World Applied Sciences Journal 26 (10): 1351-1356, 2013
ISSN 1818-4952

DOI: 10.5829/idosi.wasj.2013.26.10.1239

Numerical Model of Baffle Location Effect on Flow Pattern in Oil and Water Gravity Separator Tanks

[1]Haitham A. Hussein, [2]Rozi Abdullah, [3]Sobri Harun and [4]Mohammed Abdulkhaleq

[1,2]School of Civil Engineering, Universiti Sains Malaysia,
14300 Nibong Tebal, Seberang Perai Selatan, P. Penang, Malaysia
[3]Faculty of Civil Engineering, Universiti Teknologi Malaysia, 81310 Johor Bahru, Malaysia
[4]Civil Engineering Department, Alnahrain University, Baghdad, Iraq

Submitted: Sep 12, 2013; **Accepted:** Nov 14, 2013; **Published:** Nov 30, 2013

Abstract: Rectangular separator tanks used gravity settling to separate oil from water in treatment units. In this study, in order to increase the performance of separator tanks, the effect of different positions of baffle structure on improving the uniformity of flow was investigated. Two dimensional computation fluid dynamics (CFD) and one phase flow model was built to simulate the flow properties in the gravity separator tanks including the velocity profile distribution and volume of circulation region. A CFD programme, Flow 3D ver.10 was used in which finite volume method is used for solution of water flow equations and RNG turbulent model with the Navier-Stokes equations. The volume of fluid (VOF) method was used for tracking of free surface in simulation program. The result of numerical simulation indicate that the best location of baffle is obtained when the standard deviation of the velocity distribution and volume of circulation zone along the separator tanks is minimized. In other words, if the single baffle structure constructed in the last fourth part from the inlet tanks it can be achieved to the best separation between oil and water inside separator tanks.

Key words: Gravity separator tanks · Numerical modeling · Baffle · Oil droplet · CFD · Navier-Stokes equation

INTRODUCTION

Environmental consciousness is presently manifesting itself in tightening environmental legislation throughout the world. There are many discharge liquid waste polluted with hydrocarbon, such as source waste include petroleum and petrochemicals refining and processing, tramp oils from mechanical repair stores, utility operations, restaurants, sanitary sewage, bilge and ballast water, contaminated surface runoff. Oily wastewater contains toxic constituent such as phenols, petroleum, polyaromatic hydrocarbon, which effect to plant and animal growth and caused mutagenic and carcinogenic to human therefore must treated wastewater and separate the oil from water more as possible [1].

The removal of fat, oil and grease (FOG) from water and wastewater by gravity separation rectangular tank is one of the most widely used in treatment units. Normally for the mixture of oil in water, oil has less density than water and gravity will cause the lighter oil to rise to the water surface. A uniform flow is fundamental point to the efficiency of separation between oil and water inside rectangular tanks, so adjustment the flow pattern leads to increase the coalescing of oil droplet and as a result rise rate of oil droplet will be faster to reach to water surface of separation tanks (Stokes' law).

Morrow *et al*. [2] described installed flat plat infront of the internal flow at several location and showed that the inlet liquid flow of oil and water in separator tank have a high important effect on separation performance. Experiments were obtained in 1.83 m long and 0.46 m wide rectangular tank. Axial velocity values at the centreline were validated with computer predictions. Chen *et al*. [3] optimized two dimensional numerical modelling of the hole spacing and the diameter of holes in perforated baffle inside separator tank. The simulation with 40 mm

Corresponding Author: Rozi Abdullah, School of Civil Engineering, USM, Engineering Campus, 14300 Nibong Tebal, Seberang Perai Selatan, P. Pinang, Malaysia Tel: 04-5995834,6281.

had better improved the flow uniformity than 69.3 mm spacing with an optimum diameter of 10.5 mm and 28% as the optimum free area. Aziz *et al*. [4] analyzed numerical and experimental models (in several restaurants) results and alternative inlet geometry, outlet and baffle wall design for fat, oil and grease (FOG) mixture with water and evaluated the removal efficiency in gravity separators compartment by measured the (FOG) concentration at influent and effluent of grease abatement device by collecting sample. Highest removal performance (90%) occurred with configuration at 1-hour hydraulic retention time. Abdulkadir *et al*.[5] analyzed numerically, using the same volume fraction, the relation between the inlet velocity values and oil diameter on oil-water separator tanks by using computation fluid dynamics, the results shows the mixture velcity at 0.5 m/sec more produced when compared with 1.0 m/sec, in contrast the oil diameter at 1 mm produced the best results when compared with 0.5 and 0.25mm.

The main aim of this research is to find the best single baffle location inside the rectangular gravity separation tanks that corresponds to optimum floatation of oil droplet performance in the tank, this case is achieved when the flow inside the tank is close to minimum circulation zone of flow in separator tanks or the dead zone is divided into smaller part [6]. In this study the investigation of baffle position effect on the separation efficiency is performed via simulation flow 3D program[7] using single phase flows, water only.

Mathematical Model

Time-Averaged Flow Equations: In hydraulic numerical modeling the incompressible steady state flow with viscous effect are usually considered to determine the flow conditions. The mass continuity and momentum equations are generally used in governing equations and the turbulence flow model used with these equations to calculate the Reynolds stresses and the analysis of the separation tank model was assumed to be two dimensional flow with x and z directions [8]. the general mass continuity equation is [9-10].

$$V_f \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}\left(\rho u A_x\right) + \frac{\partial}{\partial z}\left(\rho w A_z\right) = 0 \tag{1}$$

Where $V\ f$ is the fractional volume open to flow in calculation cell; ρ is the fluid density; u and w are the velocity components in the length and height (x,z) directions, respectively. The momentum equation for the fluid velocity components in the two directions are the Navier-Stokes equations expressed as:

$$\frac{\partial u}{\partial t} + \frac{1}{V_f}\left\{uA_x \frac{\partial u}{\partial x} + wA_z \frac{\partial u}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial P}{\partial x} + G_x + f_x \tag{2}$$

$$\frac{\partial w}{\partial t} + \frac{1}{V_f}\left\{uA_x \frac{\partial w}{\partial x} + wA_z \frac{\partial w}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial P}{\partial z} + G_z + f_z \tag{3}$$

Where, G_x, G_z are body accelerations and f_x, f_z are viscous accelerations that form a variable dynamic viscosity μ given by:

$$\rho V_f f_x = wsx - \left\{\frac{\partial}{\partial x}\left(A_x \tau_{xx}\right) + \frac{\partial}{\partial z}\left(A_z \tau_{xz}\right)\right\} \tag{4}$$

$$\rho V_f f_z = wsz - \left\{\frac{\partial}{\partial x}\left(A_x \tau_{xz}\right) + \frac{\partial}{\partial z}\left(A_z \tau_{zz}\right)\right\} \tag{5}$$

where:

$$\tau_{xx} = -2\mu \frac{\partial u}{\partial x}; \qquad \tau_{zz} = -2\mu \frac{\partial w}{\partial z}; \qquad \tau_{xz} = -\mu\left\{\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right\} \tag{6}$$

In the terms above, the parameter *wsx* and *wsz* refer to wall shear stresses. in modeled the wall stress assuming a zero tangential velocity on the part of area closed to flow. However mesh boundaries are an exception because they can be assigned non-zero tangential velocities. For turbulent flows, a law-of-the-wall velocity profile is assumed near the wall, which modifies the wall shear stress magnitude [7].

The volume of fluid (VOF) method was used for tracking of free surface in simulation program Flow3D. With the VOF method, grid elements are classified as empty, full, or partially filled with fluid. Cells are allocated in the fluid fraction varying from zero to one, depending on quantity of fluid. Thus, in *F=0*, void region, whereas *F=1* represents a fluid exists. This function appears the VOF per unit volume and satisfies the equation [10].

$$\frac{\partial F}{\partial t} + \frac{1}{V_F}\left\{\frac{\partial}{\partial x}\left(FA_x u\right) + \frac{\partial}{\partial z}\left(FA_z w\right)\right\} = 0 \tag{7}$$

RNG Turbulence Model: The RNG turbulence model solves for the turbulent kinetic energy (k) and the turbulent kinetic energy dissipation rate (**ε**) using numerical techniques for differential equation. This RNG model applies statistical methods to derive the mean equations for turbulent quantities, such as turbulent kinetic energy (k) and its dissipation rate (ε). In addition the RNG model is useful for swirl streamline, as in circulation inside tanks. Renormalization group models depend less on empirical constants while setting a framework to derive a range of parameters to be used at different turbulence scales [11-12]. The RNG model more improved prediction for high stream line curvature and strain rate in comparison with k-ε model and RNG model uses equations like to the equations for the k-ε model.

The equation constants that are established empirically in the standard k-ε model are derived explicitly in the RNG model [11]. The turbulence kinetic energy, k and its rate of dissipation, ε, in two directional flow are obtained from the following transport equations:

$$\frac{\partial k}{\partial t}+\frac{1}{V_f}\left\{uA_x\frac{\partial k}{\partial x}+uA_z\frac{\partial k}{\partial z}\right\}=P+G+Diff-\varepsilon \tag{8}$$

$$\frac{\partial \varepsilon}{\partial t}+\frac{1}{V_f}\left\{uA_x\frac{\partial \varepsilon}{\partial x}+uA_z\frac{\partial \varepsilon}{\partial z}\right\}=\frac{C_{1\varepsilon}\varepsilon}{k}\left(P+C_{3\varepsilon}G\right)+ DDif-C_{2\varepsilon}\frac{\varepsilon^2}{k} \tag{9}$$

Where P is shear production, G is buoyancy production, *Diff* and *Ddif* represent diffusion and $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constant. In the RNG model, $C_{1?}$ = 1.42, $C_{2?}$ = 1.68 and $C_{3?}$ = 0.2 [11-12].

Numerical Model: In hydraulic numerical modeling, the Navier-Stokes equation has been well adapted as an effective solution to the governing equation. This equation is an incompressible form of the conservation of mass and momentum equations and is contained of non-linear advection, diffusion, rate of change and source term in the partial differential equation. The mass and momentum equations coupled through velocity can be considered to derive an equation for the pressure term. The calculation becomes more complex if the turbulent flow phenomenon be considered in the Navier-Stokes equation. In this case, the Reynolds-Averaged Navier-Stokes (RANS) equation are used. it is modified form of the Navier-Stokes equation with Reynolds stress term, which approximates the random turbulent fluctuation by statistics.

In this study, the numerical modeling was used to simulate the gravity separator tanks by using water for different single baffle position inside the tank. The available program for Computation Fluid Dynamic (CFD) model is the flow 3D program (version 10) that developed by Flow Sciences. This soft ware solves the (RANS) equations by the finite volume formulation acquired from a rectangular finite difference grid. For each cell, the mean values for the flow parameter, velocity and pressure, are computed at discrete times. The new velocity in each cell is calculated from the both continuity and momentum equations using the initial conditions or the values of previous time step values and the pressure terms are solved and adjusted by estimated velocity to satisfy the continuity equation [7, 13].

In mesh geometry on the finite control volume the Fraction Area/Volume Obstacle Representation (FAVOR) method was used [9]. This method is a porosity technique which defines the obstacle in a cell with a fraction value zero to one as an obstacle fills the cell. Geometries are embedded in the mesh by adjusting the area fraction on the cell sides with the volume fraction open to flow [14].

Comparison Test: In order to validate the numerical simulation (Flow 3D) of gravity separator tank, the numerical and experimental model used by Morrow, T. and F. Dodget [2] were considered. For this test, the flow is clear and has no mixture with oil. The inlet liquid flow rate was 0.0025 m^3/sec and the volume of fluid retained in the separator tank model was about 0.212 m^3. The dimensions of the horizontal separator tank model are: height, 0.46 m; width 0.46 m; length 1.83m. The numerical model was applied to simulate this separation tank using a uniform rectangular mesh and the cell size in two directions is 10 mm. In this verification the RNG turbulent model was selected to find the effect of turbulent on the flow field.

Figure 1 shows a comparison between results of numerical model using flow 3D and predicted with measured and numerical velocity profile by Morrow, T. and F. Dodget [2] in three different location downstream of the inlet vessel 30.48 cm, 60.96 cm and 90.44 cm from the inlet. The x and y axis in Fig (1) represent the velocity of flow (u) and height of tank (z), respectively. For these three profiles shows the velocities are in good agreement between the presented an experimental with numerical results of T. Morrow and F. Dodget 1991.

Geometry Generation: The geometry of the horizontal gravity separator rectangular tank with baffle is shown in Figure 2. The tank is 130 cm long, 42 cm water depth and 50 cm wide and the water flow rate is 0.002 m^3/sec. The height of inlet slot 10 cm, the baffle plate is located in four position from the vertical inlet tank, d= 60, 80, 100 and

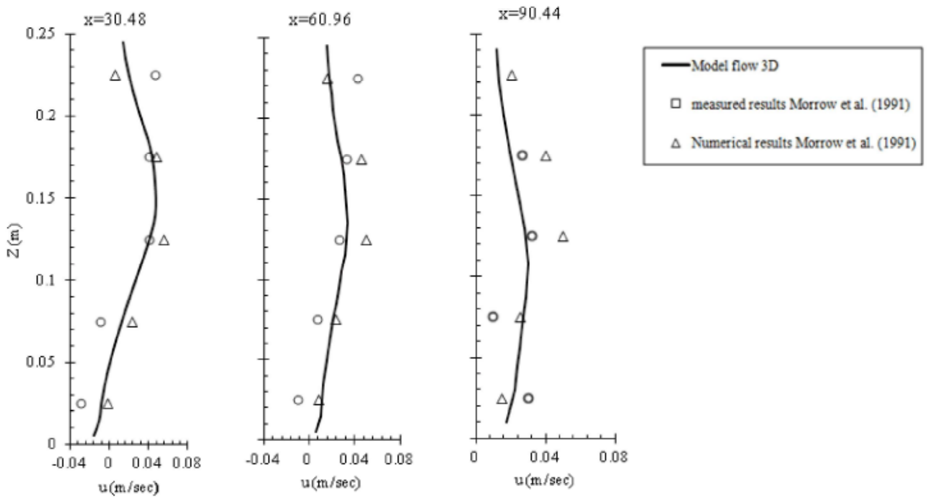

Fig. 1: Comparison of velocity distribution from present work with experimental and numerical simulation Morrow, T. and F. Dodget[2].

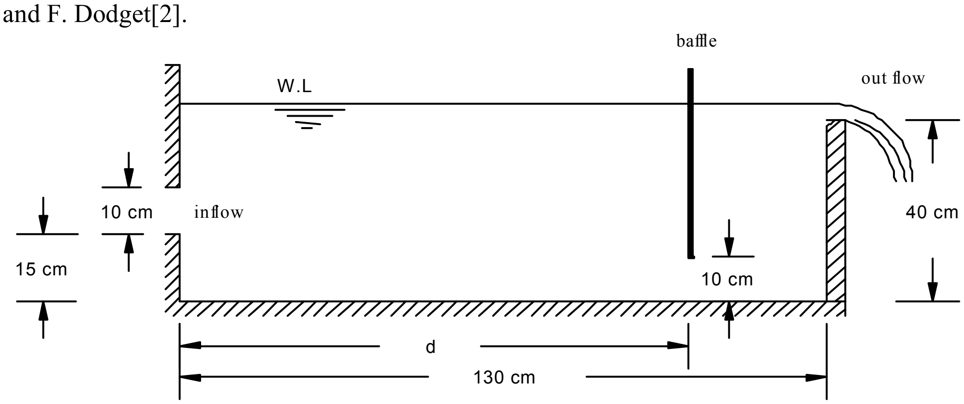

Fig. 2: Cross section of gravity separation tanks

120 cm, the height of the baffle from the tank bottom fixed on 10 cm. A weir is built in the end of the tank to regulate the flow level of H=40 cm.

In this study, the separator tank model operates under atmospheric pressure; the rectangular separation tank was modeled in two dimensions flow to reduce the computation power requirements.

The mesh pattern must be built very carefully in order to obtained accurate results. The computation mesh was established such that they conducted recirculation area, inlet, outlet and baffle. The grids used for the simulations consist of 9150 rectangular cells.

No slip boundary condition was applied for all the fixed walls. Top water surface was modeled as free surface and calculated by the VOF method. Inlet flow velocity was 0.04 m/s in the x direction and the outlet was assumed as the outflow boundary condition.

RESULTS AND DISCUSSIONS

Velocity Profile: In general, the fluid velocity pattern in separator tank is not uniform. The rise of oil droplet (vertical velocity) in water depend on the density and viscosity of water and additional the vertical velocity of oil droplet is highly dependent on the droplet diameter, with small oil droplet rising mush more slowly than larger one[15]. For this reason the separator tank must be designed to reduce the inlet horizontal velocity as more as possible and produce more uniformity to allow the oil droplet coalescing with other droplets to achieve high rates of droplets rise toward the separator surface.

For each case, the x-velocities were retrieved at 41 equally spaced points along 9 vertical positions in separator tank. The standard deviation (SD) was calculated along these specific vertical positions; it indicated the flow uniformity by evaluating the discrepancy of velocity from the mean velocity.

Table 1: Standard deviation of velocity in different location of the baffle

Baffle distance (d) from the inlet (cm)	60	80	100	120
Standard deviation (m/s)	2.247×10^{-3}	2.167×10^{-3}	1.679×10^{-3}	1.848×10^{-3}

Table 2: Volume of circulation zone in different location of the baffle

Baffle distance (d) from the inlet (cm)	60	80	100	120
Circulation volume %	66.42	64.7	58.7	59.8

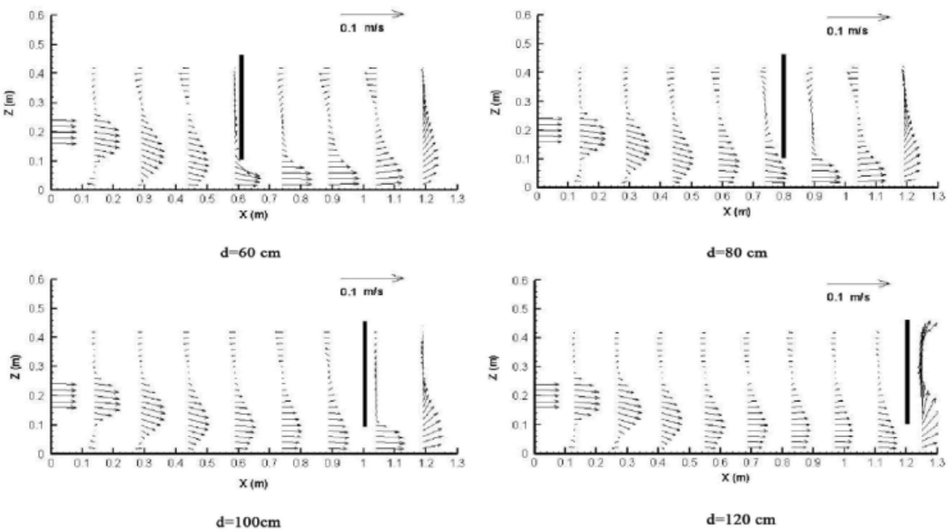

Fig. 3: X-velocity vectors of different baffle position in gravity separator tanks

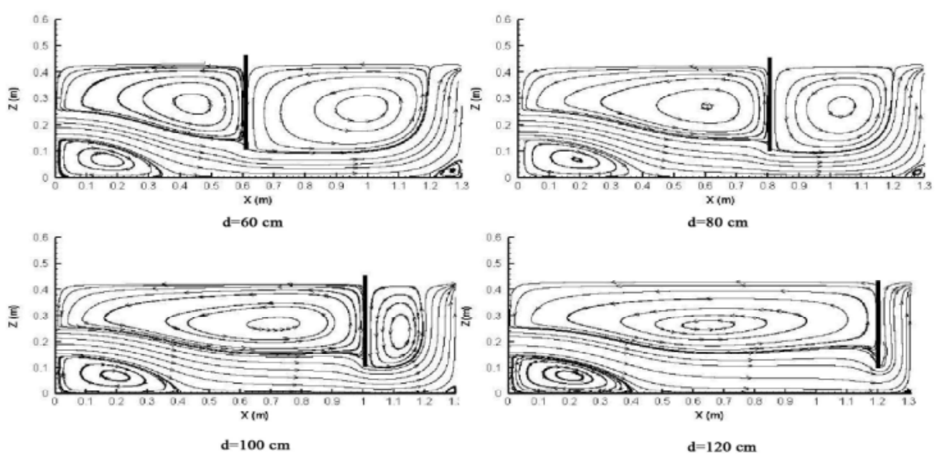

Fig. 4: Streamlines of different baffle position in gravity separator tanks

Table 1 shows the baffle distance from the inlet and standard deviation of velocity, the baffle at 100 cm position from the inlet has minimum magnitude of (SD) 1.679×10^{-3} and consequently exhibit the best performance (a better separator).

In table 1 the values of standard deviation baffle location at 100 and 120 cm are close from them, this table indicates that a single baffle in separation tank must be constructed in the last fourth part from the total length.

Figure 3 shows the x-velocity vector, In cases at d = 100 and 120 cm the value of negative x-velocity vector is smaller than in cases at d = 60 and 80 cm, especially has been clear near the water surface of the tank and this part is very important for gravity separator tank to make the oil droplet stable and then removed by skimmer.

Flow Field: The performance of separator tanks depend on the flow pattern. Therefore the baffle structure in the separator tanks is used as on energy dissipaters, reduce the circulation zone volume inside the tank and in additional to collect the oil droplet from the water surface.

The best position of the baffle is acquired when the volume of circulation zone is minimized. Thus, the proper position for the baffle may achieve to obtain more uniform distribution of velocity in the separator tanks and minimize dead zone.

In this study, four baffle positions were modeled in the separator tanks. Figure 4 shows the streamlines inside separator tanks for different baffle structure locations. Table 2 shows the baffles locations with the volume of vortices or circulation zone, which is intended by the total water volume in separator tanks and calculated by the numerical method. This table indicates that the baffle location at 100 cm from the inlet flow has the minimum value of circulation volume, while the maximum magnitude of circulation volume appears when construct the baffle at 60 cm from inlet flow.

Furthermore, if it is considered the valueof the circulation volume in cases baffle locations at 100 and 120 cm are close from them, the volume of circulation zone reduce whenever the position of baffle constructed on the end of separator tanks. Consequently, the removal efficiency of separator tanks increased.

CONCLUSION

The study was performed to evaluate the oil and water gravity separator tank performance. Best uniformity of flow in separator tank will improve the development to the maximum removal efficiency of oil droplet from water. Investigation the effect of baffle structure in separator tank is one method to improve flow uniformity in separator tank. Two dimensional CFD with VOF method predictions had confirmed that comparison of four locations of single baffle structure enhance uniformity of flow across separator tanks. Numerical model was conducted to find the effect of baffle position on the velocity profile. Standard deviation served as an indicator of flow uniformity along the separator tank sections, minimum (SD) lead to best uniformity (a better separator). Moreover, the volume of vortices or circulation zone are normalized to show the less circulation zone in each cases of baffle position in separator tank. The results show that the constructed baffle on the 100 cm distance from the inlet lead to minimum SD and volume of circulation zone percentage. Consequently, Table 2 show some indication that when design the separator tank single structure baffle must be built between (75-80) percent from the total length to achieve to best uniformity.

REFERENCES

1. Abass O, A., *et al*., 2011. Removal of Oil and Grease as Emerging Pollutants of Concern (EPC) in Wastewater Stream. IIUM Engineering Journal, 12: 4.
2. Morrow, T. and F. Dodget, 1991. Fluid Flow Modelling of Gravity Separators. Multi-phase production, pp: 364.
3. Chen, G.M., M. Iskandr and M. Nor, 2011. Flow pattern in a horizontal primary separator with a perfoated baffle. Journal-the Institution of Engineers, Malaysia, 72: 3.
4. Aziz, T.N., *et al*., 2011. Performance of grease abatement devices for removal of fat, oil and grease. Journal of Environmental Engineering, ASCE, 137(1): 84-92.
5. Abdulkadir, M. and V. Hernandez-Perez, 2010. The effect of mixture velocity and droplet diameter on oil-water separator using computational fluid dynamics (cfd). World Academy of Science, Engineering and Technology, 61: 35-43.
6. Razmi, A., B. Firoozabadi and G. Ahmadi, 2009. Experimental and numerical approach to enlargement of performance of primary settling tanks. Journal of Applied Fluid Mechanics, 2(1): 1-12.
7. Flow Science, 2009. Flow-3D user manual: Santa Fe, NM, USA.
8. Rostami, F., *et al*., 2011. Numerical modeling on inlet aperture effects on flow pattern in primary settling tanks. Applied Mathematical Modelling, 35(6): 3012-3020.
9. Hirt, C. and J. Sicilian, 1985. A porosity technique for the definition of obstacles in rectangular cell meshes. in Proc. Fourth International Conf. Ship Hydro.
10. Hirt, C.W. and B.D. Nichols, 1981. Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of Computational Physics, 39(1): 201-225.
11. Yakhot, V. and L.M. Smith, 1992. The renormalization group, the □-expansion and derivation of turbulence models. Journal of Scientific Computing, 7(1): 35-61.
12. Yakhot, V. and S.A. Orszag, 1986. Renormalization group analysis of turbulence. I. Basic theory. Journal of Scientific Computing, 1(1): 3-51.
13. Ho, J., *et al*., 2007. Numerical modeling study for flow pattern changes induced by single groyne. in Proceedings of the Congress-international Association for Hydraulic Research.
14. Hirt, C., 1992. Volume-fraction techniques: Powerful tools for flow modeling. Flow Sci. Rep. FSI-92-00, pp: 2.
15. API, 1990. Design and Operation of Oil-Water Separators. American Petroleum Institute, Washington, Publication, pp: 421.