Basrah Journal for Engineering Sciences, Vol. 23, No. 2, (2023), 78-86

Review Article

Journal homepage: www.bjes.edu.iq
ISSN (Online): 23118385, ISSN (Print): 18146120

OPEN ACCESS

A Review Study of Sweeping and Normal Impingement Jets

Ahlam A. Hassan [1,]*, Muneer A. Ismael [2]

[1,2] *Department of Mechanical Engineering, College of Engineering, University of Basrah, Basrah, Iraq*
E-mail addresses:shainshahram@gmail.com , muneer.ismael@uobasrah.edu.iq
Received: 6 April 2023; Revised: 2 May 2023; Accepted: 6 May 2023; Published: 30 December 2023

Abstract

This review focuses on the experimental and numerical studies of sweeping impingement jets that serve in cooling of hot surfaces. It is known that the impinging jets produce high-localized heat transfer coefficient. The sweeping jet covers a wider area on a hot target to improve the heat transfer rate, they could be used to increase the cooling rate of the impingement surface by disturbing the boundary layer. To display a readable survey, the current review was partitioned to four groups based on engineering configurations. The review shows that the sweeping nozzle gives better efficiency in heat transfer, improved Nusselt number and uniform target surface temperature, compared with the conventional normal jets. The current review reveals that the sweeping-jet mechanism can be achieved either by fluidic oscillator or by exciting a flexible wall forming an oscillating jet. Most of the fluidic oscillator researches are conducted experimentally (27%), while the researches that use flexible wall are about 24%.

Keywords: Sweeping jet, Impingement jet, Fluidic oscillator, Fluid-structure interaction (FSI).

https://doi.org/10.33971/bjes.23.2.10

1. Introduction

The fundamental objective of this review is to acquire an in-depth understanding of the progression of sweeping and regular impinging slot jet procedures, fluid-structure interaction, and recent advancements in this field of study. Publications that include unique ideas have been given higher priority. In this survey of the relevant literature, a variety of impingement jet configurations is discussed.

2. Heat transfer and the Fluidic Oscillator

For cooling purposes, impinging jets are utilized in a range of technological applications, such as jet engine fuel and rotor blades. When more dispersion and rapid mixing are required, the addition of an oscillation to a jet improves the jet's mixing capabilities, and the oscillation motion helps in the management of jet flows in gas turbines, diesel engines, and heat removal from heat sinks. The efforts of researchers in this field are categorized according to different geometries and innovative ideas.
Lundgreen et al. [1] implemented the CFD and focused on the pressure difference and how is introduced across such a fluidic oscillator to produce an oscillatory jet. As portrayed in Fig.1, several jets generate this periodic fluctuation often. The design and mass flow rate of the fluidic oscillator determine its harmonic distortion. They vary the spacing between the jet opening and the flat plate (target) through 3, 4, 6, and eight times the hydraulic diameter. They took into account, additionally, the Reynolds numbers, 5jetflow rates, and oscillation frequencies. They as compared the thermal performance of a fluidic with an instantly jet with identical hydraulic diameter, plate spacing, and flow rate. As the spacing of jets to the target goes high, the cooling performance turned into negatively impacted. As a result, the researchers deduced that the fluidic oscillator should be designed carefully to acquire its advantages in heat removing.

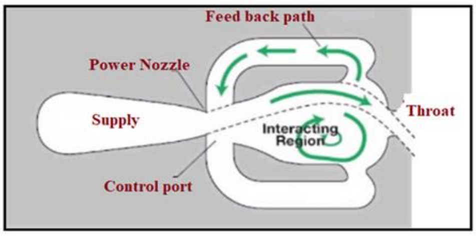

Fig. 1 Schematic of a fluidic oscillator

Hossain et al. [2] Performed a numerical simulation of a fluidic oscillator impinging perpendicularly on a cross flow with a free stream velocity of U∞. Three inclination angles of the throat of the fluidic oscillator, α = 30°, 60°, and 90°. The ratio of the throat velocity U_t to the free stream velocity was termed as blowing ratio and investigated within the range; 1, 2, and 3. The numerical solution was based on the unsteady RANS algorithm. They found that the cross-flow does not affect the frequency of the oscillating jet, while the blowing ratio affects the height of jet penetration and the lateral spreading of the jet. They also observed two alternative vortices in all blowing ratios and found that the higher the blowing ratio is the higher the mixing between the two streams. Wen et al. [3] conducted experimental measurements of a sweeping impinging jet's using time-resolved PIV. They fixed the spacing between the jet opening and the target at L/D = 8, and they focused on the effect of the Reynolds number. They recorded different findings; one of these is that the frequency of the oscillation raises almost linearly when the Reynolds number (Re) is raised from 2.7×10^3 to 9.2×10^3. In addition, they recognized that when Re is set at 4000, the oscillation motions in the near-exit region exhibit an equal distribution. While at Re = 9300, the jet in the same region displays irregular oscillation motions. Wen and Liu [4] undertook experiments to assess the flow dynamics produced by fluidic oscillator using time-resolved particle image velocimetry (TR-PIV). The Lagrange remodel technique is employed to explain the flux learning equation for current flow outside the generator. They found that the jet-spreading angle becomes wider when Reynolds ranges from 2500 to 11700. They also found that the size of the two bilateral bubbles, formed inside the chamber of the oscillator, is proportional to the Reynolds number, which in turn made the spreading angle wider. To examine the flow exiting from the oscillator, they implemented the Lagrangian dynamic mode decomposition (DMD) and recorded different behaviors of the flow distribution. Wen et al. [5] [Interaction of dual sweeping impinging jets at different Reynolds numbers] built a different fluidic oscillator by merging a pair of these devices in a single fluidic oscillator with two exit nozzles (as shown in Fig. 2). The merged devices are shared by a feedback port; thus, the two jets were correlated in their output characteristics. The experiments were focused on the flow behavior with the aid of time-resolved PIV against the Reynolds number. Three ranges of Reynolds numbers were tested. The outcomes of these experiments showed that the two jets work individually and stable with good in-phase motion at the lower Reynolds number (Re = 1800), while at the higher Reynolds number (Re = 9200), the two jets were randomly oscillating and the variations in their velocities were highly irregular.

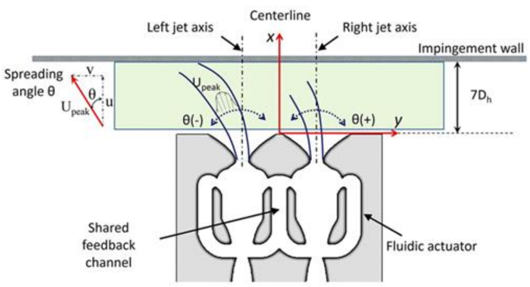

Fig. 2 Dual fluidic oscillators

Kim et al. [6] achieved an experimental heat transfer inspection of oscillating jet produced by a fluidic oscillator and impinging upon a curved heater. They inspected two curvatures of the heater-like plate, namely, convex and concave. In their studies, different complex phenomena were detected; the most immunizing of theme are the contribution of the centrifugal force in enhancing heat transfer in the concave surface. In addition, they detected that when the curvature is moderate, the heat transfer is better that the severe curvature. Kim et al. [7] conducted an experimental study on the fluidic sweeping jet striking upon a hot aluminum plate and compared the results with a steady square jet. The technique of phosphor thermometry was adopted to measure the temperature of aluminum plate. The measurements were conducted for different wall-to-jet spacing and Reynolds number. The oscillating flow of the jet exiting from the chamber was captured using the two-dimensional planner PIV. The measurements of the temperature and the flow fields served in calculating the local Nusselt number. They endorsed the merit of the sweeping jet over the square steady jet and they attributed this advantage to the turbulence made by the oscillation of the sweeping jet. They also concluded that the spacing of jet-wall should not exceed a critical value, 5 in their study, because with narrow spacing, the velocity of the jet is drastically decreased. Zhou et al. [8] performed an experimental study on the heat transfer characteristics of an incident sweep jet with a time interval function. They used the temperature-sensitive paint (TSP) technique to acquire the variations of the thermal fields. The influence of Reynolds number of Re = 5000, 10,000, 15,000 and close spacing H/D = 0.5, 1.0, 2.0, 3.0 on heat transfer was quantified. Particle image velocimetry (PIV) was used to connect heat transfer measurements with jet impingement flow fields. At Re = 5000 and 10,000, the sweeping jet showed decreased heat transfer near the nozzle (x/D = 4) but somewhat increased accuracy as one moved away from the jet. At Re = 15,000, it showed improved performance across the domain at H/D = 0.5 and 1.0. Sweeping jets can outperform circular jets with high Reynolds numbers and small spacing. In general, they reported that as Reynolds number and spacing decrease, the thermal performance rate of a moving jet increased. The oscillation of the coolant allows the sweeping current to cover a wider area and improve heat transfer in the remote area. At Re = 15,000, the broad jet has a significantly improved level of turbulent kinetic energy (TKE) compared to the circular example, leading to better convection at H/D = 0.5 and 1.0. Kim et al. [9] Quoted the idea of augmenting the transfer of heat by enlarging the jet-wall spacing. For this idea, they conducted the sweeping jet by using a fluidic oscillator with no feedback ports, i.e., free-feedback sweeping jet, and evaluated it by comparing its performance with the regular feedback sweeping jet and steady square jet. For this task, they used proper orthogonal decomposition analysis, time-resolved PIV and smoke-visualization technique. The temperature of the target, hot plate, was measured using thermographic phosphor thermometry. They obtained that the frequency of free-feedback oscillator generates higher jet frequency than the regular oscillator, while the sweeping angle was lesser with the free-feedback oscillator. They fulfilled the main purpose of the free-feedback oscillating jet that is the heat transfer is enhanced greatly than regular oscillator at higher jet-wall spacing's. Hossain et al. [10] did experimental and numerical studies on the impact of varying the exit nozzle angle of a fluidic oscillator on the fluid exit angle and the rate of heat removal from a flat plate. They tested eight nozzle exit angles and three rates of mass flow (0.97, 1.48 and 1.97 kg/s) against a fixed jet-wall spacing at H/D = 5. They showed that there is not a significant relationship between the oscillation frequency and the nozzle exit angle. On the other hand, a crucial role of the nozzle angle on both the jet angle and the rate of heat transfer was recorded. Namely, the jet angle showed an increasing function with the nozzle angle up to 70 degrees. Beyond this nozzle angle (70º – 130º), the jet angle tends to be unchanged. The nozzle angle affected the heat transfer negatively, but on the other hand, the temperature uniformity was improved.

3. Other sweeping jets

In this category, some works were found proposing sweeping jets but, in a mechanism rather than the passive fluidic oscillator. Camci and Herr [11] conducted an experimental study on a sweeping jet induced by using two communication ports supplying pressure to alternatively push the main jet before leaving the nozzle hole, as shown in Figure 3. They widened the zone that was subjected to the oscillating jet and proved its ability to remove heat from a flat plate more than a stationary square jet. The researchers used a triple decomposition method to a crossed hot-wire signal for the sake of describing the detailed fluid structures of the oscillating jet.

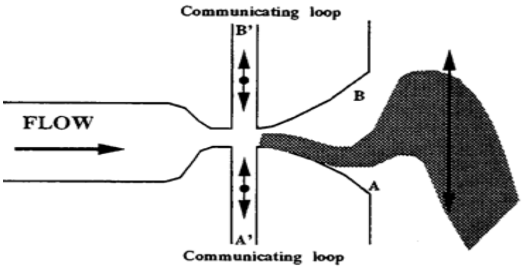

Fig. 3 A configuration of fluidic oscillator suggested by Camci and Herr [11]

Eghtesad et al. [12] achieved a numerical study for the sake of replacing the array of jets when the demand is the uniformity of temperature of a cooled flat plate. For this aim, they suggested two holes impinge two jets with a giving function having a specified frequency and pulsation as shown in Fig.4. Since their study is numerical, they tested a wide range of parameters, such as the Reynolds number, jet-wall spacing, spacing between the two jets, frequency and pulsation of the two jets and the phase shift between the two jets. The equations governing the flow structure were based on the k-ε turbulent model. Their main task was to bring the hot surface with uniform temperature in order to avoid thermal stresses. This task was achieved with aid of optimization using an artificial neural network ANN. They declared that the aim was obtained with a maximum accuracy of 98%. However, they did not discuss the mechanism by which the jets are oscillated. For this reason, we excluded this paper from the category of the fluidic oscillator. It is worth mentioning that this paper was an extension to the work of Hewakandamby [13] which he pointed out to the need of a fluidic oscillator.

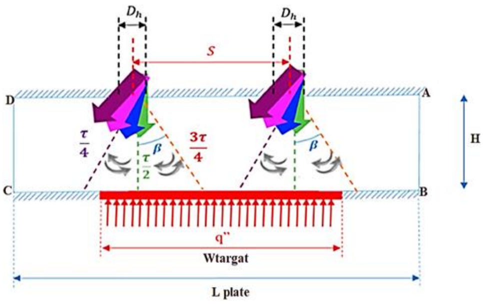

Fig. 4 Two oscillating jets proposed by Eghtesad et al. [12]

Before terminating this category, it is useful to point out to a special class of non-steady impinging jets namely, the swirling jets. This class of jets are conventional circular nozzles provided by means responsible for making a swirl in the jet before leaving the nozzle opening. The swirling mean is two (or more) angular suppliers connected at the tangent of the circular main nozzle to produce the swirling jet (as depicted in Fig. 5). Hence, there are three flow passages, one axial and two tangent passages. All these passages derive their flow from one source. Another proposed technique of generating a swirling jet is represented by using special guide vanes produced say using 3D printing techniques. The present literature review has revealed that the pioneer works regarding this field of investigation can be recorded for Herradad et al. [14], Ichimiya and Tsukamoto [15] and Browun et al. [16].

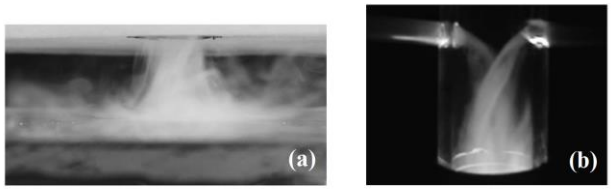

Fig. 5 (a) Swirling jet (b) the tangential flow that generates swirling [15]

4. Multiples impinging slot jets

Looking at the field of early impinging jets, leads to the conviction that this field has been started many decades and plentiful works were directed to this field. Several sorts of jets have evolved to enhance their performance through changing characteristics of fluid flow. It is not always remarkable for jets to be used for heating purposes; however, the main task of impinging jets is located at using them for programs that require cooling such as cooling of turbine blades and electronic devices and in heat treatment of steel production. In this category, the review is focused on conventional steady impingement jets. Lin et al. [17] reported a useful detail of experimental works on an impinging slot jet and heated target. The parametric adjustment in the examination of the Reynolds number and the ratio of jet-wall spacing was set at $190 \le Re \le 1537$ and $1 \le H/D \le 8$, respectively. They detected two distinct regimes, laminar and transitional with the studied range of Reynolds number. In addition, correlations for the stagnant and average Nusselt number have been derived from the experimental data. Rady and Arquis [18] proposed an array of jets impinging upon a flat hot plate with exit holes aligned with impinging jets i.e. along the same horizontal walls as portrayed in Fig. 6. In this design, the horizontal flow output is marginal. To keep the inlet and exit flows from disturbed interaction, the authors suggested a protrusions like-baffle extending from the upper wall where the holes exist. They treated the existing of baffles as a solid like-porous media, thus they added Darcy's drag term in the momentum equations. They performed their numerical solution using the computational software AQUILON, which is based on the finite volume method. Because the exit flow from exhaust holes reduces the attachment of impinging jet on the hot target, they recorded a reduction in heat transfer comes from the upper exit holes, thus they suggested the protrusion baffles and found them very effective in increasing the rate of heat removal from the hot target. Moreover, the pressure drops arising from this arrangement was not so significant. The Reynolds number were varied up to 500 (within the laminar flow), the jet-hot wall spacing, and the distance between the inlet and exit holes were also investigated.

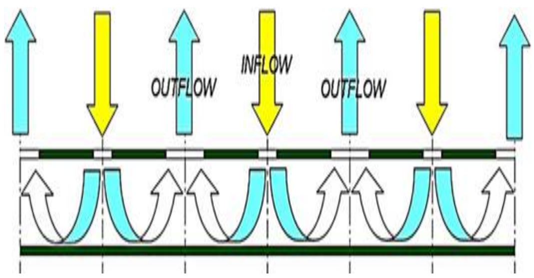

Fig. 6 Multiple inlet jets and exit holes proposed by [18]

Sivasamy et al. [19] studied numerically the characteristics of a slot jet impinging on a confined flat plate using the upwind finite difference method. They transformed the laminar governing equations into the stream function-vorticity form to discard the pressure gradient term, which in turn needs special effort to deal with. The range of the laminar Reynolds number was from 100 to 500 and the range of jet-wall spacing was from 2 to 5. They did not consider the thermal filed and focused on the flow behavior. However, they found that the reattachment length is a function of the Reynolds number and the jet-wall ratio. Arquis et al. [20] studied the characteristics of convection and conduction heat transfer in system composed of an impinging jet directed vertically on a channel involving a series of heated blocks separated by a specified distance. The problem is solved within the laminar region, where Reynolds number ranges from 100 to 500 and the effect of the drag generated from the existing of the hot blocks is simulated by adding Darcy's terms to the momentum equations. Finite volume method was followed in the numerical solution. They considered a wide range of the geometrical parameters such as the ratio of channel height to block length (0.5 – 1), slot width to block length (0.25 – 1), block aspect ratio (0.0627 – 0.25) and the ratio of block to fluid thermal conductivity ratio (10 – 1000). Beside to the primary vortex on the upper confining wall, they observed a secondary vortex in two zones, these are the space between the blocks and above the top surfaces of the downstream blocks. They recorded the maximum Number at the block underneath the slot jet while the downstream blocks experience lesser Nusselt number. In addition, the spacing between the blocks offered insignificant effect on the Nusselt number. Reynolds number had a positive role on the Nusselt number, while the height of the channel and the blocks do negative effect. Sharif and Banerjee [21] did a numerical study of convective heat transfer in geometry composed of slot jet impinging on a moving isothermal plate. An upper fixed plate confines the jet. The problem was studied within the turbulent two-dimensional regime, where the Reynolds number (based on the jet velocity) was ranging from 5000 to 20000. The k-ε turbulent model was implemented in the formulation of the problem. They also tested the jet wall spacing within 6–8. The speed of the isothermal moving wall was normalized with the jet velocity and was assumed 0 to 2. They deduced a significant relation between the average Nusselt number on the hot plate and the Reynolds number and the plate speed as well. Nevertheless, the friction, represented by the skin friction coefficient, was found to irrespective of the Reynolds number but it depends on the plate speed in a notable manner. Lee et al. [22] conducted experimental studies on a slot jet impinging upward on a flat plate heated by constant heat flux. The size of the jet is a couple of millimeters and the flow was controlled to be within the laminar regime. They varied the Reynolds number Re = 120 – 200, jet-wall spacing 0.75 – 12.5, and width of the slot 0.5 mm – 1.5 mm. Their outcomes were focused on the local Nusselt number at stagnation point. They established several correlations for the stagnant Nusselt number with the Reynolds number and the jet-wall spacing. In all these correlations, the role of the Reynolds number is stronger than the role of the jet-wall spacing. Agrawal et al. [23] used a circular jet impinging on a hot stainless-steel plate heated up to 800 ^{o}C. Their main goal was to use the water flowing from the circular jet in rewetting the hot surface. The experimental setup was done for turbulent Reynolds numbers ranging from 5000 to 24000 and jet-plate spacing (z/d) ranging from 4 to 16. Transiently, the rewetting occurred initial at the stagnation point faster, while apart from the stagnation point, the period of rewetting becomes higher. With increasing Reynolds number, the authors reported an increase in the heat transfer and faster rewetting action. Contrary, the jet-plate spacing showed irrelevant action on the rewetting. They established a correlation showing a strong proportional between the maximum heat transfer and Reynolds number and weak inverse relation among the radial distance from the stagnation point and the jet-plate spacing. Zukowski [24] performed a classical experimental work on a slot jet impinging on a flat plate heated by a 300 W/cm^2 constant heat flux. They varied three parameters namely, jet velocity 5 – 20 m/s, slot jet width 1 – 2 mm and the jet-wall spacing 4 –10. In their data, they focused on predicting the location at which the local Nusselt number is minimal. The relation revealed that this location becomes farther from the stagnation point (the center of the slot jet) with higher Reynolds number and wall-jet spacing while the width of slot jet brings the minimum location closer to the stagnation point. Caliskan et al [25] focused on the effect of the geometry of the nozzle opening by making several experiments and numerical analysis to an array of nozzles. The considered six nozzle geometries, these are generated from circular and square main shapes, i.e., by varying the aspect ratios of these two geometries. These geometries are holes of circular, horizontal ellipse, vertical ellipse, square, tall rectangular, and shallow rectangular made on a flat plate as shown in Fig.7. The flow fields were measured using laser Doppler velocimetry LDV while the thermal fields were measured by infrared thermal camera. The Reynolds number was ranged in the turbulent regime from 2000 to 10000, while the jet-wall spacing (H/d) was changed from 2 to 10. Their study revealed that the elliptical nozzle is the best from the thermal performance point of view.

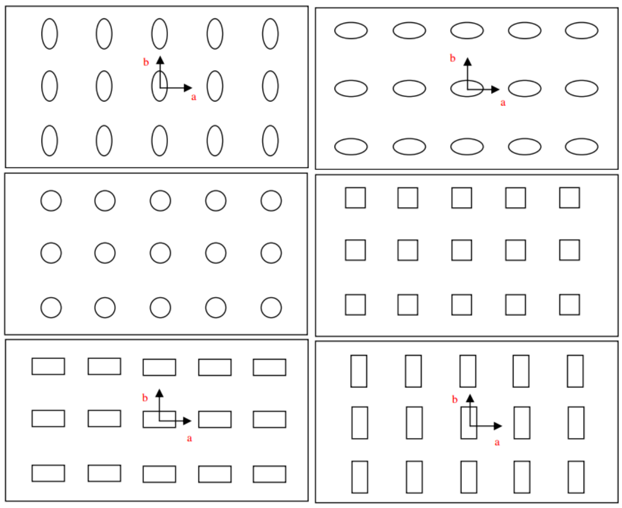

Fig. 7 Various geometries of nozzle-hole suggested by Caliskan et al. [25]

Lafouraki et al. [26] Conducted a computational simulation on an impinging jet towards a constant temperature concave plate. The confining wall has a convex geometry with same inclination angle to the lower concave one thus, forming a converging channel. The studied angle of this channel was from 0 to 5 degrees and the Reynolds number was with the laminar regime (100 – 400), while the spacing between the jet and the lower hot wall (H/W) was taken as 4 to 10. They used finite volume method in the numerical solution to compute the Nusselt number with three fashions, at stagnation point, locally and average along the hot lower plate. Their results showed a positive action of the converging angle of the channel on the average Nusselt number, where the augmentation in the Nusselt number was about 24% when the channel angle is raised from 0 to 5 degrees. On the other hand, the stagnation Nusselt number declines with the channel angle. They also observed an increase in skin friction with increasing the channel angle and attributed this to the rise of the velocity gradient. Rattner [27] focused on the purpose of avoiding the cross-flow interaction between the impinging jets and horizontal exit flow. Thus they, chose the strategy of using multiple inlets and multiple aligned holes. Their main goals were to improve the heat transfer and thermal stress uniformity of the horizontal plate and minimize the pressure drop. Their studied parameters were in the laminar regime and limited by the jet speed-based Reynolds number by 20 – 500, Prandtl number from 1 to 100, jet-wall spacing from 0.1 to 4 and the ratio of jet pitch to its diameter ratio from 1.8 to 7.1. The lower target plate was 5 x 4 mm and kept at constant heat flux 500 W/cm^2 heat flux. They used the Multiphysics COMSOL Ver. 5 to achieve the numerical simulations. Their main findings were that this geometry could bring the temperature of the plate with more uniformity and lower magnitude. The pressure drop was relatively moderate. Penumadu and Rao [28] discussed the heat transfer and fluid flow characteristics of a large number of impinging jets on a flat plate using the steady and unsteady RANS along with LES (large eddy simulation). Their scale was in microchannel thus, they considered low Reynolds numbers and focused on the produced drops of pressure. The two-dimensional array of the impinging jet was approximated to a single line of multiple 400 μm-diameter array. The simulations showed that the impinging jets closer to the output port (downstream of the central jet) are significantly influenced by the cross flow which leads to oscillate flow there. They addressed also the main source of the pressure drop, which are the contraction in the nozzle at inlet and due to the kinetic energy losses in the nozzles. They also addressed an effect of the jet-wall spacing on heat transfer even these spacing is relatively low, while the diameter of the jet showed negligible effects on the heat transfer.

5. Flexible baffles for flow direction

Employing of flexible walls or segment to interact with the fluid flow has a wide variety of applications in the engineering and industrial sectors. This field of investigation is termed as fluid-structure interaction (FSI). Microelectronic cooling systems, heat exchangers, and the temperature control of nuclear reactors are few examples of the numerous applications of this technology. The present review has revealed that this baffle can be used passively, i.e. it is fixed from one end and free from another, or actively i.e. applying an external excitation to its free end. The following Lambert and Rangel [29] investigated numerically the ability of a flexible flap in boosting the mixing of momentum in a microchannel. A force distributed along the entire length of the flap actuated the flap. This force generated a Reynolds number (based on the flap motion) varying from 0.3 to 80. In order to model the interactions between fluids and solids, the fictitious-domain DLM approach is utilized. The equations for momentum are solved one at a time for the fluid and the solid using the finite-volume approach and the finite-difference method, respectively. Equations are coupled using Lagrange multipliers that are dispersed throughout the system. According to their findings, mixing is enriched whilst the flap displacements are bigger, and whilst the dimensionless frequency is ranging from 1 and 2. When the length of the flap is 67% of the microchannel height, the mixing is optimal. Khanafer et al. [30] demonstrated a computational study to investigate a structural model of fluid interaction between fluid flow and heat transfer around a flexible micro-cantilever in a fluid cell involved in a microchannel. The flexible micro-cantilever was attached to a rectangular bluff body, and subjected to a noise at the other end. This study demonstrated the effects of random noise of free end, inlet velocity, dimension of the bluff body and micro-cantilever elasticity. Their outcomes predicted that the micro-cantilever oscillates in harmonically with the introduced random noise and with higher inlet fluid velocity. They predicted also that the higher bluff body led to preceded excitation. Ali et al [31] examined the laminar mixed convection in a parallel plate channel provided with four elastic flaps. These flaps are attached appositely on the upper and lower walls of the channel and given the same initial tilt. The main purpose of these flaps was to generate vortices, which in turn help in generating a motionless mixer. They considered two Reynolds numbers, 1000 and 1850 and compared their results with similar geometry but with rigid flaps. They observed that the elastic flaps serve in generating vortices at their free ands and these vortices detach transiently to introduce velocity gradients. The predicted enhancement in mixing was 98% and heat transfer enhancement of 134% compared with rigid flaps. The authors solved the problem using FSI analysis implanted in ANSYS Fluent software. Soti et al [32] used an in-house built numerical code to solve the mixed convection through a large-scale flow induced mixer. This mixer is composed of a flexible baffle attached horizontally to the back of a circular bluff body immersed in the cross flow between parallel plate channel. The walls are heated isothermally while the entering fresh fluid play the role of coolant. They worked within the laminar regime (Re = 100 – 500) and tested the flexibility of the baffle within dimensionless elasticity modulus $E = 1.4 \times 10^3$ to 2.8×10^3 N/m^2 and two values of Prandtl number Pr = 1 and 2. They left the free end of the baffle passive; thus, they investigated its vibrating action as a flow mixer. They got their aims by observing the vortices that generated at the wake zone serve in sweeping those vortices adjacent to the channel walls leading to disturb the boundary layer there and thus enhancing the heat transfer. Their numerical code was based on strong FSI in which the flow solver was based on sharp interface immersed boundary, while the structure was based on finite element method. Ma et al. [33] [Mechanism of enhancement of heat transfer for plate-fin heat sinks with dual piezoelectric fans] conducted an interested experimental study regarding the cooling of a U-shaped heat sink using two active baffles equipped with piezoelectric patches. Each patch was bonded on each elastic baffle. Hence, the two baffles formed what is called as piezoelectric fan. They achieved their experiments in two categories, parallel and vertical orientations as shown in Fig. 8. The measurements of fluid and thermal fields were executed by high-speed PIV and infrared thermometer, respectively. They inspected, relatively, wide range of parameters like the spacing between excited baffles, their height, phase shift between their vibration and the orientation.

Their results showed that the vertical orientation produces unlimited fresh air into the heat sink while the parallel orientation serves in unclogged ventilation inside the heat sink. In addition, they found that the lower spacing between baffles and lower height serve in reducing the thermal resistance. The in-phase operation of the dual baffles produced less thermal resistance than the out of phase operation.

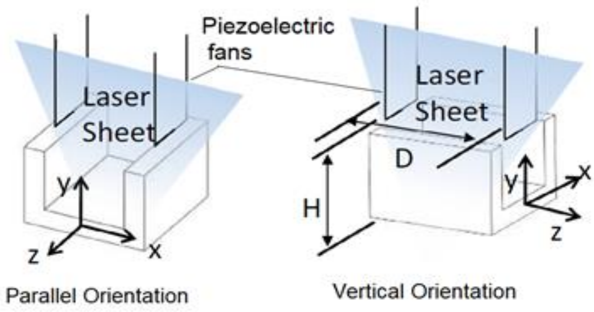

Fig.8 Dual piezoelectric fan of Ma et al. [33]

Li et al. [34] carried out a thermal measurement of the thermal filed of a heat sink composed of an array of square-pin fins. This heat sink was cooled by a dual flexible baffle piezoelectric fan. They inspected three geometrical parameters; these are the spacing between the tips of dual fan and the heat sink, the elevation of the heat sink itself and the orientation of the dual fan, face to face or edge to edge (as shown in Fig. 9). They studied additional operating parameter that is the phase shift between the vibrating baffles, in-phase (0 degrees) or out of phase (180 degrees) between them. They revealed that the out-of-phase operation of edge-to-edge arrangement could reduce the thermal resistance of the heat sink. Nevertheless, they pointed out that the action of these two parameters vary according to the various combinations of the other parameters.

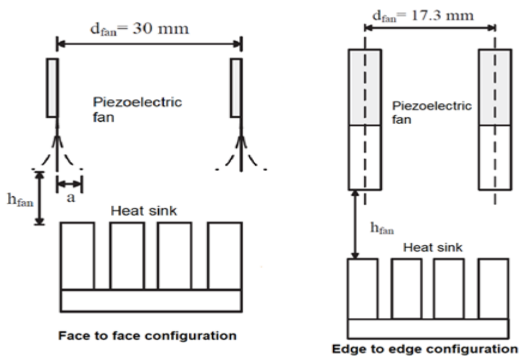

Fig. 9 Configurations of the dual piezoelectric fan of [34]

Generally, the piezoelectric fan consumes very low power and remove considerable thermal power from electrical devices and more reliable in operation. The research-attention in this field was started at the end of 1970's and continuing up to date. The review work of Hales and Jiang [35] gives a good survey to the works regarding with the PE fans. Ghalambaz et al. [36] utilized the FSI simulation to investigate the role of an excited flexible baffle attached horizontally on a vertical hot wall as shown in Fig. 10. The role of the active baffle was to disturb the thermal boundary layer and hence reduce thermal resistance and increase the heat transfer. The non-dimensional parameters that were studied are the Rayleigh number, baffle elasticity modulus, baffle length, baffle tip frequency and amplitude, and its thermal conductivity. They observed an increasing in the Nusselt number with increasing the amplitude of the baffle. Moreover, they reported that when the frequency is higher than 0.1, an enhancement of the Nusselt number is observed.

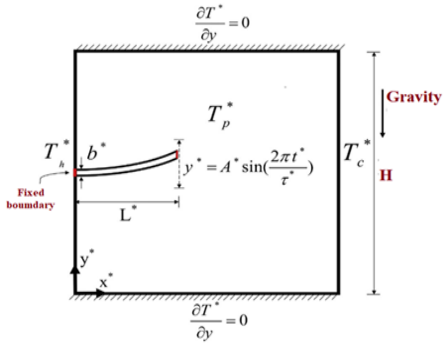

Fig. 10 The excited baffle used in the geometry of [35]

Ismael and Jasim [37] addressed the use of a passive flexible baffle in controlling the mixed convection inside a vented square enclosure. The enclosure had two apertures, an inlet at the lower left wall and an outlet at the upper right wall. The baffle is attached at the base of the enclosure in such a way to face the inlet port and interact with the inlet jet as portrayed in Fig. 11. They achieved their study numerically using the finite element. Their parametric study was achieved by considering the flexibility of the baffle which is governed by the Cauchy number (from 10^{-12} to 2×10^{-4}), the spacing between the baffle and the inlet aperture (from 0.2 to 0.8), Richardson quantity (Ri = 0.1 – 100), and Reynolds number (Re = 50 – 250). Their findings showed the flexible baffle could augment the Nusselt number by about 11% than the conventional rigid baffle. The spacing of the baffle play a notable role in augmenting the Nusselt number, and especially when it becomes closer to the inlet aperture.

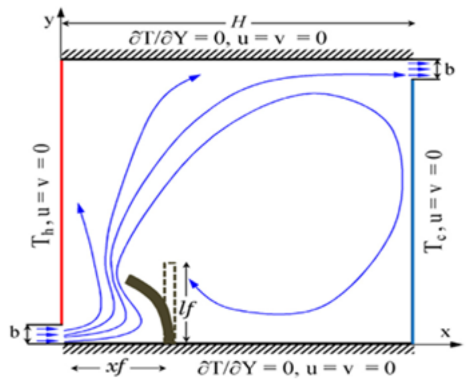

Fig. 11 Freely passive baffle facing an inlet jet, the geometry of [37]

5. Summary and motivation

The present study surveys work regarding the oscillating impingement jets. The oscillating jets serves in providing swept larger area of the hot surfaces and this task can be performed either using fluidic oscillator or using excited nozzles. The last two decades showed that the computational fluid dynamics (CFD) contributed in publishing abundant papers, while the experimental works are limited to fluidic oscillator. The following remarks are deduced from the present survey.

1. Most of the studies regarding the fluidic oscillator are achieved experimentally, and in these experiments, the need to accurate measurement instruments like PIV and thermal camera are mandatory.
2. There are few strategies used in oscillating the jet; fluidic oscillators which are passive devises used in sweeping jets impinging on hot surfaces; and piezoelectric baffles used in generating a mixing of rest fluid with hot sinks.
3. In experimental studies, the parameters are limited because of the difficulty in changing the pre-manufactured devises, for example, in the fluidic oscillator; the main parameters that were studied are the Reynolds number and the spacing between the jet and the target.
4. In piezoelectric fans, there are wider range of parameters, because this technique uses an active actuator, for example, the frequency and the phase shift between the fans can be controlled and added to the studied parameters.
5. In experimental studies, there is no restriction in using high values of Reynolds number, while most of the numerical studies prefer the laminar limits of Reynolds number. This is to avoid the convergence problem arising from the turbulent models.

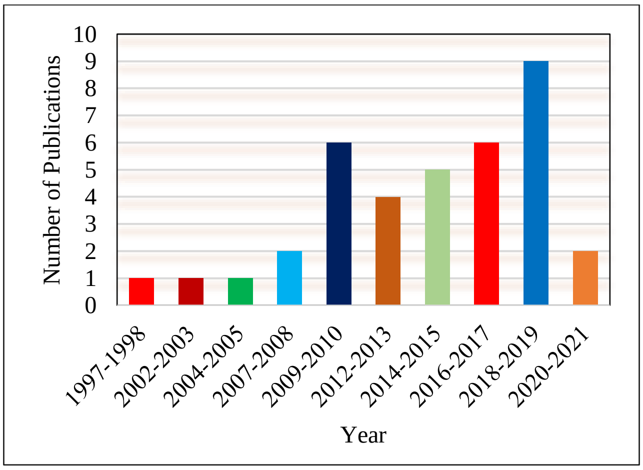

Fig. 12 Number of published papers arranged chronologically

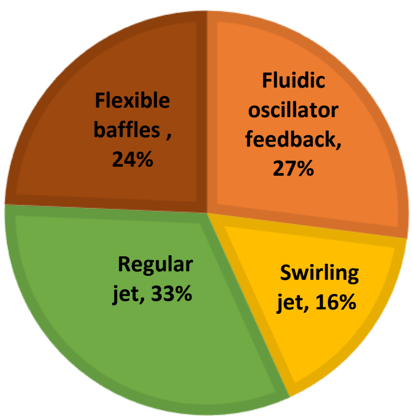

Figure 13: Percentages of impingement jets configurations

Symbols

U∞ = Free stream velocity
Ut = throat velocity
H/D = Fixed jet-wall spacing
α = inclination angles of throat
Ri = Richardson number
Re = Reynolds number
Ca = Cauchy number
Nu = Nusselt number
Pr = Prantal number
E = Modulus of elasticity
q"= Heat flux

Abbreviation

CFD = Computational fluid dynamics
RANS = Reynolds average Naiver-Stokes
PIV = Particle Image Velocimetry
TR-PIV = Time-Resolved Particle Image Velocimetry
DMD = Dynamic mode decomposition
TSP = Technique sensitive paint
TKE = Turbulent kinetic energy
ANN = Artificial neural network
LDV = Laser Doppler velocimetry
LES = Large eddy simulation

References

[1] R. K. Lundgreen, M. A. Hossain, R. Prenter, J. P. Bons, J. Gregory, and A. Amer " Impingement Heat Transfer Characteristics of a Sweeping Jet" American Institute of Aeronautics and Astronautics, /DOI:10.2514/6.2017-1535 https://arc.aiaa.org

[2] M. A. Hossain, R. Prenter, R. K. Lundgreen, L. M. Agricola, A. Ameri, J. W. Gregory, J. P. Bons " Investigation of Crossflow Interaction of an Oscillating Jet", American Institute of Aeronautics and Astronautics, 10.2514/6.2017-1690.https://www.researchgate.net/publication/312110373

[3] X. Wen, Y. Liu, and H. Tang," Unsteady Behavior of a Sweeping Impinging Jet: Time-Resolved Particle Image Velocimetry Measurements", Experimental Thermal and Fluid Science96 (2018) 111-127.https://doi.org/10.1016/j.expthermflusci.2018.02.033

[4] X. Wen, Y. Liu ""Lagrangian Analysis of Sweeping Jets Measured by Time-Resolved Particle Image Velocimetry", Experimental Thermal and Fluid Science, 97 (2018) 192-204.https://doi.org/10.1016/j.expthermflusci.2018.04.

[5] X. Wen, Z. Li, W. Zhou, and Y. Liu1," Interaction of Dual Sweeping Impinging Jets at Different Reynolds Numbers", the American Institute of Physics, Physics of Fluids 30, 105105 (2018); doi: 10.1063/1.5054161. https://doi.org/10.1063/1.5054161

[6] D. Kim, S. Jeong, T. Park, and D. Kim," Impinging Sweeping Jet and Convective Heat Transfer on Curved Surfaces", International Journal of Heat and Fluid Flow, 79 (2019)108458 . https://doi.org/10.1016/j.ijheatfluidflow.2019.108458

[7] S. Kim, H. Kim, and K. Kim, "Measurement of Two-Dimensional Heat Transfer and Flow Characteristics of an Impinging Sweeping Jet", International Journal of Heat and Mass Transfer 136 (2019).415-426. https://doi.org/10.1016/j.ijheatmasstransfer.2019.03.021

[8] W. Zhou, L. Yuan, Y. Liu, Di Peng, and X. Wen, "Heat Transfer of a Sweeping Jet Impinging at Narrow Spacings", Experimental Thermal and Fluid Science 103, (2019).89-98.
https://doi.org/10.1016/j.expthermflusci.2019.01.007.

[9] M. Kim, D. Kim, E. Yeom, and K. Kim," Experimental Study on Heat Transfer and Flow Structures of Feedback-Free Sweeping Jet Impinging on a Flat Surface", International Journal of Heat and Mass Transfer 159 (2020) 120085.
https://doi.org/10.1016/j.ijheatmasstransfer.2020.120085

[10] M. Hossain, A. Ameri, J. Gregory, and J. Bons, "Effects of Fluidic Oscillator Nozzle Angle on the Flow field and Impingement Heat Transfer", AIAA Journal April (2021), DOI: 10.2514/1.J059931
https://doi.org/10.2514/1.J059931

[11] C. Camci and F. Herr, "Forced Convection Heat Transfer Enhancement Using a Self-Oscillating Impinging Planar Jet" Transactions of the ASME 770 Õ Vol. 124, (2002).DOI: 10.1115/1.1471521.
http://heattransfer.asmedigitalcollection.asme.org

[12] A. Eghtesad, M. Abad bozchelou, and H. Afshin," Heat Transfer Optimization of Twin Turbulent Sweeping Impinging Jets", International Journal of Thermal Sciences 146 (2019) 106064.
https://doi.org/10.1016/j.ijthermalsci.2019.106064

[13] B. N. Hewakandamby," A Numerical Study of Heat Transfer Performance of Oscillatory Impinging Jets", International Journal of Heat and Mass Transfer 52 (2009) 396–406
b.n.hewakandamby@shef.ac.uk

[14] M. Herrada, C. Pino, and J. Casanova, " Confined Swirling Jet Impingement on a Flat Plate at Moderate Reynolds Numbers", American Institute of Physics of Fluids 21, 013601 (2009). DOI:10.1063/1.3063111.
http://ojps.aip.org/phf/phfcr.jsp

[15] K. Ichimiya and K. Tsukamoto," Heat Transfer Characteristics of a Swirling Laminar Impinging Jet", Journal of Heat Transfer, Vol. 132 / 012201-1 (2010).
http://heattransfer.asmedigitalcollection.asme.org

[16] K. Brown, G.Byrne, T. Donovan and D. B. Murray," Effect of Thermal Boundary Condition on Heat Dissipation due to Swirling Jet Impingement on a Heated Plate", Journal of Physics: Conference Series 395 (2012) 012039, doi:10.1088/1742-6596/395/1/012039.
(http://iopscience.iop.org/17426596/395/1/012039)

[17] Z. Lin, Y. Chou, and Y. Hung, "Heat Transfer Behaviors of a Confined Slot Jet Impingement", Int. J. Heat Mass Transfer. Vol. 40, No. 5, pp. 1095-1107, (1997).

[18] M. Rady, and E. Arquis, "Heat Transfer Enhancement of Multiple Impinging Slot Jets with Symmetric Exhaust Ports and Confinement Surface Protrusions" Applied Thermal Engineering 26,1310–1319 (2006).
www.elsevier.com/locate/apthermeng

[19] A. Sivasamy, V. Selladurai, and P. Rajesh Kanna," Numerical Simulation of Two-Dimensional Laminar Slot-Jet Impingement Flow Confined by a Parallel Wall", International Journal for Numerical Method in Fluids, 2007 55:965–983, DOI: 10.1002/fld.1492.
sivasamy@yahoo.com

[20] E. Arquis, M. Rady, and S. Nada," A Numerical Investigation and Parametric Study of Cooling an Array of Multiple Protruding Heat Sources by a Laminar Slot Air Jet "International Journal of Heat and Fluid Flow 28, 787–805, (2007) 787-805
www.elsevier.com/locate/ijhf

[21] M. Sharif, and A. Banerjee, "Numerical analysis of heat transfer due to confined slot-jet impingement on a moving plate" Applied Thermal Engineering 29, 532–540, (2009)) 532–540.

[22] D. Lee, J. Bae, M. Ryu, and P. Ligrani, "Confined, Milli scale Unsteady Laminar Impinging Slot Jets: Effects of Slot Width on Surface Stagnation Point Nusselt Numbers " Journal of Electronic Packaging, Vol. 134 / 041004-1, December (2012) DOI: 10.1115/1.4007317
http://www.asme.org/about-asme/terms-of-use

[23] C. Agrawal, R. Kumar, A. Gupta, and B. Chatterjee, " Rewetting and maximum surface heat flux during quenching of hot surface by round water jet impingement" International Journal of Heat and Mass Transfer, Volume 55, May (2012).
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2012.04.045

[24] M. Zukowski," Heat Transfer Performance of a Confined Single Slot Jet of Air Impinging on a Flat Surface" International Journal of Heat and Mass Transfer, Volume 57, Issue 2, February (2013).
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2012.10.069

[25] S. Caliskan, S. Baskaya, and T. Calisir, "Experimental and Numerical Investigation of Geometry Effects on Multiple Impinging Air Jets" International Journal of Heat and Mass Transfer 75 (2014) 685-705.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.04.005 0017

[26] B. Lafouraki, A. Ramiar, and A.Ranjbar," Laminar Forced Convection of a Confined Slot Impinging Jet in a Converging Channel", International Journal of Thermal Sciences 77 (2014) 130-138.
http://dx.doi.org/10.1016/j.ijthermalsci.2013.10.014

[27] A. Rattner, "General Characterization of Jet Impingement Array Heat Sinks with Interspersed Fluid Extraction Ports for Uniform High-Flux Cooling" Journal of Heat Transfer, Vol. 139 / 082201-1,(2017) . [DOI: 10.1115/1.4036090].
http://heattransfer.asmedigitalcollection.asme.org

[28]P. Penumadu, and A. Rao," Numerical Investigations of Heat Transfer and Pressure Drop Characteristics in Multiple Jet Impingement System" Applied Thermal Engineering 110 (2017) 1511-1524.
http://dx.doi.org/10.1016/j.applthermaleng.2016.09.057 1359

[29] R. Lambert and R. Rangel, "The Role of Elastic Flap Deformation on Fluid Mixing in a Microchannel", American Institute of Physics (2010); DOI: 10.1063/1.3410268.
rhrangel@uci.edu.

[30] K. Khanafer, A. Alamiri, and I. Pop," Fluid-Structure Interaction Analysis of Flow and Heat Transfer Characteristics Around a Flexible Microcantilever in a Fluidic Cell" International Journal of Heat and Mass Transfer 53 (2010)1646–1653

[31]S. Ali, C. Habchi, S. Menanteau, T. Lemenand ,and J. Harion," Heat Transfer and Mixing Enhancement by Free Elastic Flaps Oscillation" International Journal of Heat and Mass Transfer 85 (2015) 250-264.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.01.122

[32] A. Soti, R. Bhardwaj, and J. Sheridan," Flow-induced Deformation of a Flexible Thin Structure as Manifestation

of Heat Transfer Enhancement" International Journal of Heat and Mass Transfer 84 (2015) 1070-1081.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.01.048

[33] S. L. Ma, J. W. Chen, H. Y. Li, J. T. Yang," Mechanism of Enhancement of Heat Transfer for Plate-Fin Heat Sinks with Dual Piezoelectric Fans", International Journal of Heat and Mass Transfer,90 (2015) 454-465.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.03.050

[34] H. Y. Li and Y.X. Wu," Heat Transfer Characteristics of Pin-Fin Heat Sinks Cooled by Dual Piezoelectric Fans" International Journal of Thermal Sciences, 110(2016)26-35.
http://dx.doi.org/10.1016/j.ijthermalsci.2016.06.030

[35] A. Hales, X. Jiang," A Review of Piezoelectric Fans for Low Energy Cooling of Power Electronics", Applied Energy, 321-337.
https://doi.org/10.1016/j.apenergy.2018.02.014

[36] M. Ghalambaz, E. Jamesahar, M. Ismael, and A. Chamkha," Fluid-Structure Interaction Study of Natural Convection Heat transfer over a Flexible Oscillating Fin in a Square Cavity ", International Journal of Thermal Sciences 111 (2017) 256-273.
http://dx.doi.org/10.1016/j.ijthermalsci.2016.09.001

[37] M. Ismael, and H. Jasim," Role of the Fluid-Structure Interaction in Mixed Convection in a Vented Cavity" International Journal of Mechanical Sciences 135 (2018) 190-202. https://doi.org/10.1016/j.ijmecsci.2017.11.001