Computers and Fluids 000 (2017) 1–11

Contents lists available at ScienceDirect

Computers and Fluids

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

Effects of nozzle arrangement on uniformity of multiple impinging jets heat transfer in a fast cooling simulation device

Zhe-Xi Wen, Ya-Ling He*, Zhao Ma

Key Laboratory of Thermo-Fluid Science and Engineering of MOE, School of Energy and Power Engineering, Xi'an Jiaotong University, Xi'an, Shaanxi 710049, China

ARTICLE INFO

Article history:
Received 30 March 2016
Revised 12 April 2017
Accepted 15 May 2017
Available online xxx

Keywords:
Ground cooling simulation
Multiple impinging jets
Heat transfer uniformity
Nozzle arrangement
Numerical simulation

ABSTRACT

A cooling simulation device utilizing multiple impinging jets is built for ground cooling simulation tests of high speed flight vehicles. To help the design of the device and supply a relatively uniform heat flux on the test piece surface, a three-dimensional numerical study is performed in this paper to investigate the effects of nozzle arrangement on the convective heat transfer uniformity of the impinging jets. A simplified physical model with the size of 200 mm × 200 mm × 50 mm is built and the shear-stress transport (SST) k-ω turbulence model is used in the calculation. The nozzle quantity is varied from 8×8 to 32×32 for uniform nozzle arrays with a fixed total area of the nozzles. The corresponding numerical results of uniform nozzle arrangements are analyzed in details. Based on the 16×16 uniform arrangement results, the effects of diameter varying nozzle arrangements on heat transfer uniformity are further examined. Finally, an overall performance evaluation indicator named *QU* ratio is used in the optimization of nozzle arrangement, which proves to be effective and convenient in practice.

1. Introduction

In recent years, the fast development of high speed flight vehicles has become a trend around the world [1–4]. Different kinds of supersonic and hypersonic flight vehicles have been developed and tested. Among all the problems followed, the tough thermal environment, which greatly deteriorates the thermal environment of high speed flight vehicles, brings many new challenges. For example, aerodynamic heating grows rapidly with the increase of the flight velocity and the surface temperature of such flight vehicles may exceed 1000 °C during the flight. In addition, some flight vehicles may further go through a fast cooling process after the aerodynamic heating stage because of deceleration or other reasons during the flight. The resulting rapid cooling of the high temperature surfaces will possibly lead to large thermal stresses, sealing failure, deformation of the structures and many other problems that could be critical to the safety of the flight vehicles. Therefore, some materials, structures and body parts of high speed vehicles should be tested on the ground and ground fast cooling simulation devices should be built. Ground cooling simulation can be classified into thermal tests [5–8], which is commonly seen in the development of flight vehicles. However, most of the reported thermal testing devices focus on the aerodynamic heating simulation using infrared lamps or other heaters. To simulate the cooling process, cooling techniques should be adopted on the high temperature surfaces after heating.

Based on the feasibility study and experimental validation results of different cooling techniques by the authors [9], multiple impinging air jets are selected to cool the test piece in the cooling simulation devices. A cooling simulation device [10] is designed and built for the tests of plate materials, of which the maximum size is 400 mm × 400 mm. Thus, uniformity of heat transfer on the test piece surface should be guaranteed for the homogeneousness of the cooling processes in different positions as required. Besides, good heat transfer uniformity can minimize the accessary thermal stresses which will interfere with the effects of the fast cooling itself. Since heat transfer of multiple impinging jets is closely related to the design of the nozzle array, the effects of the nozzle arrangement on heat transfer uniformity has to be examined to help the detailed design of the device.

A lot of researchers performed experimental or numerical investigations on impinging jets heat transfer. Xing and Weigand [11] studied the heat transfer of a 9×9 jet array impinging on a flat plate at jet Reynolds numbers from 15,000 to 35,000. An optimum jet-to-plate spacing was found in the experiments for different crossflow schemes. Goodro et al. [12] experimentally studied the effects of nozzle hole spacing on jet array impingement heat transfer in different conditions and a new correlation consid-

* Corresponding author.
E-mail addresses: yalinghe@mail.xjtu.edu.cn, hylepe@gmail.com, hylxjtu@gmail.com (Y.-L. He).

http://dx.doi.org/10.1016/j.compfluid.2017.05.012
0045-7930/© 2017 Elsevier Ltd. All rights reserved.

Nomenclature

c_1, c_2, c_3	parameters in determination of the *QU* ratio
c_p	specific heat (kJ/kg K)
D	diameter of nozzles (mm)
H	height of nozzles (mm)
i	sequence number of nozzles
k	turbulent kinetic energy(m^2s^2)
N	quantity of nozzles
Nu	Nusselt number
P	pressure (Pa)
Q	flow rate of air (m^3/s)
QU	overall evaluation indicator
$\bar{q}$	average surface heat flux (W/m^2)
$\Delta\bar{q}$	normalized extreme surface heat flux differences
$\bar{q}_s$	spanwise-averaged surface heat flux
Re	Reynolds number
S	total area of nozzle (mm^2)
T	temperature (°C)
U_i, U_j	time averaged velocity components (m/s)
u_i',u_j'	velocity fluctuations (m/s)
u_τ	shear velocity (m/s)
V_j	jet velocity at nozzle exit (m/s)
x, y, z	axis of Cartesian coordinates
y^+	dimensionless wall distance

Greek symbols

λ	thermal conductivity (W/mK)
μ	dynamic viscosity (Pa s)
ν	kinematic viscosity (m^2/s)
ρ	density (kg/m^3)
σ_q	standard deviation of the surface heat flux(m^2/s^2)
σ_q^*	normalized standard deviation of surface heat flux
ω	specific dissipation rate(1/s)

Subscripts

h	high
l	low
j	jet
w	wall

ering the Mach number effect was proposed. In addition, the effects of temperature ratio and Mach number were also investigated [13,14]. Lee et al. [15] discussed the separate and combined effects of hole spacing, jet-to-target plate distance and Reynolds number on cross-flow and heat transfer. The complex variations of heat transfer in different positions were shown. Wae-hayee [16] studied the effect of crossflow velocity both experimentally and numerically. The shifts of impingement region and the changes of Nusselt number with crossflow were observed and analyzed. Chougule et al. [17] used the SST k-ω turbulence model to calculate the multi-jet impingement heat transfer of air and the results were validated against experimental data. The effects of different influencing parameters were examined. Wang et al. [18] compared the accuracy of 15 turbulence models and 4 wall functions in modeling an impinging jet. It was found that the SST k-ω turbulence model is suitable for jets calculation. Xing et al. [19] conducted a numerical study on the heat transfer characteristics of impinging jets to evaluate the effectiveness of CFD codes. The SST k-ω turbulence model was used in their study. They got a well agreement between the numerical results and the experimental data. A conclusion was drawn that CFD codes were helpful in the design of multiple jets configurations. Zuckerman and Lior [20] reviewed the related investigations on jet impingement heat transfer. A detailed discussion was given on physics, heat transfer correlations and numerical modeling of impinging jets. The computational costs, heat transfer coefficient prediction accuracy and abilities to predict the secondary heat flux peak of different turbulence models were compared with each other. The SST k-ω and v^2-f turbulence models were recommended for the computational accuracy and reasonable computation time. Chiu et al. [21] compared the performance of elliptic nozzle arrays with different aspect ratios in experiment and an optimum aspect ratio was found. Caliskan [22] studied the nozzle geometry effect on flow and heat transfer characteristics with circular, elliptic and rectangular nozzle arrays. The best heat transfer performance was obtained with the elliptic arrangements. The effects of surface ribs [23], detached ribs [24] and twisted-tape swirl generators [25] on multiple impinging jets heat transfer are also investigated by researchers either numerically or in experiments. Despite all the above studies, most investigators focused on overall effects of heat transfer or detailed distributions of heat flux on target surfaces. Systematic investigations on heat transfer uniformity of impinging jets were not commonly seen. The authors compared the effects of 21 configurations with different nozzle geometries and arrangements on heat transfer uniformity in Ref. [26]. However, impinging jets from circular nozzles are most favorable in machining and further investigation should be performed.

In this paper, the nozzle arrangement effects on impinging jets are examined to help the design of a ground cooling simulation device. Different arrays of circular nozzles are used in the numerical study. Due to the complexity of impinging jet heat transfer, two typical nozzle heights and two flow rates of air are considered. Firstly, heat transfer uniformity indicators are calculated from CFD results with uniform nozzle arrangements. The diameters and nozzle number are different between cases. Detailed discussion is given on the resulting differences in heat transfer. Secondly, 3 diameter varying laws are applied within the 16×16 array and the effects of diameter variation on heat transfer uniformity are discussed. Finally, the overall heat transfer performance evaluation is made by using an indicator of *QU* ratio, which illustrates a convenient way of assessing different nozzle arrangements in designing the cooling simulation device.

2. Physical and numerical model

2.1. Physical model and boundary conditions

A schematic layout of the jet plenum and the test piece in the cooling simulation device is shown in Fig. 1. Compressed air coming from the air tanks enters the plenum and expands through the top plenum. The distributor helps to evenly distribute the flow while the buffer reduces the noise in the expansion process. In the middle plenum, two layers of honeycomb materials are installed as the flow straighteners. Pressure and temperature sensors are also mounted in this section. An $N \times N$ nozzle array is distributed on the replaceable jet plate in the bottom of the plenum. By machining circular holes with different diameters in different positions as required, different nozzle configurations can be obtained. In this study, all nozzles in an array are with the same diameter and positioned with uniform spacing. The impinging jets are generated by these nozzles at a height H and are used to cool the test piece on the pedestal. The test piece is heated to the required temperature by an infrared lamp heater and transferred on a rail to the position below the jet plate when the target temperature is reached. According to the jet plate designs, the test piece can then be cooled by different impinging jets. The size of the test piece and the cross section of the plenum are both 400 mm × 400 mm.

Despite the actual unsteady cooling process, steady air flow and convective heat transfer is numerically investigated for designing purpose. In addition, radiation heat transfer is not considered. Such simplification is reasonable because non-uniformity of convection

Please cite this article as: Z.-X. Wen et al., Effects of nozzle arrangement on uniformity of multiple impinging jets heat transfer in a fast cooling simulation device, Computers and Fluids (2017), http://dx.doi.org/10.1016/j.compfluid.2017.05.012