CFD Letters 15, Issue 5 (2023) 1-17

SEMARAK ILMU PUBLISHING
202103268166(003316878-P)

CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

Current Review on Numerical Simulation of Synthetic Jet (SJ) Cooling

Nawal Radhiah[1], S.M. Firdaus[1,*], Azmi Yusof[1]

[1] Advanced Mechanics Research Group, Centre of Mechanical Engineering Studies, College of Engineering, Universiti Teknologi MARA Cawangan Pulau Pinang 13500 Permatang Pauh, Penang, Malaysia

ARTICLE INFO

Article history:
Received 5 September 2022
Received in revised form 7 October 2022
Accepted 9 November 2022
Available online 1 May 2023

Keywords:
Synthetic jet; cooling; heat transfer

ABSTRACT

This paper reviews the working principles and numerical approach for the synthetic jet cooling in the previous studies. It is essential to reviews relevant previous research within the study scope in order to further enhance the improvement of synthetic jet technology. Furthermore, numerical approach can cut the cost and save time compared to experimental works. Numerical simulation is crucial to expedite the synthetic jet product enhancement as it opens up big potential in electronic device application. Studies carried out by many scholars within the scope of this publication have demonstrated that resonance frequency enhances the performance of synthetic jet. Reynold numbers vary with frequency, and greater Reynold numbers produced higher heat transfer coefficients. The majority of researchers have also chosen the cylindrical cavity type because it offers superior velocity output, which improves cooling performance. According to previous studies, a smaller diameter leads to a higher velocity output and a higher Reynold number.

1. Introduction

The rapid growth of electronic and communication technologies, particularly for 5G Wireless Communication Technologies, has resulted in the widespread use of small and compact electronic devices. Due to its integrative and intelligent features, portable electronic systems that combine numerous intelligent functional units in a restricted space have gotten a lot of attention [1]. Modernized electronic equipment is made to be smaller, faster, and capable of doing more complex tasks. Due to their compact size, electrical devices have high power densities and operating temperatures thus lowering their performance. If the electronic device temperature rises, it will begin to degrade and then shorten its life to the greatest extent. The growth of electronic equipment necessitates a steady increase in heat transmission intensity [2].

Thermal management will be critical for all types of electronics goods in the coming decade as there is a link between an electrical component's performance, especially its lifespan or lifecycle, and its specific working temperature range [3]. In many tiny electronics, existing temperature management strategies are ineffective. Heat sinks, heat sinks with fans, and other classic cooling solutions have reached their limits. As a result, more powerful heat transfer improvement methods

* *Corresponding author.*
E-mail address: sh.firdaus@uitm.edu.my (S.M. Firdaus)

https://doi.org/10.37934/cfdl.15.5.117

or cooling techniques are needed to keep the temperature of electronic equipment within a safe range. Synthetic jet (SJ), a novel type of increased heat transfer technology, has progressively gained popularity as a solution to the above concerns with electronic device heat dissipation.

It was found that SJ had a maximum heat transfer coefficient (HTC) that was 11 times greater than natural convection [4]. Several cooling systems for removing excess heat have been developed in the past. Active cooling techniques and passive cooling techniques are the two types of cooling procedures. An external source of energy is required for active cooling method such as fan, and SJ. Passive cooling method, such as heat pipes and heat sinks, remove heat without using any extra energy sources. Due to its low cost, simple structure, light weight, and ease of installation, research into SJ heat transfer use has accelerated in the previous decade, making it more promising than other currently used technologies. It comprises of an actuated cavity with a narrow opening (orifice/slot) and a moving diaphragm under the effect of fluctuating voltage. It goes through two stages within one operation cycle: suction and ejection. The shape and size of the orifice or slot can affect the flow field and heat transfer performance [5].

The dimensions and form of the SJ’s cavity have an impact on the velocity produced. As it affects the average velocity, a shallow cavity has a negative impact on SJ performance [6]. The orifice's dimensions have a significant impact on the SJ's performance. When compared to the rectangular orifice, the velocity produced by the circular orifice has a larger peak centerline velocity along the stream-wise direction [7]. The average Nusselt number (Nu) rises when the nozzle to distance ratio (H/d) declines at all frequencies. With increasing frequency, the appropriate value of H/d drops. For small H/d, high frequency SJ were shown to be more effective than low frequency SJ at removing heat, whereas low frequency SJ were found to be more effective for bigger H/d [4]. The flow field of the SJ may be separated into three distinct flow area due to its transient feature which are near field region, transitional region and far field region [7]. The impulse of the vortex has significant impact on the reversed flow created near the orifice. Each vortex's impulse must be large enough to the forces connected with the reversed suction flow [8].

The mechanics of flow in this synthetic model can be well understood through simulation analysis in computational fluid dynamics (CFD), which lowers the cost and analysis time needed in experimental processes. The purpose of this research is to review the working principle and numerical technique of a synthetic jet in order to understand the critical parameters involved in enhancing heat transfer.

2. Working Principles

The research was done by Le Clainche [9] to forecast the spatiotemporal structures of SJ. The simulations were run with an axis-symmetric flow condition with a cylindrical chamber and circular jet nozzle. Higher-order dynamic mode decomposition was the method employed (HODMD). A data-driven approach called HODMD is used to examine data gathered throughout the time period to build the previous DMD expansion. Interpolation and extrapolation technique can contribute to precise computations of such expansion. His method has reduced memory requirements by 70%, which has decreased processing time.

The simulations by Chen *et al.,* [10] established the beginning temperature ratio for jet formation and the threshold value of a non-dimensional parameter at the transition from no jet to a distinct jet. His research has found that the linear thermos-acoustic theory-based reduced-order network model offers quick, precise forecasts of the thermos-acoustic instability inside the TAE in the frequency domain. Ziadé *et al.,* [11] conducted a numerical study to determine whether changes to the inner geometry of a SJ cause variations in flow behaviour at the exit. It can be seen that the cavity with the sharpest transition from orifice to cavity conveys the most momentum at the exit. Lehnen *et al.,* [12] came to the conclusion that the cavity depth had a considerable impact on the centreline velocity of the jet.

Numerical simulation of flow control over NASA hump with uniform blowing jet and SJ was performed by Tang *et al.,* [13]. On the hump, two active flow control methods are used to lessen the flow separation. According to calculations, the SJ with the same momentum coefficient as the uniform blowing jet is more efficient than the uniform blowing jet at reducing or removing the separation in the downstream section of the hump. In order to enhance the performance of a SJ, the shape of the heating surface and the signal of the diaphragm have been altered by Benayad *et al.,* [14]. Numerical analysis has been done to compare the basic and modified case. The obtained data indicated that the changed case's Nu increased by around 51% in comparison to the basic instance. A numerical study on the three stages of the vortex self-similarity evolution in an impinging synthetic jet has been done by Silva-Llanca *et al.,* [15]. His research indicate that the vortex pairs showed parametric dependence on the vortex-distance-to-radius ratio and self-similarity.

In a numerical study, Huang *et al.,* [16] investigated the HTC brought about by impinging a SJ flow onto a longitudinal fin employed in an electronic cooling system. The impacts of several factors, such as H/d and diaphragm movement frequency and amplitude, are noted. It was discovered that operation frequency increased the fin surfaces' average HTC almost linearly. The time average HTC on the fin surfaces is shown in Figure 1 together with variations in diaphragm amplitude. The calculation was performed again at various fixed jet operating frequencies between 153 Hz and 4880 Hz. As the amplitude of the jet grows at a constant operating frequency, the HTC tends to almost increase linearly. The HTC rises linearly with increasing diaphragm amplitude at a constant operating frequency. The work of Miró *et al.,* [17] intends to study the cooling properties of a SJ confined between two parallel isothermal plates with a forced temperature differential. The analysis of the conditions at the outlet has revealed that depending on the Reynolds number (Re), the outlet temperature is approximately 1/10 to 1/5 of the temperature difference between the hot and cold plates and a strong asymmetry appears in the flow between the ejection and suction phases.

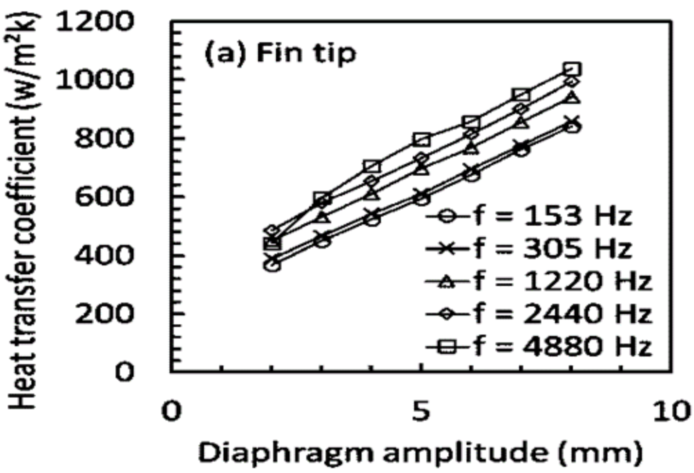


Fig. 1. Heat transfer coefficient vs diaphragm amplitude [16]

A special piston-type SJ with a quick-return characteristic structure has been explored by Liu *et al.,* [18]. The quick-return piston-type SJ has a considerable impact on the cavity pressure, jet velocity, mass flow, and momentum; the more pronounced this characteristic, the better the performance. Mohammadshahi *et al.,* [19] assesses the efficacy of a self-oscillating jet produced by a fluidic oscillator with two folds feedback for cooling films. The findings with various Re were examined, and it was discovered that while the mean Nu was about constant, raising Re causes the oscillating jet to occur more frequently. The Q-criterion calculation revealed strong vortices as Re rose, however because of the vortices' high velocity and shorter contact time with the hot plate, the heat transfer was not considerably altered. The oscillating jet may cover a bigger area and is better for blade cooling, according to comparisons made between it and a steady jet with a constant Re. Javadi *et al.,* [20] carried out numerical studies on impinging heat transfer in cross-flow with and without the influence of a vortex generator (VG). Due to the strong structures the VG has built, the flow may steer around the jet and ascend to the top wall of the channel. As a result, the VG lessens the negative impacts of cross-flow on heat transfer during impingement. Heat transport is improved by the location of the VG.

The features of transient heat transfer from circular air-jet impingement on a flat plate have been studied numerically by Guo *et al.,* [21]. For Re values between 14,000 and 53,000 at H/d = 4, numerical investigations were also carried out to determine the regional Nu distributions on the plate. The air jet started impinging at time t = 13 s, and the temperature at each site clearly dropped. The temperature gradually dropped as the air-jet impingement proceeded. When the air jet started to impinge, Nu at each site grew quickly. Re climbed while the Nu grew. At the point of standstill, the Nu grew somewhat until peaking. Nu is significantly impacted by the H/d at the stagnation zone.

Li *et al.,* [22] has performed a numerical investigation of how St, frequency, and Re affect the behaviours of a triangular SJ flow in terms of heat transfer. The main findings are that variations in St, or dimensionless stroke length, result in inconsistent frequency effects on heat transfer at various intervals. The amplitude of the area averaged Nu would typically increase with an increase in Re or frequency within one cycle. Due to the great coherence between heat and mass transmission, Re and frequency's respective effects on heat transfer may be traced back to their impact on the flow field. The flow field would be significantly altered by modifying either of the two parameters separately. The study's findings by Jacob *et al.,* [23] indicate that heat transmission increases as frequency rises. At low H/d, there is recirculation and an air entrainment effect, which has an impact on heat transferability. But this effect diminishes and uniform heat transmission takes place at large H/d and frequencies. Due to air recirculation, SJ transmit heat less effectively at very low H/d. As the diameter gets smaller, the heat transmission gets better.

The numerical simulations by d'Alencon *et al.,* [24] demonstrate that, regardless of the Re and H/d, the k SST was a trustworthy turbulence model for predicting the flow and heat transfer in conditions with the lower frequency. It was illustrated how a SJ would strike a uniformly heated wall. The varying vortex dynamics led to a change in the Nu distribution along the wall. Yeom *et al.,* [25] investigated the application of SJ arrays in a single channel of an air-cooled heat sink to enhance heat transmission. The increased proximity to the jet orifices, which enhances the effectiveness of the created vortex ring structures, was shown to make heat transfer in the fin tip region more susceptible to SJ impingement. The objective of the numerical analysis was to quantify the effects of system parameters on the efficiency of heat transfer over various regions of the channel (fin) wall. The channel side surfaces, which are further from the jet orifice, are subject to a reduced cooling impact from the SJ, according to the numerical study's findings. It is discovered that, when compared to a situation with cooling by channel through-flow alone, the SJ can increase locally-averaged HTC at the fin tip by up to 413 percent.

A numerical study of using high amplitude acoustic waves was conducted by Rulik and Wróblewski [26]. Two steps were used to split the works that were presented in his study. The structure and method of producing the flow field unsteadiness in the cavity were the sole things that were examined in the initial stage. The second stage of the project involved analysing the heat transfer in the cavity and figuring out how the parameters of the acoustic wave that was generated related to the heat transfer intensification process. The data collected demonstrate that the number of vortices coincides with the cavity maximal mode. It is discovered that the HTC distributions for the steady-state and the transient-state solutions differ significantly if the generated acoustic wave is characterised by a large amplitude. Lau *et al.,* [27] investigates the cooling and heat transport characteristics of Nano-fluids in a microchannel assembly using a single SJ. The results demonstrate improvement in efficiency of HTC and the cooling capacity of the microchannel. Images of the velocity field inside the microchannel with various Nano-fluids for the expulsion (t* = 10) and suction (t* = 10.5) stages are shown in Figure 2. The former represents the moment when the diaphragm is moving at its fastest rate toward the orifice, while the latter represents its fastest rate away from the opening. In all instances, during the expulsion stage, the fluid ejected from the orifice mixes with the cross flow in the microchannel to create a clockwise vortex just downstream. The deflection of the expelled streamlines from the SJ towards the direction of the microchannel flow leads to the creation of the vortex, which corresponds to flow mixing. The degree of flow mixing and vortex generation within the microchannel has a significant impact on how well heat transfer is enhanced. It is proven that the creation of vortices is essential for disturbing the thermal boundary layer at the micro channel's upper wall, which removes heat from the silicon wafer.

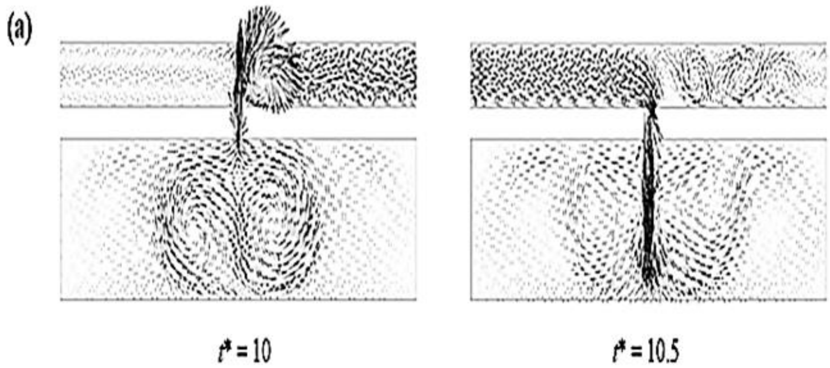


Fig. 2. Velocity field of expulsion (t* = 10) and suction (t* = 10.5) [24]

Huang *et al.,* [28] examines the impacts of the SJ, which is utilised in the study to control the tip leakage flow in a centrifugal compressor. It is placed at the shroud and close to the blade leading edge. The following conclusions can be taken from the current work: The tip leaking vortex trajectory is forced to bend in an upward manner by the SJ. The SJ placement should be close to the leading edge with a small injection angle in order to have a greater impact on the flow at the tip region close to the leading edge. The flow close to the leading edge becomes more stable when the SJ is used, and the dominant frequency is changed to the frequency at which the SJ is activated.

Kumar *et al.,* [29] investigates the mechanism of vortex creation and its bifurcation process. During the first axial switching, it has been noted that the vortex ring splits up close to the orifice exit and flows away from the centre of the SJ in the minor axis plane. In contrast, the vortex ring in the major axis plane is drawn toward the SJ's centreline and eventually vanishes. Haque *et al.,* [30] presents a numerical analysis of the effect of VG on flow through a heated block. For a range of Re (500–1050), it is discovered that the number of VG dramatically increases the total HTC. The degree of improvement is largely influenced by the location of the VG.

Three waveform of SJ were used in Zhang's [31] numerical research. He found that the rate of heat transmission will typically increase as Re rises. When St is close to the critical St, the transmission rate of the T-heat wave is higher than the S-wave and R-wave. The time-dependent inlet velocity profile for various waveforms is shown in Figure 3. The right portion of streamlines and velocity contours at crucial St in three waveforms is illustrated in Figure 4 to help better comprehend this phenomenon. The inlet velocity of an S-wave steadily increases from zero to the maximum value. For R-waves, the inlet velocity will quickly decrease to its lowest value and stay there throughout the period of rest. The behaviour of the vortices in a T-wave is comparable to an S-wave. It is clear from the velocity contour that the T-wave has a higher velocity than the S-wave. As a result, T-wave has a greater average Nu than S-wave and R-wave as shown in Figure 5. It can be observed that the HTC is strongly influenced by the waveform.

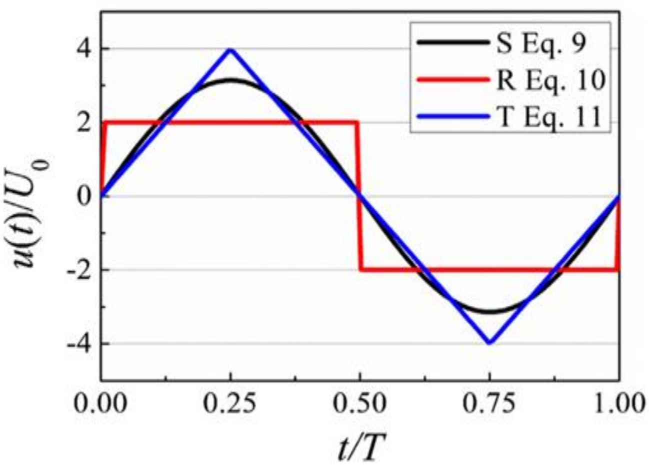


Fig. 3. Velocity profiles for different waveform [31]

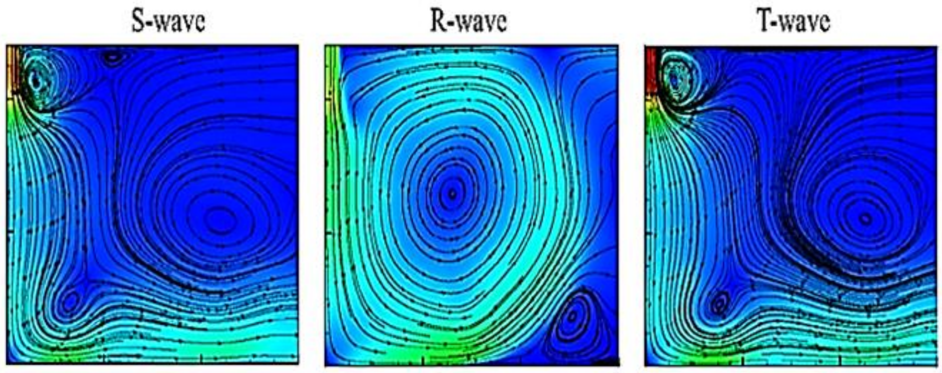


Fig. 4. Velocity contours for different waveform [31]

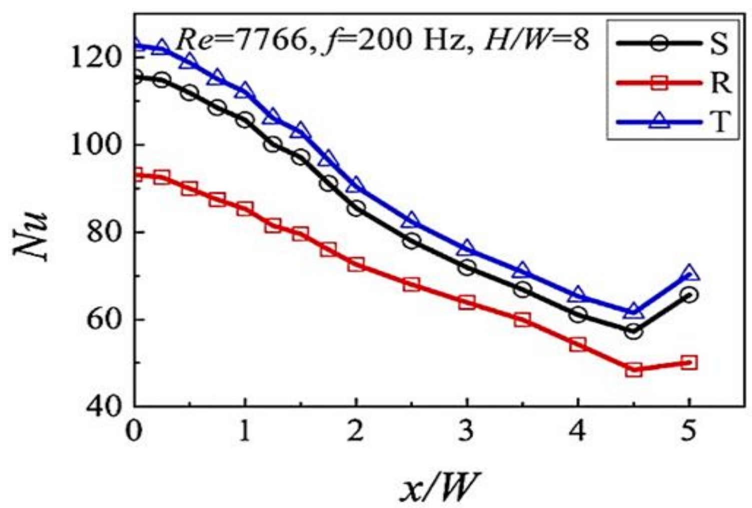


Fig. 5. Time-averaged Nusselt number for different waveform [31]

For an orifice with a variety of opening angles, numerical predictions are made by Husin *et al.,* [32]. The findings demonstrated that a diffuser-shaped orifice improves heat transfer at larger vibration amplitudes and that an opening angle of 90° is superior compared to 45°. Additionally, an increase in SJ cooling performance may not be possible for vibration amplitude increments greater than 0.6mm. The effect of SJ interaction with crossflow in channel on the cooling of heat wall is investigated by Xiang *et al.,* [33]. The interaction of the SJ with the channel flow enhances heat dissipation at the heated wall. In the jet hole, the downstream region's heat transmission has improved while the upstream region's heat transfer has deteriorated. According to research by Rhakasywi *et al.,* [34] heat transport in a heat sink with an impinging SJ may be mathematically calculated in three dimensions using several sinusoidal and non-sinusoidal wave excitation modes. The results of this investigation showed that when the SJ used a 120 Hz square wave, the impinging SJ applied to the heat sink revealed the optimum heat transfer capacity for cooling. It is clear from the Figure 6 that compared to the frequencies of 80 Hz and 160 Hz, the square wave excitation mode with a frequency of 120 Hz produces a greater Nu. According to estimates, air exchange into and out of the SJ cavity can occur for a long enough amount of time during vibrations that happen at a frequency of 120 Hz.

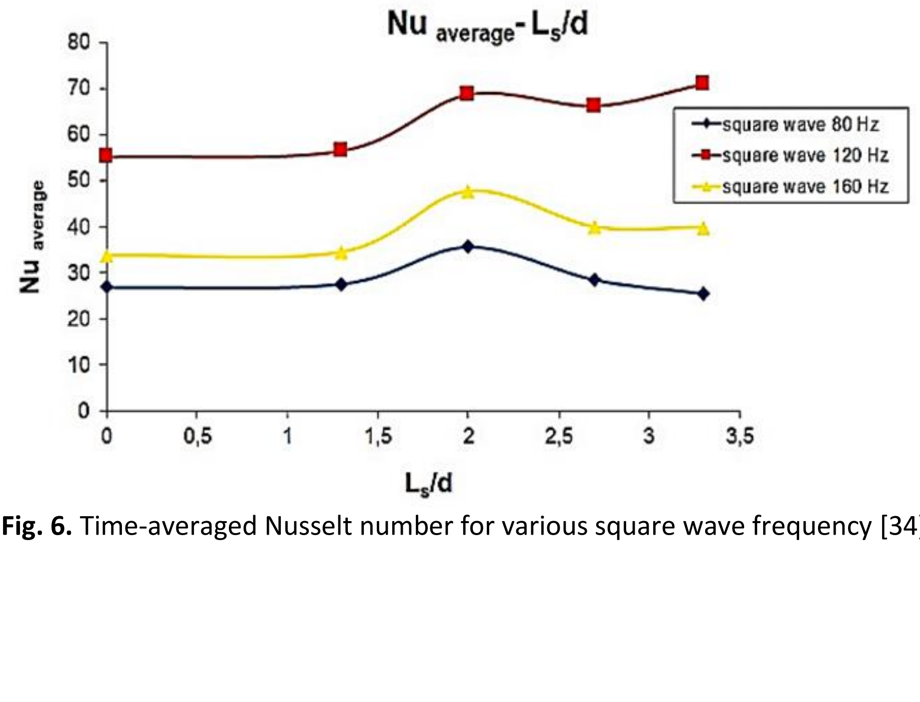


Fig. 6. Time-averaged Nusselt number for various square wave frequency [34]

The effects of jet stream injection from a channel's bottom wall on the vibration and heat transfer of a cylinder with transverse motion in a low Re zone have been the subject of a numerical investigation by Rabiee *et al.,* [35]. The findings demonstrate that for slots near to the cylinder, the vibrations of the cylinder begin to significantly decrease from an injection velocity onward, which is associated with a decrease in the force coefficients applied to the cylinder. The entire Nu reaches its minimum value at a certain jet injection velocity that is controlled by the cylinder-slot distance and the channel's input flow velocity. Zhu *et al.,* [36] used direct numerical simulation to study the flow via a slotted cylinder. The change in flow direction or the variability in flow rate in slits is caused by the alternating vortex shedding and varying pressure distribution. The cylinder with a straight transverse slit has the best control, taking the slit's inner wall into account. The decreases in drag and lift at Re=100 is 1.7% and 17%, respectively.

Qiu *et al.,* [37] has numerically examined the flow and heat transmission properties in a microchannel outfitted. A hairpin vortex makes up the majority of the vortex structure. The legs of the hairpin vortex lengthen as it moves downstream and combine with a pair of large-scale longitudinal vortices. The parametric investigation shows that while H/d determines the transient heat transfer performance, Re mostly influences the improvements of the time-area-averaged Nu and the overall pressure drop. Qayoum *et al.,* [38] has carried out comprehensive computational research on the effect of orifice on the optimum excitation frequency of a single cavity with different orifices. A piezoelectric actuator's SJ velocity is greatest when it is operating at its resonance frequency. The H/d at close ranges for single rectangular-slot and single-hole orifices have a greater impact on the SJ actuator. Each SJ quickly combines with the central one in the case of the several synthetic jets, resulting in a combined jet with a sizable momentum flux.

Krishan *et al.,* [39] has extensively researched the impact of sidewalls for continuous jets. With the use of Large Eddy Simulations (LES), a slot SJ is examined in order to comprehend how sidewalls affect the behaviour of the SJ flow-field. In order to prevent flow entrainment over the span-wise axis of the jet, the sidewalls are attached to the shorter side of the slot and extend in the stream-wise direction of the jet. The SJ's flow-field has undergone a considerable shift as a result of the sidewalls' presence. Due to the sidewalls, one of the unique characteristics of axis-switching in slot SJ flow field is reduced. Further simulations showed that the continuous contact of the vortex with the wall boundary layer in the span-wise plane causes the plane SJ to expand slowly in comparison to the free SJ in the lateral plane. The sidewalls' existence would have a major impact on the SJ's ability to transmit heat because they have a considerable impact on its flow-field in the context of the augmentation of heat transfer by SJ impingement.

Tan *et al.,* [40] presents a study for increasing the impingement heat transfer of array continuous jets by including a SJ into the array unit. A particular hybrid-jet impingement arrangement with a square-layout continuous-jet array and a centrally located SJ actuator is the subject of his numerical research. As a result of the localised heat transfer enhancement at the central zone of the array unit on the coordination mechanism of active pulsating SJ and passive pulsated wall jet, the results demonstrate that the integrity of a central synthetic jet in a continuous-jet array is indeed an effective way to improve the overall heat transfer performance. The heat transmission in the continuous-jet stagnation zones is essentially unaffected by the presence of a SJ at the continuous-jet array centre. However, this integrated SJ effectively enhances the local heat transfer in the central zone of a square configuration array unit, which enhances both the heat transfer uniformity and the overall heat transfer. The target surface mostly suffers from the periodic separate impact of the principal vortex rings at a close impinging distance. A trailing jet that is in front of the main vortex rings dominates the downward flow of a SJ while it is at a great impinging distance.

The results of numerical modelling by Sadaghiani *et al.,* [41] show that adding nanoparticles to the base fluid significantly changed velocity profiles and enhanced heat transfer. The thermal and hydrodynamic impacts of Nano-fluid flows were found to be greatly underestimated by the homogeneous (single-phase) model. Kashyap *et al.,* [42] researches the impact of secondary surfaces (SS) fixed over a rectangular VG while providing a summary of dynamic mode decomposition. When SS is present, the flow behind the VG significantly shears the generated primary vortex, causing it to stretch. Stretching reduces the span of the created vortices while increasing the angular momentum of the vortex. A greater tilt of the primary vortex toward the hot surface results from the addition of SS in z-axis. As a result, primary vortex bounces back more violently, which finally causes the helicity to fall. This demonstrates that primary vortex's slow propagation speed with the inclusion of SS.

Jacob *et al.,* [43] uses numerical analysis to determine the HTC of a SJ emerging from a circular orifice at H/d=6. The outcome demonstrates the importance of the H/d and sinusoidal wave frequencies on the measured heat transfer rate. Nu for stagnation rises with the Re for the tested range of values, as shown in Figure 7. By increasing the Re, the axial momentum increases and turbulence forms in the flow direction. As a result, the fluid's velocity increases along with the HTC. According to Mohammadpour *et al.,* [44], heat transfer increase for in-phase and 180° out-of-phase twin jets depends on orifice spacing. This arrangement caused a distinct pattern of vortex to emerge, which disrupted the main channel flow and encouraged rapid thermal mixing inside the channel. The biggest cooling improvement was achieved with the channel design configuration using two SJs running at 180° out-of-phase and 2 mm orifice spacing.

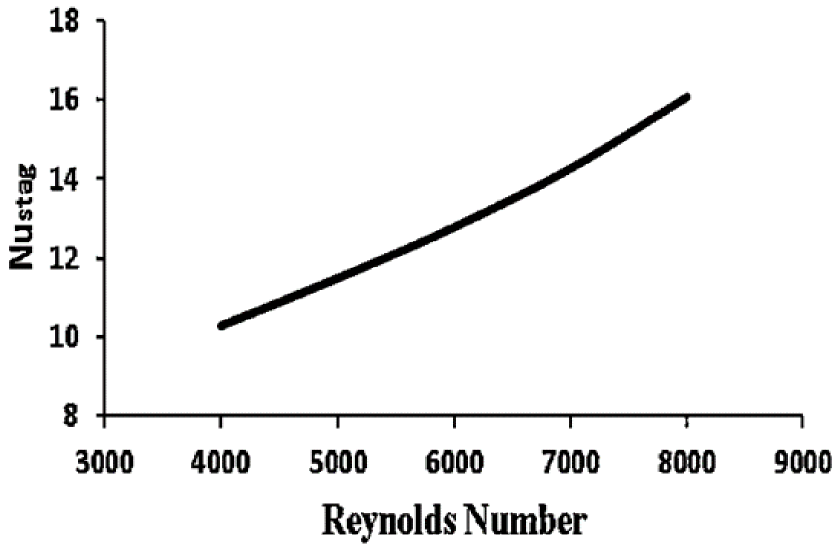


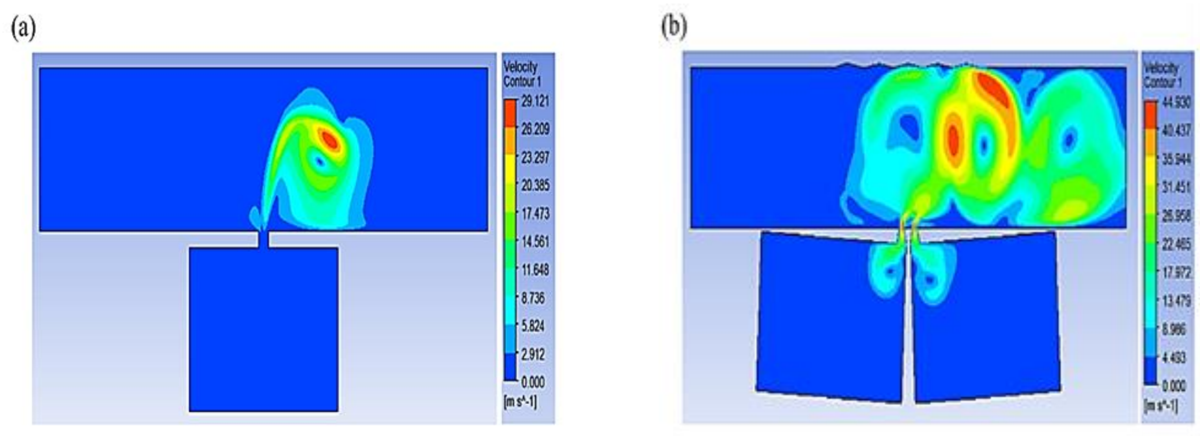
Fig. 7. Variation of stagnation Nusselt Number with Reynolds Number [43]

Geng *et al.,* [45] has found that when the nozzle diameter is small, the thrust is driven by velocity. While the thrust is acceleration-dominated when the diameter is large. There are two distinct patterns in the thrust fluctuation with nozzle height. The first variation in thrust pattern is dominated by the pressure force. The accelerating force has a greater impact on the second pattern of thrust fluctuation. The bidirectional SJ was studied by Fisher *et al.,* [46] in order to determine any potential benefits. One of the main conclusions of this study is that if the input (suction) and outflow (ejection) directions of a SJ are tuned individually, the performance of the jet might be greatly enhanced. The findings also indicate that the size of the separation bubble is not significantly affected by the various parabolic velocity profiles provided at the jet exit. The results, however, are dependent on the kind of jet exit conditions used. In the current research, Hoh *et al.,* [47] has constructed a CFD model of a fluid mixer supported by a SJ. The simulation outcomes demonstrate that this model can forecast the time-averaged mixing degree for a range of stroke ratio with a maximum difference of roughly 20% very effectively.

Lemanov *et al.,* [48] has found that the stagnation Nu rises at short H/d. The intensity of heat transfer is found to diminish as the distance is increased farther and as Re rises. Liu *et al.,* [49] suggests a formation criterion for the two-dimensional SJ. For the covered range of simulations, SJ may be divided into four stages by looking at the trend of normalised stroke distance against dimensionless parameter, Lam *et al.,* [50] study to determine the fluid flow and heat transfer characteristics of various discrete electronic components mounted on the bottom plate of the channel with and without a porous layer. The primary recirculation period's duration is shown to grow near to the jet intake as Re increases. The thermal boundary layer is found to be thickest along the secondary recirculation, which eventually causes overheating, while the thickness is thinnest in the stagnation region, creating a high Nu. The average Nu for all heat sources is observed to rise with rising Re and fall with decreasing H/d. The addition of a porous layer fully eliminates recirculation bubbles on heat sources.

Belova *et al.,* [51] has carried out computational studies of the impact of SJ on the flow in the model transition channel. Due to the huge diffuser expansion angle, a detachment flow forms in the channel, which is physically induced by a positive pressure gradient on the wall; Heat flux into the wall increases in the zone of separated flow; the highest heat flux is seen at the attachment point of the separation flow. The total pressure loss is decreased as the separation region's size is reduced. Li *et al.,* [52] has looked into the heat transmission and flow structure of periodic pulsating slot-jet impingement of Nano-fluids with rectangular wave and triangular wave. The findings show that T-wave is more effective at enhancing heat transmission. And on the subject of heat transfer performance, Re and show an excellent mutual promotion relationship.

Benayad *et al.,* [14] has modified the SJ with obstruction orifices and undulant heated wall with a slope and it is result indicate enhancement on the heat transfer due to increase of vortex formation in the flow domain. The magnitude velocity contours for the basic case with a periodic signal and the modified case with the bi-periodic signal are shown in Figure 8. Results are consistent with a jet amplitude of 75 mm and a transverse flow velocity of 1 m/s. The findings clearly demonstrate that the modified case's magnitude velocity is larger due to the bi-periodic signal's acting force and the creation of the jet vortex caused by the little inclined flow originating from the modified orifice's inclination. It is shown in Figure 9 that the case with modification gives higher values of Nu.



(a) (b)

Fig. 8. Velocity contour (a) Basic case (b) Modified case [14]

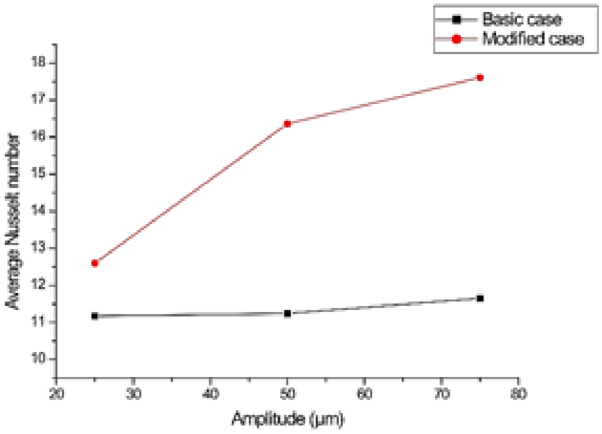


Fig. 9. Average Nusselt number comparison (a) Basic case (b) Modified case [14]

2.1 Frequency Analysis

According to Wang *et al.,* [53] research, the SJ excitation frequency significantly affects the control of two DOF VIVs. The cycles of the in-flow and cross-flow have many different amplitudes when excitation frequency below 4.0. Lehnen *et al.,* [12] came to the conclusion that the H/d and frequency have the most effects on the SJ ability to dissipate heat. It was discovered by Huang *et al.,* [16] that frequency increased the fin surfaces' average HTC almost linearly. At a fixed operating frequency, the HTC increases linearly with increasing diaphragm amplitude. Liu *et al.,* [18] conclude that in under any conditions, the suction time would get longer as the frequency rose, but the structure with rapid return can mitigate this effect.

The main findings by Li *et al.,* [22] are that variations in St, result in inconsistent frequency effects on heat transfer at various intervals. The amplitude of the area averaged Nu would typically increase with an increase in Re or frequency within one cycle. Due to the great coherence between heat and mass transmission, Re and frequency respective effects on heat transfer may be traced back to their impact on the flow field. The flow field would be significantly altered by modifying either of the two parameters separately. According to Li *et al.,* [22], Re affects the ideal frequency that results in the highest time-area averaged Nu. Most of the time, an increase in Re or frequency would demonstrate an increase in area averaged Nu during a period. The highest area averaged Nu's normalised occurrence moment at identical Re progressively lags with a frequency in the region of 10-35 Hz. The variation trend of the area averaged Nu with normalised time differs from Re even at the same frequency.

Three waveform SJ were used in Zhang's [31] numerical research. The rate of heat transmission will typically increase as Re rises. One can infer that low frequency is good at removing heat from a system at low Re, whereas high frequency is more efficient at high Re. Qayoum *et al.,* [38] has found that a piezoelectric actuator's SJ velocity is greatest when it is operating at its resonance frequency. The multi-orifice SJ’s optimum frequency is lower than the single-orifice SJ’s. Dong *et al.,* [54] has found that the primary clockwise vortex is favoured by low frequency and small amplitude, whereas the anticlockwise vortex is favoured by high frequency and large amplitude. Ahmad and Qayoum [55] has found that a SJ with a single orifice and a double orifice operates at its resonance frequency when its velocity reaches a maximum value to produce upward sweep vortices. The average HTC brought on by the double orifice SJ is 32% greater than the single orifice SJ at the resonance frequency and voltage. Li *et al.,* [52] has found that the frequency-induced trend suggests that as the frequency increases, the effect of frequency on the enhancement of heat transfer weakens.

3. Numerical Approach

An earlier attempt had been made to explore the SJ effect through numerical simulation research. In order to simulate the moving diaphragm of the jet as a moving wall boundary, user-defined function (UDF) has been used [29,40,45,52,55]. The governing equations of continuity, momentum and energy based on URANS (Unsteady Reynolds Averaged Navier-Stokes) has been mostly used [26,46,48,49,52,55].

Geng *et al.,* [45] and Hoh *et al.,* [47] have runs the simulations with ANSYS Fluent while take into account the flow as incompressible and in transient state. The pressure-velocity coupling scheme is chosen to be the pressure implicit with the splitting of operators (PISO). CFD software ANSYS Fluent with the Semi-Implicit Method for Pressure Linked Equations (SIMPLE) technique has been widely use among the researchers in order to simulate the incompressible flow. [16,25,29,40] In order to take into account turbulent and viscous effects, the k-omega SST model is mostly chosen to solve the mathematical model [10,16,19,25,26,40]. The turbulence model was used because it accurately depicts the flow in the vicinity of the wall and performs better when dealing with the large flow fluctuation [56]. Qiu *et al.,* [37] runs simulations in three dimensions, incompressible flow and constant physical attributes. In this study, the pressure-velocity coupling is handled by the SIMPLEC algorithm. Fisher *et al.,* [46] uses ANSYS FLUENT's PISO method for transient calculations and the SIMPLE technique for pressure-velocity coupling in steady-state simulations.

4. Discussions

Table 1, Table 2 and Table 3 summarize studies regarding synthetic jet application using numerical simulation concerning on frequency and Reynold numbers (Re), design parameter and method review. The majority of researchers employed various resonance frequency values, choosing the ones that were good for their diaphragms, according to the observation from summarised studies in Table 2. At its resonance frequency, where the diaphragm displays the greatest amplitude, a synthetic jet operates more effectively. The greatest amplitude of the diaphragm is achieved while using the resonance frequency. More fluid is trapped inside the volume chamber as a result of a higher amplitude producing a higher swept volume. As a result, the most air is evacuated at the fastest speed.

Other than that, Re is a crucial factor in fluid dynamics since it may be used to assess the flow conditions. The Re of a SJ varies with driving frequency and diaphragm amplitude. Given the ideal jet-to-surface separation, higher Re resulted in a higher HTC. The performance of the SJ is also impacted by the volume cavity design. According to Table 2, previous researchers frequently examine the cylindrical cavity as literature has found that the cylinder cavity provides a better velocity output. The orifice is a key element of the volume chamber, providing fluid access during the suction and ejection phases and having an open diameter. It is clear that researchers favoured circular orifice the most since they provided superior velocity profiles and enhanced cooling. When compared to the rectangular orifice, the SJ produced by the circular orifice has a larger peak centreline velocity along the stream-wise direction. In addition, smaller orifice’s diameter results in higher Re, according to a common finding in the literature.

It can be shown that the research procedures employed in the studies make use of simulation tools such CFD techniques for thermal analysis. Table 3 shows that ANSYS is a highly well-liked software platform that researchers favour for carrying out CFD flow studies. Due of its high accuracy in boundary layer simulations, experts advise using the shear stress transport (SST), thermal-fluid model, and K-Omega turbulence model while performing the CFD analysis. URANS (Unsteady Reynolds Averaged Navier-Stokes) solvers are also frequently employed. Researchers frequently utilise the Semi-Implicit Method for Pressure Linked Equations (SIMPLE) technique to model incompressible flow because it works well for unsteady flow with tiny time steps.

Table 1
Frequency and Reynold numbers

Reference	Frequency (Hz)	Reynold Number
Lehnen *et al.,* [12]	1, 4, 100, 200, 400	300,1200
Benayad *et al.,* [14]	10,000	5-20
Liu *et al.,* [18]	100-200	-
Li *et al.,* [22]	10,20,50	10,000 - 20,000
Zhang *et al.,* [29]	0, 10, 20, 50, 100, 200, 400	1553, 3883, 7767
Qayoum and Malik [35]	10,32,50	120,137,145
Lemanov *et al.,* [45]	1–200 Hz	-
Li *et al.,* [48]	10,20,75	10,000 - 20,000
Dong *et al.,* [50]	800-1000	-
Ahmad and Qayoum [51]	15,475, 951	247, 330

Table 2
Design parameter

Reference	Cavity	Orifice
Le Clainche [9]	Cylindrical	Circular
Kane *et al.,* [18]	Cylindrical	Circular
Guo *et al.,* [21]	Cylindrical	Circular
Kumar *et al.,* [27]	Cylindrical	Slot
Zhu *et al.,* [33]	Cylindrical	Slot
Qiu *et al.,* [34]	Cylindrical	Circular
Jacob *et al.,* [40]	Cylindrical	Circular
Ahmad and Qayoum [51]	Cylindrical	Circular

Table 3
Method review

Reference	CFD software	Pressure-velocity coupling scheme	Type of flow	Turbulence model
Chen *et al.,* [10]	ANSYS Fluent	PISO	-	k-omega SST
D' Alencon *et al.,* [11]	ANSYS Fluent	SIMPLE	Transient	conventional k-SST
Benayad *et al.,* [14]	ANSYS Fluent	-	Transient, incompressible	k-omega SST
Huang *et al.,* [16]	ANSYS Fluent	SIMPLE	Incompressible	k-omega SST
Mohammadshahi *et al.,* [19]	-	SIMPLE	Incompressible	k-omega SST
Li *et al.,* [22]	-	-	Transient, incompressible,	k-turbulence SST
Yeom *et al.,* [52]	ANSYS Fluent	SIMPLE	Incompressible	k-omega SST
Rulik and Wroblewski [24]	ANSYS Fluent		Transient, compressible	k-omega SST
Lau *et al.,* [25]	-	SIMPLE	Incompressible	-
Kumar *et al.,* [27]	ANSYS Fluent	SIMPLE	Transient, incompressible	-
Qiu *et al.,* [34]	-	SIMPLEC	Transient, incompressible	-
Tan *et al.,* [37]	ANSYS Fluent	SIMPLE	Incompressible	k-turbulence SST
Geng *et al.,* [42]	ANSYS Fluent	PISO	Transient, incompressible	k-omega SST
Fisher *et al.,* [43]	ANSYS Fluent	PISO, SIMPLE	Transient, steady	k-turbulence SST
Hoh *et al.,* [44]	ANSYS Fluent	PISO	Transient, incompressible	-

5. Conclusions

This research reported on the working principle and numerical approach selection. The main goal was to provide the researcher with a clear understanding of the direction of numerical analyses in synthetic jet applications. In order to properly analyse the efficacy of synthetic jet cooling, numerical modelling is essential. The nature of the flow field and the properties of the micro heat transfer were the main topics review in the study. Studies conducted by various researchers covered in the scope of this paper have shown that synthetic jet operates more effectively with resonance frequency. Reynold number are varied with the frequency and higher Reynold numbers resulted in higher heat transfer coefficient. Other than that, cylindrical cavity type has been chosen by most researchers as it provides better velocity output resulting to better cooling performance. Based on the previous researchers, smaller orifice's diameter result to higher velocity output thus led to higher Reynold number. Other than that, ANSYS is a highly well-liked software platform that academics favour for carrying out CFD flow studies. Finally, yet importantly the advantages of SJ are it is low cost, it has simple structure, lightweight and ease of installation.

Data Availability Statement (DAS)

The data that support the findings of this study are available from the corresponding author, N.R., upon reasonable request.

References

[1] Li, Wanwan, Fei Wang, Wenlong Cheng, Xi Chen, and Qing Zhao. "Study of using enhanced heat-transfer flexible phase change material film in thermal management of compact electronic device." *Energy Conversion and Management* 210 (2020): 112680. https://doi.org/10.1016/j.enconman.2020.112680

[2] Smyk, Emil, Paweł Gil, Rafał Gałek, and Łukasz Przeszłowski. "Acoustic and flow aspects of novel synthetic jet actuator." In *Actuators*, vol. 9, no. 4, p. 100. MDPI, 2020. https://doi.org/10.3390/act9040100

[3] Sharma, Hardik, Harsh Sharma, and Manish Mukhija. "Smart Door Lock Using Bluetooth." *Journal homepage: www. ijrpr. com ISSN* 2582: 7421.

[4] Jacob, Arun, Shafi KA, and K. E. Roy. "Experimental investigations on synthetic jet impingement heat transfer." *International Journal of Advanced Research in Engineering and Technology (IJARET)* 11, no. 5 (2020). https://doi.org/10.1016/j.ijft.2021.100104

[5] Arshad, Adeel, Mark Jabbal, and Yuying Yan. "Synthetic jet actuators for heat transfer enhancement–A critical review." *International Journal of Heat and Mass Transfer* 146 (2020): 118815. https://doi.org/10.1016/j.ijheatmasstransfer.2019.118815

[6] Smyk, Emil, Joanna Wilk, and Marek Markowicz. "Synthetic Jet Actuators with the Same Cross-Sectional Area Orifices-Flow and Acoustic Aspects." *Applied Sciences* 11, no. 10 (2021): 4600. https://doi.org/10.3390/app11104600

[7] Hong, Mun Hoh, See Yuan Cheng, and Shan Zhong. "Effect of geometric parameters on synthetic jet: A review." *Physics of Fluids* 32, no. 3 (2020): 031301. https://doi.org/10.1063/1.5142408

[8] Xu, Yang, Chanhee Moon, Jin-Jun Wang, Oleg G. Penyazkov, and Kyung Chun Kim. "An experimental study on the flow and heat transfer of an impinging synthetic jet." *International Journal of Heat and Mass Transfer* 144 (2019): 118626. https://doi.org/10.1016/j.ijheatmasstransfer.2019.118626

[9] Le Clainche, Soledad. "Prediction of the optimal vortex in synthetic jets." *Energies* 12, no. 9 (2019): 1635. https://doi.org/10.3390/en12091635

[10] Chen, Geng, Gopal Krishan, Yi Yang, Lihua Tang, and Brian Mace. "Numerical investigation of synthetic jets driven by thermoacoustic standing waves." *International Journal of Heat and Mass Transfer* 146 (2020): 118859. https://doi.org/10.1016/j.ijheatmasstransfer.2019.118859

[11] Ziadé, Paul, Mark A. Feero, and Pierre E. Sullivan. "A numerical study on the influence of cavity shape on synthetic jet performance." *International Journal of Heat and Fluid Flow* 74 (2018): 187-197. https://doi.org/10.1016/j.ijheatfluidflow.2018.10.001

[12] Lehnen, M. V., C. Y. Y. Lee, and F. L. D. Alves. "Nusselt number correlation for synthetic jets." *Journal of the Brazilian Society of Mechanical Sciences and Engineering* 38 (2016): 2161-2171. https://doi.org/10.1007/s40430-015-0337-

1
[13] Tang, Gongyu, and Ramesh K. Agarwal. "Numerical simulation of flow control over NASA hump with uniform blowing jet and synthetic jet." In *2018 Flow Control Conference*, p. 4017. 2018. https://doi.org/10.2514/6.2018-4017
[14] Benayad, Zouaoui, Samir Laouedj, and Abdelkader Filali. "Numerical investigation on the cooling of electronics components with synthetic multi-jets and non-sinusoidal bi-periodic forcing functions." *Energy Reports* 6 (2020): 1-9. https://doi.org/10.1016/j.egyr.2019.10.011
[15] Silva-Llanca, Luis, and Jean Paul d'Alençon. "Vortex self-similarity in an impinging synthetic jet and its three-stage evolution." *International Journal of Heat and Mass Transfer* 161 (2020): 120219. https://doi.org/10.1016/j.ijheatmasstransfer.2020.120219
[16] Huang, Longzhong, Taiho Yeom, Terrence Simon, and Tianhong Cui. "An experimental and numerical study on heat transfer enhancement of a heat sink fin by synthetic jet impingement." *Heat and Mass Transfer* 57 (2021): 583-593. https://doi.org/10.1007/s00231-020-02974-y
[17] Miró, A., M. Soria, C. Moulinec, J. C. Cajas, Y. Fournier, and France MFEE. "Numerical investigations on rectangular and circular synthetic jet impingement." In *Tenth International Conference on Computational Fluid Dynamics (ICCFD10)*, vol. 18. 2018.
[18] Liu, Zheng, and Liang Hong. "Numerical Study of a novel Piston-type synthetic jet actuator with a quick-return characteristic." In *IOP Conference Series: Materials Science and Engineering*, vol. 187, no. 1, p. 012030. IOP Publishing, 2017. https://doi.org/10.1088/1757-899X/187/1/012030
[19] Mohammadshahi, Shabnam, Hadi Samsam-Khayani, Tao Cai, and Kyung Chun Kim. "Experimental and numerical study on flow characteristics and heat transfer of an oscillating jet in a channel." *International Journal of Heat and Fluid Flow* 86 (2020): 108701. https://doi.org/10.1016/j.ijheatfluidflow.2020.108701
[20] Javadi, Ardalan. "Numerical study of an impinging jet in cross-flow within and without influence of vortex generator structures on heat transfer." *Heat and Mass Transfer* 56, no. 3 (2020): 797-810. https://doi.org/10.1007/s00231-019-02728-5
[21] Guo, Qiang, Zhi Wen, and Ruifeng Dou. "Experimental and numerical study on the transient heat-transfer characteristics of circular air-jet impingement on a flat plate." *International Journal of Heat and Mass Transfer* 104 (2017): 1177-1188. https://doi.org/10.1016/j.ijheatmasstransfer.2016.09.048
[22] Li, Ping, Xinyue Huang, and Dingzhang Guo. "Numerical analysis of dominant parameters in synthetic impinging jet heat transfer process." *International Journal of Heat and Mass Transfer* 150 (2020): 119280. https://doi.org/10.1016/j.ijheatmasstransfer.2019.119280
[23] Jacob, Arun, K. A. Shafi, and KE Reby Roy. "Heat transfer characteristics of piston-driven synthetic jet." *International Journal of Thermofluids* 11 (2021): 100104. https://doi.org/10.1016/j.ijft.2021.100104
[24] d'Alençon, Jean Paul, and Luis Silva-Llanca. "Two-dimensional numerical analysis of a low-re turbulent impinging synthetic jet." In *2016 15th IEEE Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems (ITherm)*, pp. 921-929. IEEE, 2016. https://doi.org/10.1109/ITHERM.2016.7517644
[25] Yeom, Taiho, Longzhong Huang, Min Zhang, Terrence Simon, and Tianhong Cui. "Heat transfer enhancement of air-cooled heat sink channel using a piezoelectric synthetic jet array." *International Journal of Heat and Mass Transfer* 143 (2019): 118484. https://doi.org/10.1016/j.ijheatmasstransfer.2019.118484
[26] Rulik, Sebastian, and Włodzimierz Wróblewski. "A numerical study of the heat transfer intensification using high amplitude acoustic waves." *Archives of Acoustics* 43, no. 1 (2018): 31-47.
[27] Lau, G. E., J. Mohammadpour, and A. Lee. "Cooling performance of an impinging synthetic jet in a microchannel with nanofluids: an Eulerian approach." *Applied Thermal Engineering* 188 (2021): 116624. https://doi.org/10.1016/j.applthermaleng.2021.116624
[28] Huang, Wenlin, Huijing Zhao, Zhiheng Wang, Guang Xi, and Haijun Liu. "Numerical study on interaction of tip synthetic jet with tip leakage flow in centrifugal impeller." In *Turbo Expo: Power for Land, Sea, and Air*, vol. 51005, p. V02BT44A019. American Society of Mechanical Engineers, 2018. https://doi.org/10.1115/GT2018-76330
[29] Kumar, Abhay, Ramesh Donga, and Ashish Karn. "A Numerical Study of the Bifurcation of Rectangular Synthetic Jets." In *Proceedings of 46th National Conference on Fluid Mechanics and Fluid Power (FMFP) December*, pp. 9-11. 2019. https://doi.org/10.2139/ssrn.3475910
[30] Haque, Mohammad Rejaul, and Amy Rachel Betz. "Heat Transfer Enhancement by Insertion of Vortex Generators in Electronic Chip Cooling: A Numerical Study." In *International Conference on Nanochannels, Microchannels, and Minichannels*, vol. 58301, p. V001T05A001. American Society of Mechanical Engineers, 2017. https://doi.org/10.1115/ICNMM2017-5523
[31] Zhang, Yanyao, Ping Li, and Yonghui Xie. "Numerical investigation of heat transfer characteristics of impinging synthetic jets with different waveforms." *International Journal of Heat and Mass Transfer* 125 (2018): 1017-1027. https://doi.org/10.1016/j.ijheatmasstransfer.2018.04.120

[32] Husin, Azmi, Mohd Zulkifly Abdullah, Azmi Ismail, Ayub Ahmed Janvekar, Mohd Syakirin Rusdi, and Wan Mohd Amri Wan Mamat Ali. "Heat Transfer Performance of a Synthetic Jet Generated by Diffuser-Shaped Orifice." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 53, no. 1 (2019): 122-128.
[33] Xiang, Liping, Yueyue Liu, Kunrong Huang, and Yi Huang. "Influence of synthetic jet in crossflow configuration on heat transfer enhancement." In *Journal of Physics: Conference Series*, vol. 1633, no. 1, p. 012032. IOP Publishing, 2020. https://doi.org/10.1088/1742-6596/1633/1/012032
[34] Rhakasywi, Damora, Engkos A. Kosasih, and Ridho Irwansyah. "Computational and Experimental Study of Heat Transfer on the heat sink with an impinging synthetic jet under Various Excitation Wave." *Case Studies in Thermal Engineering* 26 (2021): 101106. https://doi.org/10.1016/j.csite.2021.101106
[35] Rabiee, Amir Hossein, and Somayeh Davoodabadi Farahani. "Effect of synthetic jet on VIV and heat transfer behavior of heated sprung circular cylinder embedded in a channel." *International Communications in Heat and Mass Transfer* 119 (2020): 104977. https://doi.org/10.1016/j.icheatmasstransfer.2020.104977
[36] Zhu, Hongjun, Honglei Zhao, and Tongming Zhou. "Direct numerical simulation of flow over a slotted cylinder at low Reynolds number." *Applied Ocean Research* 87 (2019): 9-25. https://doi.org/10.1016/j.apor.2019.01.019
[37] Qiu, Yun-Long, Wen-Jie Hu, Chang-Ju Wu, and Wei-Fang Chen. "Flow and heat transfer characteristics in a microchannel with a circular synthetic jet." *International Journal of Thermal Sciences* 164 (2021): 106911. https://doi.org/10.1016/j.ijthermalsci.2021.106911
[38] Qayoum, A., and A. Malik. "Influence of the excitation frequency and orifice geometry on the fluid flow and heat transfer characteristics of synthetic jet actuators." *Fluid Dynamics* 54 (2019): 575-589. https://doi.org/10.1134/S0015462819040086
[39] Krishan, Gopal, K. C. Aw, and Rajnish N. Sharma. "A Numerical Investigation of the Influence of Sidewalls on the Flow-field of a Slot Synthetic jet." In *21st Australian Fluid Mechanics Conference*, pp. 1-4. 2018.
[40] Tan, Jun-wen, Yuan-wei Lyu, Jing-zhou Zhang, and Wen-jing Sun. "Heat Transfer Enhancement of Array Continuous-Jet Impingement with an Integrated Central Synthetic Jet."
[41] Sadaghiani, Abdolali Khalili, Mehmet Yildiz, and Ali Koşar. "Numerical modeling of convective heat transfer of thermally developing nanofluid flows in a horizontal microtube." *International Journal of Thermal Sciences* 109 (2016): 54-69. https://doi.org/10.1016/j.ijthermalsci.2016.05.022
[42] Kashyap, Uddip, Ashish P. Pawar, Sandip Sarkar, and Sandip K. Saha. "Experimental and numerical study of effect of secondary surfaces fixed over rectangular vortex generator with an overview of dynamic mode decomposition." *Physics of Fluids* 32, no. 5 (2020): 057101. https://doi.org/10.1063/5.0004044
[43] Jacob, Arun, R. Anandhu, R. Leena, and K. A. Shafi. "Experimental and Numerical Investigations of Synthetic Jet Used for Electronic Cooling." In *Journal of Physics: Conference Series*, vol. 1355, no. 1, p. 012008. IOP Publishing, 2019. https://doi.org/10.1088/1742-6596/1355/1/012008
[44] Mohammadpour, J., G. E. Lau, S. Cheng, and A. Lee. "Thermal performance of a pair of synthetic jets equipped in microchannel." *Continuity* 1 (2021): 0.
[45] Geng, Lingbo, Zhiqiang Hu, and Yang Lin. "Numerical investigation of the influence of nozzle geometrical parameters on thrust of synthetic jet underwater." *Sensors and Actuators A: Physical* 269 (2018): 111-125. https://doi.org/10.1016/j.sna.2017.11.018
[46] Fisher, Rodrigo, Takafumi Nishino, and Mark Savill. "Numerical analysis of a bidirectional synthetic jet for active flow control." *AIAA Journal* 55, no. 3 (2017): 1064-1069. https://doi.org/10.2514/1.J055081
[47] Hoh, Hong Mun, Cheng See Yuan, and Lim Kim Chuan. "Numerical Modelling of Synthetic-Jet-Assisted Mixing." *CFD Letters* 11, no. 4 (2019): 16-31.
[48] Lemanov, V. V., M. A. Pakhomov, V. I. Terekhov, and Z. Travnicek. "Non-stationary convective heat transfer in an air synthetic impinging jet. Experiment and numerical simulation." In *Journal of Physics: Conference Series*, vol. 2119, no. 1, p. 012024. IOP Publishing, 2021. https://doi.org/10.1088/1742-6596/2119/1/012024
[49] Liu, Zhiyong, Zhenbing Luo, Qiang Liu, Xiong Deng, and Wenqiang Peng. "Self-support phenomenon and formation characteristics of dual synthetic jet." *Sensors and Actuators A: Physical* 299 (2019): 111597. https://doi.org/10.1016/j.sna.2019.111597
[50] Lam, Prasanth Anand Kumar, and K. Arul Prakash. "A numerical investigation of heat transfer and entropy generation during jet impingement cooling of protruding heat sources without and with porous medium." *Energy Conversion and Management* 89 (2015): 626-643. https://doi.org/10.1016/j.enconman.2014.10.026
[51] Belova, V. G., V. A. Stepanov, and A. Yu Chirkov. "The effect of synthetic jets on heat fluxes in a transitional channel with flow separation." In *Journal of Physics: Conference Series*, vol. 1129, no. 1, p. 012005. IOP Publishing, 2018. https://doi.org/10.1088/1742-6596/1129/1/012005
[52] Li, Ping, Dingzhang Guo, and Ruirui Liu. "Mechanism analysis of heat transfer and flow structure of periodic pulsating nanofluids slot-jet impingement with different waveforms." *Applied Thermal Engineering* 152 (2019): 937-945. https://doi.org/10.1016/j.applthermaleng.2016.09.151

[53] Wang, Haibo, Lin Ding, Li Zhang, Rajnish N. Sharma, and Lin Yang. "Numerical study on two-degree-of-freedom vortex induced vibrations suppression of a circular cylinder via synthetic jets at different excitation frequencies." *International Journal of Heat and Fluid Flow* 84 (2020): 108593. https://doi.org/10.1016/j.ijheatfluidflow.2020.108593

[54] Dong, Xiangrui, Chunyang Hao, Yinlin Dong, Chaoqun Liu, and Yalu Li. "Investigation of vortex motion mechanism of synthetic jet in a cross flow." *AIP Advances* 12, no. 3 (2022): 035045. https://doi.org/10.1063/5.0086084

[55] Ahmad, Mukhtar, and Adnan Qayoum. "Investigation of Impingement of Double Orifice Synthetic Jet for Heat and Fluid Flow Characteristics in Quiescent Flow." *Pertanika Journal of Science & Technology* 27, no. 3 (2019).

[56] Lacombe, Francis, Dominique Pelletier, and Andre Garon. "Compatible wall functions and adaptive remeshing for the k-omega SST model." In *AIAA Scitech 2019 Forum*, p. 2329. 2019. https://doi.org/10.2514/6.2019-2329