Effect of Orifice Geometry on Heat Transfer Characteristics of Jet Array

Leena R¹, Arunraj B² and M. Jose Prakash³
T.K.M college of Engineering, Kollam, Kerala, India

E-mail: ¹leenakalidas@gmail.com, ²arunrajbabu.95@gmail.com, ³jpmmech@yahoo.co.in

Abstract. Jet impingement is one of the best methods for achieving high heat transfer coefficient in a single phase and has been a topic of active research for several decades, involving both experiments and computations. Most of the research on multiple jets have been carried out for an array of jets in the high Reynolds number regime. Experimental and numerical investigations were carried out to study the effect of inline three jet array impinging on a flat plate with varying Reynolds Number. Air was considered as the cooling medium. This study investigates the fluid flow and heat transfer characteristics of circular and square jet arrays impinging orthogonally on a flat-plate with inline arrangements. The number of jets is 3. The Reynolds number is varied from 3000 to 6000. The numerical methodology is validated using the experimental local Nusselt number distribution for inline three jet array with the same geometry and boundary conditions. In this work, circular and square of equal hydraulic diameters were selected for a comparative study. The results reveal that square jet array has better heat transfer characteristics compared to a circular array. The optimum value of jet to target plate distance is 6 times the jet diameter for both circular and square jet array. Also, the optimum value of pitch to diameter ratio is 2.

Keywords: Jet impingement; inline array; Nusselt number

1. Introduction
Thermal management has become very important in industries due to the advancement of technology. The primary objective of thermal management is to remove heat from the thermal devices to ensure that they operate properly and avoid triggering the temperature-activated failure mechanisms. This is generally accomplished by conducting the heat away from the thermal bodies to a gas or liquid coolant provides significantly high local heat transfer coefficient. In this method, a thin boundary layer is formed due to the collision of the fluid with the wall surface. But the major disadvantage of jet impingement is that the local heat flux can be highly non-uniform. Processes such as cooling of gas turbine components, tempering of glass, drying of paper and textile, cooling of high heat flux supercomputers, power devices and advanced military avionics require a promising technique like jet impingement. A single jet is usually employed to produce localized heating and cooling. In many applications a large area is required to be heated or cooled or enhancement of global heat transfer is needed. Thus, it is necessary to apply multiple jet system or array of jet system. Uniform temperature distribution can be obtained by array of jets. Since multiple jets cover a large area the key factors to be considered are the high average heat transfer coefficient and uniformity of heat transfer over the target surface. In a multi jet configuration, individual jet can be affected by essentially two types of interactions. The first is the jet-to-jet interaction between pairs of adjacent jets prior to the impingement. Secondly, there is interaction between the impinging jets and the flow formed by the spent air of the neighbouring jets. Therefore, studies on multiple jets have been mainly focused on identifying the optimum multiple conditions by considering the interaction among the jets.

Ozmen [1] found that the interaction between the twin jets decreases with the increase in nozzle-to-plate distance and jet-to-jet spacing. Goodro et al. [2] conducted experimental studies on the thermal characteristics of circular jet array of 8d and 12d spacing. Local and spatially-averaged Nusselt number show a strong dependence on the jet Reynolds number for both situations, with negligible variations between Mach number = 0.1 and 0.2.
Chander et al. [3] conducted experimental studies on three interacting circular jets arranged in triangular configuration at low Reynolds number of 800 with inter-jet spacing (S/d) of 3, 4, 6 and 7.58. The interaction between the jets becomes stronger when the jet-to-jet spacing is 3d. It was found that the average heat flux is more uniform at a moderate separation distance of 5d and for a small inter jet spacing of 3d. Katti et al. [4] studied the influence of streamwise pitch on local heat transfer characteristics for in-line arrays of circular jets. They concluded that the stagnation Nusselt number decreases with increase in nozzle-to-plate distance and with the reduction in jet spacing due to increase of interaction between the jets and cross flow. Katti et al. [5] studied the effect of stream wise pitch on the local, stagnation and strip average heat transfer coefficients of multiple round jets.

Puneet Gulati et al. [6] studied the effect of jet-to plate spacing and Reynolds number on the local heat transfer distribution to normally impinging submerged circular air jet on a smooth and flat surface. San and Lai [7] studied the effect of jet to jet and jet to wall distances to optimize the heat transfer in stagnation point and obtained a correlation as a function of Reynolds number, s/d and H/d ratios. Heyerichs and Pollard [8] assessed the performance of turbulent models k–ω and several versions of the k–ε by considering both separating and impinging turbulent flows with heat transfer. Lance Fisher [9] conducted a numerical study to investigate the heat transfer to an axisymmetric circular impinging air jet using the k-ε turbulence model.

Srivalli et al. [10] analysed numerically laminar flow of multiple staggered array of square jets for Reynolds number ranging from 100 to 500. Leon et al. [11] has revealed that the thermal pattern in an in-line arrangement of circular jets. They reported that the interaction with the neighbouring jets affects the flow more strongly for the jet from a sharp-edged orifice than that from a contoured orifice. Yoshisaburo et al. [12] conducted experimental studies with square in-line array of circular jets for minimum cross flow condition. They found that the average Nusselt number shows its maximum value for a jet-to-jet spacing of 4d compared to 6d and 8d.

2. Experimental Setup

![Schematic layout of jet impingement system](image_url)

1. Dryer 2. Compressor with storage Tank 3. Air Filter 4. Pressure Regulator 5. Needle Valve 6. Flowmeter 7. Stainless steel pipe 8. Nozzle 9. Target Plate 10. Heater 11. Insulation 12. Power Supply 13. Data Acquisition System.

**Figure 1** Schematic layout of jet impingement system
Figure 1 shows the schematic diagram of the experimental setup. Air is supplied by an air compressor through a calibrated flowmeter. Air filter and pressure regulator are installed upstream of the flow meter to filter the air and to maintain the downstream pressure at 2 bars. The thermal environment of the microprocessor is simulated using a copper plate of size 100 X 100 ×6 mm with a heater placed underneath. A thin heating coil having the same dimensions as that of the cross section of the target plate is used for ensuring uniform heating. The bottom of the heater is well insulated to prevent heat loss. The target plate is placed on a two-dimensional traverse system so that the relative position of the target plate with the nozzle plate can be varied conveniently. The power input to the heater is varied using a dimmer stat and the voltage and current are measured using voltmeter and ammeter respectively. T-type thermocouples were used to measure the temperature at various locations. The temperature data were acquired using a temperature scanner.

Power is supplied from DC power source through a voltage stabilizer, a variac and a current transformer. The voltage across the heater and the current are measured by digital meters whose ranges and the accuracies are of 0 to 20 ± 0.5% V and 0 to 400 ± 0.5% A, respectively. The velocity of the jet exit air close to the nozzle was measured with a calibrated anemometer. In the experimental and numerical studies, the size of the orifice is 3 mm. Three circular holes, each with a diameter of 3 mm are drilled on the nozzle.

3. Numerical Setup

In this section, an overview on the numerical setup investigations is presented. The numerical grid density is taken into consideration

3.1. Boundary and Initial Conditions

The flow is assumed to be steady, incompressible, three dimensional and turbulent. Heat transfer across the thickness of the plate, buoyancy and radiation heat transfer effects are neglected. Properties of the fluid such as density, viscosity and thermal conductivity are assumed to be constant. The impinging plate was kept at constant heat flux of 2100W/m². The three-dimensional Navier-stokes and energy equation with standard turbulent model are solved using Ansys Fluent software. The numerical simulation has been done using SST k-ω-turbulent model for jet impingement application. This model used to predict capture gradient effect and heat transfer distribution near the wall accurately with moderate computation cost. The schematic diagram 3-D computational domain and section view of three jet circular and square array is shown in Figure 2.

Figure 2. computational domain of circular and square jet array of hydraulic diameter 3mm
3.2. Numerical grids independence study
Figure 3 shows the grid independent study performed for the physical model to obtain the optimum number of elements. The number of elements is varied from 0.05 million to 0.35 million. Result from the calculation using 0.25 million cells agrees well with the case using 0.3 million cells. From the study between 0.25 million cells and 0.3 million cells there is a slight variation which is below the order 5%. So, the mesh with 0.3 million cells has been chosen as the suitable mesh for further investigations.

![Figure 3 Grid independence study.](image)

3.3 Numerical solver
The simulation was performed using Computational Fluid Dynamics (CFD) Ansys Fluent that solves equations of continuity, momentum and energy using the Reynolds-Averaged Navier-Stokes approach. Fluid properties (Air):
- Density = 1.225 kg/m³
- Dynamic Viscosity = 1.983e-5 Pa·s
- Thermal Conductivity = 0.024 W/m·K

CFD model:
- Software: Ansys Fluent
- Steady analysis of single jet impingement on the flat plate
- Dimension of plate (100mmx100mmx6mm)
- Diameter of nozzle jet (3mm)
- Side of square jet (3mm)

The computational domain is modelled as a 3D steady system in the present investigation. Multiple circular jets and square jets are impinging on a flat surface. A finite volume based CFD package can be used for solving the governing continuity, momentum, energy and turbulence model equations. K-ω SST turbulence models are used for closure and power law scheme is used for discretization. A second order spatial discretization with SIMPLE algorithm can be used for pressure velocity coupling. The solution convergence is achieved when the sum of the normalized residuals was on the order of $10^{-4}$ for continuity and momentum. Convergence was achieved generally in 4,000 iterations with the use of under-relaxation parameters.
4. Results and discussion

This section provides the analysis of the flow field behavior of the array of circular and square impingement jets. Finally, the heat transfer characteristics in terms of the Nusselt number is presented.

4.1 Validation of Experimental results.

Figure 4 shows the variation of stagnation temperature at the top surface of the plate obtained from numerical studies. The result obtained from experimental studies is also shown for comparison. Here orifice diameter is 3 mm, jet to jet spacing is 6D and the Reynolds number is 3155. There is very good agreement between the values obtained from numerical and experimental studies and the maximum difference is only 2%. The results from present experiments and numerical analysis show that the heat transfer depend on distance from stagnation point in the stagnation region. The presence of the optimum nozzle-to-plate distance can be explained by the relationship between the fluid flow and heat transfer rates.

The heat transfer rate depends on the turbulence intensity of the jet nearer to the impingement surface. The turbulence intensity depends on the velocity of jet. At low values of the nozzle-to-plate spacing the jet axial velocity decreases as soon as it emerges from the nozzle and it increases as the distance increases. Thus, due to the increase of turbulence, the heat transfer rate increases with the increase of nozzle-to-plate distance up to the optimum value. At the optimum nozzle-to-plate distance, the mean velocity of the jet at the centre remains constant. This distance is almost equal to the length of the potential core. When the nozzle-to-plate spacing increases further, the fluid velocity decreases due to the spreading of jets due to the presence of surrounding air. But the presence of the confinement can reduce the spreading rate of the jet by restricting the flow of surrounding air.

4.2 Effect of stagnation Nusselt number with Reynolds number for Z/D=6

Figure 5 shows the effect of Reynolds number on stagnation Nusselt number. As the graph indicates, the stagnation Nusselt number increases with the Reynolds number for the range of values tested. The heat transfer rate of the jet impingement depends on the turbulence of the impinging jet. The turbulence of the jet depends on the velocity of the jet. Thus, the stagnation Nusselt number increases with the jet Reynolds number.
4.3 Effect of jet to jet spacing on stagnation Nusselt number

*Figure 7* shows the variation of the local Nusselt number with the jet-to-jet spacing for the Reynolds number equals 3155 and for the optimum orifice-to-plate distance of Z/D equals 6. The graph indicates that the Nusselt number reaches maximum values when the jet-to-jet spacing is 2D. Thus, the stagnation Nusselt number increases with the increase in jet-to-jet spacing until spacing between the jets up to 2D. The Nusselt number decreases with further increase of the jet-to-jet spacing the cooling area becomes smaller at larger jet-to-jet spacing. Thus, the stagnation Nusselt number shows a negative slope. Two relative maxima of Nu exist for the jet array considered. First relative maximum occurs at S=4mm and second relative maximum at larger S/D ratio. For small S/D ratio an increase in its value results in reduction in the jet interference, so the Nu increases. Further increase at S=6mm due to the fountain effect. The impingement introduces an up-wash motion which is called fountain because of which a recirculation occurs between the jets.
4.4 Effect of Nusselt number with Reynolds Number for Circular and square orifice

Figure 8 shows the variation of Nusselt number with Reynolds number of different orifice shape (square and circular) at stagnation point. Figure shows the variation of Nusselt number with axial distance for various nozzle shape having equivalent diameter and constant Reynolds number. The square orifice has larger heat transfer coefficient has compared to circular orifice this is because of larger covered area of square orifice. Due to larger entrainment the square orifice attains a large mass flux at the same distance from nozzle. It is evident that the square orifice has 15% more heat transfer rate as compared to circular orifice.
5. Conclusions
Numerical and experimental investigations were carried out to study the effect of geometrical and flow parameters on the heat transfer characteristics of multiple jets. The present investigations provide a good understanding of the fluid flow and heat transfer characteristics of air impinging jets. The numerical results of the present study agree satisfactorily with experimental data. Experimental and numerical results reveal that the stagnation Nusselt number attains its maximum value when the nozzle-to-plate distance is six times the diameter of the orifice.

The heat transfer rate is a strong function of the jet Reynolds number, nozzle-to-plate distance, jet-to-jet spacing. The experimental results for 3-in-line arrangement showed excellent agreement with the theoretical analyses. Maximum percentage error of approximately 2% occurs of temperature value at stagnation point. From the numerical study for circular and square jet array it is evident that the square orifice has 15% more heat transfer rate as compared to circular orifice.

6. Nomenclature
- D: Jet or nozzle diameter (mm)
- Z: Distance between orifice and target plates (m)
- T_{stag}: Stagnation Temperature (K)
- Re: Reynolds number
- S: Jet to jet spacing (mm)
- Nu_{local}: Local Nusselt Number

7. References
[1] Y Ozmen 2011 Confined impinging twin air jets at high Reynolds numbers Experimental thermal and fluid science 35pp 355-363
[2] Matt Goodro, Jongmyung Park, Phil Ligrani, Mike Fox, Hee-Koo Moon 2008 Effects of hole spacing on spatially-resolved jet array impingement heat transfer International Journal of Heat and Mass Transfer 5pp 6243–6253.
[3] Subhash Chander, Anjan Ray 2007 Heat transfer characteristics of three interacting methane/air flame jets impinging on a flat surface International Journal of Heat and Mass Transfer 50pp 640–653.
[4] Vadiraj Katti, S.V. Prabhu 2008 Influence of span wise pitch on local heat transfer distribution for in-line arrays of circular jets with spent air flow in two opposite directions Experimental Thermal and Fluid Science 33pp 84–95.
[5] Vadiraj Katti, S.V. Prabhu 2009 Influence of stream wise pitch on local heat transfer characteristics for in-line arrays of circular jets with cross flow of spent air in one direction International Journal of Heat Mass Transfer 45pp 1167-1184.
[6] Puneet Gulati, Vadiraj Katti, and S.V. Prabhu 2009 Influence on the shape of the nozzle on local heat transfer distribution between smooth flat surface and Impingement Air Jet International Journal of Thermal Sciences 48pp 602-617.
[7] San, J.-Y. and M.-D. Lai 2001 Optimum jet-to-jet spacing of heat transfer for staggered arrays of impinging air jets International Journal of Heat and Mass Transfer 44pp 3997-4007.
[8] K. Heyerichsand A. Pollard 1996 Heat transfer in separated and impinging turbulent flows International Journal of Heat and Mass Transfer 39pp 2385–2400.
[9] Lance Fisher 2001 A Numerical Investigation of Jet Impingement on a Heated Flat Plate The Pennsylvania State University ME pp 513-16.
[10] G. Srivalli, B. Raghavarao, M.R.CH. Sastry 2012 Fluid flow and heat transfer analysis of laminar multiple square jets impinging on a flat plate *International journal of engineering research and applications* 2pp 048-056.

[11] Leon F. G. Geers, K. Hanjalic, M. J. Tummers 2006 Wall imprint of turbulent structures and heat transfer in multiple impinging jet arrays *J. Fluid Mech* 546pp 255–284.

[12] Yoshisaburo Yamane, Yoshiyasu Ichikawa, Makoto Yamamoto and Shinji Honami 2012 Effect of Injection Parameters on Jet Array Impingement Heat Transfer *International Journal of Gas Turbine, Propulsion and Power Systems* 4pp 27-34.

**Acknowledgments**

The authors acknowledge the financial support by CERD research project.