Applied Thermal Engineering 65 (2014) 85–93

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Experimental study of convective heat transfer from in-line cam shaped tube bank in crossflow

Arash Mirabdolah Lavasani [a,*], Hamidreza Bayat [b], Taher Maarefdoost [b]

[a] Department of Mechanical Engineering, Central Tehran Branch, Islamic Azad University, Tehran, Iran
[b] Young Researchers and Elite Club, Central Tehran Branch, Islamic Azad University, Tehran, Iran

HIGHLIGHTS

- Heat transfer from cam shaped-tube bank has been studied experimentally.
- Tubes were mounted in an in-line arrangement with two longitudinal pitch ratios 1.5 and 2.
- Pressure distribution and drag coefficient and heat transfer of each tube in tube bank were measured.
- Lower pressure drop compared with circular tube bank.
- Higher thermal-hydraulic performance of cam-shaped tubes compared to circular tube.

ARTICLE INFO

Article history:
Received 13 August 2013
Accepted 30 December 2013
Available online 8 January 2014

Keywords:
Experimental
Cam shaped
Tube bank
Heat transfer
Crossflow

ABSTRACT

Heat transfer from cam-shaped tube bank has been studied experimentally. Tubes were mounted in an in-line arrangement with two longitudinal pitch ratios 1.5 and 2. Reynolds number based on equivalent diameter varied in range of $27{,}000 \leq Re_D \leq 42{,}500$. Drag coefficient and heat transfer from tubes in second row were obtained. Results show that drag coefficient and heat transfer of cam shaped tubes depend on position of tubes in tube bank. Tubes that located at first and second column have the maximum and minimum value of drag coefficient, respectively. In addition, Heat transfer from subsequent tubes is greater than tube in first column. Thermal hydraulic performance of cam shape tube is about 6 times greater than circular tube with same equivalent diameter.

1. Introduction

Study of viscous flow past single and multiple bluff bodies has wide engineering applications of which it can be mention the followings: heat exchangers, cooling towers, oil and gas pipelines, electronic cooling and so on. There are several aspects in understanding the detail of fluid–structure interaction with these structures. Such as, avoiding structural failure which could be caused by flow induced vibration under several conditions. Decreasing pressure drop and drag coefficient of cylinders and enhancing convective heat transfer to or from cylinders surface. Kays and London [1], Horner [2], Zukauskas and Ziugzda [3], Zukauskas and Ulinskas [4], and Zdravkovich [5,6] published books about flow and heat transfer characteristic around bluff bodies.

The difficulty of predicting flow around multiple cylinders is increased when they are placed in proximity to each other. Flow characteristics past each cylinder is affected by its neighbors via wake interaction which results in alternation of overall flow pattern. Many researchers have been motivated by this and have immersed themselves in studying wake interaction of the two cylinders with equal diameter in crossflow as a basic wake interaction model. Deng et al. [7] studied 3-D transition in wake of flow passing around two circular cylinders in tandem arrangement. Their spacing ratio L/D varied in range of 1.5–8 and their Reynolds number was 220–270. Their results show that flow around two tandem circular cylinders can be treated as 2-D system for $L/D \geq 3.5$, whereas 2-D treatment will be invalid for $L/D \geq 4$. Mahir and Altac [8] numerically investigated unsteady laminar convective heat transfer from two isothermal cylinders in tandem arrangement. Their Reynolds numbers were 100 and 200 and their center-to-center ratios, L/D, varied from 2 to 10. They found that mean Nusselt number of the upstream cylinder approaches to that of single cylinder for $L/D \geq 4$. Sumner [9] summarized the literature

* Corresponding author. Tel.: +98 9381299907.
E-mail addresses: arashlavasani@iauctb.ac.ir, arashlavasani@yahoo.com (A. Mirabdolah Lavasani).

1359-4311/$ – see front matter
http://dx.doi.org/10.1016/j.applthermaleng.2013.12.078

on the flow around two circular cylinders with equal diameter immersed in a steady cross-flow.

Rocha et al. [10] studied numerically elliptical and circular section in one and two row tubes and plate fin heat exchanger. They reported that elliptic tubes and plate fin heat exchangers have considerably better overall performance than circular one due to lower pressure drop and higher fin efficiency of elliptic tubes and plate fin.

Matos et al. [11] studied numerically forced convection of staggered circular and elliptic tubes. Their Reynolds number varied in range of $300 \le Re_L \le 800$. They found that elliptic configuration performs more efficiently than circular one. Bouris et al. [12] studied numerically alternate tube configurations for particle deposition rate reduction in tube bundle. Their results show that elliptic-shaped tubes had the lowest fouling rates and pressure drop compared with circular tubes. Matos et al. [13] numerically and experimentally optimized heat transfer from heat exchangers with finned circular and elliptic tubes. Their results show that by using optimal elliptic arrangement, heat transfer gains of up to 19% relative to the optimal circular tube arrangement.

Ibrahim and Gomaa [14] studied experimentally and numerically thermo-fluid characteristics of elliptical tube bank in cross-flow. Their Reynolds number varied in range of $5600 < Re < 40{,}000$ and angle of attack varied from 0° to 150°. They found that elliptic tube bank at zero angle of attack has maximum thermal performance, and using elliptic tube arrangement in heat exchangers could lead to energy conservation.

Lam et al. [15] studied effects of wavy cylindrical tubes in a staggered tube bundle. They used experimental measurement and large eddy simulation technique. Their Reynolds number varied in range of $6800 < Re < 13{,}400$. Their results showed that by using wavy tube drag coefficient was reduced and the fluctuating lift was suppressed.

Nouri-Borujerdi and Lavasani [16] experimentally studied convective heat transfer from a cam shaped tube in crossflow. Their Reynolds number and angle of attack varied in ranges of $0° < \alpha < 180°$ and $15{,}000 < Re < 27{,}000$, respectively. Their results show that thermal performance of cam shaped tube is maximum at $\alpha = 0$ and cam shaped tube had larger thermal hydraulic performance compared to circular tube.

Tang et al. [17] experimentally and numerically studied air-side heat transfer and friction factor of five type of fin-and-tube heat exchanger in Reynolds number in range of 4000–10,000. Their results show that by using vortex generator with higher angle of attack and smaller height, overall performance of heat exchanger will be better.

Næss [18] studied heat transfer and pressure drop of ten finned tube bundle using serrated fins experimentally. He found that increasing the fin pitch reduced the heat transfer coefficient and increasing fin height resulted in the increase of the heat transfer coefficient. Ehwany et al. [19] worked on an elbow-bend heat exchanger as a heater and cooler in an alpha-type Stirling engine. Their results showed that elbow-bend heat exchanger reduces the hydraulic losses. Moawed [20] studied experimentally forced convention from outside surfaces of helical coiled tube. He studied ten helical coiled-tubes with various design parameters and Reynolds number in range of 6.6×10^2–2.3×10^3. He found that with small values of pitch ratio, higher average Nusselt number can be achieved.

Kukulka et al. [21] investigated the overall thermal performance of enhanced heat transfer tubes. Their results show that their tube minimizes the fouling rate and provides heat transfer performance gains in excess of 100% compared to smooth tubes. Simo Tala et al. [22] performed unsteady-RANS simulation to investigated effects of tube pattern on thermal-hydraulic characteristics in a two-row finned-tube heat exchanger. Their results indicate that iso-sectional tube increases thermal-hydraulic performance of heat exchanger compared to classical finned-tube heat exchangers. Tan et al. [23] studied experimentally heat transfer and pressure drop performance of twisted oval tube heat exchanger. Their results show that heat transfer coefficient of the twisted tube is higher than smooth round tube at the cost of some incensement of pressure drop.

Lee et al. [24] studied numerically the effects of uneven longitudinal pitch on heat transfer from in-line tube bank. It can be found from their results that increasing the longitudinal space for uniformly distributed cylinders enhanced overall heat transfer. Bilirgen et al. [25] numerically study effects of fin spacing, fin thickness, and fin material on overall heat transfer of annular finned tubes. Their results show that increasing fin thickness leads to minor increases in heat transfer and modest increases in pressure drop. Increasing thermal conductivity of material is also leads to higher heat transfer. Sun and Zhang [26] used CFD model to investigate simultaneously fluid flow and heat transfer on both air side and water side of elliptical tube in finned-tube heat exchangers. Their results show that impact of axis ratio on overall thermal-hydraulic performance is dependent on air velocity and water volumetric flow rate. At lower volumetric flow rate increasing axis ratio is advantageous to overall thermal-hydraulic performance, while it has adverse impact on higher volumetric flow rate.

Streamlined shaped tubes due to low hydraulic resistance need less pumping power. So, the purpose of this study is to experimentally investigate the flow and heat transfer characteristics around cam shaped tube bank subject to crossflow of air.

2. Experimental setup

Fig. 1 shows the cross section profile of the cam shaped tube that comprised some parts of two circles with two arcs segments tangent to them. The cylinder have identical diameters are equal to $d = 8$ mm and $D = 16$ mm where distance between their centers is $l = 15.75$ mm. Tubes are made of a commercial steel plate with 0.7 mm of wall thickness.

For measuring drag coefficient of cam shaped tube in tube bank a test tube with length of 31 cm was made. Fourteen holes (1 mm diameter) were drilled on the surface of test tube to measure the static pressure on the tube surface by a digital differential pressure meter. For measuring heat transfer, four test tubes with length of 22 cm were made. The two ends of test tubes were insulated for decreasing heat transfer from these surfaces. In order to neutralize the effects that sudden deformation of tube could cause on the passing flow, the insulation was formed like the cam shaped tube. By doing this, the two end's surfaces of each tube will be insulated without any kind of effects that sudden deformation could cause on passing flow through tube bank.

Fig. 2 shows sixteen cam-shaped tubes located at wind tunnel test section. The distance between the upper and lower tubes to the

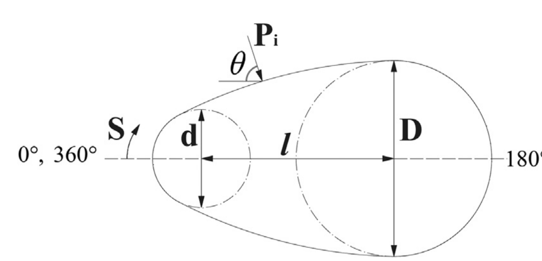

Fig. 1. Schematic of a cam shape tube.