

Effect of jet-mainstream velocity ratio on flow characteristics and heat transfer enhancement of jet on flat plate flow

N Puzu¹, S Prasertsan¹ and C Nuntadusit¹

¹ Department of Mechanical Engineering, Faculty of Engineering, Prince of Songkla University, Songkhla, Thailand

E-mail: chayut@me.psu.ac.th

Abstract. The aim of this research was to study the effect of jet-mainstream velocity ratio on flow and heat transfer characteristics of jet on flat plate flow. The jet from pipe nozzle with inner diameter of $D=14$ mm was injected perpendicularly to mainstream on flat plate. The flat plate was blown by mainstream with uniform velocity profile at 10 m/s. The velocity ratio (jet to mainstream velocity) was varied at $VR=0.25$ and 3.5 by adjusting velocity of jet flow. For heat transfer measurement, a thin foil technique was used to evaluate the heat transfer coefficient by measuring temperature distributions on heat transfer surface with constant heat flux by using infrared camera. Flow characteristics were simulated by using a computational fluid dynamics (CFD) with commercial software ANSYS Fluent (Ver.15.0). The results showed that the enhancement of heat transfer along downstream direction for the case of $VR=0.25$ was from the effect of jet stream whereas for the case of $VR=3.5$ was from the effect of mainstream.

1. Introduction

Heat transfer enhancement is one of key technology for energy saving. Especially, convective heat transfer is commonly found in the engineering application, e.g. engine cooling fin, electronics cooling system, heat exchanger etc. In the meanwhile, the method for heat transfer enhancement are various such as to install ribs, vanes, or pins on heat transfer surface to produce mainstream vortex on the surface. Many researchers presented the effect of vortex relate with the heat transfer of the surface.

M. Fiebig [1] studied the heat transfer enhancement by using wing-type vortex generators experimentally and numerically. The results showed that heat transfer rate was higher than the case of plane surface. However, the increasing of flow friction due to attachment wing-type vortex generators causes additional pressure loss and the need of more pumping power. For minimizing pressure loss penalty, vortex generating jet (VGJ) are attractive method by injecting jet into the boundary layer flow. Generally, the injecting jet in mainstream is used for film cooling application e.g. gas turbine engine blade cooling [2-5]. Ahn et al. [6] numerically investigated the interaction between jet and mainstream effect on film cooling effectiveness in gas turbine cooling application. The result showed that film cooling effectiveness depended on velocity ratio (jet to mainstream velocity) and the configuration of jet holes. Maio and Wu [7] numerically investigated the blowing and hole's shape are influencing film cooling effectiveness. In case of heat transfer enhancement, Qayoum et al. [8] used piezo-actuator for displacing the diaphragm to generate vortex and reported the heat transfer coefficient of downstream surface become increase. Jabbal and Zhong [9] shown that the heat transfer rate of underneath vortex surface increased significantly. Akdag et al. [10] experimentally investigated the cooling of constant heat flux surface by using thermocouple to measure surface temperature. The



temperatures are varying with distance from the jet hole and the nearest position temperature is lowest. Whereas understanding of flow characteristics of vortex generating jet is important, the provide results from previous studies are limited.

The objective of this study was to investigate the flow and heat transfer characteristics of a vortex generating jet on a flat plate flow. The jet discharged from pipe nozzle with normal to the heat transfer surface. The effects of jet-to-mainstream velocity ratio on flow and heat transfer characteristic was focussed.

2. Experimental setup and method

The experimental setup is shown in Fig. 1. An open wind tunnel having bell shape nozzle was used to generate mainstream with uniform velocity profile over a flat plate. A VGJ nozzle was installed on the flat plate. The dimensions of bell shape nozzle exit of wind tunnel were 400 mm x 400 mm. The measured mainstream turbulence at the nozzle exit was nearly isotropic with turbulence intensity of 3.5% and a decay rate, $\partial V_m / \partial (X/D)$, of 0.175%, approximately. The difference of turbulence intensity and mean velocity along the nozzle exit was lesser than 2.0% respecting their mean value. The jet pipe was made of aluminum tube having the inner diameter of $D=14$ mm and the length of $10D$ for achieving fully developed flow of jet at the exit. The inlet of nozzle pipe was connected to blower supplier with flexible tube for reducing noise vibration.

The details of test section are shown in Fig. 1. The test section was made of acrylic plate with sharp edge ($480 \text{ mm} \times 800 \text{ mm}$ and 20 mm in thickness) and was opened with rectangular hollow. A stainless-steel foil with $300 \text{ mm} \times 300 \text{ mm}$ and 0.03 mm in thickness was used for measuring heat transfer coefficient. It was tightly stretched between two copper bus bars over the rectangular hollow.

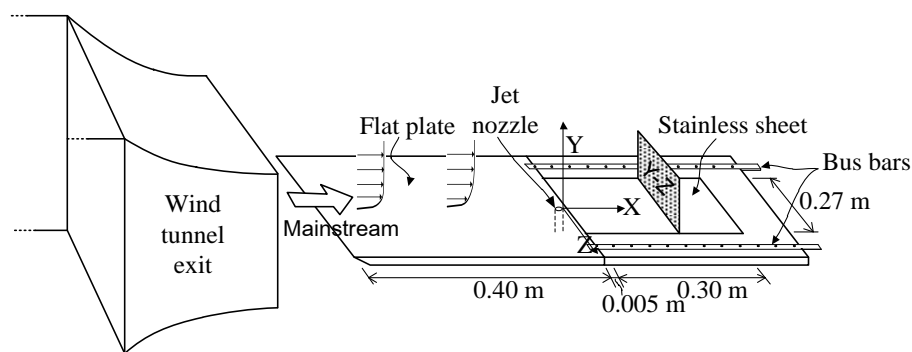


Figure 1. Detail of test section

The temperature distributions on the heat transfer surface were measured from the rear side of stainless foil using an infrared camera. The temperatures on both flow side and measurement side can be considered being the same since the stainless foil was sufficiently thin. The heat transfer surface was heated to obtain constant heat flux by supplying electrical current from power supply through the copper bus bars. The bottom side of heat transfer surface was sprayed with black paint which has emissivity at 0.95.

The temperature of mainstream and jet were controlled at $26.0 \pm 0.2^\circ\text{C}$. The mainstream velocity at wind tunnel exit was fixed at 10 m/s . The jet velocity from pipe nozzle was varied corresponding to jet-to-mainstream velocity ratio was varied at $VR=0.25$ and 3.5 .

For data reduction, heat generated in heat transfer surface was defined as

$$\dot{Q}_{input} = I^2 R \quad (1)$$

Where, I and R were electrical current and electrical resistant of stainless foil. The heat losses from the rear side of heat transfer surface were considered from,

$$\dot{Q}_{losses} = \dot{Q}_{conv} + \dot{Q}_{rad} \quad (2)$$

where \dot{Q}_{conv} and \dot{Q}_{rad} were the heat losses from the rear side of stainless sheet to the environmental air by free convection and radiation, respectively. The local convective heat transfer coefficient can be obtained from following equations:

$$h = \frac{\dot{Q}_{input} - \dot{Q}_{losses}}{(T_w - T_{aw})A} \quad (3)$$

where, A was the area of heat transfer surface; T_w and T_{aw} were the local wall temperature (with wall heat flux) and local adiabatic wall temperature (with no heat flux) on heat transfer surface.

The convective heat transfer coefficient ratio, h/h_0 is used to represent the ability of heat transfer enhancement that compare between the case with jet and the case without jet. Here, h_0 was the convective heat transfer coefficient for the case with no jet.

3. Numerical Simulation

3.1. Computational model and calculation detail

In order to understand the effect of velocity ratio on flow characteristics of vortex generating jet, three-dimensional flow fields were simulated by using ANSYS FLUENT (version 15.0). The solutions for turbulence flow were achieved by solving steady incompressible Reynolds Averaged Navier-Stokes (RANS) equation through the finite volume method. The SIMPLEC (Semi-Implicit Method for Pressure Linked Equations-Consistent) algorithm was used for reveal velocity and pressure field in order to reduce the calculating time. In the present study, SST k- ω turbulence model was used.

The computational domain was divided into three regions as shown in Fig. 2. The first region is from nozzle exit to flat plate. The second region is on flat plate before the jet nozzle. The third region is on flat plate with constant heat flux. The grid was concentrated near the flat plate surface. The y^+ on flat surface was less than 2 for all cases.

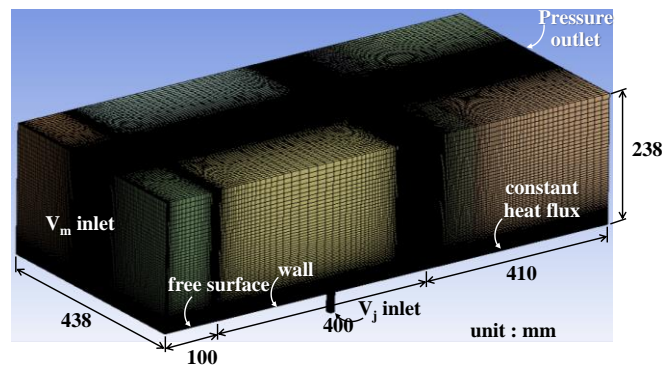


Figure 2. Computational model and grid generation

3.2. Computational verification

In order to verify computational results, the near wall velocity profiles that were measured with hot wire anemometry were compared with the simulation results. Figure 3 shows the comparison of velocity profiles above flat surface from experiment and simulation with no jet. The results show good agreement of both experimental and numerical results.

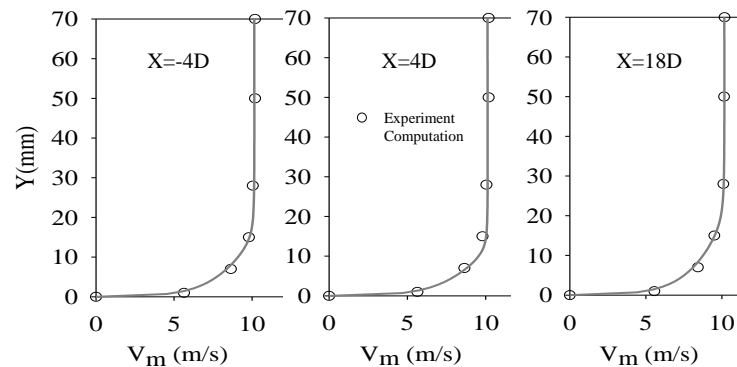


Figure 3. Comparisons of experimental velocity profile over flat plate of test section with computational result (no jet flow)

4. Results and discussions

4.1. Heat transfer

The h/h_0 from experimental results are shown in Fig. 4. For the case of $VR=0.25$, it is found that the area of high heat transfer peaks appeared into two region along downstream of jet. The characteristics of these two peaks were elongated parallel, along downstream direction. The highest heat transfer area was occurred at $X/D=1-3$ with h/h_0 up to 1.35 and gradually decrease along downstream direction. For closed area behind jet exit between $X/D=0.36\sim1$, h/h_0 became slightly increase when compare to the case of without jet. The jet effect on h/h_0 increasing in the surrounding area $Z/D=-4\sim-2$, $2\sim4$ was insignificant. With the increasing of VR to 3.5, the heat transfer enhancement region expanded more in spanwise and streamwise directions. The highest heat transfer area ($h/h_0=1.6$) was initial immediately after jet exit to $X/D=3$, and its contour geometry became candle flame shape. Moreover, the effect of jet stream can increase h/h_0 on the surrounding area.

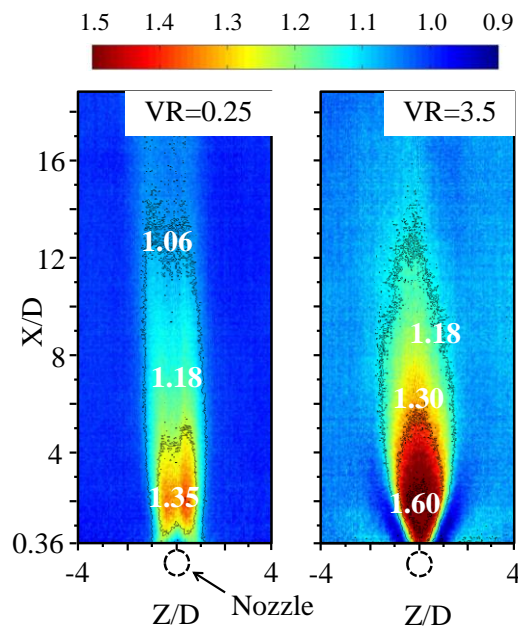


Figure 4. Effect of VR on contours of heat transfer coefficient ratio for vortex generating jet

4.2. Flow characteristics

The results of jet-mainstream mixing flow from simulation result are shown in figure 5. The two cases of VR=0.25 and VR=3.5 cases, jet stream path was considered fluid flow through jet exit and mainstream path was restricted pass normal plane with 2 mm height locate at upstream of jet exit position. After jet exit, jet stream lift away from surface for VR=3.5 case and move close with heat transfer surface for VR=0.25 case. For flow path of mainstream, it seem that mainstream flow covered over jet stream on VR=0.25 case and entrained upward mixing with jet stream for VR=3.5 case. From this effect, heat transfer surface was dominated from mainstream.

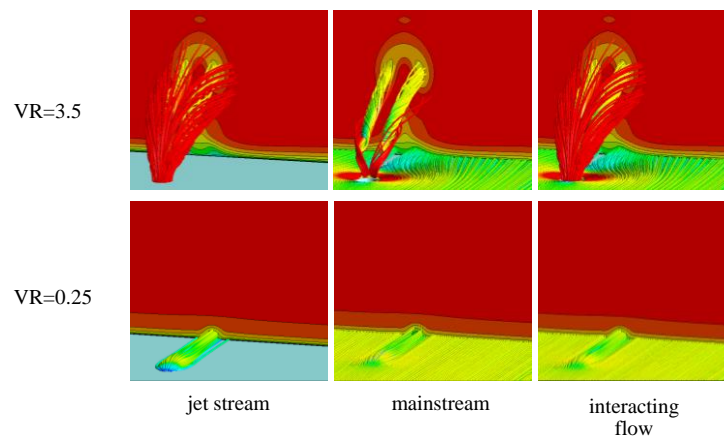


Figure 5. Flow path for jet, mainstream and interaction between jet and mainstream at different VR

4.3. Swirling strength

The swirling strength and vector of flow on a plane after jet exit are shown in Fig. 6. The counter rotating vortex pairs (CRVP) rotate close with surface and lift away for case VR=0.25 and VR=3.5, respectively. However, CRCP on the case VR=3.5 generate small rotation flow adjacent with surface and stronger than VR=0.25 case. From this result, the heat transfer rate becomes higher.

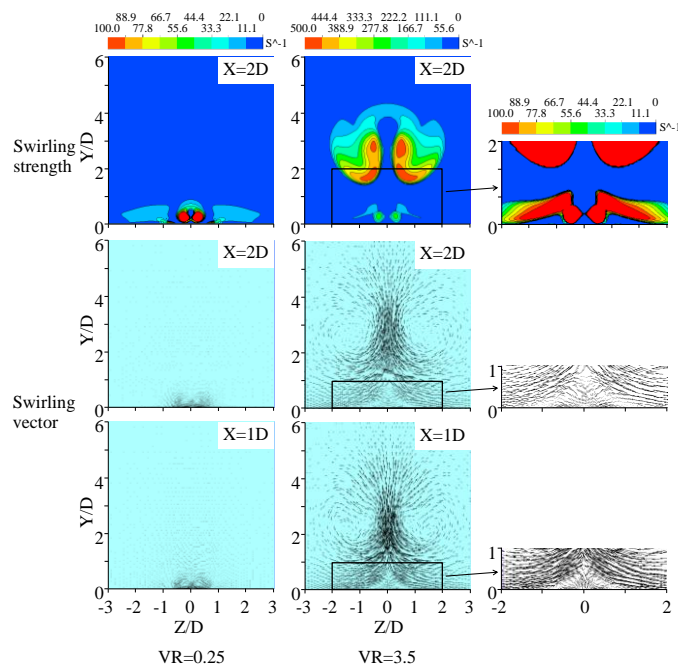


Figure 6. Swirling strength and vector of flow after jet exit

The flow mechanism after jet exit can be clearly explained from streamline on close parallel plane in Fig. 7. Figure 7 shows 3D streamline on ZX plane above heat transfer surface 1.0 mm at VR=0.25 and 3.5. For case of VR=0.25, it is clearly see the interaction between the jet flow and mainstream produce two of counter rotating flow. This two of rotating flow entrained the mainstream to attach on the heat transfer surface. This will result in two peaks of heat transfer behind the jet hole for this case. For case of VR=3.5, CRVP moved away from surface and introduced small vortices under upwash side. This phenomenon led heat transfer peak behind the jet exit.

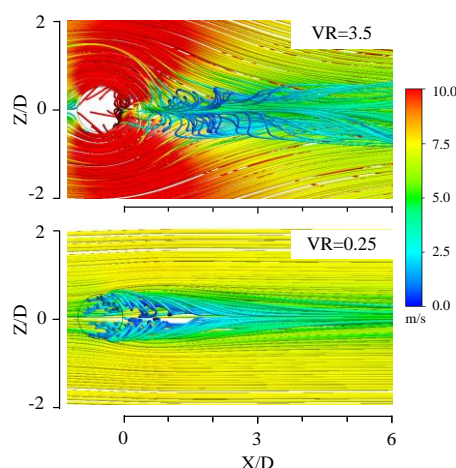


Figure 7. Plane streamline above heat transfer surface at 1 mm

4.4. Heat transfer mechanism on downstream surface

The temperature distribution closing to surface on normal planes after jet exit are shown in Fig. 8. The effect of CRVP to distribution of temperature at the centre of jet exit along downstream direction was different with beside. The different of thermal boundary layer thickness at close jet exit position was

higher than away position. On $VR=0.25$ case, the thickest and lowest point accorded with area between and two high heat transfer, respectively. For $VR=3.5$ case, the thermal boundary layer thickness was the thinnest along jet centre. This result occurred from mainstream that flow around jet stream and generate swirl flow to impinge heat surface.

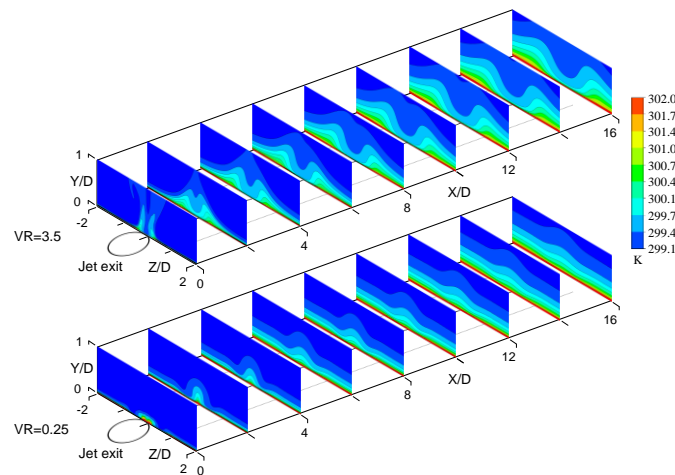


Figure 8. The temperature distribution closing to surface on perpendicular planes

5. Conclusion

In this paper, heat transfer enhancement on a flat plate in case of using a vortex generating jet from pipe nozzle was investigated for case $VR=0.25$ and 3.5 . The flow characteristics were explored with numerical simulation. The main findings are shown as follow:

1. High heat transfer area on $VR=0.25$ case occur from CRVP of jet stream to move close heat surface along downstream direction.
2. High heat transfer area on $VR=3.5$ case occur from CRVP of mainstream that flow around

References

- [1] Fiebig M 1998 *Trans IChemE* **76** 108-123
- [2] Aris M S, McGlen R, Owenc I and Sutcliffe C J 2011 *Appl. Therm. Eng.* **31** 2230-2240
- [3] Smulsky Y I, Terekhov V I and Yarygina N I 2012 *Int. J. Heat Mass Transfer* **55** 119-130
- [4] Cho H H, Wu S J and Kwon H J 2000 ribs *J Turbomach* **122** 579-586
- [5] Compton A D and Johnston P J 1992 *AIAA J* **30(3)** 640-647
- [6] Ahn J, Jung I S and Lee J S 2003 *Int. J. Heat Fluid Flow* **24** 91-99
- [7] Miao J-M and Wu C-Y 2006 *Int. J. Heat Mass Transfer* **49** 919-938
- [8] Qayoum A, Gupta V, Panigrahi P K and Muralidhar K 2010 *Int. J. Heat Mass Transfer* **53** 5035-5057
- [9] Jabbar M and Zhong S 2008 *Int. J. Heat Mass Transfer* **29** 119-130
- [10] Akdag U, Cetin O, Domiral D and Ozkul I 2013 *Int. C. Heat Mass Transfer* **49** 96-103