Flow Analysis of Laminar Wall Jet over Curved Cavity with a Channel Mounted Fin

M. Arul Prakash, K. Mayilsamy, P. Maheandera Prabu, A.Ravinthiran, G. Murali

Abstract: ***Wall jet flow is used for industrial cooling process, cooling of electronic component mounted on circuit board etc. Numerical simulations have been carried out for laminar two dimensional wall jet flows along curved cavity having a channel mounted thin fin. The commercial finite volume code FLUENT is chosen to resolve the mass balance and momentum balance equations. Fluid flow characteristics are investigated for different Reynolds number (Re=100 to 600) and for different fin geometry. The results are plotted in the form of velocity profiles and streamline contours. The effect of fin length on the laminar wall jet characteristics is also investigated.***

Index Terms: ***Laminar flow, Wall jet, Curved cavity, Fin, Effect of geometry.***

I. INTRODUCTION

Wall jet flow with different geometrical configuration is predominantly employed in electronic cooling devices, industrial cooling processes, heat exchangers etc. The most widespread application of the wall jet is to modify heat and mass transfer close to walls in the automobile demister. Many researchers in the past published results from both computational fluid dynamics (CFD) tools and experimental studies for wall jet flow and cavity flow problems. The combinational free and forced convection heat transfer behavior in a 2D, laminar, incompressible wall jet along a vertical wall is investigated by[2].

The profile for velocity and temperature is chosen to be power series. The Prandtl number, modified Grashof number and Reynolds are considered. The skin friction and surface temperature details is also reported. Flow and thermal study of a mixed convection, laminar, incompressible wall jet was carried out by [3]. Buoyancy assisted forced convection case was considered for the investigation. It was reported that the average Nusselt number increases with Re, Gr, and Pr . An experimental work on laminar wall jet was carried out by [4]. The similarity theory by[5] is made comparison with the experiments. It helps understanding the similarity character in connection with its source.

Investigations on conjugate heat transfer in a flat plate by a laminar two-dimensional plane wall jet was investigated by [6]. An analytical solution was presented for two cases of Pr variations with k, Re and λ as the parameters. To get the entrainment boundary and exit boundary conditions in laminar incompressible wall jet flow,[7] made an attempt. Numerical simulations of laminar, 2D wall jet flows over solid obstacle were carried out by [8]. An incompressible flow analysis for laminar wall jet with obstacle was considered. Stream line contour, u-velocity profiles and v-velocity profiles were plotted for different Reynolds numbers. The pattern of flow and the formation of vortices were critically analyzed and reported. [9] investigated the effect of obstacle in an incompressible laminar wall jet flow. To solve the problem a newly written CFD code was used. The code is based on vorticity-stream function method and written in c-language. The streamline, growth of each recirculation, and length of re-attachment, and x and y component velocity profiles were compared for the simple flow and that with obstacle. The results from the investigations are reported that the impacts are more for higher Reynolds numbers and around the obstacle.

Lid-driven cavity case problem is the traditional benchmark problem [10]. The laminar behavior of wall jet flow characteristics in a shallow-cavity and constant temperature condition at the wall is solved for the Reynolds number range from 25 to 600 by [11]. The flow characteristics of different shapes of the cavities (square, Rectangular, semi-circular cavities) are examined and studied experimentally[12]. Similarly, the studies of Vortex development in arc-shape cavity for high values of Reynolds numbers condition have been carried out [13]. This two dimensional flow analysis using vorticity-stream function method summarizes effects of aspect ratio on the flow establishment and vortex structures. It was also reported about the considerable effects on the solutions of flow through cavities. The flow and heat transfer for lid driven flow combined with curvature shape cavity have been analyzed by[14]. The parameters taken for this analysis are Reynolds number (Re), Grashof number (Gr) and Inclination (φ).

The flow and mixed convection behavior of arc cavity has been analyzed by[15]numerically and experimentally. The Nusselt number (Nu) reaches minimum value at transition regime.

Manuscript published on 30 September 2019
* Correspondence Author

M. Arul Prakash*, Department of Mechanical Engineering, Sri Sairam Engineering College, Chennai, India.
K. Mayilsamy, Department of Mechanical Engineering, PSG College of Technology, Coimbatore, India.
P. Maheandera Prabu,Fluid Mechanics and Machinery Lab, Indian Institute of Technology, Indore, India.
A. Ravinthiran, Department of Mechanical Engineering, Sri Sairam Engineering College, Chennai, India.
.

This also reduces the heat transfer. In a lid-driven rectangular shape cavity, the vortex structure of steady flow is investigated by [16] using a lattice Boltzmann method. The aspect ratios ranging between 0.1 and 7, and the Reynolds number range of 0.01 to 5000 are chosen. The impact on the size of vortices, location of vortices, center of vortices are determined. The flow pattern is also obtained. As non-dimensional velocity Re increases, the vortex shape is changed. For higher Reynolds numbers, more number of large vortices was observed. A strong effect of Reynolds number on size and locations of center of large vortices has been noticed near the lid for the deep cavity flow. [17]carried out Numerical simulations on two-dimensional incompressible slot jet flows. The reattachment length, center of vortex and coefficient of friction (C_f) are analyzed. The aspect ratio (AR) and the Reynolds number have more effect on flow pattern than width of the block and the height.

A CFD based optimization is done on the curved cavity by[18]using neural network. The numerical study of buoyancy induced oscillations in a lid-driven cavity of arc shape is carried out by [19]. Numerical investigations on 'wall driven flow with viscous effects' over semi circular shaped cavity are carried out by[20]. The effect of the inlet slot dimension of a plane wall jet under laminar conditions on the formation and growth of recirculation and on the velocity profiles are investigated numerically by [21] when the jet is allowed to flow over an obstacle. At upstream locations, the maximum value of stream wise wall jet velocity profile increases with the increase in the inlet slot height.

Analysis on 'Multiple Branch Pipe' and on Flow Header used in Tube Heat Exchangers was done by [22]. The primary, secondary and tertiary vortices are clearly captured when Reynolds number (Re) increases. The flow structure analysis on lid-driven arc cavity is done by[23] using power law fluids. The results depicts that the central vortex moved to upper right corner under shear thinning for the lower Reynolds number (Re=100). Experiments on flow past cavities for different shapes are carried out by [24]using Particle Image Velocimetry technique (PIV). It was found that the rectangular shape cavities and triangular shape cavities have maximum amplitudes. It is also established that the semi-circular shaped cavity has the lowest amplitude.

Based on literature survey, it is observed that the laminar wall jet through curved cavity with channel mounted thin fin has not been solved yet. So the aim of present work is to solve the laminar wall jet flow in a curved cavity with fins of different lengths (b). The impact of geometry (b) and Reynolds number (Re) in the cavity flow under the laminar wall jet conditions are studied numerically in the preset study.

II. PROBLEM DESCRIPTION AND GOVERNING EQUATIONS

Schematic illustration of the physical configuration with computational-domain of the analysis is shown in Figure 1. At the inlet, laminar flow of wall jet is assumed. Inlet slot dimension is *h* and the curved cavity width is *4h*.

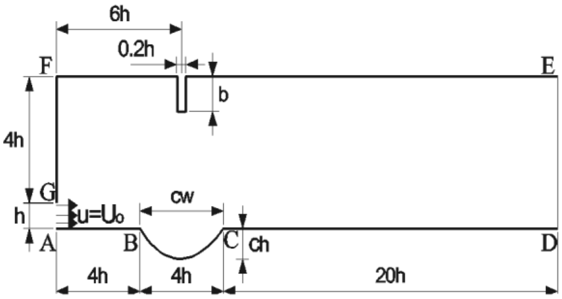


Figure 1. Schematic illustration of the problem

The cavity is at a distance of *4h* from the inlet slot as shown in the schematic. The wall jet enters at the inlet slot with uniform velocity ($U_o = 1$). Air enters at the atmospheric temperature condition. The arc cavity is maintained at 400K (constant temperature). In this present work, the Reynolds numbers is taken from the range 100 to 600 (Re = 100,200, 300, 400, 500 and 600)for the laminar wall jet at the inlet slot . The inlet slot dimension of the wall jet, h = 1 unit is assumed. The jet inlet velocity is considered based on the hydraulic diameter (D_h) of the jet.

A. *Governing Equations*

The two-dimensional, incompressible laminar flow governing equations of continuity and momentum are expressed as shown below:

Continuity equation:

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{\partial p}{\partial x}+\frac{1}{\mathrm{Re}}\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \quad (1)$$

x-Momentum:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \quad (2)$$

y-Momentum:

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{\partial p}{\partial y}+\frac{1}{\mathrm{Re}}\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \quad (3)$$

B. *Boundary Conditions*

Boundary conditions along boundaries of the domain (Figure. 1) are :

Along jet inlet AG,
Uniform jet velocity is considered.

Following no slip conditions are chosen along wall-AB, wall-BC, wall-CD, wall-EF, wall-FG and fin as:

$$u = 0 \quad \text{and} \quad v = 0 \qquad (4)$$

Fully developed flow condition is assumed along DE,

$$\frac{\partial \varphi}{\partial x} = 0, \qquad \varphi(u, v, p, \omega) \quad (5)$$

The wall BC is maintained at constant Temperature 400K.

Along all the wall boundaries,

$$u = 0 \quad \text{and} \quad v = 0 \qquad (6)$$

III. NUMERICAL PROCEDURE AND VALIDATION

The solution is obtained by using finite volume based CFD code FLUENT. The continuity equation, x-momentum & y-momentum equations are solved subjected to the boundary conditions with initial conditions. In the present work, a third-order QUICK scheme is used to solve momentum balance equations. The SIMPLE algorithm is used to couple the pressure-velocity equations. The meshes with cluster grids are used near the wall region in order to reduce the error in solution. The convergence residual10^{-8} is taken to get higher accuracy.

A. *Validation*

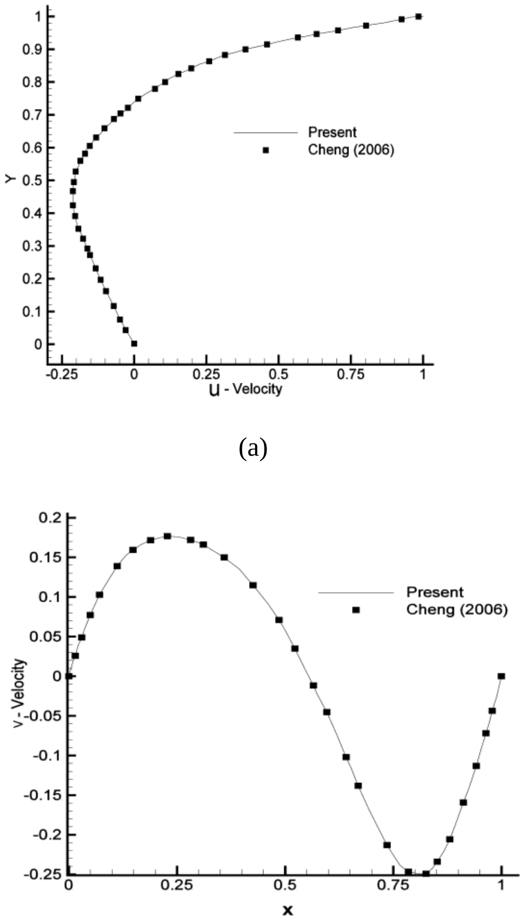
The Numerical procedure was validated with well know published results. Figure 2 compares the velocity profiles of the rectangular cavity with [16]. In Figure 3, the comparison of u/u_{max} profile of the laminar wall jet along a flat plate with experimental results of[25] are shown. These comparisons show that the numerical procedure is right and hence validated.

IV - RESULT &DISCUSSION

Flow behavior of laminar wall jet flow through curved cavity with a channel mounted fin is simulated. The effect of the length of fin and Reynolds number are investigated. The results are presented for velocity profile and streamline contours.

A. *Flow Characteristics of Wall Jet over Curved Cavity with Fin*

Effects of Reynolds number and fin length on various flow characteristics are investigated. Three Reynolds numbers 200, 400, and 600 are considered. The fin lengths chosen for the study are: b=2,3,and 4. The results on streamline contours, and velocity profiles at different downstream locations were presented for the above parameters.



(a)

(b)

Fig 2. Comparison of Velocity Profiles of Steady Flow along Rectangular Cavity with [16] (a) u-velocity (b) v-velocity

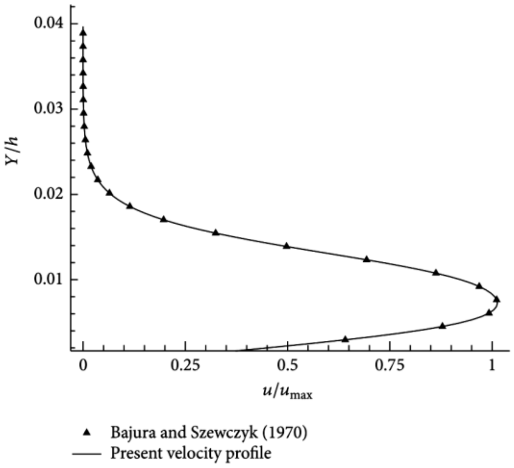
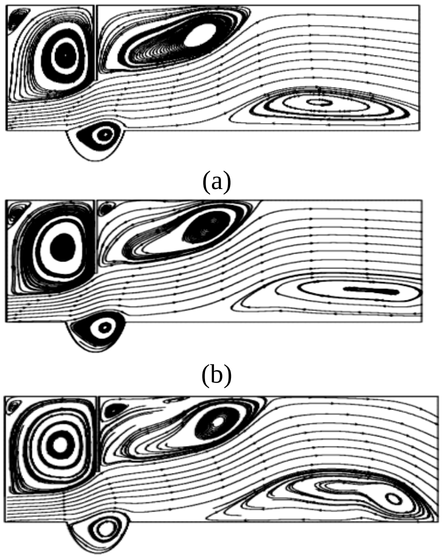


Fig 3.Comparison of *u/u_{max}*of Laminar wall jet flow along flat plate with Experimental Results of [25, 26].

B. *Effect of Reynolds number on streamline contour*

The impact of Reynolds number on streamline contour for b=2 is shown in Figure4. The streamlines are spread towards the normal direction due to nature of wall jet for low Reynolds number cases.



(a)

(b)

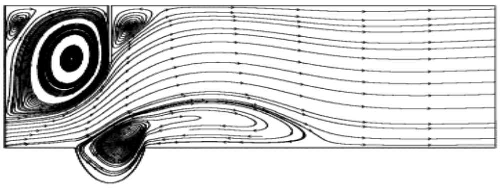
(c)

Fig 4.Effect of Reynolds Number on Streamline Contour for b = 2(a) Re = 200 (b) Re = 400 (c) Re = 600.

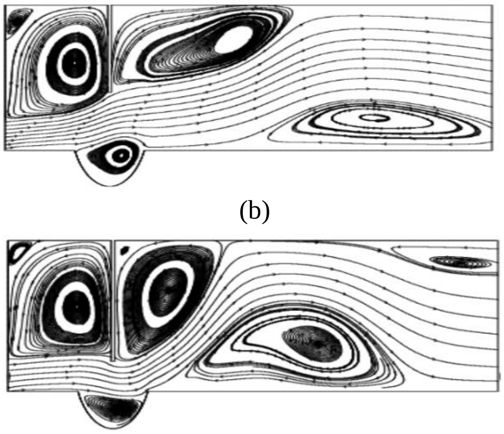
There are four major vortices are formed except corner vortex. For Re=200, the primary vortex isnoticed at the left top of the computational domain and three secondary vortices are observed. One at the cavity region (secondary vortex-1), second at right side of the fin ((secondary vortex-2) and the third is at the end of the downstream wall (secondary vortex-3). The same phenomena are noticed for all other Reynolds numbers too (Figure 4). It is identified that the centers of secondary vortices are moved towards downstream direction for increase in Reynolds number. The size of the secondary vortex-2 is gradually decreased with the formation of a new corner vortex when Reynolds number is increased. It is also observed that the domination and size of secondary vortex-3 is increased for the increase in Reynolds number.

C. *Effect of fin length on streamline contour*

The effect of fin length (b) on streamline contour for Re=100 is shown in Figure 5. For the fin length, b=2, 3 & 4, the streamline pattern and the number of vortices formed is shown. For b=2, it is noticed that there are three vortices appeared on the computational zone (except the corner vortex). However, there are four vortices noticed for the cases of b=3 and 4. For the low fin length case (b=2), the cavity vortex is more dominant and spread on the downstream wall. When b is greater than 3, the secondary vortex-3 on downstream wall is formed and it moves towards the negative flow direction. The size of the secondary vortex-1 is decreased when the value of fin length (b) is increased. It occurs due to the sudden momentum variation and change in flow direction of wall jet. The direction and momentum changes occur due to the change in fin length.



(a)



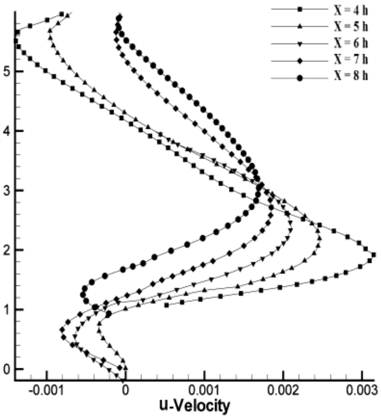
(b)

(c)

Fig 5.Effect of Fin Length on Streamline Contour for Re = 100 (a) Fin length = 2 (b) Fin length = 3 (c) Fin length =4.

D. *Effect of Reynolds number on velocity profile*

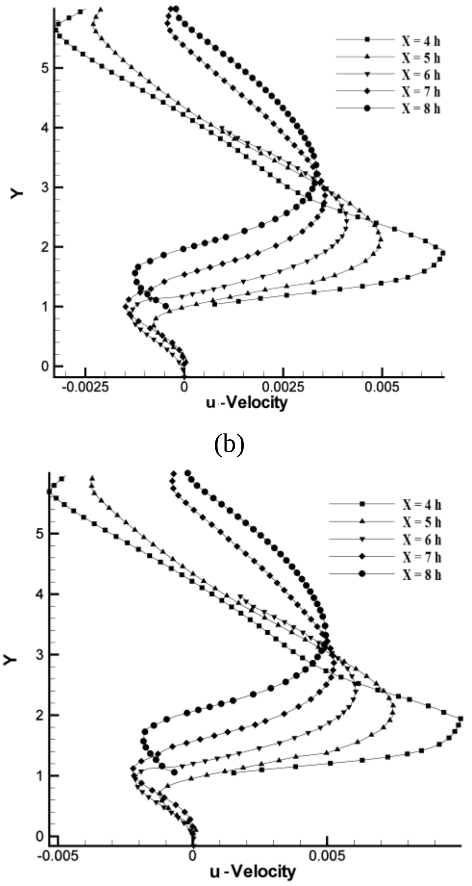
The velocity profile at various stream-wise locations for different Reynolds numbers (Re= 200, 400 & 600) are shown in Figure 6. It shows the horizontal velocity profile (u-velocity profile) for the Reynolds number, Re=200. From figure, the existence of negative velocity is noticed at the top left region. This is the indication of the primary vortex that is formed around top of left corner. The local peak velocity value of 0.0015 is observed at the location of *x/h=4*.



(a)

After this peak, When move along the stream wise direction, it is noticed that the local velocity (u-velocity) is reduced and the velocity profile moves along the normal direction.

This reduction is due to the expansion of wall jet along the direction of flow. Negative velocities are identified also at the bottom of the profile at the downstream locations of *x/h=6*, *x/h=7*, and *x/h=8*. This is due to the Formation of the vortex in this region.



(b)

(c)

Fig. 6.Effect of Reynolds Number on Velocity Profiles for b = 2 (a) Re = 200 (b) Re = 400 (c) Re = 600.

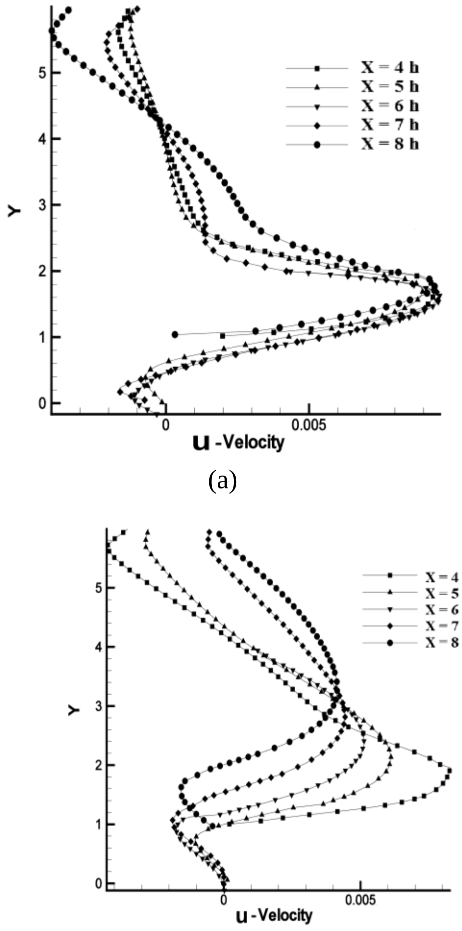
The observations based on Figure 6, show that the horizontal velocity is increased while the Reynolds number is increased. This behavior is due to increased momentum of the wall jet. Negative horizontal velocity on cavity region (bottom) is increased up to *x/h* = 7 when Reynolds number increased. It means that the cavity vortex is dominated up to the location *x/h* = 7. The peak horizontal velocity is gradually increased and moving towards the normal direction when the Reynolds number (Re) is increased. This is due to the increased momentum and wall jet behavior. In general, the wall jet moves towards normal direction when the flow distance is increased.

E. *Effect of fin length on velocity profile*

The effect of fin length on velocity profile is studied for Re=500 and shown in Figure 5.7 . The peak horizontal local velocity of 0.0082 for b=2 & 4 is identified at the location *x/h* = 4. The local peak velocity is decreased and it moves towards the normal direction along the downstream direction. It occurs because of momentum loss in wall jet along the flow direction. It is also noticed that the horizontal velocity profiles is moved towards the negative normal direction when b is increased. It means that the wall jet spreading area is reduced when the fin length (b) is increased. The peak local horizontal velocity is increased along the downstream direction due to increase in fin length.

F. *Variation of velocity profiles at the mid of the Cavity*

The effect of Reynolds number (Re) and fin length (b) on velocity profile at the stream wise location corresponding to mid of the cavity are investigated. Variation of Velocity Profiles at this location for various Reynolds numbers(Re=100, 200, 300, 400, 500 & 600) and fin lengths (b = 2 to 4) is shown in Figure 8.

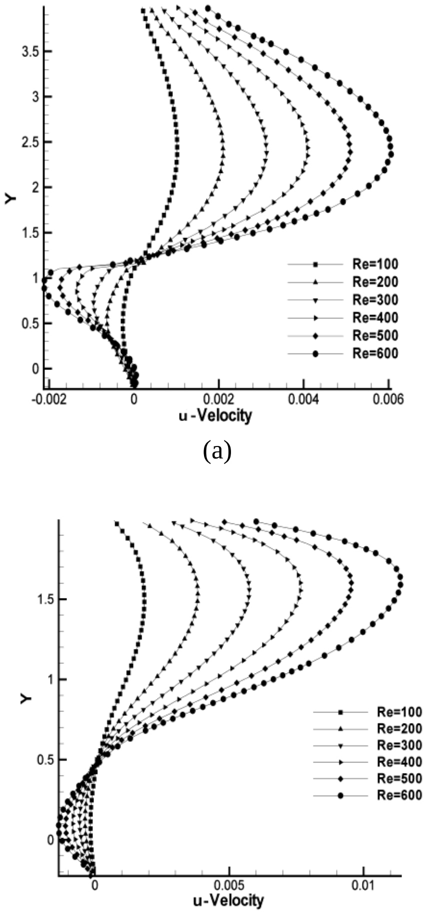


(a)

(b)

Fig. 7.Effect of Fin Length on Velocity Profile for Re = 500, (a) b = 2 (b) b = 4.

The positive horizontal velocity profile is moved towards the downstream direction and the negative horizontal velocity profile is moved towards the upstream direction when Reynolds number (Re) increased. The effect of geometry (fin length) on velocity profile is observed and compared. The shearing location between cavity and wall jet is decreased while fin length is increased.



(a)

(b)

Fig 8.Effect of Reynolds Number on Velocity Profilesat the Mid of theCavity for (a) b = 2 (b) b = 4

The shearing location is noticed for the case b=2 at Y=1.2 and the case b=4 at Y=0.5. This is obviously signifies that the geometry is influenced on shear layer between the cavity and wall jet. The horizontal component of velocity is increased when there is an increase in the fin length. It occurs due to the pressure increase at the leading edge location of the fin. It leads to the increase in momentum near the tailing edge of the fin.

V - CONCLUSION

The flow behavior of laminar and two-dimensional wall jet over the curved cavity with channel mounted fin is investigated. The conclusions of this study are listed below:

- The centers of secondary vortices are moved towards the downstream direction while Reynolds number is increased.
- The size of secondary vortex-2 is gradually decreased with the formation of a new corner vortex when there is a increase in Reynolds number.
- The size of secondary vortex-1 is decreased when fin length (b) is increased. This occurs due to the sudden momentum variation and change in flow direction of the wall jet.
- The horizontal velocity is increased while Reynolds number is increased. This is due to the increased momentum of the wall jet.
- The spreading area of wall jet is reduced when fin length (b) is increased. The peak local horizontal velocity is increased along the downstream direction due to increase in fin length.
- At the mid of the cavity, the positive horizontal velocity profile is moved towards the downstream direction and the negative horizontal velocity profile is moved towards upstream direction while Reynolds number is (Re) increased.
- The horizontal velocity increases when the fin length is increased. It occurs due to the increase in pressure at leading edge location of the fin.

REFERENCES

1. Schwarz WH, Caswell B. Some heat transfer characteristics of the two-dimensional laminar incompressible wall jet. Chemical Engineering Science. 1961; 16(3-4): 338-351.
2. Gorla RSR. Combined natural and forced convection in a laminar wall jet along a vertical plate with uniform surface heat flux. Applied Scientific Research. 1976; 31(6): 455-464.
3. Kumar Raja K, Rajesh Kanna P, Kumar Das M. Numerical simulation of mixed convection in a two-dimensional laminar plane wall jet flow. Numerical Heat Transfer, Part A: Applications. 2007; 52(7): 621-642.
4. Peters F, Ruppel C, Javili A, Kunkel T. The two-dimensional laminar wall jet. Velocity measurements compared with similarity theory. Forschung im Ingenieurwesen. 2008; 72(1): 19-28.
5. Glauert M. The wall jet. Journal of Fluid Mechanics. 1956; 1(6): 625-643.
6. Kanna PR, Das MK. Conjugate forced convection heat transfer from a flat plate by laminar plane wall jet flow. International Journal of Heat and Mass Transfer. 2005; 48(14): 2896-2910.
7. Rajesh Kanna P, Das MK. A short note on the entrainment and exit boundary conditions. International journal for numerical methods in fluids. 2006; 50(8): 973-985.
8. Arul Prakash M, Mayilsamy K, Kanna PR. Numerical Simulation of Two Dimensional Laminar Wall Jet Flow over Solid Obstacle. in Applied Mechanics and Materials.Trans Tech Publ. 2014;592: 1935-1939.
9. Arul Prakash M, Mayilsamy K, Rajesh Kanna, P. Numerical Investigations on Effect of Obstacle in an Incompressible Laminar Wall Jet Flow. in Applied Mechanics and Materials. Trans Tech Publ. 2016; 852: 747-753).
10. Botella O, Peyret R. Benchmark spectral results on the lid-driven cavity flow. Computers & Fluids. 1998; 27(4): 421-433.
11. Maheandera Prabu P, Padmanaban K. Laminar wall jet flow and heat transfer over a shallow cavity. The Scientific World Journal. 2015; 2015: 1-16.
12. Migeon C, Texier A, Pineau G. Effects of lid-driven cavity shape on the flow establishment phase. Journal of Fluids and Structures. 2000; 14(4): 469-488.
13. Mercan H, Atalık K. Vortex formation in lid-driven arc-shape cavity flows at high Reynolds numbers. European Journal of Mechanics-B/Fluids. 2009; 28(1): 61-71.
14. Chang MH, Cheng CH. Predictions of lid-driven flow and heat convection in an arc-shape cavity. International communications in heat and mass transfer. 1999; 26(6): 829-838.
15. Chen CL, Cheng CH. Experimental and numerical study of mixed convection and flow pattern in a lid-driven arc-shape cavity. Heat and mass transfer. 2004; 41(1): 58-66.
16. Cheng M, Hung K. Vortex structure of steady flow in a rectangular cavity. Computers & fluids. 2006; 35(10): 1046-1062.
17. Muthukannan M, Kanna PR, Bajpai A, Jeyakumar S. Numerical investigation on the fluid flow characteristics of a laminar slot jet on solid block mounted on a horizontal surface. Arabian Journal for Science and Engineering. 2014; 39(11): 8077-8098.
18. Asok SP, Sankaranarayanasamy K, Sundararajan T, Rajesh K, Ganeshan GS. Neural network and CFD-based optimisation of square cavity and curved cavity static labyrinth seals. Tribology International. 2007; 40(7): 1204-1216.

19. Cheng CH, Chen CL. Numerical study of effects of inclination on buoyancy-induced flow oscillation in a lid-driven arc-shaped cavity. Numerical Heat Transfer, Part A: Applications. 2005; 48(1): 77-97.
20. Glowinski R, Guidoboni G, and Pan TW. Wall-driven incompressible viscous flow in a two-dimensional semi-circular cavity. Journal of Computational Physics. 2006; 216(1): 76-91.
21. Arul Prakash M, Mayilsamy K, Murali G.Effect of Inlet Slot Dimension on Laminar Plane Wall Jet Flow over an Obstacle. Advances in Natural and Applied Sciences. 2017; 11 (4): 580-587
22. Ravinthiran A, Sankar N, Vijaya Rajan P and Priyadarshi Dutt, A Comparative Study and Flow Analysis of Multiple Branch Pipe Flow Header used in Tube Heat Exchangers, Indian Journal of Science and Technology, Vol 9(48), December 2016
23. Mercan H, Atalik K. Flow structure for Power-Law fluids in lid-driven arc-shape cavities. Korea-Australia Rheology Journal. 2011; 23(2): 71-80.
24. Ozalp C, Pinarbasi A, Sahin B. Experimental measurement of flow past cavities of different shapes. Experimental Thermal and Fluid Science. 2010; 34(5): 505-515.
25. Bajura RA, Szewczyk AA. Experimental Investigation of a Laminar Two Dimensional Plane Wall Jet. The Physics of Fluids. 1970; 13(7): 1653-1664.
26. Prabu PM, Padmanaban KP, Laminar Wall Jet Flow and Heat Transfer over a Shallow Cavity, The Scientific World Journal. 2015,1-16 , DOI: 10.1155/2015/926249.

AUTHORS PROFILE

Dr. Arul Prakash M is a Faculty Member in the Department of Mechanical Engineering of Sri Sairam Engineering College, Chennai. His Teaching Experience is 19 years in the area of Thermal Engineering, Fluid Flow analysis, Heat transfer, and Mechatronics. He obtained his post graduation in Thermal Engineering in the year 2000. He obtained his doctoral degree Ph.D from Anna University Chennai in the year 2018. Presently his research work is in the field of Computational Fluid Flow simulations and Heat Transfer. He has published five international publications in peer-reviewed journals. His areas of interest are in Computational Fluid Flow, Heat Transfer, Renewable Energy, jet impingement, nano-fluids and Automation.

Dr. K Mayilsamy, Professor, Department of Mechanical Engineering, PSG College of Technology, Coimbatore - 641004. Having Total teaching experience of 32 years, with 57publications till now. Life membership: FMFP, ISHMT and ISTE. No. of Ph.D guided -10. Area of research: Heat transfer, Thermal storage systems, Alternative Fuels.

Dr. Maheandera Prabu Paulraj is presently doing postdoc at IIT Indore. He received doctoral degree from Anna University Chennai (2016) and master degree from PSG College of technology Coimbatore under Anna University Chennai (2008). His Bachelor of Engineering degree from Kurinji College of Engineering and Technology Manappari (2005), He has 8 year teaching and 2 years research experience. He has published 9 international journals in various area of research like, Jet flow, impinging jet, solar energy, bio mass energy, multiphase flow modeling, etc.

Mr. A. Ravinthiran, working as Assistant professor in Mechanical Department of Sri Sairam Engineering College, Chennai, Tamil Nadu, India. Having 8 years of experience in the field of Engineering Design and Analysis.

Dr.G.Murali is working as Professor in Mechanical Engineering at Koneru Lakshmaiah Education Foundation (Deemed to be University), Guntur District, A.P.,India. He obtained his Ph.D. from Anna University, Chennai. He is having 19 years of Experience in teaching with 29 papers in reputed international journals and 1 patent. He has presented 8 papers in international conferences.

He was a Board of studies member of Mechanical Engineering Department at Adhiyamaan college of engineering (Autonomous), Hosur