International Journal of Thermal Sciences 108 (2016) 174–184

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Effects of mist and jet cross-section on heat transfer for a confined air jet impinging on a flat plate

Fifi N.M. Elwekeel [a], [*], Antar M.M. Abdala [b]

[a] Helwan University, Faculty of Industrial Education, Cairo, Egypt
[b] Helwan University, Faculty of Engineering, Matareya Branch, Cairo 11817, Egypt

ARTICLE INFO

Article history:
Received 30 December 2015
Received in revised form
7 May 2016
Accepted 8 May 2016

Keywords:
Air jet
Heat transfer
Mist cooling
Jet cross-section

ABSTRACT

Convective heat transfer to an impinging air jet is known to yield high local and area-average Nusselt numbers. We simulate this heat transfer over a wide range of study parameters, including Reynolds number and mist mass fraction, and three jet shapes: circular, half circuit, and quarter circuit. Simulations show that when compared with air only, mist provides higher heat-transfer enhancements for the first two shapes but is insignificant for the third. The area-average Nusselt number is higher by ~100% for the half-circuit jet than for the circular jet for both single-phase air and mist. With 0.5% mist, the area-average Nusselt number of the circular jet is enhanced by 41% at Reynolds number 10000.

1. Introduction

The impingement of a jet on a plane surface is a flow of interest in many engineering applications. This flow is used extensively in process engineering applications such as annealing of metals, cooling of gas turbine blades, and cooling in laser/plasma cutting processes [1]. In the annealing industry, impinging jets are used to quench products after rolling. In gas turbine engines, impinging jets are applied for cooling of turbine blades/vanes. In laser/plasma cutting processes, the thermal deformation of products can be reduced with jet impingement cooling. Besides the above applications, impinging jets are also adopted to enhance electronic cooling [2,3].

Guerra et al. [4] and Webb and Ma [5] investigated the round impinging jet using air as cooling fluid. They studied the effects of some parameters on heat transfer. These parameters were Reynolds number, the nozzle-to-plate spacing, and the ambient air temperature relative to the jet temperature. Colucci and Viskanta [6] and Brignoni and Garimella [7] studied the effects of orifice geometry on heat transfer in the impinging jet and plate. They found that orifice geometry can improve heat transfer by 20–30% above that of simple square orifice nozzle. The nozzle geometry parameters such as orifice shape and diameters, as well as orifice inlet and outlet shapes, do play a determining role in heat transfer rate, which depends also on the thickness of the nozzle orifice [8–10].

Lee et al. [11] investigated the increase in ellipticity on heat transfer. They found that the local Nusselt number in the impingement region for a nozzle-to-plate spacing of six-jet diameters was approximately equal to the result for a jet exiting a round contoured orifice and 15% larger than for a jet exiting a long round pipe. The local Nusselt number for the major and minor axes of an elliptical jet decreased with radius more rapidly than for jets exiting a round orifice or pipe. The heat transfer produced by the jet in the impingement region is related to the turbulence intensity of the jet and can be increased by adding a grid at the jet exit when the nozzle-to-plate distance is less than six-jet diameters [12,13].

Lou et al. [14] investigated the effects of nozzle and nozzle-to-plate spacing on impinging heat transfer. According to their results, the decreasing in nozzle width and nozzle-to-plate spacing leads to enhancing each of heat transfer coefficient and Nusselt number.

Zhou and Lee [15] studied the performance of rectangular jets. This focused on the nozzle-to-plate spacing (1–30) and jet Reynolds number (2715–25005). The results showed a significant influence on heat transfer behaviors of impinging jets.

Puneet et al. [16] conducted experiments of heat transfer to the

---

[*] Corresponding author.
E-mail addresses: fifinew2000@yahoo.com (F.N.M. Elwekeel), antar451@yahoo.com (A.M.M. Abdala).

http://dx.doi.org/10.1016/j.ijthermalsci.2016.05.007
1290-0729/

impinging jet from a flat plate employing three different nozzle cross sections, circular, square, and rectangular, and investigated the effect of nozzle shape. They showed that the heat transfer characteristics of square and circular jets are similar whereas the Nusselt number of the rectangular jet is higher in the stagnation region than those of circular and square jets.

Further, the two-phase impinging jet is of great interest in developing modern turbulence models. Pakhomov and Terekhov [17] developed a model for the theoretical study of heat transfer in two-phase mist-dispersed jets falling normally onto a flat plate. The simulation conditions were droplet sizes of 5–100 μm and a fixed Reynolds number of 26600. The theory applies the Reynolds-averaged Navier–Stokes (RANS) equations for two-phase flow. They found that by droplets addition the heat transfer rate increases several times compared with single-phase air.

Terekhov and Pakhomov [18–21] developed a computational model to study the flow and heat transfer in a laminar mist flow over an isothermal flat plate, and numerically studied the flow dynamics as well as heat transfer in a turbulent two-phase gas-droplet ducted flow. Garbero et al. [22] presented their numerical study of a round impact gas-droplet jet performed using the commercial CFD FLUENT package. The gas phase is described with the RANS method taking into account the coupling between the phases. For the description of droplet dynamics, the Lagrangian approach was applied.

Tan et al. [23] examined heat transfer of a free jet in both oblique and normal impinging jets with mist/air. They found that the heat transfer coefficient in the stagnation region for the oblique impinging jet has a much lower value than that for the normal impinging jet, whereas away from the stagnation region, the local heat transfer coefficient for the oblique jet has a higher value than that for the normal impinging jet.

Li et al. [24–27] made a series of studies on enhancements afforded using a mist/steam impinging jet. They concluded that the heat transfer is enhanced over 200% by the addition of 1.5% mist at the stagnation point. Mist enhancement was found to decline to near zero by five slot-widths downstream. Furthermore, they developed a model for mist/steam jet cooling in which their results followed the experimental data closely. Wang et al. [28] experimentally investigated the mist/steam heat transfer of three rows of circular jet impingement in a confined channel. The results indicated that the cooling enhancement region of the three-row jet impinging jets was more extensive than those employing a single row of circular jets or jet slots. The highest enhanced region spanned about five-jet diameters and became negligible downstream. The maximum local enhancement was up to 800% by injecting 3.5% of mist at a low heat-flux condition and 150% enhancement at a high heat-flux condition. It should be noted that the mist/air flow and mist/steam flow are thermodynamically different. Mist/air is a two-component, two-phase flow where evaporation of the water droplets is determined by the partial pressure of the water vapor in the air and not by the total pressure of the mixture. Mist/steam is a one-component, two-phase flow in which the water droplet evaporation is determined by the steam pressure.

From the literature, we conclude that most studies have focused on the air/mist impinging jet; moreover, there is no information on the effects of jet shape on confined mist jets. In the current study, a simulation study with three different jet cross-sectional shapes was conducted to illustrate some of the heat transfer characteristics that occur when air and an air/mist jet impinge on a flat plate. Two objectives were established. The first is to assess the effects of shape of the jet cross-section on the heat transfer characteristics during jet impingement. The second objective is to assess the enhancement of heat transfer by addition of mist, in particular, heat transfer produced with circular (C), half-circular (H) and quarter-circular (Q) shapes with mist concentrations ranging from 0.2% to 1% by weight. The heat transfer tests reported here were performed for jets with values of the Reynolds number ranging from 10000 to 30000 (based on the hydraulic diameter of circular jet).

2. Turbulence modeling

In the present work, a suitable turbulence model for the problem needs to be chosen first. With the standard k-ε model, the renormalization group (RNG) k-ε model, the standard k-ω model, omega Reynolds stress model (ωRS), and the shear stress transport (SST), each was tested for suitability in simulating single-jet impingement flow and heat transfer.

Launder and Spalding [29] proposed the k-ε model in the standard form. The RNG k-ε model is based on the renormalization group analysis of the Navier–Stokes equations. The RNG model was developed by Yakhot et al. [30], to account for the effects of smaller scales of motion. Wilcox [31] developed the standard k-ω model, which is sensitive to inlet freestream turbulence properties. In the ωRS model, the eddy viscosity approach has been discarded and the Reynolds stresses are directly computed. This modeling approach originates from the work by Launder et al. [32]. The shear stress transport (SST) turbulence model is a combination of the k-epsilon in the free stream and the k-omega models near the walls. The SST two-equation turbulence model was developed by Menter [33] to deal with the strong freestream sensitivity of the k-omega turbulence model and improve pressure-gradient predictions. Also Bardina et al. [34] proved that the SST model predicts more accurately the near wall turbulence that plays a vital role in the accurate prediction of turbulent heat transfer. Garg and Ameri [35] compared the SST, k-ε and k-ω models with experimental data, finding that the SST model resolved the passage vortex better than other two models.

3. Computational setup and methodology

Fig. 1 shows the geometry of the computational domains. Air and air/mist flow through a jet with both length and diameter *d* of 6 mm with uniform velocity profile, vertically impinging on the confined target plate with two side walls positioned at spanwise distances of $y = \pm 31.26$ mm. The jet after impingement was restricted to discharge in two opposite directions parallel to the *x*-axis, and two channel outlets are placed at $x = \pm 125.1$ mm. The inlet jet takes one of three shapes C, H, and Q (Fig. 2).

The flow is assumed to be statistically steady turbulent flow, incompressible, and three-dimensional. The buoyancy and

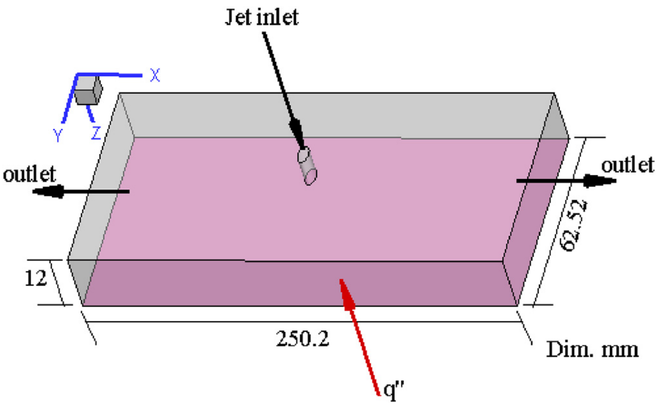

Fig. 1. Geometry of the computational domain.