



On the numerical–experimental analysis and scaling of convective heat transfer to pulsating impinging jets



Sajad Alimohammadi*, Darina B. Murray, Tim Persoons

Trinity College Dublin, Dept. Mechanical and Manufacturing Engineering, Dublin, Ireland

ARTICLE INFO

Article history:

Received 28 November 2014

Received in revised form

17 July 2015

Accepted 17 July 2015

Available online 14 August 2015

Keywords:

Heat transfer enhancement

Pulsating impinging jets

Unsteady CFD

Local heat transfer coefficient

Experimental validation

Near-wall turbulence

Strouhal number

ABSTRACT

This paper presents a comprehensive numerical–experimental study performed on unsteady impinging jets with flow pulsation and builds on an experimentally validated numerical Reynolds-averaged Navier Stokes (RANS) CFD model for fluid flow and heat transfer simulations of steady impinging jets. The model accounts for intermittent pulsation with a square-wave form as the inlet velocity signal. Comparing against experimental local heat transfer data as reference, accurate CFD simulation results are presented. The study is performed over an extensive range of operating conditions for an axisymmetric air jet impinging on a flat surface for nozzle-to-surface distances $1 \leq H/D \leq 6$, Reynolds numbers $1300 \leq Re \leq 2800$ and Strouhal number $0.0029 \leq Sr \leq 0.333$ (pulsation frequency $5 \text{ Hz} \leq f \leq 260 \text{ Hz}$). The numerical model is used to extend the findings beyond the experimental range and simulates considerably higher pulsation frequencies. The numerical results also provide a more detailed insight into the near-wall behaviour in the viscous sub-layer, which helps to understand the governing heat transfer mechanisms. Correlations for the averaged and stagnation point Nusselt number enhancement in pulsating flow have been least-square fitted to the numerical data as a function of suitably modified Strouhal numbers.

© 2015 Elsevier Masson SAS. All rights reserved.

1. Introduction

Increased awareness about the energy consumption of data centres and telecommunication systems have made electronics thermal management a very active research field. Research activities cover a wide range of cooling techniques, from high density heat transport and two-phase convection systems, to high and moderate density cooling using forced liquid and air convection in micro-structured heat sinks. Impinging jets, consisting of fluid issued from a nozzle hitting a solid surface, can achieve high local heat transfer rates [1,2]. Impinging jets have been used in many cooling/heating engineering applications, including cooling of electronic devices and turbine blades, de-icing of aircraft, and drying of textiles, among many others.

Steady impinging jets are considered as an effective technique to increase convective heat transfer due to a very thin thermal boundary layer forced over a surface [1–11]. Pulsating or intermittent impinging jets are generally believed to enhance the heat

transfer when compared to steady jets, although there remains a lack of consensus on the level of enhancement and for some conditions the heat transfer can even be impaired [12–21]. The intermittent nature of a pulsating jet effectively redevelops and breaks up the boundary layer within each pulsation period, which may result in a thinner boundary layer than that of an equivalent steady jet. This can be achieved using little additional energy with no need to increase the pressure or flow rate [13]. The shortening and spreading of the pulsating jet also allows for higher turbulence levels and promotes flow instabilities [17].

There is a lack of consensus in the scientific literature about the effect of flow pulsation on convective heat transfer enhancement. Although many studies have been carried out, the findings about the potential heat transfer enhancement induced by pulsating jets are quite varied [14–20]. Thus, some researchers report a significant heat transfer enhancement [14–17], while other studies reported no enhancement or even a reduction of the heat transfer using pulsating jet impingement [18–20]. Camci and Herr [14] found enhancement factors in stagnation heat transfer of up to 70% for $Re = 14,000$, $H/D = 24$, and $f = 100 \text{ Hz}$ ($Sr = 0.033$). Intermittent pulsation with a square-wave form has been experimentally investigated by Zumbrennen and Aziz [15]; they reported significant heat transfer enhancement of up to 100% using the

* Corresponding author. Tel.: +353 1896 1034.

E-mail addresses: alimohas@tcd.ie (S. Alimohammadi), dmurray@tcd.ie (D.B. Murray), tim.persoons@tcd.ie (T. Persoons).

Nomenclature

D	nozzle pipe inner diameter (m)	q	local convective heat flux (W/m^2)
f	pulsation frequency (Hz)	Sr	Strouhal number (fD/U_m)
h	heat transfer coefficient ($\text{W/m}^2 \text{K}$)	Re	Reynolds number
H	nozzle-to-surface distance (m)	T	local surface temperature (K)
k	thermal conductivity (W/m K)	T_{ref}	fluid temperature in jet nozzle (K)
L	nozzle length (m)	u, v	radial, axial velocity components (m/s)
Nu	Nusselt number (hD/k)	u', v'	radial and axial fluctuating u, v (m/s)
Nu_0	stagnation Nu	U_m	mean flow velocity (m/s)
$\overline{Nu}$	area-averaged Nu	x, r	axial and radial coordinate (m)
Pr	Prandtl number	y^+	non-dimensional distance from the wall
		ω_z	vorticity, normal to $\{x, r\}$ plane ($1/\text{s}$)

square-wave pulsation at high frequency $f = 142 \text{ Hz}$ ($Sr = 0.37$). Different waveform shapes were also investigated by Herwig and Middelberg [16], showing that a square-wave form pulsation produces much higher enhancement (for $Sr = 0.08$, $\delta Nu_0 = 25\%$) than triangular or sinusoidal forms. The numerical results presented by Xu et al. [17] show that increase in the Reynolds number and frequency augments the Nusselt number compared to steady flow (up to 47%). In contrast, Liu and Sullivan [18] have measured changes in heat transfer due to pulsation of -10% to $+10\%$, depending on the pulsation frequency and H/D . Using an acoustically excited air jet from a contoured nozzle, O'Donovan and Murray [19] have also reported a heat transfer reduction of up to -21% for $10,000 \leq Re \leq 30,000$ and $0.5 \leq H/D \leq 2$. This lack of consistency in findings is mainly due to the different ranges of parameters used in different investigations. Persoons et al. [12] have recently observed a change in behaviour due to flow pulsation in axisymmetric impinging jet. Thus, for $H/D = 1$ and a low pulsation frequency ($Sr < 0.025$), a reduction in stagnation point heat transfer rate by 13% is observed, increasing to positive enhancements for $Sr(H/D) > 0.1$ up to a maximum enhancement of 48% at $Sr(H/D) = 0.6$.

Since direct numerical simulations (DNS) and even large eddy simulations (LES) remain impractical for many current industrial engineering environments, there is a need to establish a reliable numerical methodology to investigate the fluid flow and heat transfer of unsteady impinging jets at low computational costs [12,19,22–26]. This can be achieved by using the computationally efficient Transient Reynolds-Averaged Navier Stokes (TRANS) turbulence modelling method. However to ensure good accuracy, the numerical methodology should be accompanied by experimental validation of heat transfer results. There have been few studies in this field with a major focus on enhancing the accuracy of unsteady heat transfer results using computationally low-cost turbulence modelling schemes [17,20,21]. The current paper tries to achieve accurate simulations by validating the results via thorough comparisons against experimental data.

The motivation for this research is to characterise the stagnation and area-averaged enhancements (or degradations) in convective heat transfer rate achieved by flow pulsation, when compared to steady jets, for an extended range of operating conditions. The results will be analysed in terms of the effect of different key parameters such as Reynolds number Re , dimensionless nozzle-to-surface distance H/D , nozzle diameter D , pulsation frequency f , and Strouhal number Sr .

Following a rigorous experimental validation, the CFD model will be used to simulate operating conditions beyond the frequency range for reliable operation of the experimental pulsating valve. This shows how the numerical model assists to avoid experimental errors in order to account for much higher frequency jets.

To better understand the heat transfer enhancement due to flow pulsation, near-wall flow characteristics such as the radial velocity gradient, the normalized vorticity and the Reynolds stress components, are examined in detail. This approach reveals that these flow characteristics are directly proportional to the heat transfer distribution of steady and pulsating jets over the impingement surface, and how the variation of jet frequency itself can affect the enhancement.

2. Numerical approach

The step-by-step approach toward the development of the unsteady CFD simulation methodology is briefly described in this section. Fig. 1 depicts the solution domain, the generated mesh and the boundary conditions used in the simulation. The CFD model reproduces the exact configuration used in the accompanying experiments (see Section 3). The governing equations were formulated to eliminate all gradients in the circumferential direction (see Ref. [27]). This assumption imposed an axial symmetry condition, and reduced the necessary simulated region to a radial slice through the domain. The computational domain extends far enough from the area of interest (up to a radial distance of $16D$ from the jet centreline) to prevent any outlet boundary effects on the results.

A fully developed inlet velocity profile is imposed at the inlet to the domain. At the radial outlet and unconfined top boundaries of the domain which are free to the environment, an opening boundary condition with zero relative pressure is used. Turbulence intensities of 3% and 5% are chosen for the domain inlet and far-field boundaries, respectively. It is required to prove the domain size independence. The procedure to determine the minimum required vertical distance from the unconfined top boundary (distance above the nozzle exit) is described in Alimohammadi et al. [23]. This is done to show that the entrainment from the surrounding region above the nozzle exit is modelled correctly. The planar heated wall surface at the bottom of the domain is set to a constant temperature of 60°C , in accordance with the experiments (see Section 3).

The basic mesh topology is generated based on the structured approach with a quadrilateral mesh; the mesh is then refined and adapted iteratively in regions with large velocity, pressure, temperature and turbulence gradients, such as the impingement wall region and the mixing region of jet and surrounding entrained air, in order to attain a stable solution. This is shown in Fig. 2. An adequate value of near-wall cell thickness is ensured by keeping the y^+ (non-dimensional distance of the first grid point off the wall, scaled with the local friction velocity and kinematic viscosity) below unity for the near-wall cells.

Regarding the spatial discretization scheme for the convection term in the conservation equations, firstly, the entire domain is

Download English Version:

<https://daneshyari.com/en/article/667998>

Download Persian Version:

<https://daneshyari.com/article/667998>

[Daneshyari.com](https://daneshyari.com)