Applied Thermal Engineering 39 (2012) 163–170

Contents lists available at SciVerse ScienceDirect

Applied Thermal Engineering

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

Numerical characterization of multi-nozzle spray cooling

Yan Hou [a,b], Yujia Tao [a], Xiulan Huai [a,*], Zhixiong Guo [c]

[a] Institute of Engineering Thermophysics, Chinese Academy of Sciences, P.O. Box 2706, Beijing 100190, China
[b] Graduate School of the Chinese Academy of Sciences, Beijing 100049, China
[c] Department of Mechanical and Aerospace Engineering, Rutgers, The State University of New Jersey, 98 Brett Road, Piscataway, NJ 08854, USA

ARTICLE INFO

Article history:
Received 9 October 2011
Accepted 11 January 2012
Available online 20 January 2012

Keywords:
Multi-nozzle
Spray
Two-phase flow
Droplet
Simulation

ABSTRACT

This work aims to study the characteristics of multi-nozzle spray cooling using CFD method based on the fundamentals of air flow and liquid droplet collision dynamics. A mathematical model for the two-phase flow was presented. The simulations were performed using a Eulerian–Lagrangian approach. Focus was placed on revealing the flow behavior with multiple nozzles, the droplet trajectory, and the influencing factors. The predictions by the present simulations matched well with the experimental results available in the literature, with a comparison showing deviation below 10%. It is concluded that the multi-nozzle spray characteristics including the Sauter Mean Diameter (SMD) of droplets and the mass weighted average droplet velocity are influenced by the nozzle inlet pressure, the mass flux, the nozzle-to-surface distance and the number of nozzles. With increase of the inlet pressure, the droplet SMD decreases and the mass weighted average droplet velocity increases. With increase of the mass flux, both the droplet SMD and the mass weighted average droplet velocity increase. The nozzle-to-surface distance is a very sensitive parameter to the droplet velocity distribution. Nevertheless, the droplet velocity distribution is not a monotonic function of the nozzle-to-surface distance. With increasing nozzle number, the change in droplet size is not appreciable; whereas the mass weighted average droplet velocity decreases and the distribution of the droplet size is improved significantly.

1. Introduction

Spray cooling is important in a wide spectrum of industrial applications. It has great potentials in the cooling of high power systems. Traditionally, spray cooling was utilized to cool highly heated surfaces for equipments and processes in metallurgy, chemical and nuclear industry. Recently, it has received increasing attention in the development of modern technologies, such as the cooling of electronic devices and high power solid-state lasers. In the process of liquid spray, the liquid is usually injected into a chamber through an atomizing nozzle, resulting in the production of a spray comprising a large number of liquid droplets. The research of spray cooling involves three major subtopics: liquid atomization, interaction between droplets and surface, and the associated mass and heat transfer. A well-controlled particle size distribution is desirable in liquid atomization because the huge interfacial contact area between liquid and vapor can enhance heat and mass transfer rate and directly determines the cooling performance [1]. Hence, understanding the details of liquid atomization, such as the droplet diameter and velocity profiles, is essential in the design and performance optimization in spray cooling applications.

A number of review papers in spray cooling have addressed important issues focusing on the critical heat flux (CHF) and parametric studies [2–5]. However, little work has been done toward the disclosure of the spray characteristics. Experiments providing comprehensive information with regard to the droplet characteristics close to the injection point are very scarce since it is difficult to perform accurate measurements at locations very close to the atomizing nozzle. The experimental studies of Kurt and Mudawar [6] and Cheng et al. [7] both used the Phase Doppler Anemometry method to test a single nozzle and obtained the Sauter Mean Diameter (SMD) of droplets at different nozzle-to-surface distances. Such experiments are very costly, not to mention that it is not so easy to realize non-intrusive measurement. To this end, numerical simulation is a good means which can predict the spray characteristics and provide a theoretical guidance for experiments. However, numerical simulation of droplet dynamics in a two-phase flow is a particularly challenging problem because of the complicated process of film formation, droplet breakup, collision, coalescence and evaporation.

* Corresponding author. Tel./fax: +86 10 8254 3108.
E-mail address: hxl@mail.etp.ac.cn (X. Huai).

1359-4311/$ – see front matter © 2012 Elsevier Ltd. All rights reserved.
doi:10.1016/j.applthermaleng.2012.01.030