Modelling Stilling Basins for Sewage Networks

Shahad R. Mohammed¹, Basim K. Nile ², Waqed H. Hassan ³

¹ Graduate student, Civil Engineering Department, University of Kerbala, Kerbala, Iraq. E-mail: Civileng416@gmail.com
² Professor, University of Kerbala, Kerbala, Iraq. E-mail: Basimnile@yahoo.com
³ Professor, University of Kerbala, Kerbala, Iraq. E-mail: Waaqidh@uokerbala.edu.iq

Abstract. Concrete elements play an effective role in stilling basin design in terms of enhancing the dissipation of energy to prevent immoderate scouring of downstream structures. Proper appurtenances, such as wedge-shaped splitter blocks, baffle walls, inverted t-sections, rectangular walls, inverted L sections, and impact walls, are thus utilised with circular pipe outlets to create stilling basins that are shorter yet more effective, and thus more economical. This research was based on numerical analysis of stilling basins using FLOW-3D software; the resulting Froude numbers were compared with the Froude number of a case study, a stilling basin manhole created by the Karbala Government. This research thus examined the influence of Froude number, the arrangement and location of dissipators, and the height of the pipe inlet under different hydraulic condition in order to better simulate stilling basin models. The turbulent energy dissipation was found to vary from 16% to 85%. The results from the models showed that L-sections offered maximum turbulent energy dissipation (ΔE/E₁), while the rectangular wall shape gave minimum turbulent energy dissipation (ΔE/E₁). It was also observed that the maximum turbulent kinetic energy almost always occurred in the initial sections, with flow becoming generally less turbulent at the end sections. Thus, at the end of all stilling basins, the turbulent kinetic energy dissipation and turbulent kinetic energy profiles became more uniform. The proposed numerical simulation using FLOW-3D was shown to be a reliable tool for predicting and discussing the turbulent energy dissipation in stilling basins. Further conclusions included the fact that the maximum turbulent energy dissipation (ΔE/E₁) increases as values of Fr increase. The results of this research could thus help hydraulic designers to consider their selections for the optimum design of stilling basins more carefully.

Key words: FLOW-3D, Hydraulic jump, Stilling basins, Turbulent energy dissipation.

1. Introduction

Stilling basins are hydraulic structures adopted for dissipating the energy of downstream flow; they also decrease the velocity and energy of such flow. The control and dissipation of energy can thus be said to be the main purpose of stilling basins. When wastewater floods down manholes, it flows from a pump station to the stilling basin manhole with very high kinetic energy. The molecules of wastewater have a great deal of potential energy, which is converted into kinetic energy in the downstream flow. The main reason for
this kinetic energy formation is the high velocity of downstream flow; scouring and erosion will thus occur in outlet structures if this energy cannot be reduced or dissipated.

Several contemporary studies have investigated the issue of downstream energy dissipation in hydraulic structures, and various methods and techniques have been developed to either improve the efficiency of existing dissipaters or to find new dissipaters that satisfy the requirements of both high efficiency and low cost. These methods may be classified as (a) energy dissipation using stilling basins; (b) energy dissipation using counter (reverse) flow; and (c) energy dissipation by means of slopping surfaces. Inserting a hydraulic jump in a stilling basin has thus been determined to be the most efficient way to dissipate excess energy, as well as being less prone to corrosion [1]. In pipe outlets, Goel and Verma [2] developed more efficient energy appurtenances in comparison with Grade’s energy appurtenance, while for deep and narrow outlets, new designs for stilling basins were presented by Verma and Goel [3], who suggested models for efficient stilling basins based on studying the influences of appurtenances such as wedge shaped blocks, weir walls, impact walls, stepped walls, and grids on the hydraulic performance of such stilling basins.

Different designs of stilling basins can help to reduce energy in various ways, and Goel [4] developed a new design by implementing systematic experimentation on various models of stilling basins in which the pipe outlet was kept level with the basin floor while the appurtenance arrangements and locations differed in each experiment. Similarly, Ghamari and Nekoufar [5] found that chute block dimensions played a significant role in terms of selecting the correct type and size of stilling basin, and using the same approach, Peyman [17] showed that the central blocks offer high dissipation of energy and decreasing hydraulic jump depth in stilling basins.

Babaali [6] illustrated how flow pattern formation can be affected by stilling basin geometry and size, leading to influences on the hydraulic performance of the system in general, while Soori [7] proved that both energy dissipation and hydraulic jump stability increased when adverse steps were used at the end of the stilling basin.

Babaali and Soori [8] indicated that adding converging walls with an end adverse step in a stilling basin improved performance with regard to static pressure; the static pressure increased as the convergence angle increased, based on the distribution of pressure in the stilling basin bottom. Similarly, Eshkou [9] demonstrated that a stable hydraulic jump could be produced by utilising baffle blocks with an adverse slope in the hydraulic jump, as well as showing that the hydraulic jump characteristics were improved in a converging arrangement as compared to a diverging arrangement.

Hydraulic issues can often be solved by applying IT programs. Recently, computer technology development has thus allowed the addressing of further hydraulic problems, particularly by allowing the application of turbulence models, which are often important parts of these issues. The k-ε two-equation turbulence model is a useful technique used to simulate complex water flow [10-13], and Fluent and FLOW-3D software now offer useful tools for 3D simulation with the ability to assess otherwise non-measurable parameters experimentally [14,16] using numerical analysis; studies focused on these methods may thus improve the performance of stilling basin.

FLOW-3D [18] is a commercial computational fluid dynamics (CFD) package that includes special modules for hydraulic engineering applications; these use numerical techniques to solve fluid motion equations to
obtain transient, three-dimensional solutions to multi-scale, multi-physics flow problems. Its array of physical and numerical options allows users to apply FLOW-3D to a wide variety of fluid flows and heat transfer phenomena, and thus this software is widely in use for solving hydraulic problems.

In this paper, the numerical simulation method was similarly utilised in FLOW-3D to identify optimal stilling basin models. The results were then compared with a case study for validation by changing a number of parameters that influence the flow of the stilling basin. These parameters included the shape of the dissipators (baffle wall with gap, rectangular wall, inverted T-section, inverted L-section, wedge-shape splitter block, and baffle wall with two gaps), the position of the dissipators (0.4, 0.6, 0.9 and 1.2 m), the Froude number (1.5, 2.5, and 3.5), and inlet pipe height inside the stilling basin. Hydraulic jump characteristics were also studied using the 3D RNG k-ε turbulence model in FLOW-3D, allowing the optimal stilling basin to be selected.

2-Methodology
This paper focused improving stilling basins in terms of making them more economical, shorter, and practical, by utilising suitable appurtenances to increase the energy dissipation after circular pipe outlets.

2.1 Case study model
To verify the effectiveness of the numerical simulations of stilling basins, field data was collected from Karbala city (case study), and a comparison between the analysis and case study was made using the Froude number for the case study in the FLOW-3D software at a scale of 1:3, in order to evaluate the software and check the validity of the results. For pipe outlets, the design of stilling basins is based on the inflow Froude number, Fr, as shown in equation (1).

\[
Fr = \frac{v}{\sqrt{g \cdot d}}
\]  

where
\[V\] is the jet efflux velocity,
\[d\] is the outlet pipe diameter, and
\[g\] is gravity acceleration

2.2 Numerical model
2.2.1 Volume of Fluid (VOF) turbulence model
In general, flow in stilling basins is very complex, though the Navier-Stokes equations offer a mathematically accurate numerical method of determining outcomes. Thus, numerical simulation of free surface turbulent flow has become a salutary and extensively used method [15]. FLOW-3D software solves the governing equations using the finite volume method, which is used to represent and evaluate partial differential equations in algebraic equations form and employs both Fractional (Area/Volume) Obstacle exemplification (FAVOR) and Volume of fluid (VOF) methods to promote accuracy both in modelling
rigid bodies and in simulating fluid behaviour. FLOW-3D software defines the equations of continuity; momentum and the free surface profile as follows:

**Continuity equation**

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0
\]

**Momentum equation**

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial u_i}{\partial x_j} \right] + \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

**K - equation**

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon
\]

**\( \varepsilon \) – equation**

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial \varepsilon}{\partial x_j} \right] + c_1 \varepsilon \rho \frac{k}{\varepsilon} G_k - c_2 \rho \frac{\varepsilon^2}{k}
\]

where

\( \rho \) and \( \mu \) are the density of the averaged volume fraction and the molecule viscosity coefficient, respectively.

\( \mu_t \) is the turbulent viscosity coefficient, obtained through the turbulent kinetic (k) and the turbulent dissipation (\( \varepsilon \)) rates

\[
\mu_t = c_\mu \frac{k^2}{\varepsilon}
\]

\[
c_1 \varepsilon = 1.42 - \frac{\eta}{1 + 5 \eta}
\]

\[
\eta = \frac{sk}{\varepsilon}
\]

\[
S = \sqrt{2sij \delta ij}
\]

In the tensor expression, \( x_i \) represents x, y, z and \( u_i, u, v, \) and \( w \) where \( i = 1, 2, 3 \).

\( i \) and \( j \) are thus used as summation subscripts.

The general model constants are \( \eta = 4.38, \beta = 0.012, c_\mu = 0.085, c_2 = 1.68, \partial_k = 0.7179, \partial \varepsilon = 0.7179 \)
2.2.2 Boundary conditions
In numerical flow analysis, one of the most essential steps is determining suitable boundary conditions, which should agree with the physical condition of the problems. In terms of Cartesian coordinates, FLOW-3D utilizes perpendicular hexahedral meshes to limit the domains of three-dimensional flows. The rectangular mesh prism thus requires six types of boundaries. The boundary conditions used in this study are illustrated in Figure 1; these are specific velocity (V) at the upstream mesh plane (inlet boundary), outflow (O) in the downstream mesh plane (outlet boundary), the walls on the top and bottom boundary (W), and symmetrical right and left boundary sides (S).

![Figure 1: Boundary conditions of numerical simulation model](image)

2.2.3 Mesh Generation
In FLOW-3D, determining the correct grid is one of the most significant matters in terms of generating an accurate solution. If the generated mesh is of good quality, more accurate results can be obtained from the numerical model. Determining a suitable mesh domain along with an appropriate size of mesh for each cell thus plays a critical role in such simulation. Both the cell size and grid can also influence simulation time as well as accuracy of results, however, making it important to reduce the volume of cells while retaining sufficient resolution to reflect the chief factors of the geometry and enough detail of flow.

The meshes used in FLOW-3D software have cubic cells, which also has an influence on the simulation process. In this study, various different cell sizes were thus tested (2.5, 2,1.5, and 1cm) to allow identification of the best cell size to satisfy the phenomena conditions. Figure 2 shows the results of the tests carried out to determine the best number of cells, which suggests that energy dissipation does not change greatly with further reductions from a cell size of 1.5 cm, which was thus determined to be the best cell size for modelling energy dissipation, giving a corresponding number of cells of 72,085,300. Figure 3 further shows the differences between fine and coarse meshes.
Figure (2) : Relationship between size of cells and ratio of turbulent energy dissipation ($\Delta E/E_t$)

Figure (3): Mesh generation by FAVOR (Fractional Area–Volume Obstacle Representation),

(a) Fine mesh; (b) Course mesh.

2.2.4 Shapes of dissipater used in stilling basin models

Stilling basins are seldom constructed to contain the whole length of hydraulic jump, and several means of reducing the length of the hydraulic jump are applied, including the construction of appurtenances within the basin. The appurtenances shown in Figure 4 also provide better performance in shortened basins [13] and are thus more normally used.
3 Results and Discussion

3.1 Validation of numerical models

The numerical simulation of the stilling basin was conducted using field data collected from the case study. The calculations were conducted in two cases: (1) one pump was run; or (2) two pumps were run. The latter was considered the critical case. In the first case, the dimensions of the basin were initially calculated, then the incoming velocity and water depth before and after the addition of dissipators were calculated. The Froude number was then calculated at the pipe inlet. All of these calculations are shown in Table 1. Figure 5 shows the relationship between depth of water in the case study before and after collision with the dissipators ($y_1$, $y_2$, respectively), showing a decrease in water depth after impact with the dissipator. Figure 6 shows the a 2D sketch of the case study stilling basin.
## Table 1. Characteristics of the case study stilling basin

| Case study model calculation |       |
|------------------------------|-------|
| Basin length (m)             | 5.5   |
| Basin width (m)              | 2     |
| Basin depth(m)               | 1.8   |
| Inlet diameter (mm)          | 500   |
| Outlet diameter (mm)         | 500   |
| Beginning Velocity (m/sec)   | 2.5   |
| Fr actually*                 | 1.12  |

\[ Fr = \frac{v}{\sqrt{g \cdot d}} \]

\[ Fr = \frac{2.5}{\sqrt{9.81 \cdot 0.5}} \]

Fr = 1.12
Verification of the model was carried out by comparing the resulting Froude numbers of the case study and the model. Figure (7) shows the results for the Froude number in the numerical model, with a final outcome of 1.19, which offers good agreement with the case study results of 1.12, suggesting an error rate of only 6%.
Figure (7): Froude number in the numerical model

The coloured bar in figure 7 represents the minimum and maximum values of the Froude number in blue and red, respectively.

3.2 Dimensional analysis

This theoretical study relied on applying dimensional analysis to obtain an equation without dimensions that links various relationships between different parameters and their effects on energy dissipation in stilling basins. This was done by changing the form of the energy dissipative structures, the Froude number, the height of entrance pipe, and the location of the energy dissipators in the test stilling basins. The functional relationship of these quantities can be written as

\[
(\Delta E/E_1) = f (Fr, (d/Y_1), (XS/Y_1))
\]

where:

- \(Fr\) = Froude number, as defined in Eq. (1)
- \(d\) = diameter of pipe inlet for stilling basin
- \(Y_1\) = height of inlet circular pipe
- \(XS\) = position of dissipators in stilling basin

\[10\]
3.3 Effect of Froude number on energy dissipation
Various stilling basin models for outlet diameter $d = 10\text{cm}$ were tested with different Froude numbers (1.2 to 3.5). Figure 8 shows the ratio of energy dissipation increased as the Froude number increased for different shapes of energy dissipater; it also shows that the L-section shape offered maximum energy dissipation while the rectangular wall gave minimum energy dissipation. Moreover, the Froude numbers values at the basin entrance are higher than those inside the basin, and the flow after collision with the dissipater turns from supercritical to subcritical flow by the end of the basin.

![Figure (8): Relationship between the Froude number and various shapes of energy dissipater](image)

3.4 Effect of inlet pipe height
To show the effect of inlet pipe height on turbulent energy dissipation, different heights of inlet pipe were used with Froude number values 1.5, 2.5, and 3.5, with the positions of the various dissipators kept constant in order to determine the maximum turbulent energy dissipation, as illustrated in figures 9, 10, and 11. In general, these figures show that turbulent energy dissipation increases with increases in the ratio of the diameter of pipe inlet to height of entrance pipe above the stilling basin floor ($d/y_1$), in this case, 0.14, 0.2, and 0.25.
Figure (9): Effect of ratio of diameter to height of inlet pipe (d/y1) on turbulent energy dissipation for Fr=1.5

Figure (10): Effect of ratio of diameter to height of inlet pipe (d/y1) on turbulent energy dissipation for Fr=2.5

Figure (11): Effect of ratio of diameter to height of inlet pipe (d/y1) on turbulent energy dissipation for Fr=3.5
3.5 Effect of energy dissipator position \( \left( \frac{x_s}{y_1} \right) \) inside stilling basin when \( \frac{d}{y_1} = 0.2 \)

Figures 12 and 13 demonstrate the positions of two invert L-section dissipators at 0.6 m and 1.2 m inside the relevant stilling basin models. From these figures, the turbulent energy dissipation is higher when the dissipator position is closer to the inlet pipe. Moreover, the maximum kinetic energy occurs at the beginning of the basin and the flow shows less turbulence at the end of a basin, after collision with the dissipator. This suggests that the length of stilling basin can be shortened if the disperser is placed near the inlet pipe, making the stilling basin will be shorter and more economical. The coloured bar in figure 12 used yellow and cyan to indicate the high initial kinetic energy \((E_1)\) before the hydraulic jump and the low kinetic energy \((E_2)\) after the hydraulic jump, respectively. The specific energy equation of flow used to calculate the result of turbulent energy dissipation in FLOW-3D is

\[
\Delta E/E_1 = \frac{E_1 - E_2}{E_1} \times 100
\]

The same procedure was used in figure 13 to calculate the turbulent energy dissipation. The value of \(\Delta E/E_1\) in case of a 1.2 m position was equal to just 33\% of the maximum energy dissipation seen at 0.6m. Figures 14 to 16 present the ratio of turbulent energy dissipation \((\Delta E/E_1)\) based on change of position of dissipaters’ ratio \((xs/y_1)\) for all shapes under different values of Froude number. The turbulent energy dissipation decreases when the ratio \((xs/y_1)\) increases for all dispersed shapes, though the optimum shape of energy dissipator is the L-section.

![Figure (12): Stilling basin simulation with dissipater position 0.6 m from inlet pipe](image)

\(\Delta E/E_1 = 60\%\)
Figure (13): Stilling basin simulation with dissipater position 1.2 m from inlet pipe

Figure (14): Effect of ratio of position of dissipator to height of inlet pipe (xs/y1) on turbulent energy dissipation for Fr=1.5
**Figure (15):** Effect of ratio of position dissipator to height of inlet pipe \((xs/y1)\) on turbulent energy dissipation for \(Fr=2.5\)

**Figure (16):** Effect of ratio of position dissipator to height of inlet pipe \((xs/y1)\) on turbulent energy dissipation for \(Fr=3.5\)

4-Development of turbulent energy dissipation formula

Many variables affect turbulent energy, as discussed previously. The accuracy of the suggested relationship was thus evaluated based on the coefficient of determination \((R^2)\). Eureqa-Formulize statistics software was used to analyse the resulting data by means of a nonlinear regression method. The formula in table 2 was
thus suggested as a means to predict turbulent energy dissipation based on the numerical data in terms of Froude number (Fr), position of dissipators (xs), and height of inlet circular pipe (Y₁).

Figures 17 to 22 shows the convergence of the predicted data to the observed data for each shape, and the corresponding coefficient of determination (R²).

**Table (2) Turbulent energy dissipation Formulas**

| Shape of dissipater                  | Equation                                                                 |
|--------------------------------------|--------------------------------------------------------------------------|
| Baffle wall with gap                 | \[(\Delta E/E₁) = 2.244 + 0.06547*Fr*(xs/Y₁) + 2.655*(xs/Y₁)^2 - 3.9084*(xs/Y₁) - 0.58321*(xs/Y₁)^3\] |
| L-section                            | \[(\Delta E/E₁) = 0.1015 + 0.09405*Fr + 0.3663/(xs/Y₁)\]                |
| T-section                            | \[(\Delta E/E₁) = 0.8146 + -1.0131*(xs/Y₁)/(3.525 + Fr)\]               |
| Rectangular wall                     | \[(\Delta E/E₁) = 0.396 + 0.09106*Fr - 0.152*(xs/Y₁)\]                 |
| Baffle wall with two gaps            | \[(\Delta E/E₁) = 0.9098 + (-0.3091 - 0.2413*(xs/Y₁))/Fr\]            |
| Wedge-shaped block                   | \[(\Delta E/E₁) = 0.4736 + 0.11786*Fr - 0.05984*(xs/Y₁)^2\]            |
Figure (17): Equation for baffle wall with gap and numerical data comparison

Figure (18): Equation for inverted L-Section and numerical data comparison

Figure (19): Equation for inverted T-Section and numerical data comparison
Figure (20): Equation for rectangular shape and numerical data comparison

Figure (21): Equation for baffle wall with two gaps and numerical data comparison

Figure (22): Equation for wedge-shaped block and numerical data comparison
5- Conclusion

In this paper, validation of a numerical model generated in FLOW-3D was undertaken by means of a comparison of Froude numbers between the numerical model and case study data, which indicated only about a 6% error rate for prediction of turbulent energy dissipation using suitable appurtenances. Based on this, the proposed numerical model in FLOW-3D is an effective tool to analyse and predict the energy dissipation, indicating that the turbulence models are valid. Six different disperser shapes were then subject to simulation of flow in stilling basin models; calibration was done between the models and the best model selected based on the ratio of maximum energy dissipation (ΔE/E1). The turbulent energy dissipation ratio (ΔE/E1) varied from 16% to 85%; in particular, the performance of the stilling basin in figure 12 is clearly better than that in figure 13, and the length of stilling basin can be shortened whenever the disperser is near the inlet pipe, making the stilling basin shorter and more economical. As seen in figure 8, the maximum ratio of ΔE/E1 also increases when values of Fr increase as a result of increased velocity, and the inverted L-section disperser shape gives maximum turbulent energy dissipation (ΔE/E1), while the rectangular wall shape gives the least. This may be because increasing the surface area causes skin friction to increase, dissipating more energy in the basin and enabling higher performance.

References:

[1] Y. K. Murthy and E. Divatia, “Behaviour of stilling basins in large spillways,” Irrig. power, 1982.
[2] A. A. Goel and D. V. S. Verma, “Improved design of energy dissipators for pipe outlets,” Irrig. Drain. Syst., vol. 13, no. 4, pp. 313–320, 1999.
[3] D. V. S. Verma and A. Goel, “Development of efficient stilling basins for pipe outlets,” J. Irrig. Drain. Eng., vol. 129, no. 3, pp. 194–200, 2003.
[4] A. Goel, “Design of stilling basin for circular pipe outlets,” Can. J. Civ. Eng., vol. 35, no. 12, pp. 1365–1374, 2008.
[5] A. Ghamari and K. Nekoufar, Optimum design of type I stilling basin, vol. 5. 2015.
[6] H. Babaali, A. Shamsai, and H. Vosoughifar, “Computational Modeling of the Hydraulic Jump in the Still Basin with Convergence Walls Using CFD Codes,” Arab. J. Sci. Eng., vol. 40, no. 2, pp. 381–395, 2014.
[7] S. Soori, H. Babaali, and N. Soori, “An Optimal Design of the Inlet and Outlet Obstacles at USBR II Stilling Basin,” Int. J. Sci. Eng. Appl., vol. 6, no. 5, pp. 134–142, 2017.
[8] H. Babaali, N. Soori, S. Soori, A. Mojtahedi, and A. Hamedi, “Static Pressure Estimation on Converging USBR II Stilling Basin: Numerical Approach,” vol. 7, no. 79, pp. 89–97, 2018.
[9] Z. Eshkou, A. A. Dehghani, and A. Ahmadi, “Forced Hydraulic Jump in a Diverging Stilling Basin Using Angled Baffle Blocks,” vol. 144, no. 2012, pp. 1–10, 2018.
[10] X. Wei-lin, L. Hua-sheng, Y. Yong-quan, and W. Chi-gong, “Numerical simulation of 3-D turbulent flows of plunge pool and energy dissipation analysis [J],” J. Hyrodyn., vol. 5, 1996.
[11] L. Lu, Y. Li, B. Teng, and B. Chen, “Numerical simulation of turbulent free surface flow over obstruction,” J. Hyrodyn., vol. 20, no. 4, pp. 414–423, 2008.
[12] Y. Wang, B. Wang, H. Zhang, Z. Wang, S. Zhou, and L. Ye, “Three-dimensional Numerical Simulation on Stilling Basin of Sluice in Low Head,” in 2016 5th International Conference on Civil, Architectural and Hydraulic Engineering (ICCAHE 2016), 2016.

[13] Pillai, N. N., & Unny, T. E. (1964). Shapes for appurtenances in stilling basins. Journal of the Hydraulics Division, 90(3), 1-21.

[14] Y. Wang, Z. Bao, and B. Wang, “Three-dimensional numerical simulation of flow in stilling basin based on Flow-3D,” Eng. J. Wuhan Univ., vol. 4, 2012.

[15] C. Jian-gang, Z. Jian-min, X. U. Wei-lin, and W. Yu-rong, “Numerical simulation of the energy dissipation characteristics in stilling basin of multihorizontal sub merged jet” J. Hydrodyn., vol. 22, no. 5, pp. 732–741, 2010.

[16] Shi Z.P. 2010. Numerical simulation of energy dissipater with low-head flood discharging structure. Northwest A&F University.

[17] Varjav Peyman, Khosravinia Payam and Salmasi Farzin (2008). “Evaluation of energy dissipation in stepped spillway using fluent numerical model”. Fourth National Congress on Civil Engineering, Tehran University, Iran.

[18] Flow Science, 2014, “FLOW 3D version 11 user manual"