Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 161 (2019) 275–282

Energy Procedia

www.elsevier.com/locate/procedia

2nd International Conference on Sustainable Energy and Resource Use in Food Chains, ICSEF 2018, 17-19 October 2018, Paphos, Cyprus

CFD Modelling of Finned-tube CO_2 Gas Cooler for Refrigeration Systems

Xinyu Zhang[a], Yunting Ge[a,*] , Jining Sun[b], Liang Li[c], Savvas A. Tassou[b]

[a]*Sustainable Environment Research Centre, University of South Wales, Pontypridd, CF37 1DL, United Kingdom*
[b]*Brunel University London, Institute of Energy Futures, RCUK Centre for Sustainable Energy use in Food chains (CSEF), Uxbridge, UB8 3PH, United Kingdom*
[c]*School of Engineering and Technology, University of Hertfordshire, Hatfield, Hertfordshire, AL10 9AB, United Kingdom*

Abstract

As a main component in a refrigeration system, finned-tube CO2 gas cooler plays an important role to the system performance and thus needs to be thoroughly investigated. To achieve this, some effective parameters including the CO2 and air fluid velocity fields, temperature profiles and heat transfer characteristics at different operating conditions are predicted and analysed by means of Computational Fluid Dynamics (CFD) modelling and simulation. It is noted that CFD modelling can accurately obtain the local heat transfer coefficients of both air and refrigerant sides, which are difficult to be predicted by conventional empirical correlations. This paper investigates the effect of varied operational parameters on local heat transfer coefficients and temperature profiles of the working fluids in a finned-tube CO2 gas cooler by means of CFD modelling. As one of the simulation results, it is found that the approach temperature decreases with increased air inlet velocity. The model has been compared and validated with experimental measurements and literature correlations. The research methods and outcomes can be used for further investigation and optimization in this area.

Selection and peer-review under responsibility of the 2nd International Conference on Sustainable Energy and Resource Use in Food Chains, ICSEF2018

Keywords: CO2 finned-tube gas cooler; heat transfer coefficient; refrigeration system; Computational Fluid Dynamics (CFD);

* Corresponding author. Tel.: +44-1443 482165.
E-mail address: yunting.ge@southwales.ac.uk

1876-6102 © 2019 The Authors. Published by Elsevier Ltd.

Selection and peer-review under responsibility of the 2nd International Conference on Sustainable Energy and Resource Use in Food Chains, ICSEF2018.
10.1016/j.egypro.2019.02.092

Nomenclature

A	Area (m^2)	*Greek symbols*	
C_p	specific heat at constant pressure (J kg^{-1} K^{-1})	Δ	difference
d	tube diameter (m)	ρ	density (kg m^{-3})
D	Depth (m)	ξ	friction coefficient
f	fanning friction factor	μ	dynamic viscosity (Pa·s)
j	Colburn factor		
H	height (m)		
h	heat transfer coefficient (W m^{-2} K^{-1})	*Subscripts*	
k	thermal conductivity (W m^{-1} K^{-1})	a	air
L	flow length (m)	i	*i*th grid
$\dot{m}$	mass flow rate (kg/s)	r	refrigerant
Nu	Nusselt number	w	wall
P	pressure (Pa)		
Pr	Prandtl number		
Q	heat transfer (W)		
Re	Reynolds number based on fin spacing		
T	temperature (k)		
u	velocity (m/s)		
va	air velocity(m/s)		
W	width (m)		

1. Introduction

Chlorofluorocarbons (CFCS) and hydrochlorofulorocarbons (HCFCS) can no longer be used as refrigerants due to their long term impact on environment such as high ozone depleting potentials (ODP) and high global warming potentials (GWP). Subsequently, environmentally friendly natural working fluids with none or negligible ODPs and GWPs need to be developed and applied. As a natural working fluid, CO_2 has been widely employed for refrigeration, heat pump, air conditioning systems as well as environmental control units owing to its superb thermophysical properties and negligible environmental impact. However, the performance of CO_2 refrigeration and heat pump systems need to be further improved with some advanced technologies. These include replacement of expansive valve with expansion turbine, installation of liquid line/suction heat exchanger and utilisation of high efficient compressor, evaporator and gas cooler etc. Considering CO_2's high critical pressure (73.8 bar) and low critical temperature (30.98°C), the heat rejection process in a CO_2 refrigeration system operates conventionally under supercritical condition without condensation if the temperature of heat rejection medium is relatively high. The corresponding heat exchanger is thus called gas cooler instead of condenser.

As a main component in a CO_2 refrigeration system, CO_2 gas cooler plays an important role in the system performance and thus needs to be further investigated and designed optimally. When air acts as the heat rejection medium, air-cooled finned-tube CO_2 gas cooler is normally utilised in the refrigeration system. A great deal of researches have been carried out to investigate the heat transfer and hydraulic behaviours of the finned-tube CO_2 gas cooler with both theoretical and experimental analysis. For the CO_2 side, the heat transfer coefficients could be calculated from the correlations by Gnielinski [1] or by Pitla et al. [2]. Sparrow and Samie [3] compared the hydraulic behaviours of CO_2 gas coolers with one and two row staggered arrays and found that CO_2 pressure drop of the latter heat exchanger was nearly doubled that of the former one. As to the external air flow side, the heat transfer and friction correlations from Wang, Jang and Chiou [4] are commonly utilised for the heat exchangers with wavy fins. These heat transfer and hydraulic calculations are essential to understand and analyse further the heat exchanger performance. A number of modelling methods could be applied to analyse the performance of CO_2 finned-tube gas coolers at different operating conditions including ε-NTU or LMTD i.e. lumped, tube-in-tube and distributed. Since there is rapid change of the CO_2 thermophysical properties with temperature during an isobaric gas cooling process, it is not practical to use the ε-NTU or LMTD method to simulate gas coolers (Kim et al. [5]). The tube-in-tube method developed from the research of Domanski [6] was utilized in the simulation of a gas cooler by Chang and Kim [7]. By means of the model simulation, the effects of some coil structural parameters on the performance of the gas cooler were investigated. It was found from the simulation results that the approach temperature could be decreased with increased heat exchanger front area. Ge and Cropper [8] used a distributed method to calculate CO_2 temperature profile along CO_2 flow process using localised working fluid heat transfer and hydraulic parameters. The simulation results showed that the larger heat exchanger circuit number could increase its heating capacity. Similar modelling method was also applied by Gupta and Dasgupta [9] to evaluate the effect of airflow velocity on the heat exchanger performance. It was revealed that higher airflow velocity could decrease the working fluids approach temperature and thus increase gas cooler's heating capacity.

It is noted that some empirical correlations of airside heat transfer coefficients and pressure drop calculations are needed for the above CO_2 gas cooler models. These correlations may not be applicable for the CO_2 finned-tube gas coolers with different pipe and fin structures and various airflow profiles. To obtain accurate modelling of the CO_2 gas cooler with complicated structural and operational parameters, 3D CFD modelling of CO_2 finned-tube gas coolers is therefore expected. Yaïci et al. [10] used CFD modelling to investigate the effect of inlet air flow maldistribution on the performance of heat exchanger. Although there are some CFD modelling development in the areas of heat exchangers by a number of researchers, to the authors' knowledge, there is no detailed CFD modelling on the CO_2 finned-tube gas cooler found in public literatures.

This paper presents a numerical investigation of CO_2 finned tube gas cooler using three-dimensional CFD modelling. The airflow side heat transfer coefficient is predicted by CFD modelling and imported into the simulation model of the whole heat exchanger. The modelling approach is new and applicable to model heat exchangers with complicated structures. The developed CFD model has been validated by comparing calculations and measurements from public literatures. The model is then applied to predict the refrigerant temperature profiles at different operating conditions. Further research and development in this area has also been discussed.

2. Model description

A 3D CFD model has been developed to simulate the performance of a CO_2 finned-tube gas cooler at different operating conditions. The modelling approach is summarised and listed below:

- Symmetrical condition is assumed on the mid plane between two consecutive fins in the heat exchanger.
- Due to the coil symmetry structure, the airside heat transfer coefficient of each passage between two consecutive fins is assumed to be the same. The model of two fins with air flow region is then used to obtain local airside heat transfer coefficient which is then exported into a txt type file. The User Defined Function (udf) from CFD and a code developed in Visual Studio 2017 are then applied to assign the heat transfer coefficient to the coil's external surface.
- The gas cooler is divided into 10 segments along the pipe length direction. Since there are 375 fins in the coil, each segment contains about 37 fins. To simplify the simulation process, the entire gas cooler model is developed based on one segment model. Meanwhile, the transverse heat conduction is also considered in the model.
- The developed CFD model is then validated with the test results from public literature.

The modelling routine starts firstly from the airside, and then the simulation runs through each numbered pipe. The whole modelling calculation depends on conservation equations of mass, momentum and energy of each pipe segment. For one coil segment, each pipe is further divided into a number of sections or small finite elements. As shown in Figure 1, there are totally 54 pipes in the heat exchanger. Each numbered pipe consists of a number of finite elements. Setting up energy conservation equation for each element will help to calculate the CO_2 temperature of each pipe segment and thus the whole pipe. The output temperature of each pipe is then used as the inlet temperature for next numbered pipe. After iteration, the refrigerant outlet temperature of the final coil pipe can be calculated. Iteration should be carried on until energy conservation equation reaches convergence.

2.1. Physical model

An identical CO2 finned-tube gas cooler is selected for CFD modelling which is used by Ge and Cropper[8] as shown in Figure 1. Airflow passes from right to left and refrigerant flows into the top tube numbered 0 and out from the bottom tube numbered 54. The specification of the modelled gas cooler is listed in Table 1.

Table 1. Specification of the modelled gas cooler.

Dimensions	
W × H × D (m)	0.61 × 0.46 × 0.05
Front area (m^2)	0.281
Fin shape	Raised lance
Fin pitch (mm)	1.5
Fin thickness (mm)	0.13
Number of tubes row	3
Tube outside diameter (mm)	7.9
Tube inside diameter (mm)	7.5
Tube shape	Smooth

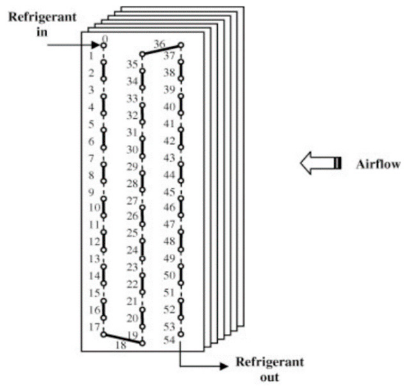
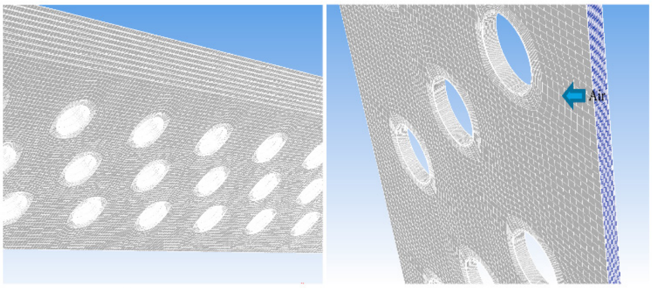


Fig.1. Modelled gas cooler of numbered tube pipes

2.2. CFD model

As explained above, the 3D CFD model of the plain finned-tube CO_2 gas cooler has been developed and demonstrated in this present work. The overall modelling process is composed of three parts. In the first part, the created geometry consists of fins and tubes having one refrigerant inlet (pipe '0') and one refrigerant outlet (pipe '54') while SOLID WORK software is applied to draw the geometry. Then the geometry of 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 model accuracy increases with larger cell numbers but is limited by computer memory. For the airside model, it consists of 2 consecutive fins and 54 tube pipes while the region between 2 fins is selected as air domain. As for the CO_2 side model, it consists of 10 consecutive fins and 54 tube pipes. The geometry and meshes of one coil segment and two fins only are depicted in Figure 2 while the mesh details for both CO_2, fins and airflow sides are described in Table 2. In the second part, the meshed gas cooler is imported to ANSYS Fluent 18.2 to solve mass, momentum and energy governing equations. After iteration, each equation achieves convergence with the maximum error of 10^{-6}. The final part of the model is post-processing including the demonstrations of temperature, velocity and pressure contours as well as other key characteristics to evaluate gas cooler performance.



(a) (b)

Fig.2. Meshing of geometry, (a) 10 fins gas cooler model, (b) 2 fins and air flow region model.

Table 2. Mesh size

Case	Number of nodes	Number of faces	Number of cells
Airside model	1263000	3393606	1060928
CO_2 side model	2220000	5113610	1427024

2.3. Air and Refrigerant side heat transfer coefficients

Heat transfer coefficient is an important parameter to calculate convective heat transfer between solid surface and fluid. The local airside heat transfer coefficient is determined for each individual surface of either fin or tube. The overall heat transfer coefficient of both fin and tube surfaces is calculated with surface integrals from FLUENT:

$$h_{a,i} = \frac{Q_i}{A_i\left(T_{w,i} - T_a\right)} \tag{1}$$

On the refrigerant side, Gnielinski correlation is used to calculate the respective heat transfer coefficient.

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

where, Filonenko's correlation is used to predict the friction coefficient,

$$\xi = \left(0.79\ln\left(Re\right) - 1.64\right)^{-2} \tag{3}$$

Refrigerant side heat transfer coefficient can be determined by equation (4):

$$h_r = \frac{Nu}{d}k \tag{4}$$

Furthermore, the refrigerant outlet temperature is calculated from heat transfer rate. 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 software.

$$T_r = \frac{Q_r}{h_r A} + T_w \tag{5}$$

$$Q_r = \dot{m}c_p(\Delta T_r) \tag{6}$$

2.4. Boundary conditions

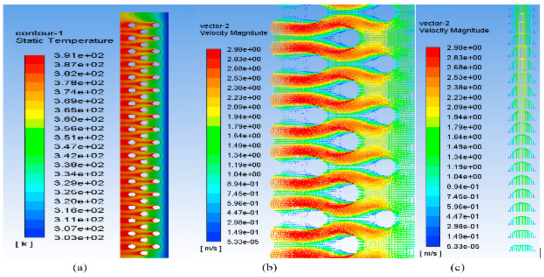
In this study, since the calculated Reynolds number of airflow based on fin space is less than 2000, laminar flow and viscous model is selected. The following boundary conditions have been used for investigating airside heat transfer coefficient.

- The materials of fin and tube are aluminum and copper respectively.
- The outer side walls of fins are assigned as adiabatic wall.
- Airflow region between two consecutive fins is selected and studied.
- The air thermo-physical properties of density, viscosity, specific heat capacity and thermal conductivity are all functions of temperature and pressure, which are obtained from REFPROP software. Each property can be input into FLUENT by means of piecewise-linear function.
- For each pipe section, tube inside wall temperature is set as constant.
- The air inlet is set as velocity inlet and outlet as pressure outlet.

Fig.3 shows the simulation results for temperature and velocity contours of airflow. Based on the results, the airflow temperature around the first row pipe from the right has the lowest temperature compared to the other two rows. When the air flows externally through tubular area, it separates into two side streams and then forms a stagnant area. The maximum velocity of airflow is thus occurred close to the transverse plane of boundary layers. The performance of airside heat transfer coefficient in one passage between two consecutive fins will be used for the whole gas cooler model. It is noted that the average airside heat transfer coefficient increases with the higher air inlet velocity, as shown in Fig.4.

To acquire the refrigerant temperature and pressure changes, the following boundary conditions have been used for the entire gas cooler model.

- Import local airside heat transfer coefficient derived from separate CFD calculation using User Defined Function (udf). A code has been developed in Visual Studio 2017 for this purpose.
- Fin, tube outside and inside surfaces are set as convection boundary conditions.



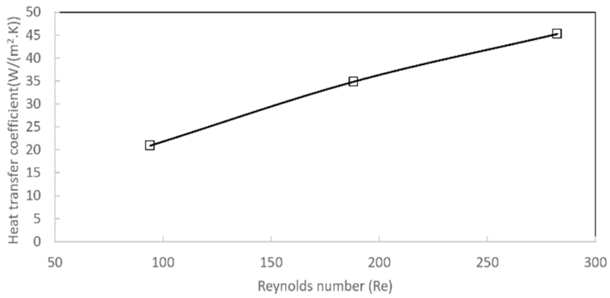


Fig.3. Temperature and velocity contour of air (air inlet velocity = 1m/s). (a)temperature contour of middle plane in air flow region, (b)velocity contour of middle plane in air flow region, (c) air inlet velocity contour between fins.

Fig.4. Variation of airside heat transfer coefficient with Reynolds number

The fanning f-friction factor is defined as the ratio of sheer stress and flow kinetic energy density, relating to the pressure drop of refrigerant in tube pipe. It should not be confused with friction coefficient. The difference is that the value of friction coefficient is four times the value of fanning f-friction factor.

$$\mathrm{f} = \frac{\Delta P d}{\frac{1}{2}\rho u^2 4L} \tag{7}$$

The Colburn j-factor is expressed as:

$$j = \frac{Nu}{RePr^{1/3}} \tag{8}$$

3. Validation

For the CFD modelling of finned-tube CO_2 gas cooler, The air inlet Reynolds number based on the fin pitch is in the range of 90-300, while based on the tube collar diameter is in the range of 1400-4600. The numerical research of Ge and Copper [8] used correlations of Wang et al. [11] to determine the airside fanning f-friction and colburn j-factor. According to the experiment test conditions, the airflow and refrigerant inlet temperatures are maintained at 302.55 k and 391.25 k respectively. In this CFD modelling study, three different values of f-factor and j-factor at various Reynolds numbers have been calculated and compared with those calculated by Wang et al.'s correlations, as shown in Fig.5. The maximum deviations of f-friction factor and j factor between present study and Wang et al.'s research are in the order of 34% and 30% respectively. When Reynolds number is 188.2, the deviation of j-factor deviation is minimum. The deviation of f factor reaches minimum value when the Reynolds number is 282.3. Subsequently, reasonable agreement can be obtained between CFD results and the literature correlations.

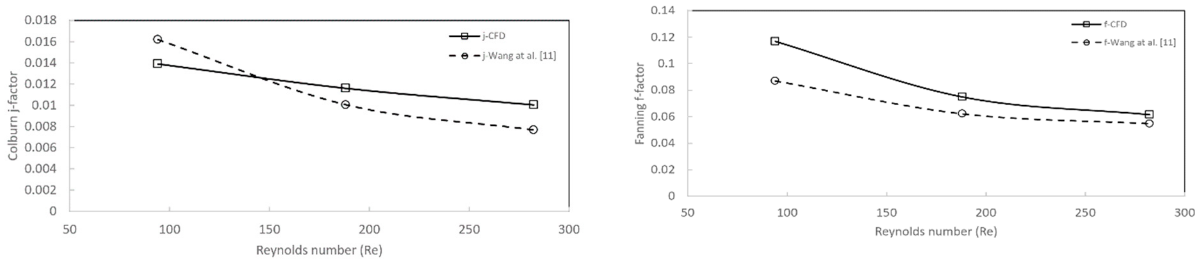


Fig.5. Comparison of performance parameters of varying Reynolds number. (a) Colburn j-factor, (b) Fanning f-friction factor.

In addition, three operating conditions for simulation and validation are listed in Table 3, including the changes of air inlet velocity and refrigerant inlet temperatures. Refrigerant outlet temperature is a significant parameter to evaluate the performance of gas cooler and facilitate controls for its associated system. It can be seen from Figure 6, refrigerant temperature drops dramatically from the first pipe to the eighteenth pipe, which are all the third-row pipes from the airflow direction. And then the refrigerant temperature changes marginally. However, when the CO_2 flows from the pipe number 24 to pipe number 36, its temperature increases and then turns to decrease after pipe number 36. This phenomenon may be caused by the sudden airflow temperature changes from pipe 35 to pipe 36. Under the condition of only air inlet velocity being variable, the higher velocity leads to lower the refrigerant temperature. This is because, higher velocity can improve the heat transfer coefficient and thus heat transfer rate. Compared to the numerical simulation results by Ge and Cropper [8] for refrigerant outlet temperature, when the air velocity is the lowest at 1m/s, the largest error is realised. However, with the increase of air inlet velocity, less errors are found. When the velocity reaches to the maximum at 3 m/s, the temperature discrepancy is about ±2 K, as listed in Table 3.

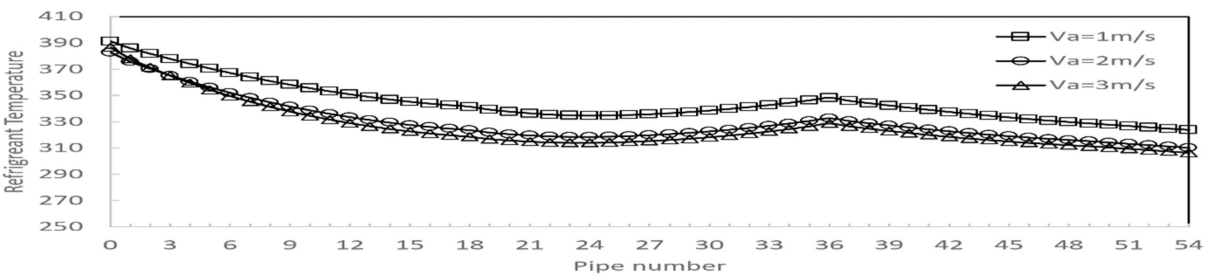


Fig.6. Comparison of modelling results of varying air inlet velocity

Table 3. Modelling conditions.

Test condition	Air velocity (m/s)	Air inlet temperature (k)	Refrigerant inlet mass flow rate (kg/s)	Refrigerant inlet pressure (MPa)	Refrigerant inlet temperature (K)	CFD Simulated Refrigerant outlet temperature(K)	Refrigerant outlet temperature (K)
1	1	302.55	0.038	9	391.25	323.69	311.15
2	2	302.55	0.038	9	382.65	310.3	306.65
3	3	302.55	0.038	9	386.65	306.37	304.65

4. Conclusion and Further Work

A 3D CFD model of finned-tube CO_2 gas cooler has been developed. Although some researchers have carried out different modelling investigations on the heat transfer performance of finned-tube gas coolers, the modelling method utilised in this study is new. This method can accurately obtain the local heat transfer coefficients of airside directly by CFD, which is difficult to predict accurately by empirical correlations. The calculated airside heat transfer coefficient is then imported into the complete heat exchanger model by means of user defined function (udf) in CFD. The model simulation results show that the approach temperature decreases with higher airflow inlet velocity. Although the simulation results have been validated with published literature and showed reasonable agreement, this CFD model still needs to be further improved in terms of the calculation of air side heat transfer coefficients at different air velocities. This will thus enhance the model's palpability. Effects of geometric parameters including tube diameter, tube row, tube pitch, tube arrangement, fin pitch, fin thickness as well as pipe circuit arrangement will be investigated in the future work. This will lead to the optimal coil designs.

Acknowledgements

The authors would like to acknowledge the support received from GEA Searle and Research Councils UK (RCUK) for this project.

References

[1] Gnielinski V. New Equations for Heat and Mass Transfer in Turbulent Pipe and Channel Flow. International Chemical Engineering.1976;16(2):359-368.
[2] Pitla S, Groll E, Ramadhyani S. New correlation to predict the heat transfer coefficient during in-tube cooling of turbulent supercritical CO2. International Journal of Refrigeration. 2002; 25(7):887-895.
[3] Sparrow E, Samie F. Heat transfer and pressure drop results for one- and two-row arrays of finned tubes. International Journal of Heat and Mass Transfer. 1985; 28(12):2247-2259.
[4] Wang C, Jang J, Chiou N. Technical Note A heat transfer and friction correlation for wavy fin-and-tube heat exchangers. International Journal of Heat and Mass Transfer.1999; 42(10):1919-1924.
[5] Kim MH, Pettersen J, Bullard CW. Fundamental process and system design issues in CO2 vapor compression systems. Progress in Energy and Combustion Science; 2004; 30:119-174.
[6] Domanski PA. EVSIM-An evaporator simulation model accounting for refrigerant and one-dimensional air, NISTIR-89-4133. Washington, DC:NIST,1989.
[7] Chang Y, Kim M. Modeling and Performance Simulation of a Gas Cooler for a CO2 Heat Pump System. HVAC&R Research. 2007; 13(3):445-456.
[8] 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.
[9] Gupta D, Dasgupta M. Simulation and performance optimization of finned tube gas cooler for trans-critical CO2 refrigeration system in Indian context. International Journal of Refrigeration. 2014; 38:153-167.
[10] Yaïci W, Ghorab M, Entchev E. 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.2016; 101:527-541.
[11] Wang CC, Lee WS, Sheu WJ. A comparative study of compact enhanced fin-and-tube heat exchangers, Int. J. of Heat Mass Transfer .2001;44:3565-3573.