Heat Transfer Effect of Louvered Baffle in Jet Impingement Heat Transfer with Cross-flow

Sabu Kurian*, Tide P S and Biju N 1

Department of Mechanical Engineering, School of Engineering, Cochin University of Science and Technology, Cochin, Kerala-682022, India

*email: sabukurian74@gmail.com

Abstract. The objective of this research is to numerically investigate heat transfer characteristic of a louvered baffle assisted multi-jet impingement of air on a heated plate subjected to constant heat flux and cross flow. Two h/d configurations were considered for the present study. An array of jets with 3 x 3 configurations discharging from round orifices of diameter d=5 mm and with jet-to-target plate distance ranging from 2d to 3 d were studied. SST k-ω turbulence model was used for numerical simulation to examine the effect of blow ratio, louver angle and baffle clearance on heat transfer. Blow ratios of 0.25, 0.5, 075 and 1.0 and louver angle 30°, 45° and 60° were considered for CFD simulations. In all the cases, heat transfer increases with blow ratio and blow ratio with 1 has the maximum heat transfer. It is also noticed that heat transfer rate increases with decrease in louver angle h/d ratio.

1. Introduction
Jet impingement has been widely used in many fields such as turbine cooling, rocket launcher cooling and electronics cooling due to its intensive heat transfer characteristics. The air-jet impinging on a normal small surface will enhance heat transfer than any other conventional cooling techniques. Impinging jets have the three different zones as shown in Figure 1. a) core jet, b) free jet, c) stagnation point. Core jet has the same velocity that of the nozzle exit velocity and velocity decrease is observed due to the entrainment of outside air into the jet.

In multiple air jet impinging, the effect of nozzle geometry was experimented by Sinan Caliskan et al. (1). Heat transfer effects of rectangular and elliptical jets impinging over a target plate at Reynolds numbers varying from 2000 to 10000 were investigated by them. They found that, due to the augmented entrainment of surrounding air, performance of elliptical jets is better than rectangular jets. Nakod et al. (2) investigated the heat transfer characteristics of circular jet impinging over a finned flat plate and vortex generator. Cube type fins and equilateral triangle type vortex generators were used in this experiment. It is revealed from this study that vortex generators are better than cube shape finned surface. The effect of nozzle inlet geometry on heat transfer was experimentally and numerically investigated by M F Koseglu and S. Baskaya (3) and found that elliptical and rectangular jets are superior to other geometries. Mixing characteristics and entrainment phenomenon of interrupted jets were studied by Harekrishna Yadav et al.(4). They found that frequency and magnitude of pulsation jet determine the length of fully developed zone which characterizes the mixing and entrainment. Nusselt number distributions of a staggered five jet nozzle array were investigated by Jung-Yang san and Jenq-Jye Chen (5). They investigated about the influence jet to jet spacing and h/d ratio on heat transfer. Junsik Lee et al.(6) experimentally studied about the cross-flow effect with varying jet to jet spacing.
and h/d ratio on heat transfer. They noticed the deterioration of heat transfer with the increase of cross-flow.

Hollworth and Durbin investigated the application of jet impingement in electronics cooling [7]. Jet impingement on the vehicle windscreen for moisture removal was investigated by Roy et al. [8]. Jet impingement for the cooling of a grinding process was investigated by Babic et al. [9] and the methods of controlling the vortex flow of a free jet were reported by Hussain and Zaman [10], Ho and Huang [11]. Liu and Sullivan [12] have showed that, the heat transfer characteristics of a jet will be enhanced when the jet is excited acoustically at certain frequencies. Hwang et al. [13] investigated about the methods to regulate the vortex roll-up in the jet flow. Heat generation and dissipation in a grinding process were investigated numerically by Lavine and Jen [14], [15], Jen and Lavine [16], [17] and Liao et al. [18] and proposed a new model. Smoke wire technique was employed by Fleischer et al. [19] to visualize the initiation and development of vortices in an impinging jet flow. Weigand and Spring [20] investigated the heat transfer behavior of a multi jet impingement and analyzed the results with single jet. Suitability of different CFD models for multiple impinging jet system was also analyzed by them. Influence of the initial cross-flow on both heat transfer was studied by Florschuetz et al. [21, 22] and Florschuetz and Su [23, 24]. They found that the influence of the initial cross-flow depends primarily on h/d ratio and the blow ratio. They also observed the overall decrease in heat transfer due to cross-flow. Roa et al. [25] and zu et al. [26] reported that SST k-ω model is best suitable for modeling multiple impinging jets due to the good agreement between numerical and experimental data at fair costs.

Heat transfer can be further augmented by providing ribs, baffles, vortex generators etc in the flow domain. Lot of experiments have been noticed with ribs and vortex generators. Insertion of baffles may be a contributing factor, especially when cross flow is present. It has been noticed that baffles create extra turbulence in the flow [27]. Turbulence is directly proportional to heat transfer in most cases. From the elaborate literature review, it can be concluded that, the fundamental parameters affecting multi jet impingement heat transfer is known. But role of baffles and louvers were not properly investigated in the past.

This study focuses on numerical and experimental investigation on the heat transfer properties of 3 x 3 jet array impingement over a flat plate with cross-flow and louvered baffle. Investigation was carried out at different cross-flow velocities, target plate distance (h/d) and louver angles. A clearance of 1 mm is provided between the baffle and target plate which is constant throughout this research. Thermocouples are deployed at different locations of the target plate to measure the temperature. Constant velocity air jets are used throughout the experiment. Cross-flow velocities have been varied at four levels. Mean while louver angle and target plate distance varied at 3 and 2 levels respectively. Numerical techniques were used to simulate the flow patterns under different conditions by using suitable turbulence models and the predictions were compared with experimental measurements. Commercial CFD package ANSYS Fluent is used to conduct the numerical study.
2. Experimental setup and procedures

Important parts of the present test facility are test section fitted with data acquisition system, and system to develop high speed jet flow, nozzle, cross-flow arrangement and an impingement surface. The schematic diagram of the test facility is shown in Figure 2. Blower (3 hp) draws air from atmosphere and supplies to the air pre-heater, where it is pre-heated to a fixed temperature of 305 K. One portion of the air moves to the plenum chamber to produce air jets, while another portion passes to the cross-flow duct. Separate orifice-meters and manometers are provided in the plenum chamber and cross-flow duct pipe line. The target plate is an aluminium flat impingement surface instrumented with thermocouple and foil type 50 W foil type heater, is mounted on a z traverse which can be moved in vertical direction. Movement of the target plate in the vertical direction is used to fix the h/d ratio. Dimension of the target plate is 120 x 120 mm. Temperature of the target plate is measured with 18 K-type thermocouples deployed at the different locations. DAQ is connected to a personal computer and all thermocouples, to record the real time temperature. All temperature and flow measuring devices are NABL calibrated.

Nine nozzle of five mm diameter is used. The length of the nozzle is 10mm. The cross section of the cross-flow region is 120 x 10 mm. Baffle with louvers are used with louver angle 45°. Estimated uncertainty of jet velocity is ±5.385 %.

Blow ratio is defined as the ratio of cross-flow velocity to jet velocity. Experiments were conducted with four blow ratios ranging from 0.25 to 1.0. Four louvered baffles having a length of 60 mm are attached alternatively on either side of the confinement. Louver angle for the experiment is fixed as 45°.

![Figure 2 Schematic diagram of experiment setup](image)

**Figure 2** Schematic diagram of experiment setup [Parts list: 1. 3 hp blower, 2. Air Pre-heater, 3. & Valves, 5, 6. Orifice-meters, 7. Plenum chamber, 8. Nozzle plate, 9. Target plate, 10. Z-traverse, 11 Thermocouples, 12. DAQ, 13. Computer]

3. Numerical work

3.1. Geometry and boundary conditions

The domain of the CFD model is shown in figure 4. The domain is a three dimensional space consist of an inline 3 x 3 nozzles array, a target plate of 120 x 120 mm, made of aluminium and louvered baffles. Of the four sides, two sides are open for the entry and exit of cross-flow, while other two sides are confined. Air inlet into the domain is through 9 nozzles with diameter, d=5 mm and through the cross-flow inlet section. Domain discretisation is done through structured hexahedral mesh. Grid study was conducted by varying the cell size from 0.97 million to 1.87 million cells which shows that grid having 1.3 million cells was the best suited for present study. Viscous, steady, incompressible, and three dimensional are the initial assumptions of the study. Also gravitational effects, radiation and viscous dissipation are neglected. Air properties such as specific heat, density, and thermal conductivity are considered constant.
A heat flux of 3472 W/m² is supplied to the target plate. All the confinement walls of the domain are adiabatic in nature. ‘Velocity inlet’ boundary conditions were given at the nozzle inlet and cross-flow inlet. Pressure outlet is the exit boundary condition at the exit of domain. Air velocity at the inlet of nozzle is fixed as 38 m/s and cross-flow velocity varies with blow ratio. Temperature of air at the inlet is kept at 305 K. For the numerical analysis, h/d considered is 2 and 3 and louver angles are 30°, 45°, 60°.

Commercial CFD package, ANSYS Fluent is used to carry out the numerical simulation. Second order discretization scheme was used for the pressure, continuity, and momentum. For pressure velocity coupling, standard coupled algorithm is adopted. Turbulence model selected is SST k-ω as it provides a good balance between accuracy and computational time, steady state impingement is assumed throughout this simulation. To show temperature and Nusselt number in numerical results, a diagonal line is considered over the heated plate as shown in Figure 3.

Figure 3 Wireframe model with location of louvered baffles and jets and diagonal line of study

3.2. Grid study
Grid independence study is performed to reduce the influence of the number of grids on the computational results. It is always a good practice to follow this for every geometry which is tedious. Grid independence is achieved at 1314937 elements. Table 1 showing average temperature variation at different grid elements, temperature at .55mm and .5mm is almost same. So current study deals with edge element of 0.55 were chosen as a safe limit for the created model.

Table 1 Grid independence study

| Edge sizing          | 0.65mm | 0.6mm | 0.55mm | 0.5mm |
|----------------------|--------|-------|--------|-------|
| Average Temperature (K) | 328.24 | 327.656 | 323.609 | 323.65 |
| No of cells           | 972154 | 1120796 | 1319521 | 1877774 |

3.3. Validation
Numerical results are validated against the corresponding experimental study, by considering the temperature at specific location of target plate. Temperature obtained from the experimental and numerical investigation is shown in Figure 4. Louver angle of 45° and blow ratio 0.25 is the case considered for the validation. It can be concluded that experimental results are in close agreement with numerical result and experimental results and the maximum deviation is limited to ±4.38 %.
4. Results and discussion

4.1. Experimental results

Average surface temperature for the blow ratio 1, 0.75, 0.5 and 0.25 determined from experiment at h/d of 2 and louver angle 45° are shown in the Table 2. It is evident that, as the blow ratio increases, the average temperature decreases and hence the heat transfer increases. The high velocity air coming from the nozzle mixes with cross-flow air and passes through the baffles which create turbulence above the target plate which results in more heat transfer.

Table 2 Experimentally determined average surface temperature

| BR  | Average surface temperature (K) |
|-----|---------------------------------|
| 1   | 316.167                        |
| 0.75| 318.472                        |
| 0.5 | 320.732                        |
| 0.25| 324.532                        |

Figure 5 shows the variation of surface temperature along the reference diagonal line with respect to the changes in the blow ratio. In all the cases, as the blow ratio increases, surface temperature at the salient points also increases. General trend in the changes of surface temperature is almost same and hence, we could infer that blow ratio is the important factor which determines the surface heat transfer.

Average surface heat transfer coefficient (h) in W/m²K and surface Nusselt number (Nu) is estimated using the following relations $q = h A (T_s - T_\infty)$ and $Nu = (hd)/k$, where $q = $ Heat flux, $3472.2$ W/m², $A =$
Area of plate (120 x 120mm), $T_s =$ Average surface temperature from 18 thermocouples, (K), $T_e =$ Ambient temperature, 305K, $d =$ Jet diameter, 5mm and $k =$ Thermal conductivity of fluid, W/m K.

Experiments were performed for the validation purposes only and the rest of the experiments were conducted numerically.

4.2. Numerical results

Total number of cases considered for the present study is 24, keeping blow ratio, louver angle and h/d ratio as the variable. Blow ratio considered at 4 levels such as 0.25, 0.5, 0.75 and 1.00, louver angle at 3 levels such as 30°, 45° and 60° and h/d at 2 levels such as 2 and 3.

From Table 3, it is evident that, heat transfer is directly proportional to blow ratio. An appreciable Increase in Nusselt number can be noticed between the blow ratio 0.5 and 0.75 while the increase in Nusselt number between the blow ratio 0.75 and 1.00 is minimal. This is due to the increased turbulence along with the increase in cross-flow velocity. An increase in Nusselt number can be noticed with decrease in louver angle. This happens due to the increase in cross-flow deflection. Increasing h/d ratio from 2 to 3 brings decrease in Nusselt number. As the h/d ratio increases, the jet travel distance increases and downward shift in the impingement location causes the delay in impingement. Increase in h/d ratio decreases the jet momentum is also a factor which cause decreased heat transfer rate.

| Table 3 | Numerically determined average surface temperature and Nusselt number |
|---------|------------------------------------------------------------------|
| h/d ratio | Louver Angle | Blow Ratio | Average Temperature (K) | Average Nusselt Number |
| 2 | 30° | 0.25 | 319.203 | 50.51022 |
| | | 0.5 | 313.479 | 84.60864 |
| | | 0.75 | 306.632 | 439.5813 |
| | | 1 | 306.596 | 449.4967 |
| | 45° | 0.25 | 320.229 | 47.10728 |
| | | 0.5 | 314.682 | 74.09592 |
| | | 0.75 | 307.164 | 331.5142 |
| | | 1 | 307.092 | 342.9239 |
| | 60° | 0.25 | 321.687 | 42.99135 |
| | | 0.5 | 316.356 | 63.17336 |
| | | 0.75 | 308.164 | 226.808 |
| | | 1 | 308.097 | 231.6425 |
| | 3 | 30° | 0.25 | 321.164 | 44.38237 |
| | | 0.5 | 314.251 | 77.54802 |
| | | 0.75 | 306.912 | 375.2075 |
| | | 1 | 306.826 | 392.8788 |
| | 45° | 0.25 | 322.015 | 42.1626 |
| | | 0.5 | 315.337 | 69.40086 |
| | | 0.75 | 307.238 | 320.5526 |
| | | 1 | 307.222 | 322.8608 |
| | 60° | 0.25 | 322.959 | 39.94636 |
| | | 0.5 | 316.126 | 64.4793 |
| | | 0.75 | 307.631 | 272.6707 |
| | | 1 | 307.624 | 273.3981 |

4.3. Streamline

For better understanding the nature of flow, velocity streamlines of cross-flow and air jets are shown separately in Figures 6 and 7 corresponding to blow ratio 0.25, 0.5, 0.75, 1.00. Figure 6 exhibits the variations in cross-flow velocity due to the effect of louvered baffles. Stream lines originating from the
jets are kept hidden. Presence of baffles largely affects the flow dynamics in the domain. In all the cases, it is evident that, as the flow passes across the baffle, there is acceleration in the velocity.

Figure 7 shows the velocity stream lines of the jet air movement, keeping the cross-flow steam lines hidden. Stream lines originating from jets seems to be quite complex. Flow emanating from jets gets deflected toward the downstream direction, especially in the region where jets get exposed directly to cross-flow. But jets protected by the baffles show better impingement on the target plate. Velocity of the flow gets accelerated towards the downstream in all the cases. Figure 8 shows the flow dynamics of combined jet flow and cross-flow.

![Figure 6 Velocity stream lines of cross flow alone for different blow ratios](image1)

![Figure 7 Velocity stream lines of jet flow alone for different blow ratios](image2)
4.4. Temperature contours

Figure 9 and 10 shows the surface temperature contours for blow ratio 0.25, 0.5, 0.75 and 1.00 at h/d ratio 2 and 3 and constant louver angle 30°. For h/d ratio 2 and blow ratio 0.25, temperature drop at the point of jet impingement is clearly visible. As the blow ratio increases, temperature drop beneath the jet location slowly disappears showing the wash away of impinging jets by the cross-flow. It could be inferred that, heat transfer due to jet impingement deteriorates as cross-flow velocity increases. Cross-flow becomes the dominating factor for increased heat transfer, as blow ratio increases. For h/d = 3 and blow ratio 0.25, temperature drop at the point of jet impingement is less visible compared to previous case. Momentum loss due to the increased jet travel distance is the reason for that. This comparison reveals that, jet impingement cooling with 1 cross-flow is best suited for low h/d applications.

Presence of baffle creates two distinct region of temperature difference in target plate. In the upstream side, baffle creates blockage in the cross-flow passage causes decrease in flow velocity and increase in temperature. Flow through baffle clearance and louver hole creates increased velocity towards the downstream of baffle. Increased jet velocity scores away the boundary lay and causes increased heat transfer and reduced temperature. This phenomenon is same in both h/d ratios considered in this research.

4.5. Temperature variations along reference line

Figures 11, 12 and 13 shows the local surface temperature on the target plate at louver angle 30°, 45° and 60° along diagonal line considered as reference for this study. Axis of three diagonal jets passes through reference line. For blow ratio of 0.25, jet impingement at the target plate is evident, represented by the sharp dip in temperature. Decrease in temperature at the center jet impingement location is less sharp because of larger mass flux concentration in that area causes sweep away of jets in that location. As blow ratio increases, temperature drop at the impingement location decreases due to the larger interruption by the cross-flow on jets. For blow ratio 0.75 and 1.00, the temperature profile is almost flat and same which is due to the extensive sweep away of incoming jet flow. For higher louver angle and lower blow ratios, temperature drop at the impingement locations become shallow as shown in Figures 12 and 13. But secondary impingement from the louvers is covered by larger area. For larger blow ratio, temperature profile is almost same at all louver angles.

Figure 14 represents the graphical representation of overall performance of the jet impingement with louvered baffles. Among the considered cases, jet impingement with h/d 2, louver angle 30° and blow ratio 1 has the lowest average surface temperature (306.596 K). Meanwhile, jet impingement with h/d 32, louver angle 60° and blow ratio 0.25 has the highest average surface temperature (322.959 K).
Figure 9 Temperature contours for different blow ratios at h/d=2 and louver angle 30°

Figure 10 Temperature contours for different blow ratios at h/d=3 and louver angle 30°
Figure 11 Temperature along diagonal line for h/d = 2 and louver angle 30°

Figure 12 Temperature along diagonal line for h/d = 2 and louver angle 45°

Figure 13 Temperature along diagonal line for h/d = 2 and louver angle 60°
Figure 14 Comparison of average temperature at different blow ratios and louver angles at h/d=2

5. Conclusion
Numerical and experimental investigations of jet array impingement on a flat plate with louvered baffle and cross-flow subject to constant heat has been carried out. Effect of blow ratio, h/d ratio and louver angle on heat transfer were the factors considered in the investigation. From the detailed study, it has been found that, numerical investigation is in good agreement with the results obtained from the experiment. Maximum deviation of average surface temperature between experimental and numerical results limited to 3.55%. Important observations derived from this study are:

- As the blow ratio increases, heat transfer from the target surface increases. This is due to the increased turbulence as the flow passes through the baffles. As the blow ratio increases, cross flow become the dominating factor in heat transfer. This is due to the sweep away of impinging jets at higher blow ratio.
- Nusselt number increases with decrease in louver angle. This happens due to the increase in cross-flow deflection. Decrease in louver angle causes creates a secondary impingement zone immediately after the baffle with will increases the heat transfer.

Increasing h/d ratio from 2 to 3 brings decrease in Nusselt number. As the h/d ratio increases, the jet travel distance increases and downward shift in the impingement location causes the delay in impingement. Increase in h/d ratio decreases the jet momentum is also a factor which cause decreased heat transfer rate.

References
[1] Sinan Caliskan, Senol Baskaya and Tamer Calisir, “Experimental and numerical investigation of geometry effects on multiple impinging air jets,” International journal of heat and mass transfer (2014)75 pp. 685-703.
[2] P M Nakod, S V Prabhu and R P Vedula, “Heat transfer augmentation between impinging circular air jet and flat plate using finned surfaces and vortex generators,” Experimental thermal and fluid science 32 (2008)1168-1187.
[3] M F Koseoglu and S Baskaya, “The role of jet inlet geometry in impinging jet heat transfer, modeling and experiments,” International journal of thermal science49(2010)1417-1426.
[4] Harekrishna yadav, Amit Agrawal and Atul Srivastava, “Mixing and entrainment characteristics of a pulse jet,” International journal of heat and fluid flow000(2016)1-13.
[5] Jung-Yang San and Jenq-Jye Che, “Effects of jet to jet spacing and jet height on heat characteristics of an impinging jet array,” International journal of heat and mass transfer71(2014)8-17.
[6] Junsik Lee, Zhong Ren, Phil Ligrani, Dae Hee Lee, Michael D. Fox and Hee-Koo Moon, “Cross flow effects on impinging array heat transfer with varying jet-to-target plate distance and hole spacing,” International journal of heat and mass transfer75(2014)534-544.
[7] Hollworth, B. R and Durbin M., “Impingement cooling of electronics," *ASME Journal of Heat Transfer* (1992) 114, pp. 607-613

[8] Roy S, Nasr K, Patel P and AbdulNour B, "An experimental and numerical study of heat transfer of an inclined surface subject to an impinging air flow," *International Journal of Heat and Mass Transfer* (2002) 45, pp. 1615 - 1629.

[9] Babic, D. M., Murray, D. Band Torrance, A. A, “Mist jet cooling of grinding processes,” *International Journal of Machine Tools and Manufacture* (2005) 45, pp. 1171 – 1177

[10] Hussain, A. K. M. Fand Zaman, K. B. M. Q., “Vortex pairing in a circular jet under controlled excitation. part 2. coherent structure dynamics," *Journal of Fluid Mechanics* (1980), 101, pp. 493 - 544.

[11] Ho, C. Mand Huang, L. S., “Sub harmonics and vortex merging in mixing layers," *Journal of Fluid Mechanics* (1982) 119, pp. 443 – 473

[12] Liu, Tand Sullivan, J. P., “Heat transfer and flow structures in an excited circular impinging jet,” *International Journal of Heat and Mass Transfer* (1996) 39, pp. 3695 – 3706

[13] Hwang, S. D, Lee, C. Hand Cho, H. H., “Heat transfer and flow structures in axisymmetric impinging jet controlled by vortex pairing," *International Journal of Heat and Fluid Flow*, (2001)22, pp. 293 – 300

[14] Lavine, A. Sand Jen, T. C., 1991, Thermal aspects of grinding: heat transfer to work piece, wheel and fluid," *ASME Journal of Heat Transfer*, (1991)113, pp. 296 -303

[15] Lavine A. S., “An exact solution for surface temperature in down grinding, "*International Journal of Heat and Mass Transfer*, (2000)43, pp. 4447 – 4456

[16] Jen, T. Cand Lavine, A. S., “A variable heat ux model of heat transfer in grinding: model development," *ASME Journal of Heat Transfer*, (1995)117, pp. 473-478

[17] Jen, T. C and Lavine, A. S., “A variable heat u model of heat transfer in grinding with boiling,” *ASME Journal of Heat Transfer*, (1996) 118, pp. 463 – 470

[18] Liao, Y. S and L. S. Y. Y. T. H., “A thermal model of the wet grinding process," *Journal of Materials Processing Technology*. (2000)101, pp. 137 – 145

[19] Fleischer, A. S., Kramer, Kand Goldstein, R. J., Dynamics of the vortex structure of a jet impinging on a convex surface, "*Experimental Thermal and Fluid Science*, (2001)24, pp. 169 – 175

[20] Bernhard Wigand, Sebastian Spring," Multiple Jet Impingement — A Review”, Heat Transfer Research, 2011, Vol. 42, No. 2, pp. 101-142

[21] Florschuetz, L. W., Metzger, D. E., Su, C. C., Isoda, Y., and Tseng, H. H. Jet Array Impingement Flow Distributions and Heat Transfer Characteristics. Effects of Initial Cross-flow and a Non-uniform Array Geometry, NASA-CR-3630, 1982

[22] Florschuetz, L. W., Metzger, D. E., and Su, C. C. Heat transfer characteristics for jet array impingement with initial cross-flow, *ASME J. Heat Transfer*, 1984, Vol. 106, pp. 34–41.

[23] Florschuetz, L. W. and Su, C. C. Heat Transfer Characteristics within an Array of Impinging Jets. Effects of Cross-flow Temperature Relative to Jet Temperature, NASA-CR-3936, 1985

[24] Florschuetz, L. W. and Su, C. C. Effects of cross-flow temperature on heat transfer within an array of impinging jets, *ASME J. Heat Transfer*, 1987, Vol. 109, pp. 74–82

[25] Rao G. A., Kitron-Belinkov M., and Levy Y. Numerical analysis of a multiple jet impingement system. In: Proc. ASME Turbo Expo 2009: Power for Land, Sea, Air, June 8–12, Orlando, Florida, USA, GT2009-59719, 2009

[26] Zu Y. Q., Yan, Y. Y., and Maltson, J. D. CFD prediction for multi-jet impingement heat transfer. In: Proc. ASME Turbo Expo 2009: Power for Land, Sea Air, June 8–12, Orlando, Florida, USA, GT2009-59488, 2009

[27] Berner, Durst, F., and McEligot, D. M., 1984, "Flow Around Baffles," ASME journal of heat transfer, Vol. 106, pp. 743-749