(IJISE) 2017, Vol No. 5, Jan-Jun **e-ISSN: 2454-6402, p-ISSN: 2454-812X**

A COMPREHENSIVE ANALYSIS OF FLUID FLOW IN IMPINGING JET SYSTEM

Dr Arun Prakash Singh

Associate Professor Department of Physics, Hindu College Moradabad

ABSTRACT

Examples of impinging jets include quenching of metals and glass, cooling and drying of paper and other materials, drying of food products for e.g. freezing , drying and baking , textiles, age hardening, cooling of gas turbine blades and outer wall of the combustion chamber, manufacturing process such as grinding , deicing of aircraft system, thermal management of electronic equipments, anti-sediment jets in thermal and nuclear power plants, control of thermo-acoustic instability in gas turbine combustors, and safety requirements in the storage of cylinders containing liquefied gas etc. Investigators presented those three regions characterize the development of the single impinging jet as it flows from the nozzle toward the impingement surface namely, the free-jet region, the impingement region and the wall jet region

Keywords: Impinging jet, Convective heat transfer enhancement, Confinement, Thermal measurements, Pressure distributions.

INTRODUCTION

Impinging jets cool the surfaces instantly as the heat transfer efficiency is very high[1-9], it provide an effective way to transfer energy or mass between a surface and the fluid. Compared to other heat or mass transfer configurations that do not employ phase change, the jet impingement device offers efficient use of the fluid and high heat & mass transfer rates.

Heat transfer rates in these cases are affected by various parameters like Reynolds number, nozzle to plate spacing, oblique angle of impingement, radial distance from stagnation point, Prandtl number, target plate inclination, confinement of the jet, Nozzle geometry, jet array configuration, crossflow, jet to jet spacing in multiple jets, the magnitude of the imposed swirl velocity, curvature of target plate, roughness of the target plate and turbulence intensity at the nozzle exit and the ambient air temperature relative to the jet temperature[10-22].

A review of the most important result of impinging jet systems available so far is presented here. The review focuses on fluid flow and heat transfer in impinging jets of various configurations. The structure of this review comprises of two main parts, namely the results on experimental and numerical methodologies. Due to page limitations, the equations and governing physics are not presented in detail, however sufficient citations are included.

METHODOLOGY

Experimental methodologies for Flow and Heat Transfer

In the upcoming section, studies on single and multiple jet impingement comprising of circular, rectangular, slot, impinging air jet through a cross flow, array impinging air jet, twin air jet, swirl jet, round jet with triangular tabs are discussed.

1. Single Jet Impingement

Single Jet Impingement is a phenomenon in which the fluid exiting from a nozzle or orifice hits a wall or solid surface usually at normal angle. In order to produce localized heating and cooling, single jet impingement can be used.

2. Circular Jet

The heat transfer enhancement achieved by acoustically forcing round impinging jet with nozzle-to-plate distances of 0.625-2 diameter. It was found that the force caused a small change in the heat transfer at the stagnation point and in the impingement region. There was a significant change in the secondary peak of the heat transfer that occurred at a distance of approximately 1.5-2 diameters[9]. The heat transfer at the secondary peak could be increased up to 10% when the jet was forced at its natural frequency and suppressed by 10% when the jet was forced at lower frequencies.It has been proposed that the secondary peak in the heat transfer for small nozzle-to-plate distances is because of the unsteady separation of the secondary vortices that form near the wall, as the primary vortex rings from the jet shear layer pass over the wall [10, 11].

Heat transfer experiments on jet Impingement cooling on convex semi-cylindrical surfaces was conducted and observed that the stagnation Nusselt number increases with decreasing (cylinder to jet diameter) D/Dj [12]. Local heat transfer on convex surfaces subject to a round impinging jet was examined and also reported that the stagnation Nusselt number increases with decreasing D/Dj [13].

Experimental results contradicting those discussed above have also been reported [14] in which naphthalene mass transfer technique to measure mass transfer coefficient on a circular cylinder subject to a circular impinging jet was used. Both the stagnation point and circumferential distributions on a cylinder surface were measured. The stagnation mass transfer was found to be increased with increasing D/Dj at a fixed impinging distance and jet Reynolds number. Circumferential and axial heat transfer distribution on an isothermal circular cylinder subject to a round impinging jet was investigated [15]. The results showed that the increased cylinder diameter enhances the stagnation heat transfer. [16] Experimental and numerical investigation on a circular air impinging jet on a cylinder was performed and observed that the stagnation heat transfer increases as the diameter ratio D/Dj increases.

The heat transfer distribution between circular jet and flat plate for the nozzle plate spacing was greater than two times the diameter of jet, for the single jet as well as an array of jets[17-22].

The effect of very low nozzle-to-plate spacing ($z/d<1$) on the local heat transfer distribution on a flat plate impinged by a circular air jet issued through long pipe nozzle which allows for fully developed flow at the nozzle exit was studied and found that in the acceleration range of the nozzle plate spacing ($z/d<0.25$), maximum Nusselt number shifts from the stagnation point to the point of secondary peak with the effect being more pronounced at higher Reynolds number [23].

3. Slot jet

The surface curvature affects the various sizes of slot airjet impingement cooling flow structure and heat transfer around a concave/convex side of a heated semi cylindrical surface were studied. In the convex surface case, it was found that a series of three-dimensional counter-rotating vortices around the surface could increase the momentum transport in the flow structure and enhance the heat transfer process on the wall near the stagnation point. However, the heat transfer magnitude was reduced in the region away from the stagnation point where the flow became more stable due to the centrifugal force around the surface curvature [1] and [24].

The fluid flow and heat transfer characteristics of a slot jet impinging on a semi-circular concave surface was examined and they have laid emphasis on interpreting the heat transfer data in association with the measured mean velocity and velocity fluctuations of impinging and evolving wall jets region along the concave surface, specifically, the occurrence and location of a secondary peak [25]. The heat transfer characteristics of semi-cylindrical convex surfaces (with a diameter of D) subject to slot jet cooling with a slot width of b was investigated and observed that decreasing the ratio of D/b enhances heat transfer at the stagnation point and concluded that the increased counter-rotating vertex size is responsible [26].

4. Multiple Jet Impingements

Given a specific heat transfer coefficient, the flow required for an impinging jet device may be two orders of magnitude smaller than that required for a cooling approach using a free wall-parallel flow. Multiple jets may be used for more uniform coverage. The impingement cooling approach offers a compact hardware arrangement with no additional moving parts.

In many industrial applications usually, a large surface area of the equipment needs to be heated or cooled. In order to enhance the global heat transfer, multiple jets impingement can be used. The use of multiple jets enhances cooling uniformity by creating several impingement zones which cover a significant fracture of the heated surface. Nevertheless, multiple jets promote flow blockage between the jets and complicate fluid distribution downstream from the impingement zone. These problems become critical in the cooling of multi-chip modules, requiring uniform cooling of a large number of chips as well as ease of fluid introduction into, and rejection from the module in the smallest volume possible [27].

Potential solutions to these problems were to use a two-dimensional or slot jet which provide a larger impingement zone and ensured uniform coolant rejection following impingement. In addition, cooling via a two-dimensional slot jet maintains nearly isothermal chip surface conditions. Therefore, a two-dimensional or planar air jet obviously offers some beneficial features, such as cooling effectiveness, uniformity and controllability on compact electronic packages [28].

5. Cross Flow

Cross-flow is defined as the fluid flow in the direction perpendicular to the jet impingement flow. Impinging jets in a cross flow are encountered in industrial processes such as the Separated Over Fire Air (SOFA) jets in the furnace of a large capacity utility boiler and the air liquid two-phase jets in the Selective Non-Catalytic Reduction (SNCR) technology in reading NOx emissions. In order to achieve an efficient heat transfer, the cross-flow effect must be minimized.

The influence of jet-to-jet distance and jet-to-plate distance on heat transfer characteristics of round impinging jets with in-line arrangement on heat transfer was investigated. Their results revealed that the maximum heat transfer rate was achieved at $S = 4D$ and $H = D$.[31] visualized the flow and temperature patterns on the impinged surface of round jet arrays in a cross-flow. They suggested that the cross-flow significantly deforms the impingement area that located at downstream in confined channel [29] and [30]. The local impingement cooling effectiveness and heat transfer coefficients over the interaction area of an air jet impinging on a wall and subjected to a cross-flow of air was evaluated and it was reported that an impingement cooling effectiveness decreases with the blowing rate especially near the stagnation point. Main disadvantage of cross flow is that heat transfer coefficients reduced due to the blockage effect of cross-flow [32].

6. Twin Jets

The visualization and characterization of a twin jets flow, while impinging vertically to a solid surface, is of primary importance to the understanding of the phenomena relevant to V/STOL type of aircraft performance and stability when operating in ground vicinity, during short take-off and landing. Many studies presented about this subject are concerned with heat transfer rates of unconfined and confined single and multiple jets impinging on a plate. However, little is reported about flow structure of the multiple impinging jets. [33] have investigated experimentally the flow structure and heat transfer characteristics of air jet impingement for nozzle-to-plate spacing less than one nozzle diameter in the Reynolds number range of $3600 < Re < 27,600$.

A particular feature of the two-dimensional parallel flow of twin jets was the appearance of a sub atmospheric region between the jets, owing to the entrainment of the fluid by the turbulent jet [34]. A sub atmospheric region occurs on the impingement plate and its effect decreases with increasing nozzle-to-plate spacing at the impinging circular twin jet flow [35].

The heat transfer characteristics of multiple impinging air jets were reviewed and observed that there are two types of interaction between the jets in the multiple-jets system. The first type was the interaction between adjacent jets prior to impingement on the surface. The second type was the collision of two wall jets, which were generated after impingement. Such collision became significantly important when the jets were closely spaced, the nozzle-to-plate spacing was small, and the jet velocity is high [36-38].

[39] obtained high heat transfer coefficients for multiple jets in the stagnation region and at the second stagnation point (the midpoint between the two neighboring jets). The average Nusselt number of the twin jets system increased with decreasing jet-to-jet spacing was explained [40]. The influence of spent air exits located between the jets on local heat transfer coefficient for impinging array of air jets were examined [41]. The turbulence, mean flow and heat transfer characteristics of an array of confined impinging slot jet with those of a single jet was compared [42].

7. Swirl Jets

The impingement of swirling jets against heated surface is frequently used as a tool to increase the heat transfer from the surface just especially on the stagnation point. A rapid decay of heat transfer in the wall jet region is developed along the impinged wall due to the boundary layer development. The swirling jet is created by blades mounted with five different angles and swirling jets with five different swirl intensities were generated for the same flow rate. When the blades are mounted radially, no swirl is imported to the jet and the swirl parameter will be practically zero. However, with blades rotated at the maximum possible angle, the jet will have highest levels of swirls.

8. Triangular Tabs

The heat transfer produced by the orifice nozzles with contoured outlets is up to 20-30% higher than that produced by a simple square orifice nozzle [43, 44]. The heat transfer produced by the orifice nozzle also depends on the thickness of the orifice relative to the orifice diameter [45]. These changes in the orifice geometry modify the heat transfer to the impinging jet because they affect the development of the jet before it impinges on the plate, particularly in the case of confined jets [46, 47].

The breakdown of formation of ring structures should be promoted to increase mixing in the jet which enhances the heat transfer near the impingement region [48]. There are a number of techniques that can be used to promote this mixing, including adding tabs that protrude into the flow at the jet exit. For example, a delta tab at the exit of a round jet produced a pair of counter-rotating vortices that increased mixing in the jet shear layer. They also found that the delta tab produced a low-speed region directly behind the tab and high-speed regions on each side of the tab [49].

It is well known that the heat transfer produced by the jet in the impingement region is related to the turbulence intensity of the jet and can be increased by adding a grid at the jet exit when the nozzle-to-plate distance is less than 6mm diameter [50].

By increasing the number of tabs, the local and average Nu number can be increased; when the Nozzle to plate distance is increased the heat transfer in triangular tabs is reduced [51].

THERMAL MEASUREMENTS

Majority of the authors who studied the impingement of jet experimentally employed direct thermal measurements to estimate the convective heat transfer coefficient. Some of the methods used to measure cooled/heated surface temperature followed by various researchers are reviewed below.

1. Thermochromic Liquid Crystals (TLC)

The most precise and advanced method used to determine thermal performance of impinged jets is the Liquid Crystal Thermography. A thermo chromatic Liquid-Crystal technique is used to visualize and record isotherms on the uniformly heated impingement surface. The full field liquid crystal thermographs were recorded on a super video tape which produced a color image that has twice the quality as that of ordinary composite video tape. It improved the color resolution by absorbing the unreflected light. The captured color liquid crystal thermographic images were used to determine the surface temperature. This method is mainly used to determine the temperature distribution on the surface.

Only a few authors used the TLC as a transient technique [52 - 57]. However it has a serious disadvantage of having high cost, low color density, very much sensitive to temperature change and it is difficult to work with.

2. Thermocouple

Thermocouple is one of the most commonly used devices to measure temperature. It consists of two dissimilar metal conductors which connect each other at one or more spots, when there is temperature difference between those parts; thermocouple produces voltage which converts the temperature gradient into electricity. Several authors had used thermocouple to measure temperature [58-63]. The main advantage of using thermocouple is that it is self-powered and it does not require any external form of excitation. It can measure a wide range of temperature at low cost. The only disadvantage of thermocouple is that it is difficult to determine the system errors less than 1^0C accurately.

3. Infrared Thermography (IR)

Infrared thermography is also one of the widely used methods to detect Infrared energy emitted from objects. This emitted energy is captured as the temperature distribution on the surface of the object and it is displayed as a color photo. IR thermography can be used to give a relative comparison of temperature distribution over a wide surface area [64-68]. Some of the main advantages of using IR thermography are the measurement of temperature from a distance without contacting the object, measurement of temperature of a moving or hazardous object and, a drastic change in temperature of an object.

NUMERICAL METHODS

A preliminary prediction of the heat transfer characteristics of engineering applications can be obtained using mathematical correlations, normally in terms of the system governing parameters. Numerical methods relevant to our current topic are discussed below.

1. Direct Numerical Simulation (DNS)

Direct Numerical Simulation (DNS) is a simulation in computational fluid dynamics in which the Navier–Stokes equations are numerically solved without any turbulence model and this method solves the full Navier-Stokes, continuity and energy/mass diffusion equations using discrete units of time and space. DNS captures all the physical phenomena of turbulent flows completely. It requires an extremely small grid in order to fully resolve the smallest scales of motion. The DNS approach has the ability to capture unsteady vortex behavior and to resolve different time and length scales [69, 70].

DNS is usually used for simulations of flows at low and moderate Reynolds Number (up to 10000) and geometrics like two dimensional slot jets. So far, DNS has been used for analysis of single jet impingement, to study the unsteady flow dynamics, to study the dynamics and formation process of coherent structures and for the studies related to the heat transfer process [71-74]. DNS is also used for the analysis of entrainment process at the turbulent/non-turbulent interface [75, 76]. Consequently, long computation time limits the use of DNS to Reynolds Number much lower than those in the gas turbine impingement heat transfer application. In an attempt to rectify this situation, one of the CFD models Large Eddy Simulation (LES) can be used.

2. Large Eddy Simulation (LES)

LES is a time-variant approach. It tracks flow properties with the full equations down to user-defined length scale (typically the grid spacing), and then uses additional subgrid-scale equations to describe turbulent flow behavior at smaller scales. Low-pass filtering, the principle operation in LES eliminates the small scales (space and time) of the solution and thus reduces the computational cost.

LES is very much suitable for modeling of the flow of round jets at high Reynolds numbers since large scales are solved directly by the solver while the small ones are being modeled [77]. The LES study of multiple round impinging jets is performed at high Reynolds number [78].

They had successfully predicted the mean flow characteristics of the jet although only one quarter of the geometry was simulated by applying the symmetry boundary condition in the middle of the jets, which is not usual for LES.

Despite the additional sub-grid-scale modeling, LES still requires fine mesh in order to properly predict the fluid flow and heat transfer near the solid walls [79, 80]. With careful use, LES can provide very accurate and meaningful results. Though the LES method has shown a good and clarified understanding of formation, propagation, and effects of flow eddies upon the velocity fields and jet transfer characteristics [81-84], it requires high resolution in both space and time for stability and accuracy, and therefore still needs a great amount of computing power of time to produce satisfactory solutions for the transitional and turbulent flows. The use of LES does not necessarily have an upper or lower limit on Re, but the influence of turbulence is small enough for laminar flows ($Re<1000$) such that the DNS approach offers little improvement in accuracy over the time-averaged techniques detailed below. For such cases where computational cost is not a primary concern, the LES method provides the greatest information about the impinging jet flow field [85].

3. The Reynolds-Averaged Navier-Stokes (RANS)

RANS is a steady-state time-averaged solution technique which uses some version of the Navier-Stokes equations adjusted for the presence of turbulent flow. The commonly used RANS models for jet flows comes under two categories, the eddy-viscosity models and Second Moment Closure (SMC) models. Eddy viscosity models treat the turbulent viscosity as a scalar quantity, assuming an isotropy in the normal stresses, whereas the SMC models track all Reynolds stresses and heat fluxes using semi-empirical equations based on expected physical trends rather than direct derivations. The Computational simulation of SMC model is costlier than eddy-viscosity model [86].

Pre-conceptual and optimization studies require efficient analyzing technique. In RANS modeling only the mean flow characteristics are resolved by governing equations while the turbulence is strictly modeled by a semi-empirical turbulence model. The success of RANS simulation therefore depends on the accuracy of the model used. Models are usually based on assumptions, and therefore it is very important that the assumptions in the model derivation apply to the process under investigation [77].

4. Near Wall Treatment

In order to get accurate results the steady and transient models require a description of how the flow behaves next to the wall (the target surface) in addition to the portions of the CFD model that describes the fluid flow inside the computational domain.

Numerical models of turbulence near the wall commonly feature one of the following two approaches. In the first approach, the grid near the wall is constructed at sufficiently high resolution. It properly resolves the flow in the entire viscous sub-layer and turbulent boundary layer with turbulence equations at low cell Reynolds numbers. This requires a model capable of resolving turbulent behaviors very close to the wall, and a large computation effort. Another method uses algebraic equations which relate the steady and fluctuating velocity and scalar profiles to wall distance and surrounding fluid properties. These functions predict the flow properties in and above the viscous sub-layer. This method requires only a single cell in the sub-layer, and thus requires less computational time.

Modeling with various wall equations was performed by [87]. He concluded that the prediction of shear stresses by standard logarithmic law was poor (errors of up to -30% in the stagnation region). He also proved that a generalized non equilibrium law of the wall had performed well in the stagnation region but only under predicted wall shear stress (errors of up to -12%). By using the non-equilibrium law in the stagnation region and switching to the logarithmic law in the wall-jet region their "hybrid law of the wall" model gave improved results. For turbulent jet impingement studies, three second moment closure type models and one eddy viscosity model were compared. They stated that both the eddy viscosity model and the basic Reynolds stress model achieved very poor agreement with experiments. They had also found that the Near wall reflection model results were in better agreement with experiments [88]. Second moment closure model generally performs better than the standard k-ε model with wall functions [89].

5. k-ε model

k-ε model is one of the most widely used models on Computational Fluid Dynamics to simulate the mean flow characteristics of turbulent flow conditions. It is a two equation model with two transported variables such as turbulent kinetic energy (k) which determines the energy in turbulence and the turbulent dissipation rate (ε) which determines the rate of dissipation of the turbulent kinetic energy independently with a dissipation equation based on expected trends. It uses the Boussinesq hypothesis in order to calculate Re stresses as a direct function of velocity gradient. Experimentally determined constants are required to fully complete the equations.

It has been found by several authors that for impinging jet problem, it gives good results in the free jet region but very poor results in the wall jet region and stagnation region. It failed to predict the occurrence of secondary peaks in Nusselt number for the impinging jet problem. The standard *k-ε* model is modeled for flows at high Reynolds number and does not apply for low Reynolds number. The model uses wall function to determine the velocity profiles in order to get good results in such cases.

A two-dimensional implementation of the *k-ε* model and test data from the experimental work was compared [90-93].He found that the test case at Re=23,000, the center line wall-normal-root-mean-square (rms) velocity level of the model was predicted to be four times larger than those measured in the experimental work. The main problem found in the model was that it showed larger value of turbulent kinetic energy in the stagnant region. It over predicted the root mean square velocity normal to the wall at $r/D = 0.5$ which was corresponding to the edge of the jet. The model also over predicted Nusselt number in the center of the impingement region upto 40% and had failed to predict the secondary Nusselt number peak at $r/D = 2$.

An alternate *k-ε* model was again developed by [90] which produced a greatly improved impingement center line wall - normal fluctuating velocity values and better Nusselt number prediction in the $r/D < 2$ region. The largest error in Nusselt number were typically found to be 15%, which occurred in the range $1 < r/D < 3$. [91] used the low Re number model which was previously studied [92] in a comparative CFD study of various turbulence models used for the impinging jet problem. They had included the Yap correction term to adjust the dissipation rate *ε* as a function of *k*, and distance from the wall y in the model. A damping function was added to adjust the turbulent viscosity used in the conservations equations at low Reynolds Number. The dissipation rate was increased in order to reduce the turbulent length scale. Without the correction factors, the *k-ε* model will over predict turbulent length scale and turbulent viscosity. These model constants depend on experimental data, and are somewhat arbitrary. It produces poor results in the impinging jet problems, but remains as benchmark against which models are compared [93]. The *k-ε* models remains in use due to its common implementation and comparatively low computational cost [94-98].

6. *k-ω* model

The *k-ω* model differs from the *k-ε* models by solving turbulent kinetic energy (*k*) and energy dissipation rate per unit of turbulent kinetic energy (*ω*). In *k-ω* model, *ω* is determined using a conservation equation which contains experimentally determined functions, rather than direct calculation from the velocity field [99]. The *k-ω* model can generate good prediction of flow properties in the wall jet region, both in the logarithmic region and sub layer, without using damping functions. . The value of *ω* at or near the wall – adjacent cell may be set proportional to ν/y^2, meaning the user can fully specify the turbulence conditions at the wall, unlike in the *k-ε* model. The latest version of the *k-ω* model includes correction terms to improve predictions in the low Reynolds number flow regions. The *k-ω* model typically produces Nusselt number profiles with a local error of up to 30% of the experimental Nusselt number value. It can produce better predictions of the turbulent length scale than the *k-ε* model. Unfortunately, the *k-ω* model is sensitive to far – field boundary conditions.

[100, 101] demonstrated that *k-ω* model over predicted the Nusselt Number. With the inaccurate free jet modeling, dense wall grid requirement, and undesirable sensitivity to known far-field conditions, it has been found that the k-ω model is only moderately better than the *k-ε*; it offers better predictions of Nusselt Number, with a higher computational cost.

7. v^2 *f* Model

The v^2-*f* model is also known as the "normal velocity relaxation model". [91] has shown some of the best predictions with calculated Nusselt number values falling within the spread of experimental data [102]. The v^2-*f* model uses an eddy viscosity to increase stability (rather than using a full RSM) with two additional differential equations beyond those of the *k-ε* model, forming a four-equation model. It uses the turbulent stress normal to the streamlines (referred to as v^2) to determine the turbulent eddy viscosity. It incorporates upper and lower limits on the turbulent time and length scales. In implementations, the limits on these terms have been further modified to impose reliability constraints in few cases.

As with the *k-ω* model, the v^2-*f* model requires a dense wall grid. In some cases the v^2-*f* model has been shown to predict realistic levels of turbulence in the decelerating jet core but excessive turbulence levels in the shearing flow outside the core and in the wall jet. Despite this difficulty and its moderately high computational cost, it is acknowledged as one of the best predictors of Nu distribution. It has an advantage over the standard *k-ε* series of models because it can predict the occurrence, position, and magnitude of the secondary Nu peak for low H/D. This model is highly recommended for the impinging jet problem, and its moderate computational cost is offset by its ability to closely match experimental results.

RESULTS AND DISCUSSION

Pressure Distribution

Authors had analyzed the pressure distribution for orifice and air flame impinging vertically on a horizontal flat plate and reported that,

- Pressure distribution is independent of the Reynolds number, but dependent on nozzle-to-plate spacing and jet-to-jet spacing in multiple impinging jets.
- The pressure coefficient distributions are found to be maximum at the stagnation point.
- The maximum values of the pressure at the primary stagnation point decrease with increasing H/D spacing.

The pressure gradient distribution on the impingement plate is irrelevant to nozzle diameter, but strongly depended on nozzle exit configuration, nozzle-to-plate spacing, jet Reynolds number and jet type.

HEAT TRANSFER CHARACTERISTICS

1. Effect of Jet to Plate distance (H/D) and Jet to Jet spacing (L/D)

By changing the jet to plate distance, distinct change in heat transfer characteristics has been found by the researchers. They had observed that,

- The temperature gradient increases with reduction in the values of H/D ratio.
- The heat transfer coefficient varies inversely with nozzle-to-resistor spacing.
- The local Nusselt number decreases with increase in H/D.
- Formation of two local maxima are evident in the Nusselt number in case of small separation distances i. e when $H/D < 1.0$.
- The primary and secondary sub atmospheric regions occur on impingement surfaces and lie up to the same location on both surfaces in case of $H/D \leq 1$.
- The sub atmospheric regions become stronger with decreasing nozzle-to-plate spacing due to the recirculation zones affecting impingement.
- Secondary sub-atmospheric regions are stronger than primary sub-atmospheric region for larger jet-to-jet spacing.
- Nusselt number of the twin jets system increased with decreasing jet-to-jet spacing.
- When the jet-to-jet spacing (L/D) is increased, the primary stagnation point shifts in radial direction and the value of secondary stagnation point decreases.

2. Effect of Reynolds number

The heat transfer characteristics of impinging jet, including stagnation, local and average Nusselt numbers are significantly affected by jet Reynolds numbers. The results concluded by most of the authors are

- When Reynolds number is increased, both the area weighted average Nusselt number, Nu_{ave} and stagnation point Nu_o also increases.

- For swirl jets once the distance H/D is fixed, the stagnation point,Nu_o and the area weighted average Nusselt number, Nu_{ave} almost collapse and increase linearly with Reynolds number independently of the jet swirl intensity.
- The location of second maxima of Nusselt number depends on the Reynolds number.

3. Effect of Nozzle Geometry

Many authors had studied the heat transfer characteristics by using different nozzle geometries like sharp edged orifice, square edged orifice, standard edged orifice and sharp edged rectangular nozzle.

It had been observed that

- For large separation distances (H/D = 6), the local Nusselt number is independent of the nozzle geometry.
- In triangular tabs impinging jet, as the protrusion depth of tabs into the jet flow increase, the heat transfer rate is also enhanced.
- The surface temperature decreases with reduction in the jet diameter.
- The local Nusselt numbers in the region corresponding to $0<=H/D<=0.5$ increases with increasing nozzle diameter.
- In sharp edged orifice impinging jet, due to higher turbulence intensity at the exit, heat transfer is significantly enhanced.

4. Effect of Oblique angle

The heat transfer characteristics of impinging jet by varying the angle of inclination had been analyzed. It had been proved that the maximum local Nusselt number and the maximum pressure on the impingement surface move downstream while the inclination angle was increased.

5. Effect of turbulence Intensity

The effect of the turbulence intensity along the jet centerline in an impinging jet was investigated. They observed that when approaching the impingement surface, in contrast to the velocities, centerline turbulence levels increase throughout the development of the jet. They also concluded that when the jet flow is fully developed, the turbulence intensity decreases rapidly near the plate.

6. Effect of Jet arrays

Researchers used a square array of jets with pitches varying in span wise and stream wise direction. Correlations were proposed for the distribution of segment-wise average Nu numbers.

conducted experiments with square, equilateral triangular array of jets. The effect of different parameters on the average heat transfer coefficients was discussed. studied the local heat transfer distribution using the technique of the liquid crystal thermography by employing a square array of jets. Both stream wise and spam wise pitches are varied for the three configurations studied. Researchers studied the local heat transfer distribution due to an array of jets with one side exit using liquid crystal thermography. These studies were reported for a single z/d and pitch.

7. Effect of swirl Jet

Many studies discovered that the multichannel nozzle (Swirl Number, S = 0) is characterized by a quite higher average heat transfer rate at all the considered nozzle-to-plate distances, with very high heat transfer peaks only at the shortest considered distance (z/D = 2). This effect at short distances is probably due to a different entrainment mechanism along with a higher flow exit speed and is caused by the area reduction induced by the swirling insert. This advantage is less evident at higher nozzle-to-plate distances. In the circular impinging jet, the global heat transfer increases with the nozzle-to-plate distance of up to eight diameters, while in all the tested cases, the mean heat transfer, related to all the swirling jets, decreases with increasing the nozzle-to-plate distance. At high nozzle-to-plate distances, the multichannel jet produces an enhancement in heat transfer that seems to be due to the higher exit speed, which is caused by the area reduction induced by the swirl insert.

At a fixed nozzle-to-plate distance, by increasing the swirl number, it is possible to see that the swirling jets are characterized by a broadening of the impingement region and by a decrease of the global heat transfer. The swirling jets show the best performance in heat transfer uniformity, at a nozzle-to-plate distance higher than six diameters; the jet with Swirl number(S) = 0.8 represents an exception because its higher degree of swirl causes the formation of four distinct stagnation zones, which are highly decentralized because of a vortex breakdown that causes the highest standard deviation percentage. At high nozzle-to-plate distance, swirling nozzle with S = 0.6 shows an almost constant heat transfer distribution with a Nusselt number standard deviation percentage lower than 8%.

8. Effect of cross Flow

The cross flow in case of the jets issued from the orifices with AR = 4 is considerably less significant than that in cases of the ones delivered from the orifices with AR = 1 and 8. The influence of the cross-flow on the impinged jets with an in-line arrangement is less prominent than that on the ones with a staggered arrangement.

The in-line jet array at AR = 4 gives the most efficient heat transfer on the impinged surface corresponding to the minimum cross-flow effect. The heat transfer enhancement factor is found to increase with the jet-to-cross flow mass ratio when the Reynolds number and the jet diameter remain constant, and it increases with the Reynolds number when the jet-to-cross flow mass ratio and the jet diameter remain constant. It was observed to decrease with jet diameter when other conditions were maintained constant. The presence of a cross flow degraded the heat transfer performance by 13.6% which was caused by the blockage effect of the cross flow. Impinging jets are found to be capable of enhancing the heat transfer process even though the jet was not designed to impinge on the target surface. The vortices created in a jet in cross flow system moved downstream and helped to improve the heat transfer process. The heat transfer enhancement factor was found to be considerably large up to 2.3.

Several study conducted experiment and the measurements were reported for the local heat transfer to an impinging air jet with and without cross flow. The experiments were carried out in a wind tunnel and the cross flow is provided by a squirrel cage fan located at the exit of the tunnel. The cross flow velocity to jet flow velocity ratio (M) is varied from 0.055 to 0.2 and the Reynolds number from 78000 to 121000. Two jet-to-plate ratios (z/d) of 6 and 12 were considered for the experiment. made measurements of local heat transfer coefficients resulting from the impingement of a turbulent jet which interact with cross flow. The impingement surface was made up of 21 electrically heated segments. Adiabatic temperature was used in the calculation of the heat transfer coefficient.

Recent studied the local impingement cooling effectiveness and heat transfer coefficients with an air jet in presence of a cross flow. Heat transfer measurements were made for jet-to-plate ratio(z/d) of 6 and 12 and heat flux varying from 750 to 1200 W/m^2. It was observed from the available literature that the heat transfer on the target plat was provided only on the center-line along the direction of cross flow. Hence, there was a need to provide a complete spatial distribution of the heat transfer on the entire target plate.

The center-line distribution was also provided in addition to the Nusselt number contour-maps and a correlation was presented for the stagnation point Nusselt number in terms of the Reynolds number Re, cross flow velocity to jet flow velocity ratio, M and jet to plate ratio, z/d.

NUMERICAL RESULTS AND DISCUSSION

A large number of informative studies had been conducted using the DNS, LES, k-ϵ, k-ω, RANS and Near-wall treatment model. These models are found to give large errors compared to data sets. The v^2-f model is the best compromise between solution speed and accuracy as it can produce better predictions of fluid properties in impinging jet flows in a minimum computational time. Moreover, v^2-f model will provide more accurate predictions than the other models.

CONCLUSION

Fluid flow and convective heat transfer in jet impingement systems are of great importance in turbo machinery and power engineering. A review of the most important results for the jet impingement systems presented here elucidated the most important finding relating to investigations and predictions of heat transfer in single and multiple impinging jets. For the methodological side, the most reliable modern experimental techniques were outlined (thermocouples, infra-red cameras, laser Doppler and particle Image velocimetry) for experimental predictions. Numerical studies viz., direct numerical simulation, widely used Reynolds-averaged Navier-Stokes equations techniques, large-eddy simulation approaches, self-similar solutions and integral methods were also discussed.

This review concludes that,

- Heat transfer is more in multiple impinging jets than single impinging jets since the stagnation points are more in multiple impinging jets.
- The heat transfer characteristics are better for lower nozzle to plate spacing and nozzle diameter.
- Pressure drop coefficient is less for circular impinging jets.
- Area weighted average Nusselt Number, Nu_{avg} and stagnation point Nusselt number, Nu_s increased with increase in Reynolds number.
- Heat transfer increased with decreasing oblique angle.
- v^2-f model is reported to be best among all the other models.

Though the existing literature available on fluid flow and heat transfer of impinging jets are found to be exhaustive, the investigation on impinging jet can be further extended

- To study the heat transfer performance of intermittent jet flow, impinging jet system with varying power input.
- To investigate the effect of other type of working fluids (Air mist), the effect of plate material and thickness of the plate on the heat transfer and determine the effect of coating on plates.
- To understand the effect of dimpled/protruded plate on the heat transfer rate.

REFERENCES

1. Chan T. L. *International Journal of Heat and Mass Transfer.* 2002. 45. 993-1006p.
2. Sarkar A. et al. *J. Food Sci*. 2004. 69(4). 113-122p.
3. Moraga N.O.et al. *Int. J. Heat Mass Transfer.* 2000. 43. 53–67p.
4. Moreira R.G. *J. Food Eng.* 2001. 49 (4). 291–295p.

5. Olsson E.E.M. et al. *J. Food Eng*. 2005. 67. 273–280p.
6. Hoefler F. et al. *Journal of Turbo machinery*. 2012. 134 (3). 031022.
7. O'Donovan T.S. et al. *J. Mechanical Engineering Science.* 2006. 220 (6).837-845p.
8. Garimella S.V.et al. *ASME Journal of Electronic Packaging*. 2001. 123. 165-172p.
9. Liu T.et al. Int. *J. Heat Mass Transfer*. 1996. 39. 3695–3706p.
10. Didden N.et al. *J. Fluid Mech*. 1985. 160. 235–256p.
11. Landreth C.C et al. *Expt. Fluids.* 1990. 9. 74–84p.
12. Cornaro C.et al. *Int. J. Therm. Sci*. 2001. 40. 890-898p.
13. Lee D.H.et al. *Int. J. Heat Fluid Fl*. 1997. 18. 160-169p.
14. Sparrow E.M.et al. *J. Heat Transfer-Trans. ASME.* 1984. 106. 570-577p.
15. Tawfek A.A. *Heat Mass Transfer*. 1999. 35. 327-333p.
16. Singh D.et al. *Int. J. Heat Mass Transfer*. 2013. 60. 672-688p.
17. Puneet G.et al. *International Journal of Thermal Sciences*. 2009. 48. 602–617p.
18. Livingood J.N.B.et al. *A lit. survey, NASA Tech. Memo.*1970. NASA TM X-2778.
19. H. Martin. *Advances in Heat Transfer.* 1977. 13. 1–60p.
20. Jambunathan K.et al. *Int. Journal of Heat and Fluid Flow.* 1992. 13. 106–115p.
21. Viskanta R. *Experimental Thermal and Fluid Science* . 1993. 6. 111–134p.
22. Gardon R.et al. *Int. Developments in Heat Transfer, ASME.* 1962. 454–460p.
23. Lytle D.et al. *Int. Journal of Heat and Mass Transfer*. 1994. 37. 1687–1697p.
24. Gau C.et al. *J. Heat Transfer*. *1991*. 113. 858-864p.
25. Choi M.et al. *Int. J. Heat Mass Transfer*. 2000. 4. 1811-1822p.
26. LIN Z.H.et al. *Int. J. Heat Mass Transfer*. 1997. 40. 1095-1107p.
27. Wadsworth D.C.et al.*ASME J. Heat Transfer*.1990. 112. 891-898p.
28. Nuntadusit C.et al. *Int. Com. in Heat Mass Transfer*.2012. 39. 1154–1164p.
29. Katti V.et al. *Experimental Thermal and Fluid Science*. 2008. 33. 84–95p.
30. Brizzi L.E.et al. *Journal of Thermal Science.* 2000. 9. 217–223p.
31. Bouchez J.P. *International Journal of Heat and Mass Transfer*. 1975. 18. 719–730p.
32. Lytle D.et al. *Int. J. Heat Mass Transfer.* 1994. 37. 1687–1697p.
33. Tanaka E. *ASME.* 1974. 17. 920–927p.
34. Abdel-Fattah. *Exp. Therm. Fluid Sci.* 2007. 31. 1060–1072p.
35. Martin H. *Adv. Heat Transfer.* 1977. 13. 1–60p.
36. Polat S. et al. a review, Ann. Rev. *Numer. Fluid Mech. Heat Tra*. 1989. 2. 157–197p.
37. Koopman R.N. *Int. J. Heat Mass Transfer*. 1976. 19. 673–683p.
38. Koopman R.N. *Thesis. University of Minesota.* 1975.
39. Mikhai S. *Heat Transfer.* 1982. 3. 377–382p.
40. Yang Y. *Numer. Heat Transfer.Part A*. 1998. 33. 23–37p.
41. HuberA.M. *ASME J. Heat Transfer*. 1994. 116. 570–576p.
42. Saad N.R. *Int. Heat Fluid Flow.* 1992. 13. 2–14p.
43. Colucci D.W. *Expt. Thermal Fluid Sci.* 1996. 13. 71–80p.

44. Brignoni L.A. et al. *Int. J. Heat Mass Transfer*. 2000. 43. 1133–1139p.
45. Fitzgerald J.A. et al. *J. Heat Transfer.* 1997. 119. 630–632p.
46. Lee J. et al. *Int. J. Heat Mass Transfer*. 2000. 43. 3497–3509p.
47. Fitzgerald J.A. et al. *Int. J. Heat Mass Transfer.* 1998. 41. 1025–1034p.
48. Kataoka K. et al. *Int. J. Heat Mass Transfer.* 1987. 30. 559–567p.
49. Reeder M.F. et al. *J. Fluid Mech.* 1996. 311. 73–118p.
50. Gardon R. et al. *Int. J. Heat Mass Transfer.* 1966. 8. 1261–1272p.
51. Hoogendoorn C.J. et al. *Int. J. Heat Mass Transfer*. 1977. 20. 1333–1338p.
52. Koseoglu M.F. et al. *Int. J. Therm. Sci*. 2010. 49. 1417–1426p.
53. Colucci D.W. et al. *Exp. Therm. Fluid Sci.* 1996. 13. 71–80p.
54. Ireland P.T. et al. *AGARD Conf. Proc*. 1985. 390. 28p.
55. Baughn J.W. *Int. J. Heat Fluid Flow.* 1995. 16. 365-375p.
56. Yan W.M. et al. *Int. Journal of Heat and Mass Transfer.* 2004. 47. 5235–5245p.
57. Nuntadusit C. et al. *Int. Com. in Heat and Mass Transfer.* 2012. 39.202–207p.
58. SinanCaliskan. et al. *Int. Journal of Heat and Mass Transfer.* 2014. 75. 685–703p.
59. Gao N. et al. *International Journal of Heat and Fluid Flow.* 2006. 27. 33–41p.
60. Gao N. et al. *Int. Journal of Heat and Mass Transfer*. 2003. 46. 2557–2569p.
61. Schertzer M.J. et al. *Int. Journal of Heat and Mass Transfer*.2006. 49. 4200–4208p.
62. MiroslawZukowski. *Int. Journal of Heat and Mass Transfer.*2013. 57. 484–490p.
63. Khammar M. et al. *Int. Journal of Heat and Mass Transfer*. 2014. 76. 144–152p.
64. Patil V.A. et al. *Measurement Science and Technology*. 2005. 16. 472-476p.
65. Martin R.H. et al. *Int. J. Therm. Sci.* 2011. 50. 1199–1206p.
66. Gulati P. et al. *Int. J. Therm. Sci*. 2009. 48. 602–617p.
67. Fenot M. et al. *Int. J. Therm. Sci*. 2005. 44. 665–675.
68. Narayanan V. et al. *Int. J. Heat Mass Transfer*. 2004. 47. 1827–1845p.
69. Luo K.H. et al. *J. Eng. Math.* 1997. **32**. 121–142p.
70. Jiang X. et al. *Theor. Comput. Fluid Dyn.* 2000. **14**. 55–74p.
71. Chung Y. et al. *Int. J. Heat Fluid Flow*. 2002. 23. 592–600p.
72. Tsubokura M. et al. *Int. J. Heat Fluid Flow.* 2003. 24. 500–511p.
73. Hattori H. et al. *Int. J. Heat Fluid Flow.* 2004. 25. 749–758p.
74. Satake S.I. et al. *Int. J. Num. Meth. Heat Fluid Flow.* 1998. 8. 768–780p.
75. Mathew J. et al. *Phys. Fluids.* 2002. 14. 2065–2072p.
76. Ooi A. et al. *J. Fluid Mech.* 1999. 381. 141–174p.
77. Martin Draksler. et al. *Int. Journal of Heat and Fluid Flow.* 2014. 46. 147–157p.
78. Kharoua N. et al. *J. Eng.* 2011. 8. 40–48p.
79. Hadz M. *Ph.D. thesis, Delft University of Technology, Netherlands.* 2006.
80. Gant S. et al. *Springer, Netherlands*. 2010 .13. 217–222p.
81. Beaubert f. et al. *C. R. Mecanique*, 2002. **330**. 803–810p.
82. Gao S. et al. *Int. J. Heat Fluid Flow*. 1995. **16**. 349–356p.

83. Ha¨llqvist T Licentiate *Thesis*, *Royal Ins. of Technology*, *Stockholm*, Sweden. 2003
84. Olsson M. et al. *Phys. Fluids.* **10** (2). 476–486p.
85. Yue-Tzu Yang. et al. *Int. Journal of Heat and Mass Transfer.* 2007. 5o. 799–807p.
86. Chen C.J. et al. *Fundamentals of Turbulence Modeling*. 1998.
87. Bouainouche M. et al. *Int. J. Numer. Methods Heat Fluid Flow*. 1997. **7**. 548–564p.
88. Craft T.J. et al. *Int. J. Heat Mass Transfer*. 1993. 36. 2685–2697p.
89. Dianat M. et al. *The Pennsylvania State University*, *University Park*, 1995.
90. Craft T.J. et al. *Int. J. Heat Fluid Flow.* 1996. **17.** 108–115p.
91. Neil Zuckerman. et al. *Journal of Heat Transfer*. 2005. 127.
92. Chuang S.H. *Int. J. Numer. Methods Fluids.* 2000. 33. 475–498p.
93. Craft T.J. et al. *Int. J. Heat Mass Transfer.* 1993. **36**. 2685–2697p.
94. Merci B. et al. *Int. J. Numer. Methods Heat Fluid Flow.* 2003. **13**. 110–132p.
95. Papageorgakis G.C. et al.*Numer. Heat Transfer*. 1999. 35. 1–22p.
96. Hosseinalipour S.M. et al.*Numer. Heat Transfer Part A.* 1995. 28. 647–666p.
97. Mujumdar A.S. et al. *Appl. Therm. Eng.* 2005. 25. 31–44p.
98. Gibson M.M. et al. *Int. J. Heat Fluid Flow.* 1997. 18. 80–87p.
99. Wilcox D.C. *DCW Industries.* 2002.
100. Park T.H. *Int. J. Heat Mass Transfer.* 2003. **46**. 251–262p.
101. Heyerichs K. *Int. J. Heat Mass Transfer.* 1996. **39**. 2385–2400p.
102. Behnia M. et al. *Int. J. Heat Mass Transfer.* 1998. **41**. 1845–1855p.