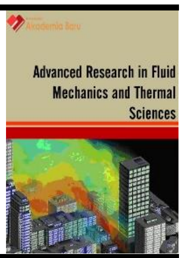
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Experiment Study on Heat Transfer and Flow Structure of The Single Row Circular Orifice Jets Injecting into Turbulent Boundary Layer

Open Access

Panupong Juera-ong[1], Nioh Puzu[2], Chayut Nuntadusit[1,*]

[1] Department of Mechanical Engineering, Faculty of Engineering, Prince of Songkla University, Hat Yai, Songkhla 90110, Thailand
[2] Department of Mechanical Engineering, Faculty of Engineering, Princess of Naradhiwas University, Mueang, Narathiwat 96000, Thailand

ARTICLE INFO

Article history:
Received 26 December 2019
Received in revised form 6 August 2020
Accepted 8 August 2020
Available online 6 December 2020

Keywords:
Heat transfer enhancement; Orifice Jets; Turbulent boundary layer; TLC; Laser-induced fluorescent

ABSTRACT

Thermal management is a crucial issue in electrical equipment. Most of past studies focused on passive technique using solid turbulators. Active technique by using jet flow was investigated here. This paper presents a study of heat transfer and flow characteristics of a single row of orifice jets on boundary layer on a flat plate. The study was divided into two parts. Firstly, the effect of jet velocity was investigated by fixing the mainstream velocity at 10 m/s and varied the jet velocity corresponding to a momentum flux of 0.01, 0.06, 0.25, 1.0, 2.25, 4.0 and 12.25. An infrared camera captures the study of temperature distribution on heat transfer surfaces with constant heat flux. Secondly, the flow structure was investigated using laser-induced fluorescence technique. The heat transfer enhancement can be found downstream of the row of orifices. When increasing the momentum flux ratio, the heat transfer enhancement was significantly promoted. Because the generated vortex structure introduced more mainstream flow attacking the surface.

1. Introduction

Nowadays, heat transfer enhancement is an essential issue due to the increase in heat load in thermal equipment. For example, electronic circuits in a computer device generate at high heat load per volume. Gas turbine blade operated at higher temperature load resulting in greater thermal efficiency. There are many techniques for cooling this equipment by an active and passive method. In this study, we focused on the active method using orifice jets for cooling on the surface by injected a row of jets into the turbulent boundary flow. This technique can be applied for a small channel in electronic equipment.

[*] *Corresponding author.*
E-mail address: chayut@me.psu.ac.th

https://doi.org/10.37934/arfmts.78.2.4755

Ahn [1] described the effect of the jet outlet and founded that the circulating currents from the downwash side of the jet could increase the heat transfer. Lee *et al.,* [2] studied the geometric variation of film cooling holes on the cooling performance to increase the film cooling effectiveness. The result showed that the film cooling effectiveness was improved through optimization of the holes. Sun *et al.,* [3] studied for film cooling at a fixed coolant mass flow rate, and they found that when the blowing ratio was increased, the cooling effectiveness of the fan-shaped hole increased. Singh *et al.,* [4] investigated for film cooling over a flat plate. The results showed that the longest hole could maximize the cooling efficiency of the film cooling. Yuen and Martinez-Botas [5] described the heat transfer surface for enhancing the film cooling effectiveness and found that the performance of effectiveness and heat transfer coefficient for the two inline rows were prone to be an advantage in the film cooling for a turbine blade. Fiebig [6] deployed a wing-type vortex generator on a flat surface and compared for heat transfer enhancement between generated longitudinal vortices and transverse vortices. Prince [7] carried out the numerical simulation for a vortex jet to delay boundary layer separation on the trailing edge of a plane wing surface. The result showed that the vortex jet can significantly delay the separation on trailing-edge and subsequent stall to higher attack angles.

Jacobi and Shah [8] studied for solid vortex generators and found that could considerably increase heat transfer and friction loss. Aris *et al.,* [9] improved the thermal performance of fin stacks by changing attack angles that can enhance the heat transfer based on the fin surface temperature about 37% compared with the normal fin stack. Smulsky *et al.,* [10] investigated the effect of the rib attachment orientation angle on a heat transfer surface and found that the heat transfer at an angle of 50° became highest. Rao *et al.,* [11] investigated for dimples in the pin-fin channel. The experiment results showed that the pin-fin with dimple could improve convective heat transfer up to 19% compared to the normal baseline pin fin channel and the pin-fin with deeper dimples shows relatively higher Nusselt numbers, but the pin-fin with shallower dimples shows relatively lower friction factors. Qayoum [12] studied for a synthetic jet interacting with laminar boundary layer on a flat plate. The result shows that the average heat transfer coefficient increased with excitation and become maximum at about 44%. Fric [13] studied for single jet issuing from the wall into a crossflow and showed four types of flow structures which were investigated for heat transfer enhancement.

Compton and Johnson [14] studied the vortex flow generated by the jet flow blocking the main flow and found that the vortex flow can reduce the thickness of the boundary layer. Zhang [15] studied the vortex flow occurred between the main flow and the circle orifice jet. Kwanmon and Asi [16] studied the interaction between mainstream and jet flow near the jet exit. The results showed that the velocity contour showed a kidney-shaped close to the jet exit. Dai [17] showed the detail of the flow structure of the jet when it interacted with the mainstream. Suzuki [18] studied for a pulse jet to improve heat transfer rate on surface. Jabbal and Zhong [19] studied the characteristics of flow and heat transfer using the synthetic jet. The characteristics of jet flow on a smooth surface and a rough surface were also studied by Jovanovic [20].

However, most of the studies focused on flow characteristics of jet interacting with the mainstream flow. There appeared only a few studies concerned on heat transfer characteristics on the downstream surface of the jet holes. In this study, we interested in heat transfer enhancement on a surface using jets from a row of orifices in a turbulent boundary layer on a flat plate, which the heat transfer can control actively by changing the velocity of the jet flow. This research aims to study for the flow and heat transfer characteristics for a row of orifice jets. The effect of the momentum ratio between the jet flow and mainstream was investigated experimentally.

2. Methodology
2.1 Experimental Setup

Figure 1 shows the experiment setup details using in this study. The open wind tunnel blows a uniform velocity-profiled mainstream through a square exit over the horizontal flat plate with a sharp leading edge. There is a row of circular orifices on the flat plate for cooling the constant heat flux surface at downstream of the row.

The open flow wind tunnel is designed with a square exit, with a bell-shaped with the exit at 400 mm x 400 mm, as shown in Figure 1 The wind tunnel from the settling chamber, which was installed coarse and fine meshes for obtaining uniform flow. The flow was narrowed to re-accelerate the airflow and removing separation flow near the exit. The mainstream was uniform-velocity profile at 10.0 m/s. The velocity at the exit was measured with hot wire anemometer and found that the velocity was controlled to be almost constant along a downstream direction within 4.5%. The turbulence intensity was small, about 0.175%. During the experiment, the air temperature in the room was fixed at 25°C. The temperature of the air jet and mainstream were fixed at 26.0 ± 0.2°C by the heaters, which controlled with the temperature controller.

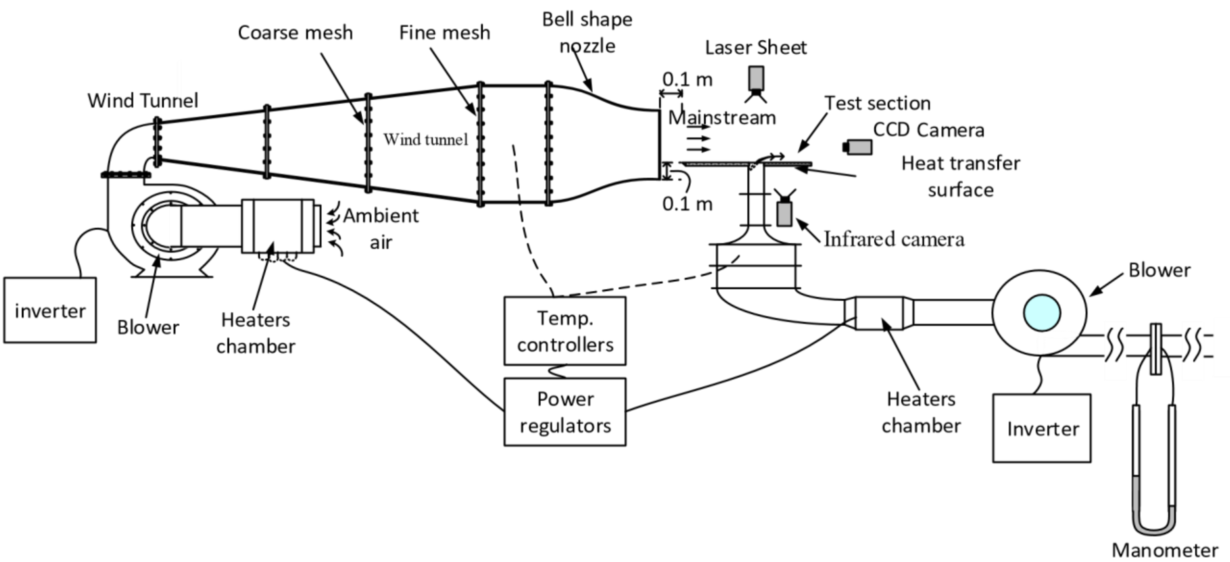


Fig. 1. Experimental setup

Figure 2 shows the details for a flat plate with a heat transfer surface and a row of orifices. The flat plate was made of an acrylic plate with 20 mm in thickness and the length of 800 mm long, 480 mm wide. The leading edge of the flat plate was sharp edge at 15° to keep an initial laminar flow near the leading edge (detail in Figure 2). A tripwire with 1.5 mm in diameter was fixed on the surface at 1.0D from the leading edge to trigger a turbulent boundary layer flow. The boundary layer thickness (δ) at the jet orifice position was measured to be 1.01D based on the streamwise distance from the leading edge, and the mainstream Reynolds number (Re_x) was 2.5×10^5 which in range of turbulent boundary layer.

A single row of circle orifices with diameter D = 14 mm and pitch distance S = 2D was mounted at about 28.6D from the leading edge. The heat transfer surface was made of a stainless sheet (270 mm x 300 mm and thickness of 0.030 mm) downstream of the orifice holes at 0.36D from the orifices. The stainless sheet tightly stretched between two copper bus bars, which was connected with the DC supply that generated uniform heat flux on the stainless sheet.

For heat transfer measurement, the temperature distribution on the stainless sheet was measured by the infrared camera (FLIR T420), which could be captured up to 120 fps and the range of -20°C to 120°C with an accuracy of 0.2%. The resolution of the camera was 320 × 240 pixels. The camera was installed underside the heat transfer surface and the temperature were measured from the rear side of the surface, which was sprayed with black paint having an emissivity of 0.95. The stainless sheet surface was heated to constant heat flux by supplying electricity via the bus bars that were installed at the lateral of the test section, as shown in Figure 2. The temperature was measured on the rear side of the heat transfer surface. The surface temperature on the heat transfer could be considered without the heat conduction effect.

For flow visualization of jet structure in the mainstream, the smoke generated from smoke generator (Ate-AEROTECH model: SGS-90) with oil (SHELL ONDINA 15) mixed with the jet flow in jet chamber. The argon ion laser with cylindrical lens was used to generate a laser sheet. The fluorescence could show the jet flow structure when the smoke was passed the laser sheet. In this study, the cross-section of the jet flow structure was visualized and recorded with a CCD camera (Sony model: HP9531PE) with the resolution 1920 x 1080 pixels. The evolution of jet flow was investigated at different downstream locations from the orifice exits.

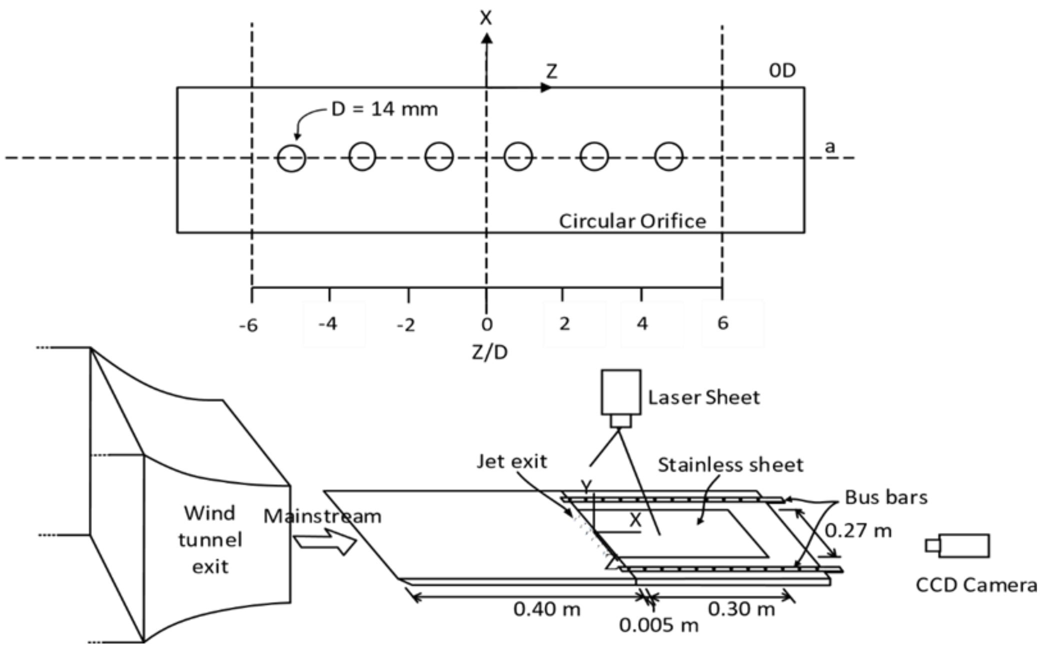


Fig. 2. Test section

2.2 Experimental Parameters

In this study, the effect of jet velocity was studied and varied corresponding to seven values of the mainstream to jet momentum flux ratio (J) of 0.01, 0.06, 0.25, 1.0, 2.25, 4.0 and 12.25. The jet momentum flux ratio was defined as:

$$J = \frac{\rho_j {v_j}^2}{\rho_m {v_m}^2} \tag{1}$$

where ρ_j is the density of jet flow, ρ_m is the density of mainstream, v_j is the velocity of jet flow and v_m, is the velocity of mainstream. The density of mainstream and jet were the same in this study. Moreover, the jet Reynolds number was corresponded to 8.96 × 10^2, 2.2 × 10^3, 4.5 × 10^3, 8.96 × 10^3,

1.34×10^4, 1.79×10^4 and 3.14×10^4, respectively. Where the jet Reynolds number was defined based on the centre velocity of the orifice as

$$Re_j = \frac{v_j D}{\nu} \quad (2)$$

where ν is the kinematic viscosity of jet flow.

2.3 Data Measurement and Data Processing

The total heat flux via the adjustable DC power supply could be calculated from Joule heating as

$$Q_{in} = VI \quad (3)$$

where I is the electrical current (A), V is the voltage (V) between the two-bus bar. The total heat loss from the rear side of heat transfer surface could be calculated from

$$Q_{loss} = Q_{conv} + Q_{rad} \quad (4)$$

where Q_{conv} is the heat loss from free convection and Q_{rad} is the heat loss from radiation. The local convection heat transfer coefficient could be obtained from Newton's law of cooling as

$$h = \frac{Q_{in} - Q_{loss}}{(T_w - T_{aw})A} \quad (5)$$

where, T_w is the local wall temperature (with wall heat flux), T_{aw} is the adiabatic wall temperature (without wall heat flux) and A is the area of the heat transfer surface.

3. Results

3.1 Heat Transfer Characteristics

The convective heat transfer coefficient ratio of h/h_0 is defined to show the heat transfer enhancement, where h is the coefficient for the case with the jets and mainstream, h_0 is the coefficient for the case with mainstream and without the jets.

Figure 3 shows the experimental results of a heat transfer coefficient ratio on the downstream surface at different the jet-mainstream momentum flux ratios was measured with an infrared camera. All seven momentum flux values increased the heat transfer rate from no jet condition.

For the case of small momentum flux ratio at J = 0.01, 0.06 and 0.25, the heat transfer enhancement can be seen downstream of each orifice. The heat transfer enhancement increased and elongated to downstream as increasing the momentum flux ratio. It was found two footprints of heat transfer enhancement behind each orifice. This phenomenon corresponded to the jet flow structure near the heat transfer surface. There appears clearly low heat transfer region between the orifices.

For the case of medium momentum flux ratio at J = 1.0, 2.25 and 4.0, the heat transfer enhancement can be seen clearly in the region behind each orifice. The region of low heat transfer as seen in case of small momentum flux ratio disappeared. This resulted from the interaction between adjacent jet flow promoted the heat transfer in the region between the orifices and can be seen on the footprint of heat transfer results.

For the case of maximum momentum flux ratio at J = 12.25, the higher heat transfer enhancement can be seen behind the orifice compared to the case of medium momentum flux ratio. The interaction between jet flow resulted in the heat transfer enhancement moved to the region between the orifices from about X/D=4.0 and elongated to downstream. It was found that the heat transfer can be promoted significantly and almost uniform extended to X/D=12.

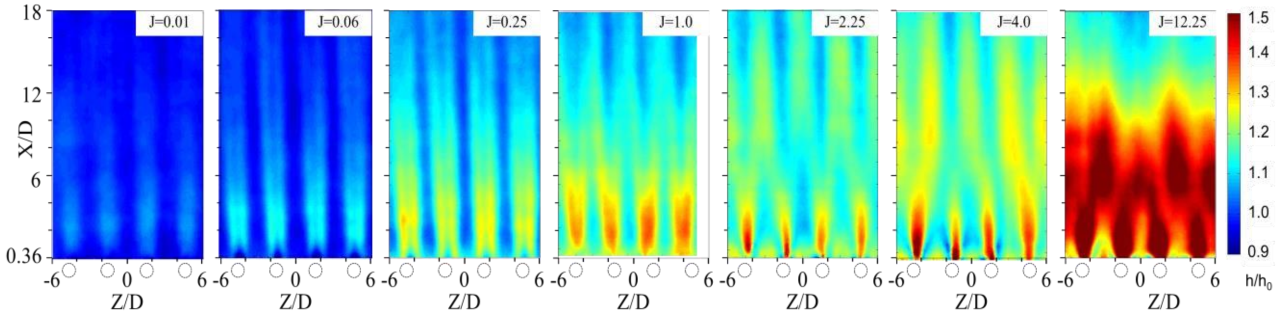


Fig. 3. Contours of heat transfer coefficient ratio at different momentum flux ratios

3.2 Flow Structure Development

Figure 4 shows the cross-section of jet flow structure from a row of circular orifices in the mainstream compared at different momentum flux ratios. The cross-section of jets was shown at the downstream distance at X/D = 1, 4, 8 and 16.

For the case of low momentum flux ratio at J = 0.06 and 0.25, the cross-section of each jet attached on the surface and the area of cross-section increased as going downstream. This corresponded to high heat transfer enhancement on the surface. Moreover, the jet disappeared at far downstream X/D = 16. There appeared no covered region on the surface between the jets. This region corresponded to the low heat transfer area on the surface. When increased the momentum flux ratio to J = 2.25, the cross-section of each jet became kidney shape which agrees to past research. This is due to the interaction between jet structure and mainstream flow.

Moreover, this resulted in the line of two peaks of heat transfer on the surface at near downstream of each orifice. As going downstream, the jet flow attached to the surface and spreading in a spanwise direction. So, the region of low heat transfer between downstream of the orifice became small and disappeared when increasing the momentum flux ratio.

For the case of medium momentum flux ratio at J = 2.25 and 4.0, the kidney-shaped jet may introduce the mainstream flow to attack the surface in the near downstream region as shown at X/D = 1 and 4. This promoted the heat transfer enhancement behind the jet orifices. Then the jet spreading in spanwise direction and covered over the surface as shown at X/D = 8 and 16. The interaction between the adjacent jets at far downstream was very complicated and produced intricate heat transfer pattern on the far downstream region.

For the case of maximum momentum flux ratio at J = 12.25, the kidney-shaped jet still appeared at X/D = 1. The size of the jet became larger and appeared far from the surface. This was due to the strong momentum of jet penetrating the mainstream. The more mainstream was introduced to attack the surface and promote more the heat transfer on the surface in near and far downstream.

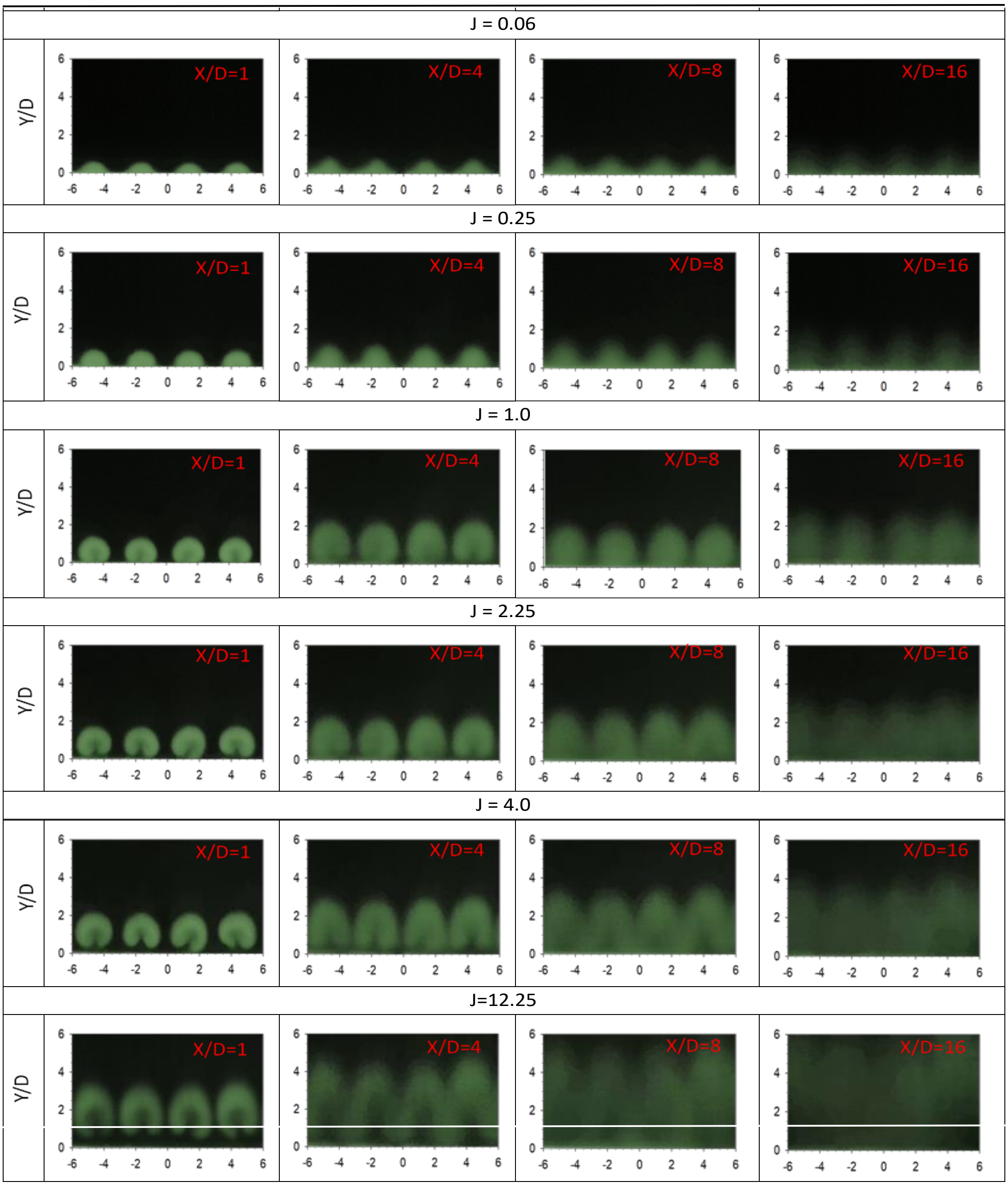


Fig. 4. Cross section of jet flow structure from a row of circular orifices in the mainstream at different downstream locations compared at different momentum flux ratios

4.Conclusions

An experimental study for flow structure and heat transfer characteristics of a row of orifice jets into turbulent boundary layer on the flat plate was carried out. The effect of jet velocity corresponding to momentum flux ratio was investigated at J = 0.01, 0.06, 0.25, 1.0, 2.25, 4.0 and 12.25 for a constant mainstream velocity at 10 m/s. The main results can be summarized as follows.

i. The heat transfer on the surface could be controlled actively by varying the jet velocity from a row of orifices on a flat plate.
ii. The heat transfer enhancement can be found downstream of the row of orifices. When increasing the momentum flux ratio, the heat transfer enhancement was significantly

promoted due to the kidney shape vortex structure induced mainstream to attack the surface. The high heat transfer region extended to further downstream.

iii. The kidney-shaped vortex in jet flow is the key for heat transfer enhancement on the surface. The higher momentum flux ratio can generate more robust vortex structure in jet and more promote the heat transfer in the near and far region from the row of orifices.

Acknowledgment
This research was funded by a grant from the Department of Mechanical Engineering, Faculty of Engineering, Prince of Songkla University, Thailand.

References

[1] Ahn, Joon, In Sung Jung, and Joon Sik Lee. "Film cooling from two rows of holes with opposite orientation angles: injectant behavior and adiabatic film cooling effectiveness." *International Journal of Heat and Fluid Flow* 24, no. 1 (2003): 91-99. https://doi.org/10.1016/S0142-727X(02)00200-X

[2] Lee, Ki-Don, and Kwang-Yong Kim. "Shape optimization of a fan-shaped hole to enhance film-cooling effectiveness." *International Journal of Heat and Mass Transfer* 53, no. 15-16 (2010): 2996-3005. https://doi.org/10.1016/j.ijheatmasstransfer.2010.03.032

[3] Sun, Xiaokai, Gang Zhao, Peixue Jiang, Wei Peng, and Jie Wang. "Influence of hole geometry on film cooling effectiveness for a constant exit flow area." *Applied Thermal Engineering* 130 (2018): 1404-1415.. https://doi.org/10.1016/j.applthermaleng.2017.11.117

[4] Singh, Kuldeep, B. Premachandran, and M. R. Ravi. "Experimental assessment of film cooling performance of short cylindrical holes on a flat surface." *Heat and Mass Transfer* 52, no. 12 (2016): 2849-2862. https://doi.org/10.1007/s00231-016-1777-6

[5] Yuen, C. H. N., and R. F. Martinez-Botas. "Film cooling characteristics of rows of round holes at various streamwise angles in a crossflow: Part II. Heat transfer coefficients." *International journal of heat and mass transfer* 48, no. 23-24 (2005): 5017-5035. https://doi.org/10.1016/j.ijheatmasstransfer.2005.05.020

[6] Fiebig, M. "Vortices, generators and heat transfer." *Chemical Engineering Research and Design* 76, no. 2 (1998): 108-123. https://doi.org/10.1205/026387698524686

[7] Prince, Simon A., Vahik Khodagolian, Chrisminder Singh, and T. Kokkalis. "Aerodynamic stall suppression on aerofoil sections using passive air-jet vortex generators." *AIAA journal* 47, no. 9 (2009): 2232-2242. https://doi.org/10.2514/1.41986

[8] Jacobi, A. M., and R. K. Shah. "Heat transfer surface enhancement through the use of longitudinal vortices: a review of recent progress." *Experimental Thermal and Fluid Science* 11, no. 3 (1995): 295-309. https://doi.org/10.1016/0894-1777(95)00066-U

[9] Aris, M. S., R. McGlen, I. Owen, and C. J. Sutcliffe. "An experimental investigation into the deployment of 3-D, finned wing and shape memory alloy vortex generators in a forced air convection heat pipe fin stack." *Applied thermal engineering* 31, no. 14-15 (2011): 2230-2240. https://doi.org/10.1016/j.applthermaleng.2011.03.015

[10] Smulsky, Ya I., V. I. Terekhov, and N. I. Yarygina. "Heat transfer in turbulent separated flow behind a rib on the surface of square channel at different orientation angles relative to flow direction." *International journal of heat and mass transfer* 55, no. 4 (2012): 726-733. https://doi.org/10.1016/j.ijheatmasstransfer.2011.10.037

[11] Rao, Yu, Chaoyi Wan, and Yamin Xu. "An experimental study of pressure loss and heat transfer in the pin fin-dimple channels with various dimple depths." *International journal of heat and mass transfer* 55, no. 23-24 (2012): 6723-6733. https://doi.org/10.1016/j.ijheatmasstransfer.2012.06.081

[12] Qayoum, Adnan, Vaibhav Gupta, P. K. Panigrahi, and K. Muralidhar. "Perturbation of a laminar boundary layer by a synthetic jet for heat transfer enhancement." *International journal of heat and mass transfer* 53, no. 23-24 (2010): 5035-5057. https://doi.org/10.1016/j.ijheatmasstransfer.2010.07.061

[13] Fric, Thomas F. *Structure in the near field of the transverse jet*. CALIFORNIA INST OF TECH PASADENA GRADUATE AERONAUTICAL LABS, 1990.

[14] Compton, Debora A., and James P. Johnston. "Streamwise vortex production by pitched and skewed jets in a turbulent boundary layer." *AIAA journal* 30, no. 3 (1992): 640-647. https://doi.org/10.2514/3.10967

[15] Zhang, Xin, and Michael W. Collins. "Flow and heat transfer in a turbulent boundary layer through skewed and pitched jets." *AIAA journal* 31, no. 9 (1993): 1590-1599. https://doi.org/10.2514/3.11819

[16] Kwanmon, S., and Asi, B., "Structure Jet-and Cross flow Interactions and Cross-plane Entrainment Mechanism of a jet in cross flow", The 30th Conference of Mechanical Engineering Network of Thailand, Songkhla Thailand. TSF0015, (2016).

[17] Dai, C., L. Jia, J. Zhang, Z. Shu, and J. Mi. "On the flow structure of an inclined jet in crossflow at low velocity ratios." *International Journal of Heat and Fluid Flow* 58 (2016): 11-18. https://doi.org/10.1016/j.ijheatfluidflow.2015.12.001

[18] Jabbal, M., and S. Zhong. "The near wall effect of synthetic jets in a boundary layer." *International Journal of Heat and Fluid Flow* 29, no. 1 (2008): 119-130. https://doi.org/10.1016/j.ijheatfluidflow.2007.07.011

[19] Jovanović, M. B., H. C. De Lange, and A. A. Van Steenhoven. "Influence of hole imperfection on jet cross flow interaction." *International Journal of Heat and Fluid Flow* 27, no. 1 (2006): 42-53. https://doi.org/10.1016/j.ijheatfluidflow.2005.06.003

[20] Smulsky, Ya I., V. I. Terekhov, and N. I. Yarygina. "Heat transfer in turbulent separated flow behind a rib on the surface of square channel at different orientation angles relative to flow direction." *International journal of heat and mass transfer* 55, no. 4 (2012): 726-733. https://doi.org/10.1016/j.ijheatmasstransfer.2011.10.037