International Journal of Thermal Sciences 96 (2015) 45–69

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Effect of nozzle shape on jet impingement heat transfer from a circular cylinder

Dushyant Singh, B. Premachandran*, Sangeeta Kohli

Department of Mechanical Engineering, Indian Institute of Technology Delhi, Hauz Khas, New Delhi 110 016, India

A R T I C L E I N F O

Article history:
Received 30 August 2014
Received in revised form
17 April 2015
Accepted 17 April 2015
Available online 22 May 2015

Keywords:
Jet impingement
Nozzle shape
Circular cylinder
Heat transfer

A B S T R A C T

Experimental and numerical investigations were carried out to study the effect of nozzle shape on unconfined jet impingement heat transfer from a heated circular cylinder. Air was considered as the working fluid. The heated cylinder surface was maintained at a constant heat flux. In this work, circular, square and rectangular nozzles of equal hydraulic diameters were selected for a comparative study. The Reynolds number, Re_{hyd} defined based on the hydraulic diameter of the nozzle was varied from 10,000 to 25,000. The ratio of hydraulic diameter of the nozzle to the diameter of heated cylinder, d_{hyd}/D was maintained at 0.2 for the parametric study. The non-dimensional distance between the nozzle exit and the cylinder, h/d_{hyd} was varied from 4 to 16. For a fixed jet Reynolds number, the mass flow rates through different nozzles are different. Hence, a parametric study was also carried out to know the effect of nozzle shape for various fixed mass flow rates. For this, a modified Reynolds number $\overline{Re}_{hyd}$ was kept constant for all nozzle shapes. For a fixed jet Reynolds number, heat transfer from the cylinder was higher for the case of rectangular nozzle. On the other hand, for a fixed modified Reynolds number $\overline{Re}_{hyd}$, heat transfer rate from the cylinder was found to be higher when the circular nozzle was used. For the same geometry, numerical simulations were also carried out using three different two-equation turbulence models. Using the experimental data, two correlations for stagnation Nusselt number $Nu_{stag.}$ have also been provided.

1. Introduction

Jet impingement heating and cooling are widely used in many engineering applications to achieve very high heat transfer rate from a target surface. Notable applications of jet impingement heat transfer are cooling of electronic components, processing of materials, cooling of turbine blades, anti-icing system of aircraft wings and processing of food. Martin [1], Jambunathan et al. [2] and Viskanta [3] reviewed jet impingement heat transfer from flat surfaces. Recently, Zuckerman and Lior [4] described in detail the physics of jet impingement flows and presented an extensive review of the correlations available in the literature.

Many researchers have investigated the effect of nozzle shape on heat transfer rate from a flat target surface. The nozzles could be in the form of an orifice (Fig. 1(a) and (b)) or long pipe or channel type nozzles (Fig. 1(c)) with different cross-sectional shapes. The experimental investigations in the literature on a variety of nozzle shapes cover a wide range of Reynolds number (based on the hydraulic diameter) from 5000 to 60,000; with the ratio of spacing to hydraulic diameter (h/d) varying from 2 to 12 [5–14].

For orifice type nozzles, some researchers have studied the effect of the different contours of the orifice as shown in Fig. 1(b) [5–8] or the shape of the orifice such as circular, rectangular or elliptical [9–13]. Obot et al. [5] considered both sharp edged and contoured nozzles, for Re_d ranging from 15,000 to 60,000. They found considerable effect of nozzle shape on heat transfer for h/d less than 6, but this effect was insignificant for $h/d > 12$. Colucci and Viskanta [6] conducted experiments with a sharp-edged orifice and two hyperbolic nozzles using a thermochromatic liquid crystal technique to get the temperature distribution over the impinging surface. They reported that the stagnation Nusselt number with hyperbolic shaped nozzles performed was 25–36 % higher than that with orifice sharp-edged. Lee and Lee [7] investigated the effect of square-edged, standard-edged and sharp-edged orifices on jet impingement heat transfer from a flat surface and found that the sharp-edged orifice resulted in the highest value of, the sharp-edged orifice type nozzle resulted in the highest value of in the stagnation Nusselt number. Brignoni and Garimella [8] found that

* Corresponding author.
E-mail address: prem@mech.iitd.ac.in (B. Premachandran).

http://dx.doi.org/10.1016/j.ijthermalsci.2015.04.011
1290-0729/

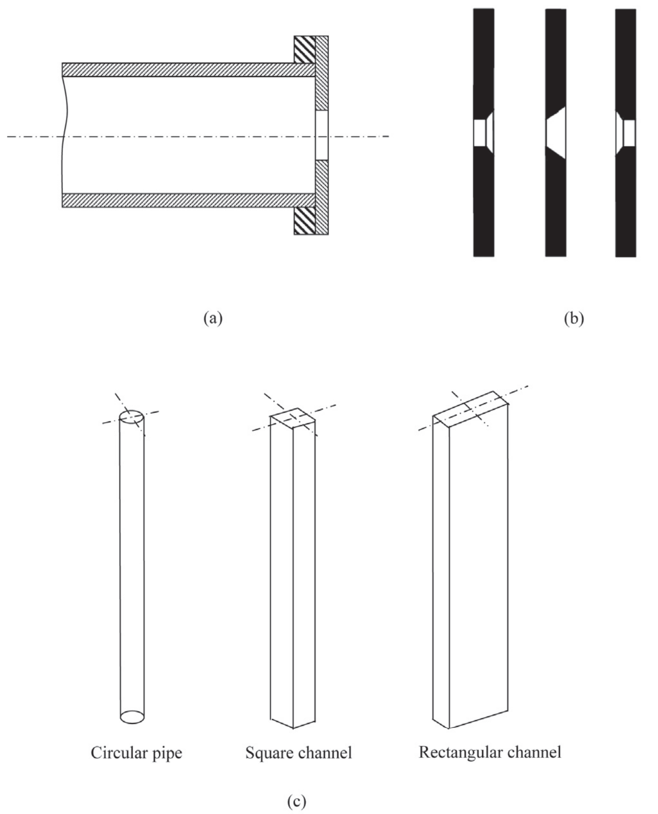

Fig. 1. Types of nozzles used in the jet impingment studies (a) an orifice type nozzle (b) Types of orifices (c) long pipe or channel type nozzles.

chamfering the nozzle increased the ratio of average heat transfer coefficient to the pressure drop by 30.8% as compared to an unchamfered nozzle. Lee et al. [9] observed that the stagnation Nusselt number was higher in the case of elliptic jet compared to a circular jet. Lee and Lee [10] worked with elliptical orifices of five different aspect ratios and found the stagnation Nusselt number to be highest for the aspect ratio of 4 and h/d = 2, with 15% increases as compared to a circular jet. Zhou and Lee [11] studied the effect of turbulence intensity on heat transfer from a flat surface due to impingement of a jet from a sharp-edged rectangular nozzle. Koseoglu and Baskaya [12] carried out both experimental and numerical investigations to compare the effect of different orifice shapes viz. circular, elliptical and rectangular on jet impingement cooling of a flat surface at fixed mass flow rate for all shapes. They found reduction in average heat transfer with increase in aspect ratio or increase in cross-sectional area of the jet. Zhao et al. [13] numerically studied the flow and heat transfer characteristics of jet impingement on a flat surface. with circular, square, rectangular and elliptical nozzles with same Reynolds number. It was observed that the non-circular jets can provide higher heat transfer coefficient than the circular jets for $h/d_{hyd} = 2$ to 4, but circular nozzle was found to perform better for $h/d_{hyd} = 5$ to 6.

Gulati et al. [14] experimentally studied jet impingement heat transfer from a flat surface using long piped nozzles with circular, square and rectangular cross sections. They observed that the local Nusselt number on stagnation point is 10% higher for the rectangular nozzle as compared to square and circular nozzles for $Re_{hyd} = 5000$ to 15000 at $h/d_{hyd} = 0.5$. However, the average Nusselt number calculated over 60 mm × 60 mm area was almost the same for all the nozzle configurations tested.

The studies on jet impingement heat transfer from circular cylinders are much less as compared to those for flat target surfaces. These studies include slot jets [15–18] and circular jets [19–25] investigating the effect of parameters such as Reynolds number, non-dimensional distance between the nozzle exit to the cylinder surface and the ratio of the diameters of the jet and the target cylinder. However, no study is reported on the effect of nozzle shape on the heat transfer from a cylindrical target surface. Considering the important role that the nozzle shape has shown to play in case of flat targets for certain set of parameters, extending these studies to cylindrical target surface can help in providing practically useful information for applications involving such surfaces. In addition, such a study acquires significance considering the fact that the flow characteristics over a cylindrical target surface can be quite different from those over a flat target surface. The present work, therefore, focuses on experimental as well as numerical investigation of the effect of three nozzle shapes, viz., circular, square and rectangular on jet impingement heat transfer from a circular cylinder subjected to a constant wall heat flux. Extensive experimentation has been done by varying the cooling jet flow rate and the distance between the nozzle and the target. Numerical simulations have been carried out for selected cases to check how different two-equations turbulence models perform for different nozzle shapes.

2. The physical problem

Fig. 2 shows the schematic of the physical problem addressed in this work. The air jet emerging from the nozzle and impinging on the circular cylinder is unconfined. Long pipes of circular, square or rectangular cross-section with hydraulic diameter, d_{hyd} are used as nozzles, their dimensions being so chosen that for all the three cross-sectional shapes, the d_{hyd} was the same (Table 1). This meant that the actual cross-sectional area of each nozzle is different, with the circular nozzle having the smallest cross-sectional area followed by the square and the rectangular nozzles. Thus for a given average velocity of flow through each nozzle, the fluid mass flow rate would be the highest for the rectangular nozzle and smallest for the circular nozzle.

Considering the above, the nozzle shapes of the same hydraulic diameter can be compared in two ways (i) by operating them with the same average velocity of flow through the nozzles or (ii) by operating them with the same fluid mass flow rate through the nozzles. It is important to note that in most of the literature on flat target surfaces, the comparison of nozzle shapes is based on same average velocity of flow, since this also corresponds to maintaining the same Reynolds number based on the hydraulic diameter $\left(Re_{hyd} = \frac{\rho V_{avg} d_{hyd}}{\mu} = \frac{\dot{m} d_{hyd}}{A_n \mu}\right)$ for all the nozzle shapes. Only, Koseoglu and Baskaya [12] carried out the comparison of different nozzle shapes for a flat target by keeping mass flow rate and the area of the nozzles same for all the shapes, which would mean that the average velocity of flow would be the same but hydraulic diameter of the different shapes would be different. In the present work, the comparison has been carried out for both the conditions (i) and (ii) discussed above. If the mass flow rate of the fluid is same for all the nozzle shapes having the same hydraulic diameter, it amounts to maintaining can be considered as a modified the non-dimensional number $\frac{4\dot{m}}{\pi d_{hyd} \mu}$ constant. This also can be considered as a modified Reynolds number $\overline{Re}_{hyd}$. For circular nozzles, both Re_{hyd} and $\overline{Re}_{hyd}$ will have the same value for a given flow rate and nozzle diameter. However for non-circular nozzles, the will have different values even for a given flow rate.