SEMARAK ILMU
PUBLISHING
202103268166(003316878-P)

CFD Letters

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

CFD Air Flow Evaluation of Finned Tube Evaporator for Refrigerated Display Cabinet Application

I Dewa Made Cipta Santosa[1,*], I Gede Nyoman Suta Waisnawa[1], Putu Wijaya Sunu[1], I Wayan Temaja[1], Liang Li[2]

[1] Department of Mechanical Engineering, Politeknik Negeri Bali, Jalan Kampus Bukit Jimbaran, Kuta Selatan, Badung, Bali, 80363, Indonesia
[2] College of Engineering, Design and Physical Science, Brunel University London, Uxbridge, UB8 3 PH, United Kingdom

ARTICLE INFO

Article history:
Received 5 May 2023
Received in revised form 8 June 2023
Accepted 9 July 2023
Available online 30 April 2024

Keywords:
Air side heat transfer coefficient; air flow characteristic; finned tube evaporator; numerical modelling

ABSTRACT

This study is aimed to develop a simulation to improve the performance of the finned tube evaporator which is applied to the refrigerated display cabinet. CFD model was developed to be able to analyse the characteristics of air flow inside the fin gap and air side heat transfer coefficient. Geometry of the model of overall finned tube evaporator is considered covering two aluminium wavy fins with an air flow in between, combination of staggered cooper tubes with refrigerant flow inside. Fin gap is designed 4 mm to anticipate frost on the fin surface that can block air flow. Turbulence models used in the study is the realizable k-ε turbulence which had the best performance turbulence model and it was validated with secondary data from previous studies and shows the lowest error only 5.9 %. The use of CFD was found to be sufficiently representative of the heat transfer characteristics of evaporators, and acted as an effective simulation tool to determine the heat transfer coefficient in order to improve efficiency in terms of improved design. The characteristics of air flow between the fin gap and around the tube was obtained various and complex. In the case study the entry velocity of 1.7 m /s at the highest turbulence condition of the first row can reach speeds of 2.75 m/s. Hight turbulence regime in flow can indicate higher the heat transfer coefficient of the evaporator.

1. Introduction

Refrigeration system has developed both in term of component development and in term of being environmentally friendly, since the refrigeration system contribute to increased CO_2 levels in atmosphere and can also damage the ozone which protects earth living from dangerous radiation of the sun shine [1, 2]. However, natural refrigerant such as carbon dioxide (CO_2-R744) is certainly excellent for the environment friendly but at present it still has several weaknesses, especially in term of high-pressure safety and efficiency of the system [3], so that investment and operation are still more expensive being compared with the conventional refrigerant such as, R134a, R600a. Thus, improving the efficiency of each component becomes more attention to improve the efficiency of

---

[*] *Corresponding author.*
E-mail address: idmcsantosa@pnb.ac.id (I Dewa Made Cipta Santosa)

https://doi.org/10.37934/cfdl.16.9.5263

overall refrigeration system. Using exergy analysis, four main components in vapor compression refrigeration system were investigated by Choi *et al.,* [4], the components that consist of condenser, evaporator, compressor and expansion valve. Result of analysis revealed that there were losses of 30%, 25%, 25%, and 20%, respectively. It means that the condenser and evaporator are urgently needed for efficiency improvement in order to obtain higher coefficient of performance (COP) of the overall refrigeration system. And in this study evaporator become concerned to be analysis since this equipment more risk from frost that the air flow quality significant useful to be investigated to get good effectiveness for the evaporator [5].

Computer simulations, especially Computational Fluid Dynamic (CFD) have been widely applied to several heat exchanger type simulations. Where, simulation can analyze flow and temperature in depth instead of expensive and complex measurement of experimental methods. Using ANSYS Fluent to simulate a finned tube type heat exchanger have been considered and analyzed properly and the characteristics of this type of heat exchanger are very compatible with the Refrigeration, Heating and Air Conditioning (RHVAC) system [6, 7]. Proper simulation is very important to evaluate thermal performance under various operating conditions and especially, for optimal design. Computational Fluid Dynamics (CFD) is also very appropriate for analyzing fluid flow design, static and thermal analysis, to obtain design optimizations that can be applied to specific needs. In addition, with valid CFD modeling, analysis becomes more flexible and enable us to obtain a very specific data. With precision simulation, the optimization of heat exchanger design can be done in depth [8, 9].

Besides thermal approach analysis, the CFD model is also designed to allow evaluation of the 3D effects of mal-distribution on variable temperature, heat transfer coefficient, and cooling power. From the air side study, the CFD model is appropriate for observing the effects of mal-distribution of flow on a finned tube heat exchanger applied for an open display cabinet. The model shows the significances of uneven air flow at the evaporator inlet, and can affect to the air temperature and heat flux in the evaporator. Poor air distribution causes a 10% difference in terms of thermal load between the right and left side of the evaporator [10]. In addition, other open display cabinet studies which were investigated by 2D and 3D CFD model showed very good results in air flow analysis. The ambient conditions (temperature and humidity) as well as air flow through heat exchanger have contribution on the overall performance and energy efficiency of display cabinets [11-13]. The type of flow in the pipe also significantly affects the relative thermal resistance of the refrigerant. The result obtained is the importance of considering the homogeneousness of angular heat transfer coefficient, which if uniformed will affect the results of the precision of the model. In pipe flow regime investigation with two phase condition also can be using other interesting computational simulation program, such as MATLAB [14]. Furthermore, in ventilation application, optimization can be done by optimizing the degree of inclination of the airflow output so that cooling can be effectively reached into the room and energy can be saved significantly. The CFD simulation is very suitable to analyze accurately for the best ventilation design to get optimization of pressure drop and distribution of flow into the room [15, 16].

Wang *et al.,* [17] examined the finned tube evaporator by using thermal-hydraulic characteristics approach. This characteristic is analyzed using the CFD method. The 3D numerical model is derived from the finned tube evaporator technique diagram, and the boundary condition is determined according to the measured data obtained by experimental test rig. High and low temperatures alternately seem in the middle plane of the adjacent tube line. The shape of the front end and the back end that connects the main body to the exhaust pipe is an important factor in the flow area. In term of cooling evaporator and condenser, the application of finned tube heat exchangers is the formation of ice layers on the surface of fins that can reduce performance. This happens because there are obstacles until the air flow is blocked in the fin and tube gaps. With computer modeling simulations, thermal resistance can be evaluated and shown that the occurrence of frost on the fin surface is the main cause of the decrease in cooling capacity. So, it is recommended to improve the evaporator design by reducing fin density in the first line pipe. In addition, some simulation model can predict accurately previous were performed by using Reynolds-averaged Navier-Stokes (RANS) equations, steady-state condition and in conjunction with Re-Normalization Group (RNG) k-ε as the turbulence model [18, 19].

In summary, finned tube heat exchanger is very suitable for evaporator of the refrigerator system. In addition, the CFD simulation is very precise and accurate for a finned tube heat exchanger simulation. So, the results of this study will provide a significant contribution to the development of evaporator in order to increase the Coefficient of Performance (COP) of fridge temperature refrigeration systems. The heat transfer coefficient of finned tube heat exchanger for the evaporator will be simulated, especially in working conditions of moderate temperature (0^{o}C) with air humidity of 90% RH. As far as the authors are concerned, this special condition has never been done in previous studies. Although at this early stage of research is aimed to simulations were used to analyze the characteristics of the air flow between the fins and the tube, as well as to analyze the heat transfer coefficient. For the next stage, it will be continued with the optimization of the fin gap and wavy angle in order to reduce the impact of frost on the evaporator performance.

2. Methodology

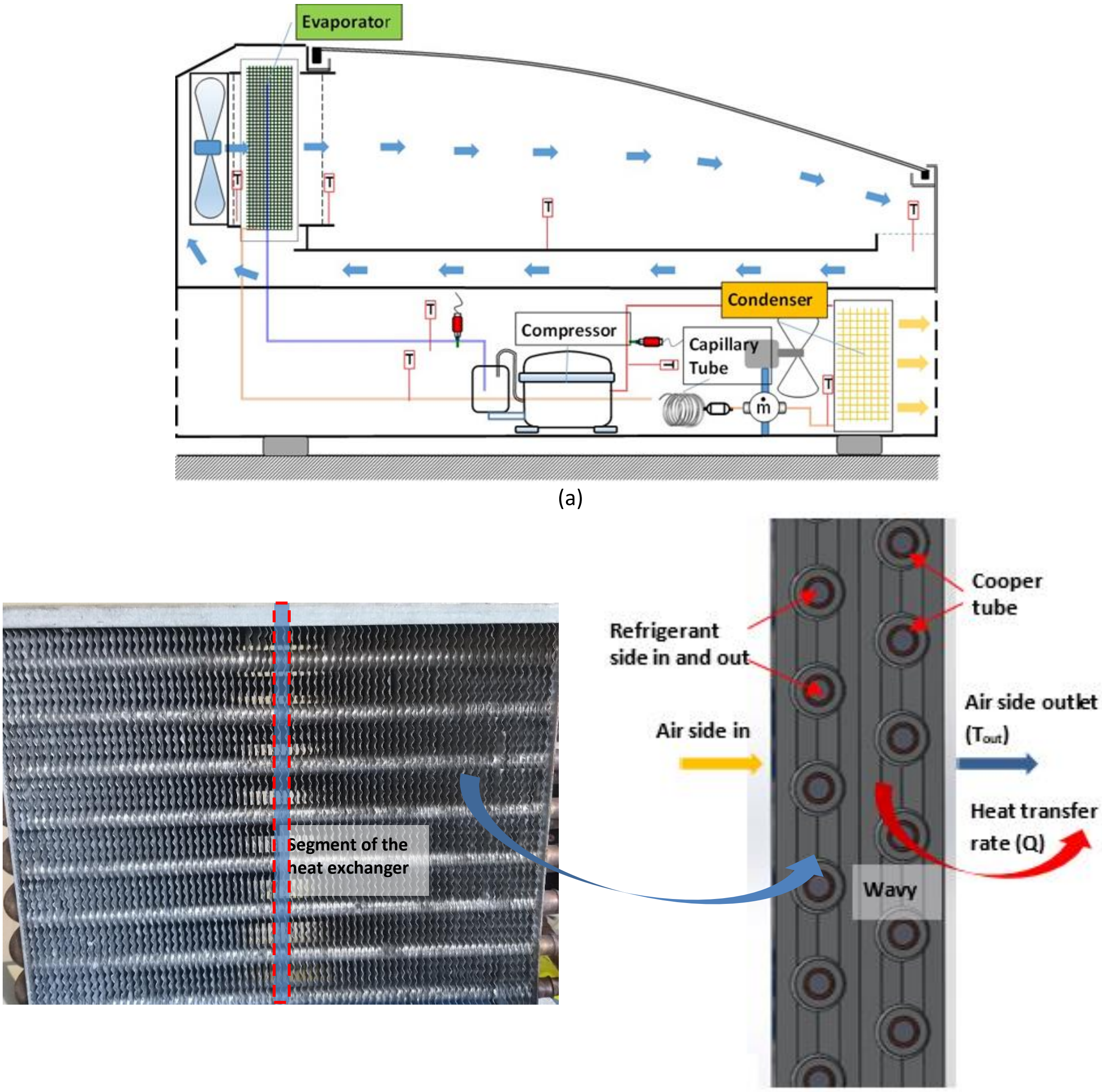
2.1 Geometry and Boundary Condition

Figure 1 shows the schematic diagram of refrigerated display cabinet and also show the evaporator to be modeled with the CFD. Since the evaporator to be modeled has a specification described in Table 1, so it is too difficult to model whole geometry of the evaporator. It will not only need very high computer but also long-time running iteration. To obtain an effective processing time and sufficient computer capacity for the CFD model, geometry is arranged with two fins as shown in Figure 1 (c). The geometry of the simulation is considered to consist of two fin and others exactly similar with the real evaporator which including, 2 rows pipes arrangement with fin type is wavy and staggered pipe position with longitudinal distance of 25.4 mm and axial distance of 22 mm. Fin is made of aluminum, with a thickness of 0.4 mm fin and 4 mm air gap between the fins. The pipe is made of 8 mm outer diameter copper, with a thickness of 1.68 mm. In this case the air as a working fluid is modeled with 90% RH humidity and room temperature, while the properties are derived using the Engineering Equation Solver program [20] and input in the fluid properties in the CFD program. For working fluid refrigerant which flow in the tubes is modeled with R134a refrigerant with conditions at medium temperature at 0^{o}C.

Table 1
Specification of the finned and tube evaporator

Fined tube evaporator	Unit	Characteristics/number
Tube arrangement (material)		Staggered (Cu)
Longitudinal tube spacing	(mm)	25.4
Axial tube spacing	(mm)	22
Inside tube diameter	(mm)	8
Outside tube diameter	(mm)	10
Inside tube surface		Smooth
Fin spacing	(inchi)	4 mm
Fin thickness	(mm)	0.6
Fin geometry (material)		Aluminium
Tube length	(mm)	240

Since the fins are only 0.6 mm thick, joining on such a thin surface can be error especially when considered in the context of the dimensional ratio to the overall evaporator. This is also due to the type and size of the mesh. In order to more conveniently represent heat conduction in fins, and also recommended by ANSYS Fluent® for finned heat exchanger models, the thin conducting wall shell concept available in ANSYS Fluent® is used. This refers to simplifying the discretization of heat transfer materials to a single node in thickness, thereby avoiding splicing to a very small degree.



(a)

(b) (c)

Fig. 1. Evaporator of display cabinet modelled to the CFD (a) Display cabinet schematic diagram (b) Overall finned tube evaporator (c) CFD Simplified segment geometry

The thermos-physical properties (density, viscosity, heat capacity, thermal conductivity) of air and working fluid (R134a) as a function of temperature and pressure were obtained using the Equation Engineering Solver software. These are combined in Fluent using a linear-piece formulation. The thermo-physical properties of copper and aluminum are obtained from the Fluent database. The mass flow rate and temperature of the working fluid (refrigerant) at the inlet for each tube is assumed to be constant for the tube segments since the real evaporator temperature variation is linier with very small variation between inlet and outlet temperature. Air enters between the two fins (x-direction), with constant velocity and temperature. Fins and collar fins are modelled as thin walls are available in the Fluent. And the equation for the starting boundary was assumed steady state and constant at temperature of air 28^{o}C and RH 90%.

2.2 Meshing and CFD Governing Equation

The type and quality of the mesh plays a very important role in order to get high accuracy of numerical calculations and mesh quality is also very important to match and validate with the geometry [21]. Furthermore, the proper turbulence model should be determined after selecting the type and quality of the mesh. The discontinuous meshing approach is particularly useful in the early design stages of heat exchangers to save time and memory at the minimum disadvantage of accuracy. In order to achieve optimal mesh elements, a grid independence test was performed on the volume tetrahedral mesh element geometry. Therefore, it is recommended to explore the use of the discontinuous connection approach for the turbulent flow encountered in many heat exchanger applications [22, 23]. Meshing in this model uses the tetrahedral type with a simulation of three different cell density numbers. Meshing sensitivity analysis is carried out with respect to model convergence. Using a medium (3,200,000 cells), the convergence of residuals is at least 10^{-4} for continuity, 10^{-7} for energy, 10^{-3} for x, y and z, 10^{-3} for k and 10^{-2} for ε, while fine grids are found to have residues of 10^{-5}, 10^{-8}, 10^{-6}, 10^{-4} and 10^{-4}. A very satisfying residue is obtained from a fine grid. However, this smoother grid also requires longer computing time. Final meshing is shown in Figure 2, where a high grid density is used in all areas where the temperature gradient is high, which occurs at the fin collar and the adjacent closest to the nearest heat source.

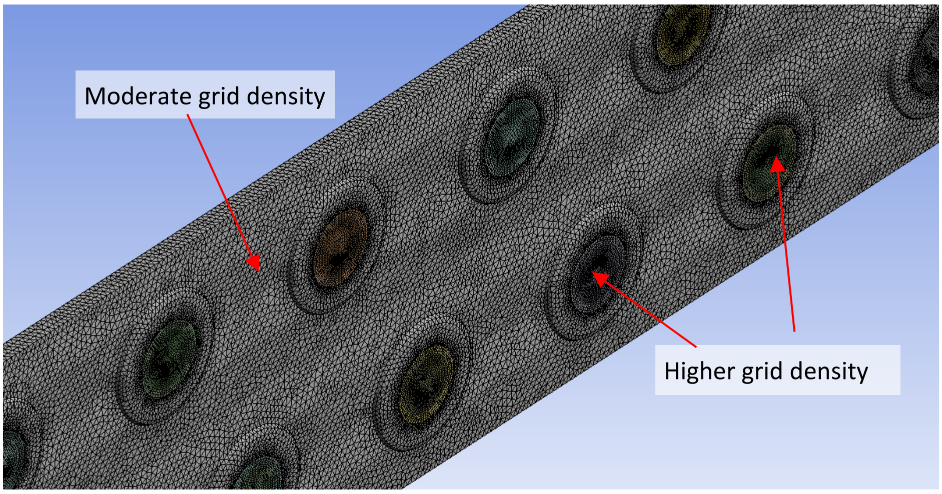


Fig. 2. Three dimension (3D) tetrahedral meshing geometry

This fundamental physical principle is conveyed as a set of Navier-Stokes Equations Eq. (1) - Eq. (5). The equation governing the flow and heat transfer associated with the fluid is based on the principle of conservation of mass, momentum and energy, and because they are non-linear Second Order Equations (SOE), the solution procedure is complicated. CFD concerns and solves the discretized form of this equation for a domain, through iteration, where pressure (p), temperature (T), density (ρ) and velocity components (*u, v, w*) on each grid cell can be predicted with high correctness. The three equations governing to be developed as follows:

Continuity equation:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=S_M \tag{1}$$

Momentum equation:

$$\text{X-Momentum}: \frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho\, u^2)}{\partial x}+\frac{\partial(\rho\, uv)}{\partial y}+\frac{\partial(\rho\, uw)}{\partial z}=-\frac{\partial(P)}{\partial x}+\frac{\partial}{\partial x}(\bar{\tau})+\rho g_i+F_i \tag{2}$$

$$\text{Y-Momentum}: \frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho\, uv)}{\partial x}+\frac{\partial(\rho\, v^2)}{\partial y}+\frac{\partial(\rho\, vw)}{\partial z}=-\frac{\partial(P)}{\partial y}+\frac{\partial}{\partial y}(\bar{\tau})+\rho g_j+F_j \tag{3}$$

$$\text{Z-Momentum}: \frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho\, uw)}{\partial x}+\frac{\partial(\rho\, vw)}{\partial y}+\frac{\partial(\rho\, w^2)}{\partial z}=-\frac{\partial(P)}{\partial z}+\frac{\partial}{\partial z}(\bar{\tau})+\rho g_k+F_k \tag{4}$$

Energy equation:

$$\frac{\partial}{\partial t}(\rho H)=-\frac{\partial}{\partial x}(\rho u c_p T)-\frac{\partial}{\partial y}(\rho v c_p T)-\frac{\partial}{\partial z}(\rho w c_p T)+\frac{\partial}{\partial x}\left[\lambda\frac{\partial T}{\partial x}\right]+\frac{\partial}{\partial y}\left[\lambda\frac{\partial T}{\partial y}\right]+\frac{\partial}{\partial z}\left[\lambda\frac{\partial T}{\partial z}\right]+S_E \tag{5}$$

In these equations, SM dan SE represent mass source term (kg/m^3) and energy source term (W/m^2), respectively in the system. In accordance with the designed geometry, it allows air flow in the fin gap in complex flows with dual fluid flow regimes consisting of high or low turbulence regimes, especially depending on the surface topology of the heat exchanger. A high Reynolds number of current gives a higher heat transfer rate, compared to a lower Reynolds flow, and therefore the model must capture the turbulence aspect of the flow. Higher turbulent flow occurs in the fin, the effectiveness and heat transfer coefficient of the heat exchanger is better.

3. Result and Discussion

3.1 CFD Simulation Validation

The results and discussion explain about the validation of the model developed by the CFD method to show the effectiveness of the CFD model with this method to the experimental results that have been previously studied. And the core of the discussion in this study is the investigation of the characteristics of air flow between the gap and the tube, as a basis for further investigation for the application of the evaporator. The validation indicators are the heat transfer rate (Q) error (%) dan air outlet temperature error (°C) which is shown in Figure 3.

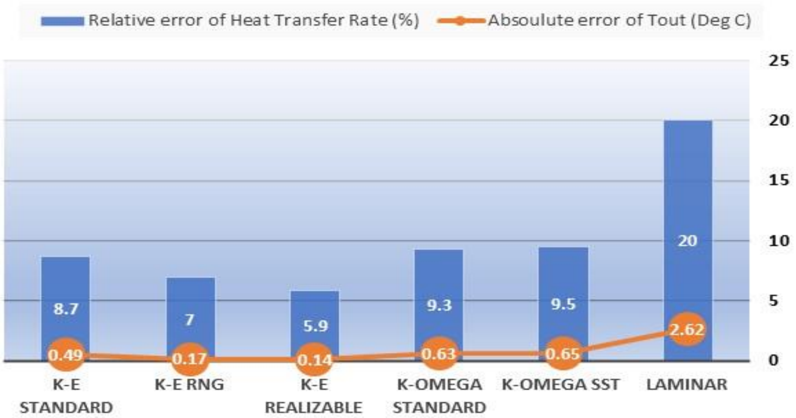


Fig. 3. Error of turbulence model with experimental data derived from Santosa *et al.,* [24]

To determine the accuracy of the numerical model, turbulence model performance is determined in different experimental test conditions which was already described by Santosa *et al.,* [24]. Air-outlet temperature (T_{out}) and heat transfer rate (Q) are obtained from the average value of the segments for each experimental condition. Validation procedures include a comparison between predictions CFD parameters and results of secondary experimental data. Data collection conditions for finned tube heat exchangers have been carried out at an air flow rate of 1.7 m/s, 2.0 m/s, 2.4 m/s. Environment temperature was assumed to be constant at 28 ^oC. Temperature input to the refrigerant (R134a) boundary in each tube is constant at 0^oC according to the working fluid conditions of the evaporator at medium temperature. The heat transfer rate of the air side is calculated directly by CFD.

The models compared were five turbulence models consisting of k-ε standard, k-ε realizable, k-ε RNG, k-omega standard and k-omega SST. Laminar model error was also calculated to get wide range of comparison between turbulence and laminar model. The Realizable turbulence model obtained the best relative error results for Q (%) and the absolute error T_{out} (^oC). Meanwhile, the k-ω turbulent model showed a slightly worse performance for all than the k-ε turbulence one. While the Laminar model had an error of Q: 20%, T_{out}: 2.62. Therefore, it was found that the realizable k-ε turbulence had the best performance turbulence model and was automatically used in this study. This turbulence model consideration and error acceptable was very good agreement with previous study conducted by Morales and Loredo [25].

3.2 Heat Transfer Coefficient Evaluation

The heat transfer coefficient (h_c) of the evaporator model was evaluated using a comparative analysis with the experimental research conducted by Wen and Ho [26]. The heat exchanger specifications in their experimental study and the geometry of the CFD modeling are shown in Table 2. In general, the designs of the two heat exchangers are comparable, but specifically there are differences in fin spacing, fin thickness and working fluid. Comparable observations are in terms of fin type, number of row and overall construction.

As mentioned earlier, the heat transfer coefficient is very urgent to determine the overall heat exchanger performance. As opposed to previous studies, this study does not impose heat transfer correlations on the model; rather, it is calculated implicitly in the model. Therefore, this section describes the effectiveness of CFDs to determine the heat transfer coefficient. The air side heat transfer coefficient was adopted from calculations developed by Wen and Ho [26] which applied calculations on segments developed in this study in accordance with the design of the designed geometry. In each segment for tube / fin wall bundles were inferred from heat transfer rate (Q), heat-transfer surface area (A_t) = A_{colar} + A_{fin}, wall temperature (T_W) = (T_{colar} + T_{fin}) / 2 and bulk air temperature (T_{bulk}). The following equation, Eq. (4) and Eq. (5) are used for the calculation of heat transfer coefficient- h_c (W / m^{2}K) and heat transfer rate (W), respectively. The format of this equation was chosen primarily to comply with the formulations used in the literature.

Table 2
Comparison heat exchanger specification between CFD model and Wen and Ho experiment [26]

Specification	CFD model	Experiment of Wen and Ho [26]
Fin type	Wavy fin	Wavy fin
Number of row	2	2
Tube outer diameter	8 mm	10.30 mm
Inlet diameter	6.32 mm	10.10 mm
Fin spacing	4 mm	2.54 mm
Fin thickness	0.2 mm	0.12 mm
Number of pipe investigation in circuit	20	20
Working fluid	R143a	Water

$$h_c = \frac{\dot{Q}}{(A_t)(T_w - T_{bulk})} \tag{4}$$

$$\dot{Q} = \dot{m}_{air} \cdot \Delta h_{air} \tag{5}$$

The mean heat transfer coefficient in the segments was evaluated for each air inlet velocity investigated: 1 m / s, 1.3 m / s, 1.7 m / h, 2 m / s and 2.4 m / s. Figure 5 indicates that air velocity increases, the coefficient of heat transfer also increases.

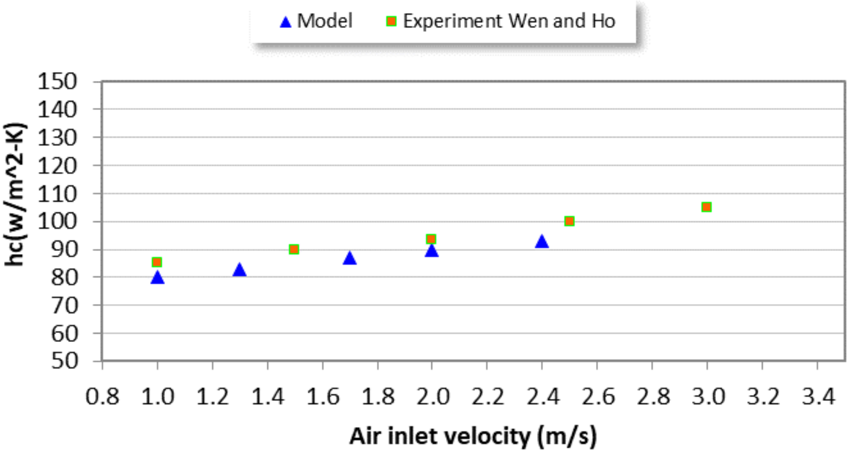


Fig. 4. Variation of heat transfer coefficient with air inlet velocity

As a result, there is a slight deviation between the model and the experimental results, this is most likely due to differences in specifications, especially because of fin geometry. However, according to the comparison of the value of the heat transfer coefficient (h_c) it can be assumed that

CFD is sufficient to calculate the heat transfer coefficient depend on the air side calculation. The model shows more linear correlation because it is more ideal than the actual results (experiment), in fact various real factors are assumed to be ideal in the modeling.

3.3 Air Flow Characteristic Investigation with Velocity Contour

Figure 5 shows the characteristics of air flow in between two fins with CFD velocity vector approach. The flow characteristics in the heat exchanger flow section are strongly influenced by the presence of tubes and fins. To explain this phenomenon, the flow characteristics in each row of the evaporator is also presented. Each row has a weak or stagnant formation on the behind of tube. The movement of air between fins and tubes is very complex which is certainly very difficult to observe with the experimental method. With the experiment method it can only be clearly observed in the average entrance speed and the average exit speed only. Turbulence speed increases significantly when turbulence is highest between the left and right sides of the pipe while the opposite condition, stagnant conditions occur at the behind of the tube, especially in the second row. This stagnant condition greatly reduces the heat transfer coefficient [27]. Thus, in accordance with the objectives of this study, the stages of further research will be simulated with different fin distances to determine the optimization of the heat transfer coefficient. In a narrow fin evaporator distance will increase the potential of frost blocking. In addition, a larger fin spacing contributes an advantage especially on pressure drop and geometry should be designed to ensure even flow distribution to get less sensitive to thermohydraulic deterioration. With larger fin spacing and fin length also allows longer operation time between defrost cycles [28]. Meanwhile, to maintain turbulence conditions, wavy angle at fin will be simulated to obtain the optimum angle to ensure evenly air distribution, especially from the first row to the second row.

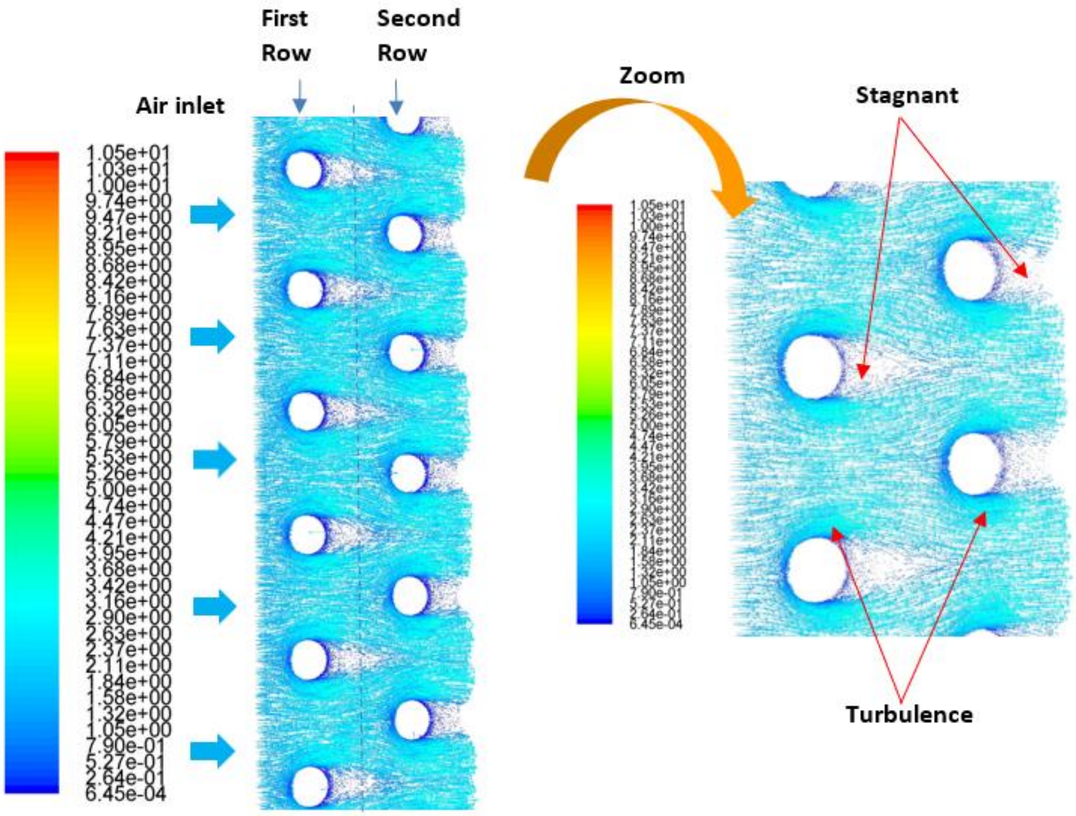


Fig. 5. Air flow characteristic with velocity vector

In addition, the data plot is described in Figure 6. The problem of heat transfer in the heat exchanger is closely related to the flow structure. From the CFD data plotted based on the evenly entering air velocity of 1.7 m/s, where turbulence occurs and stagnant flow also occurs in certain segments. From CFD data, the velocity varies from 0.5 to the maximum at an average speed of 2.75 m / s. it is indicated that the air flow inside the fin gap really complicated.

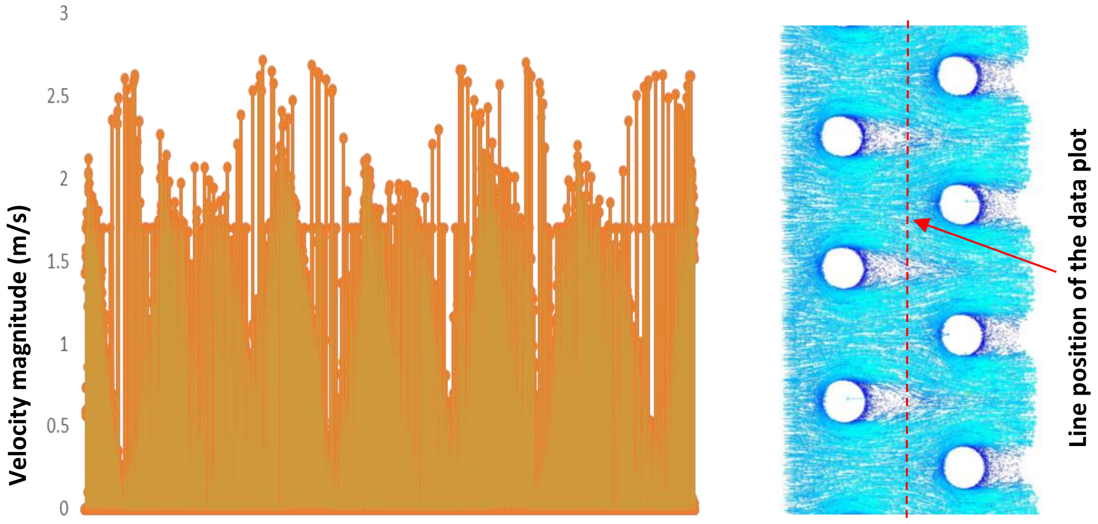


Fig. 6. Air side velocity magnitude

4. Conclusions

Development of the Computational Fluid Dynamics (CFD) model was used to evaluate and investigate the air flow characteristic in a finned tube heat exchanger for evaporator application. In addition, the heat transfer coefficient (h_c) can also be analyzed deeply and the CFD clearly showed the effect of the h_c with variation of air inlet velocity and also high turbulence occurrence in the fin gap. In this study, the finding can be highlight as follows.

i. The relative errors of heat transfer rates (Q) only 5,9% and absolute error of temperature outlet (T_{out}) is found at 0.14 ^oC much smaller that real K type thermocouple with 0.5 ^oC accuracy. Depend on error value indicates that the model can be categorized valid because the error is still in the range of the type K thermocouple error which used in the experimental measuring instrument and also get good agreement with the relevant previous studies.
ii. In term of the design of the CFD, model geometry is considered the evaporator segment which consists of two fins, all pipes, air flow, and refrigerant flow, obtained time effectiveness in computer iteration. The use of CFD was found to be sufficiently representative of the heat transfer characteristics of evaporators, and acted as an effective simulation tool to determine the heat transfer coefficient in order to improve efficiency in terms of improved design.
iii. The characteristics of air flow between fin gaps are very interesting to observe with flow characteristics because this is very difficult to study with the experimental method. Flow characteristics show the phenomenon of turbulence flow and stagnant contrast in the fin gap between the tubes. The difference between the first and second lines and stagnant flow conditions behind the tube can reduce overall heat exchanger performance. A wider gap will reduce turbulence and air contact to the fin.

iv. The mean heat transfer coefficient in the segments was evaluated for each air inlet velocity investigated: 1 m / s, 1.3 m / s, 1.7 m / h, 2 m / s and 2.4 m / s, indicates that air velocity increases, the coefficient of heat transfer also increases.
v. The flow characteristics in the segment could be an indicator, the optimum evaporator can be designed for the refrigerator application, especially for medium temperature or refrigerated display cabinet.

Acknowledgement

The author expresses gratitude/thanks to the Directorate Academic and Vocational Higher Education (DAPTV), Directorate General of Vocational Education, Ministry of Education, Culture Research and Technology-Indonesian Government, for the research funding in year 2023 with Letter of Acceptance No. 0463/D4/AL.04/2023. Thank is also forwarded to all members and research assistants in the Mechanical Engineering Department of Bali State Polytechnic and, finally to Research Centre (P3M) Bali State Polytechnic for the excellent administrative support. And we also deliver thank for the College of Engineering, Design and Physical Science, Brunel University London for the in- kind support of simulation computer program.

References

[1] Santosa, IDewa MC, and ING Suta Waisnawa. "Performance analysis of transcritical CO2 refrigeration system for supermarket application." *GEOMATE Journal* 15, no. 50 (2018): 70-75. https://doi.org/10.21660/2018.50.IJCST1

[2] Santosa, I. D. M. C., I. G. N. S. Waisnawa, P. W. Sunu, and I. W. Temaja. "Simulation of transcritical CO2 refrigeration system with booster hot gas bypass in tropical climate." In *Journal of Physics: Conference Series*, vol. 953, no. 1, p. 012044. IOP Publishing, 2018. https://doi.org/10.1088/1742-6596/953/1/012044

[3] Ahammed, Md Ezaz, Souvik Bhattacharyya, and M. Ramgopal. "Thermodynamic design and simulation of a CO2 based transcritical vapour compression refrigeration system with an ejector." *International journal of refrigeration* 45 (2014): 177-188. https://doi.org/10.1016/j.ijrefrig.2014.06.010

[4] Choi, Jong Min, Yonghan Kim, Mooyeon Lee, and Yongchan Kim. "Air side heat transfer coefficients of discrete plate finned-tube heat exchangers with large fin pitch." *Applied Thermal Engineering* 30, no. 2-3 (2010): 174-180. https://doi.org/10.1016/j.applthermaleng.2009.08.001

[5] Da Silva, Diogo L., Christian JL Hermes, and Claudio Melo. "First-principles modeling of frost accumulation on fan-supplied tube-fin evaporators." *Applied Thermal Engineering* 31, no. 14-15 (2011): 2616-2621. https://doi.org/10.1016/j.applthermaleng.2011.04.029

[6] Yaïci, Wahiba, Mohamed Ghorab, and Evgueniy Entchev. "3D CFD study of the effect of inlet air flow maldistribution on plate-fin-tube heat exchanger design and thermal–hydraulic performance." *International Journal of Heat and Mass Transfer* 101 (2016): 527-541. https://doi.org/10.1016/j.ijheatmasstransfer.2016.05.063

[7] Banu, PS Arshi, DNS Ramesh Lohith, M. Praveen Kalyan, Dilip Sai Vempati, and B. Hemanth Sai. "Simulation of fin and tube heat exchanger and validation with CFD analysis." *Materials Today: Proceedings* 66 (2022): 1471-1476. https://doi.org/10.1016/j.matpr.2022.05.552

[8] Zhang, Guohui, Baolong Wang, Xianting Li, Wenxing Shi, and Yang Cao. "Review of experimentation and modeling of heat and mass transfer performance of fin-and-tube heat exchangers with dehumidification." *Applied Thermal Engineering* 146 (2019): 701-717. https://doi.org/10.1016/j.applthermaleng.2018.10.032

[9] Halim, Sufi Suraya, Md Tasyrif Abdul Rahman, Abdul Hamid Adom, Muhammad Sofwan Mohamad, Azizul Mohamad, and Nasrul Amri Mohd Amin. "Intake Manifold Material Selection and Fluid Flow Analysis for Formula Society of Automotive Engineers (FSAE) Race Car." *CFD Letters* 15, no. 5 (2023): 18-28. https://doi.org/10.37934/cfdl.15.5.1828

[10] Rossetti, A., S. Minetto, and S. Marinetti. "A simplified thermal CFD approach to fins and tube heat exchanger: Application to maldistributed airflow on an open display cabinet." *international journal of refrigeration* 57 (2015): 208-215. https://doi.org/10.1016/j.ijrefrig.2015.05.014

[11] Gaspar, Pedro Dinis, LC Carrilho Gonc¸ alves, and Xiao Ge. "Influence of ambient air velocity orientation in thermal behaviour of open refrigerated display cabinets." In *Engineering Systems Design and Analysis*, vol. 49163, pp. 453-462. 2010. https://doi.org/10.1115/ESDA2010-24124

[12] Gaspar, Pedro Dinis, LC Carrilho Gonçalves, and R. A. Pitarma. "Detailed CFD modelling of open refrigerated display cabinets." *Modelling and Simulation in Engineering* 2012 (2012): 9-9. https://doi.org/10.1155/2012/973601

[13] Gaspar, Pedro Dinis, LC Carrilho Gonçalves, and R. A. Pitarma. "CFD parametric studies for global performance improvement of open refrigerated display cabinets." *Modelling and Simulation in Engineering* 2012 (2012): 54-54. https://doi.org/10.1155/2012/867820

[14] Khan, Umair, William Pao, Nabihah Sallih, and Farruk Hassan. "Flow Regime Identification in Gas-Liquid Two-Phase Flow in Horizontal Pipe by Deep Learning." *Journal of Advanced Research in Applied Sciences and Engineering Technology* 27, no. 1 (2022): 86-91. https://doi.org/10.37934/araset.27.1.8691

[15] Oliet, C., C. D. Pérez-Segarra, J. Castro, and A. Oliva. "Modelling of fin-and-tube evaporators considering non-uniform in-tube heat transfer." *International journal of thermal sciences* 49, no. 4 (2010): 692-701. https://doi.org/10.1016/j.ijthermalsci.2009.11.009

[16] Enam, Samin, Muhammad Noor Afiq Witri Muhammad, and Nor Azwadi Che Sidik. "Numerical Investigation on The Effect of Grille Blockage Ratio on Air Flow Characteristics of Air Vents." *CFD Letters* 14, no. 10 (2022): 32-41. https://doi.org/10.37934/cfdl.14.10.3241

[17] Wang, Enhua, Hongguang Zhang, Boyuan Fan, Minggao Ouyang, Kai Yang, Fuyuan Yang, Xiaojuan Li, and Zhen Wang. "3D numerical analysis of exhaust flow inside a fin-and-tube evaporator used in engine waste heat recovery." *Energy* 82 (2015): 800-812. https://doi.org/10.1016/j.energy.2015.01.091

[18] Kumar, Ankur, Jyeshtharaj B. Joshi, Arun K. Nayak, and Pallippattu K. Vijayan. "3D CFD simulations of air cooled condenser-II: Natural draft around a single finned tube kept in a small chimney." *International Journal of Heat and Mass Transfer* 92 (2016): 507-522. https://doi.org/10.1016/j.ijheatmasstransfer.2015.07.136

[19] Ibrahim, Muhammad Fatih, Mohd Faizal Mohamad, Naoki Ikegaya, and Azli Abd Razak. "Numerical Investigation of Flow and Dispersion over Two-Dimensional Semi-Open Street Canyon." *CFD Letters* 15, no. 2 (2023): 53-70. https://doi.org/10.37934/cfdl.15.2.5370

[20] Santosa, I. D. M. C., IGN Suta Waisnawa, and I. W. Temaja. "Engineering Equation Solver (EES) Computer Simulation For CO 2 Refrigeration Performance." In *2018 International Conference on Applied Science and Technology (iCAST)*, pp. 728-731. IEEE, 2018. https://doi.org/10.1109/iCAST1.2018.8751503

[21] Abobaker, Mostafa, Sogair Addeep, Lukmon O. Afolabi, and Abdulhafid M. Elfaghi. "Effect of Mesh Type on Numerical Computation of Aerodynamic Coefficients of NACA 0012 Airfoil." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 87, no. 3 (2021): 31-39. https://doi.org/10.37934/arfmts.87.3.3139

[22] Kumar, Mukund, and Satish Kumar. "CFD Analysis of Fly Ash Slurry Flow Across Horizontal Pipe." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 104, no. 2 (2023): 153-160. https://doi.org/10.37934/arfmts.104.2.153160

[23] Alfarawi, Suliman SS, Azeldin El-sawi, and Hossin Omar. "Exploring Discontinuous Meshing for CFD Modelling of Counter Flow Heat Exchanger." *Journal of Advanced Research in Numerical Heat Transfer* 5, no. 1 (2021): 26-34.

[24] Santosa, IDewa MC, Baboo L. Gowreesunker, Savvas A. Tassou, Konstantinos M. Tsamos, and Yunting Ge. "Investigations into air and refrigerant side heat transfer coefficients of finned-tube CO2 gas coolers." *International Journal of Heat and Mass Transfer* 107 (2017): 168-180. https://doi.org/10.1016/j.ijheatmasstransfer.2016.11.011

[25] Morales-Fuentes, A., and Y. A. Loredo-Sáenz. "Identifying the geometry parameters and fin type that lead to enhanced performance in tube-and-fin geometries." *Applied Thermal Engineering* 131 (2018): 793-805. https://doi.org/10.1016/j.applthermaleng.2017.12.057

[26] Wen, Mao-Yu, and Ching-Yen Ho. "Heat-transfer enhancement in fin-and-tube heat exchanger with improved fin design." *Applied Thermal Engineering* 29, no. 5-6 (2009): 1050-1057. https://doi.org/10.1016/j.applthermaleng.2008.05.019

[27] Taler, Dawid, and Paweł Ocłoń. "Thermal contact resistance in plate fin-and-tube heat exchangers, determined by experimental data and CFD simulations." *International Journal of Thermal Sciences* 84 (2014): 309-322. https://doi.org/10.1016/j.ijthermalsci.2014.06.001

[28] Morales-Fuentes, A., O. M. Chapa-Contreras, S. Méndez-Díaz, and J. M. Belman-Flores. "Analysis of the heat transfer area distribution in a frosted plain fin-and-tube geometry." *International Journal of Refrigeration* 75 (2017): 26-37. https://doi.org/10.1016/j.ijrefrig.2017.01.016