NUMERICAL INVESTIGATION OF AN FREE JET IMPINGEMENT TO FLAT TARGET PLATE

KAZIM PIHTILI  PERihan OCAL
Faculty of Engineering, Department of Mechanical Engineering Bingol University, Turkey

Abstract
Because of the providing highest heat transfer coefficient in heating and cooling areas, there is amount of studies on jet impingement either single or multi jet. So in this study free slot jet impingement take into consideration. Numerical investigation in question for this study bases on Ansys CFX software. In this numerical investigation, different fluids were used. Study in question was investigated for different aspect ratio of rectangular slot duct (A/B), Reynold number and different downstream ratio of the duct exit (Z/B). From results, heat transfer coefficient, local Nusselt number, velocity distribution and also axis switching phenomenon proved. According to results in question it has been understood that, the greater Reynold number taken, the greater local Nusselt number occur, the greater downstream ratio of the duct handled, the lower Nusselt number observed. Also axis switching phenomenon for different aspect ratio of rectangular slot jet has been demonstrated as a velocity contour diagram. According this velocity contour diagrams demonstrations strong axis switching effect observed.

Keywords: Slot jet, free jet, confined jet, numerical analysis, impingement, cooling

1. Introduction

Because of having high heat transfer coefficient, being economical and easy implement, studying on jet impingement has gained importance for the last twenty years. Also the studies that are handled vary widely according to the topics. This topics can be determined as different boundary conditions (i.e. free jets, confined jets), different nozzle shapes (i.e. slot jets, circular jets, elliptical jets, triangular jets etc.), different fluids (i.e. air, gases, water etc.), different jet or plate orientations (i.e. impinging on flat plate/jet or inclined plate/jet) etc. As understood from the headings of the subject, jet impact has a wide working area. Broad literature presentation can be made in this sense, but because of our work has been investigated in this direction, the literature about slot jet impact will be presented here detailed. There are amount of investigation on convective heat transfer coefficient with jet impingement. But there are few studies on physical jet structure which effect directly convective heat transfer and mixing phenomenon for slot jet. Compared with circular jet impingement slot jet have better mixing. In this sense rectangular nozzles are of particular interests because they offer passively improved mixing due to axis switching and enhanced small-scale mixing near corner regions and farther downstream due to faster breakdown of vortex ring coherence and hence faster transition to turbulence [1]. Yu and Grimaji [2] have been studied on axis-switching and stability of laminar rectangular jets using Lattice Boltzman method. The objective of this study is to establish how the jet behavior changes with aspect ratio (AR). The study has been investigated for different aspect ratio (AR= 1, 1.5, 2), different Reynold numbers which changed as 10,100,150,200. At the end of the study they handled three main conclusion for low Reynold number (Re<=100) they observed three jets are stable and no axis-switching observed, for intermediate Reynold number (Re = 150) they observed axis-switching clearly, and for high Reynold number (Re >= 200) they observed all three jets were unstable. Gorman et al. [3] investigated slot jet impingement heat transfer in the presence of jet-axis switching. Their focus of the work to investigate heat transfer and flow characteristic of rectangular slot jets which experiences axis switching. Related jet has been initial cross sectional aspect ratio of 5:1 and 10:1. At the end of the investigation they observed that higher Reynold number, greater downstream distance of impingement plate, greater blockages served to enhance Nusselt number values.
Kumaraswamy et al. [4] study on computational analysis of supersonic jets from rectangular nozzles. In their study, four aspect ratios have been taken into consideration and nozzles for each aspect ratio have been designed using Catia V5. The same research has been analyzed using Ansys CFX. The total pressure decay along the centerline of jet has been concentrated since mixing is enhanced more with rapid total pressure decay along centerline. In addition, to that the potential core length for each nozzle has been evaluated. According to results, it is observed that potential core length is largest for Aspect Ratio AR=2 and least for AR=8. As potential core length decreases, rate of total pressure decay increases. Then they compared the static pressure for different aspect ratios, they observed AR=2 has the maximum peak value and this peak value decreases with increase in aspect ratio. So the atmospheric pressure is attained by AR=8 at the earliest. The static pressure increases due to the shock strength increase, so AR=2 has the highest shock strength. And also they observed from comparison plots the shock strength is maximum for AR=2 and minimum for AR=8. The peak value of maximum density gradient is attained by AR=8.

A study called ‘Assessment of turbulence models for free and confined impinging jet flows have been investigated by Pattamatta [5]. In this study he focused on the validation of turbulence models for impinging jet flows. Six different turbulence models are considered for comparison, which include models of Realizable k- ε (RKE), Reynolds Stress Model (RSM), Spalart Almaras (SA), Standard k- ε (SKE), Shear Stress Transport (SST), and v2f. It has been observed that no single model could predict all the fluid flow and heat transfer characteristics consistent with the experimental data. However, amongst these models Spalart Almaras, realizable k- ε (RKE) model along with near-wall two-layer approach called “enhanced wall treatment”, and shear stress transport (SST) models show consistently good agreement for heat transfer coefficient. The pressure loss coefficient for various nozzle configurations could be predicted well using RKE model.

Craft et al.[6] modelled flow and heat transfer characteristics of an unconfined (free) circular jet discharging perpendicular to a flat plate and examined the performance of four turbulence models for H/D =2 and 6 and R = 2.3x10^4 and 7x10^4. They used k – ε eddy viscosity model as well as three second moment closure model. Compared the their data with the experimental data their numerical results indicated that low Reynolds k – ε model proposed that by Launder and Sharma and also the basic Reynold stress model predicted a very high level of turbulence near the stagnation point. This also resulted in an overestimation of heat transfer coefficient in this region. In conclusion these authors have reported that the effect of the Reynolds number unpredictable satisfactorily by four models studied.

In this study free slot jet impinging on target wall with constant temperature has been investigated to observe convective heat transfer performance for different Reynold numbers, different aspect ratio (A/B), different downstream ratio (Z/B) and different fluids. From results local Nusselt number, stagnation region Nusselt number, heat transfer coefficient contour, velocity distribution has been shown.

2. Physical model and boundaries
The physical model is represented in Fig 1. As it is seen in Fig 1a., the flow model is consist of a target wall, three opening, two jet walls, jet inlet and two symmetry axis. And also it can be seen from Fig. 1a. Z axis shown streamwise (stream down) direction, Y axis shown lateral direction and also X axis shown spanwise direction. Fig 1b. shown slot jet detail picture, which consists of channel dimension with H, W and jet dimension with A, B. Because of in our study dimensionless sizes taken into consideration, jet aspect ratio (A/B), downstream ratio (B/Z) has been used. The length and high of slot jet has been consideration as 5 mm and 5mm in dimension respectively for aspect ratio A/B = 1:1; this dimension taken as 10mm and 5 mm respectively for aspect ratio A/B = 2:1. Downstream length (Z) taken as 5 mm for Z/B = 1 and 20 mm for Z/B = 4.
Two different fluids have been used for flow domain, which consists of air and water. The properties of air and water are specified for static temperature at 25 °C. At the inlet of the solution domain, the temperature is assumed to be uniform and equal to the ambient temperature (25°C). Inlet velocity specified for different Reynold numbers which were handled in this study for the relevant fluid. Reynold number(R) has been calculated from (1) equation.

\[ R = \frac{U \sigma}{\mu} \]  

(1)

In this equation U, B, \( \rho \), \( \mu \) indicated flow velocity, small edge of channel, density of fluid, dynamic viscosity of fluid in use respectively. The opening boundaries of solution domain were taken as under the condition of atmospheric pressure, for which relative pressure taken as zero Pa. Target plate was considered to be at a constant temperature with 100 °C. Also jet walls taken as adiabatic smooth wall. As known the symmetry boundary condition does not require a special definition. Simulation is performed for thermal energy, because fluid in use was considered as incompressible flow. For water flows this acceptance also suitable for all fluid water states. For air flows this acceptance based on Mach number magnitude. This acceptance only will be true for air flow if Mach number less than 0.3. In this sense relevant Reynold number chosen suitable magnitude as 500 and 1000 respectively. The study is performed by using Ansys CFX 14.5 solver manager.

It is well known that mesh accuracy more important for numerical investigations on flow domains. In this respect quadratic mesh has been used, also relevant mesh construction has been shown in Fig.2. mesh independence provided there by taking three edges of element length sizes; 0.18, 0.2 and 0.22 mm. Hence to solve for short time all solutions made for element length size of 0.2mm. Relevant nods are equal to 221933, elements = 1268040 for aspect ratio A/B = 2/1; nods = 219776, elements =1255631 for aspect ratio A/B = 1/1 Z/B = 1. Accuracy is ensured by requiring that all residuals reduced to \( 10^{-6} \) at the end of the computer run. As seen in fig.3. Approximately 100 % mesh accuracy is obtained for all domains in question.
2.1 Mathematical model and governing equations
Numerical simulation done by Ansys CFX solver is based on RANS form of momentum and mass conservation equations. RANS form of momentum and mass conservation equations for incompressible flows using by CFX fluid domain solver with cartesians coordinate system shown in equation (2) and (3). As you know in region close to wall high speed, high temperature and high turbulence gradient occurs. In order to the viscous effects to be included in the results suitable turbulence model must be used. In this sense k-ε turbulence model has been used. In this model, the first variable (k) determines the energy in the turbulence, second variable (ε) indicated turbulent dissipation. In this respect the turbulent kinetic energy k and the dissipation ε are represented by (4) and (5) equations respectively. \( u_i \), \( E_l \) and \( \mu_k \) represent the velocity component in corresponding direction, the component of rate of deformation and the turbulent viscosity. The viscosity can be characterized by equation (6). The equations also consist of some adjustable constants \( \alpha_k \), \( \sigma_k \), \( C_{d1} \), \( C_{d2} \). The values of these constants have been arrived at by numerous iterations of data fitting for a wide range of turbulent flows; \( C_p=0.09 \), \( \alpha_k=1.0 \), \( \sigma_k=1.30 \), \( C_{d1}=1.44 \) and \( C_{d2}=1.92 \). If a turbulent Prandtl number was defined as equation (7), then the energy equation can be represented by equation (8). In this equation, \( \kappa \) is the molecular thermal conductivity (a fluid property), and \( k_t \) is the turbulent thermal conductivity respectively.

\[
\rho \left( u_i \frac{\partial u_i}{\partial x_i} \right) = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left( \mu + \mu_k \right) \frac{\partial u_i}{\partial x_i} \quad (i = 1, 2, 3) \tag{2}
\]

\[
\frac{\partial u_i}{\partial x_i} = 0 \tag{3}
\]

\[
\frac{\partial \rho k}{\partial t} + \frac{\partial \rho u_i u_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu + \mu_k \right) \frac{\partial u_i}{\partial x_i} + 2 \mu_k E_i E_i - \rho \varepsilon \tag{4}
\]

\[
\frac{\partial \varepsilon}{\partial t} + \frac{\partial \varepsilon u_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu + \mu_k \right) \frac{\partial \varepsilon}{\partial x_i} + C_{d2} \frac{\varepsilon^2}{k} + \frac{\varepsilon}{k} \frac{\partial u_i}{\partial x_i} \left( \frac{\partial u_i}{\partial x_i} - \frac{\partial u_j}{\partial x_j} \right) \tag{5}
\]

\[
\mu_t = \rho C \mu \varepsilon^2 \tag{6}
\]

\[
Pr_t = \frac{C_p \mu_t}{k_t} = 0.9 \tag{7}
\]

\[
\rho \varepsilon u_i \frac{\partial T}{\partial x_i} = \frac{\partial}{\partial x_i} \left( (k + k_t) \frac{\partial T}{\partial x_i} \right) \tag{8}
\]
3. Results and Discussions
The results of this study were presented in a dimensionless form for sake of generality. All fluid properties were assumed to be constant due to small variation of temperature. Data from numerical result are collected under subheadings and presented in its own categories which consist of heat transfer dimensionless number (Nusselt number (Nu), velocity distribution \(v\) (m/s^2) and axis switching phenomenon.

3.1. Velocity distribution
Velocity vector distribution in different parallel planes away from jet exit (\(D = 2, 3, 3.5, 4, 4.5\) mm away from jet exit respectively) has been shown in Fig.4 for aspect ratio 2:1 and downstream ratio 1. As it seen from that figure for both air flow and water flow velocity vector distributions change from first planes to last one. It is understood from first plane (\(D = 2\)mm) at the jet exit, regional speed differences from centerline to away from centerline on lateral and spanwise directions so low for both flows. But from the last plane (\(D = 4.5\)mm) it is understood that the bigger regional speed differences occur. There is another situation understood from Fig.4 that for the same Reynold number (\(R = 500\)) water flow velocity distribution the same as air flow velocity. This situation valid for other aspect ratio \(A/B = 1:1\) and other downstream ratio \(Z/B = 4\). According to Fig.4., jet center zones velocity magnitude has its minimum as environment velocity magnitude. We well known that velocity magnitude which is shown in relevance figure is radial velocity or it can be described as lateral and spanwise velocity too. In this sense it can be point that the reason of this state may be spread out of flow. Which make it minimum at jet cross section zone and near environment while make it maximum on the other zones. Also it can be concluded that maximum radial velocity occur because of intensive flow interaction away from stagnation zone.
Another figure about velocity distribution is Fig. 5. Which illustrate downstream velocity magnitude with \(A/B = 1:1\) aspect ratio, \(Z/B = 1\) downstream ratio, \(R= 500\) for water and \(air\) respectively. Firstly it can be seen that both for water flow and air flow velocity taken on symmetry plane profile has almost same configuration. Then second point understanding from same figure that downstream velocity magnitude decreased before impinging plate region. For this reason decreasing zone has circular shape for both water and air water in scope of 2D configuration, for 3D configuration it will be spherical shape. This situation valid for other aspect ratios, downstream ratios and Reynolds number of course.
Figure 4. Velocity vector distribution in different distance away from jet exit, A/B = 2:1, Z/B = 1, R = 500

a) Water flow  b) Air flow
3.2 Local Heat transfer magnitude

3.2.1 Effect of aspect ratio (A/B)
In Fig. 6. Nusselt number shows for two different aspect ratio 1:1 and 2:1, in constant downstream aspect ratio 1 and constant Reynold number (R = 1000). Fig. 6.a. indicated Nusselt number for air flow, second one indicated Nusselt number for water flow. As understood from the Fig. 6.a Nusselt number for aspect ratio 2:1 higher than aspect ratio 1:1 for air flow the same state can be seen for water flow from Fig. 6.b. According to Fig. 6.b. for water flow 2:1 aspect ratio Nusselt number higher than 1:1 aspect ratio. Nusselt number around stagnation region higher than Nusselt number away from stagnation region, on the other hand away from stagnation zone for both aspect ratio Nusselt number value approximately the same. Another aspect ratio effect can be seen as that for aspect ratio A/B = 2:1 stagnation region Nusselt number effect is larger than aspect ratio A/B = 1:1. This situation is the same as for air and water flow.

3.2.2 Effect of downstream ratio (Z/B)
Effect of downstream ratio illustrated via Fig. 7. in which heat transfer coefficient of water and air is observed for different downstream ratios as Z/B = 1 and Z/B = 4. As we can see from related figure heat transfer coefficient has different distribution for both downstream ratios. Firstly it must be specified that according Fig 7.a. in general for
water and for air heat transfer coefficient has its maximum value at stagnation zone for downstream ratio \(Z/B = 1\), on the other hand from Fig.7, b. it is can be understood that for related downstream ratio heat transfer coefficient decreases around stagnation zone compared away from stagnation zone and it take its shape with respect to axis switching at that location. That is to say for the bigger downstream ratio the lower local heat transfer coefficient occur, additionally minimum heat transfer coefficient zone change from rectangular to circular shape for related downstream range from \(Z/B = 1\) to \(Z/B = 4\) respectively. These configurations valid for both flow. In generally change in heat transfer coefficient in respect to different downstream ratio is the same as for water and air flow. But heat transfer coefficient of water flow higher than airs flow for both downstream ratios.

![Heat transfer coefficient contour](image)

**Fig.7.** Heat transfer coefficient contour, \(A/B = 2:1\) , \(R = 1000\) a) \(Z/B = 1\), b) \(Z/B = 4\)

### 3.2.3 Effect of Reynold number (R)

To understand heat transfer magnitude, dimensionless Nusselt number parameter has prescription. As it is known to talk about Nusselt number there must be a wall interacting with the fluid. According to this study wall in question is of course target plate which has been cooled by jet flow; fluid in question is water or air of course. Change of local Nusselt number with Reynold number has been investigated for incase of aspect ratio \(2:1\), downstream ratio 1, which shown in Fig.8., from Fig.8. a. it can be seen local Nusselt number for air flow and Fig.8. b for water flow respectively. We understand from relevant figures that for air flow stagnation zone local Nusselt number higher than away from the stagnation zone for both air and water flow. This situation is valid for both Reynold number in each flow in question. When looking at Fig. 8. it is understood that the higher Reynold number hanles the higher local Nusselt number occur for both flows. Another important result also that local Nusselt number of water flow higher than air flow with a big difference. Fig.8 and Fig.9. shows that the higher Reynold number in use the higher Nusselt number occur. There is another significant point to understand from Fig.9. Nusselt number distribution for low Reynold numbers on the impingement plate around stagnation zone is inclined to maximum value; while it is inclined to increase away from stagnation zone for either air flow or water flow. On the other hand for higher Reynold number this state is reversing. As we seen from Fig.9 air flow and water flow have their minimum Nusselt numbers around stagnation zone while they have maximum value away from stagnation zone for Reynold number equal to 10000.
3.3 Axis switching Phenomenon

Axis switching phenomenon that is known as change in cross sectional shape with increasing downstream distance from the jet origin. This state is encountered for free jets of non-circular cross section. In particular, it is found that a rectangular jet whose initial configuration is defined by a horizontal long dimension and a vertical short dimension undergoes changes so that its downstream demonstration is characterized by a horizontal short dimension and a vertical long dimension [3]. As we know axis switching more significant event for mixing, entrainment etc. situations. In this sense according to results axis switching phenomenon has been shown in Fig.10 which illustrated for aspect ratio A/B = 1 and downstream ratio Z/B = 4; while Fig.11. for aspect ratio A/B = 2:1. And downstream ratio Z/B = 4 via colored by velocity contour. As seen from Fig.10. change in cross sectional shape of free jet from rectangular form at jet exit to circular form at 17 mm away from jet inlet. At the end of the jet, just before impinging target plate (away from jet exit 19 mm), 45° axis switching case clearly observed. As it is known 45° axis switching occur only in square jets like seen in Fig.10 for both water and air. In their study Yu and Grimaji [2] emphasized that 45° axis switching case observed for square slot jet (which mean also aspect ratio = 1) and additionaly 90° axis switching case observed for rectangular slot jets (for aspect ratio = 1.5, 2 etc.). There is another point observed from Fig.10. that velocity spreaded large distances just before impinging on to target plate (Fig.10., D = 19mm and D = 19.75mm). Because of flow interactions, velocity contour has been spread out for both flows and both downstream ratios (Fig.10 and Fig.11). For both flows on the 19 mm away from jet exit velocity
contour diagram shown decay of velocity magnitude about stagnation zone, while on the other planes (5 and 15 mm away from jet exit) velocity contour has its maximum value at about stagnation zone.

Figure 10. Axis switching phenomenon, A/B = 1:1, Z/B = 4, R = 1000, a) Air flow, b) Water flow

Figure 11. Axis switching phenomenon, A/B = 2:1, Z/B = 4, R = 1000, a) Air flow, b) Water flow

In generality change of cross shape of jet flow the same for both water flow and air flow. This state can be seen from Fig. 11. too. Spread out of air velocity larger than water with small difference for aspect ratio A/B = 2:1. The most significant issue must be specified that for square jet, for which A/B = 1:1, Z/B = 1 and R = 1000, illustrated in Fig.10. 45° axis switching has been occurred completely for both flows. But for rectangular slot jet axis switching hasn’t been complete yet, the state may because of taking insufficient downstream length. For this reason rectangular slot jets axis switching issue occur for bigger streamwise ratio than Z/B = 4 as understand from Fig.11. May be downstream ratio handled as Z/B = 5 or 6 will be sufficient to observed axis switching for rectangular slot jet with aspect ratio A/B= 2:1. In this respect a lot of study on this state can be taken under microscope for enlightening researchers.
To understand aspect ratio effect on axis switching it must be noticed that for aspect ratio $A/B = 2:1$ spread out of flow begins earlier than aspect ratio $A/B = 1:1$, but completion of axis switching phenomenon earlier for $A/B = 1:1$ than aspect ratio $A/B = 2:1$.

4. Conclusions

There are two main areas where jet flow is used heating/cooling and mixing. Slot jets have better positions for mixing state compared to round jets. In this sense present study has been taken consideration for free slot jet impingement on to the target plate. Jet impingement in question have two different aspect ratio $A/B = 1:1$ and 2:1, two different downstream ratio $Z/B = 1, 4$ and also two different fluid flow air and water. The study has been solved numerically with fine mesh accuracy, and also for $k-\varepsilon$ turbulence model. Because of presence of target plate with constant temperature, for domain thermal energy was chosen of course. Under this condition taken data from solver manager are velocity distribution, heat transfer efficiency (Nusselt number) and finally axis switching case has been demonstrated. Whilst according us present study will be important reference for both areas in terms of heat transfer of impinging plate which be exposed to jet impingement and in terms of mixing structure (axis switch). For all this matters we understanding from results, it is possible to say briefly:

1. For different planes which were taken as perpendicular to streamwise direction velocity component known as radial velocity generally has its maximum value at jet cross section zone then decreases throughout environmental zone (Fig.4. and Fig.5).
2. About heat transfer it should be emphasized that water flow heat transfer coefficient, also relevant Nusselt number higher than air flow with huge differences. Nevertheless radial and streamwise velocity profiles similar to each other (Fig.4. and Fig.5.).
3. Streamwise velocity component magnitude which is shown in Fig.5. has its maximum value in centerline throughout the streamwise (Z). It has its minimum value at just before impinging target plate because of flow interaction and on scattered region. And also has its zero value on target plate due to stagnation issue.
4. To avaluate aspect ratio effect according to present study results, it can be said that Aspect ratio 1:2 Nusselt number higher than aspect ratio 1:1 nusselt number(Fig.6.). It is necessary a lot verification to said that the higher aspect ratio the higher Nusselt number will be taken.
5. Change in heat transfer coefficient in respect to different downstream ratio is the same as for water and air flow. In low downstream ratios heat transfer coefficient on target plate around stagnation zone has maximum value, while higher downstream ratios it decreases and take its minimum value at region in question(Fig.7.); at the same time heat transfer coefficient of water flow higher than air flow for both downstream ratios.
6. Nusselt number distribution for low Reynold numbers on the impingement plate around stagnation zone is inclined to maximum value; while it is inclined to increase away from stagnation zone for either air flow or water flow. On the other hand for higher Reynold number this state is reversing. Air flow and water flow have their minimum Nusselt numbers around stagnation zone while they have maximum value away from stagnation zone for Reynold number equal to 10.000 (Fig.8. and Fig.9.).
7. The higher Reynolds number used the higher velocity will be occur then of course the higher Nusselt number occurs (Fig.8 and Fig.9).

8. On target plate Nusselt number around centerline throughout spanwise direction (X) have the same profile for water and air. Althought the same Reynold numbers have been use for flows in questions, Nusselt number of water flow higher than air flow for all Reynolds number (Fig.6., Fig.7., Fig.8., Fig.9.).

9. About axis switching it can be specified that for square slot jet cross section shape of jet flow colored by velocity is changed from square to circular shape until 17mm away from jet exit, then just before impinging on the target plate radial velocity component have its full axis switching 45° which valid only for square. For rectangular slot jets axis switching issue occur for bigger streamwise ratio than Z/B = 4 as understand from present results (Fig.11.). In generally configuration of axis swithing state of water the same as air flow’s configurations.


REFERENCES