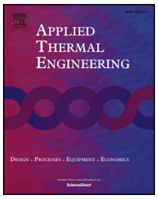
Applied Thermal Engineering 146 (2019) 356–363

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng



Research Paper

Numerical and experimental study on the influence of top bypass flow on the performance of plate fin heat exchanger

Huikun Cai[*], Lijun Su, Yidai Liao, Zeju Weng

Department of Mechanical and Electronic Engineering of Xiamen University, Room 263 of Aerospace Engineering Building, 4221-134 Xiangan South Road, Xiangan South District, Xiamen, Fujian 361102, China

HIGHLIGHTS

- Top bypass flow has great influence on thermal performance of PFHE.
- Experiments agree with simulations well with an average difference of 13.5%.
- A relationship between top bypass duct and its optimized air velocity is gained.
- A ratio of 0.5 for frontal fin to heat exchanger area is suggested.

ARTICLE INFO

Keywords:
Plate-fin heat exchanger
Top bypass flow
Thermal performance analysis
Height ratio of top bypass flow duct to fin
Optimized frontal area ratio

ABSTRACT

Bypass flow deserves a deep research as it is the important phenomenon in plate-fin heat exchanger and is of great influence on the thermal performance. Present studies had consulted this problem, but did not conduct a throughout study and restricted in a limited geometry. Therefore, in this paper an experimental bench is built for the research. Numerical simulations are also carried out on the models with and without bypass flow. The discrepancy of average heat transfer coefficient in simulation with bypass flow increases with air velocity and the maximum difference reaches 82.76% at the velocity of 10 m/s when compared to the model without that, but reduces to be 19.1% when compared to experimental results. After experimental validation, top bypass flow is analyzed with different ratios of top duct height to fin height and varied air velocities. A relationship between air velocity and its minimum height ratio is gained for the estimation in initial design of plate-fin heat exchanger. The optimization with varied fin numbers and interval distances is also carried out, and it is found that the optimized ratio of frontal fin area to frontal plate-fin heat exchanger area is almost keeping at an approximate value of 0.5 for all cases, but thinner fin and narrower air flowing channel can make better heat transfer effect. This paper reveals the influence of top bypass flow on thermal performance and design principle of plate-fin heat exchanger, which is hopeful to be an instruction for its future application in many fields.

1. Introduction

As a compact but high-efficiency component, plate-fin heat exchanger (PFHE) has been widely applied in chemical engineering, aerospace, electronics, power engineering and other fields, and is well beneficial to energy utilization, waste heat recovery, resource saving and cost reduction [1–3]. As a result, researchers have paid great attention to the developments on design theory, experimental test, manufacturing technique and newly application of PFHE, which will thereby make improvements of relevant fields mentioned above.

For PFHE, Kays et al. [4] did an extensive research on fifty-six types of the fins and gained their heat transfer and flow resistance curves and relations. Gupta et al. [5] focused on heat loss problem in a crossing-flow PFHE due to low temperature and conduction in vertical direction. Sahin et al. [6] analyzed the effect of flowing structure on time-average and phase-average turbulence characteristic. Zhang et al. [7] built a distributed-parameter model to optimize the PFHE based on minimum entropy generation principle. Liu et al. [8] designed a new channel structure and using Fluent analysis they found that heat exchange efficiency of improved channels could be enlarged by enhancing internal turbulent flow when compared with traditional channels. Zhe et al. [9] experimentally investigated single-phase and two-phase flow distribution in PFHE and found that nonuniform distribution of fluid flow could affect system thermal performance. Cai et al. [10] demonstrated the

---

[*] Corresponding author.
E-mail address: caihuikun@xmu.edu.cn (H. Cai).

https://doi.org/10.1016/j.applthermaleng.2018.10.007
Received 8 August 2018; Received in revised form 30 September 2018; Accepted 2 October 2018

performance of louver, straight, wavy and serrated fins at various altitudes by simulations and experiments, and found that louver fin was of the strongest cooling capacity while serrated fin was of the best adaptability to various altitudes.

According to these open-reported literatures, it can be observed that the design and performance analyses of PFHE have been treated analytically, numerically and experimentally and gained amount of achievements, but there are still many problems left and new challenges raised. Among of these challenges, bypass flow deserves a deep research as it is the important phenomenon in PFHE and is of great influence on the thermal performance.

The bypass flow phenomenon is always induced by a fan which distributes air through the parallel channels formed by plates fins and the baseplate as well as around the heat sink in the flow bypass region. That is that bypass flow is the possibility that the forced air can flow bypass the heat exchanger core entirely, and leak from the core to the bypass duct. Knight [11], Lee [12] and Teertstra [13] had studied particularly about the bypass flow, and accounted not only for flow by-pass (side bypass), but also for that part of the flow that enters the heat sink and exits through the top (top bypass). Furthermore, Chapman et al. [14] carried out a comparative thermal performance evaluation using aluminum heat sinks made with extruded fins, cross-cut rectangular pins, and elliptical shaped pins in environments characterized by a low air flow. The overall thermal resistance of a straight fin was lower than those of the other two designs owing mainly to the combined effect of enhanced lateral conduction along the fins and lower flow bypass characteristics where the heat source was localized at the center of the heat sink base plate. Butterbaugh et al. [15] investigated the effects of tip and lateral bypass on a heat sink with small fin spacing using compact modelling. Instead of balancing total energy (kinetic and pressure) of the fluid, they only balanced the pressure drop associated with the heat sink and bypass area in their iterative procedure to calculate channel velocity. The predictions of their flow model were within 10% of most of their experimental data. Hans et al. [16] developed an empirical bypass correlation for different fin designs. The Reynolds number and duct height had the largest influence on the prediction of Nusselt number while all investigated parameters (including Reynolds number, duct height, duct width, fin height, fin thickness and fin-to-fin distance) were important for dimensionless pressure drop. Dogruoz et al. [17] created a “two-branch by-pass model”, in which a one-dimensional difference approach was used to model the fluid flow through the heat sink and its top by-pass duct. It was shown that there was a good agreement between the temperature predictions based on the model and experimental data at high approach velocities for tall heat sinks, however the discrepancy increased as the approach velocity and heat sink height decreased. Rakib et al. [18] built an analytical model for predicting air flow and pressure drop across the heat sink by applying conservation of mass and momentum over the bypass regions and in the flow channels established between the fins of the heat sink. The simulated pressure drop varied significantly with bypass configurations (in terms of height, width and flowing velocity), and was found in good agreement with the experimental data. Villafane et al. [19] used numerical flow analyses to guide the design of the complete wind tunnel and analyze the reproduction of the bypass flow structure. Their study would lead to improvements in the thermal management of future aircraft power plants, particularly by taking benefit of the cold bypass air to refrigerate the lubrication oil, without penalizing the propulsive efficiency. Sangkeun et al. [20] conducted an efficient numerical procedure for the installation study of a cooler having a bypass duct to study the influence of an air–oil heat exchanger location and orientation on engine performance, and important design variables were clearly identified.

From these studies, bypass flow has been proved to affect thermal and flow characteristics of the PFHE remarkably, but there are still many problems needed solutions by numerical and experimental investigations. One of the problems is that how to design a suitable bypass flow duct. For a suitable bypass flow duct, it firstly should be a duct for fully developed flow and consequently induce no additional contraction or expansion effect which can affect flow characteristics. Secondly, its dimensions should meet the requirements of compactness and lightweight. It should not be a big one which will enlarge system size and space greatly. And if for the determined bypass flow duct or system dimension, the coming problem is altered to be seeking the largest air velocity which is fit to these requirements. It can be concluded that there is a relationship between air velocity and its minimum bypass flow duct size. The present studies had consulted this problem, but they did not conduct a throughout study and only restricted in a limited geometry (a height range from 0.33 to 1 times of fin height for Hans [16], and a range from 1 to 2 times of heat sink height and width for Rakib [18]). Therefore, this paper will focus on the influence of bypass flow duct on thermal performance of the PFHE and a throughout study of the optimized ratio of bypass flow duct height to fin height for different air velocities. The numerical simulations and experimental tests are both carried out and their results are found in good agreement. After the validation, the PFHE is optimized with varied fin numbers and interval distances. This paper aims to reveal the influence of top bypass flow on PFHE performance and design principle of optimized PFHE, which is hopeful to be an instruction for future PFHE application in many fields.

2. Material and methods

Since the whole radiator is of a large size (referred to our work [21]), the PFHE presented here is just one part of the radiator due to calculation resource reduction and convenient establishment of experimental bench. In addition, its bottom surface is attached to another component closely, there is only top bypass duct left in flowing region so that the influence of top bypass flow is considered whereas side bypass flow is neglected in this work. The particular dimensions of PFHE are shown in Table 1. Subjected to limited lab conditions, an experimental model with a fixed fin number and thickness is applied in Section 2 and 3, whereas an actual model with different fin parameters is developed in Section 4 for a throughout study using the practical working conditions after experimental validation.

Table 1
PFHE analysis parameters and boundary conditions.

Material properties	Value	
	Experimental model	Actual model
Coolant fluid	Air	Air
Coolant viscosity (Pa s)	0.0000155	0.0000189
Coolant conductivity (W m^{-1} K^{-1})	0.0263	0.0280
Coolant specific heat at 298 K (J kg^{-1} K^{-1})	1005	1005
Coolant density (kg m^{-3})	1.768	1.110
Prandtl number *Pr*	0.702	0.699
PFHE material	Aluminum alloy	
PFHE conductivity (W m^{-1} K^{-1})	180	
PFHE specific heat (J kg^{-1} K^{-1})	871	
PFHE density (kg m^{-3})	2690	
Dimensions		
PFHE width *W* (mm)	100	100
PFHE length *L* (mm)	150	132
PFHE height *H* (mm)	45	49
Fin height *h* (mm)	35	35
Baseplate thickness *H-h* (mm)	10	14
Fin thickness *t* (mm)	2	1/1.5/2/2.5
Fin number *N*	18	18–52
Boundary conditions		
Coolant inlet velocity (m s^{-1})	1–10	50
Coolant inlet temperature (K)	298	327
Coolant outlet pressure (Pa)	0	0
Uniform heat flux (W m^{-2})	3333	–
Uniform surface temperature (K)	–	413