Performance analysis of finned-tube CO_2 gas cooler with advanced 1D-3D CFD modelling development and simulation

X.Y. Zhang[a], Y.T. Ge*[a], J.N. Sun[b]

[a]Sustainable Environment Research Centre, Faculty of Computing, Engineering and Science, University of South Wales, Pontypridd, CF37 1DL, UK

[b]RCUK National Centre for sustainable Energy Use in Food Chains (CSEF), Institute of Energy Future, Brunel University London, Uxbridge, Middlesex, UB8 3PH, UK.

ABSTRACT

Due to natural refrigerant applied, CO_2 transcritical refrigeration and heat pump systems have been widely applied and attracted more attentions. As a main component, CO_2 gas cooler plays an important role in the system performance and thus requires further development for design and control optimizations with advanced technology. Correspondingly, a new coupled 1D and 3D Computational Fluid Dynamics (CFD) model on a finned-tube CO_2 gas cooler has been proposed and developed. The CFD model has been validated by comparing with literatures for parameters including airside heat transfer coefficient, refrigerant side temperature profile as well as heating capacity. The model has been applied to predict the heat exchanger performance at different operating conditions of both air and refrigerant sides and maldistributions of air flow inlet. It is found from the simulation results that the refrigerant temperature decreases abruptly in the first coil row and the refrigerant temperature profile along the heat exchanger tubes is affected by thermal conduction between two adjacent tube rows through fins. In addition, the higher air flow inlet velocity can reduce greatly the coil approach temperature and thus improve the system efficiency. Similar effect can also be found from refrigerant pressure. Furthermore, the non-uniform air flow patterns can affect considerably the coil performance in terms of the refrigerant temperature profile, coil approach temperature, coil heating capacity and system energy efficiency. The developed CFD model can be an efficient tool for the performance evaluation and optimisation of the CO_2 gas cooler and its associated system.

Keywords: CO_2 finned-tube gas cooler, thermal conduction through fins, refrigeration system efficiency, Computational Fluid Dynamics (CFD) modelling, air flow maldistribution.

* **Corresponding author:** Yunting Ge, Email:yunting.ge@southwales.ac.uk.

Nomenclature

A	area (m^2)
C_p	specific heat at constant pressure (J kg^{-1} K^{-1})
d	tube inner diameter (m)
D	depth (m)
f	fanning friction factor
Fp	fin pitch (m)
F_s	factor of safety
GCI	grid convergence index
H	height (m)
h	heat transfer coefficient (W m^{-2} K^{-1})
$h_{a,CFD}$	airside average heat transfer coefficient calculated by CFD
j	Colburn factor
k	thermal conductivity (W m^{-1} K^{-1})
L	flow length (m)
$\dot{m}$	mass flow rate (kg/s)

N number of tube row

Nu Nusselt number

P pressure (Pa)

Pr Prandtl number

Q heat transfer (W)

Re Reynolds number

Re_{in} Reynolds number based on air frontal velocity

T temperature (K)

u velocity (m/s)

va air velocity(m/s)

W width (m)

Z order of grid convergence

Greek symbols

Δ difference

$\boldsymbol{\rho}$ density (kg m^{-3})

ξ friction coefficient

$\boldsymbol{\mu}$ dynamic viscosity (Pa·s)

τ grid refinement ratio

ε relative error

α, β, γ correlation parameters (dimensionless)

Subscripts

a air

ain air inlet

i *i*th grid

r refrigerant

rin refrigerant inlet

w wall

1. Introduction

As a natural, non-toxic and non-flammable working fluid with superb thermophysical properties including higher values of density, latent heat, specific heat, thermal conductivity and volumetric cooling capacity as well as lower value of viscosity [1]. CO_2 plays an important role in many energy conversion systems particularly refrigeration and heat pump. Compared with conventional refrigeration or heat pump systems with subcritical vapour compression cycles, the heat rejection process of a CO_2 refrigeration system takes place mostly at supercritical pressure and temperature without condensation when a fined-tube heat exchanger is applied. This is due to the low CO_2 critical temperature (30.98°C) and high temperature of heat rejection medium. Subsequently, CO_2 refrigeration systems normally operate in transcritical cycles and CO_2 gas coolers are used for the heat rejection instead of condensers. The optimal design and operation of CO_2 gas coolers are thus very important to the performance of CO_2 transcritical cycles and need to be further investigated.

It is known that the performance of CO_2 transcritical cycles can be modified and improved by several aspects such as improvement of component operational performances and gas cooler CO_2 exit temperature profiles[1], the use of internal heat exchangers , the employment of compression and expansion integrations , controls [2]as well as the optimisation of gas cooler pressures. A theoretical investigation [3] was carried out on a two-stage transcritical CO_2 refrigeration cycle equipped with a sub-cooler and intercooler. The results showed that the decrease of the CO_2 gas cooler exit temperature led to higher system COP. A CO_2 refrigeration system with an internal heat exchanger was experimentally set up by Kim [4] to investigate the control of gas cooler pressure. The system COP could be enhanced with an internal heat exchanger installed and appropriate control applied. In addition, the thermodynamic analysis of a CO_2 transcritical cycle revealed that the expansive valve component had the largest percentage of total irreversibility in the system. Its replacement with an turbo-expander could greatly reduce the expansion process's contribution to total cycle irreversibility [5]. It was known that there existed an optimal gas cooler pressure to achieve the peak value of COP under the conditions of specific amount of refrigerant charge at the same heat sink temperature. The author [6] proposed a method to control gas cooler pressure in order to maximize practically the system COP at some specific conditions such as evaporating temperature and approach temperature.

As explained previously, a finned-tube gas cooler is typically used in a CO_2 transcritical cycle due to its characteristics of simplicity, durability and versatility. In the past decades, many efforts have been put to investigate the refrigerant side heat transfer and hydraulic behaviours of such a heat exchanger. Based on wall and bulk Nusselt numbers, general correlations of refrigerant in-tube heat transfer coefficients were evaluated and developed by Gnielinski's [7], which could also be applied to the calculations of CO_2 refrigerant. More specially, a number of researchers [8-10] investigated experimentally the heat transfer coefficients of supercritical CO_2 in-tube cooling processes with different tube diameters. Pitla et al. [11] obtained a new correlation to calculate the heat transfer coefficient of supercritical CO_2 in cooling process based on analyses of experimental and numerical data. On the other hand, Wang et al. [12] proposed correlations to calculate the airside heat transfer coefficient and pressure drop for plain finned-tube heat exchangers based on the measurements of 74 heat exchanger samples with different geometric dimensions. These heat transfer and hydraulic calculations of both refrigerant and air flow sides are essential to understand and analyse the heat exchanger performance and modelling development.

Many mathematical models have been developed to simulate the performance of various heat exchangers and the ε-NTU and LMTD are two common numerical methods applied. Lee and Domanski [13] used tube-by-tube approach to model finned-tube air to refrigerant evaporators and simulate their performances at different operating conditions. Ge and Cropper [14] applied a detailed distributed method to model CO_2 gas coolers and predict CO_2 refrigerant temperature profiles along refrigerant pipe flow direction. Due to the complicated performance of airflow side and abrupt property changes of supper critical CO_2 flow side in the CO_2 gas cooler, a 3D computational fluid dynamics (CFD) simulation method is therefore expected owing to its high simulation accuracy. Comparing to experimental investigation, the CFD model can reduce significantly the construction time of physical prototype and enable researchers to perform experiment in a virtual laboratory. The design of a finned-tube gas cooler can be specified by a number of parameters including fin pitch, fin thickness, fin area, tube diameter, tube transverse and longitudinal pitches as well as tube arrangements. These design parameters will affect greatly the heat exchanger performance no matter what refrigerants are applied. Starace et al. [15] proposed a "hybrid method" using both numerical and analytical approaches to obtain the overall performance of heat exchanger starting from CFD simulations at micro-scale. A suggestion was concluded that multi-scale approach leads to a better accuracy level compared to full-scale one. Starace et al. [16] applied this "hybrid method" to the plate-finned tube evaporator geometry. The results showed that the refrigerant evaporated completely in the first row, however, for the last row the vapour quality was 28% lower at outlet. Erek et al. [17] performed CFD simulation to analyse the influence of some effective geometric parameters of a water-flue gas finned-tube heat exchanger on flue gas heat transfer and pressure drop and found that fin pitch had significant impact on the flue gas pressure drop. Another important result was that the elliptical tube presented better heat transfer performance than that of circle tube. Similar results were obtained by Bhuiyan et al. [18] on air flow side CFD simulation of finned-tube heat exchangers. In addition, Yogesh et al. [19] investigated the effects of elliptical tube dimensions on the air flow heat transfer performance with ellipticity ratio ranging between 0.6 and 0.8. It was found that when the ellipticity ratio reached to 0.6, the airside heat transfer coefficient achieved the peak value. It is known that un-uniform of airflow in practical finned-tube heat exchanger operation can lead to different heat transfer characteristics compared with that uniform airflow. Singh et al. [20] used both segment-by-segment approach and 3D CFD simulation method to investigate the effects of complex airflows for R410A air cooled condenser. One of the important results was that airflow rate distribution and air propagation can be identified accurately with the integration of distributed heat exchanger model and CFD methods. Fiorentino and Starace [21] proposed a 2D CFD model of falling film evaporation on horizontal tubes to study the temporal change characteristics of the film flow process by changing water-to-air mass flow ratio and tubes arrangement. A trade-off curve of a specific geometry was achieved. A detailed thermal-hydraulic performance analysis for the effect of airflow maldistribution on finned-tube heat exchanger was presented by Yaïci et al. [22]. It was found that Colburn j-factors could increase or decrease by 50% at airflow maldistribution conditions compared that at uniform airflow circumstance. Although there have been a number of CFD modelling developments in the areas of finned-tube heat exchangers by a number of researchers, to the authors' acknowledge, the CFD modelling works are mostly on the fin-side fluid heat transfer and hydraulic characteristics and very few include the simultaneous analyses on the effect of tube- side fluid behaviours particularly for the CO_2 gas coolers. Consequently, the effect of thermal conduction through fins on the refrigerant side temperature cannot be detected appropriately. The complete CFD modelling of CO_2 gas coolers, however, can provide accurate boundary conditions for the air side modelling and also predict detailed refrigerant temperature profiles along refrigerant flow direction.

Correspondingly, in this paper, a novel coupled 1D and 3D Computational Fluid Dynamics (CFD) model on a finned-tube CO_2 gas cooler has been proposed, developed and explained. The Combined CFD model can predict accurately the effect of thermal conduction between tubes through connected fins on the CO_2 refrigerant side temperature profile and thus the refrigeration system efficiency. The conventional CO_2 gas cooler models could not detect this effect properly due to less detailed analyses on the fin sides, which will not obtain better instruction for the optimal fin designs. The CFD model has been validated by comparing with literatures. It has then been applied to predict the heat exchanger performance, temperature profiles of refrigerant and fins, and system energy efficiency at different operating conditions of both air and refrigerant sides and maldistributions of air flow inlet. The developed CFD model can be an efficient tool for the performance evaluation and optimisation of the CO_2 gas cooler and its associated system.

2. Numerical Methodology

A one-dimensional (1D) CFD numerical model is a feasible way to investigate the performance of finned-tube gas cooler due to simplified coil geometry and largely reduced computing time. However, the 1D model cannot capture the temperature gradients or profiles vertical to the flow direction in the pipe and detect the heat conduction between two pipes through connected fins. On the other hand, a three-dimensional (3D) model divides the whole heat exchanger into a number of small elements, applies and solves the mass, energy and momentum conservation equations for each element by using finite volume method. It is worth to mention that the 3D CFD model is more accurate and is able to capture most characteristics of the heat exchanger. However, for the fined-tube CO_2 gas cooler to be investigated in this paper, a full-scale 3D CFD model alone is not an effective and applicable method to complete the model simulation considering of the complicated coil geometry and remarkable computation time.

Correspondingly, in this paper, a coupled 1D-3D CFD numerical model is proposed and developed to analyse the performance of CO_2 finned-tube gas coolers at different operating conditions. This can be a feasible modelling method to ensure comprehensive and accurate simulation results and simultaneously maintain reasonable computing time. In this study, the whole modelling procedure is divided into phase I model and phase II model. In both models, 1D model developed by C language is used to customize thermo-physical properties and calculate heat transfer coefficient of CO_2 according to empirical correlations from published literature. The CO_2 thermo-physical properties of density, viscosity, specific heat capacity and thermal conductivity are all functions of temperature and pressure, which are obtained from REFPROP 8.0 software and then written in the C language program. For the fins and air flow, the fin surface temperatures and air flow parameters vary in three dimensions such that a 3D CFD model is necessarily employed. These models are then processed by a routine that couples 1D model and 3D CFD model to predict the overall performance of gas cooler.

In detail, the modelling route firstly starts from airside to calculate the airside heat transfer coefficient, in which fluid flow and heat transfer are processed in a passage between two consecutive fins in phase I model. The calculation is based on the conservation equations applied of mass, momentum and energy. Then, the simulation route turns into phase II model including 10 fins, airside heat transfer coefficient of each grid achieved from phase I model are assigned to surface of fins and tubes of phase II model as boundary conditions. In this case, the number of mesh elements of each fin in phase II model should be same as that of phase I model such that heat transfer coefficient of each grid can be perfectly matched. During the simulation process, a routine written in C was loaded into ANSYS FLUENT 18.2 by User Define Function that each pipe is divided into a number of segments to calculate tube side heat transfer rate, refrigerant heat transfer coefficient and refrigerant temperature for each segment. The refrigerant temperature of one tube segment can be used as the input for its next segment based on its pressure, physical-thermal properties and mass flow rate. The calculation run through each number of pipes along the refrigerant path. As inlet temperature and mass flow rate of refrigerant is known, the other temperature could be updated in each iteration and finally converged by setting up energy conservation equation. Consequently, CO_2 temperature profile and the temperature distributions of fin surface as well the velocity distribution of air domain can be computed by this 1D-3D CFD simulation method. The following governing equations are employed for the present study.

Conservation of mass:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

Conservation of momentum:

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial P}{\partial x}+\mu(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}) \tag{2}$$

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial P}{\partial y}+\mu(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}) \tag{3}$$

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial P}{\partial z}+\mu(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}) \tag{4}$$

Conservation of energy:

$$\rho C_p\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) \tag{5}$$

2.1. Physical Model

In this study, a typical staggered CO_2 finned-tube gas cooler depicted in Fig. 1 is selected and investigated. The air flow passes from right to left and refrigerant flows into the top tube numbered ‘0’ and out from the bottom tube numbered ‘53’. Table 1 shows the specification of the coil parameters.

The CFD model is developed based on the following assumptions:

(a) The model is developed under steady state condition.

(b) The actual raised lance fins are simplified as plain fins.

(c) A small coil element consists of two consecutive fins and connected short tubes as well as associated air domain which are used to calculate local airside heat transfer coefficient of each short tube.

(d) Air flow hydraulic behaviours between each small coil element is assumed the same under the consideration of symmetrical geometry.

(e) The refrigerant temperature does not change when it flows within a short distance.

2.2. 1D-3D CFD model: Phase I

In phase I model, as shown in Fig. 1, a small coil element containing two adjacent fins and connected short tubes as well as air domain is purposely selected to calculate airside heat transfer coefficient. The coil element highlighted in Fig. 1 has two fins and a number of short tubes between them, as shown in Fig.1 (c). This model is built in SolidWorks 2017. Then the 3D geometry of this model in STEP format is imported to ANSYS ICEM CFD 18.2 while in ICEM each part of the geometry is named. The geometry is meshed using hexahedral type elements. Meshing is an important step for pre-processing simulation since the quality of mesh could significantly influence the accuracy of simulation results, each element of the mesh holds specific solutions of the conservation equations applied. The detailed mesh specification can be seen in table 2.

To facilitate the modelling, the small coil element is further divided into a number of smaller elements based on the number of short tubes contained. For each smaller element, the airside model is applied to solve the mass, momentum and energy equations at a steady state heat transfer condition. Heat transfer coefficient is an important parameter to calculate the convective heat transfer between solid tube surface and heat transfer fluid (airflow). For the selected smaller element, the local airside heat transfer coefficient can be determined by the heat flux and temperature difference between tube outer surface and incoming air flow. If air inlet average temperature is used for the heat flux calculation of each tube, the heat transfer coefficient near the second and third tube rows could be inaccurate. The reason is primarily caused by the lager air temperature changes when flowing through fins. The feasible method is to use air bulk temperatures in different sections to obtain various heat transfer coefficients. A modified method for obtaining heat transfer coefficient of each particular point using the results of CFD model contains two consecutive fins and an air domain. In this method, local airside heat transfer coefficient is determined by air temperature distribution in fluid domain. The total air temperature increase equals to the summation of temperature increases over the first row, the second row and the third row. Air temperatures along the gas cooler are changed through three sections, which are section 0, section 1 and section 2. The evaluation planes between two consecutive fins are assumed to obtain the average air temperature of $T_{1,air}$ and $T_{2,air}$, as shown in Fig. 2. The local heat transfer coefficient is determined by the temperature difference between surface and average temperature of different sections.

The airside heat transfer coefficient at each particular point is calculated as:

$$h_{a,i} = \frac{Q_{a,i}}{A_i(T_{w,i} - T_{a,average})} \tag{1}$$

The Colburn j-factor is expressed as:

$$\mathrm{j} = \frac{Nu}{Re_{in} Pr^{1/3}} \tag{2}$$

The fanning f-friction factor is defined as the ratio of sheer stress and flow kinetic energy density, relating to the pressure drop of air in passages:

$$\mathrm{f} = \frac{\Delta P f_p}{2\rho u^2 L} \qquad (3)$$

The meshed gas cooler models are imported to ANSYS Fluent 18.2 to solve mass, momentum and energy governing equations. The boundary conditions used in this model are listed in table 3.

2.3. 1D-3D model: Phase II

In phase II model, the entire gas cooler is divided into 10 segments along the pipe length direction in which the length of each segment is ΔZ, as shown in Fig. 1(b). In each segment, it contains approximately 19 consecutive small coil elements. It is assumed that when refrigerant fluid flow through the length of ΔZ in each pipe, its temperature does not change. Following the assumptions of (d) and (e), the entire gas cooler model is developed based on 10 consecutive fins to simplify the model development and simulation processes. This geometry is also built in SolidWorks 2017 and the model is meshed in ANSYS ICEM CFD 18.2. The mesh details are shown in table 2. The airside heat transfer coefficient profile developed and calculated from phase I analysis is used in phase II model as the boundary condition of coil fin and tube surfaces. The boundary conditions are indicated in table 3. In addition, to predict the performance of gas cooler precisely, transverse heat conduction is considered in the model.

On the refrigerant side, Gnielinski correlation is used to calculate the respective heat transfer coefficient [7]:

$$Nu = \frac{\xi/8(Re-1000)Pr}{12.7\sqrt{\frac{\xi}{8}}\left(Pr^{\frac{2}{3}}-1\right)+1.07} \qquad (4)$$

where Filonenko's correlation is used to predict the friction coefficient [7]:

$$\xi = (0.79\, ln(Re) - 1.64)^{-2} \tag{5}$$

While Reynolds number (Re), Nusselt number (Nu) and Prandtl number (Pr) are calculated respectively in Eqs. (6), (7) and (8).

$$Re = \frac{\rho u d}{\mu} \tag{6}$$

$$Pr = \frac{\mu c_p}{k} \tag{7}$$

$$Nu = \frac{h_r k}{d} \tag{8}$$

Refrigerant side heat transfer coefficient can be determined by following equation:

$$h_r = \frac{Nu}{d} k \tag{9}$$

There is a heat balance between surface and refrigerant, where refrigerant temperature of each segment will be calculated:

$$Q_r = \dot{m} c_p \left(T_{r,i} - T_{r,i+1}\right) = h_{r,i} A_i (T_{r,i} - T_w) \tag{10}$$

2.4. Grid independency test

Grid independence test was performed to ensure the accuracy of the CFD modelling results. Due to the number of mesh elements of each fin in phase II model is same as that of phase I model according to the simulation methodology as mention in section 2. Three hexahedral type mesh structures of phase I model with different mesh elements including 769,120, 993,168 and 1370572 sizes were performed to achieve the optimized grid number. The refrigerant outlet temperature was used to evaluate the influence of grid size. Fig. 3 shows the variations of predicted refrigerant outlet temperatures with different grid node numbers. The relative temperature difference for the model with last two mesh sizes is less than 1%. Moreover, the grid convergence index (GCI) was used to report the results of grid convergence studies.

The order of grid convergence can be calculated as Eq. (11):

$$Z = \ln\left(\frac{f_3 - f_2}{f_2 - f_1}\right) / \ln(\tau) \tag{11}$$

where f_1, f_2, f_3 are the grid solution of finest, medium and coarse mesh respectively.

The GCI for fine grid is defined as:

$$GCI_{fine} = \frac{F_s|\varepsilon|}{\tau^Z - 1} \tag{12}$$

where F_s is a factor of safety, and $F_s = 1.25$ for comparisons over three or more grids.

The relative error is defined as:

$$\varepsilon = \frac{f_2 - f_1}{f_1} \tag{13}$$

The refinement of 1.32 was used for increasing mesh elements. Two parameters of airside pressure drop and heat transfer coefficient for CFD simulations were used to check the accuracy of the grid as shown in table 4. In this study that the values of $GCI_{23}/(\tau^{p} * GCI_{12})$ are considered, which are both approximately one for pressure drop and heat transfer coefficient and indicate that the solutions are well within the asymptotic range of convergence. Therefore, the finest mesh contains 1370572 elements were selected for phase I model and 3132924 elements were used for phase II model to ensure the accuracy of CFD simulation results.

3. Model Results and Validations

The numerical research of Ge and Copper [14] used correlations of Wang et al. [12] to determine the airside fanning f-friction and colburn j-factor. Besides, the experimental results carried out by Hwang [23] were used to validate the refrigerant temperature profile along pipe flow direction. A purposely designed test rig was built up by Hwang and run at different operating conditions in CO_2 transcritical cycles to explore the heat transfer performance of a CO_2 finned-tube gas cooler. The test facilities included an airflow duct, two environmental chambers, a finned-tube gas cooler (as shown in Fig. 1), an expansion valve, an evaporator and a compressor. Several important operating parameters were varied and measured to evaluate and compare the performance of the finned-tube gas cooler. These included air inlet velocity, air inlet temperature, refrigerant inlet pressure, temperature and mass flow rate. The airflow velocity was adjusted by controlling airflow fan speed. An inverter was used to control the speed of reciprocating compressor and therefore adjust the refrigerant mass flow rate. Refrigerant temperatures at each pipe bend point along the refrigerant flow direction were measured with totally 52 thermocouples. There are totally 36 test and CFD simulation conditions. The validation of the CFD model is mainly on the results of air-side heat transfer coefficients and refrigerant temperature profiles at different operating conditions. Correlations of Wang et al. [12] are used to validate the CFD predictions of airside fanning friction factor and colburn j-factor in which 88.6% of j factors are within 15% errors and 85.1% of the friction factors are within 15% errors. As shown in Fig. 4, the database of Hwang's experiment has also validated the CFD results of CO_2 temperature profile and CO_2 gas cooler outlet temperatures in which the discrepancies between test and simulation are all within +5K.

3.1. Airside heat transfer coefficient and pressure drop

In terms of choosing turbulent or laminar flow during the CFD simulation, the air inlet Reynolds numbers based on the fin pitch were calculated in the range of 94.1-282.3 such that laminar flow and viscous models were selected. Different values of Colburn j-factor at various Reynolds numbers and different operating conditions have been calculated and compared with those calculated by Wang et al.'s [12] correlations to evaluate and validate the calculations of airside heat transfer coefficients , as shown in Figs. 5(a) and 5(b). Accordingly, the airside heat transfer coefficient increases from 47.71W/m^{2}K to 73.37 W/m^{2}K with Reynolds number rising from 94.1 to 282.3. The maximum deviation between the CFD predicted j-factor and that of Wang et al.'s correlation is 4% showing a good agreement between the CFD simulation results and the literature correlations.

Airside pressure drop only changes with Reynolds number. Pressure drop increases as the airflow inlet velocity increases. When air flows through a crossflow finned-tube gas cooler, pressure drop created and it can be affected by several factors, such as fin dimensions, tube rows, fin structures and air velocity. If the pressure drop is excessive high, more electricity will be needed for the airflow fan. The comparison of Fanning friction f-factor between CFD simulation and Wang's correlation [12] at different Reynolds number is shown in Fig. 5(a), indicating maximum deviation value of 13%. The present study aims to investigate the airflow pressure drop of lanced finned-tube gas cooler with the CFD models. Since plain fins are assumed in the CFD model, the simulation shows relatively larger discrepancy for the pressure drop at various Reynolds number comparing with the measurements [20], as shown in Fig. 6. It can be learned from the CFD simulation results that fin structure has considerable impact on pressure drop. To compensate that, the following equation is derived in order to predict the relation between air flow pressure drop with lance and plain fins (CFD) for this specific finned-tube gas cooler:

$$\Delta \mathrm{P} = \alpha * \Delta P_{CFD}^2 + \beta * \Delta P_{CFD} + \gamma \quad (10)$$

Where, $\alpha = -0.003109, \beta = 2.272, \gamma = -0.1912$

Pressure drop is caused by friction when fluid flows through passages between fins. The lanced fins have relatively higher friction leading to significantly higher pressure drop than that of plate fins. Air pressure drop is a function of air flow rate such that with the increase of air inlet velocity, the pressure drop increases. When the air inlet velocity increases from 1m/s to 3m/s, the airside pressure drops change from 20Pa to 100Pa. However, for a plate fins gas cooler, the airside pressure drop changes from 9 Pa to 47 Pa with the same air inlet velocity changes.

3.2. Temperature and velocity distribution

Figs. 7(a) and 7(b) show the velocity vector contour and path line along middle plane in airflow region. When the air flows externally through a tubular area, it separates into two side streams and then forms a pair of symmetric vortexes. The vortexes allow better mix of air. The maximum velocity of airflow thus occurs close to the transverse plane of boundary layers. This

1D-3D coupled simulation method allows researchers to observe directly the temperature distribution on the fin surface. The temperature at each point on the fin surface can be obtained. As shown in Fig. 8 (a) (b) (c), the average temperature of fin surface reaches to the lowest when air inlet velocity increases up to 3m/s. The fin surface temperature around the first tube row where the refrigerant inlet is located is the highest since it is close to the refrigerant inlet. Similarly, the fin surface temperature around the third tube row is the lowest where the refrigerant outlet is located. Fig.8 (d) (e) (f) shows the temperature distribution of air middle plane. Lower air velocity leading to higher air outlet temperature. Due to symmetric vortexes can be formed behind tubes, air is bounded in this area and thus heated by fins and tubes surface. The average temperature of air around refrigerant inlet pipes has higher temperature than other position. This is one of the reasons causes reverse heat transfer phenomenon during the process of refrigerant cooling. Besides, for the 1D model, generally the heat conduction along longitudinal direction of fin is neglected. However, from the results of present study, longitudinal heat conduction has a great influence on fin temperature, and therefore both the air and refrigerant temperatures will be affected. Heat is transferred from hotter tube to colder tube across fins. That is the main reason causes reverse heat transfer such that the refrigerant temperature might not decrease continuously along CO_2 refrigerant flow direction. However, this phenomenon is difficult to detect by other simulation methods. In summary, heat transferred from hotter tubes to fins and colder tubes by thermal conduction has significantly influence on the refrigerant and fin surface temperature distributions.

3.3. Analysis of gas cooler performance

As shown in Fig. 9, refrigerant temperature profiles along refrigerant flow direction at various operating conditions are predicted. Refrigerant temperature drops dramatically in the first-row tubes numbered from ‘0’ to ‘17’, as indicated in Fig. 1. It can be observed that approximately 90% of overall temperature drop takes place in the first-row tubes. When air flows through the gas cooler, its temperature increases greatly after the row tubes numbered from ‘36’ to ‘53’ than that after the middle row tubes. Although airflow temperature is increased at airflow exit, the temperature difference between airflow around the first-row tubes and tube wall surface is still large, causing large amount of heat transfer rate. In addition, particular thermos-physical properties of CO_2 can contribute to this phenomenon. There is a slight temperature step-down trend when CO_2 flow turns from pipe ‘18’ to ‘19’, since the air temperature around the middle row is lower than that of around first tube row, leading higher heat transfer rate and thus more temperature drop. In the middle row, refrigerant temperature decreases slightly from pipe ‘18’ to ‘26’. However, there is an upward trend when refrigerant flows from pipe ‘32’ to ‘36’. This phenomenon is prominent when air inlet velocity is at a lower value of 1 m/s. The main reason is that heat is conducted across fins from hotter tubes in the first tube row to the adjacent tubes in the middle and third tube rows. The thermal conductivity of a specified fin material is determined by temperature, material properties and path length. Higher temperature causes higher heat transfer rate through fins. Besides, when air flows after the upstream tube, it could be constrained for a long time due to the formation of vortexes. This confined air can be heated by adjacent hotter tubes, and then the heat will be transferred reversely from air to tube. Subsequently, to enhance the heat exchanger performance and the efficiency of its associated system, it is suggested to apply split fins between the first and middle tube rows so as to prevent the thermal conductions.

Under the condition of varied air inlet velocity only, the higher velocity leads to lower the refrigerant exit temperature. This is because, higher velocity can improve the heat transfer coefficient and thus heat transfer rate. At a specified refrigerant pressure, mass flow rate and similar refrigerant inlet temperature, the refrigerant temperature at any position at air inlet temperature of 302.55K is always lower than that at air inlet temperature of 308.15K. This is because higher temperature difference between surface and air leads to larger heat transfer rate and therefore lower refrigerant temperature. Compared Fig. 9(e) and Fig. 9(f), when the refrigerant pressure was 11MPa, although the refrigerant inlet temperature is approximately 10K higher than the condition of 9MPa, their refrigerant exit temperatures are close. It can be summarized from Fig. 9 that refrigerant temperature decreases with higher refrigerant pressure when other parameters are kept same. Consequently, increasing high side pressure will increase heating capacity. The COP trend of CO_2 transcritical cycle is different with traditional cycles, as there does not exist optimum COP in tradition cycles. As for a CO_2 transcritical cycle, when the optimum pressure is achieved, the maximum peak value of COP can be reached [24]. Higher refrigerant mas flow rate leads to lower refrigerant inlet temperature. Besides, it is seen from both modelling and experimental results that with increased refrigerant mass flow rate, the temperature of refrigerant decreased due to the conservation of energy. The lowest temperature discrepancies between the test and CFD results for refrigerant temperature profile along flow direction take place when air inlet velocity is 3 m/s.

Lower refrigerant exit temperature makes contribute to higher heating capacity of finned-tube gas cooler and better COP of refrigeration system. Fig. 10 depicts that the heating capacity increases with increasing air frontal velocity and refrigerant pressure. In comparison of Figs. 10(a) and 10(c), when air inlet temperature is at 302.55K, the heating capacity is always higher than that of air inlet temperature at 308.15K. In addition, heating capacity increases with the increase of refrigerant mass flow rate. For a certain refrigerant pressure at 9MPa, when the refrigerant mass flow rate increases from 0.038 kg/s to 0.076 kg/s, the heating capacities can be improved by 30.14%, 33.08% and 46.36% with air frontal velocity varies from 1,2 and 3 m/s respectively as shown in Figs. 10(a) and 10(b). Similarly, the heating capacities can be improved by 27.88%, 29.2% and 29.98% as indicated in Figs. 10(c) and 10(d). The highest heating capacity occurred at the condition that air inlet temperature is at 302.55K, CO_2 mass flow rate is at 0.076 kg/s and gas cooler pressure is at 11MPa.

The varied operating conditions of the gas cooler can indirectly affect the performance efficiency of its associated system, as shown in Fig. 11, assuming that the system evaporator exit temperature and pressure are 268.15K and 3.0459MPa respectively. As depicted in Fig. 11, similar effect can be found between the coil heating capacity and system cooling COP at different operating conditions of the gas cooler. The lower air inlet temperature and higher refrigerant mass flow rate can both benefit to the system efficiency. Meanwhile, it also verifies that the gas cooler pressure of 11MPa is close to the optimal pressure for the system operation.

4. Model applications

Most of the researches on finned-tube heat exchanger CFD modelling were based on uniform airflow velocity. There is a lack of research investigation and data for the analyses of airflow misdistribution effect on the CO_2 gas cooler performance. The validated model is thus used to investigate the effect of airflow velocity maldistribution on the performance of CO_2 finned-tube heat exchanger. As shown in Fig. 12, four inlet air velocity profiles are studied in the CFD simulation: (a) uniform velocity profile ; (b) linear-up velocity profile; (c) linear-down velocity profile; (d) parabolic velocity profile. The four velocity profiles have the same average face velocity. The uniform airflow parttern is used as the baseline model. Each airflow pattern is studied for different Reynolds number ranges from 94.1 to 282.3.

There is an obvious trend that Colburn j-factor and Fanning f-factor decrease with higher Reynolds number. The j factor of linear-up airflow is always lower than that of uniform velocity profile with Reynolds number increasing. When air average inlet velocity is at 1 m/s, the j factor of linear-up velocity profile is 19.13% lower than that of uniform airflow as indicated in

Fig. 13(a). However, the j factors of linear-down airflow pattern are always higher compared with uniform airflow when air inlet average velocity varies from 1m/s to 3m/s. When air inlet average velocity reaches to 3m/s, the j factor of parabolic case is higher than that of uniform. Colburn j-factor is a dimensionless parameter and a function of heat trasnfer coefficient, airside heat trasnsfer coefficients of linear-up, linear-down and parabolic cases increase from 38.57 to73.09 W/m^{2}K, 50.15 to 78.07 W/m^{2}K and 43.63 to 76.97 W/m^{2}K respectively with Reynolds number varying from 94.1 to 282.3. In comparision of the linear-down and linear-up velocity profiles, they have the similar heating capacity as shown in Fig. 15(b), but the linear-up velocity pofile generates higher air average temperature. Therefore, the average heat transfer coefficient of linear-up case is lower than that of linear-down case. Fig. 13(b) indicates the difference of airside pressure drop between uniform and maldistribution airflows. The linear-down case has the highest pressure drop, which is 7.9%, 12.1% and 15.7% more than that of uniform case in terms of f factor when Reynolds numbers change from 94.1 to 282.3. Results are achieved based on plate fins, air non-uniform distribution could have more significant impact on raised lance fin gas cooler pressure drop. It is concluded that maldistribution air velocity profile always cause higher pressure drop. Fan power is the only energy required by airside, the high pressure drop requires high power of fan. The most important method to reduce fan power is to decrease pressure drop.

The inlet airflow pattern can directly influence refrigerant temperature profile as shown in Fig. 14. Although the airflow is non-uniform, most heat of refrigerant is rejected through the first-row of gas cooler similiar to that with uniform airflow pattern. Non-uniform airflow patterns also cause reverse heat transfer, especially when air average velocity is at 1m/s. However, this phenomon is greatly minized due to the velocity characteristics of linear-up pattern, which means air velovity near high temprature pipes is larger such that the heat exchange is improved. Approach temperature is defined as the temperature difference between air inlet and refrigerant exit temperature, which has considerable impact on cooling capacity and heat transfer performance of heat exchanger. From Fig. 14(b), it is known that the parabolic airflow pattern has higher heating capacity compared with linear-up and linear-down, while the performance with airflow linear-up and linear down is quite close. However, uniform case has the lowest approach temperature and thus highest heating capacity in all cases. When Reynolds number varies from 94.1 to 282.3, the approach temperature differences between uniform velocity profile and parabolic and linear-down as well as linear-up are 0.479K, 2.948K and 4.471K respectively as shown in Fig. 15(a). The influence of air maldistribution on heating capacity is not prominent when air average velocity is low, as shown in Fig.15 (b). Although the average heat transfer coefficient of linear-down velocity profile is the largest, the gas cooler performance is however the worst. This is because different airflow pattern causes different local heat transfer coefficient, affecting the heat transfer rate dramatically and thus the refrigerant temperature. For improving the performance of gas cooler, uniform airflow and high airside velocity can make the best contribution. When the refrigerant temperature and pressure evaporator exit are assumed as 268.15K and 3.0459MPa, the cooling COP of its associated system can also be calculated at the conditions of gas cooler air maldistribution, as shown in Fig. 15 (c). Similar results can be obtained between the coil heating capacity and system cooling COP. Therefore, at a constant evaporating temperature, uniform air velocity profile has the highest system cooling COP compared with those with air maldistribution velocity profiles.

5. Conclusions

CO_2 finned-tube gas cooler plays an important role in transcritical refrigeration systems. It needs to be well investigated and controlled to achieve better performance of refrigeration cycles. A 1D-3D coupled CFD model method for a fin-and-tube CO_2 gas cooler has been proposed and explained in this paper. The model simulation results have been validated with experimental measurements and literature correlations at different test conditions. This validated model is used to investigate the effect of airflow velocity maldistribution and different operating conditions on the performance of CO_2 finned-tube gas cooler and its associated system efficiency. Followings are the key points of this study:

- The proposed method provided additional and valuable results that other methods can not be achieved. The present study has higher accuracy of results and less computation time.
- This 1D-3D model not only allows to predict airside average heat transfer coefficient, airside pressure drop and the effect of different operating conditions on refrigerant temperature profile along pipe flow direction, but also it can obtain fin surface temperature distribution and air velocity distribution and their effects on the coil performance. From the simulation results, it is suggested that split fins be applied to minimise the reverse heat transfer between tubes.
- Airside heat transfer coefficient increases with higher air inlet velocity, higher air inlet temperature as well as higher refrigerant pressure. For uniform airlfow pattern, if air inlet temperature of 302.55K and refrigerant pressure of 9MPa, airside heat transfer coefficient increases from 47.71W/m^{2}K to 73.37 W/m^{2}K with Reynolds number varies from 84.1 to 282.3.
- For both uniform airflow velocity profile and non-uniform velocity profile, airside pressure drop increases as the air inlet average velocity increases. Reducing pressure drop is an effective method to decrease the fan power consumption.
- Approximately 90% of the refrigerant temperature drop occurs in the first tube row of gas cooler due to the larger temperature difference between air and tube surfaces.

- Reverse heat transfer is an important result from this research. It can be minimized by increasing velocity. Linear-up airflow pattern can effectively improve the phenomenon of reverse heat transfer. However, its heating capacity is not as good as uniform airflow pattern.
- Approach temperature is decreased with increasing air inlet velocity. Uniform airflow pattern has the best performance of approach temperature compared with air flow maldistribution. Lower approach temperature leads to better cooling capacity, heat recovery capability of gas cooler and system cooling COP.
- Heating capacity of gas cooler and system Cooling COP are improved with the increase of refrigerant pressure (close to optimal pressure), air frontal velocity and refrigerant mass flow rate. Under the condition of only airflow pattern is the variable, uniform air velocity profile can produce the best performance of heating capacity and system cooling COP. Therefore, at a constant evaporating temperature, uniform air velocity profile has the highest COP of system compared with those at air flow maldistribution conditions.
- This detailed 1D-3D CFD modelling method can make contribute to better analyse and controls of gas cooler as well as refrigeration system in practical operations.

References

[1] Ge YT, Tassou SA, Santosa ID, Tsamos KM. Design optimisation of CO_2 gas cooler/condenser in a refrigeration system. Applied Energy 2015; 160: 973-981.

[2] Tsamos KM, Ge YT, Santosa ID, Tassou SA. Experimental investigation of gas cooler/condenser designs and effects on a CO_2 booster system. Applied Energy 2017; 186:470–479.

[3] Jamali S, Yari M, Mohammadkhani F. Performance improvement of a transcritical CO2 refrigeration cycle using two-stage thermoelectric modules in sub-cooler and gas cooler. International Journal of Refrigeration 2017; 74:105-115.

[4] Kim M, Kang D, Kim M, Kim M. Investigation on the optimal control of gas cooler pressure for a CO 2 refrigeration system with an internal heat exchanger. International Journal of Refrigeration 2017; 77:48-59.

[5] Robinson D, Groll E. Efficiencies of transcritical CO2 cycles with and without an expansion turbine. International Journal of Refrigeration 1998; 21(7):577-589.

[6] Ge YT, Tassou SA. Control optimisation of CO_2 cycles for medium temperature retail food refrigeration systems. International Journal of Refrigeration 2009; 32: 1376 – 1388.

[7] Gnielinski V. New equations for heat and mass transfer in turbulent pipe and channel flow. International Chemical Engineering 1976;16(2):359-368.

[8] Dang C, Hihara E. In-tube cooling heat transfer of supercritical carbon dioxide. Part 1. Experimental measurement. International Journal of Refrigeration 2004; 27(7):736-747.

[9] Oh H, Son C. New correlation to predict the heat transfer coefficient in-tube cooling of supercritical CO2 in horizontal macro-tubes. Experimental Thermal and Fluid Science 2010; 34(8):1230-1241.

[10] Pettersen J, Rieberer R, Leister A. Heat transfer and pressuredrop characteristics of supercritical carbon dioxide inmicrochannel tubes under cooling. The Fourth IIR-Gustav Lorentzen Conferenceon Natural Working Fluids, West Lafayette, IN. 2000; 99 – 106.

[11] Pitla S, Groll E, Ramadhyani S. 2002. New correlation to predict the heat transfer coefficient during in-tube cooling of turbulent supercritical CO2. Int. J. Refrigeration 2002; 25(7):887-895.

[12] Wang C, Chi K, Chang C. Heat transfer and friction characteristics of plain fin-and-tube heat exchangers, part II: Correlation. International Journal of Heat and Mass Transfer 2000; 43(15):2693-2700.

[13] Lee J, Domanski PA. Impact of Air and Refrigerant Maldistributions on the Performance of Finned-tubeEvaporators with R-22 and R-407C. Report No.: DOE/CE/23810-81, July 1997.

[14] Ge YT, Cropper R. Simulation and performance evaluation of finned-tube CO2 gas coolers for refrigeration systems. Applied Thermal Engineering 2009; 29(5-6):957-965.

[15] Starace, G., Fiorentino, M., Longo, M. and Carluccio, E., 2017. A hybrid method for the cross flow compact heat exchangers design. Applied Thermal Engineering, 111, pp.1129-1142.

[16] Starace, G., Fiorentino, M., Meleleo, B. and Risolo, C., 2018. The hybrid method applied to the plate-finned tube evaporator geometry. International Journal of Refrigeration, 88, pp.67-77.

[17] Erek A, Özerdem B, Bilir L, İlken Z. Effect of geometrical parameters on heat transfer and pressure drop characteristics of plate fin and tube heat exchangers. Applied Thermal Engineering 2005; 25(14-15):2421-2431.

[18] Bhuiyan A, Amin M, Islam A. Three-dimensional performance analysis of plain fin tube heat exchangers in transitional regime. Applied Thermal Engineering 2013; 50(1):445-454.

[19] Yogesh S, Selvaraj A, Ravi D, Rajagopal T. Heat transfer and pressure drop characteristics of inclined elliptical fin tube heat exchanger of varying ellipticity ratio using CFD code. International Journal of Heat and Mass Transfer 2018; 119:26-39.

[20] Singh V, Abdelaziz O, Aute V, Radermacher R. Simulation of air-to-refrigerant fin-and-tube heat exchanger with CFD-based air propagation. International Journal of Refrigeration 2011; 34(8):1883-1897.

[21] Fiorentino, M. and Starace, G., 2016. Numerical investigations on two-phase flow modes in evaporative condensers. Applied Thermal Engineering, 94, pp.777-785.

[22] Yaïci W, Ghorab M, Entchev E. 3D CFD analysis of the effect of inlet air flow maldistribution on the fluid flow and heat transfer performances of plate-fin-and-tube laminar heat exchangers. International Journal of Heat and Mass Transfer 2014; 74:490-500.

[23] Hwang Y, Jin DDH, Radermacher R, Hutchins J. Performance measurement of CO2 heat exchanger. ASHRAE Trans 2005;306-16.

[24] M.H. Kim, J. Pettersen, C. Bullard, Fundamental process and system design issues in CO2 vapor compression systems, Progress in Energy and Combustion Science 30(2004)119-174.