Experimental and computational study of fluid flow on a flat plate with three rectangular impinging slot jets

Malladi R Ch Sastry*a, B V S S Prasadb & A V S K S Gupta

aDepartment of Mechanical Engineering, Gudlavalleru Engineering College,
Gudlavalleru 521 356, India

bThermal Turbo Machines Laboratory, Department of Mechanical Engineering,
Indian Institute of Technology Madras, Chennai 600 036, India

cDepartment of Mechanical Engineering, JNTUH College of Engineering, Hyderabad 500 085, India

Received 1 December 2014; accepted 7 July 2015

Computational and experimental studies on a flat plate are carried out to investigate flow characteristics with an array of three rectangular air jets impinging on its top surface. Pressure distribution on the impingement surface and flow visualization experiments with oil lamp black technique are carried out to qualitatively corroborate the computational flow structure with visualized patterns. Computations are carried out by shear stress transport (SST) \( \kappa - \omega \) turbulence model. The jet-to-plate spacing (\( H \)) to hydraulic diameter of slot (\( D \)) ratio (\( H/D \)) is varied from 0.5 to 4.0 and jet Reynolds number (\( Re \)) is varied from 5282 to 8450. The computationally obtained flow structure reveals the complex interaction of the wall jets. Primary pressure peaks are observed at the stagnation point and the secondary peaks are noticed at the interaction points of the wall jets. Position of the secondary peaks depends upon \( H/D \) and is independent of Reynolds number.

Keywords: Jet impingement, Array jet, Flow visualization, Experiments, CFD

Impinging jets have been widely used in many industrial applications to enhance the heat transfer rate in heating, cooling or drying processes. Applications include drying in paper industry, cooling and heating in food industry, annealing metallurgy, and cooling of heated components in gas turbine engines, computers and electronic instruments. A single air jet and an array of jets are used for different purposes. A single jet is usually employed to produce localized heating and cooling. In many applications, a large surface area is required to be heated or cooled, or enhancement of global heat transfer is needed. Thus, it is necessary to apply multiple-jet system. The ability to control heat transfer from the surface by varying flow parameters such as jet exit velocity and geometrical parameters such as jet exit opening, jet-to-plate spacing, and nozzle-to-nozzle spacing in arrays are some of the key factors that have lead to the sustained and widespread use of jet impingement technologies. The most commonly used geometries are axisymmetric (circular orifice or pipe), slot (two-dimensional) nozzles, rectangular (three-dimensional) and square (three-dimensional) nozzles.

Many previous studies presented about this subject are concerned with heat transfer rates of unconfined and confined single and multiple jets impinging on a plate. However, little is reported about flow structure of the multiple impinging jets. Popiel and Trass studied the vortex structure of the impinging round jets by the smoke wire flow visualization technique and the observed changes in the vortex structures with \( H/d \) values were attributed to explain the differences in the transport rate values. Lytle and Webb have investigated experimentally the flow structure and heat transfer characteristics of air jet impingement for nozzle-to-plate spacing's less than one nozzle diameter in the Reynolds numbers \( 3600 < Re < 27,600 \). An experimental investigation to determine velocity and turbulence characteristics of a confined impinging slot jet has been made by Ashfort-Frost et al. and measured low levels of turbulence in the stagnation region and its subsequent development in the wall jet, suggesting that transition to a turbulent wall jet occurs when the impingement plate is placed within the potential core of the jet. Chimiya and Yamada have reported that the presence of the recirculation regions on both impingement and confinement surfaces for low spacings and that, with

*aCorresponding author (E-mail: mrchsastry@gmail.com)
the increase in Reynolds number and nozzle-to-plate spacing, the recirculation flow on the impingement surface moves downstream and its volume increases correspondingly. Tawfek\textsuperscript{6} reported a monotonic trend of decreasing pressure coefficient ($C_p$) with increasing radial distance on the surface of an isothermal flat plate, where the jet was impinging normal to it and concluded that the pressure distributions along the impingement surface are similar and closer to the heat transfer variations at the same configurations for the nozzle-to-plate spacings $H/D > 6$. An experimental investigation on heat transfer characteristics of slot jets impinging on a cylinder was made by Nada\textsuperscript{7}. Behnia \textit{et al.}\textsuperscript{8} noted that the effect of confinement is only significant in low nozzle-to-plate spacings. Only a few studies paid attention to the sub-atmospheric region. Bernard \textit{et al.}\textsuperscript{9} made a comparison between flow visualization results and wall pressure measurements with jets impinging on a plane wall. Experimental studies highlight links between a quantitative approach such as the wall pressure coefficient distribution and a qualitative one such as wall streamline visualizations by the spreading over method. In the studies made by Baydar\textsuperscript{10} and Baydar and Ozmen\textsuperscript{11} for Reynolds numbers ranging from 500 to 50,000 in confined impinging jets, it has been observed that a sub-atmospheric region occurs on the impingement plate for $Re > 2700$ and the nozzle-to-plate spacing less than 2 and that there existed a linkage between the sub-atmospheric region and the peaks in local heat transfer coefficients. Koopman and Sparrow\textsuperscript{12} noted that there are two types of interaction between the jets in the multiple-jets system. The first type was the interaction between adjacent jets prior to impingement on the surface. The second type was the collision of two wall jets, which were generated after impingement. Such collision became significantly important when the jets were closely spaced, the nozzle-to-plate spacing was small, and the jet velocity is high. Huber and Viskanta\textsuperscript{13} have examined the influence of spent air exits located between the jets on local heat transfer coefficient for a confined impinging array of air jets. Saad \textit{et al.}\textsuperscript{14} compared the turbulence, mean flow and heat transfer characteristics of an array of confined impinging slot jet with those of a single jet. Barata\textsuperscript{15} investigated the flow field created by twin jets impinging on a ground plane with laser Doppler measurements. Results show the presence of a complex 3D scarf vortex formed around the impinging jets. Many numerical studies have been made on a jet impinging on a flat plate using various turbulence models. Dianat \textit{et al.}\textsuperscript{16} made predictions of axisymmetric and two-dimensional impinging turbulent jet with a Standard $k$–$\varepsilon$ turbulence model. The flow and heat transfer characteristics of impinging laminar square twin jets have been numerically investigated by Aldabbagh and Sezai\textsuperscript{17}. The calculated results show that the flow structure of square twin jets impinging on a heated plate is strongly affected by the jet-to-plate distance. Panda \textit{et al.}\textsuperscript{18} studied experimentally and computationally the detailed flow structure and the effect of jet-to-plate spacing for round jets on the pressure distribution and flow structure. Ozmen\textsuperscript{19} studied experimentally flow characteristics of confined twin jets issuing from the lower surface and impinging normally on the upper surface at high Reynolds number ranging from 30000 to 50000.

The broad objectives of the paper are therefore to computationally investigate the effect of varying $H/D$ and Reynolds number (i) the flow structure of the interacting jets, (ii) pressure distribution. Flow visualization experiments with oil-lampblack technique are carried out to qualitatively corroborate the computational flow structure with visualized patterns. Similarly the computational pressure distributions are compared with measured data.

**Experimental Procedure**

Figure 1 shows the schematic diagram of experimental set-up. The important parts of the set-up are: the air supply system, test section and instrumentation. The air supply system consists of reciprocating air compressor, flow regulating valves and the air supply lines.
A twin cylinder reciprocating air compressor is installed with a 0.32 m³ receiving tank to minimize the fluctuation in the output of the compressor. The compressor provides a maximum free air delivery of 40 m³/h at a maximum pressure of 8 bar. It is run by a three phase induction electric motor with stabilized power supply. A Bourdon gauge (0-12 kgf/cm²) is used to measure the tank pressure.

A conical diverging section is used to connect the 25 mm diameter main supply line to the test section. The included angle of diverging section is maintained at 7° to avoid flow separation. A seamless rectangular chamber of 600×300×300 mm and 6 mm thick with one side closed, large enough to act as a plenum chamber or settling chamber to produce uniform pressure and reduce the fluctuations. The dry air from the main supply line enters the plenum chamber through the hose pipe via a rotameter. The rotameter (0-50 m³/h) is used for flow measurement in the supply line.

The impingement chamber is fastened to the plenum chamber. It has the internal dimensions of 160×160×160 mm with 4 mm thick plate. The exit side of the impingement chamber has semi-confining plate which contains three numbers of 6×12 mm rectangular orifices (slots) through which air exits, as shown in Fig. 2. A 160 × 160 mm square plate made of 5 mm thick acrylic (Perspex sheet) is used as target plate, on which jet is impinged from the chamber. The geometry details of the impingement chamber and the target plate are given in Table 1. In order to determine the static pressure distribution on the target surface, a close array of pressure tap holes of 0.8 mm diameter is drilled along the plate. The tap holes are closely placed because of the steep change of pressure expected due to jet interactions. Impingement plate is placed over a stand, whose height can be varied by a lead screw mechanism to obtain the required H/D ratio. Static pressure on the impingement plate is measured by multi tube manometer.

Flow visualization experiments are carried out with oil-lampblack technique. The oil and lampblack powder are mixed in proper proportion and painted on the target surface. The lampblack coated target surface is kept beneath the plenum chamber. The air jets, after impinging on the target surface, leave traces of the flow path lines on the plate. These pictures are captured by a high resolution camera.

### Computational Methodology

Figure 2 shows the geometric model that mimics the physical model used for experiment of interest. It consists of a semi-confined plate and 160×160 mm square impingement plate. The semi-confining plate consists of a row three rectangular jets of 6×12 mm in size. The jet ejects from the rectangular orifice to impinge on the target plate. Jet-to-plate spacing is varied to obtain the H/D ratio from 0.5 to 4.0.

A computational mesh suitable for finite volume method is generated using the automatic generating tool GAMBIT 2.3.16. The important features of the mesh are (i) structured mesh generated for the entire domain but includes coopered wedge elements where appropriate, and (ii) fine clustered mesh generated near all the solid walls, at the jet interface, and in the up-wash region. Figure 3 shows the computational domain with mesh.

The three dimensional flow is formulated with the following assumptions: (i) the fluid is incompressible, (ii) the fluid properties are constant and (iii) the mean

<table>
<thead>
<tr>
<th>Description</th>
<th>Dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rectangular slot (l × b)</td>
<td>12 × 6 mm</td>
</tr>
<tr>
<td>Hydraulic diameter of slot (D)</td>
<td>8 mm</td>
</tr>
<tr>
<td>Impingement plate length</td>
<td>160 mm</td>
</tr>
<tr>
<td>Impingement plate width</td>
<td>160 mm</td>
</tr>
<tr>
<td>Jet-to-plate spacing (H)</td>
<td>4 mm, 8 mm, 16 mm, 32 mm</td>
</tr>
<tr>
<td>Intra jet spacing</td>
<td>32 mm</td>
</tr>
</tbody>
</table>

![Fig. 2 – Physical model with input conditions](image1)

![Fig. 3 – Computation domain with mesh](image2)
Flow is steady. The governing equations used for simulations are the Reynolds averaged continuity and momentum equations for modeling the turbulence quantities. The equations are as follows:

Continuity equation, \( \nabla \cdot (u) = 0 \) \( \ldots (1) \)

Momentum equation, \( (u \cdot \nabla)u - \frac{1}{\rho} \nabla p + \nu \nabla^2 u = 0 \) \( \ldots (2) \)

Turbulent kinetic energy (K) equation,
\[
\frac{\partial (k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + \nabla k - Y_k + S_k \quad \ldots (3)
\]

Specific dissipation rate (\( \omega \)) equation
\[
\frac{\partial (\omega u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + \nabla \omega - Y_\omega + D_\omega + S_\omega \quad \ldots (4)
\]

Where \( \Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \), \( \Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \), \( \sigma_k,1 = 1.176, \sigma_\omega,1 = 1.168, \sigma_k,2 = 1.0, \sigma_\omega,2 = 1.168 \)

For the purpose of computation, a specified mass flow conditions is imposed at the inlet. A turbulence intensity of 5%, is chosen for the inlet fluid. Ambient air of constant temperature at 308 K is specified as the inlet fluid. On all other solid surfaces such as plenum cylinder and jet orifices, the adiabatic wall conditions is imposed. Constant (ambient) pressure outlet condition is applied on all outlet boundaries. Atmospheric pressure and temperature of 308 K are applied at these outlets. All the input values imposed are those used in the experiment.

A finite volume based solver fluent 6.3 is used for solving the governing continuity, momentum equations and turbulence model equation. Flow is considered to be in compressible and constant properties are used because of small variation in temperature and pressure. The solution is considered as converged when the residuals falls below \( 10^{-6} \) for momentum continuity and turbulence equations. Change in static pressure at a stagnation point is continuously monitored so that there will be no change in the value for consecutive 300 iterations after the solution is converged.

Grid independence study is done to find the optimum grid size and distribution. The grid independence study is carried out by analyzing the variation of static pressure at the stagnation point for the case of \( H/D = 2.0 \) and \( Re 6340 \) listed in Table 2, for different mesh sizes. It is clear from the table that difference in the value of static pressure with 1.691 million and 1.99 million is not significant. In order to reduce the computational cost, the 1.691 million is selected as final grid. Similar grid independence study for other cases as well.

The parameters investigated in the present study include (i) effect of \( H/D \), varied as 0.5, 1.0, 2.0 and 4.0, (ii) effect of jet exit Reynolds number, varied as 5282, 6340, 7395 and 8450. Figure 4 shows the planes and lines considered for results and discussion. Plane X-X passes through the perimeter and center jet, where as plane Y-Y which is perpendicular to plane X-X passes through centre jet.

<table>
<thead>
<tr>
<th>S. No</th>
<th>Mesh size in millions</th>
<th>Static pressure (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.805</td>
<td>151</td>
</tr>
<tr>
<td>2</td>
<td>1.002</td>
<td>157</td>
</tr>
<tr>
<td>3</td>
<td>1.691</td>
<td>162</td>
</tr>
<tr>
<td>4</td>
<td>1.99</td>
<td>162</td>
</tr>
</tbody>
</table>

![Fig. 4 – Semi-confining plate with plane and lines](image-url)
Results and Discussion

Comparison of experimental data with computations

Flow visualization

Figure 5 shows the flow structure on the impingement surface for $H/D = 1.0$. Figures 5(a) and (b) show the result from flow visualization experiment and the computation respectively. The qualitative comparison between the results is excellent. Corresponding to the impingement point, dark zone is observed due to low shear stress. The oil streak lines are moving outward from the impingement point. Between each pair of jet, another zone is observed which separates the oil streak lines from the jets on either side of it. In the centre part of this zone similar to jet impingement point dark zone is observed. Figure 5(b) shows computational results on the surface of the plate. Corresponding to the impingement point (dark zone in 5(a)), nodal point of attachment is observed. The flow between the centre jet and perimeter jet is described in zoomed view (Fig. 5(c)). The point where jet impinges is the nodal point of attachment, NA (where all skin friction line moves away from the nodal point). As the jet is rectangular it observe to be a line type, but originally all skin-friction line starts from a single point. The skin friction lines from nodal point of attachment moves toward each other from the centre jet and perimeter jet. As the two adjacent jets interact, with each other (centre jet and perimeter jet) two saddle points formed SP1 and SP2 shown in Fig. 5(c) (zoomed view). Two saddle points cannot be joined together hence another attachment line formed in between them (AL). The flow diverges both from the AL and NA. The diverged flow from NA and AL prevent each other to pass through the saddle point and finally forms another line termed as line of separation denoted by SL in Fig. 5(c). In y-direction all the three jets are extended up to the exit of the plate. This feature is depicted from both experiment and computation.

Pressure distribution

Figure 6 shows the comparison of computational result with the experimental data along line A-A. The pressure values are non-dimensionalized with their respective maximum pressure. The comparisons
are good with maximum deviation of about 10% mostly in the sub-atmospheric regions. From the computational results in Fig. 5 it is observed that dimensionless pressure of ‘1’ is corresponding to the impingement/stagnation point (NA). In between jet-1 and jet-2, another peak is observed which is termed as the secondary stagnation point. This peak is due to interaction of the wall jets in that region. This interaction region constitutes SL1, AL1, and SL2. So the peak center is corresponding to the attachment line. In general as discussed earlier, the sub-atmospheric zone is observed immediate downstream of the stagnation zone. The interaction pattern remains more or less same between jet-1 and jet-3.

**Effect of jet-to-plate spacing (H/D)**

**Velocity distribution**

Figure 7 shows the velocity contours for the four jet-to-plate spacing $H/D = 0.5$, 1.0, 2.0, and 4.0 for Reynolds number 6340 along plane X-X. It can be clearly seen that in case of $H/D = 0.5$ & 1.0 the potential core is extended up to target surface shown in Fig. 7 (a,b). Whereas in case of $H/D = 2.0$ & 4.0, the potential core region vanishes by the time flow reaches the target surface, shown in Fig. 7(c,d). These differences in velocity pattern happen because of the variation in momentum exchange between the fluid jet and ambient.

The wall jets from adjacent jets when interacts with each other a up-wash flow takes place as shown in Fig. 6 when $H/D$ value lower ($H/D =0.5 \text{ &} 1.0$) the interaction is prominent, besides as the distance is less the up-wash flow fountain reaches the top wall. This low velocity jet forms another wall jets on the semi confining plate, shown in Fig. 7 (a,b). But no such thing happens for higher $H/D$, because the second jet diffuses out by the time it reaches the top surface, shown in Fig. 6(c,d).

Figure 8 shows the velocity contours in plane Y-Y, on the other hand the velocity distribution is smooth in the plane Y-Y. Velocity values downstream of the stagnation zone and the wall jet region are very high for Y-Y plane compare to plane X-X. Hence the wall jets are extended up to the exit of the plate.
Flow visualization

Figure 9 shows the computed skin friction lines, the jets are arranged in such a way they are being opposed by adjacent jet in one side and other side is open to atmosphere. It is clearly observed the flow pattern for the perimeter jet is kind of similar to the single jet. In case of low $H/D$ as the jet concentrates at the stagnation point and less effective away, the entrainment is more and leads to formation of the separation line ($H/D$ 0.5&1.0) shown in Fig. 9 (a,b). Whereas due to higher spread no such separation line obtained for $H/D = 2.0$ & 4.0 here flow moves smoothly away in radial direction.

Path lines

Figure 10(a) shows the path lines along the plane X-X in case of $H/D = 0.5$. It is observed that jet-1 and jet-2 after impinging on the wall form wall jets, which after some distance separate from the wall. As discussed in earlier cases that if saddle point (SP) lies on a separation line then flow separates. After separation from the surface due to the presence of the confinement plate, these stream lines again merge into main jet to form a vortex in between the plates. The vortex formed from the wall jet-1 rotates in the opposite direction as shown in Fig. 10 (a). These two vortex pairs are termed as counter rotating vortex pair (CRVP). The separation from the surface is otherwise called as upwash flow/fountain. In between the CRVP and the impingement surface, another vortex pair is noticed termed as the wall eddy. The wall eddy separates from the AL and merges towards the saddle points on either side of it. Hence in the direction of rotation of wall eddies are in opposite direction to the CRVP.
In the right side of jet-2, the wall jet is separated from the wall corresponding to the separation line and a vortex is formed. This vortex is extended up to jet-2 which flows back again with the impingement jet as shown in Fig. 10(a). In case of $H/D = 1.0$, as shown in Fig. 10(b) the distribution follows the same pattern as discussed in case of $H/D = 0.5$. However due to shifting of the separation lines towards jet-1, the vortex size formed from wall jet of jet-1 is less compared to that of the vortex formed from jet-2 in the CRVP. In the right side of jet-2, vortex form due to the separation of wall jet, denoted as vortex-1. Another vortex is formed between vortex-1 and main jet due to the presence of free space and high momentum fluid in either side of it. From vortex-1 to the exit of the zone, it is filled with entrained fluid from ambient.

Figure 10(c) shows flow path lines in plane X-X for $H/D = 2.0$. The flow features remain almost same as $H/D = 1.0$. The wall jet diminishes after certain distance. In case of $H/D = 4.0$, Fig. 10(d) no vortex is formed from wall jet-2. Consequently, no CRVP is noticed between jet-1 and jet-2. Unlike in other cases, the vortex does not reach up to the confining plate. In the right hand side, a vortex is formed close to the exit of the plate, termed as vortex-1. Rest of the space between surface and confining plate the zone is filled with entrained fluid.

**Pressure distribution**

Figure 11 shows the dimensionless pressure distribution along line A-A on the impingement surface. It is observed that the pressure zone in the stagnation zone is broader for low $H/D$ cases and reduces with increase in $H/D$. In the present range, $H/D = 4.0$ shows most narrow stagnation pressure zone. Secondary stagnation peak is observed for all the $H/D$ cases. However their magnitudes and position vary with $H/D$. The position of secondary stagnation peak is almost at the mid of jet-1 and jet-2, whereas with increase in $H/D$ ratio the position of the peak shifts towards jet-1.

![Fig.10 – Path lines along plane X-X, for Re 6340: (a) $H/D = 0.5$, (b) $H/D = 1.0$, (c) $H/D = 2.0$ and (d) $H/D = 4.0$](image-url)
For $H/D = 0.5$, the sub-atmospheric pressure is lowest followed by $H/D = 1.0$, 2.0, and 4.0. The difference between the peak and valley is higher in case of low $H/D$ cases due to strong wall jet interaction.

Figure 12 shows the dimensionless pressure distribution along line B-B. Unlike in case the line A-A, the stagnation zone is narrower for low $H/D$ cases and increases with increase in $H/D$. Downstream of the stagnation zone, negative/sub-atmospheric pressure is obtained for all the cases. No secondary peaks are observed as the interactions of wall jets are not strong.

Effect of Reynolds number

Flow structure

Figures 13 (a), (b), (c), and (d) show the surface oil streak lines for $H/D = 2.0$, at varying Reynolds numbers of 5282, 6340, 7395 and 8450, respectively. It is observed that the flow behavior remains almost same for all the Reynolds number cases. Overall, the flow structure is independent of the Reynolds range in the present tested range.

Pressure distribution

Figure 14 shows the dimensionless pressure distribution on the impingement surface along line A-A for varying Reynolds number. With increase in
Reynolds number, the pressure value all along line A-A increases. The pressure values are non-dimensionalized with the maximum pressure among all the Reynolds number cases. The position of the secondary peak remains same. The stagnation zone where the pressure value is higher becomes broader with increase in Reynolds number.

Figure 15 shows the dimensionless pressure distribution along line B-B with different Reynolds number values. At the stagnation point the pressure values increase with increase in the Reynolds number. As in case of line A-A, the stagnation pressure zone becomes broader with the rise in Reynolds number value. Low Reynolds number cases show comparatively high sub-atmospheric pressure in the wall jet region, however the difference is marginal.

Conclusions

Three-dimensional flow on a flat plate is investigated with three-jets impinging on its surface. The following conclusions are drawn from the study:

(i) Among the different turbulent models tested, the κ-ω SST turbulence model is found to be a better predictive model for the determination of flow characteristics of impinging jet systems as validation results are good.

(ii) It is observed that the impingement of upwash fountain on the semi-confining upper plate for low H/D values, with increasing H/D separation point comes closer to the centre jet.

(iii) Larger pressure peaks are observed for lower H/D and Position of the secondary pressure peaks moves away from centre jet with decrease H/D.

(iv) Pressure peaks are found to be higher for higher Reynolds number. However, position of the secondary peak is independent of Reynolds number in the range of 5282 to 8450.

Nomenclature

\( D \) = hydraulic diameter of slot, m  
\( H \) = separation distance from jet orifice exit to target surface, m  
\( H/D \) = jet-to-plate distance  
\( P^* = P/P_{max} \)  
\( Re \) = Reynolds number, \( \frac{4m}{\pi \mu D^4} \)

Greek Symbols

\( \kappa \) = turbulent kinetic energy, \( \text{m}^2/\text{s}^2 \)  
\( \mu \) = dynamic viscosity, \( \text{kg}/\text{m-s} \)  
\( \varrho \) = turbulent dissipation rate \( \text{m}^2/\text{s}^3 \)  
\( \nu \) = kinematic viscosity, \( \text{m}^2/\text{s} \)  
\( \rho \) = density, \( \text{kg}/\text{m}^3 \)  
\( \omega \) = specific dissipation rate, \( \text{l}/\text{s} \)

References