International Journal of Heat and Mass Transfer 116 (2018) 259–272

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Conjugate analysis of wall conduction effects on the thermal characteristics of impinging jets

G. Nasif *, R. Balachandar, R.M. Barron

Department of Mechanical, Automotive & Materials Engineering, University of Windsor, ON N9B 3P4, Canada

ARTICLE INFO

Article history:
Received 21 May 2017
Received in revised form 7 September 2017
Accepted 10 September 2017
Available online 17 October 2017

Keywords:
Jet impingement
Conjugate heat transfer
Nusselt number
Thermal resistance
VOF

ABSTRACT

A transient numerical investigation using the conjugate heat transfer (CHT) technique has been carried out to evaluate the effects of boundary heat flux on the thermal characteristics due to the turbulent air and water jet impingement process. For the water jet, the volume of fluid method is employed to capture the interface in the multiphase flow. It is found that the wall conduction may alter the prediction of the fluid-solid interfacial thermal characteristics compared with that from a pure convection process. The degree of influence depends on the working fluid, nozzle size, metal thermal conductivity, metal thickness and boundary heat flux. The CHT approach tends to reorganize the uniform heat flux distribution at the boundary to a non-uniform heat flux distribution at the fluid-solid interface. This is mainly attributed to the conjugate effect of the solid. For a given jet Reynolds number and boundary heat flux, the CHT results reveal that the convective heat flux at the stagnation point is higher for the air jet than for the water jet. Contrary to the non-CHT predictions, the interfacial temperature distribution is uniform for metals that possess a higher thermal conductivity with the air jet and for all metals with the water jet.

1. Introduction

Impinging jets provide an effective mechanism to transfer energy and mass in various engineering applications ranging from textiles to electronic components. Due to its high localized heating and cooling rates, a turbulent jet of gas or liquid directed to the target can effectively heat up or cool down a particular region. The heat transfer mechanism associated with the jet impingement process has been studied, both experimentally [1–5] and numerically [6–10]. The effects of the nozzle diameter, jet Reynolds number and nozzle-to-target spacing on the thermal characteristics have been thoroughly investigated in earlier studies. Nevertheless, there are still other aspects that should be examined to provide an enhanced understanding for the jet impingement process, e.g., the influence of the conduction and wall heat flux on the thermal characteristics. The conjugate heat transfer technique is used in the present work to investigate these aspects.

Many industrial applications are subjected to a strong thermal interaction between fluids and solids. Analytical approaches can generate good results to identify the main parameters of the thermal interaction problem and to verify the numerical codes. However, the applications of the analytical methods are restricted to very simple configurations [11–15]. Experiments are considerably expensive and cannot be fully relied on in the industry. Modern computational techniques, such as conjugate heat transfer (CHT), were developed after computers came into broader application to replace the empirical expressions of proportionality of heat flux to temperature difference with heat transfer coefficient (HTC). The state-of-the-art of the computational methods involves coupling the conduction in the solid and convection in the fluid to predict the HTC at the interface. The coupled approach is more reliable and more realistic than a decoupled solution [11]. In the computational CHT approach, two separate simulations are set up, one for fluid analysis and another for solid thermal analysis. Assuming the temperature distribution on the wall boundary, the fluid flow problem is solved to determine the local HTC distribution on the wall. The HTC distribution with the reference temperature is applied to the solid thermal simulation to re-evaluate the temperature distribution in the solid. The wall temperature distribution predicted by the solid thermal analysis is fed back to the transient flow simulation and applied as a wall boundary condition to re-evaluate the modified HTC distribution at the fluid-solid interface. The iteration process continues until the solution is obtained with a suitable accuracy.

Zhu et al. [16] investigated the wall effect on the HTC of an impinging jet. They concluded that the CHT approach redistributes the boundary heat flux and changes it from a uniform heat flux

* Corresponding author.
E-mail address: nasifg@uwindsor.ca (G. Nasif).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.09.034
0017-9310/

Nomenclature

A	surface area of the control volume
C	Courant number
CHT	conjugate heat transfer
c_p	specific heat
D	disc diameter
d	nozzle diameter
H	nozzle-to-target spacing
HTC	heat transfer coefficient
h	heat transfer coefficient (HTC)
h_0	heat transfer coefficient at the stagnation point
i	internal energy
L	pipe nozzle length
k	turbulent kinetic energy
Nu	Local Nusselt number = hd/κ
Nu_0	stagnation point Nusselt number
Nu_{m}	maximum Nusselt number
$\mathbf{n}$	normal vector
Pr	Prandtl number = ν/α
Pr_t	turbulent Prandtl number
$\mathbf{q}$	heat flux vector
q_b	boundary heat flux
q_i	interfacial local heat flux
q_0	stagnation point heat flux
Re	Reynolds number of the jet = $V_j d/\nu$
R_a	conductive thermal resistance in the axial direction
R_{h0}	convective thermal resistance at the stagnation point
R_{tr}	conductive thermal resistance in the transverse or radial direction
r	radial distance measured from the jet axis
$S_{\emptyset}$	source term
t	time
t_p	plate thickness
T	temperature
T_i	local temperature at the interface
T_j	bulk temperature of the jet at the nozzle exit
T_0	minimum temperature at the stagnation point
T_{ref}	reference temperature = T_j
$\mathbf{U}$	mean velocity vector
$\mathbf{u}$	instantaneous velocity vector
V_j	bulk velocity at the nozzle exit
u^*	shear velocity = $\sqrt{\tau_w/\rho}$
V	control volume
y	normal distance from the wall (or interface)
y^+	wall non-dimensional distance = $u^* y/\nu$
α	thermal diffusivity = $\kappa/\rho c_p$
α_i	volume fraction of i^{th} phase in the control volume
$\Gamma_{\emptyset}$	diffusion coefficient (μ or κ)
Δt	time step
κ	thermal conductivity
κ_{eff}	effective thermal conductivity
κ_f	fluid thermal conductivity
κ_m	metal (solid) thermal conductivity
μ	dynamic viscosity
ν	kinematic viscosity of fluid = μ/ρ
ρ	density
τ_w	wall shear stress
τ_{w0}	wall shear stress at the stagnation point
$\emptyset$	transported scalar property
ω	specific dissipation rate

boundary to a nearly isothermal boundary. The heat redistribution is also driven by the non-uniform distribution of the HTC on the impinged surface. The degree of heat redistribution is related to both conductive thermal resistance in the solid and convective thermal resistance at the interface. The study also revealed that the CHT prediction may have a negative impact on the local Nusselt number (Nu), indicating a decrease in the local Nu as the thermal conductivity of the solid decreases. Mensch and Thole [17] used a conjugate heat transfer approach to account for the combined effects of both internal and external cooling. The geometry that was employed in the latter study is a turbine blade endwall that includes impingement, film cooling and heat conduction through the endwall. The conclusion from this study revealed that internal HTC of impingement geometries is sensitive to geometric parameters, while the average temperature of the endwall external surface is not particularly sensitive to geometric parameters.

The flow field of the jet impingement process has been extensively investigated in previous work [1–5]. Nevertheless, it is worthwhile at this point to discuss the effect of the nozzle-to-target parameter (H/d) on the local Nusselt number to justify the selection of the $H/d = 6.0$ in the current study. Here H is the distance from the nozzle to the impingement plate and d is the nozzle diameter. The effect of this parameter is well documented in [2] for air jets using a nozzle size of $d = 25.0$ mm. The study divulged that at $H/d = 2.0$, the local value of Nu begins to increase from the stagnation point (Nu_0) towards the primary maxima (Nu_{m}) at the radial location of the nozzle end-edge side. The Nu_{m} represents the maximum Nusselt number in the local distribution of the Nu profile. The Nu then decreases towards the transition zone to the local minima. After this position, the Nu increases and reaches the secondary maxima due to the flow transition from a laminar to turbulent boundary layer. After the secondary maxima, the Nu decreases steadily into the wall jet region. The primary maxima in the Nu profile might be attributed to the accelerating flow in the fluid film and the interaction between the jet's central core and wall at small nozzle-to-target spacing [18]. As the nozzle-to-target parameter increases to $H/d = 4.0$, the primary maxima Nu_{m} is shifted towards the stagnation point, while the secondary peak does not occur clearly, but the tendency is maintained with nearly constant Nu distribution at the transition zone. This indicates that the nozzle-to-plate spacing $H/d = 4.0$ can be a threshold for the secondary maxima to occur for air jets [2]. As the nozzle-to-target spacing increases beyond the $H/d = 4.0$, the central potential core of the jet will not reach the wall. The Nu_{m} occurs at the stagnation region, regardless of the jet Reynolds number. This results from the high turbulent intensity of the impinging jet. The turbulent intensity at the jet centerline continues to increase with the increase of the H/d, which will enhance the heat transfer coefficient at the stagnation point. For liquid jets, the velocity distribution approaches a uniform profile when the nozzle-to-target spacing increases, e.g., when $H/d > 5.0$ (for water), because the viscosity tends to eliminate the radial gradients within about five diameters downstream of the nozzle [19]. Therefore, the Nu_{m} occurs at the stagnation point for the water jet when $H/d > 5.0$. In the current study, a nozzle-to-target spacing of $H/d = 6.0$ is chosen to provide a unique location of the occurrence of the Nu_{m} at the stagnation point for both air and water jets. The maximum Nusselt number, i.e., $Nu_{\mathrm{m}} = Nu_0$ will be used as a normalized parameter to normalize the local Nusselt number profile.

This research is part of an ongoing study to evaluate jet impingement performance. A detailed analysis for the complex flow field, including the velocity field and second-order statistics,