Cooling performance improvement of impingement hybrid synthetic jets in a confined space with the aid of a fluid diode

Yu, Qinghua; Mei, Ziyue; Bai, Mengqi; Xie, Danmei; Ding, Yulong; Li, Yongliang

DOI:

[10.1016/j.applthermaleng.2019.113749](https://doi.org/10.1016/j.applthermaleng.2019.113749)

License:

Creative Commons: Attribution-NonCommercial-NoDerivs (CC BY-NC-ND)

Document Version

Peer reviewed version

Citation for published version (Harvard):

Yu, Q, Mei, Z, Bai, M, Xie, D, Ding, Y & Li, Y 2019, 'Cooling performance improvement of impingement hybrid synthetic jets in a confined space with the aid of a fluid diode', *Applied Thermal Engineering*, vol. 157, 113749. <https://doi.org/10.1016/j.applthermaleng.2019.113749>

[Link to publication on Research at Birmingham portal](#)

Publisher Rights Statement:

Checked for eligibility: 24/05/2019

General rights

Unless a licence is specified above, all rights (including copyright and moral rights) in this document are retained by the authors and/or the copyright holders. The express permission of the copyright holder must be obtained for any use of this material other than for purposes permitted by law.

- Users may freely distribute the URL that is used to identify this publication.
- Users may download and/or print one copy of the publication from the University of Birmingham research portal for the purpose of private study or non-commercial research.
- User may use extracts from the document in line with the concept of 'fair dealing' under the Copyright, Designs and Patents Act 1988 (?)
- Users may not further distribute the material nor use it for the purposes of commercial gain.

Where a licence is displayed above, please note the terms and conditions of the licence govern your use of this document.

When citing, please reference the published version.

Take down policy

While the University of Birmingham exercises care and attention in making items available there are rare occasions when an item has been uploaded in error or has been deemed to be commercially or otherwise sensitive.

If you believe that this is the case for this document, please contact UBIRA@lists.bham.ac.uk providing details and we will remove access to the work immediately and investigate.

**Cooling performance improvement of impingement hybrid
synthetic jets in a confined space with the aid of a fluid diode**

Qinghua Yu^{a,b,*}, Ziyue Mei^b, Mengqi Bai^a, Danmei Xie^b, Yulong Ding^a, Yongliang Li^{a, #}

^a Birmingham Centre for Energy Storage, School of Chemical Engineering, University of
Birmingham, Birmingham B15 2TT, United Kingdom

^b Key Laboratory of Hydraulic Machinery Transients (Wuhan University), Ministry of
Education, Wuhan 430072, China

* Corresponding author. E-mail address: yqh2015@hotmail.com (Q. Yu)

[#] Corresponding author. E-mail address: Y.Li.1@bham.ac.uk (Y. Li)

Abstract

For the conventional synthetic jet in a confined space, the recirculation of heated air at a small nozzle-to-surface spacing results in deterioration of its impingement cooling ability. To overcome this drawback, various novel fluid diodes are designed and introduced to form hybrid synthetic jets in this paper. Numerical analyses are carried out to reveal the cooling performance enhancement mechanism of the hybrid synthetic jets with the aid of fluid diodes. Instantaneous images of velocity contour and streamline at one cycle are comparatively presented to elaborate on their flow characteristics. Based on the simulation results, the volumetric efficiency, air temperature at the nozzle exit, average temperature on the heated surface and average Nusselt number are further calculated to quantify the improvements of the proposed hybrid synthetic jets compared to the conventional synthetic jet. The hybrid synthetic jet with a convergence nozzle generates the best cooling ability among various configurations. This study provides significant guidance for the design of novel hybrid synthetic jets with high-efficiency impingement cooling performance.

Keywords: Impingement cooling; Synthetic jet; Fluid diode; Heat transfer; Confined space.

Nomenclature

Roman letters

A	amplitude (μm)
D_c	cavity diameter (mm)
D_h	heated surface diameter (mm)
D_i	inlet outer diameter (mm)
D_n	nozzle exit diameter (mm)
D_o	orifice diameter (mm)
f	oscillation frequency (Hz)
h_c	heat transfer coefficient ($\text{W}\cdot\text{m}^{-2}\cdot\text{K}^{-1}$)
H	nozzle-to-surface spacing (mm)
H_c	cavity height (mm)
H_i	inlet chamber height (mm)
H_n	nozzle length (mm)
H_o	orifice plate thickness (mm)
L	whole target surface diameter (mm)
Nu	Nusselt number
q	heat flux on heated surface ($\text{W}\cdot\text{m}^{-2}$)
Re	Reynolds number
t	time (s)
T	oscillation period of diaphragm (s)
$u(t)$	average velocity at the orifice ($\text{m}\cdot\text{s}^{-1}$)
U_e	equivalent velocity at the orifice ($\text{m}\cdot\text{s}^{-1}$)

u_{max}	maximum ejecting velocity at the nozzle exit during one cycle ($\text{m}\cdot\text{s}^{-1}$)
$U(t)$	moving speed of diaphragm ($\mu\text{m}\cdot\text{s}^{-1}$)
V_e	air volume ejected out through the nozzle exit in one cycle (m^3)
V_s	air volume sucked back through the nozzle exit in one cycle (m^3)
$X(t)$	displacement of diaphragm (μm)

Greek letters

η_v	volumetric efficiency
θ	local time-averaged temperature ($^{\circ}\text{C}$)
λ	thermal conductivity ($\text{W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$)
μ	dynamic viscosity ($\text{Pa}\cdot\text{s}$)
ρ	density ($\text{kg}\cdot\text{m}^{-3}$)

Subscripts

a	ambient air
avg	average
c	heated surface center
s	heated surface

1. Introduction

With the developing trends of electronic devices toward miniaturization and high degree of integration, high thermal power density and small space make effective heat removal become more challenging in the design of electronic systems [1]. Heat accumulation will lead to the failure of electronic devices and shorten their service life [2, 3]. Ingenious cooling schemes are absolutely essential to achieving effective thermal management of electronic devices to keep their temperatures within a suitable range. A promising cooling technology, called an impingement synthetic air jet, has gradually attracted widespread attention [4-7], due to its obvious advantages as follows compared with conventional cooling methods, such as natural convection, fan, steady jet: (1) a synthetic jet exhibits higher heat transfer capacity, which can be 4.5-10 times [8], 1.2-1.4 times [9] and 1.4 times [10] higher than those of natural convection, fan and steady jet, respectively; (2) it has no rotating parts, which are conducive to noise reduction, low energy consumption, stability, safety and reliability; (3) it does not need fluid pipe and complex pumping, offering a compact structure and ease of integration. Based on the above advantages, an impingement synthetic air jet is considered to be a more efficient manner for cooling of electronic devices, especially in limited space.

An impingement synthetic air jet is commonly generated by a cavity which contains a small orifice on one sidewall and a periodically oscillating diaphragm mounted on the other sidewall [11]. In virtue of the periodic oscillation of the diaphragm with a certain frequency, the surrounding air is alternately sucked into and ejected from the cavity through the orifice. During the suction stroke, the near-wall low momentum air is entrained into the cavity through the orifice acting as a sink. During the ejection stroke, a coupled vortex pair is generated from the orifice in each cycle and moves toward a heated surface which needs to be cooled. There is no net mass addition but net momentum is injected into the surrounding

stagnant air. During continuous sucking-ejecting cycles, a fluid jet comprising a train of vortex pairs is formed and consecutively impinges to the heated surface. Hence, the resulting impingement synthetic jet can remove the heat and cool the heated surface. The synthetic jet velocity at the orifice can be estimated by a pressure-based model [12], a lumped element model [13] and a general reduced-order model [14]. The Reynolds number of synthetic jets can be expressed as [15]

$$Re = \frac{\rho U_e D_o}{\mu} = \frac{\rho D_o L_o}{\mu T} \quad (1)$$

where ρ denotes the air density; U_e denotes the equivalent average velocity at the orifice of the cavity; D_o denotes the orifice diameter; μ denotes the air dynamic viscosity; T is the oscillation period of the diaphragm; and L_o is the stroke length, which is defined as

$$L_o = \int_0^{T/2} u(t) dt \quad (2)$$

where $u(t)$ is the time-dependent average velocity at the orifice of the cavity.

Much work has been devoted to the heat transfer characteristics and cooling performance improvement of impingement synthetic air jets. Chaudhari et al. [16] experimentally performed a comprehensive parametric study on heat transfer characteristics of impingement synthetic air jets, including oscillation frequency ($f=100-350$ Hz), orifice diameter ($D_o = 5-14$ mm), and nozzle-to-surface spacing ($H/D_o = 1-25$). The heat transfer ability of impingement synthetic jets achieved the maximum at the resonance frequency ($f=250$ Hz) of the cavity. Small nozzle-to-surface spacing ($H/D_o < 6$) strengthened the confinement effect of the orifice plate and thus got more heated air recirculated between the orifice plate and the heated surface, causing a significant reduction in the heat transfer ability. They subsequently examined the effects of orifice shape [17] and numbers [18]. It was reported that a rectangular orifice with an aspect ratio of 3-5 provided better performance than a square or circular orifice at $H/D_o \leq 5$. The configuration of a center orifice surrounded

by multiple satellite orifices gave better heat transfer performance than the conventional configuration of a single center orifice. Lee et al. [19] numerically studied the interaction of multiple synthetic jets with cross-flow in a micro-channel for the cooling microchip. Double synthetic jets promote the flow mixing in the micro-channel and thus enhance the heat transfer. Rylatt et al. [20] proposed to add ducting to enhance heat transfer of a confined impinging synthetic air jet. Their experimental results indicated that the cold air from a remote location was drawn into the jet flow with the aid of ducting during the suction stroke and the heat transfer was thus improved. Nevertheless, it is not easy to fit ducting in such a small space. Bhapkar et al. [21] experimentally demonstrated that the cooling performance at a small nozzle-to-surface spacing ($H/D_o < 2$) was markedly affected by the orifice cavity confinement and designed various cavity shapes to provide different confinement conditions. The average Nusselt number for least confinement was 44% higher than that for maximum confinement at $H/D_o = 0.5$. However, the robustness of the improvement effect provided by the least confinement configuration was not good enough. Liu et al. [22] also reported the fundamental study concerning characteristics of flow and heat transfer of a synthetic air jet under different experimental parameters, such as actuation frequency (200-800 Hz), jet Reynolds number (500-1300), and nozzle-to-surface spacing ($H/D_o = 2-25$). It was found that the highest heat transfer coefficient was achieved using an actuation frequency of 600 Hz at a nozzle-to-surface spacing of 15. In addition, when the plate was close to the jet hole ($H/D_o < 5$), the heated air was obviously entrapped and recirculated within the small space, which consequently weakened heat transfer. They further proposed to adopt a diffusion-shaped orifice to enhance synthetic-jet heat transfer in their succeeding work [23]. It was stated that the diffusion-shaped orifice with an opening angle of 60° produced a highest heat transfer enhancement of approximately 30% with respect to a round orifice.

The above researches imply that the principal drawback of the cooling technology based on impingement synthetic jets is inevitable recirculation of the heated air in confined spaces especially when the nozzle-to-surface spacing is small, which leads to relatively poor cooling performance. Integrating fluidic diodes into the actuator of a conventional synthetic jet, which generates a so-called hybrid synthetic jet, is a potentially useful solution for this issue. With the rectification effect of fluidic diodes, the hybrid synthetic jet can entrain amounts of air from the cold surroundings far away from the heat exchange zone, i.e. introduce fresh air without preheating via fluidic diodes into the cooling process. The volumetric efficiency, which is defined as the ratio of the net output fluid volume ejected from the orifice over the whole period to the total fluid volume exchanged through the same orifice over the same period, is used to quantify the rectification performance of the actuators of hybrid synthetic jets [24]. Trávníček et al. [25] designed a hybrid synthetic jet actuator equipped with a pair of fluidic diodes shaped as conical ducts, which offered a volumetric efficiency of 13%. Subsequently, the volumetric efficiency was improved to 19% by introducing double-acting principle [26] and 70% by integrating two extremely complicated fluidic diodes [27]. Trávníček et al. [28] tested the impingement heat transfer performance of the hybrid synthetic jet actuator presented in [25] using the heat/mass transfer analogy in unconfined spaces. They reported that it offered a 26% larger flow rate than the conventional synthetic jet and thus resulted in an increase of 18% in the heat transfer rate. Hsu et al. [29] designed a double-acting hybrid synthetic jet and experimentally demonstrated that it offered higher vorticity compared with commonly used synthetic jets. They further improved the geometrical configuration of the double-acting hybrid synthetic jet actuators, achieving volumetric efficiencies of 76%~79% [30]. Although the impingement hybrid synthetic jet is a promising cooling technology, its detailed heat transfer characteristics and suppression effect on the recirculation of heated air in confined spaces remain largely unaddressed in the

literature. Further, the volumetric efficiency of impingement hybrid synthetic jets theoretically can be up to 100% like classical reciprocating pumps. Therefore, there is still plenty of room for improvement in volumetric efficiency.

This paper designs various novel hybrid synthetic jet actuators equipped with very simple fluidic diodes, aiming to achieve quite high volumetric efficiency and preferable impingement cooling performance. Since numerical simulations based on computational fluid dynamics (CFD) are able to provide precious physical insights and effective prediction without any time-consuming manufacturing [31-33], a full-domain numerical model of heat transfer is established to capture the flow and heat transfer characteristics of impingement synthetic jets in confined spaces. The drawback of the flow structure and the cooling performance at a small nozzle-to-surface for a conventional impingement synthetic jet are first explored as a baseline for comparison. The volumetric efficiencies of the designed hybrid synthetic jet actuators are then predicted whilst the suppression effects of the resulting impingement hybrid synthetic jets on the recirculation of heated air in confined spaces are examined. The cooling performances on a heated surface with a constant heat flux are evaluated in the form of target temperature and Nusselt number. Special attention is paid to the evolution of instantaneous velocity and temperature distribution for all proposed synthetic jets in this paper. This work can provide an essential reference for the proper design of hybrid synthetic jet actuators to gain preferable performance in cooling applications.

2. Numerical methodology

2.1. Numerical model and solving methods

The geometry schematic of a typical actuator generating an axisymmetric impingement synthetic jet in a confined space is shown in Fig. 1(a), which is similar to the setup in the work of Persoons et al. [34]. The actuator comprises a cylindrical cavity equipped with an

orifice and an oscillating diaphragm. With the periodic oscillation of the diaphragm, the synthetic jet is formed and impinges on the target surface via the orifice and the nozzle. The target surface is heated with a constant heat flux, which is marked by the red line. The distance between the nozzle exit and the heated surface is named as the nozzle-to-surface spacing. The nozzle exit is labeled as “Nozzle” for the sake of simplicity. The heated surface is embedded in a confined space like a channel. In this paper, the main dimensions of the geometry are set as follows: the orifice diameter $D_o = 4$ mm; the cavity diameter $D_c = 7.5D_o$ and the cavity height $H_c = 2.5D_o$; the nozzle length $H_n = D_o$; the nozzle-to-surface spacing $H = 4D_o$; the heated surface diameter $D_h = 10D_o$; and the whole target surface diameter $L = 20D_o$. Air is selected as the working fluid and the temperature of ambient air is set to 25°C .

Fig 1(a) also illustrated the boundary conditions adopted in this study. A constant heat flux of 5000 W/m^2 is specified on the heated surface. All other solid walls are set as the adiabatic boundary. Since the outlet section of the channel surrounded by the confined walls is connected to the ambient air, the pressure boundary is adopted there and the outlet static pressure is equal to the atmospheric pressure. The air can freely flow out of and into the channel through the outlet section and the backflow air temperature is set to 25°C . The no-slip boundary for air flow is applied to all solid walls. The oscillation process of the diaphragm can be simplified to a reciprocating movement like a piston with a sinusoidal displacement profile [35]. The time-dependent displacement of the diaphragm is thus defined as [36]

$$X(t) = A \sin(2\pi ft), \quad (3)$$

where t is the time; A is the amplitude; and f is the frequency. In this paper, A is set to $500 \mu\text{m}$ and f is set to 250 Hz . By taking the derivative of $X(t)$, the moving speed of the diaphragm can be derived as

$$U(t) = 2\pi f A \cos(2\pi ft). \quad (4)$$

This equation is used as the boundary condition at the diaphragm of the cavity, which is superior to the flow velocity inlet boundary condition adopted in [15, 37]. This is because the latter imposes fresh air at the inlet within every cycle and cannot allow for the recirculation of hot air, especially at a small nozzle-to-surface spacing. Moreover, the boundary condition adopted in this paper requires less computational time and resources compared with the oscillating membrane boundary condition [36].

(a)

(b)

Fig. 1 Geometry schematic of (a) a confined impingement synthetic jet actuator and (b) various cases of hybrid synthetic jet actuators based on fluid diodes.

Definitions of various cases studied in this paper are illustrated in Fig. 1(b). The basic configuration, as shown in the upper part of Fig. 1(b), remains unchanged except the connecting part between the cavity and the confined channel for all cases. As presented in the lower part of Fig. 1(b), Case A is a conventional synthetic jet actuator, which serves as the baseline case; other cases are self-designed hybrid synthetic jet actuators, where so-called fluid diodes are integrated into the connecting parts. The fluid diodes possess an inlet, which is also connected to the ambient air and where the air can freely flow in and out. They still do not require the fluid transport pipeline. The pressure boundary condition is thus applied to the inlet section. All solid walls in the connecting part are assumed to be adiabatic. Case B1 and Case B2 have the same configuration with a cylindrical nozzle and different values of H_n ; Case C1 and Case C2 have the same configuration with a divergent nozzle and different values of H_n ; Case D1 and Case D2 have the same configuration with a convergent nozzle and different values of H_n . The dimensions of the connecting parts for all cases are gathered in Table 1.

Table 1 The dimensions of the connecting parts for all cases.

Case	Dimensions (mm)					
	D_o	D_i	D_n	H_o	H_i	H_n
A	4	-	-	-	-	D_o
B1	4	$1.5D_o$	D_o	$0.5D_o$	$0.25D_o$	$1.25D_o$
B2	4	$1.5D_o$	D_o	$0.5D_o$	$0.25D_o$	$2.25D_o$
C1	4	$1.5D_o$	$1.5D_o$	$0.5D_o$	$0.25D_o$	$1.25D_o$
C2	4	$1.5D_o$	$1.5D_o$	$0.5D_o$	$0.25D_o$	$2.25D_o$
D1	4	$1.5D_o$	D_o	$0.5D_o$	-	$1.5D_o$
D2	4	$1.5D_o$	D_o	$0.5D_o$	-	$2.5D_o$

Based on the above given oscillating parameters and dimensions of the actuators, it is assumed that the resulting synthetic jets are turbulent for all cases in this study. Actually, the flow regimes of synthetic jets are determined by Reynolds number, which can be calculated by Eq. (1). According to the analysis in Section 3.2, the calculated Reynolds numbers based on the simulation results are 3820 for all cases, which justify the initial turbulent assumption in this study.

The governing equations of mass conservation, momentum conservation and energy conservation for the impingement synthetic jet in the above geometry can be expressed as

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (5)$$

$$\rho \frac{\partial u_i}{\partial t} + \rho \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \rho \overline{u'_i u'_j} \right] \quad (6)$$

$$\frac{\partial T}{\partial t} + u_i \frac{\partial T}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\alpha \frac{\partial T}{\partial x_i} - \overline{u'_i T'} \right) \quad (7)$$

where u_i is the velocity; ρ is the density; p is the pressure; μ is the dynamic viscosity; u'_i is the velocity fluctuation; T is the temperature; α is the thermal diffusivity; and T' the temperature fluctuation. In light of the Boussinesq approximation, $-\rho \overline{u'_i u'_j}$ is named the turbulent stress and $-\overline{u'_i T'}$ is named turbulent heat flux, which are defined as

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (8)$$

$$-\overline{u'_i T'} = \alpha_t \frac{\partial T}{\partial x_i} \quad (9)$$

where μ_t is the turbulent eddy viscosity; k is the turbulent kinetic energy; and α_t is the turbulent diffusivity.

The commercial CFD software, Fluent 18.0, is employed to numerically study the heat transfer of the impingement synthetic jet in this paper. The governing equations of mass, momentum and energy conservation are solved within the axisymmetric geometries under cylindrical coordinates as presented in Fig. 1. The air is considered to be an ideal gas with constant thermophysical properties [15, 31]. The pressure-based transient solver is adopted. The SST/ k - ω turbulence model is applied to capture the turbulent flow features of the impingement synthetic jet, which has proven to be more suitable than other models and has been repeatedly used in the literature [15, 35]. The pressure-velocity coupling is achieved by pressure-implicit with splitting of operators (PISO) algorithm. The least squares cell based method was applied to solve the variable gradient at each control volume center. The convective terms, diffusion terms and transient terms are discretized with the QUICK scheme, central difference scheme and first-order implicit scheme, respectively. The convergence criterion is set as that the normalised residuals of mass, momentum and energy equations decrease below 10^{-5} , 10^{-5} and 10^{-8} at each time step, respectively.

In view of the axisymmetric configuration, only a half of the region on the meridian plane is selected as the computational domain, which has been repeatedly adopted to capture the main flow and heat transfer characteristics of impinging synthetic jets in the literature [36, 37]. Further, Hatami et al. [31] and Miró et al. [32] carried out 3D simulations respectively and their results demonstrated distinctly axisymmetric characteristics of impinging synthetic jets under axisymmetric geometrical configurations. Hence, the use of a 2D axisymmetric model can obtain reasonable results with saving calculation resources. The computational domain is meshed totally by non-uniform structured quadrilateral cells for Case A as presented in Fig. 2(a). The computational grids within the connecting parts for Cases B1, C1 and D1 are shown in Fig. 2(b), which are all structured. The grid distributions maintain as similar as possible for different geometrical configurations. The grids near all the solid walls

are refined to capture large gradients of the velocity and temperature. The sizes of grids adjacent to the walls ensure $y^+ < 1$. The dynamic mesh technology is used to achieve the oscillation of the diaphragm. Since the grids of the entire computational domain are structured, the grid regeneration during the motion of the diaphragm is accomplished by the layering mesh method, which only affects the grid distribution near the diaphragm and can offer decent grid quality in the entire domain. The user-defined function (UDF) is applied to encode the moving speed of the diaphragm in Eq. (4), which is then compiled and embedded it into Fluent 18.0 as the motion profile of the diaphragm in the dynamic mesh setup to control its moving speed.

(a)

(b)

Fig. 2 Computational grids of (a) the entire domain for Case A and (b) connecting parts for Cases B1, C1 and D1.

2.2. Calculation of dimensionless parameters

The integration of fluid diodes is designed to rectify oscillating air flow to suppress the backflow of hot (heated) air through the nozzle exit in the suction stroke. The volumetric

efficiency is thus introduced to evaluate the rectifying effect of the proposed hybrid synthetic jet actuators on the air flow, which can be defined as [30]

$$\eta_V = \frac{V_e - V_s}{V_e + V_s} \quad (10)$$

where V_e represents the air volume ejected out through the nozzle exit in one cycle by the actuators and V_s denotes the air volume sucked back through the nozzle exit in one cycle.

The Nusselt number is used to evaluate the heat transfer performance of impingement hybrid synthetic jets propagating toward the heated surface with a constant heat flux, which is calculated by

$$Nu = \frac{h_c D_o}{\lambda} = \frac{q D_o}{(\theta_s - \theta_a) \lambda} \quad (11)$$

where h_c , λ , q , θ_s and θ_a are the convective heat transfer coefficient, thermal conductivity, constant heat flux on the heated surface, local time-averaged temperature of the heated surface and ambient air temperature, respectively. The average Nusselt number (Nu_{avg}) can be obtained by replacing θ_s in Eq. (11) with the area-weighted average of θ_s across the heated surface.

2.3. Independence study of grid and time step

In order to acquire adequately reliable simulation results, independence studies of grid and time step were implemented by taking Case A as an example. Four different grid sets with about 5000, 10,000, 20,000, 30,000 cells were adopted to examine the grid independence. Table 2 summarizes the predicted local time-averaged temperature at the heated surface center, maximum ejecting average velocity at the nozzle exit during one cycle and average Nusselt number across the heated surface under the four grid sets. It is obvious that the relative deviation of the fourth grid set (30,000 cells) from the third grid set (20,000 cells) is tiny in all the three variables, which is enough to justify the grid independence when

the grid cell number is greater than 20,000. For obtaining a trade-off between calculation accuracy and cost, the third grid set (20,000 cells) was thus selected to carry out the further simulations in this study. The choice of the time step should ensure that Courant–Friedrichs–Lewy (CFL) number is less than 1 to maintain numerical stability. Based on the average jet velocity and the mesh size adopted in this paper, the time step should be less than 5×10^{-5} (i.e. $T/80$). Comparative analysis among four different time steps of $T/100$, $T/200$, $T/500$ and $T/1000$ was carried out to determine adequate time step. The results in Table 3 demonstrated that the relative deviation of the average Nusselt number between $T/1000$ and $T/500$ is enough small and therefore the time step of $T/500$ is adopted for all simulations.

Table 2 The local time-averaged temperature at the heated surface center (θ_c), maximum ejecting average velocity at the nozzle exit during one cycle (u_{max}) and average Nusselt number (Nu_{avg}) for Case A under different grid sets.

Grid cell number	Predicted values			Relative Deviation from the third grid set (%)		
	θ_c (°C)	u_{max} (m/s)	Nu_{avg}	θ_c	u_{max}	Nu_{avg}
5,000	63.1	45.9	12.21	3.52	3.85	4.36
10,000	64.5	45.0	11.95	1.38	1.81	2.14
20,000	65.4	44.2	11.70	-	-	-
30,000	65.6	44.0	11.64	0.31	0.46	0.51

Table 3 Average Nusselt number (Nu_{avg}) for Case A under different time steps.

Time step	Nu_{avg}	Relative Deviation from the third time step set (%)
$T/100$	11.23	4.02
$T/200$	11.51	1.62
$T/500$	11.70	-
$T/1000$	11.72	0.17

3. Results and discussions

3.1. Validation of the numerical methods

The validation of the numerical methods adopted in the present model was conducted through two steps. In the first step, the flow characteristic of a synthetic jet predicted by the adopted numerical methods was validated based on the same geometry and boundary conditions as the work of Jian et al. [36]. Fig. 3(a) compares the instantaneous air velocities at 1 mm away from the nozzle exit along the axis during one cycle predicted by the current model with the corresponding experimental and numerical results obtained by Jian et al. [36]. It is evident that they exhibit the similar velocity profile trend and quite minor difference at each instant, which justify a good conformity. In the second step, the heat transfer performance of impingement synthetic jets predicted by the adopted numerical methods in the current model was validated based on the same geometry and boundary conditions as the work of Chaudhari et al. [16]. As shown in Fig. 3(b), the average Nusselt numbers predicted by the current model are compared with those obtained by the experimental work of Chaudhari et al. [16] for $H/D_o = 2-20$ at $Re = 1150$ and $Re = 4180$. It can be found that the present results show an appropriate agreement with the experimental data obtained by Chaudhari et al. [16] with a relative difference of less than $\pm 15\%$ for all combinations of H/D_o and Re . The above comparisons in the two steps demonstrate that the numerical methods adopted in the present model are feasible and valid.

Fig. 3 Model validation: (a) comparison of instantaneous velocity with the experimental and numerical results of Jian et al. [36]; (b) comparison of average Nusselt number with the experimental results of Chaudhari et al. [16].

3.2. Analysis of flow characteristics

By monitoring instantaneous velocities at 0 mm and 1 mm away from the center of the nozzle exit, it is found that the monitored velocities become stable only over three cycles at the same oscillation phase of the diaphragm, i.e. the periodicity of velocity variation is achieved only after three cycles. The results are in line with the work of Jain et al. [36] and Xia et al. [37]. Therefore, the analyses on the flow characteristics and heat transfer performances are carried out after three cycles. When the oscillating diaphragm is at the bottom end, i.e. the ejection stroke is just finished and the sucking stroke is about to start, we assumed that a cycle is initiated.

According to the preliminary calculations in this study, it is found that the cooling performance of the impingement synthetic jet notably deteriorated as $H/D_o < 6$. Therefore, the case with $H/D_o = 4$ is taken as an example of a small nozzle-to-surface spacing in this paper. Fig. 4 shows the streamlines and velocity contours for Case A with $H/D_o = 4$ and $H/D_o = 8$ at four typical instants of one cycle. As observed in Fig. 4(a) for Case A with $H/D_o = 4$, the flow in the cavity is dominated by a large clockwise vortex, while the flow in the

channel is dominated by an anticlockwise large vortex in the entire cycle. When the diaphragm moves from the bottom end to the neutral position ($t = 0T-0.25T$), the average sucking velocity near the orifice gradually increases from zero to the maximum. A volume of air is sucked into the cavity through the orifice and incorporated into the large clockwise vortex. Some fresh air flows into the channel through the lower outlet section, which partially flows out of the channel through the upper outlet section directly and partially flows into the cavity along the large anti-clockwise vortex inside the channel and the upper confined wall. This part of air is heated by the large anti-clockwise vortex before flowing into the cavity although it is helpful for decreasing the air temperature near the orifice. When the diaphragm moves from the neutral position to the top end ($t = 0.25T-0.50T$), the average sucking velocity near the orifice gradually decreases to zero and the sucking stroke ends. The two large vortices within the cavity and the channel are gradually decoupled and eventually become independent of each other. Meanwhile, the amount of the fresh (i.e. cold) air sucked into the cavity is progressively decreased to zero. Eventually, all the fresh air flowing into the channel through the lower outlet section immediately turns back near the outlet and directly flows out of the channel via the upper outlet section, which means the net flow rate at the outlet section is zero. When the diaphragm moves from the top end to the neutral position ($t = 0.50T-0.75T$), the average ejecting velocity near the orifice gradually increases from zero to the maximum. A volume of air is ejected from the cavity to the channel, which induces the generation of a small anticlockwise vortex near the nozzle exit. The vortex coexists with the anticlockwise large vortex and moves downstream towards the heated surface. The heated air partially flows out of the channel along the heated surface through the outlet section and partially stays within the channel in the form of the anticlockwise large vortex. When the diaphragm moves from the neutral position to the top end ($t = 0.75T-1.00T$), the average ejecting velocity near the orifice gradually declines to zero and the ejection stroke ends. The

amount of air flowing out of the channel along the heated surface also reduces to zero gradually. The flow structure tends to be similar to that at $t = 0.50T$. The moving direction of the small induced vortex changes from towards the heated surface along the axis to towards the outlet along the heated surface. More specifically, the small vortex is generated near the orifice, moves downwards and impinges on the heated surface, and then move along the heated surface towards the outlet, and eventually vanishes. The specific evolution route of the small vortex is 1-2-3-4-5-6 as marked in Fig. 4(a). It implies that the small vortex marked by 5 is evolved from that marked by 1 generated in the previous cycle. When it moves to the position marked by 6, it is incorporated into the large anti-clockwise vortex and vanishes. Most of the air is recirculated in the confined space and only very little amount of fresh air is supplemented into the confined space in one cycle. It means that the heated (i.e. used) air is repeatedly sucked into and ejected from the cavity. This leads to the deterioration of heat transfer or cooling performance, which will be discussed in the later section.

As observed in Fig. 4(b) for Case A with $H/D_o = 8$, the flow structure in the cavity is similar to that for $H/D_o = 4$, while that in the channel is totally different from the latter. The flow in the channel is no longer dominated by a large anti-clockwise vortex, which is a replaced by C-shape flow structure. There is always fresh air flowing into the channel through the upper outlet section as the supplement in the entire cycle. The fresh air is sucked into the cavity along the upper confined wall or incorporated into the jet ejected from the cavity to impinge on the heated surface. Meanwhile, the heated air continuously flows out of the channel via the lower outlet section in the entire cycle along the heated surface. It implies that the heat generated at the heated surface is endlessly transported to the outside of the channel by the involved cold air for $H/D_o = 8$ and thus the heated surface is effectively cooled. Comparison of flow structures between Fig. 4(a) and Fig. 4(b) demonstrates that the

427 small nozzle-to-surface spacing ($H/D_o = 4$) restricts the supplement of the fresh air and the
 428 discharge of the heated air.

431
 432
 433 Fig. 4 Velocity contours and streamlines for Case A with (a) $H/D_o = 4$ and (b) $H/D_o = 8$ at four different
 434 instants of one cycle.

To overcome the above-mentioned drawbacks of Case A with the small nozzle-to-surface spacing ($H/D_o = 4$), various hybrid synthetic jets based on fluidic diodes were designed. In the following study, all cases have $H/D_o = 4$. Fig. 5 presents velocity contours and streamlines of hybrid synthetic jets at four typical instants for Case C2 and Case D2 as examples. Compared with Case A as shown in Fig. 4(a), the improvements in Case C2 as illustrated in Fig. 5(a) are as follows: In the whole suction stroke ($t = 0T-0.50T$), there is some fresh air continuously flows into the cavity through the lower outlet section, and plenty of fresh air is supplemented into the cavity via the inlet of the connecting part. In the whole ejection stroke ($t = 0.50T-1.00T$), plenty of air is ejected from the cavity and impinges on the heated surface, which also induces some fresh air from the inlet to impinge on the heated surface at the same time; Further, a part of the heated air continuously flows out of the channel through the upper outlet section. More improvements are provided by Case D2 as illustrated in Fig. 5(b), which are as follows: In the whole cycle, the air nearly maintains the unidirectional downward flow in the nozzle, except that the nozzle is blocked by a vortex at the end of the suction stroke. It means that plenty of cold fresh air is continuously supplied via the inlet of the connecting part and some air from the cavity and/or the inlet continuously impinges on the heated surface. These improvements are attributed to the combined action of the inlet and the convergent nozzle. The inlet can supplement the fresh air freely. The convergence nozzle can speed up the air flowing downwards, while it serves as a diffuser for the air flowing upwards and thus slows down the air. It means the convergence nozzle exhibits large flow resistance for the air flowing upwards. From the above, the proposed cases solve the drawbacks of Case A by promoting the supplement of the cold fresh air and the discharge of the heated air, i.e. suppressing the recirculation of the heated air in the confined space. This is the main mechanism of cooling performance enhancement.

Fig. 5 Velocity contours and streamlines with partially enlarged views for (a) Case C2 and (b) Case D2 at four typical instants of one cycle.

To quantify the flow characteristics, Fig.6 illustrates the variation curves of area-averaged velocities with time at the orifice, inlet, outlet and nozzle exit for all cases. It can be seen from Fig. 6(a) that the time-dependent average velocity curves at the orifice nearly keep the same for all cases, which is in line with the similar flow structures in the cavity for all cases as shown in Figs. 4-5. The oscillation of the velocities for all cases is subject to the sinusoidal function, which is consistent with the oscillation of the diaphragm. According to Eq. (1) and Eq. (2), the calculated Reynolds numbers and dimensionless stroke length (L_o/D_o) for all cases are 3820 and 14.3, respectively. As presented in Fig. 6(b), the fresh air flows into the cavity or the channel via the inlet in most instants of one cycle except for a small time interval in $t = 0.45T-0.80T$ for all cases. In this time interval, the air flows out via the inlet. Case D2 offers the maximum instantaneous velocity while Case C1 offers the minimum instantaneous velocity at the inlet among all cases in one cycle. Moreover, the inlet velocity increases with H_n under the same geometry configuration. By comparing Fig. 6(b) and Fig. 6(a) for Case D2, it can be found that the flow-in velocity at the inlet is first greater than and then lower than but still close to the velocity flowing into the cavity in the suction stroke. It implies that most of the air in the cavity is supplemented by the inlet and a part of the fresh air from the inlet impinges on the heated surface via the nozzle in the suction stroke, which is in line with Fig. 5(b). As shown in Fig. 6(c), the oscillation of the velocity at the outlet for Case A is still subject to the sinusoidal function, whereas those for other cases deviate from the sinusoidal function. Since the area of the outlet section is the same for all cases, it is evident that less air flows into the channel via the outlet section for all the cases based on fluid diodes compared to Case A in one cycle. The amount of air flowing into the channel via the outlet section for Case D2 is the least among all cases, which is close to zero. It can be found from Fig. 6(d) that the oscillation curve shape of the velocity at the nozzle exit is similar to that at the outlet for all the cases except for Case C1 and C2. This is because

the diameter of the nozzle exit in these two cases is larger than those in other cases as demonstrated in Fig. 1(b). For better comparison, the time-dependent velocity curves in Fig. 6(d) are changed to the time-dependent volume flow rate curves in Fig. 7(a).

Fig. 6 Variations of area-averaged velocities with time for different cases: (a) at the orifice (positive: flow downwards); (b) at the inlet (positive: flow in); (c) at the outlet (positive: flow out); and (d) at the nozzle exit (positive: flow downwards).

As shown in Fig. 7(a), the amplitude of the negative volume flow rate and the corresponding time span in one cycle for all the cases based on fluid diodes are less than that for Case A. Meanwhile, the situation is inverse for the positive volume flow rate. The increase of the amplitude is attributed to the supplement of fresh air via the inlet during

ejecting. Compared to Case A, the air volume sucked into the nozzle from the channel is smaller while the air volume ejected from the nozzle is larger for the other cases in one cycle. Among all cases, Case D2 exhibits the least air volume sucked into the nozzle and the maximum time span of the positive volume flow rate in one cycle. Case D2 almost achieves the unidirectional downward flow in the convergence nozzle during one cycle. It should be noted that the air is heated before being sucked into the nozzle, as shown in Fig. 4(a) and Fig. 5. The air volume sucked into the nozzle from the channel determines the ejected air temperature while the air volume ejected from the nozzle determines the convective heat transfer ability. The both determine the eventual impingement cooling performance. In order to better evaluate the ability of all cases to reduce the amount of sucked heated air and increase the amount of ejected air via the nozzle exit, the volumetric efficiencies for all cases are calculated according to Eq. (10), which are summarized in Fig. 7(b). The descending order of the volumetric efficiencies for all cases is Case D2 > D1 > B2 > B1 > C2 > C1 > A. All the proposed hybrid synthetic jet actuators significantly increase the volumetric efficiency compared with Case A. Especially, the volumetric efficiency for Case A is zero, while that for Case D2 is 95.8% which is close to 1. Moreover, the larger H_n can obtain higher volumetric efficiency under the same geometry configuration.

Fig. 7 (a) Variations of volume flow rate with time via the nozzle exit for all cases (positive: flow downwards); and (b) volumetric efficiency for all cases.

3.3. Analysis of heat transfer performance

As previously mentioned, the air is inevitably recirculated within the channel, the nozzle and the cavity at small H/D_o . Hence, the air in the system is gradually heated. Fig. 8 illustrates the average air temperature evolutions with time at the nozzle exit for all cases. It can be found that the air temperature at the nozzle exit progressively increases and then remains stable after a period of time for all cases. The required time of achieving a stable state increases with the stable temperature. The ascending order of the stable temperature and the required time for all cases is Case D2 < D1 < B2 < B1 < C2 < C1 < A. Among all cases, Case A has the highest stable air temperature at the nozzle exit and needs the longest time to achieve a stable state. Those are about 56.3 °C and 2.5 s, respectively. In contrast, Case D2 has the lowest stable air temperature at the nozzle exit and needs a shorter time to achieve a stable state among all cases, which are about 28.7 °C and 0.8 s. The stable air temperature at the nozzle exit for Case D2 is decreased by 27.6 °C compared to Case A and only increased by 3.7 °C compared to the ambient air temperature. The temperature distributions within the computational domain for all cases at $t = 2.5$ s are illustrated in Fig. 9. The temperature distributions for all cases are relatively stable $t = 2.5$. For Case A, the air temperature within

the cavity above the heated surface increases to 55-60 °C and that within the channel increases to 55-70 °C. It means that the heated surface is ‘cooled’ by very hot air. After introducing fluid diodes, the air temperature within the cavity is decreased below 35 °C and that within the channel above the heated surface is decreased below 40-50 °C as shown in other cases. Especially for Case D2, the air temperature within the cavity is decreased to 25 °C and that within the channel above the heated surface is decreased to about 30-45 °C.

Fig. 8 Average air temperature evolutions with time at the nozzle exit for all cases.

Fig. 9 The temperature distributions within the computational domain for all cases at $t = 2.5$ s.

The temperature distribution on the heated surface is a crucial indicator for impingement cooling performance of synthetic jets. Fig. 10(a) demonstrates time-averaged temperature profiles on the heated surface at the quasi-stable state ($t = 2.5$ s). It can be seen that the temperatures on the heated surface for all the cases based on fluid diodes are significantly reduced compared with Case A. Among them, Case D2 achieves the greatest temperature reduction, where the temperatures on the heated surface are decreased to 37.0-76.9 °C from 65.4-103.9 °C in Case A. The decreased temperatures ensure the safe and high-efficiency operation of electronic devices. To more clearly compare the cooling performance among all cases, the area-weighted average temperatures on the heated surface for all cases are illustrated in Fig. 10(b). The ascending order of the average temperatures on the heated

surface for all cases is Case $D2 < D1 < B2 < B1 < C2 < C1 < A$. It is obvious that all the hybrid synthetic jet actuators based on fluid diode (Cases B1-D2) offer preferable impingement cooling performance compared to conventional synthetic jet actuator (Case A). The hybrid synthetic jet actuator with a convergent nozzle (Cases D1-D2) exhibits better impingement cooling performance than those with a cylindrical nozzle (Cases B1-B2) and a divergent nozzle (Cases C1-C2). By comparing Cases D1 and D2, it is evident that the larger nozzle length (H_n) makes the advantages of hybrid synthetic jets more pronounced. Compared to Case A, the average temperature on the heated surface for Case D2 decreases by 24.9 °C. Further, the average Nusselt numbers of synthetic jet impinging on the heated surface for all cases, calculated according to Eq. (11), are summarized in Fig. 10(c). The descending order of the average Nusselt numbers for all cases is Case $D2 > D1 > B2 > B1 > C2 > C1 > A$. Similarly with respect to Case A, Case D2 obtains the maximum heat transfer enhancement among all the cases based fluid diodes, which increases the average Nusselt number by 54.7%. Based on the combination of Fig. 7(b), Fig. 10(b) and Fig. 10(c), it can be found that the impingement cooling performance and heat transfer ability of synthetic jets are improved as the volumetric efficiency increases.

(a)

(b)

(c)

Fig. 10 Heat transfer data on the heated surface for all cases at the quasi-stable state ($t=2.5$ s): (a) time-averaged temperature profiles; (b) area-weighted average temperatures; (c) average Nusselt numbers.

4. Conclusions

Various hybrid synthetic jets based on novel fluid diodes are proposed and their performances are numerically analyzed. The following conclusions can be drawn:

(1) Compared to conventional synthetic jet, the cooling performance improvement of the proposed hybrid synthetic jets is attributed to the increase of the volumetric efficiency, which promotes the supplement of cold fresh air and suppresses the recirculation of the heated air.

(2) The hybrid synthetic jet with a convergence nozzle exhibits a higher volumetric efficiency and preferable impingement cooling ability compared to that with a cylindrical or

divergence nozzle. Such configuration nearly achieves the unidirectional downward flow in the nozzle during one cycle and obtains a volumetric efficiency of up to 95.8%.

(3) Compared to the conventional synthetic jet, the hybrid synthetic jet with a convergence nozzle decreases the average temperature on the heated surface from 95.7 °C to 70.8 °C and increases the average Nusselt number by 54.7%, which is beneficial for the safe and high-efficiency operation of electronic devices with a high power density.

Acknowledgement

The authors would like to acknowledge the financial support of the National Natural Science Foundation of China (Grant Nos. 51606135 and 51776142), the Natural Science Foundation of Hubei Province (Grant No. 2016CFB156) and the Engineering and Physical Sciences Research Council (EPSRC) of the United Kingdom (Grant No. EP/N000714/1).

References

- [1] H.M. Maghrabie, M. Attalla, H.E. Fawaz, M. Khalil, Impingement/effusion cooling of electronic components with cross-flow, *Applied Thermal Engineering*, 151 (2019) 199-213.
- [2] D. Jagannatha, R. Narayanaswamy, T.T. Chandratilleke, Analysis of a Synthetic Jet-Based Electronic Cooling Module, *Numerical Heat Transfer, Part A: Applications*, 56 (2009) 211-229.
- [3] H. Rho, Y.S. Jang, S. Kim, S. Bae, T.-W. Kim, D.S. Lee, J.-S. Ha, S.H. Lee, Porous copper-graphene heterostructures for cooling of electronic devices, *Nanoscale*, 9 (2017) 7565-7569.
- [4] C.S. Greco, G. Paolillo, A. Ianaro, G. Cardone, L. de Luca, Effects of the stroke length and nozzle-to-plate distance on synthetic jet impingement heat transfer, *International Journal of Heat and Mass Transfer*, 117 (2018) 1019-1031.
- [5] B. Giachetti, M. Fénot, D. Couton, F. Plourde, Influence of multi-perforation synthetic jet configuration on heat transfer enhancement, *International Journal of Heat and Mass Transfer*, 125 (2018) 262-273.
- [6] B. Giachetti, M. Fénot, D. Couton, F. Plourde, Influence of Reynolds number synthetic jet dynamic in crossflow configuration on heat transfer enhancement, *International Journal of Heat and Mass Transfer*, 118 (2018) 1-13.
- [7] A. McGuinn, D.I. Rylatt, T.S. O'Donovan, Heat transfer enhancement to an array of synthetic air jets by an induced crossflow, *Applied Thermal Engineering*, 103 (2016) 996-1003.
- [8] M. Arik, An investigation into feasibility of impingement heat transfer and acoustic abatement of meso scale synthetic jets, *Applied Thermal Engineering*, 27 (2007) 1483-1494.
- [9] R. Mahalingam, A. Glezer, Design and thermal characteristics of a synthetic jet ejector heat sink, *Journal of Electronic Packaging*, 127 (2004) 172-177.
- [10] X. He, J.A. Lustbader, M. Arik, R. Sharma, Heat transfer characteristics of impinging steady and synthetic jets over vertical flat surface, *International Journal of Heat and Mass Transfer*, 80 (2015) 825-834.
- [11] G. Krishan, K.C. Aw, R.N. Sharma, Synthetic jet impingement heat transfer enhancement – A review, *Applied Thermal Engineering*, 149 (2019) 1305-1323.
- [12] T. Persoons, T.S. O'Donovan, A pressure-based estimate of synthetic jet velocity, *Physics of Fluids*, 19 (2007) 128104.
- [13] Q. Gallas, R. Holman, T. Nishida, B. Carroll, M. Sheplak, L. Cattafesta, Lumped Element Modeling of Piezoelectric-Driven Synthetic Jet Actuators, *AIAA Journal*, 41 (2003) 240-247.
- [14] T. Persoons, General Reduced-Order Model to Design and Operate Synthetic Jet Actuators, *AIAA Journal*, 50 (2012) 916-927.
- [15] Y. Zhang, P. Li, Y. 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.

655 [16] M. Chaudhari, B. Puranik, A. Agrawal, Heat transfer characteristics of synthetic jet impingement
656 cooling, *International Journal of Heat and Mass Transfer*, 53 (2010) 1057-1069.

657 [17] M. Chaudhari, B. Puranik, A. Agrawal, Effect of orifice shape in synthetic jet based
658 impingement cooling, *Experimental Thermal and Fluid Science*, 34 (2010) 246-256.

659 [18] M. Chaudhari, B. Puranik, A. Agrawal, Multiple orifice synthetic jet for improvement in
660 impingement heat transfer, *International Journal of Heat and Mass Transfer*, 54 (2011) 2056-2065.

661 [19] A. Lee, G.H. Yeoh, V. Timchenko, J.A. Reizes, Heat transfer enhancement in micro-channel
662 with multiple synthetic jets, *Applied Thermal Engineering*, 48 (2012) 275-288.

663 [20] D.I. Rylatt, T.S. O'Donovan, Heat transfer enhancement to a confined impinging synthetic air jet,
664 *Applied Thermal Engineering*, 51 (2013) 468-475.

665 [21] U.S. Bhapkar, A. Srivastava, A. Agrawal, Proper cavity shape can mitigate confinement effect in
666 synthetic jet impingement cooling, *Experimental Thermal and Fluid Science*, 68 (2015) 392-401.

667 [22] Y.-H. Liu, S.-Y. Tsai, C.-C. Wang, Effect of driven frequency on flow and heat transfer of an
668 impinging synthetic air jet, *Applied Thermal Engineering*, 75 (2015) 289-297.

669 [23] Y.-H. Liu, T.-H. Chang, C.-C. Wang, Heat transfer enhancement of an impinging synthetic air jet
670 using diffusion-shaped orifice, *Applied Thermal Engineering*, 94 (2016) 178-185.

671 [24] Z. Broučková, Z. Trávníček, Visualization study of hybrid synthetic jets, *Journal of*
672 *Visualization*, 18 (2015) 581-593.

673 [25] Z. Trávníček, T. Vít, V. Tesař, Hybrid synthetic jets as the nonzero-net-mass-flux synthetic jets,
674 *Physics of Fluids*, 18 (2006) 081701.

675 [26] Z. Trávníček, V. Tesař, J. Kordík, Performance of synthetic jet actuators based on hybrid and
676 double-acting principles, *Journal of Visualization*, 11 (2008) 221-229.

677 [27] J. Kordík, Z. Trávníček, Novel Fluidic Diode for Hybrid Synthetic Jet Actuator, *Journal of Fluids*
678 *Engineering*, 135 (2013) 101101-101107.

679 [28] Z. Trávníček, T. Vít, Impingement heat/mass transfer to hybrid synthetic jets and other reversible
680 pulsating jets, *International Journal of Heat and Mass Transfer*, 85 (2015) 473-487.

681 [29] S.-S. Hsu, Z. Travnicek, C.-C. Chou, C.-C. Chen, A.-B. Wang, Comparison of double-acting and
682 single-acting synthetic jets, *Sensors and Actuators A: Physical*, 203 (2013) 291-299.

683 [30] S.S. Hsu, Y.J. Chou, Z. Trávníček, C.F. Lin, A.B. Wang, R.H. Yen, Numerical study of nozzle
684 design for the hybrid synthetic jet actuator, *Sensors and Actuators A: Physical*, 232 (2015) 172-182.

685 [31] M. Hatami, F. Bazdidi-Tehrani, A. Abouata, A. Mohammadi-Ahmar, Investigation of geometry
686 and dimensionless parameters effects on the flow field and heat transfer of impingement synthetic
687 jets, *International Journal of Thermal Sciences*, 127 (2018) 41-52.

688 [32] A. Miró, M. Soria, J.C. Cajas, I. Rodríguez, Numerical study of heat transfer from a synthetic
689 impinging jet with a detailed model of the actuator membrane, *International Journal of Thermal*
690 *Sciences*, 136 (2019) 287-298.

- [33] P. Li, D. Guo, R. 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.
- [34] T. Persoons, A. McGuinn, D.B. Murray, A general correlation for the stagnation point Nusselt number of an axisymmetric impinging synthetic jet, *International Journal of Heat and Mass Transfer*, 54 (2011) 3900-3908.
- [35] D. Zhang, K. Yang, H.E. Qu, J. Gao, Numerical Investigation of Heat Transfer Performance of Synthetic Jet Impingement onto Dimpled/protrusioned Surface, *Thermal Science*, 19 (2015) S221-S229.
- [36] M. Jain, B. Puranik, A. Agrawal, A numerical investigation of effects of cavity and orifice parameters on the characteristics of a synthetic jet flow, *Sensors and Actuators A: Physical*, 165 (2011) 351-366.
- [37] Q. Xia, S. Lei, J. Ma, S. Zhong, Numerical study of circular synthetic jets at low Reynolds numbers, *International Journal of Heat and Fluid Flow*, 50 (2014) 456-466.