International Journal of Thermal Sciences 125 (2018) 81–88

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Numerical simulation of multi-nozzle spray cooling heat transfer

Yan Hou[a,*], Yujia Tao[b], Xiulan Huai[b,**], Yu Zou[a], Dongliang Sun[a]

[a] Beijing Institute of Petrochemical Technology, Beijing 102617, China
[b] Institute of Engineering Thermophysics, Chinese Academy of Sciences, P.O. Box 2706, Beijing 100190, China

ARTICLE INFO

Keywords:
Numerical simulation
Spray cooling
Heat flux
Heat flux distribution

ABSTRACT

The aim of this paper is to study the spray cooling heat transfer characteristics using CFD method. The two phase mathematical model is established based on Euler-Lagrange approach. In this model, unsteady-state flow conditions are simulated. Heat and mass transfer between the vapor and the water droplet are computed. The averaged error of numerical results is 10% compared to experimental results in our previous papers. It is concluded that the heat flux and its distribution on heated surface are influenced by the heated surface temperature, mass flux, nozzle to surface distance and the number of nozzles. Furthermore, the heat flux distribution do not change obviously with the increasing of heated surface temperature, and the heat flux increases with the increasing of mass flux. Moreover, there is an optimal nozzle to surface distance while the spray impingement zone circumscribes in heated surface, and more nozzles lead to higher heat flux and better heat flux distribution on heated surface. Considering the disadvantage of manufacture difficulty and cost increase for more nozzles, there is an optimal nozzle number in certain space and heat dissipation requirement. The optimal number of nozzles is 8 in this article.

1. Introduction

With the rapid development of miniaturization and integration of electron device, the heat flux increases dramatically. Compared to traditionally cooling methods, spray cooling has advantages of higher heat flux and more uniform temperature distribution. Traditionally, spray cooling was utilized to cool highly heated surfaces for equipments and processes in metallurgy, chemical and nuclear industry. Recently, it has been reported that spray cooling had been applied in the cooling of Cray X1 vector supercomputers.

In the process of spray cooling, the working fluid is forced to pass through an atomizer and broken up into droplets. These droplets impact on the heated surface continually, then a liquid film layer over the heated surface is formed. Several detailed aspects in this process are as follows: droplet ejection, droplet interaction in transit to the heater surface, droplet impact on the liquid film, and vapor bubble growth and departure in the liquid film. As it is difficult to obtain the detailed information of the spray area through experimental study [1–3], the numerical calculation becomes an important method. However, the overall simulation of full spray cooling problem is very complex, so most researchers focus on numerical simulation of single droplet impingement. Wang et al. [4] investigated the heat transfer performance during the process that a single droplet with various velocities and diameters is impacting on the thin liquid film based on VOF method. Selvam et al. [5,6] studied the effect of single bubble growth on heat transfer performance of thin liquid film based on level set method. In the next year, Selvam et al. [7] studied the phenomenon in the micro environment of about 40 μm thick liquid layer with vapor bubble growing due to nucleation based on level set method. Recently, Sarkar et al. [8] studied a vapor bubble growing in thin liquid film and a droplet impacting on the thin liquid film through 3D multiphase flow modeling using multigrid conjugate gradient solver and level set method in parallel computing environment. In these articles, the symmetry boundary is considered on four sides of the domain, simulation results can represent a large spray made up of number of droplets and bubbles with same size. However, the size, velocity and distribution of the droplets have significant effect on the heat transfer in thin liquid film. How to avoid the limitations of this method is an important problem required to be solved.

Recently, some researchers developed numerical simulations of full spray cooling based on fluid dynamic principles and empirical correlation. Issa et al. [9] built numerical model of spray cooling to study the heat transfer at temperature ranging from nucleate to film boiling under atmospheric conditions. The droplets are dispersed stochastically in the continuous gas phase. Droplet dynamics and contact heat transfer are modeled based on empirical correlation that is a function of the droplet

* Corresponding author.
** Corresponding author.
E-mail addresses: houyan@bipt.edu.cn (Y. Hou), ustbiit@163.com (X. Huai).

https://doi.org/10.1016/j.ijthermalsci.2017.11.011
Received 13 November 2015; Received in revised form 28 June 2016; Accepted 15 November 2017

Nomenclature

u_v	fluid velocity (m·s^{-1})
S_m	the mass added to the continuous phase from the dispersed second phase (kg·m^{-3}·s^{-1})
U	fluid velocity vector (m·s^{-1})
F	droplet momentum change in control volume (kg·m·s^{-2})
p	pressure (Pa)
E	kinetic energy(m^2·s^{-2})
k_{eff}	the effective conductivity of continuous phase (w·m^{-1}·k^{-1})
J_j	the diffusion flux of species j (mol·m^{-2}·s^{-1})
T_{ref}	the reference temperature, 298.15 K
$C_{p,\ j}$	heat capacity of species j (J·kg^{-1}·K^{-1})
S_h	volumetric heat sources added to the continuous phase from the dispersed phase (kg·m^{-3}·s^{-1})
u_p	droplet velocity (m·s^{-1})
g	acceleration of gravity (m·s^{-2})
Re	relative Reynolds number
C_D	drag coefficient
d_p	droplet diameter (m)
d_p'	averaged droplet diameter (m)
Δt	time step size (s)
T_p	the droplet temperature (K)
T_∞	local temperature of the continuous phase (K)
h'	convective heat transfer coefficient (W·m^{-2}·K^{-1})
A_p	surface area of the droplet (m^2)
C_p	heat capacity of droplet (J·kg^{-1}·K^{-1})
m_p	mass of the droplet (kg)
k	thermal conductivity of vapor phase (W·m^{-1}·K^{-1})
Pr	Prandtl number of vapor phase
N_{h2o}	molar flux of water vapor (mol·m^{-2}·s^{-1})
$M_{w,h2o}$	molecular weight of species h_2o (kg·mol^{-1})
h_{fg}	latent heat (J·kg^{-1})
$D_{h2o,m}$	diffusion coefficient of vapor in the bulk (m^2·s^{-1})
Sc	the Schmidt number
P_{sat}	the saturated vapor pressure (Pa)
R	universal gas constant
P_{op}	the operating pressure (Pa)
X_{h2o}	the local bulk mole fraction of water vapor
$m_{p,0}$	the initial mass of droplet (kg)
Δm_p	mass change of a droplet as it passes through each control volume (kg)
$m'_{p,0}$	the initial droplet mass flux (kg·s^{-1})
M	the droplet mass change in control volume (kg·s^{-1})
m'_p	mass flux of droplets (kg·s^{-1})
$\overline{m_p}$	average mass of the droplet in control volume (kg)
$C_{p,h2o}$	heat capacity of the volatiles evolved (J·kg^{-1}·K^{-1})
ΔT_p	temperature change of the droplet in control volume (K)
q'	heat flux of heated surface (W·m^{-2})
h	averaged heat transfer coefficient of heated surface (W·m^{-2}·K^{-1})
H	nozzle to surface distance (m)
T_w	temperature of the heated surface (K)

Greek symbols

ρ	density (kg·m^{-3})
η	dynamic viscosity (Pa·s)
τ_{eff}	viscous friction coefficient (N·s·m^{-1})
μ	molecular viscosity of the fluid (kg·m^{-1}·s^{-1})
ε_p	droplet emissivity
σ	Stefan-Boltzmann constant
θ_R	Radiation temperature (K)

Subscripts

v	vapor
p	droplet
x	for x direction
y	for y direction
z	for z direction

Weber number. Cheng et al. [10–12] presented a mathematical model of spray cooling heat transfer characteristics based on empirical correlations of droplet dynamics and heat transfer. The droplet diameter, velocity and spatial distribution are defined according to Gaussian distribution. The droplet-film impaction heat transfer, film-surface convection heat transfer, the heat transfer due to surface nucleation bubbles, heat transfer due to secondary nucleation bubbles, environmental heat transfer are defined based on basic heat transfer theories and empirical correlation. Estes et al. [13,14] developed a correlation of critical heat flux (CHF) along the outer periphery of the impact area of a normal spray. Then Rybicki et al. [15] validated this correlation for upward-oriented PF-5052 sprays as well. Recently, Visaria et al. [16] built a new volumetric flux model which is one of the key parameters governing spray cooling performance and critical heat flux (CHF) based on previous empirical correlation. As the empirical correlations are obtained under specific conditions, the universality of these models based on empirical correlations is poor. How to develop a better model to simulate the full spray cooling process is important in this field.

The objective of the present work is to study the multi-nozzle spray cooling heat transfer characteristics using the computational fluid dynamics (CFD) simulations. A two-phase flow model was developed based on the mixed Euler-Lagrange approach for a three-dimensional spray system including at least two nozzles in a cuboid chamber. The model is verified using experimental results in the literature [17] under same conditions. The effects of heated surface temperature, mass flux, nozzle to surface distance, the number of nozzles on heat flux and heat flux distribution are investigated in this paper.

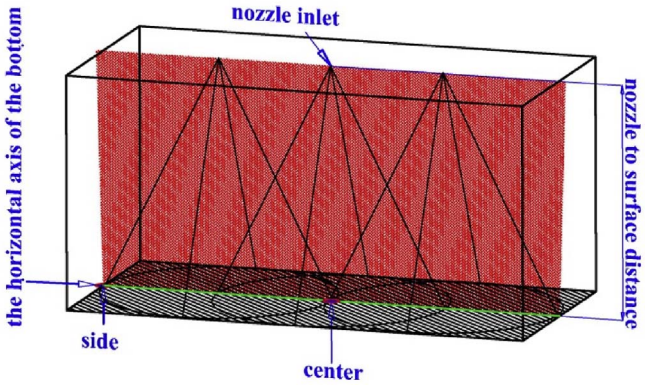

Fig. 1. The sketch for simulation domain.

2. Mathematical model for spray cooling

The fluid phase is treated as a continuum by solving the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of droplets through the calculated flow field. Then droplets can exchange momentum, mass, and energy with the fluid phase. Finally substitute the changes of momentum, mass, and energy into the flow and temperature field calculations of continuous phase as source terms. Repeat above steps until convergence.