NUMERICAL SIMULATION OF TURBULENT JETS

T. N. Aziz, J. P. Raiford and A. A. Khan*

Department of Civil Engineering, Clemson University, 218 Lowry Hall, Clemson, SC 29634-0911, USA

** E-Mail: abdkhan@clemson.edu (Corresponding Author)*

ABSTRACT: In this study, plane and circular turbulent non-buoyant jets are simulated numerically using a three-dimensional computational model. The aim of the study is to evaluate the accuracy of turbulent closure schemes employed in three-dimensional models. In particular, standard $k-\varepsilon$ and renormalized group $k-\varepsilon$ schemes with standard coefficients are evaluated. The modeled jets are deeply submerged, that is the impact of free surface and solid boundaries on jets are eliminated. The accuracy of the turbulent schemes is assessed by analyzing the decay of centerline velocity, jet growth rates, similarity of longitudinal and vertical velocity profiles, and turbulent kinetic energy profiles. The results from the two turbulent closure schemes are compared with accepted experimental and theoretical studies to determine their accuracy. It is found that the $k-\varepsilon$ scheme with standard coefficient performs equally well and in some cases better than the renormalized group $k-\varepsilon$ scheme. Finally, the model is applied to analyze flow pattern in the Sampit River, South Carolina, USA, resulting from stormwater discharge in a recreational area. Various inlet designs are investigated and box inlet is found to provide a practical means of localizing high surface currents.

Keywords: computational modeling, turbulent jets, storm water discharge, turbulent closure schemes

1. INTRODUCTION

Turbulent jets are the primary means through which waste is discharged into the environment. The performance of a jet in these circumstances relies on the orifice geometry, characteristics of the discharged and ambient fluid, and the physical environment into which the jet is discharged. In most cases, the jet interacts with solid boundaries and/or a free surface for which analytical or empirical solutions may not be available. In such cases, either physical model or numerical model studies are utilized. In recent years, numerical models have been increasingly adopted for studying complicated flow scenarios. In modeling turbulent flows, one of the key elements is the choice of a turbulent closure scheme. Two of the most popular turbulence closure schemes are the $k-\varepsilon$ and the Renormalized Group $k-\varepsilon$ (RNG). While both these models can provide accurate results, their accuracy depends on empirical coefficients which must be adjusted to calibrate these schemes. For example, the standard coefficients of the $k-\varepsilon$ scheme were determined by computer optimization of laboratory shear flows (ASCE Task Committee on Turbulence Models in Hydraulic Computations, 1988). The standard coefficients determined for the turbulent closure schemes are not universal.

Corrections to these coefficients have been established for different flows to achieve better agreement with the laboratory or analytical results; for example, the standard coefficients are modified to simulate circular turbulent jets and achieve better accuracy with established results (ASCE Task Committee on Turbulence Models in Hydraulic Computations, 1988). In other cases, where the jets may interact with boundaries, surface, or other flows, the appropriate modifications may not be available and standard values of these coefficients may have to be used.

2. MATHEMATICAL DETAILS

The $k-\varepsilon$ and RNG schemes provide a turbulent closure scheme for the Reynolds-Averaged Navier-Stokes (RANS) equations, which are given by

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

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

where i and j are indices, x_i represents coordinate directions ($i = 1$ to 3 for x, y, z directions, respectively), U_i is the time averaged velocity component, t represents time, ρ is the fluid density,

P is the piezometric pressure, ν is the kinematic viscosity of the fluid, and $\overline{u_i u_j}$ are turbulent normal and shear stresses.

Both models use the assumption of turbulent eddy viscosity to relate turbulent normal and shear stresses to time averaged velocity gradients and turbulent kinetic energy. The relationship is given by

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

where ν_t is the turbulent eddy viscosity, δ_{ij} is the Kronecker delta, and $k (= 0.5 \overline{u_i u_i})$ is the turbulent kinetic energy per unit mass. The turbulent eddy viscosity is computed as follows:

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

where c_μ is an empirical coefficient and ε is the dissipation rate of turbulent kinetic energy. The k - ε and RNG turbulent closure schemes use the above equation to determine turbulent eddy viscosity that is used to relate the turbulent shear and normal stresses to the time averaged velocity gradients.

The k - ε scheme consists of the following two equations to determine k and ε .

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

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

In the above equations, σ_k , σ_ε , $c_{1\varepsilon}$, and $c_{2\varepsilon}$ are empirical coefficients. The first two terms on the left hand side of the k and ε equations represent rate of change and advection of the respective quantities. The first term on the right hand side represents diffusion in both cases. The two remaining terms represent the generation and destruction of stress and energy, respectively. The standard values of the coefficients c_μ , $c_{1\varepsilon}$, $c_{2\varepsilon}$, σ_k , and σ_ε used in the k - ε scheme are 0.09, 1.44, 1.92, 1.0, and 1.3, respectively.

The RNG scheme takes advantage of the broad range of length scales encompassed by turbulent eddies. Energy in the large scale cascades down to the smaller scales where it is dissipated by

viscosity. At some point in this scale, the energy dissipation is equal to the energy production. This scale, referred to as the inertial scale, can be used to describe all the scales. Applying the renormalization technique produces a model that is statistically equivalent to the original Navier-Stokes equations but only describes the inertial scale of turbulence. This scale is of an order that can be efficiently handled by current computer technology. The RNG scheme uses different k and ε equations. These equations as given by Bischof, Bucker and Rasch (2004) are

$$\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\frac{\nu_{eff}}{P_{rk}} \right) \frac{\partial k}{\partial x_i} \right] + \nu_t S^2 - \varepsilon \quad (7)$$

$$\frac{\partial \varepsilon}{\partial t} + U_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\frac{\nu_{eff}}{P_{r\varepsilon}} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + c_{1\varepsilon} \frac{\varepsilon}{k} \nu_t S^2 - c_{2\varepsilon} \frac{\varepsilon^2}{k} - R \quad (8)$$

where P_{rk} and $P_{r\varepsilon}$ are Prandtl numbers for k and ε respectively, S represents the mean rate of strain, and ν_{eff} is given by

$$\nu_{eff} = \nu_t \left(1 + \sqrt{\frac{\nu}{\nu_t}} \right)^2 \quad (9)$$

The term R in the RNG scheme is one of the main differences between the RNG and k - ε schemes. The R term is a source term that is a function of S , k , and ε . This term is given by

$$R = \frac{c_\mu \eta^3 (1 - \eta/\eta_o) \varepsilon^2}{1 + \beta \eta^3 k} \quad (10)$$

where η_o and β are constants having standard values of 4.38 and 0.012, respectively, and $\eta = Sk/\varepsilon$. The standard values of the constants c_μ , $c_{1\varepsilon}$, and $c_{2\varepsilon}$ used in the RNG schemes are 0.0845, 1.42, and 1.68, respectively.

Both the k - ε and RNG schemes have been used extensively with the RNG model used for high shear and high Reynolds number flows. A sensitivity analysis of both schemes showed that the k - ε scheme is less sensitive to c_μ than the RNG scheme. It was further found that the k - ε scheme is more sensitive to $c_{1\varepsilon}$ and $c_{2\varepsilon}$ than the RNG scheme. The RNG scheme is found to be more sensitive to the rate of strain because of the presence of the source term R (Bischof, Bucker and Rasch, 2004).

3. NUMERICAL MODEL AND SETUP

In order to solve the RANS equations using the $k-\epsilon$ and RNG turbulent closure schemes, a CFD code called FLOW-3D, developed by Flow Science, Inc., is used. The code solves the RANS equations using a finite-volume/finite-differences method in an Eulerian rectangular grid. The code allows the user to choose from a zero-, one-, or two-equation turbulent closure schemes. The code also allows for a determination of boundaries independently from grid generation avoiding “saw-tooth” representations of boundaries (Rodriguez et al., 2003).

The geometries used were a circular orifice of 5 mm in diameter for the circular jet and a slot width of 5 mm for the plane turbulent jet. These jets flowed into a simulated tank of 580 cm on a side. The tank was large enough in extent so that the jets could be considered free from the effects of boundaries and water surface. A uniform velocity of 200 cm/s was applied across the nozzle for the two jets. At tank boundaries, the normal and tangential components of velocities were set to zero. At the outflow section, a continuative boundary condition was applied, which forced the normal derivatives of all the variables at the boundary to zero. At the wall, a smooth boundary was assumed by specifying a roughness height of zero. Initial conditions of zero velocity and hydrostatic pressure distribution were assumed for the inside of the tank. To compare the two turbulent closure schemes, the computational mesh as well as initial and boundary conditions for the two simulations, each of circular turbulent and plane turbulent jets, were exactly the same. However, the meshes for the circular turbulent and plane turbulent jets differed in order to appropriately capture the nozzle geometry. The computed results, at the center of each cell, were velocity components in x , y , and z directions, pressure, turbulent kinetic energy per unit mass, and turbulent dissipation rate.

4. TURBULENT JETS

Details of the circular turbulent or plane turbulent jet are shown in Fig. 1. For the plane turbulent jet, $2b_o$ represents the width of the nozzle; d is the diameter of the nozzle for the circular turbulent jet. The initial uniform velocity of the jet coming out of the nozzle is given by u_o . The virtual origin is at a distance of $\bar{x}$ from the nozzle. The centerline

velocity at any position x (distance along the jet) is given by u_m . The velocity varies from the centerline value, u_m , to zero at the edge of the jet. For linear scale and growth of jets, length b is commonly used. Distance b is measured along y or r (for plane turbulent or circular turbulent jet, respectively) coordinate direction to a point where $u = 0.5u_m$.

Fig. 1 Definition sketch of free turbulent jets.

The growth rates of the free plane and circular jets are given by

$$b/b_o = A_1 (x + \bar{x})/b_o \quad (11a)$$

$$b/d = A_2 (x + \bar{x})/d \quad (11b)$$

where A_1 and A_2 are coefficients for plane and circular turbulent jets, respectively. The value of 0.097 for A_1 was found to fit the experimental data accurately (Rajaratnam, 1976). Abramovich (1963) recommended a value of 0.097 for A_2 . The virtual origin was found to range from 0 to $2.4b_o$ behind the nozzle for plane turbulent jets, and $0.6d$ to $2.2d$ behind the nozzle for circular jets (Rajaratnam, 1976).

The decay of the centerline velocities for the plane and circular turbulent jets are given by the following equations, respectively.

$$\frac{u_m}{u_o} = \frac{A_3}{\sqrt{x/b_o + \alpha_1}} \quad (12a)$$

$$\frac{u_m}{u_o} = \frac{A_4}{x/d + \alpha_2} \quad (12b)$$

where A_3 and A_4 are given by 3.5 and 6.3, respectively. The values of α_1 and α_2 represent correction for virtual origin. The velocity profiles across the jet are found to be similar and can be approximated by a Gaussian curve of the form (Rajaratnam, 1976):

$$\frac{u}{u_m} = \exp(-0.693\lambda^2) \quad (13)$$

where λ is given by y/b or r/b for plane or circular turbulent jet, respectively.

5. SIMULATION RESULTS FOR PLANE TURBULENT JET

For the plane jet, growth rate, decay of the centerline longitudinal velocity, longitudinal and vertical velocity profiles across the jet, and turbulent kinetic energy profile are compared with experimental data and accepted empirical equations. The growth rates of the plane turbulent jet based on $k-\epsilon$ and RNG schemes are found to be 0.11 and 0.12, respectively, and compare well with the value given by Eq. (11a). The virtual origin for $k-\epsilon$ and RNG schemes are found to be $5.96b_o$ and $1.32b_o$, respectively.

Fig. 2 Centerline velocity decay of plane jet ($k-\epsilon$ scheme).

Fig. 3 Centerline velocity decay of plane jet (RNG scheme).

Figs. 2 and 3 show the decay of the centerline longitudinal velocity along the jet using $k-\epsilon$ and RNG schemes, respectively. For reference, the

decay of the centerline velocity given by Eq. (12a) is also provided in these figures. Although both schemes predict the decay of the centerline longitudinal velocity satisfactorily, the $k-\epsilon$ scheme does a better job in estimating the decay. The RNG scheme predicts lower centerline velocity immediately after the potential core and in the lower half of the jet.

The similarity of longitudinal velocity profiles across the jet at different locations is tested by comparing the predicted velocity profiles with Eq. (13). The velocity profiles obtained using the $k-\epsilon$ and RNG schemes are shown in Figs. 4 and 5, respectively, along with Eq. (13). The velocity profile immediately after the potential core is poorly predicted by both schemes, however, more so by the RNG scheme. The velocity profiles further away from the potential core follow the Gaussian curve well.

Fig. 4 Similarity of longitudinal velocity of plane jet ($k-\epsilon$ scheme).

Fig. 5 Similarity of longitudinal velocity of plane jet (RNG scheme).

The vertical velocity profiles across the jet predicted by the $k-\epsilon$ and RNG schemes are compared to Goertler's solution (Abramovich,

1963) in Figs. 6 and 7, respectively. The Goertler's solution is given by

$$\frac{v}{u_m} = \frac{1}{\alpha} \left(\frac{\alpha y}{x} - \frac{\alpha y}{x} \tanh^2 \left(\frac{\alpha y}{x} \right) - 0.5 \tanh \left(\frac{\alpha y}{x} \right) \right) \quad (14)$$

where α has a value of 7.67 and v is the vertical velocity (y -direction). Although both schemes perform well in predicting the vertical velocity profiles, the results from the k - ϵ scheme are slightly better.

Fig. 6 Similarity of vertical velocity of plane jet (k - ϵ scheme).

Fig. 7 Similarity of vertical velocity of plane jet (RNG scheme).

Figs. 8 and 9 show the profiles of kinetic energy per unit mass at $x/2b_0=50$ obtained using k - ϵ and RNG schemes, respectively. These profiles are compared with the physical model data of Heskestad (1965) at the same location. The results from the k - ϵ scheme do not conform with the experimental data near the centerline of the jet, while the RNG scheme predicts a slightly higher k value throughout.

From the above discussion, it is clear that the k - ϵ scheme performs slightly better than the RNG scheme in predicting the growth rate, similarity of longitudinal and vertical velocity profiles, and centerline velocity decay of the plane turbulent jet.

On the other hand, the kinetic energy per unit mass across the plane jet is predicted more accurately by the RNG scheme.

Fig. 8 Profile of turbulent kinetic energy of plane jet (k - ϵ scheme).

Fig. 9 Profile of turbulent kinetic energy of plane jet (RNG scheme).

6. SIMULATION RESULTS FOR CIRCULAR TURBULENT JET

For the circular turbulent jet, the same set of comparisons, as described for the plane turbulent jet, are conducted. The growth rates for the circular turbulent jet using k - ϵ and RNG schemes are 0.1 and 0.14, respectively. The corresponding virtual origins are located at $5.56d$ and $0.86d$. The growth rate based on the RNG scheme is higher than the generally accepted value of 0.097, while the k - ϵ turbulent closure scheme predicts the growth rate accurately.

The decay of the centerline longitudinal velocity obtained using k - ϵ and RNG schemes are compared with Eq. (12b) in Figs. 10 and 11, respectively. The RNG scheme predicts lower centerline velocity immediately following the potential core and the trend continues for almost the entire length investigated. The results from the k - ϵ scheme compare well with Eq. (12b).

Fig. 10 Centerline velocity decay of circular jet ($k-\varepsilon$ scheme).

Fig. 11 Centerline velocity decay of circular jet (RNG scheme).

Fig. 12 Similarity of longitudinal velocity of circular jet ($k-\varepsilon$ scheme).

Fig. 13 Similarity of longitudinal velocity of circular jet (RNG scheme).

Eq. (13) is used to test the similarity characteristics of the longitudinal velocity profiles obtained from the two turbulent closure schemes. Figs. 12 and 13 show the velocity profiles from the $k-\varepsilon$ and RNG schemes, respectively. Although the overall agreement of the velocity profiles at different locations is satisfactorily predicted by the two schemes, the RNG scheme shows discrepancy in predicting the velocity profile immediately following the potential core.

The radial velocity profiles at different locations obtained using $k-\varepsilon$ and RNG schemes are compared with the vertical velocity profile given by Abramovich (1963) in Figs. 14 and 15, respectively. The velocity profiles predicted by the two schemes compare well with the theoretical profile.

Fig. 14 Similarity of vertical velocity of circular jet ($k-\varepsilon$ scheme).

Fig. 15 Similarity of vertical velocity of circular jet (RNG scheme).

The kinetic energy per unit mass profiles predicted by the two schemes at $x/d = 61.5$ are compared with the experimental data of Wygnanski and Fiedler (1969). The comparisons for the $k-\varepsilon$ and RNG schemes are shown in Figs. 16 and 17. Both schemes perform poorly in predicting the kinetic energy per unit mass. On average, the $k-\varepsilon$ scheme performs better than the RNG scheme.

Fig. 16 Profile of turbulent kinetic energy of circular jet ($k-\epsilon$ scheme).

Fig. 17 Profile of turbulent kinetic energy of circular jet (RNG scheme).

For the circular turbulent jet, the $k-\epsilon$ scheme clearly performs better in predicting the growth rate and the decay of the centerline longitudinal velocity. Both schemes adequately predict the similarity of longitudinal and radial velocity profiles at different location along the jet. The two schemes, especially the RNG scheme, are unable to predict the kinetic energy per unit mass profile.

7. FIELD APPLICATION

The computational model with $k-\epsilon$ turbulent closure scheme is applied to predict the flow pattern resulting from a storm water discharge in the Sampit River near Georgetown, South Carolina. The scenario represents jet flow in shallow depth environment for which a general solution is not available. The Sampit River reach and the proposed outlet structure with the flow direction are shown in Fig. 18. The river reach is flat with average bottom elevation of 25.5 m. From the outlet structure, in the direction of flow, the opposite bank is about 490 m and the tip of the island is about 130 m. Although the outlet structure is located in a bend, the flow through the bend is negligible with a maximum

surface velocity of less than 0.05 m/s and oscillates back and forth. For numerical simulation, the flow through the river reach is ignored (jet discharging in a still ambient reservoir). The low and high water levels in the river, based on the historical records, are 30.2 m and 31.3 m. Stormwater outflows of 9.7 cms and 5.1 cms are considered for design purposes. The river reach is used extensively for recreational purposes and surface velocity in excess of 0.52 m/s (1 knot) is considered detrimental for recreational use of the river reach. It is required that the zone of surface velocity in excess of 0.52 m/s needs to be delineated with warning signs.

Fig. 18 Layout of the field application.

The aim is to design an outlet structure that will minimize the zone of surface velocity in excess of 0.52 m/s. Several outlet structures are considered, including box culvert (7.62 m wide and 2.1 m high), circular pipe (2.1 m in diameter), Y-pipe consisting of two 2.1-m-diameter pipes that are set 30-degrees from the normal to the shore line and are 24.4 m apart, and rectangular diffuser that consists of 6 slots (0.91 m apart center-to-center; each of which is 2.1 m high and 0.41 m wide).

The center of all outlet structures is located 3.3 m above the bed and 1.4 m below the water surface under low water level condition (the soffit is 0.35 m below the water surface). The jet issuing from the outlet structure is restricted in vertical direction owing to the proximity of water surface and channel bed. Johnston (1985) performed experimental study on jets (restricted by surface and/or bed) issuing from circular nozzles into shallow water and showed that shallow jets produced less dilution than

free jets at the same location. This means that due to limited entrainment and growth in the vertical direction, the velocity decay and the mixing of the ambient fluid with the discharged effluent will be lower than those of free jets. In addition, high velocities are expected to persist for a longer distance compared to a free jet. Thus, the worst condition that an outlet structure design is required to handle will be the low water level and high discharge scenario as it represents a minimum entrainment condition in a vertical direction.

A coarse mesh was used to determine the lateral (y -direction) spread of the jets. The maximum width of the jet was found to be approximately 40 m. A uniform mesh size of 20 cm was used in the vertical (z) direction because of shallow depth of the ambient fluid. In the lateral (y) direction, the mesh was symmetrical across the centerline of the outlet structure. Thus, the mesh description along one side of the outlet structure would be provided. The mesh size within the outlet structure was 20 cm and gradually increased to 50 cm within a distance of 10 m from the edge of the structure. The mesh size was doubled within each 10 m distance after that. In the x -direction, the mesh size was 20 cm for a distance of 5 m from the outlet. The mesh size was gradually increased to 50 cm within a distance of 20 m. The mesh size was doubled in the next 20 m and was kept constant from then on. A time step of 0.006 second was used for simulation. For all the tests, a uniform velocity was prescribed as a boundary condition at the outlet. A continuative boundary condition was applied at the downstream end as well as two side boundaries. In the case of continuative boundary condition, the flux gradient across the boundary was zero, i.e., the flow approaching the boundary was allowed to cross. Apart from the outlet structure, the upstream end was considered a no-flow boundary.

Typical velocity fields 10 cm below the initial water surface (due to surface waves), for box culvert and diffuser at high flow and low water level, are shown in Figs. 19 and 20, respectively. The results show the zone of 0.52 m/s velocity from the corresponding outlet structures. Data for all outlet structures for various combinations of discharge and water level are shown in Table 1. The Y-pipe is found to be the most efficient outlet structure, but due to construction difficulties it is not adopted. The box culvert is found to be the best outlet structure from the point of view of efficiency and constructional ease. Table 1 clearly shows that the

high velocity persists for a longer distance at lower depths as a result of restricted entrainment in the vertical direction.

Fig. 19 Flow pattern near the surface of the box culvert outlet.

Fig. 20 Flow pattern near the surface of the diffuser outlet.

Table 1 Length of the 0.52 velocity zone.

Outlet Type	Discharge	Water Level	Inlet Velocity (m/s)	Length of 0.52 m/s Velocity Zone (m)
Pipe	High	High	2.72	156
		Low		189
	Low	High	1.4	29
		Low		54
Diffuser	High	High	1.87	183
		Low		185
	Low	High	0.97	124
		Low		154
Y-Pipe	High	High	1.36	44
Box Culvert	High	High	0.60	138
		Low		160
	Low	High	0.31	-
		Low		-

8. SUMMARY AND CONCLUSIONS

The $k-\varepsilon$ and RNG schemes, employed in a three-dimensional turbulent flow model (FLOW-3D), with standard coefficients are evaluated for their accuracy in predicting the characteristics of the free plane and circular turbulent jets. In particular, the growth rate, the decay of the centerline longitudinal velocity, the similarity of longitudinal and vertical velocity profiles at different locations along the jet, and the profiles of kinetic energy per unit mass are compared with the available experimental data and theoretical analysis.

The results show that the $k-\varepsilon$ scheme provides better estimation of growth rates and decay of the centerline longitudinal velocity for both plane and circular turbulent jets. Both schemes predict the similarity of longitudinal and vertical or radial velocity profiles satisfactorily with the $k-\varepsilon$ scheme providing slightly better results. The estimation of kinetic energy per unit mass by both schemes incurs appreciable error especially in the case of circular jet. From the simulation results of ideal circular and plane turbulent jets, it is clear that the $k-\varepsilon$ turbulent closure scheme with standard coefficients, although simpler, can be used to predict effectively the characteristics of plane and circular turbulent jets.

The computational model with $k-\varepsilon$ scheme is applied to design the outlet geometry of a stormwater outflow in shallow water depth. Various types of outlet structures are considered to minimize the zone of high velocity, which is deemed detrimental for recreational navigation, and a box culvert is found to be the most practical alternative.

LIST OF SYMBOLS

A	represents a growth rate or velocity decay coefficient
b	a measure of jet growth
b_o	half the nozzle width
$c_{1\varepsilon}$	empirical coefficient
$c_{2\varepsilon}$	empirical coefficient
c_μ	empirical coefficient
d	diameter of the nozzle
k	turbulent kinetic energy per unit mass
P	piezometric pressure
P_{rk}	Prandtl number for k
$P_{r\varepsilon}$	Prandtl number for ε
R	source term in RNG scheme

S	mean rate of strain
t	time
U_i	time averaged velocity components ($i=1,2,3$)
u	velocity in the x -direction at any point in the jet
u_m	centerline velocity in the x -direction
u_o	jet velocity at the nozzle
$\overline{u_i u_j}$	turbulent normal and shear stresses
v	velocity in the r - or y -direction
x_i	coordinate directions in x, y, z ($i=1,2,3$)
$\bar{x}$	distance to the virtual origin
α	a constant
α_1	virtual origin for plane turbulent jet
α_2	virtual origin for circular turbulent jet
β	a constant
ν	kinematic viscosity of fluid
ν_{eff}	combination of fluid and turbulent kinematic viscosity
ν_t	turbulent eddy viscosity
ρ	density of fluid
δ	Kronecker delta
ε	dissipation rate of turbulence per unit mass
σ_k	empirical coefficient
σ_ε	empirical coefficient
η	a function of S, k , and ε
η_o	a constant
λ	non-dimensional distance in r - or y -direction

REFERENCES

1. Abramovich GN (1963). *The Theory of Turbulent Jets*. M.I.T. Press (English Translation), Massachusetts, 671 pp.
2. ASCE Task Committee on Turbulence Models in Hydraulic Computations (1988). Turbulence Modeling of Surface Water Flow and Transport. *Journal of Hydraulic Engineering* 114(9):970–1073.
3. Bischof CH, Bucker HM, Rasch A (2004). Sensitivity Analysis of Turbulence Models Using Automatic Differentiation. *J. Sci. Comput.* 26(2):510–522.
4. Heskestad G (1965). Hot-wire Measurements in a Plane Turbulent Jet. *Trans. ASME, J. Appl. Mech.* 87:721–728.
5. Johnston A J (1985). Aspects of Buoyant and Non-buoyant Jets Entering Shallow Tailwaters.

Proc. Intl. Assoc. for Hydraulic Research, 21st Congress, Melbourne, Australia.

6. Rajaratnam N (1976). *Turbulent Jets*. Elsevier Scientific, New York, 304 pp.
7. Rodriguez JF, Bombardelli FA, Garcia MH, Frothingham KM, Thoads BL, Abad JD (2003). High-resolution Numerical Simulation of Flow Through a Highly Sinuous River Reach. *Water Resources Management* 18:177–199.
8. Wagnanski I, Fiedler H (1969). Some Measurements in the Self-preserving Jet. *J. Fluid Mech.* 38:577–612.