© 2016 IJEDR | Volume 4, Issue 2 | ISSN: 2321-9939

Effect of Geometry on the Turbulent Jet in Convergent Ducts Using Computational Tool

Praveen K. Kasare[1*], Pankaj Mishra[2], Veerendra Patle[3]
[1*,2,3] Department of Mechanical Engineering
NRI Institute of Research and Technology, Bhopal

Abstract - **Geometrical parameters to obtain the contraction shape that produces uniform turbulent jets have been studied numerically and experimentally over several decades. This paper review the several studies done by researchers in field of fluid flow through various shaped ducts. These numerically and experimentally studies aimed to suitable nozzle contraction design and optimization. The analysis provides that the equation of third order polynomial profile gives uniform mean velocity profile. Reynold stress Model (RSM) model provide better results for mean velocity profile and turbulence intensity. Computational fluid dynamic study with Shear Stress Transport (SST – k–x) model for rectangular cross section achieves the recommended contraction ratio and maximum uniformity. An important observation comes out from review, most authors provide qualitative criteria only based on acceptable non- uniformity and avoidance of boundary layer separation to select contraction profile and none of them had determined the optimum profile within the same family group of profile.**

***IndexTerms* - CFD, Polynomial equation, duct, turbulence, RSM.**

I. INTRODUCTION

Nozzle geometry influence the characteristics of outlet jet from convergent nozzle. The main parameters which affect the geometry of convergent nozzle are contraction ratio, length of contraction, total length of the nozzle. Before the advancement of digital computer the studies are analytical nature after the advent of efficient computer computation fluid dynamics is highly emerging field[2]. Proper contractions shapes are used for generating uniform mean flow with low turbulence for wall bounded flow. Suitable nozzle design is important in boundary layer and jet flow research. The main concern of design methods for aerodynamics contractions is to produce a steady, parallel and uniform axial flow[1]. In most of these situations uniform free-stream flow with low turbulence is required. Flow through a contraction produces a rotation-free strain in the core flow. With past research it have been found that the nozzle shape has a noticeable effect on the mean and turbulent flow characteristics and the entrainment rate in the near field of turbulent jets.

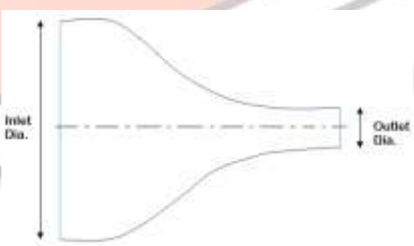

Figure 1: Convergent Duct Contraction profile

Turbulent jets are fluid flows produced by a pressure drop through an orifice. Their mechanics has recently received research attention that has resulted in a much improved understanding of the process by which they entrain surrounding fluid. Turbulent jet flows are encountered in a variety of engineering applications including combustion, chemical processes, pollutant discharge, and cooling, mixing and drying processes. Nozzles are widely used in connection with many different engineering applications, mainly to generate jets and sprays. In most laboratory studies dealing with jets, the emphasis has been on designing nozzles which create a uniform flow with low turbulence intensity at the nozzle exit.
On the other hand, in mixing applications, higher turbulence intensity at the exit may be more desirable. The nozzle exit flow serves as the initial condition for the downstream flow. In most instances, flow non-uniformity and turbulence originate within the nozzle, but the nozzle contraction is generally designed to minimize these effects. The principal concern of design methods for aerodynamics contractions is to produce a steady, parallel, and uniform exit flow. It can be shown that whenever a converging duct segment is attached to constant-area segments at its inlet and exit, regions of adverse pressure gradient will occur along the wall near each end, possibly causing boundary layer separation. If separation occurs, it will act to degrade flow uniformity and steadiness, both of which are essential in a test facility. Separation is usually avoided if the adverse pressure gradients are minimized by making the contraction sufficiently long. The control of turbulent jet flows has applications in various fields such as combustion, aerodynamic noise, and jet propulsion.

In combustors it is important to enhance turbulent mixing of the chemical species in order to achieve high combustion efficiency and to reduce the emission of pollutants. In jet engines, aerodynamic noise can be reduced by controlling flow unsteadiness that produces noise. One aim of enhanced mixing in jet propulsion applications is to decrease the plume temperature and suppress infrared radiation. The mixing rate of a turbulent jet can be significantly altered by applying a suitable excitation at the jet orifice. Since the external forcing interacts with the natural modes of the jet in a nonlinear way, it is not possible to predict which kind of actuation is optimal to increase mixing. Various experiments have been carried out that study the reaction of jets to the nozzle geometry.

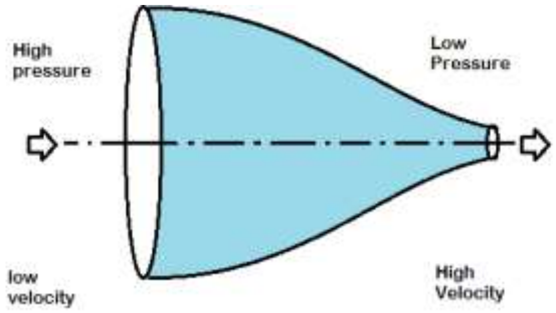

Figure 2: Pressure & Velocity change from inlet to outlet

II. LITRETURE REVIEW

In the field of geometrical variation to obtain uniform flow a lots of analytical investigation, numerical analysis and experimental works has been performed. In this section the works of some of the research groups will be reviewed in brief.

Chmielewski et al.(1974) done boundary layer consideration for aerodynamics contraction design. He had taken nozzle with different inlet curvatures. His analysis shows that at fixed contraction ratio the length to inlet diameter ratio(L/D_i) is varied in increments of 0.25 to bracket the value at which separation occurs and before contraction boundary layer grow slowly for lower curvatures[1]. Hussein and Ramjee et. al.(1976) investigated the performance characteristics of four different axisymmetric contraction shapes with the same contraction ratio (C=11)for incompressible flow experimentally. The pre and post-contraction mean and turbulent velocity profiles and spectra, and the variation of the mean and turbulent velocities along the axis as a function of local contraction ratio and axial length were presented paper. The results show that all the nozzles are of essentially equal effectiveness as far as the core flow in the exit plane is concerned. The conclusion of this study was that third order polynomial profile gives uniform mean velocity profile and low boundary layer thickness [3].

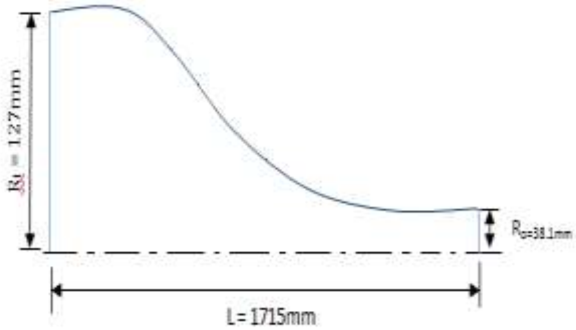

Figure 3:Geometrical Dimensions of Nozzle profile

Brown et. al. (2006) studied the evolution of grid turbulence in a planar contraction by focusing on the flow at the centre-symmetry plane. Measurements were carried out in water with inlet Taylor-microscale Reynolds number varying from 51 to 99. Detailed laser-Doppler anemometry measurements show that the streamwise fluctuating velocity component for contraction ratio C<2.5 closely follows the decay of grid turbulence in a straight channel. Furthermore, the turbulent kinetic energy reaches a minimum value in the range of contraction ratio 1.5<C<2.5. Turbulent intensity, independent of contraction angle and Reynolds number, decays exponentially[4]. Jang et al. Studied experimentally and numerically effects of axisymmetric contraction on a turbulent pipe flow the reduction in turbulence intensity in the core region of the flow was discussed on the basis of the budgets for the various turbulent stresses as they develop downstream. The contraction generates a corresponding increase in energy in the near-wall region, where the sources for energy production are quite different and of opposite sign compared to the core region, where these effects are caused primarily by vortex stretching[5].Balachandar R., Barron R. M., Shademan M., and Yu Y et. al. (2012) performed a computational study to determine the turbulent characteristics of incompressible fluid flow through nozzle at Reynolds number of 50,000. He suggest that one equation and two equation models are not appropriate for the internal flow in the nozzle, but Reynolds Stress Model(RSM) give reasonable prediction about the turbulent flow parameters.His simulation represent that turbulence intensity in each direction is very low in the core area inside the four nozzles and turbulence intensity in the longitudinal direction increases significantly in all the nozzles in wall regions [6]. Yassen et. al. (2015) investigated experimentally the three dimensional model of open circuit wind tunnel contraction and also analysis this computationally. His work concerned with design optimization of existing open circuit wind tunnel contraction[7].

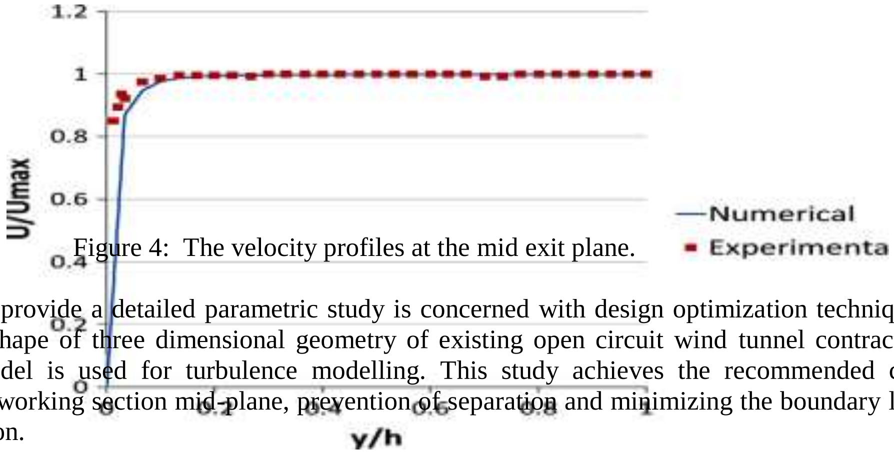

Figure 4: The velocity profiles at the mid exit plane.

His investigation provide a detailed parametric study is concerned with design optimization technique with (RANS) analysis of flow for design shape of three dimensional geometry of existing open circuit wind tunnel contraction, Shear Stress Transport (SST – k–x) model is used for turbulence modelling. This study achieves the recommended contraction ratio, maximum uniformity at the working section mid-plane, prevention of separation and minimizing the boundary layer thickness at entrance to the working section.

III. CONCLUSION

From the review of various researches we conclude that there is large research has been done to found the enhanced geometrical shape for the contraction and to get the minimum boundary layer thickness with uniformity in mean flow. This study suggest that for mean velocity profile SST-k-x model is used where on for large change in turbulence intensities RSM produce good results. It has been experimentally validated by various researchers which help new scholars for their work.

REFERENCES

[1] Chmielewski G E, 1974, "Boundary-layer considerations in the design of aerodynamic contractions" Journal of aircrafts, Vol.11 Issue 8 pp.435-438.

[2] Hussain, A. K. M. F., and V. Ramjee. 1976, "Effects of the axisymmetric contraction shape on incompressible turbulent flow." *Journal of Fluids Engineering* 98.1 pp. 58-68.

[3] Ramaseshan S., and Ramaswamy M. A., " A Rational method to choose optimum design for two-dimentional contractions " *Journal of fluids Engineering*(2002) Vol. 124, pp 544-551.

[4] Brown, Matthew L., Mehran Parsheh, and Cyrus K. Aidun. "Turbulent flow in a converging channel: effect of contraction and return to isotropy." *Journal of Fluid Mechanics* 560 (2006): 437-448.

[5] Jang, Seong Jae, Hyung Jin Sung, and Per-ÅgeKrogstad. "Effects of an axisymmetric contraction on a turbulent pipe flow." *Journal of Fluid* (2011): 376-403.

[6]."CFD study of effects of geometry variations on flow in a nozzle"*Engineering Application of Computational Fluid Mechanics* (2012) vol. 6, No.3, pp. 412-425.

[7] Abdelhamed A.S., Yassen Y.El-S.,and ElSakka M.M. "Design optimization of three dimentional geometry of wind tunnel contraction" *Ain Shams Engineering Journal* (2015)Vol. 6, pp 281-288.