Effect of Inlet Slot Dimension on Laminar Plane Wall Jet Flow over an Obstacle

M. Arul Prakash, K. Mayilsamy, G. Murali.

Department of Mechanical Engineering, Sri Sairam Engineering College, Chennai, Research Scholar, Anna university, Chennai, Tamil Nadu, India.

Received 28 January 2017; Accepted 22 April 2017; Available online 1 May 2017

Copyright © 2017 by authors and American-Eurasian Network for Scientific Information (AENSI Publication). This work is licensed under the Creative Commons Attribution International License (CC BY).

ABSTRACT

In this paper, the effect of the inlet slot dimension of a plane laminar wall jet on the formation and growth of recirculation and on the velocity profiles are investigated numerically when the jet is allowed to flow over an obstacle. Wall jet from the nozzle exit blows tangentially through the inlet slot along the bottom wall where the solid obstacle is present. The top and right side of the domain is kept open whereas the left side of the domain is bounded by a solid wall. Two-dimensional incompressible laminar flow is assumed. Continuity equation and momentum equation for the full computational domain are solved numerically by using an in-house CFD code based on stream function vorticity formulation. Simulations are carried out for different inlet slot heights and a range of Reynolds numbers. From the streamline pattern, it is observed that the increase in size and shape of the recirculation is more when the inlet dimension is increased. Investigations are extended to find the influence of slot dimension on stream-wise velocity profile in the upstream and the downstream locations. It is established that the maximum value of stream wise velocity component increases as the inlet slot height increases. These effects are more reflected on higher values of Reynolds numbers.

KEYWORDS: Effect of inlet slot dimension, Wall jet over obstacle, Laminar wall jet, Effect on flow characteristics.

INTRODUCTION

Heat exchange between wall jet and solid objects find wide engineering applications like fluid injection systems, cooling of turbine blades, wind shield defroster, electronic cooling etc. Objects present in the path of wall jet disturb the flow and the affected flow pattern plays a major role in the heat transfer. Literature published in the past has made some good contribution in analyzing the flow and heat transfer characteristics of the wall jet. Similarity solutions of the boundary layer equations for wall jet were obtained by Glauer[1]. Seidel and Fasel [2] investigated the effect of high-amplitude forcing on a laminar wall jet over a heated flat plat. It was found out that the increase in the amplitude results a reduction in skin friction, an increase in local maximum velocity and increased wall heat transfer. Law and Herlina [3] conducted experiments to investigate the velocity and concentration characteristics of three-dimensional turbulent wall jets. The existence of similarity of the velocity profiles in both the stream wise and span wise directions was reported. Using curve fitting, the mean velocity properties of the three-dimensional wall jet were approximated.

The so-called Blasius wall jet is matched to the experimental data by Levin et al. [4], and is found to be valid in the region just downstream of the nozzle. The measured base flow is matched to a boundary-layer solution developing from a coupled Blasius boundary layer and Blasius shear layer. Cohen et al. [5] found a new family of solutions describing the mean flow of an incompressible two-dimensional wall jet subjected to boundary suction or blowing. Velocity and temperature fields of a plane acoustically forced laminar wall jet...
were investigated experimentally by Quintana et al. [6] for the constant wall temperature boundary condition. It was reported that the high frequency forcing reduced the maximum skin friction and increased the maximum wall heat flux.

Experimental investigations on the behaviour of a laminar plane wall jet within the range of validity of linear stability theory were carried out by Bajura and Szewczyk [7]. It was also showed that the stability of the plane wall jet profile lies between the two extremes of the plane free jet and the Blasius velocity profiles. Kanna and Das [8] simulated the flow features of the two-dimensional laminar wall jet under a backward facing step. The effect of the presence of backward facing step on recirculation and reattachment length were studied for various Reynolds numbers and step geometry. It was found out that the distance of formation of similarity profile is reduced by increasing the size of step geometry. Velocity field measurements and comparison with the similarity solutions for a two dimensional laminar wall jet were done by Peters et al. [9].

Kumar [10] carried out a detailed analysis on the flow features of a dual jet. The comparison was made between the flow characteristics of the off jet and the dual jet for different offset ratios. The location of reattachment point, vortex centres, merge point and the combined point were identified and their relationships with offset ratio were established. It was found that the movement of vortex centre normal to stream wise direction is linear whereas its stream wise movement follows a parabolic trend.

Göransson and Trägårdh [11] studied the mechanism for sub-micron particle deposition in a laminar wall jet. The effect of particle size on deposition rate of polystyrene latex particles was investigated experimentally. It was reported that the deposition efficiency is strongly dependent on wall shear stress and particle size. A steady laminar viscous inviscid interaction that arises when a wall jet is subjected to a rapid deflection caused by the presence of a ramp was studied by Braun and Kluwick [12]. It is established that the relationship between the displacement of the flow and the resulting pressure disturbance is depend on the local curvature of the streamlines only rather than on global flow properties.

Aerts et al. [13] experimentally studied anodizing of high purity aluminium in a wall jet electrode reactor under different flow conditions. Quantitative information on the conditions of heat transfer is provided by numerical calculations of the flow field, and is applied during simulations of the anodizing process. The influence of heat transfer on the anodizing of aluminium has been demonstrated. The entrance effect on laminar flow over a backward facing step was investigated by Barton [14]. It was shown that significant differences occur on reattachment and separation lengths when using an inlet channel in backward facing step geometry.

Mudgal and Pani [15] estimated the drag force experienced by a single still, a pair of identical stills and a cube in a plane turbulent wall jet. Main focus was given on the length of the wake and the characteristics of the relaxing shear layer. In the longitudinal direction, self-similarity of the velocity profiles of the wall jet was established. Recently Kabache and Mataoui [16] examined the effects of turbulent wall jet flow and the boundary layer flow on flow and heat transfer characteristics of cooling of a hot rectangular obstacle. The influence of nozzle thickness and jet exit Reynolds number on the flow structure and the average Nusselt number were also investigated. In this turbulent flow study, only three nozzle thicknesses were considered and there was no special attention given to the velocity profiles.

Though the literature made numerous contributions in this field, the influence of inlet jet dimension on the laminar wall jet over an obstacle has not yet been reported. The attempt made in the present paper aimed to study the effects of different jet inlet dimensions on flow pattern and on velocity profiles of a laminar wall jet flow over a solid obstacle.

Perolem Definition:

A two dimensional incompressible laminar plane wall jet is considered and is assumed to be blown tangentially along the bottom wall. A solid obstacle of square cross section of size \( bh \times bh \) is placed at the bottom wall. The top and the right side of the domain are kept open whereas the left side of the domain is bounded by a solid wall. The schematic of the computational domain is shown in Fig.1. A Newtonian, viscous, incompressible fluid is considered. Buoyancy induced effects are neglected. The thermo-physical properties of the fluid are assumed to be constant.

A. Formulation of mathematical model:

The transient non-dimensional forms of continuity and momentum equations which govern the flow were reduced by stream function and vorticity formulation. The single reduced non-dimensional vorticity-transport equation is written as:

\[
\frac{\partial \omega}{\partial t} + u \frac{\partial \omega}{\partial x} + v \frac{\partial \omega}{\partial y} = 1 \left( \frac{\partial^2 \omega}{\partial x^2} + \frac{\partial^2 \omega}{\partial y^2} \right) \quad \text{Re} \left( \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} \right)
\] (1)
The Poisson equation for stream function is written as:
\[ \frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = -\omega \] (2)

The vorticity and the stream function are defined as:
\[ \omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \] (3)
\[ u = \frac{\partial \psi}{\partial y}, \quad v = -\frac{\partial \psi}{\partial x} \] (4)

The non-dimensional variables are defined as:
\[ x = \frac{x}{bh}, \quad y = \frac{y}{bh}, \quad u = \frac{u}{U}, \quad v = \frac{v}{U}, \quad t = \frac{t}{h/U} \]

Where, \( x, y, u, v, t \) are dimensional variables, \( U \) is the average jet velocity at the nozzle exit, \( bh \) is the height and width of the square obstacle.

The following non-dimensional conditions are used.
Along the jet inlet width: AE,
\[ u = U \]
Along solid walls: AF, FI, IH, HG, GB, and ED, no slip condition is assumed.
\[ u = v = 0 \]

At the top of the domain, DC:
\[ \frac{\partial u}{\partial y} = 0, \quad \frac{\partial v}{\partial y} = 0 \]

At the downstream boundary of the domain: BC, the flow reaches the developed condition. Therefore stream wise gradients were assumed to be zero.
\[ \frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = 0 \]

Along the walls, Thoms vorticity condition is applied as below:
\[ \omega_n = \frac{2(\psi_{w+1} - \psi_w)}{C_n^2} \]

Where, \( \Delta n \) is the grid spacing normal to the wall.
B. Solution approach:
Second order central difference scheme is used for the discretization of spacial terms of the governing equations and forward difference scheme is used for the temporal terms. The resulting algebraic system of equations is solved by Alternating Directions Implicit method (ADI method).

![Graph](image)

**Fig. 2:** Validation of Present Code with Barton [14]

Successive Over Relaxation (SOR) method is used for the Poisson Equation. To attain the steady state solution, the convergence criteria which is defined as the sum of the total error between two consecutive time steps is chosen as $10^{-6}$. To avoid exit effect at right boundary and to reach an optimum domain size, a systematic domain size independence test is carried out. A time step independence test is also carried out to ensure the steady state results. The size of computational domain is fixed as $AB = 3$ and $AD = 1$. At the bottom wall an obstacle is placed at a distance of $AF = 1$ from the left boundary of the domain. Square cross section is assumed for the obstacle block with the block height, as well as block width, $bh = 0.4$. The initial conditions for all the variables are considered as zero at $t = 0$. No slip ($u = 0$, $v = 0$) boundary conditions are applied at all walls including that of the obstacle block. A uniform velocity profile $U = 1$ is assumed at the jet inlet. The time step 0.001 is used for all the computations.

**Validation:**

The in-house code developed for the present simulations is validated with well established results from the literature. For this purpose, the present code is modified to solve the problems given in the literature. Variation in reattachment length of the backward facing step problem for different Reynolds numbers is compared with Barton[14]. The comparison is shown in Fig.2. Good agreement is attained in this comparison. The code is also validated for stream wise velocity profile. The results obtained from the present code is compared with the $u$-velocity profile of the Backward Facing Step problem of Gartling[17]. The comparison made for $Re = 800$, at a stream wise location $x = 7$, show a good agreement as shown in Fig.3.

**Grid Independence Study:**

A detailed grid independence study is carried out to attain the optimum grid size for the present problem. For this test, a slot height of $0.125bh$ and the Reynolds number 300 are chosen. Five uniform grids $91 \times 271$, $101 \times 301$, $111 \times 331$, $121 \times 361$, and $131 \times 391$ are tested. Stream wise velocity component ($x$-component) for all these grid sizes are plotted against normal coordinates ($y$-coordinates) of the computational domain. From this plot as shown in Fig.4, it can be seen that the deviation of velocity profile for the grid size $121\times361$ from that for grid size $131\times391$ is almost negligible. Therefore the grid $121\times361$ is chosen and used throughout the present simulations.
Simulations are carried out to investigate the influence of the inlet slot height of wall jet on various flow characteristics. The slot heights considered are: $0.05 \, bh$, $0.075 \, bh$, $0.1 \, bh$, $0.125 \, bh$, $0.15 \, bh$, and $0.2 \, bh$. The flow characteristics for different slot heights are obtained and presented in different plots. The effects on the flow pattern, formation of recirculation, entrainment, and the velocity profiles are studied for three Reynolds numbers 100, 300, and 500.

A. Effects of Inlet Slot Dimension on Flow Pattern:

The streamline contours for various slot heights are plotted from the results of the numerical simulations. Flow patterns for the Reynolds number of 100 for all the slot height values are shown in Fig. 5. For all the slot heights considered, it is observed that two recirculations are formed. One is on the left side of the obstacle and another is on the right side of the obstacle. For the slot height $0.2 \, bh$, recirculation due to the entrainment of ambient fluid is also observed on the top left of the domain. The recirculation on left and right side of the obstacle are almost same in size for the slot height of $0.05 \, bh$. A small growth of left side vortex is seen as slot height increases. The growth of right side vortex is observed to be more as compared with that of the left side vortex. For the slot height of $0.2 \, bh$, it covers almost 80% of the height of the obstacle. For the Reynolds number of 300, the streamline pattern is shown in Fig. 6. There is a considerable difference in size of vortex on left and right side of the obstacle even for the lowest slot height, $h = 0.05 \, bh$. As slot height increases, more growth of left side vortex is observed compared with that for $Re=100$. The growth of right side vortex for $Re=300$ is more when comparing with that for the left side vortex. Huge increase in size of right side vortex is noticed as slot height increases. This recirculation starts covering the top face of the obstacle for $h = 0.15 \, bh$, $0.175 \, bh$ & $0.2 \, bh$. There was no entrainment vortex for $h = 0.05 \, bh$. From slot height of $0.075 \, bh$, the entrainment vortex starts spreading towards right side and the bottom of the domain. It starts to deviate the flow and as the result, the streamlines are concentrated more near the top corners of the obstacle.

B. Effects on Reattachment Length:

It is important to investigate the influence of the inlet slot height on the formation and development of recirculation in the flow field. The growth of the right side vortex is studied by plotting the reattachment length
against the inlet slot height for three Reynolds numbers 100, 300, and 500. It is evident from the plot (Fig.7) that the reattachment length increases as the slot height increases. For lower Reynolds numbers (Re=100), gradual variations are observed on the reattachment length for all values of slot heights considered, whereas for higher Reynolds numbers, the variation is gradual only up to $h = 0.125 \, bh$ and a sudden change beyond this value of slot height. It was also observed that the rate of increase of reattachment length is high for higher Reynolds numbers and low for the lower Reynolds numbers. For the same variations in the value of Re, the reattachment length curves are closer for higher Reynolds numbers (100, 300) compared with the lower Re.
C. Effects on Entrainment:

The entrainment vortex and their growth for different slot heights are analyzed for two Reynolds numbers, Re = 300 and Re = 400. The location of the centre of the entrainment in the stream wise direction was plotted against slot height in Fig. 8. For the Reynolds number of 300, a gradual shift in the location of entrainment was observed along the stream wise direction up to the slot height of \( h = 0.15 bh \). After this slot height, there is no considerable variation in the location of entrainment for Re = 300. The centre of entrainment for the Re = 500 shifts towards the downstream direction until the slot height value \( h = 0.1 bh \), and remains in a same location thereafter. The right most location of the centre of entrainment is almost same for both the Reynolds numbers. This shows that the movement of the entrainment along the stream wise direction is restricted by the presence of the obstacle. It is also observed that for the lower values of slot heights, as Reynolds number increases the entrainment moves more towards the downstream locations.

D. Effects on Velocity Profile:

Variation of velocity in the flow field is studied by analyzing the stream wise velocity component in an upstream location and a downstream location.

The velocity profiles for the Reynolds number of 300 at an upstream location \( (x = 0.5) \) of the obstacle and at a downstream location \( (x = 1.75) \) of the obstacle for different slot heights are plotted and presented. At the upstream location \( (x = 0.5) \) all the curves starts from the bottom wall with zero no slip velocity and increases along the normal direction as shown in Fig.9.

Near to the bottom of the domain, the wall jet velocity profile is formed. This stream wise velocity increases first and decreases after reaching a maximum value. The maximum value of this velocity component increases as the inlet dimension of the jet increases. For lower values of the slot height, variation of velocity is less compared with that for higher values. The maximum velocity reached also follows similar trend. Due to the entrainment, negative velocities were observed near the top of the computational domain. Positive values of velocities for smaller slot heights in this region show that the flow is not affected by the entrainment. The velocity profiles at the downstream location \( (x = 1.75) \) for different slot heights is shown in Fig.10. The negative values of velocities near the bottom of domain are due to the vortex formation on the right side of the
obstacle. For smaller slot heights the effect of right side vortex is not felt at this location. The region near the top of the computational domain is not affected by any recirculation and hence all the velocity profiles fall in the positive region. It is clearly seen from the plots that the velocity increases as the slot height increases.

Conclusions:

Numerical simulations were carried out and the effect of inlet slot dimension on plane laminar wall jet over an obstacle is investigated. Streamline patterns, formation and growth of vortex, entrainment and variations in stream wise velocity profiles are investigated for different jet inlet slot heights and a range of Reynolds numbers. From the studies the following conclusions were attained:

1. As slot height increases, more growth in both the recirculation vortex is observed. The increase in size and shape of the right side vortex is more as compared with that of the left side vortex.
2. The effects of jet inlet dimensions are more felt in higher Reynolds numbers and are reflected in the growth of recirculation.
3. The reattachment length after the obstacle increases as the slot height increase. This is more felt in larger Reynolds numbers.
4. A gradual movement of entrainment vortex along the stream wise direction for \( \text{Re} = 300 \) is observed up to the slot height, \( h_l = 0.15 bh \). As slot height increases, no considerable variation in the location of this vortex is observed. A Similar trend is observed for other Reynolds numbers also.
5. At upstream locations, the maximum value of stream wise wall jet velocity profile increases with the increase in the inlet slot height.
6. At locations downstream to the obstacle block, higher values of stream wise velocity are observed for higher jet inlet dimensions.

REFERENCES