

Comparison of Turbulence Models in Simulation of Plane Jet Flow

Prof. K. D. Manwar¹, Dr. R. S. Tatwawadi²

¹Assitant Professor, Civil Engineering Department, Jawaharlal Darda Institute of Engineering and Technology, Yavatmal, 445001, Maharashtra, India.

² Principal, Jawaharlal Darda Institute of Engineering and Technology, Yavatmal, 445001, Maharashtra, India.

¹kunal_manwar@jdiat.ac.in, ²principal@jdiat.ac.in

Abstract

Plane turbulent jets are widely used in a variety of applications. Such as jets in drying processes, air curtains for room conditioning, heating, mixing, ventilating applications etc., therefore, jet flow is a very important research topic in both experimental and analytical research.

Depending upon the position and forces acts on the jet, jets are broadly classified as turbulent free jet and turbulent wall jet. The studies conducted herein provide the comparison between various turbulence model present Computational Fluid Dynamics(CFD). Numerical approaches were used in the present investigation. CFD technique helped to understand better the hydrodynamics of the jet flow and correlate the numerical result.

Among the all turbulence model studied, the k- ϵ turbulence model performed the best. Performance of k- ω model was found to be somewhat superior to Spalart-Allmaras and Reynolds stress model.

Keyword: Plane Jet, Turbulence Modelling, ANSYS Fluent, Jet Characteristics

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. Jets can travel long distances without dissipating. Jet fluid has higher momentum compared to the surrounding fluid medium. In the case that the surrounding medium is assumed to be made up of the same fluid as the jet, and this fluid has a viscosity, the surrounding fluid is carried along with the jet in a process called entrainment.

Jets can be classified broadly in two ways as,

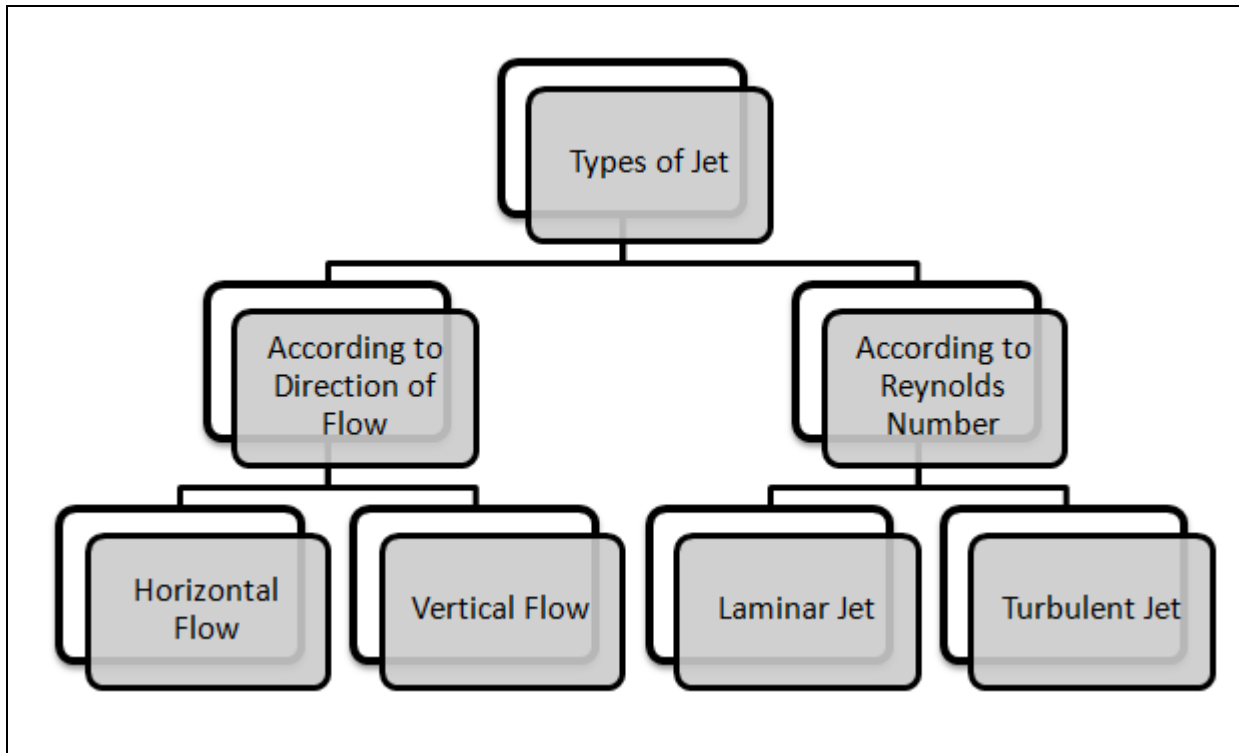


Fig.1: Types of Jet

During the work we have greatly concentrated on the study of Turbulent Jets. Whenever a moving fluid enters a quiescent body of the same fluid, a velocity shear is created between the entering and ambient fluids, causing turbulence and mixing. In nature, the situation occurs where a river empties in a lake or estuary, or occasionally when a wind blows through an gap. But, perhaps the most clearly defined jets are those produced when a fluid is discharged in the environment through a relatively narrow conduit, such as an industrial discharge released through a pipe on the bank of a river, lake, or coastal ocean. Since the properties of a turbulent flow greatly depend on the geometry of the flow domain and on the type of forces acting on the fluid, almost every situation is a separate problem requiring specific investigation. We shall therefore limit ourselves here to the most basic case, that of a jet penetrating in an otherwise quiescent fluid.

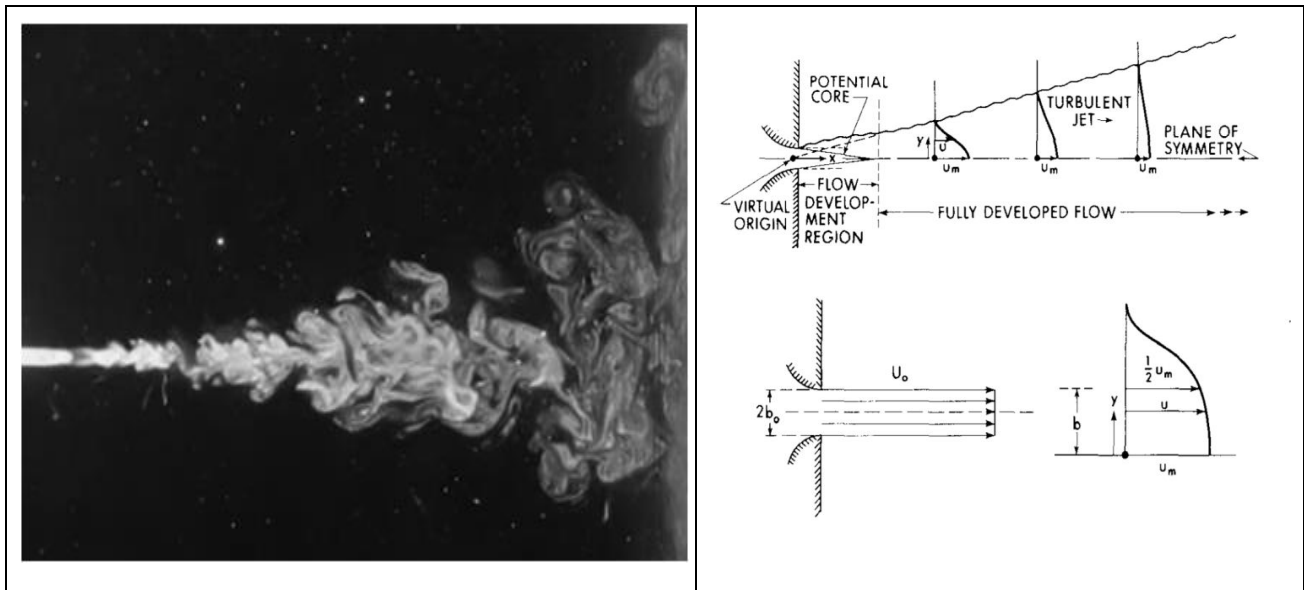


Fig.2: Turbulent Jet (Source: Rajaratnam, N, (1976), Turbulent jets)

Let us consider a jet of water coming from a plane nozzle of large length into a large body of water or a jet of air into a large expanse of air. Let the height (or thickness) of the jet be $2b_0$ and let U_0 be the uniform velocity in the jet. If we use suitable flow visualization techniques, we will find that the jet mixes violently with the surrounding fluid creating turbulence and the jet itself grows thicker. Figure shows a schematic representation of the jet configuration discussed above, which is known as the plane turbulent free jet. Experimental observations on the mean turbulent velocity field indicate that in the axial direction of the jet, one could divide the jet flow into two distinct regions. In the first region, close to the nozzle, known commonly as the flow development region, as the turbulence penetrates inwards towards the axis or centre-line of the jet, there is a wedge-like region of undiminished mean velocity, equal to U_0 . This wedge is known as the potential core and is surrounded by a mixing layer on top and bottom. In the second region, known as the fully developed flow region, the turbulence has penetrated to the axis and as a result, the potential core has disappeared. For a plane jet, the length of the potential core is about $12b_0$, and in this chapter we will consider only the fully developed flow region and we will discuss the flow development region. In the fully developed flow region, the transverse distribution of the mean velocity in the x -direction, i.e. the variation of u with y at different sections, has the same geometrical shape as shown in Fig. At every section, u decreases continuously from a maximum value of u_m on the axis to a zero value at some distance from the axis. Let us now try to compare the distributions at different sections in a dimensionless form. At each section, let us make the velocity u dimensionless by dividing it by u_m at that section and let b represent a typical length for that section. Let us take b as the value of y where u is equal to half the maximum velocity. Let us now plot u/u_m against y/b . We will find that the velocity distributions at different sections fall on one common curve. X denotes the axial distance from the nozzle. The velocity profiles at different sections which could be superposed in this manner are said to be 'similar'. The two non-dimensional quantities are called, respectively, the velocity scale and the length scale. A very large number of flows in the field of turbulent jets exhibit this

property of similarity. In order to use these similarity profiles for predicting the mean velocity field in any particular problem, we have to be able to predict the manner of variation of the velocity and length scales.

In this study of the Water Jets and the effects of particles in the designated jets using the various turbulence models flow of Turbulent Jets With help of ANSYS FLUENT 18.0 we would evaluate the performance of the Plane Jet and try to resolve the issues related to the drawbacks in the Jet-flows.

Literature Survey

The study of turbulent jets has been an important topic for many years. The earliest experimental

investigation was conducted by plane turbulent jet (Rajaratnam, 1976). Since then, free jets have

been studied with different experimental equipment's such as pitot tubes (Forthmann, 1934), Hot

Wire Anemometry (Everitt and Robins, 1978), Laser Doppler Anemometry (Ramaprian and Chandrasekhara, 1985) and Particle Image Velocimetry (Mi et al., 2007).

During the study, mainly concentrated on the study of Turbulent Jets. Whenever flow enters in a rest or moving fluid enters in a quiescent body of identical fluid through narrow conduit, the properties of flow greatly depend upon the pure mathematics of the flow domain and on the sort of forces functioning on the fluid; almost every situation is a separate problem requiring specific investigation.

The plane jet generated by fluid flow from the slotted nozzle is an important case of turbulent flow. This flow is symmetric about jet axis; hence only one side of the jet axis is taken for the simulation study. As there is no wall, simulating this case is very much easier compared to the wall jet case. Figure 3 shows the definition sketches of turbulent free jet.

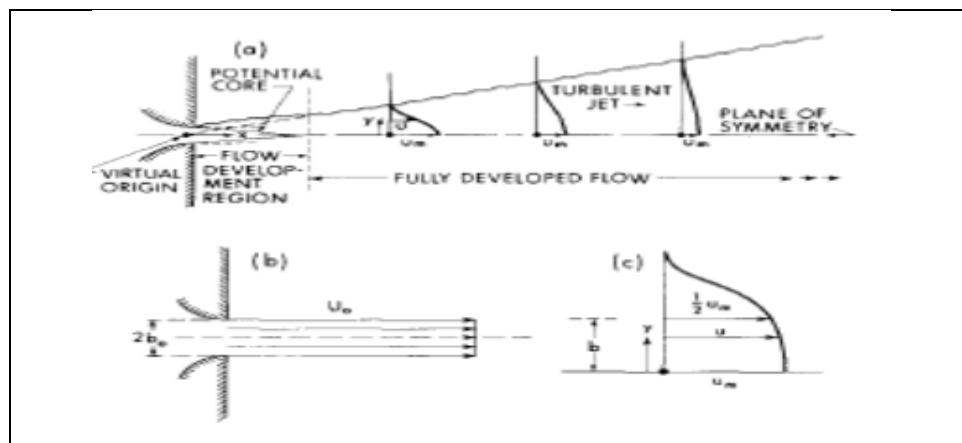


Fig. 3 Free Plane Jet

Used in many engineering applications, no really satisfactory general solution has been given until now. The solution usually presented in Rajaratnam (1976) textbook, due to tollmien's and gortler are based on the simple algebraic model of turbulence, and this leads to the serious shortcoming. Similarity solution using a modern turbulence energy transport model of turbulence together with the assumption of constant integral turbulence scale across the jet profile. The partial differential equation describing the jet flow field was transformed to five simultaneous first order ordinary differential equation for the dimensionless similarity variables, which were solved numerically.

A series of experimental investigation of air jet were performed by Liu Q. et. al., (1996). The resultant solution is in much better agreement with experimental data than any of the general solution known so far. Its distribution of eddy viscosity roughly agrees with tollmien's result in the outer part of the jet. Near the jet axis, the present solution shows instructively how the transverse diffusion transport of turbulent fluctuation which it is not possible to account for in an algebraic model is the cause underlying the deficiency in tollmien's solution.

METHODOLOGY

For the study, The CFD commercial code ANSYS 18.0 - FLUENT, was used to predict the turbulent jet flow by providing numerical solution of solving governing transport equation of continuity and momentum equation in two dimension.

Steps solving problem by ANSYS FLUENT to solve Engineering problem using ANSYS FLUENT the necessary steps are-

1. **Pre-analysis:** These steps are important, especially to obtain the correct boundary condition. For example, if want to simulate airflow over the wing of a commercial airplane you have to know the range of speed for that wing, you will also look for previous experimental or numerical result, and the relevant theories.
2. **Geometry:** In this study work using plane turbulent jet of 200 cm length, 1 cm depth slit is provided with 50 cm height of jet wall. Using ANSYS design modeller software.
3. **Mesh:** Meshing is one of the most important steps for simulation. Simulation result depend on mesh quality. Low quality Mesh can produce poor simulation result, even divergence. These steps are pre-processing.

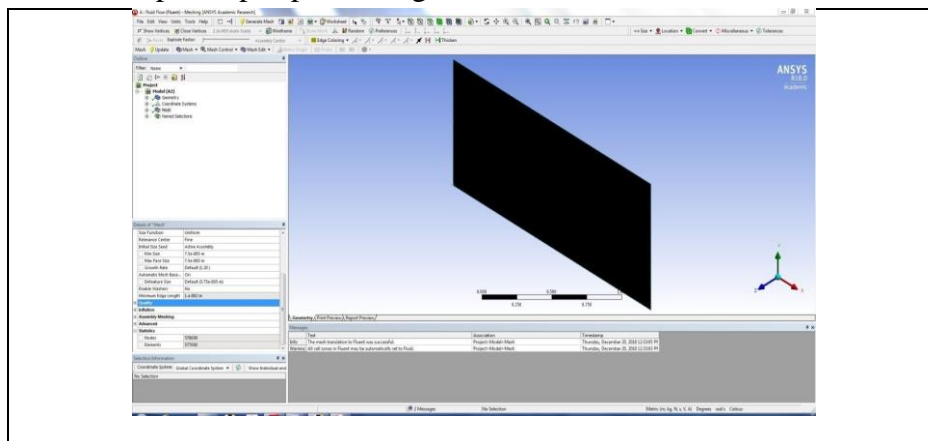


Fig.3: Meshing in ANSYS Fluent

4. **Setup:** It is done with solver ANSYS, In setup step, Give input for solution accuracy, boundary condition, physics involved, material involved, properties involved etc.

5. **Solution:** The steps are post processing, i.e., analyzing simulation result and data and verification. Whenever the solution is converge stop the iteration automatically and given the finalized answer of the solution. No further iteration is required for calculation.

6. **Result:** We used the CFD-post processing. We can also use FLUENT solver for post-processing, but CFD-post processing is easier & convenient. We want to verify whether or not our simulation is correct? For verification, we can check whether basic physical laws are maintained or not, using FLUENT. For example in this work the verify the result with the tollmien solution, Forthmann, Goertler solution.

TURBULENCE MODELLING

In computational modeling of turbulent flows, one common objective is to obtain a model that can predict quantities of interest, such as fluid velocity, for use in engineering designs of the system being modeled. For turbulent flows, the range of length scales and complexity of phenomena involved in turbulence make most modeling approaches prohibitively expensive; the resolution required to resolve all scales involved in turbulence is beyond what is computationally possible. The primary approach in such cases is to create numerical models to approximate unresolved phenomena. This section lists some commonly used computational models for turbulent flows.

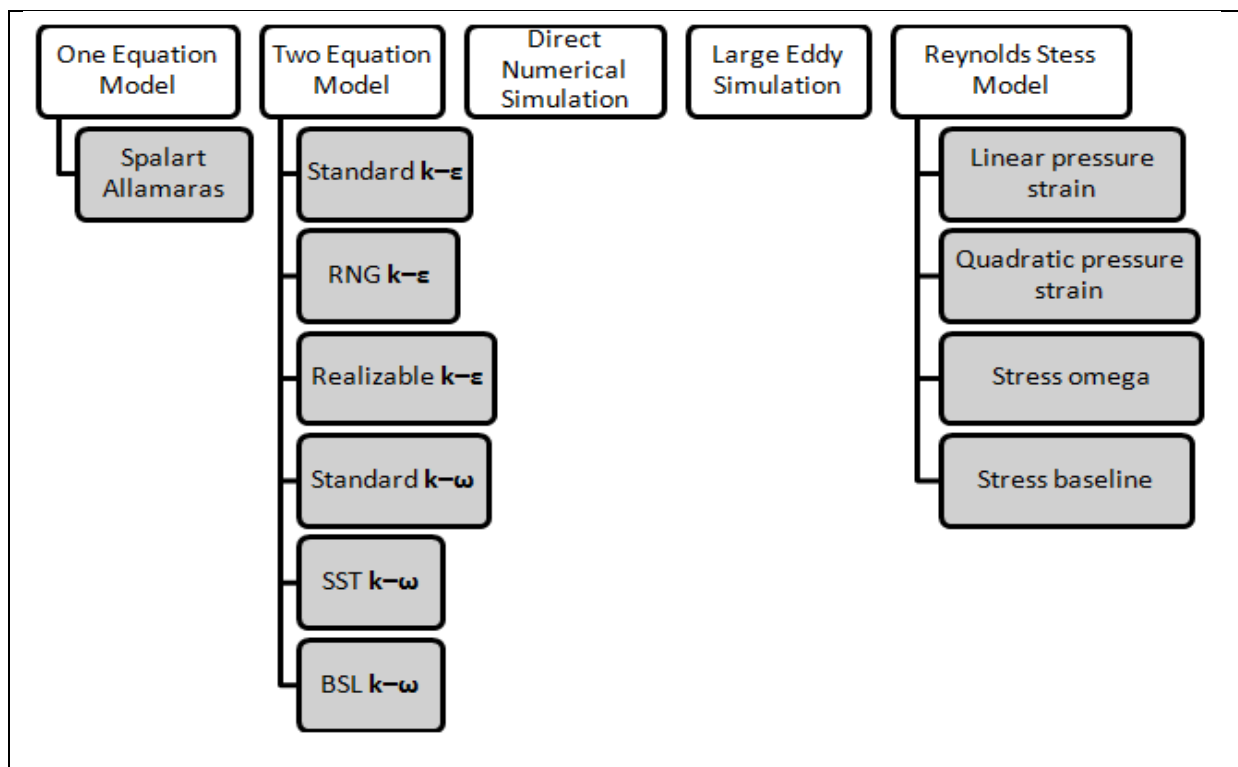


Fig.4: Turbulence Modeling

Results & Discussion

Studies on plane wall jets were compared for velocity profile, jet spread, velocity decay and entrainment ratio of the jet. Table is a list of results reported in the literature and the results were used to assess the accuracy of the numerical simulations.

Table 1: Theoretical jet characteristics value of plane free jet

Jet Characteristics	Equation	Number	Source
Velocity profile	$\frac{u}{u_m} = \exp(-0.693\eta^2)$	(i)	Rajaratnam (1976), ASHRAE(1989)
Jet Spread	$Y_{0.5} = 0.1x$	(ii)	Awbi (1991), Rajaratnam (1976)
Velocity Decay	$\frac{u_m}{u_0} = 3.5 \sqrt{\frac{b}{x}}$	(iii)	Rajaratnam (1976)
Entrainment Ratio	$\frac{Q_x}{Q_0} = \sqrt{2} \frac{u_0}{u_m}$	(iv)	Walker (1977),

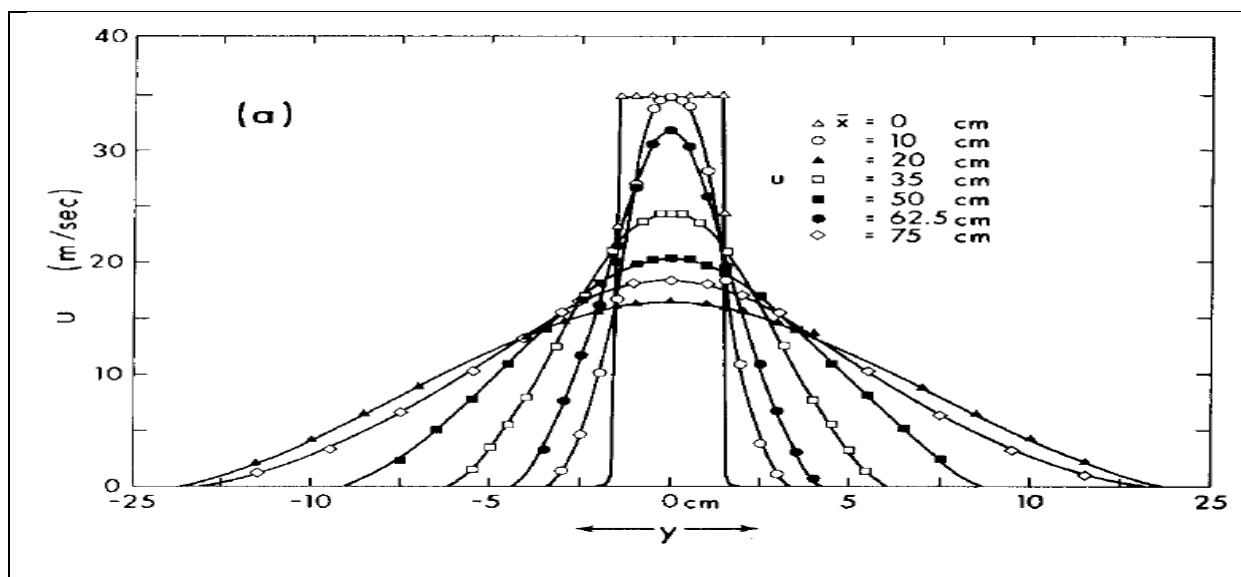


Fig. 5: Velocity Profile of Plane Turbulent Free Jet

(Source : Turbulent Jet by N. Rajaratnam 1976)

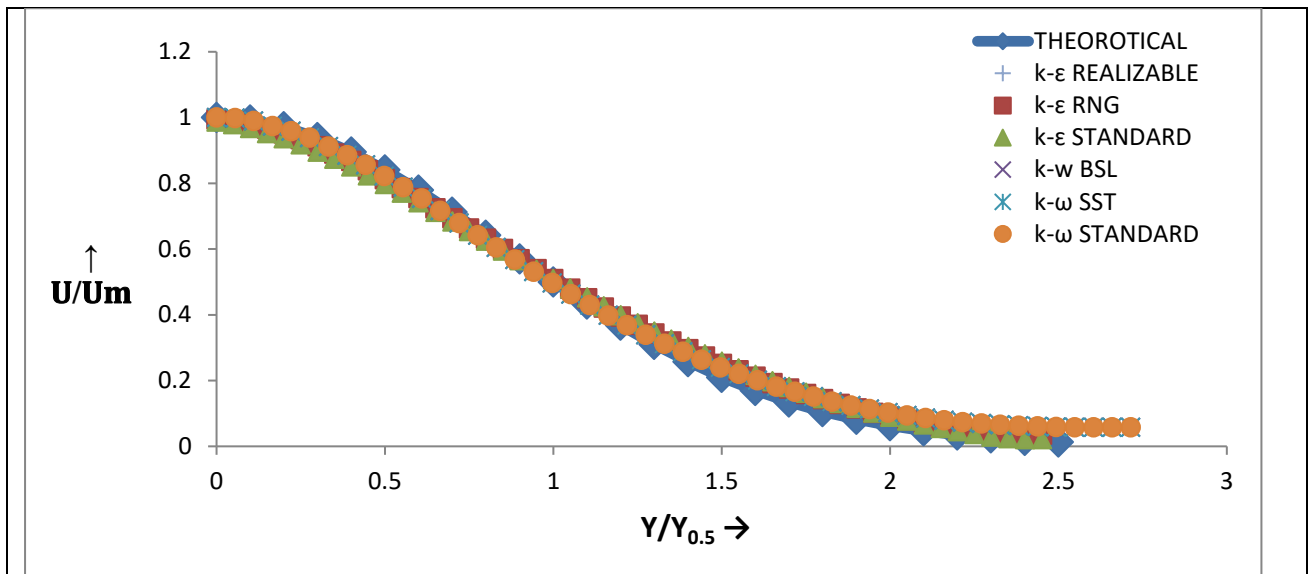


Fig. 6(a) Velocity Profile of plane turbulent free jet

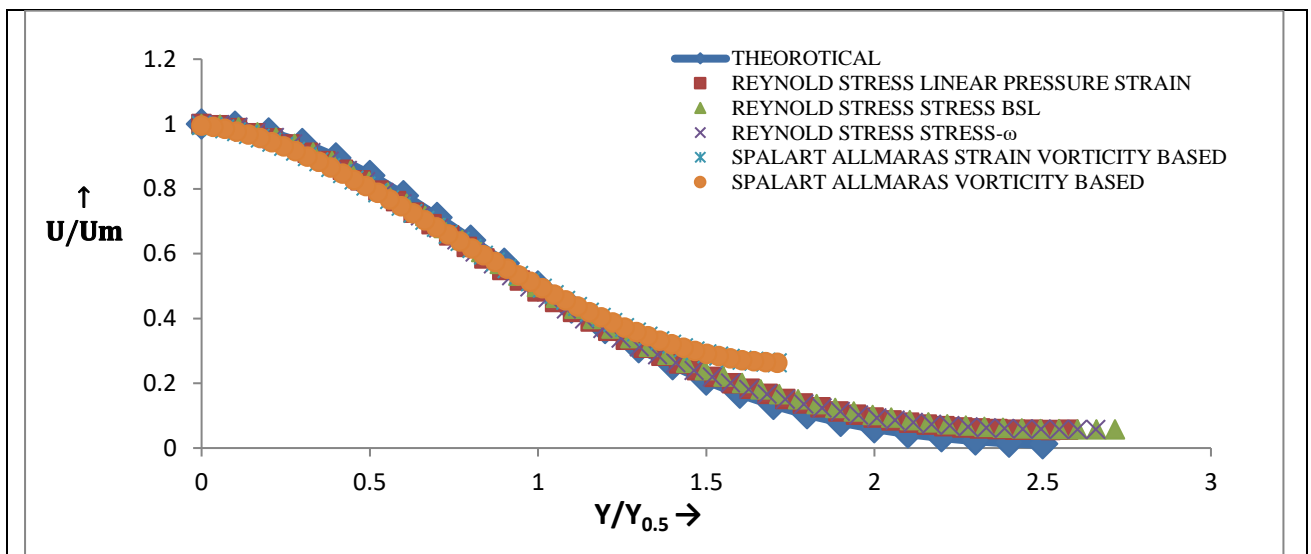


Fig. 6(b) Velocity Profile of plane turbulent free jet

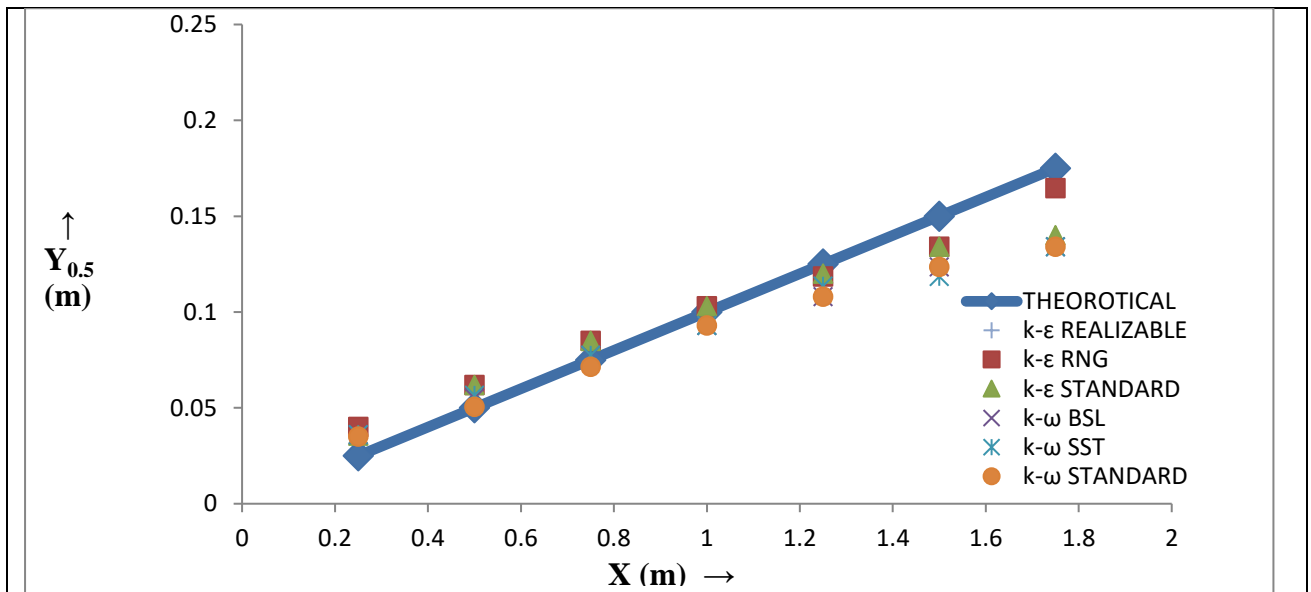


Fig. 7(a) Jet Spread of Plane Free Jet

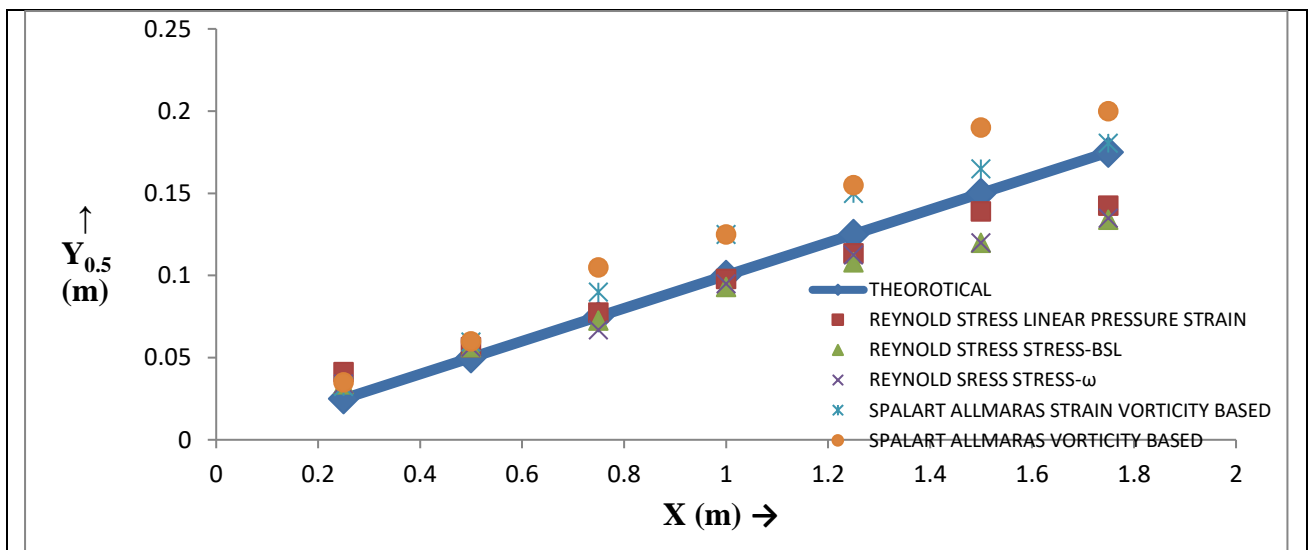


Fig. 7(b) Jet Spread of Plane Free Jet

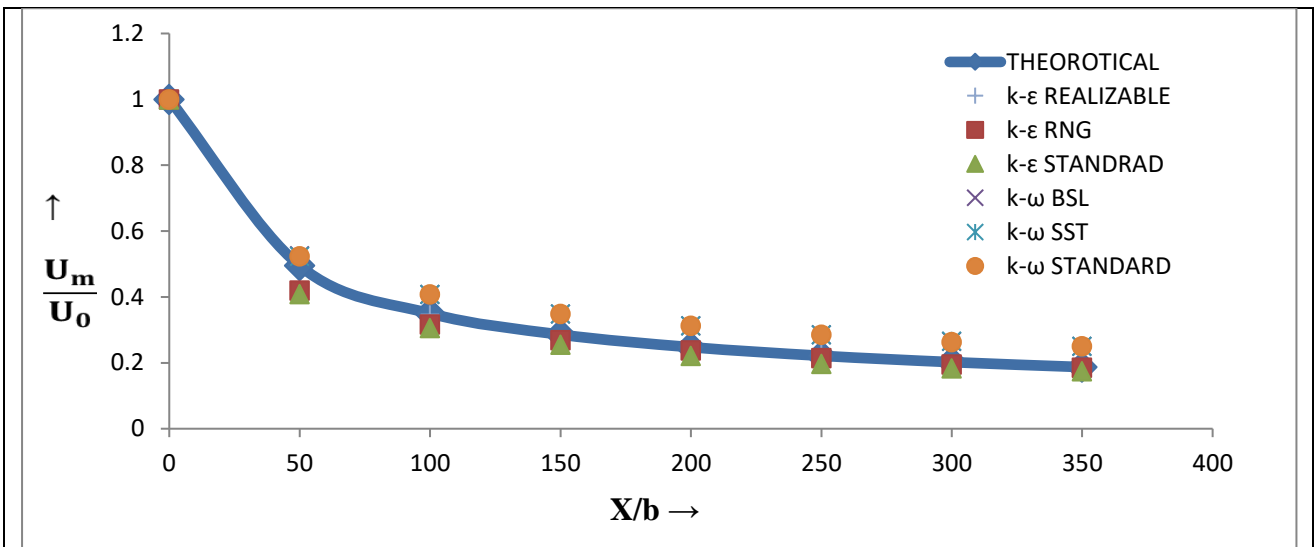


Fig. 8(a) Velocity Decay of Plane Free Jet

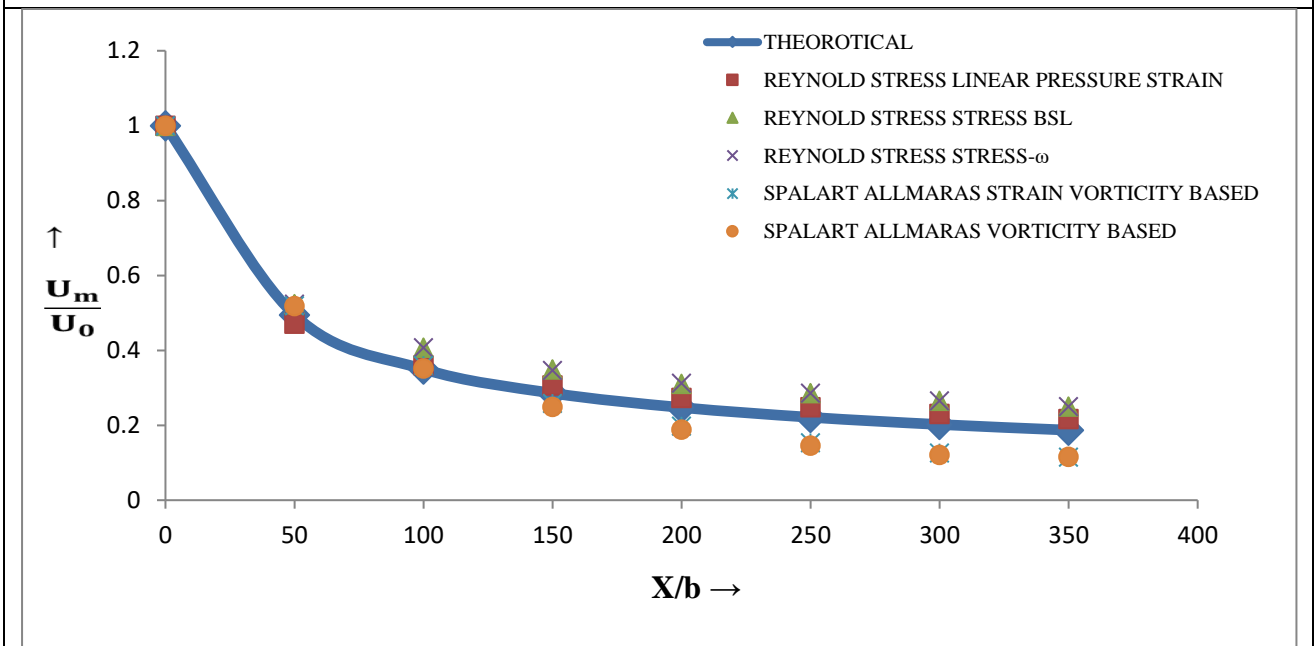


Fig. 8(b) Velocity Decay of Plane Free Jet

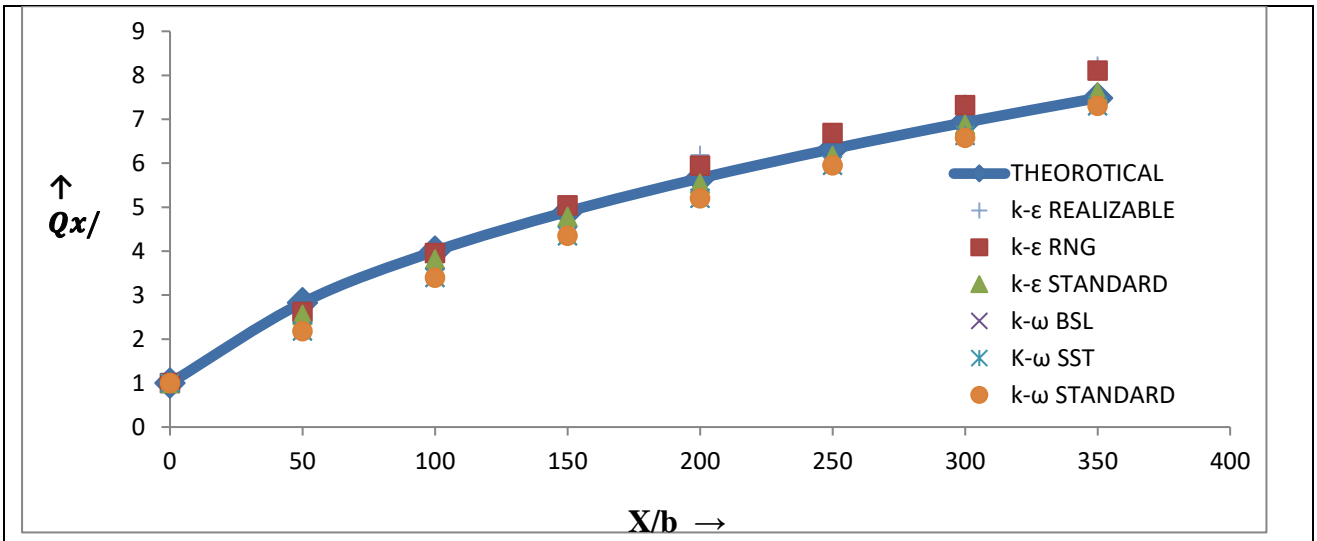


Fig. 9(a) Entrainment Ratio of Plane Free Jet

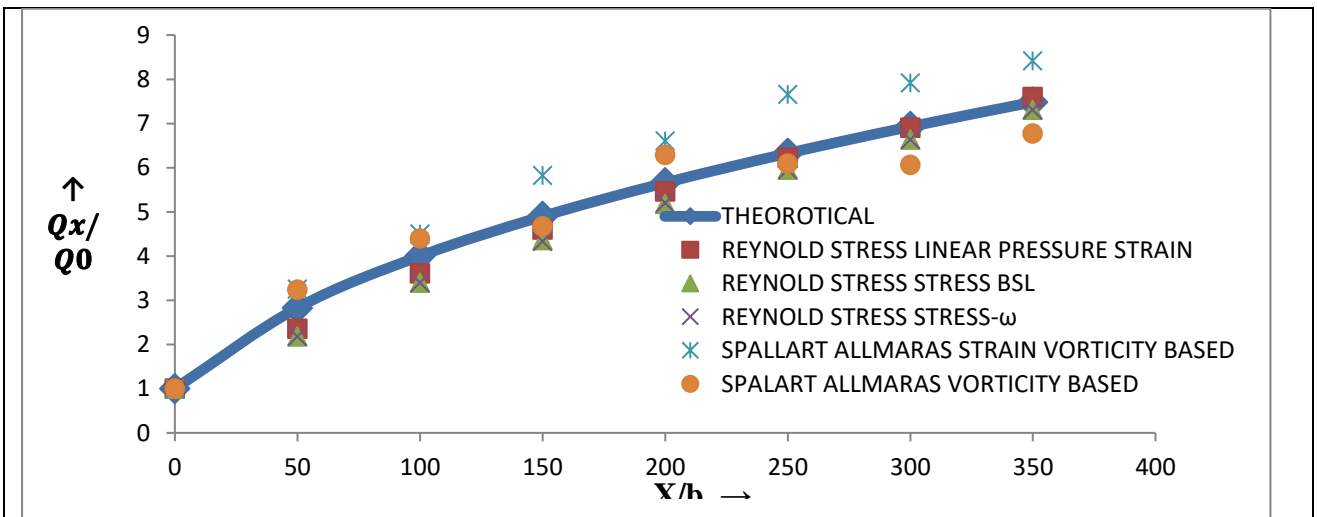


Fig. 9(b) Entrainment Ratio of Plane Free Jet

Conclusion:

Various jet characteristics such as velocity profile, jet spread, velocity decay and entrainment ratio calculated from simulation reasonably matched with the theoretical and experimental solution, CFD simulation was useful in understanding the jet flow characteristics. Among the entire type turbulence model, all varieties of k-ε model performed the best. Performance of k-ε RNG model was found to be somewhat superior compared to the other model.

References:

1. Feng Gao, Haidong Wang and Hui Wang, (2017) " Comparison of different turbulence models in simulating unsteady flow." 10th International Symposium on Heating, Ventilation and Air Conditioning, Science Direct, ELSEVIER, Vol. 205, pp: 3970-3977.
2. Law Adrian Wing-Keung and Herlina, (2002), "An experimental study on turbulent circular wall jets." J. of Hydraulic Engineering, ASCE, Vol.128, No.2, pp: 161-174.
3. Liu, Q., Hoff, S.J., Maxwell, G.M. and Bundy, D.S., (1996), "Comparison of three k- ϵ turbulence models for predicting ventilation air jet." J. of Hydraulics Engineering, ASAE, Vol. 39, No. 2 pp: 689-698
4. M. Nasimul Chowdhury, Abdul A. Khan and Firat Y. Testik, (2017), "Numerical investigation of circular turbulent jets in shallow water." J. of Hydraulics Engineering, ASCE, Vol.143, No.9.
5. Pani, B. S. and Patil, L. G., (2003), "Discussion of : An Experimental study on turbulent circular wall jets by Adrian Wing-Keung Law and Herlina." J. of Hydraulic Engineering, ASCE, Vol.129, No.9, pp: 738-740.
6. Patankar, S. V., (1980), "Numerical heat transfer and fluid flow." Hemisphere Publishing Corporation. New York.
7. Rajaratnam N. and Pani B. S., (1974), "Three dimensional turbulent wall jet." J. of Hydraulics Engineering, ASCE, Vol.100, No.1, pp: 69-83.
8. Rajaratnam N., (1976), "Turbulent jets." Elsevier Scientific Company, New York.
9. Schwarz, W.H. and Cosart, W.P., (1960), "The two dimension wall jet" J. of Fluid Mech., Vol. 10, pp: 481-495.
10. Sfroza P.M., and Herbst, G., (1967), "A study of three-dimensional incompressible turbulent wall jets." J. A.I.A.A., Vol.8, No.2, pp: 276-283.
11. Wilcox, D. D., (2006), "Turbulence modeling for CFD." DCW Industries, Inc., La Canada, California.