Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2017
(ICMERE2017) 18 – 20 December, 2017, Chittagong, Bangladesh

ICMERE2017-PI-206

NUMERICAL STUDY OF A SWIRL NOZZLE AT MODERATE REYNOLDS NUMBER

Md. M. Islam*, **Sumaiya Tasnim and Zahir U. Ahmed**

Department of Mechanical Engineering, Khulna University of Engineering & Technology, Khulna-9203, Bangladesh
mahfuz.ridoy@yahoo.com*, sumaiyatasnim114@gmail.com, zuahmed@me.kuet.ac.bd

Abstract-*The effect of swirl on the exit velocity profiles of a swirl nozzle is numerically investigated in incompressible turbulent impinging air jets with the aim of improving the understanding of flow characteristics within the nozzle. In this regard, an existing lab-based swirl nozzle (Diameter, D = 40 mm) is used for the simulation which is capable of seamless transition from a non-swirling (Q_r = 0) jet to a highly swirling (Q_r = 0.55) jet. Existing simulations on swirl nozzle is largely associated with geometric means, such as vane and helical insert within the nozzle, which, however, hinders the flow and provides an inefficient understanding of swirl flow. As such, in this study, the swirl flow is generated aerodynamically using steamwise axial flow and tangential flow via multiple tangential ports located upstream of the exit plane. Both flow then coalesce later and generate a combined flow prior to reaching the exit plane, and all flows pass through a contraction en route to a straight section of length (L=447mm). The nozzle is designed in Solidworks 2013 and numerical simulations are carried out using in ANSYS FLUENT 18 via SST k-ω turbulence model. The boundary conditions are provided from the experimentally derived data. The results show that the introduction of low levels of swirl into an impinging jet results in centerline velocity decay but a significant reduction in turbulent kinetic energy at the wall jet region. Within the range of swirl numbers investigated, drastically different velocity and pressure contours among different sections are observed.*

Keywords: Turbulent, Swirl, Nozzle, CFD, Aerodynamic

1. INTRODUCTION

A jet is a stream of fluid that is projected into a surrounding medium, usually from some kind of a nozzle, aperture or orifice. Swirl means to move with an eddying or whirling motion. A swirling turbulent jet combines the interesting characteristics of a rotating turbulent fluid motion. Swirling jets have of great practical significance in engineering applications, such as jet engines, turbo-pumps, and allow fundamental study of such complex processes. In combustion applications, their ability to create reverse flow regions near the jet nozzle has been exploited for the purpose of swirl-stabilizing the flame. The efficiency of chemical reactors and mixing devices is enhanced by making use of the faster spreading and more rapid mixing of the jet fluid with its surrounding by swirling compared with non-swirling jets. Swirl nozzles provide high velocity of fluid with a swirling motion which is pretty useful. These nozzles can be used for spray formation and impinging jet formation.

A number of methods have been used in order to generate swirl. Ullrich [1] used a combination of tangential air inlets and adjustable vanes in his examination of annular swirling jets. Rose [2] rotated a pipe at 9500 rpm so as to provide an approximately fully developed turbulent flow in solid body rotation. By this means, it was only possible to obtain a relatively weak degree of swirl. Gore and Ranz [3] imparted rotation to axial pipe flow by means of a rotating perforated plate in which holes were drilled either parallel to the axis or at 45 degree to the axis. By this means, they obtained a continuous variation in angular velocity. When using swirl vanes, they found that the flow was not axisymmetric and the flow field was complicated by additional secondary flows induced by the vanes. Chigier and Beer [4] introduced air both axially and tangentially into the swirl generator and varied the degree of swirl by varying proportions of air introduced axially and tangentially into the swirl generator. Rose [2] observed that, in comparison with the non-swirling jet, the jet with swirl spreads at a larger angle, entrains reservoir fluid more rapidly and consequently displays a more rapid reduction of mean-velocity and growth of turbulence intensity. Dombrowski and Hasson [5] found that a unique relationship exists between discharge coefficient and spray angle dependent only upon the value of the orifice length/diameter ratio. Chigier and Chervinsky [6] observed that, for the case of strong swirl, a vortex is generated in the region close to the orifice resulting in a displacement of the axial velocity maximum from the jet axis. Pratte and Keffer [7] found that the flow achieved a self-similarity for the mean velocities rather quickly while the normal turbulent intensities reached a self-similar state after a longer period of jet development. Experiments carried out by Sislian and Cusworth [8] indicate a strong dependence of the turbulent stresses on the local strain of the mean flow in most regions of the flow, which suggests that an eddy viscosity type of turbulence model, e.g., the k-ε model, rather than a Reynolds stress model could acceptable for the prediction of such flows. Panda and McLaughlin [9] reported that the vortex breakdown affects the axial velocity distribution and rapidly replaces the potential core with a large amount of turbulence. Upon interacting with the vortex breakdown, the shear layer along the jet periphery loses its organized structure and, in general, random turbulence follows. Rhode [10] showed that, the central zones exist in swirling flow fields and range in axial extent from x/D = 1.5 to 1.85 for the parameter variations of swirl vane angle φ (phi) = 0, 45, and 70 degrees with side-wall expansion angle α (alpha) = 90 and 45 degrees. Also, a relatively thin vortex core is experimentally observed downstream of the central zone.

Existing research on the analysis of flow characteristics of swirling jets shows abrupt changes in the axial and tangential velocity profile of fluid inside the nozzle with the increase in tangential-to-axial volumetric ratio. To understand the physics behind this unexplainable behavior of swirling jet this research project was carried out. This research is also important as there is no inserts or guide-vanes to generate swirl. This is because geometrical swirl generation causes dead zones to form on the axis and limits the range of swirl numbers, results in a bifurcation of a single jet into multiple jets, is likely to distort the flow and alter heat transfer characteristics as well as impingement pressure distribution and thus alter fundamental jet characteristics. So, this research will highlight the exact effect of relative proportions of axial and tangential flow on flow characteristics of swirling jets inside a nozzle.

2. METHODOLOGY

The dimensions of the swirl nozzle was selected based on the study on a Turbulent Swirl Nozzle by Ahmed et al. [11].

The inlet dia at nozzle bottom = 50 mm
The dia of three tangential inlets = 12 mm
The total length of nozzle = 623 mm
The nozzle outlet dia = 40 mm

Figure 1 shows the swirl nozzle design with necessary dimensions. Figure 2(left) presents the rendered view of swirl nozzle done in Solidworks 2013. The figure (right) also shows the cad view of the three tangential ports and their relative orientations (120^o apart circumferentially).

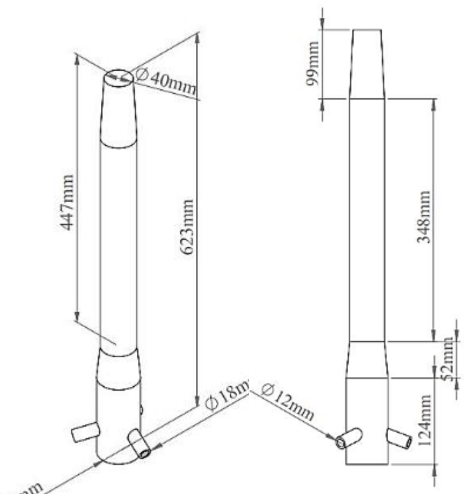


Fig. 1: Dimensions for the design of swirl nozzle

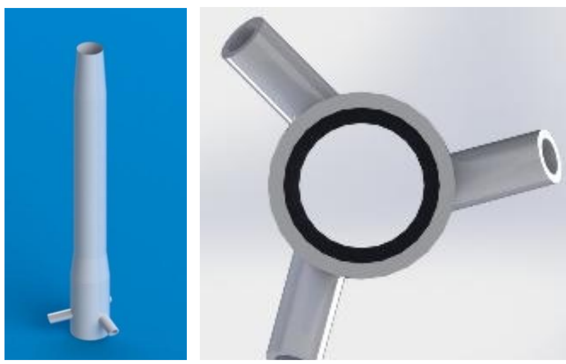
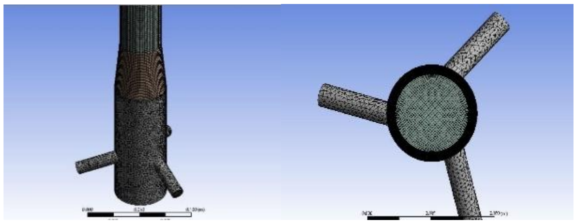


Fig. 2: Designed and rendered model of swirl nozzle and fluid element inside

ANSYS Meshing is used to generate mesh on the fluid domain, and a mesh of 294518 elements and 386645 nodes is chosen by mesh sensitivity analysis.



(a) (b)

Fig. 3: Zoomed in view for the generated mesh for (a) Isometric view, (b) Top view

Steady-state pressure-based solver with couple algorithm, absolute velocity formulation, with SST k-ω viscous model was selected for CFD analysis. The effect of gravity is neglected. In the case of shear stress transport (SST) k-ω model, a differential equation for specific rate of dissipation (ω) is solved except for dissipation rate of turbulent energy (ε). Menter introduced SST model for k-ω to integrate exact and strong k-ω equations near the wall region, with ρ independent k-ϵ equations in the far field. Turbulent viscosity in this model is calculated from below equation.

$$\mu_T = \frac{\rho k}{\omega_{max}} * \frac{1}{[\frac{1}{\alpha}\frac{SF}{a_1\omega}]}$$

Ambient air at 15^0C is used for fluid medium with standard atmosphere having density 1.2kg/m^3 and viscosity of 1.7894*10^{-5}kg/m-s. The CFD analysis was done for two cases of swirl intensities $Q_r = 0\ and\ 0.55$and the corresponding swirl number is 0.0 and 0.32 [11]. The boundary conditions for each Q_r in this thesis was determined from the experimental data of a study [12], where swirl number (S) was used to determine swirl intensity instead of Q_r.

For the pressure inlet, an atmospheric pressure with zero turbulent kinetic energy and dissipation rate is assumed, whereas for the pressure outlet, atmospheric pressure with turbulence specified by 5% intensity and hydraulic diameter equal to the nozzle exit diameter are applied. Finally, no-slip and axial symmetry conditions at the (impingement) wall and geometric axis are used, respectively.

Table 1 shows the different axial and tangential flow rates for different Q_r corresponds to given swirl numbers in Ahmed et al. [12]. In this regard, the density of air is assumed to be 1.2 Kg/m^3 for all cases, similar to the experimental data.

The pressure-based coupled algorithm is used to simultaneously solve the coupled system of continuity and momentum equations, whereas turbulence quantities are solved separately in a segregated manner; an approach which significantly improves the rate of convergence when compared to a segregated algorithm [13] whereby all variables are solved separately.

Table 1: Flow ratio for different swirl conditions

SL No.	Flow ratio Q_r	Swirl No. S	Flow rate in axial inlet	Flow rate in each tangential inlet
1.	0	0	330 litre/min = 0.0066 Kg/s	0 litre/min = 0 Kg/s
2.	0.55	0.32	150 litre/min = 0.003 Kg/s	180/3 litre/min = 0.0012 Kg/s

The PRESTO (PREssure STaggering Option) is applied for the pressure discretization as it is suitable for steep pressure gradients such as those in swirling jets, and the second-order upwind discretization scheme is used for the momentum, turbulent kinetic energy and dissipation equations. The converged solution is assumed to be achieved when the residuals of the flow parameters are less than 10^{-5}.

Table 2: Table for different boundary conditions

SL No.	Q_a (liter/sec)	Q_t (liter/sec)	$Q_r = \frac{Q_t}{Q_T}$	Swirl Number S
01.	330	0	0	0
02.	150	180	0.55	0.32
	Total mass flow rate, Q_T = 330 liter/sec			

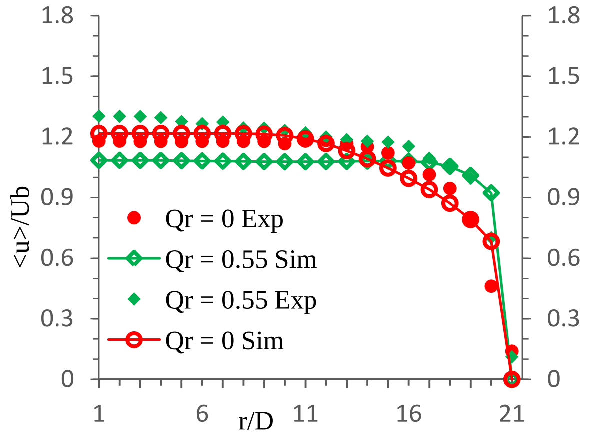


Fig. 4: Data validation of <u>/U_b vs r/D chart for Q_r= 0 and 0.55 with experimental data [12]

The results obtained from the simulations are first validated by comparing with experimental data [12] for the case Q_r = 0 and 0.55 which are equivalent to the swirl numbers, S = 0 and 0.32 at the exit plane as reported in [12]. A relatively fair agreement between the simulated data and the experiments has been found. Other discrepancy in results between experimental data and numerical simulation occurred due to the variations in inlet conditions. In the experimental study, honey combs were used before the axial inlet to ensure uniform linear flow. But in this simulation no such environment was created. Moreover there is always some uncertainty in the experimental data. So, experimental data might deviate from actual results. The deviation may also occur due to the inability of RANS approach to simulate highly shear and complex flows.

3. RESULTS AND DISCUSSION

For the analysis of axial mean velocity and pressure, a line was drawn along the centerline in the axial direction of the nozzle from the bottom inlet surface to the top outlet surface. Then results are presented for centerline axial mean velocity U_c along the centerline (r/D = 0) osition $^{x}/_{D}$ for the conditions i.e. Q_r= 0 and Q_r= 0.55. The velocity is non-dimensionalized by the bulk axial velocity U_b which is determined by the total volumetric flow divided by the area at the nozzle exit plane, similarly to the experiment. The corresponding Reynolds number, defined by $Re = \rho U_b D/\mu$, is 11,600.

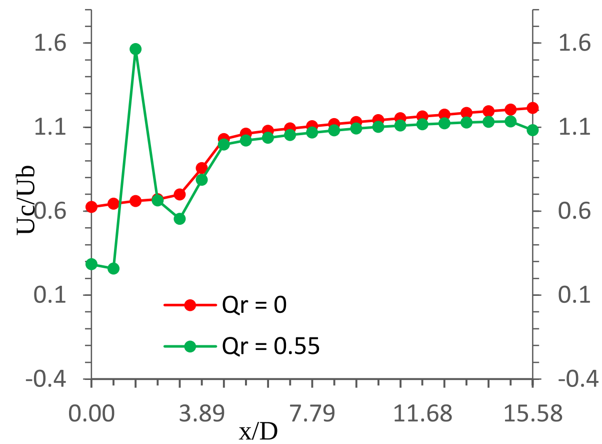


Fig. 5: Axial mean velocity, U_c/U_b along the centerline for the range Q_r= 0, 0.55.

Figure 5 shows the results for normalized axial mean velocity, U_c/U_b against normalized axial position x/D for different Q_r cases ranging from non-swirling (Q_r= 0) to high swirling (Q_r= 0.55) jets. It appears that the centerline velocity variations at different Q_r is confined to within x/D ≈4, beyond which changes in axial velocity almost linearly increases with x/D, but minimal variations with Q_r. A sharp peak is observed for Q_r= 0.55 at around x/D = 1.56, the axial distance immediately beyond tangential entry. This peak is attributed to the sudden acceleration of flow after mixing both tangential and axial flows.

Figure 6 shows the resultant velocity streamlines for different swirl conditions produced from the axial inlet and three tangential inlets.

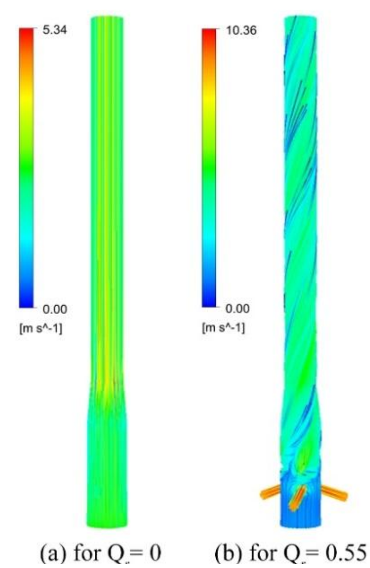
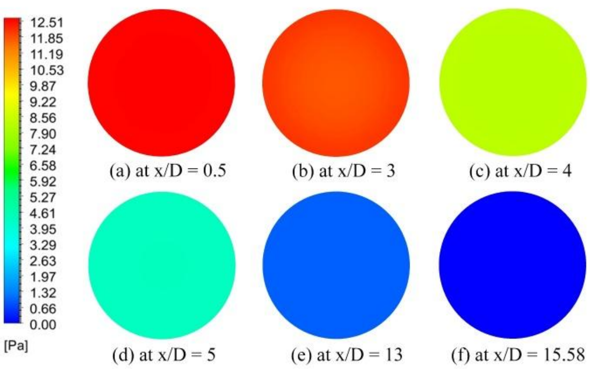


Fig. 6: Resultant velocity (V) streamline for different swirling conditions

Form the figure 6(a) it is found that, the velocity at the center of the nozzle along the axis is the highest and it reduces towards the wall surface region due to the boundary layer development at the surface. From the figure 6(b) it is found that, the tangential inlet velocity causes turbulence and low swirl effect at the center of the nozzle. In the converging section this turbulence and velocity is increased due to the higher mixing of both axial and tangential flows and the fluid eventually passes with a whirling motion along the axis to all the way to the outlet.

Figure 7 shows the pressure contours for Q_r= 0 and Q_r= 0.55 in figure 7(i) and 7(ii), respectively, at different cross-sectional planes ranging from x/D = 0.5-15.58. The results depicted in figure 7(i) shows that the pressure at the axial inlet of the nozzle is the highest and it reduces gradually towards the outlet region and is equal to the atmospheric pressure at the outlet tip. The static pressure is also found to be uniform across the plane and purely axisymmetric.

(i)



(ii)

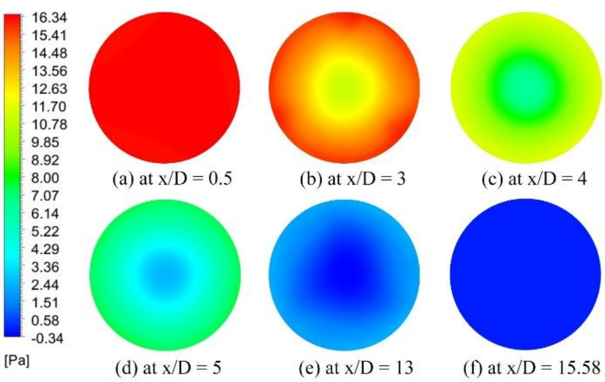


Fig. 7: Pressure contours at different cross-sections of fluid body for: (i) Q_r= 0 and (ii) Q_r= 0.55.

In contrast, figure 7(ii) shows that the pressure at the axial inlet of the nozzle is the highest and it reduces gradually towards the outlet region. Due to the tangential inlets, a little pressure drop occurs along the centerline and a little negative pressure is emerged at the outlet tip.

Figure 8 shows the wall static pressure contour on the nozzle wall surface for different swirl conditions. The pressure at the inlet region is the highest and it reduces gradually as the fluid travels towards the outlet. For the swirling flow, maximum pressure appears in the intense mixing region, at or immediately above the tangential ports. The significant difference in wall static pressure between a non-swirling and a swirling flow is found to be upto contraction region, beyond which only remarkable variation is magnitude.

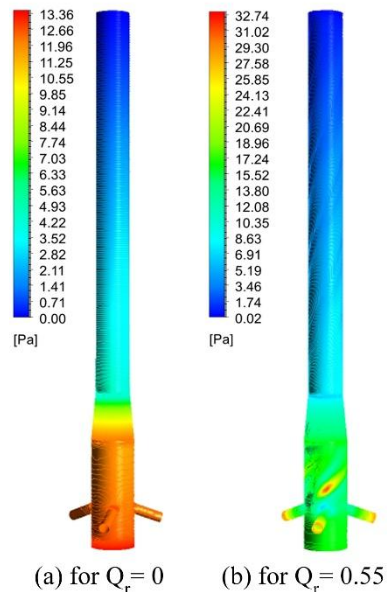


Fig. 8: Wall static pressure contour at wall for different swirling conditions

4. CONCLUSIONS

The effect of swirl is investigated numerically for an incompressible turbulent air jet in comparison with non-swirling jets using the commercial software package ANSYS Fluent (version 18). The RANS approach with SST k–ω model is applied to help study the mean flow and turbulent characteristics for these jets. The study investigates the general performance of two different swirl conditions with results validated against the experimental data [12]. The results show the centerline velocity variation is confined to within x/D ≈4, beyond which changes in axial velocity almost linearly increases with x/D. A sharp peak is observed for Q_r= 0.55at around x/D = 1.56, which is attributed to the sudden acceleration of flow after mixing both tangential and axial flows. The static pressure is found to be not uniform and purely axisymmetric in the middle of the nozzle, i.e. x/D = 3-5 when swirl is introduced. When high velocity tangential inlets are used, a remarkable pressure drop occurs and a negative pressure is emerged in the middle of the nozzle. The wall static pressure variation between non-swirling and swirling flows appears up to the contraction section.

5. ACKNOWLEDGEMENTS

University Grants Commission, Bangladesh is acknowledged for funding provided for this work via Khulna University of Engineering & Technology under the project no. CASR/41/05: S/L no. 21.

6. REFERENCES

[1] H. Ullrich, "*Stromungsvorgange in Drallbrennern mit regelbaren Drall und bei rotationsymmetrischen Freistrahlen*," Forschungsberichte auf dem Gebiete Ingenieur wesens, vol. 26, no. 1, 1960, pp.19-28

[2] W.G. Rose, *A Swirling Round Turbulent Jet*, Journal of Applied Mechanics, vol.29, TRANS. ASME, vol. 84, Series E, 1962, pp. 616-625

[3] R.W. Gore and W.E. Ranz, *Back Flows in Rotating Fluids Moving Axially Through Expanding Cross Sections*, American Institute of Chemical Engineers Journal, vol. 10, no. 1, 1964, pp. 83-88

[4] N. A. Chigier and J. M. Beer, *Velocity and Static Pressure Distributions in Swirling Air Jets Issuing From Annular and Divergent Nozzles*, Journal of Basic Engineering, TRANS. A S M E , Series D , vol. 86, 1964, pp. 788-798

[5] N. Dombrowski, D. Hasson, *The flow characteristics of swirl (centrifugal) spray pressure nozzles with low viscosity liquids*, American Institute of Chemical Engineers, 1969, pp. 604–611

[6] N. A. Chigier and A. Chervinsky, *Experimental Investigation of Swirling Vortex Motion in Jets*, ASME Applied Mechanics Division, 1966

[7] B. D. Pratte and J. F. Keffer, *The Swirling Turbulent Jet*, ASME, 1971, Paper No. 72-FE-18

[8] J.P. Sislian and R.A. Cusworth, *Measurements of Mean Velocity and Turbulent Intensities in a Free Isothermal Swirling Jet*, American Institute of Aeronautics and Astronautics, Inc., vol. 24, no. 2, 1985

[9] J. Panda and D. K. McLaughlin, *Experiments on the instabilities of a swirling jet*, Phys. Fluids 6 (I), 1994, 1070-6631/94/6(1)/263/14

[10] D. L. Rhode, *Predictions and measurements of isothermal Flow fields in axisymmetric combustor geometries*, Ph.D. thesis, Oklahoma State University, 1981

[11] B. Thomas, Z. U. Ahmed, Y. M. Al-Abdeli, M. T. Matthews. *The Optimisation of a Turbulent Swirl Nozzle Using CFD*. Proceedings of the Australian Combustion Symposium November 6-8, 2013, the University of Western Australia

[12] Z. U. Ahmed, Y. M. Al-Abdeli, F. G. Guzzomi. *Impingement pressure characteristics of swirling and non-swirling turbulent jets*. Experimental Thermal and Fluid Science, vol. 68, 2015, 722–732

[13] Z. U. Ahmed, Y. M. Al-Abdeli, M. T. Matthews. *The effect of inflow conditions on the development of non-swirling versus swirling impinging turbulent jets*. Computer and Fluids vol. 118,2015, 255-273.

7. NOMENCLATURE

Symbol	Meaning	Unit
k	Turbulent kinetic energy	(m^2/s^2)
Q_r	Flow ratio	Dimentio-nless
Re	Reynolds number	Dimentio-nless
S	Swirl number	Dimentio-nless
U_b	Bulk axial velocity at the nozzle exit plane	(m/s)
$\langle u \rangle$	Time mean axial velocity component	(m/s)