EVALUATION OF K-EPSILON MODEL FOR TURBULENT BUOYANT JET

Osman A.B, Mark Ovinis

Mechanical Engineering Department, UniversitiTeknologi PETRONAS

Email: anotood@yahoo.com

ABSTRACT

The modelling of a turbulent buoyant jet is challenging due to the complex nature of such flow, which consists of two fluids with different densities, as well as the multi-scale flow phenomena associated in both space and time. In this paper, the k-epsilon turbulence model is applied to model a turbulent buoyant jet at different flow regimes including laminar and turbulent. The velocity field and centerline velocity are in good agreement with the experiments, as well as the expected results based on jet theory. Moreover, the distribution of the radial velocity matches with Gaussian distribution. The k-epsilon model is an appropriate turbulent model that can be applied for larger Reynolds number flow simulation.

Keywords: *k-epsilon, turbulence model, CFD, turbulent buoyant jet*

INTRODUCTION

The Deepwater Horizon oil spill caused significant environmental, economic, as well as political problems due to a large amount of oil spilled into the marine environment. The type of flow from a well blowout is called "turbulent buoyant jet", as the spilled oil had less density as compared to seawater, and will be abbreviated as "jet flow" throughout this manuscript.

Numerical simulation has become an essential tool for solving engineering problems, and it has extensive applications. By using numerical simulation, complex flow dynamics can be solved. Numerical simulation has been applied to quantify the amount of oil spill in the Gulf of Mexico incident [1]. Several models were applied for fluid flow simulation including direct numerical simulation (DNS) [2], Large Eddy Simulation (LES) [3], Reynolds Average Navier-Stokes (RANS) [4] as well as Detached Eddy Simulation (DES) [5]. Each of these models has advantages and limitations. The choice between these models is usually based on two factors, namely the accuracy of the model and the computational time required. One can obtain better results with the same model, but with high computational time. Wang et al. [6], and Muppidi et al. [7] were simulated a jet flow by using DNS mode. In their work, DNS solved the flow problem with good accuracy as compared to the other existing turbulent models. This is because of the DNS model solves fluid flow problems based on Navier-Stokes equations. However, DNS to solve the flow dynamics problem with high computational time. This is due to the complex solutions required for Navier-Stoke equations. LES model has a degree of advantage of less computational time as compared to DNS. This is because of the LES ignores the smallest scales of turbulent flow in the simulation process. However, with ignoring the small scales, LES is an appropriate model only for dealing with fluid that has large-scale. Flow such as oil jet is associated with both small and large scale motion.

Several researchers applied LES for fluid flow simulation and investigated the jet flow characteristics in which only large scale was required. Akselvoll et al. [8] investigated a type of flow called confined turbulent jet, where Kannan et al. [9] were studied an axisymmetric jet flow. In their works, the LES was limited to resolve the jet flow in near-wall regions. This is because in this region the turbulent length is less than the maximum size of the grid that used for simulation. One possible solution to overcome the limitation of LES is by combining the model with RANS model. By applying RANS model the small scales can be considered. By combining LES and RANS both small and large scales will be included in the final simulation solution. The hybrid model (i.e. LES-RANS) was named also by DES as stated by Spalart et.al. [10]. RANS solve the fluid flow dynamics problem based on averaged forms of Navier-Stokes equations which called Reynolds equations, and the model can be described as a time-averaged model.

K-epsilon models have extensive applications in fluid dynamics. Three forms of K-epsilon model were applied for flow simulation, including standard k-epsilon, realizable k-epsilon, and RNG k-epsilon model [11]. The standard k-epsilon model has the advantage of fast convergence rate, low memory requirements, as well as it has higher accuracy as compared to the other forms [12]. This is because of the standard k-epsilon simulate the fluid by turbulent viscosity estimation based on a linear turbulence scale. Yakhot et al. [13] developed the RNG model which solve the mathematical methods of Reynolds Normalization Group (RNG). It has the advantage of considering various scales of the flow. Shih et al. [14] proposed a realizable k-epsilon model as an improvement of the standard k-epsilon model.

The realizable k-epsilon model is based on new formulations of turbulent viscosity and transport equations of dissipation rate, which differ from the standard model. The later model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Kannan et al. [15] simulated an axisymmetric turbulent jet flow using various turbulence models to compare their accuracy in the prediction of turbulent jet flow. Two groups of models were used, including first-order models (i.e. standard k-epsilon model, standard k-omega model, RNG k-epsilon, realizable k-epsilon model, SST k-omega model), and a second-order model (i.e. Reynolds stress model). In their work, several parameters were investigated, including decay of centerline velocity, turbulence intensity, kinetic energy, and streamlines. The outcomes of this simulation were compared to the available experimental data. In all cases, the first-order models accurately predicted the jet flow as compared to the results of the second-order model with large variations. Aziz et al. [11] were simulated a turbulent jet flow in order to predict jet centerline velocity, radial velocity, growth rate, and turbulent kinematic energy. They investigated both round and plane turbulent jet flow, in which the three forms of k-epsilon models were applied. The outcomes of the simulation were compared with the prediction of the jet theory. They concluded that the k-epsilon model with standard coefficients outperformed the others for both round and plane turbulent jets flow characteristics. In this work, a turbulent jet flow was numerically simulated considering five cases of jet flow rates. The outcomes of numerical simulations are compared to results of an experimental works as well as the expected results of jet flow theory.

METHODS

This section describes the overall methodology for simulation process of turbulent jet flow using k-epsilon model. Several steps are required including the creation of jet geometry, mesh generation, set of boundary conditions, and define the fluid model solver to simulate the turbulent jet flow using CFD. Then by running the simulation, the final results can be obtained and analyzed.

Jet Geometry

The first step required for the simulation of jet flow is the creation of jet geometry. It is because it will define the fluid flow domain including the available volume and the shape of the boundary. The jet geometry shown in Figure 1 shows the jet geometry with a size of 900 x 900 x 1000 mm. The jet nozzle at the upper surface has a diameter of 10 mm. The jet exists (in green) is immersed in the water with a height of 20 mm to avoid the negative effect of surface tension on the jet flow behaviour.

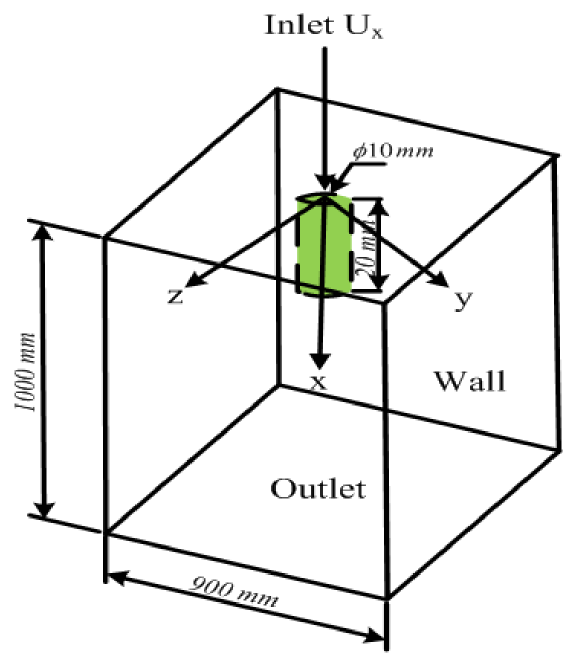

Figure 1 Geometry of jet flow.

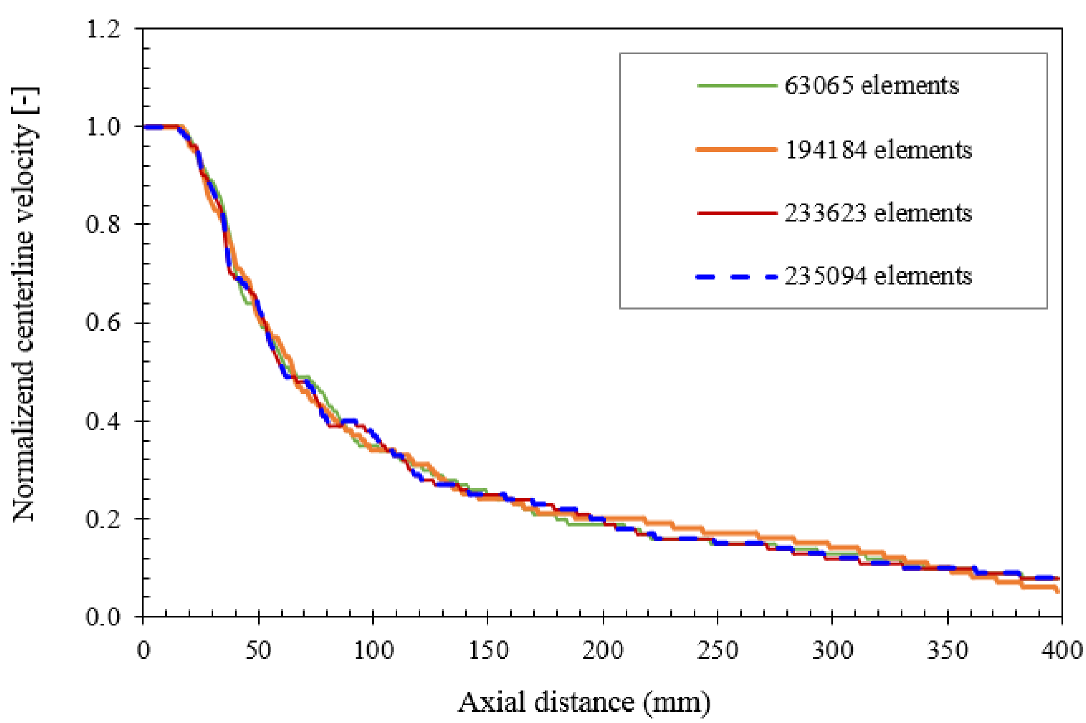

Figure 2 Normalized centerline velocity as a function of axial distance for four different mesh densities.

Mesh Generation

Mesh quality is an important factor for the numerical simulation of any fluid. The accuracy of CFD model is based on two factors: the mesh grid and the number of iterations used in the solution step. Selection of meshing technique is based on several factors includes the desired solution accuracy, available memory, size and shape of the geometry, quality of the starting surface mesh and the simulation topology. Different structures are available for geometry meshing which include triangular, or quadrilateral for 2D geometry as introduced by Versteeg et al. [16]. In this work, a tetrahedral meshing technique is used, because it is more appropriate for meshing a 3D geometry. This mesh-type has the advantage of generating high mesh quality boundary layer by creating structured grids. This is to capture the fast changes in jet flow.

One of the important factors that affect the accuracy of the numerical simulation of jet flow is the meshing structure. Finding the optimum number of mesh elements is essential to help in ensuring better results and to reduce the computational time. This is because the selection of more elements usually requires a high-performance computer with higher RAM as the computational time increases with increasing the number of elements. To quantify the effect of mesh in jet flow simulation, the centerline velocity was used as bases for comparison. Figure 2 shows the normalized centerline velocity extracted from the obtained velocity field for different mesh densities. By increasing the mesh densities, the velocity profiles become closer and insensitive to the mesh size, suggesting that the mesh resolution is adequate. Therefore, a mesh has 235,094 elements used for the geometry meshing of turbulent jet flow. This mesh size was applied for all the cases of simulation runs.

Boundary Conditions

The boundary conditions used for jet flow simulation were taken to provide the buoyant condition. A multi-phase fluid includes water and mixed-fluid were used to simulate the jet flow. The density of mixed fluid was changed by adding 5% of salt to the tap water, as done by Crone et al. [17]. Then, the jet flow was simulated by changing the nozzle velocity considering the same ranges of nozzle flow rates measured from experimental work. Table 1 summarized the boundary conditions considered in the numerical simulation of the jet flow.

Jet Flow Modeling

CFD - Fluent solver includes several fluid models usually used for modelling the fluid flow. However, based on the previous investigation on common models used for simulating turbulent jet, it was found out that the standard k-epsilon model [11] outperforms the others when tested with higher Reynolds number turbulent jet flow. K-epsilon model simulates turbulent flow with considering two variables namely as turbulent kinetic energy, and dissipation rate of kinetic energy. The standard k- epsilon model produced the best result when used for simulating turbulent jet flow [7]. The standard values for the k-epsilon model were used for jet flow simulation. Reynolds-Averaged Navier-stokes can be formulated by:

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} \frac{\partial}{\partial x_j}\left[v \frac{\partial U_i}{\partial x_j} - \overline{u_i u_j} \right] \quad (2)$$

Table 1 Summary of boundary conditions used for numerical simulation of jet flow

No.	Description	Boundary condition
1	Jet exist velocity	Five cases includes (0.18, 0.32, 0.45, 0.62 and 1.16 m/sec)
2	Jet exist in diameter	10 mm
3	Wall and output	Atmospheric pressure
4	Density of water	1000 kg/m3
5	The density of mixed water (5% salt)	1050 kg/m3

where *i, j* are indices, x_i is the coordinate in which i = 1,2,3, *Ui, Uj* is time-averaged velocity components, t represents time (sec), ρ is the fluid density, P is the piezometric pressure, v is the kinematic viscosity of the fluid, and $\overline{Ui, Uj}$ the turbulent normal and shear stresses.

The *k-epsilon* model is based on turbulent eddy viscosity to relate the normal-shear stresses to the time-averaged velocity gradients and turbulent kinetic energy is given by:

$$\overline{u_i u_j} = \upsilon_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \quad (3)$$

where v_t is the turbulent eddy viscosity, δ_{tj} is the Kronecker delta, and k= 0.5 *Ui, Uj* is the turbulent kinetic energy per unit mass, and, the turbulent eddy viscosity is given by:

$$\upsilon_t = c_\mu \frac{k^2}{\varepsilon} \quad (4)$$

where c_μ is an empirical coefficient and *k-epsilon* is the dissipation rate of turbulent kinetic energy.

The k and ε can be formulated by:

$$\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\frac{\upsilon_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + \upsilon_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} - \varepsilon \quad (5)$$

$$\frac{\partial \varepsilon}{\partial t} + U_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\frac{\upsilon_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right) + c_{1\varepsilon} \frac{\varepsilon}{k} \upsilon_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} - c_{2\varepsilon} \frac{\varepsilon^2}{k} \quad (6)$$

where c_μ, $c_{1\varepsilon}, c_{2\varepsilon}$ & $\sigma_k, \sigma_\varepsilon$ are empirical coefficients, and the standard values for these coefficients used in k - ε the model are equal to 0.09, 1.44, 1.92, 1.0, and 1.3 respectively.

To solve these models a pressure-based solver is used for the numerical simulation of jet flow. This because of the flexibility in the solution procedure and it requires less memory. The pressure-based solver is based on combining both velocity and pressure to solve the continuity and momentum equations in order to derive an equation for pressure correction. An algorithm called Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) is used for solving these equations [16].

RESULTS AND DISCUSSION

Velocity Field

Figure 3 shows the velocity fields obtained by jet simulation by considering five different jets exist flow rates. Based on visual observation, the obtained velocity fields were in a good agreement with the predicted fields of jet theory. A clear jet core region is observed for all cases with the maximum velocity being observed of the pure jet region. In the axial direction of the jet, a decay of velocity was observed. The distance of the velocity propagation is mainly based on the initial nozzle velocity, whereby increasing the nozzle velocity, the distance increases. In the radial direction, a similarity in the velocities at left and right side of the jet was obtained with not many changes in the jet angle as the nozzle velocity increases. This suggests that the k-epsilon model is able to simulate the jet flow as expected from the jet theory.

To validate the outcomes of the k-epsilon model, Figure 4 shows outcomes of our previous experimental work in which the image velocity fields were estimated using a technique called wavelet-based optical velocimetry (WOV) [18]. This is for the different cases of nozzle flow rates considered in this study. The obtained velocity field scaled from zero up to the actual nozzle velocity. The maximum velocity was observed near the nozzle region while the jet velocities decayed far from the nozzle, as expected from jet flow theory. Variation of velocities in the pure jet region (i.e. red color) could be due to the variation in images used or some biases associated with the WOV algorithm.

Centerline Velocity

Figure 5 shows the inverse of centerline velocity extracted from the velocity field obtained by CFD simulation for the five cases of nozzle velocity.

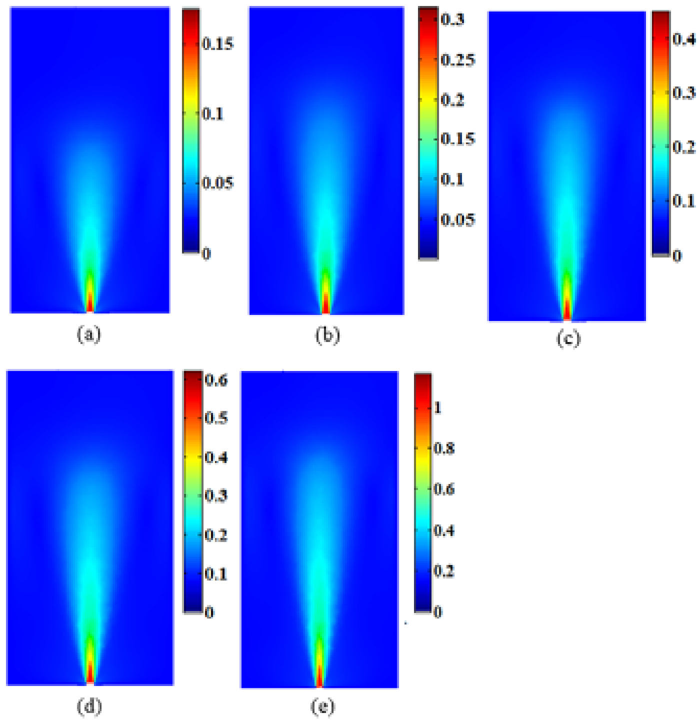

Figure 3 Velocity field simulated using CFD simulation at different jet exist velocity: (a) *U1 = 0.18 m/s*, (b) *U2 = 0.32 m/s,* (c) *U3 = 0.45 m/s*, (d) *U4 = 0.62 m/s* and (e) *U5 = 1.16 m/s*.

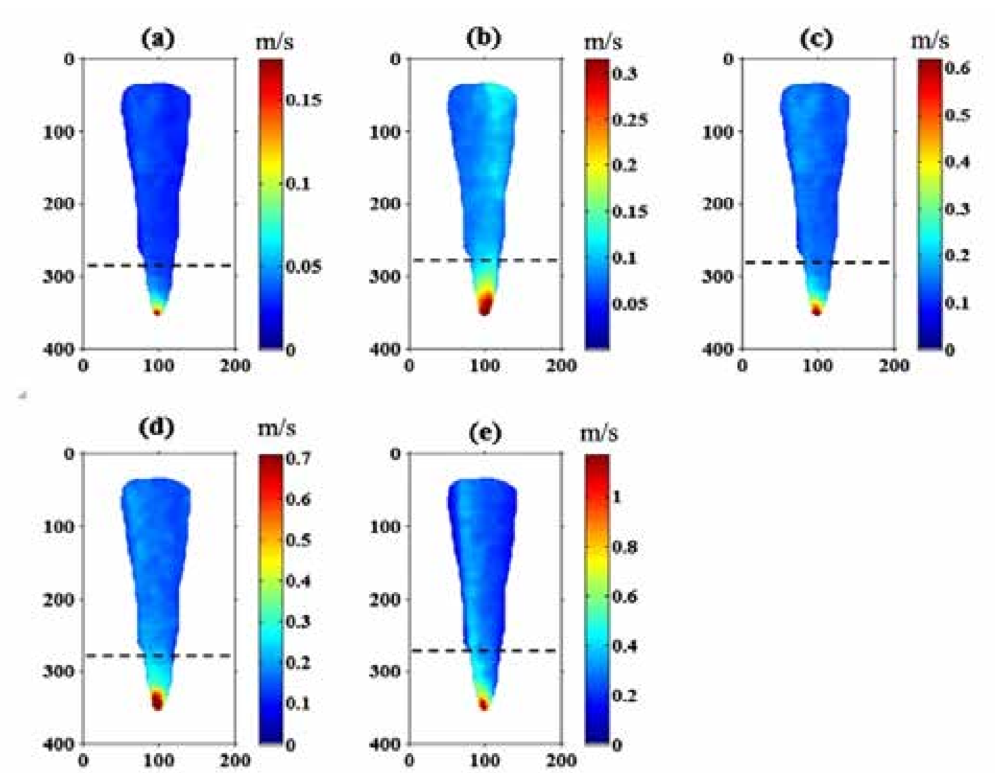

Figure 4 Velocity field estimated by WOV technique, for cases of (a) *U1 = 0.18 m/s*, (b) *U2 = 0.32 m/s*, (c) *U3 = 0.45 m/s*, (d) *U4 = 0.62 m/s* and (e) *U5 = 1.16 m/s* [18].

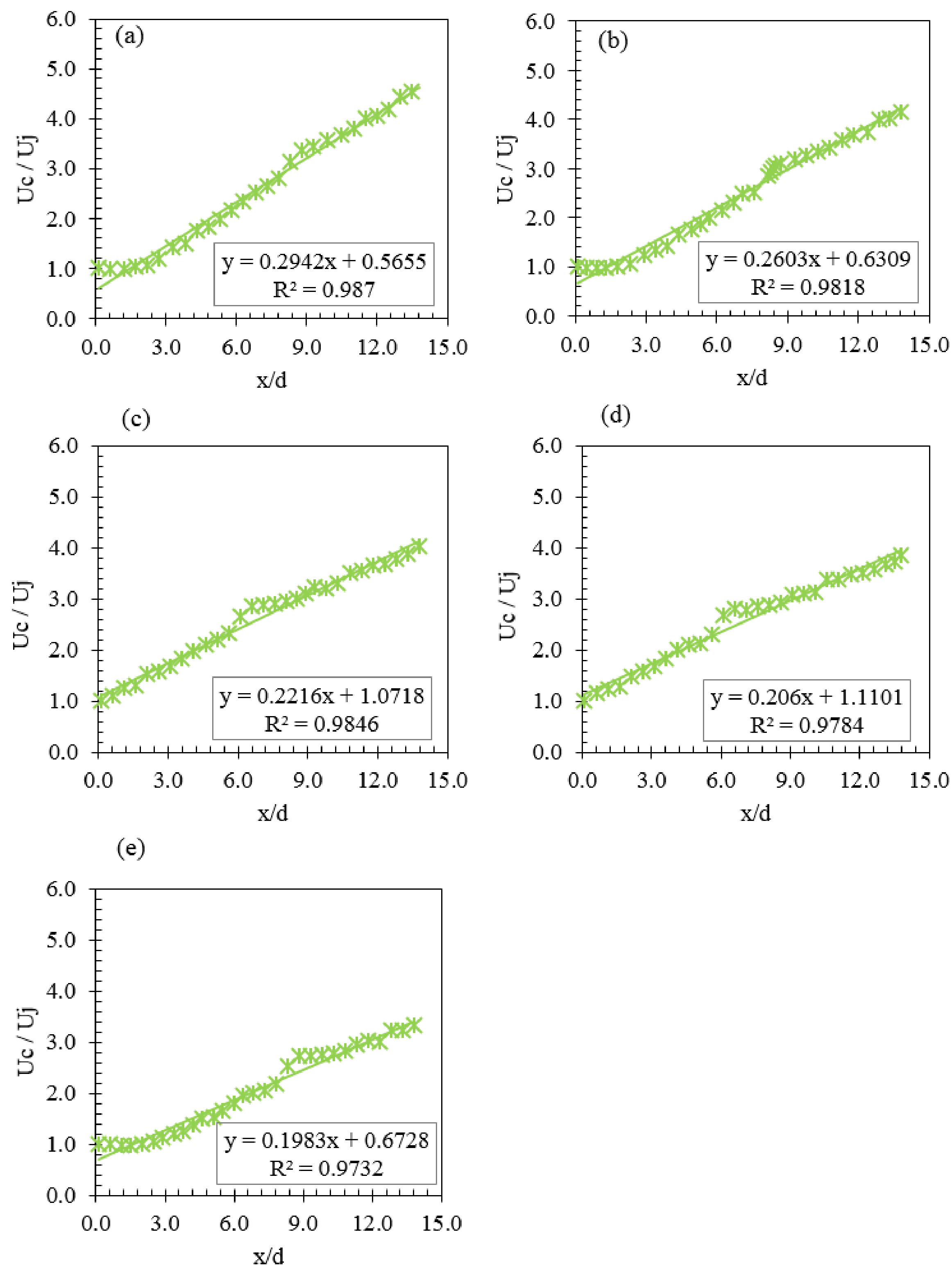

Figure 5 Decay of centerline velocity of CFD simulation for different nozzle velocity including (a) *U1 = 0.18 m/s,* (b) *U2 = 0.32 m/s,* (c) *U3 = 0.45 m/s,* (d) *U4 = 0.62 m/s* and (e) *U5 = 1.16 m/s.*

For all cases, the actual nozzle velocity was divided over the velocity along the axial distance to determine the growth rate of the turbulent jet.

Strong linear relationships were observed from which the growth rates were obtained. The growth rates are 3.4, 3.8, 4.5, 4.9 and 5.04 for the jet exist flow rate cases of Q1, Q2, Q3, Q4 and Q5 respectively. By increasing the nozzle flow rate, the jet growth rate was increased.

Radial Velocity

Figure 6 shows the normalized radial velocity at x/d = 15, 20 and 25 for the first case of nozzle flow rate as the sample. The normalization of the velocity distribution at a different location by the centerline velocity should provide a self-similarity property with Gaussian distribution profiles. A good agreement with the predicted Gaussian profile was obtained with a very small difference.

CONCLUSION

A turbulent buoyant jet flow is simulated numerically using CFD-Fluent in which five cases of jet exist velocity were considered. The outcomes of numerical simulation are in a good agreement with the experimental works as well as the expectation from jet theory. The velocity field and centerline velocity profile showed that the jet flow was high in the near-exist region, while the velocity decreases far from jet exist. The distribution of radial velocity showed a good agreement with the Gaussian profile expected from jet theory. Therefore, the k-epsilon model is a good turbulent flow model that can be applied to investigate higher Reynold's number of cases.

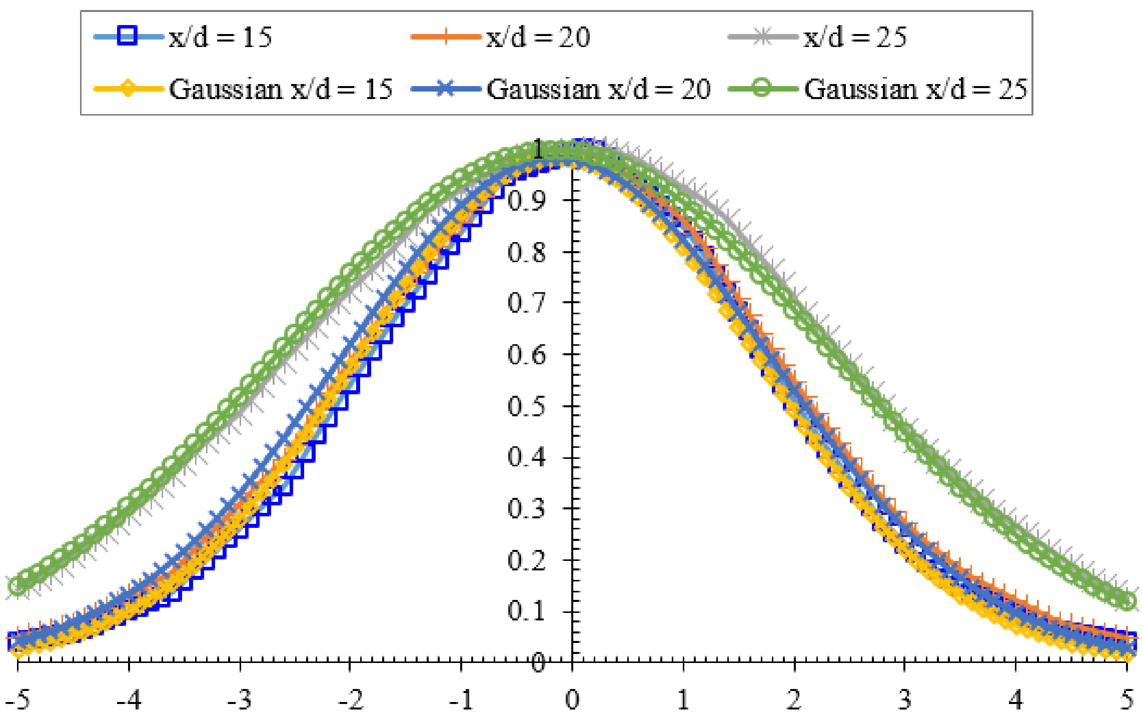

Figure 6 Normalized radial velocity profiles at a various axial location for the case of nozzle velocity *U1 = 1.16 m/sec.*

ACKNOWLEDGEMENTS

The authors would like to express their appreciation to Universiti Teknologi PETRONAS for supporting this work under YUTP 0153AA-E85.

REFERENCES

[1] B. Lehr, A. Aliseda, P. Bommer, P. Espina, O. Flores, J. Lasheras, et al., "Deepwater horizon release estimate of rate by PIV" *Report to the US Dept of interior*, 2010.

[2] I. Z. Naqavi, J. C. Tyacke, and P. G. Tucker, "Direct numerical simulation of a wall jet: flow physics" *Journal of Fluid Mechanics,* vol. 852, pp. 507-542, 2018.

[3] B. Kong, T. Li, and Q. Eri, "Large eddy simulation of turbulent jet controlled by two pulsed jets: Effect of forcing frequency" *Aerospace Science and Technology,* vol. 89, pp. 356-369, 2019.

[4] A. A. Mishra and G. Iaccarino, "Uncertainty estimation for reynolds-averaged navier–stokes predictions of high-speed aircraft nozzle jets" *AIAA Journal,* pp. 3999-4004, 2017.

[5] J. Verrière, F. Gand, and S. Deck, "Zonal Detached Eddy Simulations of a Dual-Stream Jet: Turbulence Rate Sensitivity" *AIAA Journal,* pp. 2503-2521, 2017.

[6] Z. Wang, P. He, Y. Lv, J. Zhou, J. Fan, and K. Cen, "Direct numerical simulation of subsonic round turbulent jet" Flow, *turbulence and combustion,* vol. 84, pp. 669-686, 2010.

[7] S. Muppidi and K. Mahesh, "Direct numerical simulation of turbulent jets in crossflow" in 43rd *AIAA Aerospace Sciences Meeting and Exhibit,* 2005, p. 1115.

[8] K. Akselvoll and P. Moin, "Large-eddy simulation of turbulent confined coannular jets" *Journal of Fluid Mechanics,* vol. 315, pp. 387-411, 1996.

[9] B. T. Kannan, S. Karthikeyan, and S. Sundararaj, "Comparison of Turbulence Models in Simulating Axisymmetric Jet Flow" in *Innovative Design and Development Practices in Aerospace and Automotive Engineering*: I-DAD, February 22 - 24, 2016, R. P. Bajpai and U. Chandrasekhar, Eds., ed Singapore: Springer Singapore, 2017, pp. 401-407.

[10] P. Spalart, W. Jou, M. Strelets, and S. Allmaras, "Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach" *Advances in DNS/LES*, vol. 1, pp. 4-8, 1997.

[11] T. Aziz, J. Raiford, and A. Khan, "Numerical simulation of turbulent jets" *Engineering applications of computational fluid mechanics*, vol. 2, pp. 234-243, 2008.

[12] J. SORBE, "An experimental study in the near field of a turbulent round free jet" ed: Msc thesis, Univeristy of GAVLE, 2014.

[13] V. Yakhot, S. Orszag, S. Thangam, T. Gatski, and C. Speziale, "Development of turbulence models for shear flows by a double expansion technique" *Physics of Fluids A: Fluid Dynamics,* vol. 4, pp. 1510-1520, 1992.

[14] T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu" A new k-ε eddy viscosity model for high reynolds number turbulent flows" *Computers & Fluids,* vol. 24, pp. 227-238, 1995.

[15] B. Kannan, S. Karthikeyan, and S. Sundararaj", Comparison of Turbulence Models in Simulating Axisymmetric Jet Flow", *Innovative Design and Development Practices in Aerospace and Automotive Engineering*, ed: Springer, 2017, pp. 401-407.

[16] H. K. Versteeg and W. Malalasekera, "An introduction to computational fluid dynamics: the finite volume method", *Pearson Education,* 2007.

[17] T. J. Crone, R. E. McDuff, and W. S. Wilcock, "Optical plume velocimetry: A new flow measurement technique for use in seafloor hydrothermal systems", *Experiments in fluids,* vol. 45, pp. 899-915, 2008.

[18] Osman AB, Mark Ovinis, Hashim FM, and Faye I, "Wavelet-based optical velocimetry for oil spill flow rate estimation", *Measurement,* vol. 138, pp. 485-496, 2019.