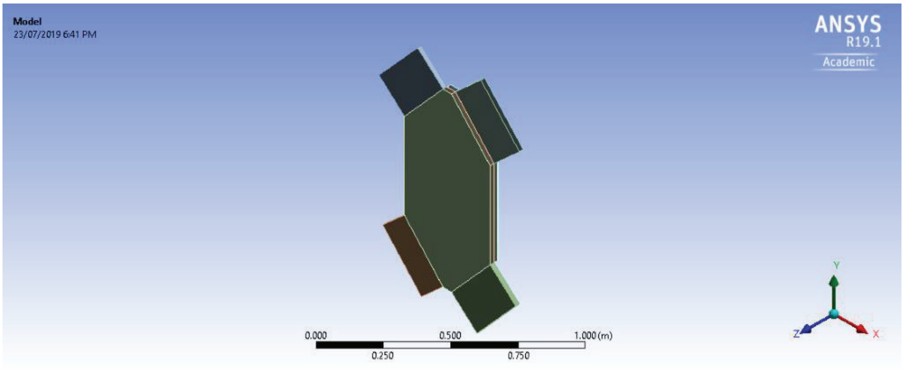

Figure 1 HRV's hexagonal-fixed-plate quasi-flow air-to-air heat exchanger

The hexagonal plate is a quasi-counter-flow plate that provides both counter flow and cross flow paths for the air (Dvořák & Vít, 2015). Theoretically, the plate for exhaust air is mounted alternately with the plate for fresh air in parallel arrangement to maximise the heat transfer rate between two airstreams. According to Figure 1, blue and red straight line illustrate the designated direction of the airstream. The air passes through the inlet duct before entering the crossflow region, then travel through counter-flow region to re-enter crossflow region before leaving the heat exchanger.

The design modelling stage focused on channel modification at the inlet duct to enhance the inlet velocity. The dimension and point of location for louvres in both modes are similar. As shown in Figure 2, six louvres are arranged inside the inlet duct with different direction angles for Mode 1 and Mode 2. The louvre orifice in Mode1 is smaller than louvre orifice in Mode 2.

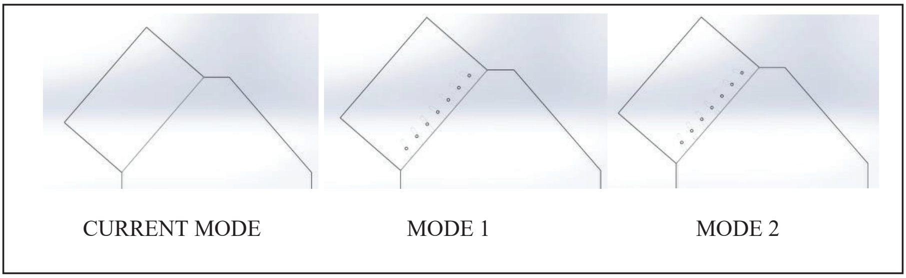

Figure 2 Design Comparison between Current Mode, Mode 1 and Mode 2

2.2 CFD Modelling

Although the CFD simulation will compute more time and resources, a validated CFD model will provide a precise design tool that can imitate a practical experimentation, with regards to the geometry of HRV's quasi-flow heat exchanger, the software and governing equations implemented, followed by the initial settings of the problem and the boundary conditions.

For better analysis in investigating the effect of different louvres arrangements on the velocity distribution across the plate of the heat exchanger, the assumptions for this study are as follow:

- Air is incompressible fluid
- Fluid at the inlet duct, near the louvre arrangement, is under turbulent flow
- With the changing temperature, the properties of fluid remain the same
- The effect of viscous dissipation of the fluid is ignored
- Fluid flows evenly at the inlet duct before entering the hexagonal plate
- The dimension of louvres is identical for both modes
- Heat exchange between the internal fluid and external fluid is ignored
- Frictional resistance and loss of fluid due to louvres arrangement is ignored
- This process is steady-state process of heat and mass transfer

2.2.1 Numerical Approach

The Direct Numerical Simulation (DNS) has no averaging or approximation, and the instant turbulence quantities can be obtained directly. DNS is considered as the most fundamental approach to solving the conservation equations of turbulent flows. Nevertheless, the drawback is, it requires intensive computational processing even for a simple flow configuration.

The Reynolds-averaged Navier Stokes (RANS) is based on the averaging of the conservation equations to model the flow turbulence. The RANS is commonly used to solve Navier-Stokes equations of a turbulence fluid flow instead of DNS due to fewer computational resources. The RANS is capable to handle the turbulence with the present of louvres in the inlet duct. However, turbulence stress and turbulent flux and other variables are introduced to the RANS due to Reynolds based averaging. Therefore, an additional turbulence model, standard k−ε turbulence model, is required when using the RANS approach to model turbulent flow fields.

2.2.2 Turbulence Modelling

The standard k−ε turbulence model is the second most accurate turbulence model flow after the Reynolds Stress Model (RMS). The standard k−ε turbulence model is simple, accurate, low computational time, which make it suitable in the simulation of engineering application that involves two-phase flow with a wide range of turbulence flows. This model was applied in the CFD simulation of HRV and IEC systems, cooling towers, heat exchangers and air conditioning systems with validated outcomes (Al-Waked, Nasif, Morrison, & Behnia, 2013; Alkhedhair, 2015; Pakari & Ghani, 2019; Saraireh, 2012; Sun, Guan, Gurgenci, Li, & Hooman, 2017). Therefore, standard k−ε turbulence model will be used to assist the study.

2.2.3 Solver Setting

In ANSYS version 19.2 software package, FLUENT is standard fluid flow analysis tool due to its flexibility, robustness and accuracy compared to other available CFD software package (Greifzu, Kratzsch, Forgber, Lindner, & Schwarze, 2016; Zou, Zhao & Chen, 2018). FLUENT is used to construct and modelling the 3D computational model using the governing equations of the fluid flow with the Finite Volume Method (FVM). The standard k−ε turbulence model is used to model the air turbulence model.

2.2.4 Governing Equations

The governing equations for the air as a continuous flow modelled with RANS and standard k-ε turbulence mode are given in Eulerian modelling as (Sun, 2019):

$$\frac{\partial(\rho_{air} v_{aj})}{\partial y_i} = S_m \tag{1}$$

$$\rho_{air} \frac{\partial(v_{ai} v_{aj})}{\partial} = \rho_{air} g_j - \frac{\partial P}{\partial y_j} + \frac{\partial \tau_{ij}}{\partial y_j} - \frac{\partial}{\partial y_j}\left(\rho_a \overline{v_{ai} v_{aj}}\right) + S_{mo} \tag{2}$$

$$\rho_{air} v_{ai} \frac{\partial E}{\partial y_i} = -p \frac{\partial v_{ai}}{\partial y_i} + \frac{\partial}{\partial y_i}\left(K_a \frac{\partial T_a}{\partial y_i}\right) - \frac{\partial}{\partial y_i}\left(\rho_{air} c_{pa} \overline{v_{ai} T_a}\right) + \phi + S_e \tag{3}$$

$$\rho_{air} v_{ai} \frac{\partial Y_i}{\partial y_i} = -\frac{\partial}{\partial y_i}\left(\rho_{air} D_f \frac{\partial Y_j}{\partial y_i}\right) - \frac{\partial}{\partial y_i}\left(\rho_{air} \overline{v_{ai} Y_i}\right) + S_m \tag{4}$$

Where ϕ is the viscous dissipation in (W/m^3), S_e is the source term of droplet energy in (W/m^3), the S_m is the source term of droplet mass in (Kg/m^{3}s) and S_{mo} is source term of droplet momentum in (Kg/m^{2}s^2), respectively.

The τ_{ij} is the stress tensor in Kg/m^{2}s and represented by (Sun, 2019):

$$\tau_{ij} = \mu_t \left(\frac{\partial v_{aj}}{\partial y_i} + \frac{\partial v_{ai}}{\partial y_j} - \frac{2}{3}\delta_{ij} \frac{\partial v_{ai}}{\partial y_i}\right) \tag{5}$$

As described by Alkhedhair (Alkhedhair, 2015) and Sun (Sun, 2019), the RANS approach components are:

$$\rho_a \overline{v_{ai} v_{aj}} = \mu_t \left(\frac{\partial v_{aj}}{\partial y_i} + \frac{\partial v_{ai}}{\partial y_i} - \frac{2}{3}\rho_{air} k \delta_{ij}\right) \quad \text{is the RANS turbulent stresses} \tag{6}$$

$$\rho_{air} c_{pa} \overline{v_{ai} T_a} = -\mu_t \frac{c_{pa}}{P_{r_1}} \frac{\partial T_a}{\partial y_i} \quad \text{is the RANS turbulent heat fluxes} \tag{7}$$

and

$$\rho_{air} \overline{v_{ai} Y_i} = -\mu_t \frac{1}{S_{c_t}} \frac{\partial Y_j}{\partial y_i} \quad \text{is the RANS turbulent mass flux} \tag{8}$$

Where k is the turbulence kinetic energy in (J/kg), P_{r1} is the Prandtl number and S_{ct} is the Schmidt number. While the μ_t is the turbulent viscosity in (Kg/m.s) and given by:

$$\mu_t = \rho_{air} c_\mu \frac{k^2}{\varepsilon} \tag{9}$$

Where the ε is the turbulent dissipation rate in (m^2/s^3) and the c_μ is an empirical constant for the standard k-ε turbulence mode defined by Launder and Spalding (Launder & Spalding, 1972).

The turbulent kinetic energy is represented by:

$$\frac{\partial(\rho_{air} v_{ai} k)}{\partial y_i} = \left[\left(\mu_{air} + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial y_i}\right] + G_k - \rho_{air}\varepsilon \quad (10)$$

While the turbulent dissipation of the kinetic energy can be written as :

$$\frac{\partial(\rho_{air} v_{ai} \varepsilon)}{\partial y_i} = \left[\left(\mu_{air} + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial y_i}\right] + \frac{\varepsilon}{k}(C_1 G_k - C_2 \rho_{air}\varepsilon) \quad (11)$$

Where the G_k is the production of turbulent kinetic energy and expressed by:

$$G_k = \tau_{ij}\frac{\partial v_{ai}}{\partial y_i} \quad (12)$$

C_1, C_2, C_μ, σ_k and σ_ε are the model constants used in the standard k-ε turbulence model given by Launder and Spalding (Launder & Spalding, 1972) used by Alkhedhair (Alkhedhair, 2015) and Sun (Sun, 2019) are shown in Table 1.

Table 1 Continuous phase turbulence model constants

C_1	C_2	C_μ	σ_k	σ_ε
1.44	1.92	0.09	1.00	1.30

σ_k, σ_ε, $C_{1\varepsilon}$, $C_{2\varepsilon}$ and C_μ are adjustable constants. The enhanced wall treatment method specifies the boundary values for the turbulent quantities near the wall are chosen for iteration of data fitting for a wide range of turbulent flows. The governing equations above are solved using the commercial computational fluid dynamics code FLUENT 19.2 with the standard $k - \varepsilon$ turbulence model.

3.0 RESULTS AND DISCUSSION

ANSYS computational fluid dynamics code FLUENT 19.2 was used to simulate the percentage of the air distribution across the hexagonal plate air-to-air heat exchanger. To examine the effect of air inlet velocity on the evaluation indices; velocity distribution, pressure loss and turbulence kinetic energy, the visual and analytical results of two different modes of louvres arrangement are compared. The simulation results will determine which mode perform best in maximising the percentage of air distribution while performing best in other evaluation indices.

3.1 Effect of Air Velocity Inlet on the Percentage of Air Distribution

Duct channel at inlet duct causes the air distribution to be non-uniform. Factors affecting the air velocity distribution over the plate of the heat exchanger are investigated to reduce air velocity maldistribution. Louvres arrangement works similarly to fin arrangement that extends the path of airstreams according to the direction of the louvres are arranged. Based on Figure 4, air velocity distribution in Mode 1 and Mode 2 is distributed uniformly after the airstream leaves the louvres. Velocity is distributed uniformly due to the existent of louvres that direct the airstream to designated directions. The increase in air velocity at the inlet may enhance the air velocity distribution over the plate of the heat exchanger. Figure 3 illustrates that the velocity distribution magnitude increases linearly with the increase of air velocity at the inlet. At the higher value of air velocity at the inlet duct, the effect on the velocity distribution magnitude becomes much stronger. According to Figure 3, with the increasing air velocity at the inlet, the velocity distribution magnitude, also known as flow rate increases. The increment of flow rate in Mode 2 is higher than Mode 1 by 3.15%. The increment of flow rate in Mode 2 is higher than the current model by 17.74%. The increment of flow rate in Mode 1 is higher than the current model by 15.1%. In this case, Mode 2 performs better than Mode 1 by 2.64%.

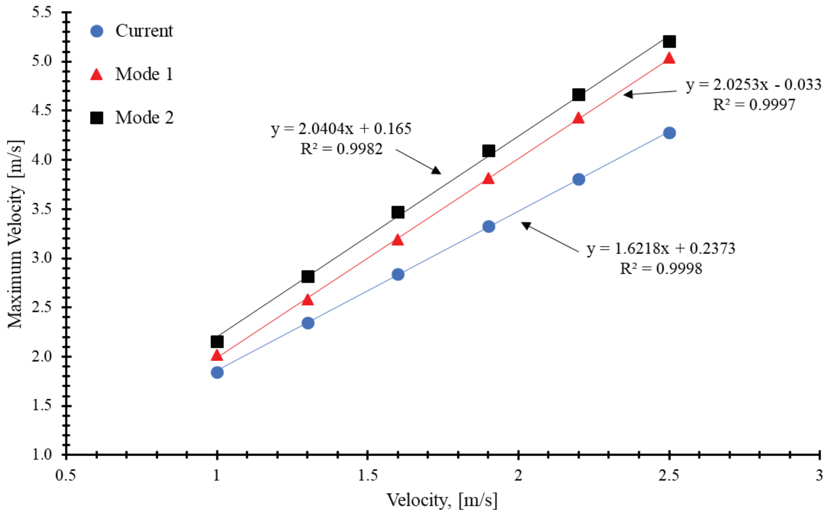

Figure 3 Effect of air velocity at the inlet on the percentage of air distribution

Since the focus of the study is to investigate the effect of different louvres arrangement on the percentage of air velocity distribution, a visual comparison must be conducted to conclude the hypothesis. In this section, the percentage of air velocity distribution is measured based on the results of simulation analysis which is from velocity contour. The area covered by the airstream is measured using the grid method to calculate an approximation of the percentage of air distribution over the plate of the heat exchanger. According to Figure 4, in Mode 1 and Mode 2, the velocity contour shows that air velocity is distributed uniformly after the airstream leaves the louvres. Thus, the percentage of air distribution inside the heat exchanger in Mode 1 is approximately higher than Mode 2 by 2%.

The velocity of the airstream is uniform across the plate of the heat exchanger, with little vorticity occurring at the edges of the hexagonal plate. The continuous airstream flows according to the designated directions, to cover most of the plate area. The vorticity occurs due to the turbulence flow of airstream after leaving the louvres. The vorticity is zero in the middle area of the plate but fully concentrate on the edges of the hexagonal plate. The edges of the hexagonal plate, blue contour, is considered as dead area since the airstreams caused vortex flow through them, causing no heat is transferred in these areas. In Mode 1, most of the area is covered with a green colour and the inconsistent yellow colour which indicates the airstream velocity is average across the plate. Whereas, in Mode 2, half of the area is covered with green and blue colour with the inconsistent yellow colour which indicates the airstream velocity is range from average to minimum value. Therefore, in this case, the smaller the louvres orifice, the higher the percentage of air velocity distribution over the plate of the heat exchanger.

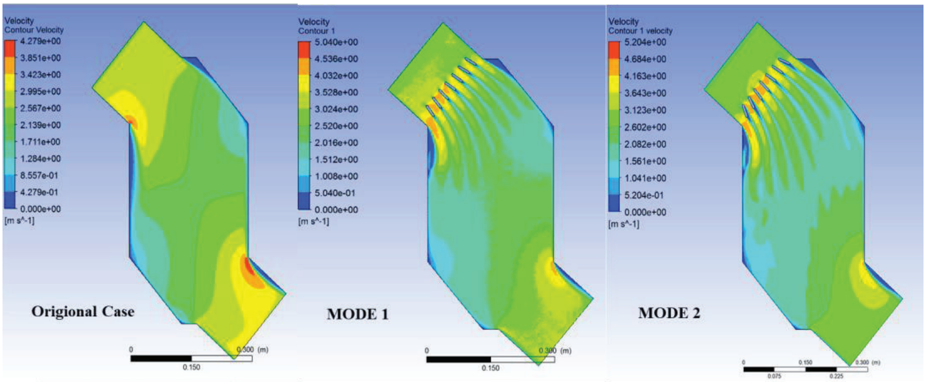

Figure 4 Comparison of velocity distribution across the plate

3.2 Effect of Air Velocity Inlet on the Pressure Losses Across Plate

The increase in air velocity at the inlet may increase the pressure losses across the plate. Initially, the pressure is maximum at the inlet duct. However, after airstreams pass through louvres arrangement, the pressure is loss across the plate, causing the pressure is minimum at the outlet duct. Pressure drop occurs due to turbulence flow generated at the louvres arrangement when airstream changes its directions drastically. Pressure drop also occurs due to the air hitting the louvres body, causing the air to lose its momentum and reduce its velocity. The pressure drop across the hexagonal plate of the air-to-air heat exchanger was measured employing the total pressure differences at the inlet duct and the outlet duct.

Figure 5 illustrates that the pressure drop increases linearly with the increase of air velocity at the inlet. At the higher value of air velocity at the inlet duct, the effect on the pressure drop becomes much stronger. According to Figure 5, with the increasing air velocity at the inlet, the pressure drop is significantly higher in Mode 1 compared to the pressure drop in Mode 2. Pressure loss in Mode 1 is higher than Mode 2 by 25.44% (8.770 Pa). By comparing with the current model, the pressure loss in Mode 1 is higher than pressure loss in current model by 34.86% (12.020 Pa). The pressure loss in Mode 2 is higher than pressure loss in current model by 12.64% (3.250 Pa).

Mode 1 has higher pressure losses due to the small louvre orifice compared to Mode 2. In this case, Mode 2 perform better in minimising pressure loss across the plate when the louvres method is employed. The wider the louvres orifice, the lower the pressure drop, the better the performance of heat exchanger. Positive results of pressure drop increase the turbulence intensity over the plate of heat exchanger, which will benefit the heat exchanger as turbulence flows create heat energy. However, too much in pressure drop may cause the power consumption to operate the system unnecessarily increases.

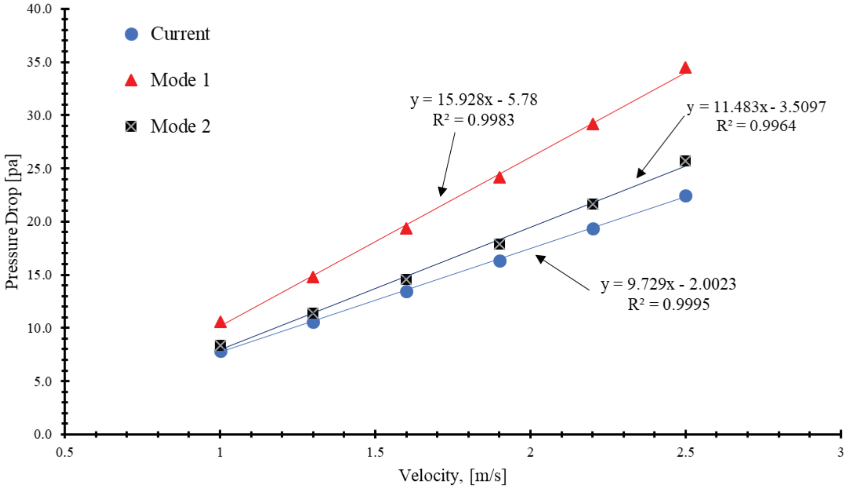

Figure 5 Effect of air velocity at the inlet on the pressure losses across plate

According to Figure 6, pressure contour is uniformly decreasing starting from the inlet duct across the hexagonal plate to the outlet duct. It is observed that Mode 1 and Mode 2 have significant pressure losses at the louvres arrangement when compared to the current model. The maximum pressure is concentrating at the inlet duct before the louvres arrangement section. After the louvres arrangement section, the pressure drops significantly across the plate of the heat exchanger. The air velocity enters the plate in a single direction, and with full velocity. Therefore, when airstreams hitting the louvre bodies, the effect of momentum is stronger that causes the pressure to accumulate in the inlet area as the airstreams are disturbed by the existence of louvres. In mode 1, the pressure drop is significant where minimum pressure is concentrating at the outlet duct, while in Mode 2, the minimum pressure only occurs at the edge of the hexagonal plate. The wider the louvres orifice, the lower the pressure drop across the plate of the heat exchanger.

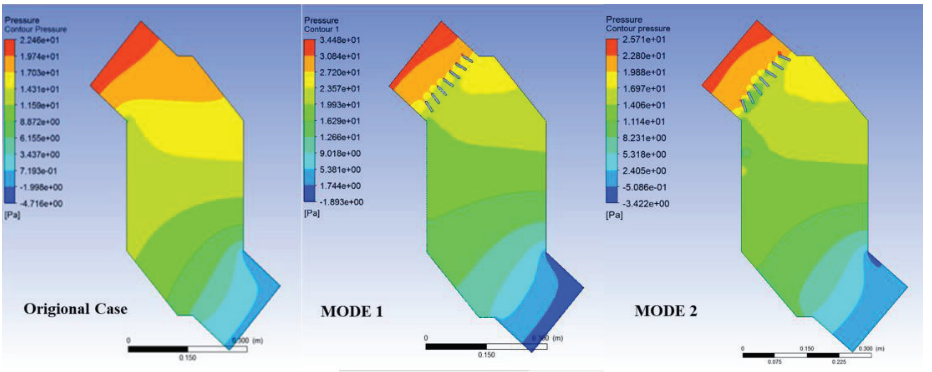

Figure 6 Comparison of pressure losses across the plate

3.3 Effect of Air Velocity Inlet on the Turbulence Kinetic Energy

The increase in air velocity at the inlet may increase the turbulence kinetic energy (TKE). Figure 7 shows the relationship between air velocity at the inlet and TKE distribution. Turbulence kinetic energy shows that the flow is having disturbance near the louvres arrangement. In order to model the fluid turbulence, TKE must be calculated using the Navier-Stokes equation as it is one of the fundamental fluid properties. Figure 7 illustrates that TKE magnitude increases linearly with the air velocity at the inlet duct. The higher the air velocity at the inlet, the stronger the effect on the TKE magnitude and distribution. According to Figure 7, with the increasing air velocity at the inlet, the TKE magnitude is significantly higher in Mode 1 compared to Mode 2. The TKE in Mode 1 is higher than Mode 2 by 40.70% (1.005 kJ/kg). The TKE difference between Mode 1 and the current model is 73.65% (1.818 kJ/kg) while the TKE difference between Mode 2 and the current mode is (0.813 kJ/kg). TKE increases with the existence of louvres as louvres are considered to be the disturbance for the flow. The TKE only occurs in the region where the velocity is maximum, refer to Figure 4. The angle that directed the louvre is the contributing factor for TKE. Since the airstream flows in a straight line with full speed, when air stream hitting the louvres, the airstream is then distributed to the designated direction. The magnitude for the momentum depends on the magnitude of the disturbance for the airstream. The wider the orifice, means that the higher the disturbance of the flow. In this case, Mode 2 has wider orifice compared to Mode 1 thus disturbance in Mode 2 is higher than in Mode 1. To conclude this index, at louvres arrangement where velocity is maximum, the turbulence kinetic energy is higher in the smaller louvre orifice due to the minimum flow disturbance generated, and the momentum is stronger resulting in better flow of airstream. Greater TKE magnitude illustrates that the flow went well throughout the distribution process.

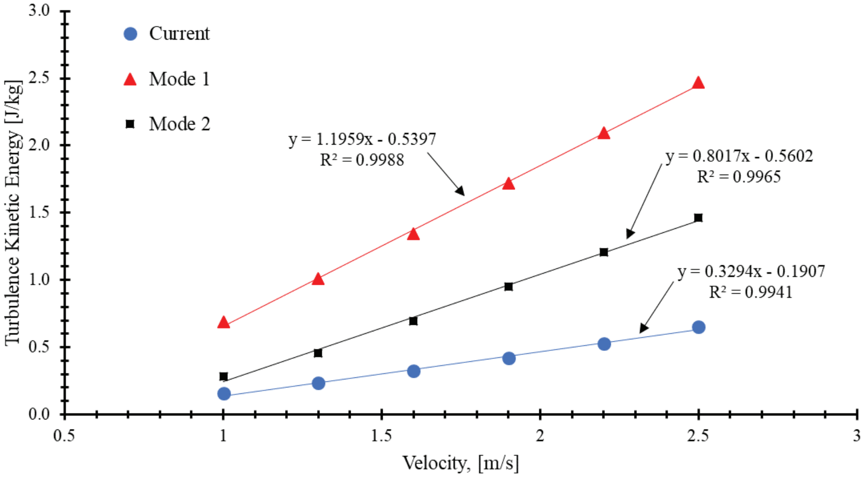

Figure 7 Effect of air velocity at the inlet on Turbulence Kinetic Energy (TKE)

According to Figure 8, in Mode 1 and Mode 2, the TKE contour shows that the turbulence kinetic energy is uniformly distributed and concentrate only at the louvres while in the current model, the turbulence is scattered across the plate of the heat exchanger. Based on observation, the distribution of air turbulence kinetic energy varies between two modes. Turbulence kinetic energy distribution means that the flow is having some disturbance, where momentum between airstream and bodies is significant. The light blue contour indicates the wake region after the flow is disturbed by the louvres arrangement. The wake region in Mode 1 is smaller than the wake region in Mode 2 illustrates that the flow of airstream in Mode 1 is better than the flow of airstream in Mode 2. Comparing the magnitude for turbulence kinetic energy for both modes, Mode 1 perform best as the TKE magnitude is high in the wake region, which reflects that the flow went well. Therefore, the smaller the louvres orifice, the flow of airstream is well distributed over the plate surface of the heat exchanger.

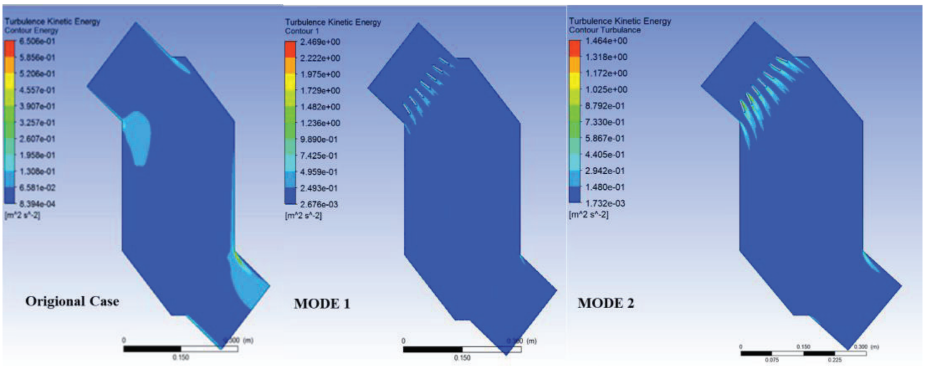

Figure 8 Comparison Of Turbulence Kinetic Energy Distribution

4.0 CONCLUSION

In this investigation, the louvres arrangement effect on maximising the air distribution over the plate of the heat exchanger to enhance the efficiency of IEC was experimentally conducted using ANSYS Fluent 19.2. Data from the analysis were collected. The proposed unique louvres arrangement shows a significant improvement in the HRV system performance. The main results were summarised as the following:

By analytical results, the increment of flow rate in Mode 2 is higher than Mode 1 by 3.15%. The increment of flow rate in Mode 2 is higher than the current model by 17.74%. The increment of flow rate in Mode 1 is higher than the current model by 15.1%. The flow rate in Mode 2 is higher than in Mode 1 by 2.64%. Then, the percentage of air velocity distribution over the plate of the heat exchanger is determined by the velocity contour comparison. Mode 1 achieved a higher percentage of air velocity distribution with the covered area is 97% while the Mode 2 only covered 95% of the plate area. Therefore, the maximum percentage of air velocity distribution across the plate of the heat exchanger is achieved by Mode 1 with average velocity magnitude across the plate while velocity magnitude in Mode 2 is close to a minimum value.

By numerical comparison, the pressure loss in Mode 1 is higher than the current mode by 34.86% (12.020 Pa). The pressure loss in Mode 2 is higher than pressure loss in current model by 12.64% (3.250 Pa). Thus the pressure losses in Mode 1 is higher than the pressure losses in Mode 2 by 25.44% (8.770 Pa). It is observed that Mode 1 and Mode 2 have significant pressure losses pass the louvres arrangement when compared to the current model. Hence, Mode 2 perform better with minimum pressure loss compare to Mode 1.

Based on the results, the TKE difference between Mode 2 and the current model is 55.56% (0.813 kJ/kg) whereas the TKE difference between Mode 1 and the current model is 73.65% (1.818 kJ/kg). The TKE in Mode 1 is higher than Mode 2 by 68.65% (1.005 kJ/kg). Based on observation, the wake region in Mode 1 is smaller than the wake region in Mode 2 which means that the flow of airstream in Mode 1 is better than the flow of airstream in Mode 2. Mode 1 perform best as the TKE mgnitude is high in the wake region, which reflects that the flow went well. Therefore, the smaller the louvres orifice, the flow of airstream is well distributed across the plate of the heat exchanger.

In conclusion, Mode 1 is selected to be an effective louvres arrangement inside the hexagonal plate of the heat exchanger to enhance the performance of HRV system, mainly by providing the highest percentage of air velocity distribution across the plate. The more area is covered by the airstream, and more heat is transferred, thus increasing the heat transfer rate, and increase the efficiency of the heat exchanger. Improvement in percentage or air distribution over the plate and TKE distribution will complement the high-pressure drop across the plate in Mode 1.

Since louvres arrangement has improved the primary evaluation indices of HRV heat exchanger especially the percentage of air distribution over the hexagonal plate, the experimental investigation will be conducted for future work to investigate the effect of different louvres arrangement on the sensible efficiency and the Coefficient of Performance (COP) of the whole systems.

ACKNOWLEDGEMENTS

The authors are grateful to University of Technology Sydney,Australia for the technical support provided for the research work.

REFERENCES

Alkhedhair, A. M. S. (2015). Modelling and experimental study of spray cooling systems for inlet air pre-cooling in natural draft dry cooling towers.

Al-Waked, R., Nasif, M. S., Morrison, G., & Behnia, M. (2013). CFD simulation of air to air enthalpy heat exchanger. *Energy Conversion and Management, 74*, 377-385.

Al-Zubaydi, A. Y. T., & Hong, G. (2018). Experimental investigation of counter flow heat exchangers for energy recovery ventilation in cooling mode. *International Journal of Refrigeration, 93*, 132-143.

Al-Zubaydi, A. Y. T., & Hong, G. (2019). Experimental study of a novel water-spraying configuration in indirect evaporative cooling. *Applied Thermal Engineering*,

Al-Zubaydi, A. Y. T., Hong, G., & Dartnall, W. J. (2016). *CFD modelling and analysis of different designs plate heat exchangers*. Australasian Heat and Mass Transfer Conference.

Chen, Y., Yang, H., & Luo, Y. (2016). Experimental study of plate type air cooler performances under four operating modes. *Building and Environment, 104*, 296-310.

Dvořák, V., & Vít, T. (2015a). Numerical investigation of counter flow plate heat exchanger. *Energy Procedia, 83*, 341-349.

Fernández-Seara, J., Diz, R., Uhía, F. J., Dopazo, A., & Ferro, J. M. (2011). Experimental analysis of an air-to-air heat recovery unit for balanced ventilation systems in residential buildings. *Energy Conversion and Management, 52*(1), 635-640.

Greifzu, F., Kratzsch, C., Forgber, T., Lindner, F., & Schwarze, R. (2016). Assessment of particle-tracking models for dispersed particle-laden flows implemented in OpenFOAM and ANSYS FLUENT. *Engineering Applications of Computational Fluid Mechanics, 10*(1), 30-43.

Launder, B. E., & Spalding, D. B. (1972). *Mathematical models of turbulence*, Academic Press.

Montazeri, H., Blocken, B., & Hensen, J. (2015). Evaporative cooling by water spray systems: CFD simulation, experimental validation and sensitivity analysis. *Building and Environment, 83*, 129-141.

Pakari, A., & Ghani, S. (2019). Comparison of 1D and 3D heat and mass transfer models of a counter flow dew point evaporative cooling system: Numerical and experimental study. *International Journal of Refrigeration, 99*, 114-125.

Saraireh, M. (2012). *Heat Transfer and Condensation of Water Vapour from Humid Air in Compact Heat Exchangers*, PhD Thesis, Victoria University, Australia.

Sun, Y. (2019). Numerical and experimental study on spray cooling system design for cooling performance enhancement of natural draft dry cooling towers, PhD Thesis, University of Quensland, Australia.

Sun, Y., Guan, Z., Gurgenci, H., Li, X., & Hooman, K. (2017). A study on multi-nozzle arrangement for spray cooling system in natural draft dry cooling tower. *Applied Thermal Engineering, 124*, 795-814.

Vera, M., & Quintero, A. E. (2015). Wall conduction effects in laminar counterflow parallel-plate heat exchangers with small-scale wall corrugations. *Heat transfer studies and applications*. London: InTech Open Limited.

Yaïci, W., Ghorab, M., & Entchev, E. (2013). Numerical analysis of heat and energy recovery ventilators performance based on CFD for detailed design. *Applied Thermal Engineering, 51*(1-2), 770-780.

Zhang, L. (2010). Heat and mass transfer in a quasi-counter flow membrane-based total heat exchanger. *International Journal of Heat and Mass Transfer, 53*(23-24), 5478-5486.

Zou, Y., Zhao, X., & Chen, Q. (2018). *Comparison of STAR-CCM and ANSYS fluent for simulating indoor airflows. Building Simulation, 11*(1) 165-174.