Numerical Investigation to Understand the Effect of Curvature on the Corner Wall Jet

S.V.H. Nagendra, Prasant Nanda, DVS Bhagavanulu

Abstract: ***The present paper is based on the measurement of mean flow parameters on the corner wall jet using different geometries of the orifice. The mean flow parameters include mean velocity profiles, both in the longitudinal and span wise directions, decay of maximum velocity and growth of half width had been recorded experimentally. Simulation is done for calculating the flow parameters for comparison purpose. It has been observed that the corner wall jet is near to the exit similar to standard three dimensional wall jet. Experiments show good agreement the existing literature and the numerical results. The decay of maximum velocity is slower on concave surface compare to the decay on plane surface.***

Index Terms: Corner wall jet, Mean Velocity profile, Decay of maximum velocity, k-epsilon.

I. INTRODUCTION

The fluid coming out of a nozzle without having contact with surface is known as free jet. The wall jet is a term used to describe the fluid flow where a jet of fluid moves over the surface parallel on it. When one half of the plane wall jet surface rotates with 90 degree and makes contact with nozzle it is known as Corner wall jet as shown in the Fig. 1. The nature of the fluid near the boundary behave like a boundary layer flow but the outer layers of the shows the behaviour of free jet. The profile of the jet is shown in Fig. 2. The existing literature shows that considerable amount of research has been done on the three dimensional jet[1,2,5,6]. There is a wide variety of application of wall jet like aerospace , heating, air condition and ventilation, turbine blade cooling etc. The behaviour characteristic of wall jet is studied by examine the flow parameters like mean velocity profiles, maximum velocity decay, growth of half width in longitudinal as well as lateral directions of flow.

In case of corner jet it has been seen that the velocities in the outer layer are higher when compared to the plane wall jet. The flow in the corner wall jet grows much more rapidly parallel to the wall than vertically along the centre line, not unlike the plane wall jet.[12] . it also has been seen that the growth of half width higher in the corner wall jet when compared to plane wall jet [3,4,7,12]. The present investigation is based on the t effect of geometry change on the formation of corner wall jet. Limited experiment has been conducted on the corner wall jet configuration i.e. is mean flow parameters only. The turbulent intensities has been referred from existing literature [4,5,7]. the mean flow parameter taken under consideration are mean velocities profiles , decay of maximum velocity , growth of half width . A blower issues fluid at a velocity of 28 m/s which is passing through a orifice. To reduce the initial turbulence a twp meter settling chamber is provided before the orifice. The surface in front of the orifice are arranged as shown in Fig.2 . The Reynolds number of the flow is 63000 at the exit of the orifice in both the geometry.

II. METHODOLOGY

A. *Experimental Setup*

The instrumentation consist of furnace control micro-manometer of specification 1.99mm of h20 to 19.99 mm of h20 and 19.99 mm of h20 to 199.99 mm of h20. (0-5 m/s and 0-50m/s velocity). A traverse which is holding the probe have a span of 2 m x 1.4m x 1.4 m . The probe is a single point total pressure probe. The data is recorded manually on a excel sheet. Two different shape is taken for orifice that is Square shape and Circular shape. Both having the characteristic length equal to 34 mm. The range of the calculation is taken up to 0 to 40 times of width of orifice which is approximately 1400 mm. The surface has a initial straight portion of 20 times the width of the orifice and then a curvature of radius 1500 mm. This curvature exist only in one direction that is Y direction where as on the other side (Z-axis) is have a plane wall. This is because it replicates the practical condition of room or any other where one side is having curvature and other adjacent wall is having straight wall. The reading are recorder at namely 1h, 5h, 10h, 15h, 20h, 25h, 30h, 35h and 40h. During the experimentation the uncertainty occurred is about 3 to 4 percent.

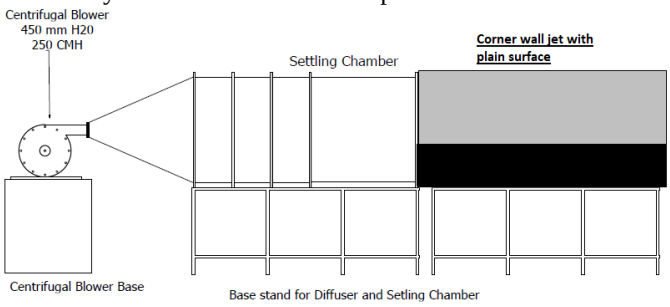

Fig. 1 The Experimental Setup with wall arrangement

Revised Manuscript Received on June 12, 2019.

S.V.H.Nagendra, Research Scholoar Department of Mechanical Engineering ,VSSUT Burla, Sambalpur, Odisha,India.

Prasant Nanda, Professor & Head, Training and placement Cell, VSSUT, Burla, Sambalpur, Odisha, India..

D.V.S.Bhagavanulu, Director, Sri Vidyaniketan, Tirupati, India.

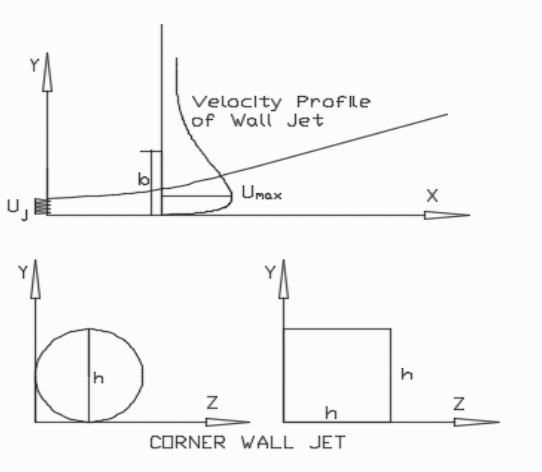

Fig 2 Wall jet Profile and Orifices position

B. *Numerical Analysis.*

The physical system is replicated exactly with true sizes in cad platform. The viscous model used to solve the problem is K-ε Realizable. The discritization is done by unstructured tetra hadrons patch independent method. inflation layer are considered based on the relation of y+ . Grid dependency test has been carried out for both the model square and circular. The code used for the numerical analysis is fluent 16.0 Academic Research *version.*

III. RESULTS AND DISCUSSION

A. *Mean Velocity profiles*

Fig.3,4 shows the velocity distribution of wall jet in vertical direction (y-axis) away from the surface. Numerical results shows good similarity with the experimental results. It also shows good agreement with Glauert . But it is observed that the velocities are higher in the outer region of the corner wall jet. The position of the maximum velocity is 0.2 in the case of plane wall jet where as the position of maximum velocity is 0.34 in the case of corner wall jet. But as it comes in the curvature region there is a shift of maximum velocity towards the wall. It can also seen that the velocity profiles at are similar up to straight portion but varies after 20h. In y direction it acts like a plane jet where as on the other side the concave curvature will show the centrifugal force effect. The centreline velocity is shown in fig 5 , which concluded that it is have same magnitude of de- acceleration in the direction of the flow.

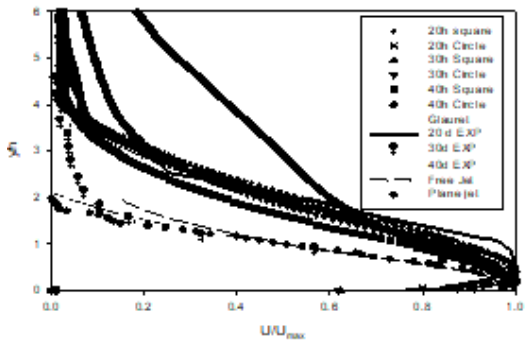

Fig 3 shows mean velocity profiles on X and Y direction

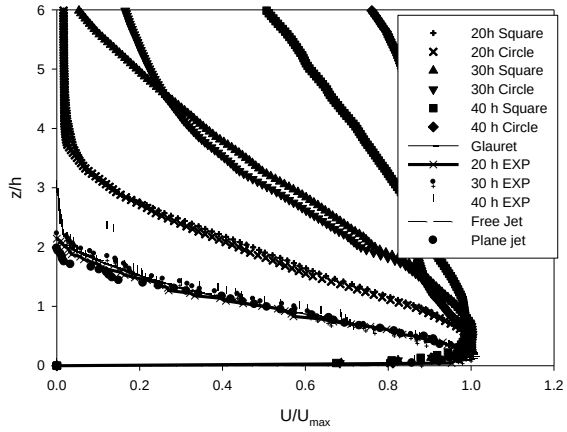

Fig 4 Shows mean velocity profiles on X and Z direction

B. *Centre Line Velocity*

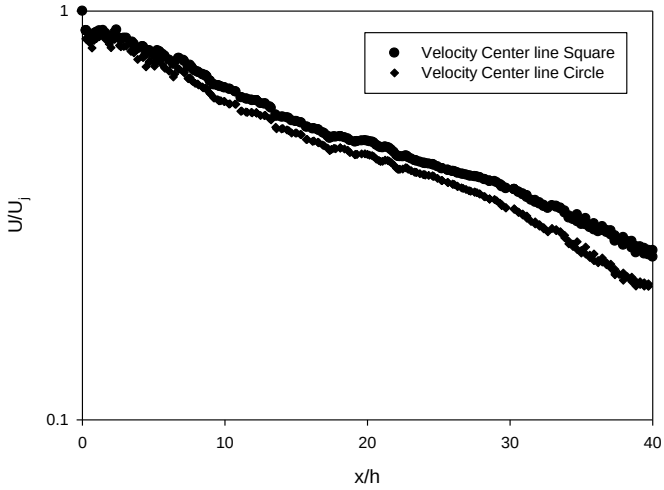

Fig 5 Shows velocity on the centre line for both the orifice.

C. *Turbulence Kinetic Energy*

The Fig.6 and 7 shows the turbulence kinetic energy of wall jet in vertical direction (y-axis) away from the surface at different stations. The behaviour well agrees with abrahamsson. It can also be seen turbulence is more when the fluid move near 30 d in the plane wall jet region. Fig 8 shows the intensity of turbulence upto 40 h on the centreline. The exit intensity of more in square orifice when compared to circular orifice.

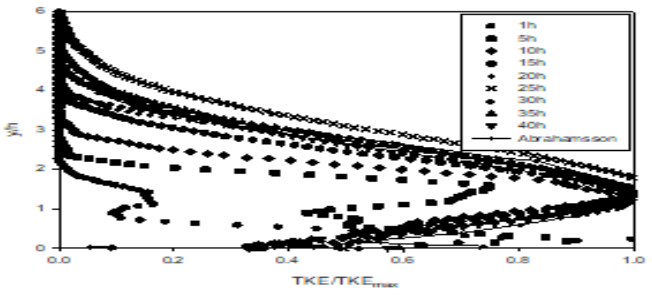

Fig 6 shows variation of Turbulence Kin. Energy on X&Y direction

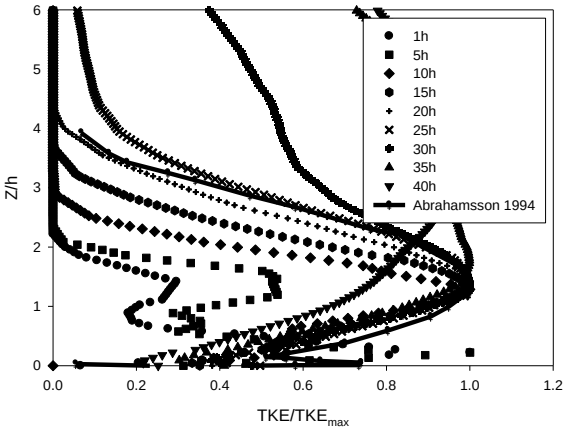

Fig 7 shows variation of Turbulence Kin. Energy on X&Z direction

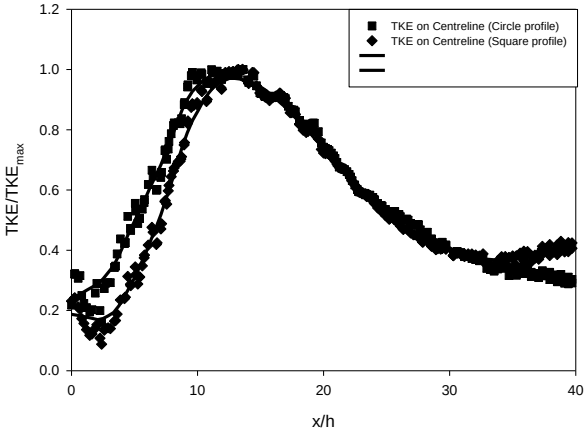

Fig 8 shows Turbulent kinetic energy over the centre line for the both the orifice

D. *Maximum Velocity Decay*

Fig.7 shows the decay of maximum velocity over the span from 0 to 40 h. It is observed that the decay is slower in the corner wall jet when compared decay for plane wall jet in the longitudinal direction. It is also found that the decay exponent is n=1.1 in the case of plane wall jet whereas it is 0.76 for the corner wall jet. Hence it can be concluded that the decay in the corner wall jet is slower when compared to the decay over the plane wall jet. The shape of the orifice does not make much difference in the decay profile upto 40 h station.

Table 1 showing decay exponents .

Orifice	Direction	range	Exponent	Reference
Square	X-Y	0-40	-0.72	Current Paper
Square	X-Z	0-40	-0.76	Current Paper
Circular	X-Y	0-40	-0.72	Current Paper
Circular	X-Z	0-40	-0.76	Current Paper
Free Jet	----	0-40	-0.93	Glauret
Plane Jet	----	0-40	-1.1	Glauret
Circular	----	0-40	-0.86	Poole 2014

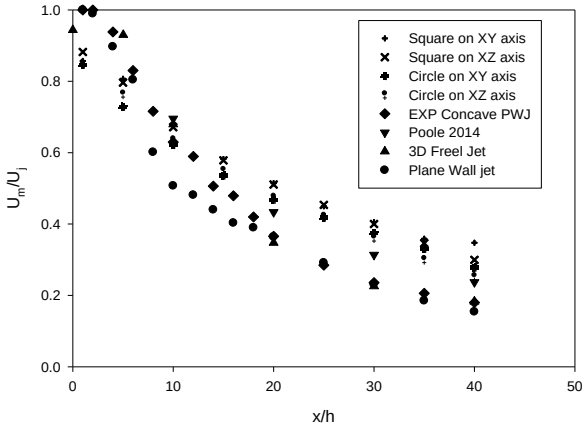

Fig 9 Shows decay of maximum velocity over the span

E. *Growth of Half Width*

Fig. 8 shows the growth of half width in the longitudinal direction. It is observed that the growth of half width in the corner wall jet is much higher when compared to the growth of half width in the plane wall, jet. The growth of half width in the corner wall jet suggests that secondary flow formation is quite significant in the outer layer of the corner wall jet. As the jet moves in the curvature zone the growth of half width reduces when compared to plane side of the setup. The same behaviour is seen in both the geometries. Fig. 9 and 10 gives the clear picture of growth of wall jet spreading over the concave as well as the plane surface . It can be seen that the jet spread normal to the surface is less on the concave surface side compared to the plane surface in both the cases of orifice.

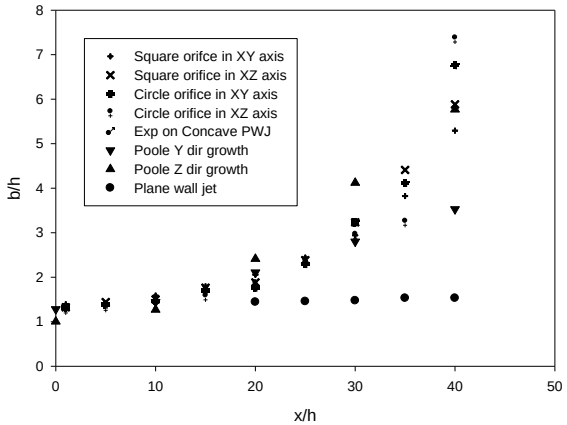

Fig 10 Shows growth in half width over the span

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

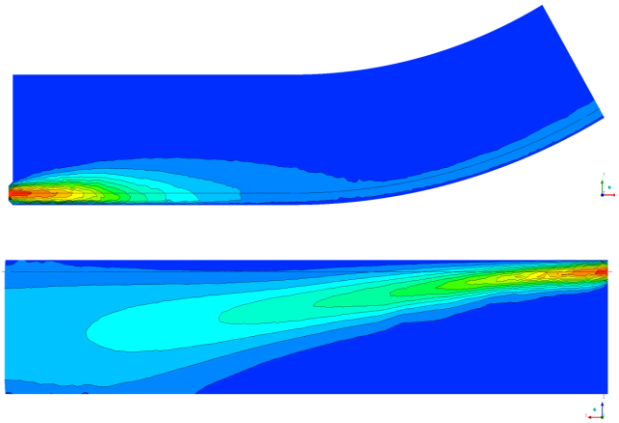

Fig 11 Contour representation of growth of half width for Square orifice

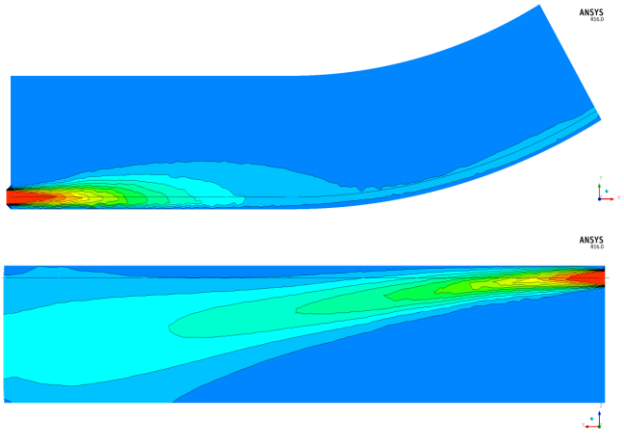

Fig 12 Contour representation of growth of half width for Circular orifice

IV. CONCLUSIONS

The orifice geometries used in the corner wall jet setup shows that the curve shape of the velocity profiles are similar to that of plane wall jet . The velocities in the outer layer region are higher when compared to the plane wall jet. The flow in the corner wall jet grows much more rapidly parallel to the wall than vertically along the centerline, not unlike the plane wall jet but as it passes over the concave region the growth tends to reduce. The decay of maximum velocity is slightly slower in the corner wall jet than the three-dimensional wall jet on plane surface and the concave region shows slower decay then the plane wall jet as well as the plane corner jet setup.

ACKNOWLEDGEMENTS

I deeply thanks AICTE INDIA for sanctioning the project under RPS scheme. The current work is the result of facilities procured during the project.

REFERENCES

1. Rajaratnam, N., and Pani, B. S., 1974. “Three-dimensional turbulent wall jets.”. ASCE J Hydraul Div, 100(HY1),pp. 69–83.
2. Launder, B. E., and Rodi, W., 1983. “The turbulent walljet - measurements and modeling”. Annual Review of FluidMechanics, 15, pp. 429–459.
3. Hogg, S. I., and Launder, B. E., 1985. “Three dimensional turbulent corner wall jet.”. Aeronautical Journal, 89(885),pp. 167–171.
4. Poole, B., 2014. “Hot-wire measurements in a three dimensional turbulent corner wall jet”. Master’s thesis,University of New Brunswick.
5. Abrahamsson H; Johansson B; Lo¨fdahl L (1994) A turbulent plane two-dimensional wall jet in a quiescent surrounding. European Journal of Mechanics B. Fluids 13: 533—556
6. Glauert. M.B., 1956, “The wall jet”, Journal of Fluid Mechanics, Vol. l.1, pp. 625-643.
7. Poole, B., & Hall, J. W. (2014). Turbulence measurements in the corner wall jet. In Proceedings of the ASME 2014 International Mechanical Engineering Congress and Exposition (pp. V007T09A092).
8. Nagendra S.V.H., Bhagavanulu D.V.S., Nanda P. (2017) Computational Study of Three Dimensional Wall Jet on Concave Surface. In: Saha A., Das D., Srivastava R., Panigrahi P., Muralidhar K. (eds) Fluid Mechanics and Fluid Power – Contemporary Research. Lecture Notes in Mechanical Engineering. Springer, New Delhi
9. S.V.H.Nagendra Prasant Nanda and DVS Bhagavanulu "Numerical Study of 3-Dimensional Wall Jet on Curved Surfaces",International Journal of Applied Engineering Research,2017,pp 5604-5609
10. Prasant Nanda, S.V.H. Nagendra and D.V.S. Bhagavanulu,"Computational Study of Effect of curvature on the Heat Transfer of Wall Jet" International Journal of Research in Advent Technology, Vol.6, No.5, May 2018, pp 797-800
11. Prasant Nanda, S.V.H. Nagendra and D.V.S. Bhagavanulu, Numerical Modeling of Wall Bound Flow for Thermal Comfort, International Journal of Mechanical Engineering and Technology 9(6), 2018, pp. 233–241.
12. Nagendra S.V.H., Bhagavanulu D.V.S., Nanda P “Flow Characteristics of a Three-dimensional Plane and Corner Wall Jet” Proceedings of the 44th National Conference on Fluid Mechanics and Fluid Power December 14-16, 2017, Amrita University, Amritapuri Campus, Kollam, Kerala, India Paper no 48.

AUTHORS PROFILE

S.V.H.Nagendra has 14 year of teaching experience, he is B.E. Mechanical engineering , M.E CAD/CAM and research scholar at VSSUT burla odisha. His field of research is Experimental fluid dynamics.

Dr. Prasant Nanda presently working as Professor & Head, Training&Placement Cell ,VSSUT,Burla. He has more than 25 years of teaching experience. Area of research is computational fluid dynamics.

Dr. DVS Bhagavanulu has 22 years of Administrative, Teaching & Research experience. He did his BE (Civil Engg.), ME (Aeronautical Engg.), Ph.D from IIT Madras, Chennai and also a Post Doctoral. He is Director to Sree Vidyanikethan Engineering College, Tirupati. And has published 136 technical Pagers in Refereed Journals, International and National Conferences. He has completed funded projects worth Nearly Rs. 1 crore. He is a Life Member in Institution of Engineers, ISG, SSI, SHRM, ISC, IAE, ISSSE, ASCE, CSI, IWRS, and ISH.