International Journal of Engineering Science Invention (IJESI)
ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726
www.ijesi.org ||Volume 8 Issue 05 Series. IV || May 2019 || PP 43-50

Reproduction of Water Surface Profile Using ANSYS – CFD

Prasanna S V S N D L[1], Suresh Kumar N.[2]
[1, 2] (Department of Civil Engg., College of Engg (A), Osmania University, Hyderabad-500 007, India)
Corresponding Author: Prasanna S V S N D L

Abstract: *The numerical analysis is a vital tool that provides detail understanding in analyzing the characteristics of the flow field which is difficult through conventional methods. The objective of the present investigation is to plot the water surface profile for different discharges ranging from 9.8 lps to 12.5 lps. The gate openings adopted for the study are 0.025 m at a slope of 1: 1200 and 0.023 m at a slope of 1: 857 and 1: 705.8. The simulation results were evaluated and plotted by adopting User Defined Function (UDF). The C-programme was executed in the fluent solver to obtain all the various parameters appropriate for plotting the water surface profile at all the cells downstream of the gate. The water surface profile for the selected discharges is in consonance with the experimental results as well as analytical calculations.*
Keywords *- Water Surface Profile, Simulation, Fluent, Volume Fraction, UDF*

Date of Submission: 26-05-2019 Date of acceptance: 08-06-2019

I. INTRODUCTION

The classification of water surface profile in channels is helpful for thorough understanding of longitudinal variation of flow depth. The profiles are based on the relation between the actual depth of flow (y), normal depth (y_n) and the critical depth (y_c) as shown in Fig. 1. Turbulence is a pattern of fluid motion characterized by chaotic changes in pressure and flow velocity. This is caused by excessive kinetic energy in parts of a fluid flow, which overcomes the dampening effect of the fluid's viscosity. Turbulence is mainly characterized by few basic parameters viz., Irregularity, Diffusivity, Rotationality, Dissipation.

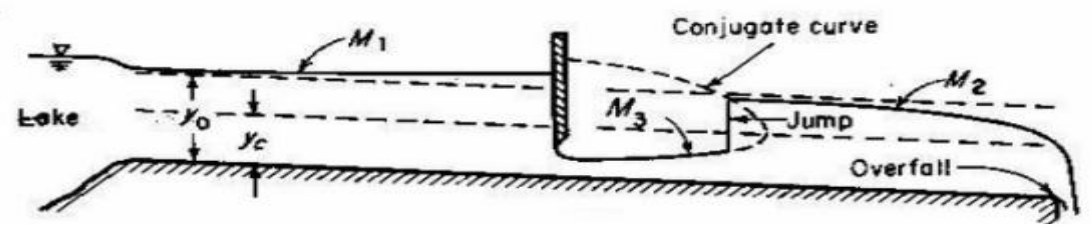

Fig. 1 sketch of water surface profile

Turbulence modeling involves the construction and use of a model to predict the effects of turbulence. These modeling techniques help in analysis and simulation of various civil engineering problems. Flow behavior that is difficult to comprehend through experimental analysis, can be easily understood by simulation. In addition, the physical modeling involves utilization of huge resources. Hence, the modeling techniques contribute to a large extent to study the effect and significance of diverse turbulence properties. The entire channel section is taken as the control volume for the analysis of any extensive property like mass, momentum, etc. of flow phenomenon and the continuum equations are solved by adopting Reynolds Transport theorem.

The flow distinctiveness can be provided through numerical techniques without resorting to experimental investigations. Further, the accompanied energy and turbulence can be analyzed by means of latest numerical simulations [1]. From the literature it is evident that, very few studies were available pertaining to the comparison of water surface profile downstream of the sluice gate. Most of the studies were observed to concentrate on the physical and analytical evaluation. The present investigation was carried out with the following objectives.

- Analytically estimate and experimentally measure and plot water surface profile for various flow conditions.
- Simulate and compare the experimental results adopting numerical modeling technique viz., ANSYS-CFD (Fluent) software.

II. Methodology

The present investigation is carried out in three phases viz., experimental, analytical and simulations as detailed in further paragraphs.

Experimental Observations:

The experiments were carried out in the Fluid Mechanics Laboratory of Department of Civil Engineering, College of Engineering (A), Osmania University, Hyderabad. The channel is of 12.0 m long prismatic tilting flume. The cross section of the channel is 0.5 m width and 0.6 m height. It is equipped with a turbulence arresting chamber on the upstream side having a length of 1.4 m. It gradually reduces from 1.0 m width to 0.5 m width, forming the entrance to the channel with a depth of 0.9 m. The plan for the experimental setup is shown in Fig. 2(a). The channel is having an aluminum base and acrylic side walls enabling a clear view of the formation of hydraulic jump. The two movable point gauges were mounted on the top of the channel sides by means of a wooden piece. A detachable sluice gate was also made from aluminum sheet. It was installed at a distance of 4.30 m downstream of the inlet gate provided in the flume. The discharge measurements were made volumetrically. The water surface levels in the channel were measured by using point gauges of 0.1 mm precision. The longitudinal section and the definition sketch of the present study are given in Fig. 2(b).

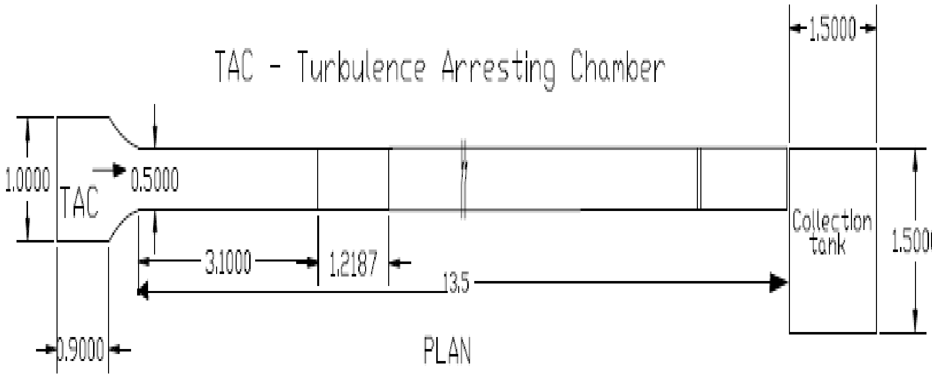

Fig. 2(a) plan of the laboratory channel setup

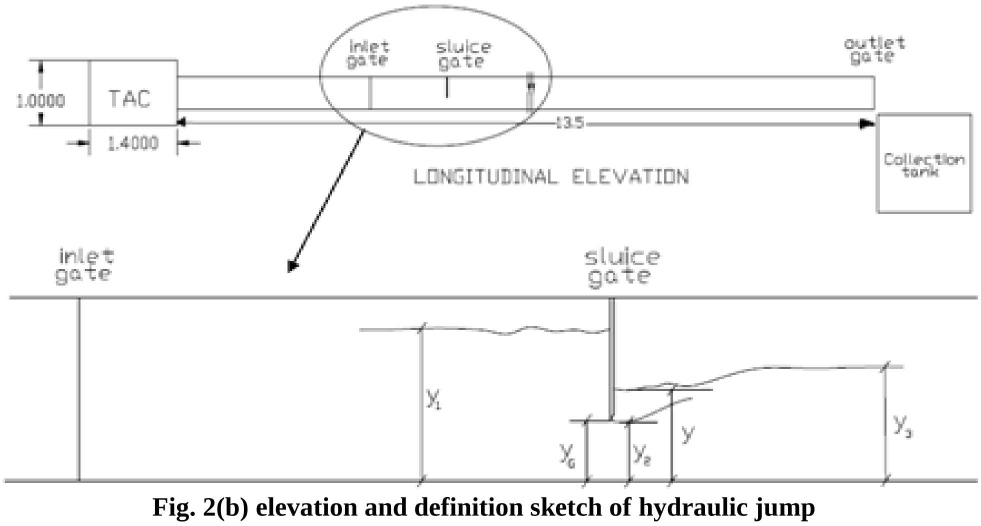

Fig. 2(b) elevation and definition sketch of hydraulic jump

The present investigation focuses on the experimental studies carried out for different discharges ranging from 9.8 lps to 12.5 lps. The gate openings adopted for the study are 0.025 m at a slope of 1: 1200 and 0.023m at a slope of 1: 857 and 1: 705.8. The water surface profiles for different discharges were observed as detailed in Table 1. The water surface for all the discharges were observed to comprise of M_3 profile. The water surface profile for a maximum discharge, at a slope of 1:1200 is depicted in Fig. 3.

Table 1 Experimental Values of Water Surface Profile

Observed Length of Profile	Observed Depths of Water Surface Profile	Observed Length of Profile	Observed Depths of Water Surface Profile
m	m	m	m
4.31	0.0213	4.739	0.058
4.371	0.0178	4.845	0.075
4.407	0.01319	5.1	0.0887
4.441	0.0126	5.418	0.0887
4.489	0.01062	5.113	0.0897
4.537	0.0105	5.428	0.0897
4.583	0.01191	5.448	0.0897
4.625	0.0135	5.443	0.0897
4.657	0.0217		

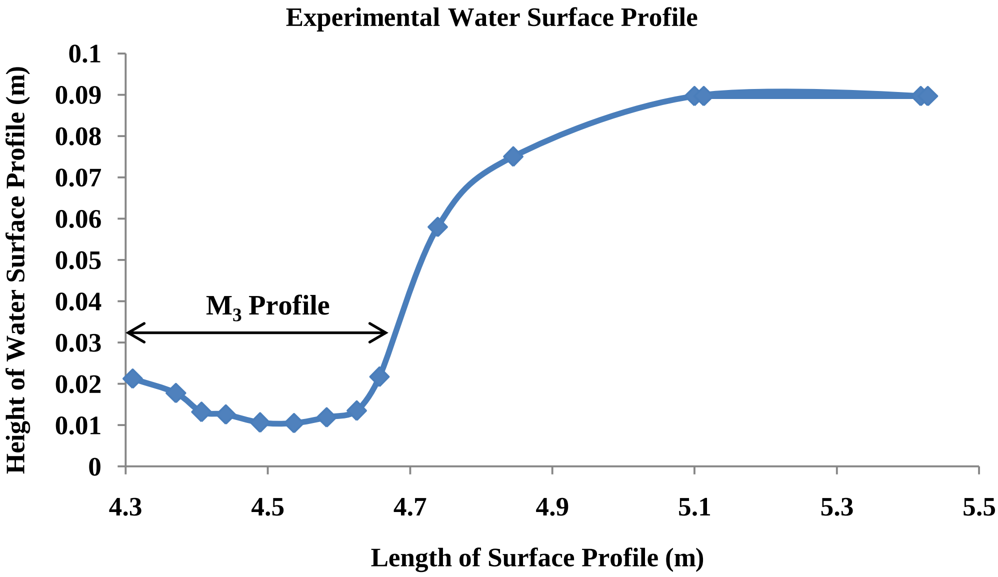

Fig. 3 observed water surface profile for 1:1200, y_G = 0.025m

Analytical calculations:
The analytical estimations were performed based on the following equations. The water surface profile was evaluated and plotted by the use of following equations [2].

$$H_1 = z_1 + \frac{V_1^2}{2g} + y_1 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots (1)$$

$$H_2 = H_1 + h_f \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots.. (2)$$

$$\left.\begin{aligned} h_f &= S_f * \Delta x \\ S_f &= \frac{n^2 * V^2}{R^{4/3}} \end{aligned}\right\} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots (3)$$

where,
H_1 = total head at the just downstream of the sluice gate, m
V_1 = velocity at the sluice gate, m/s
H_2 = total head at the point of the interest on the d/s of sluice gate, m
S_f = Slope of the energy line
R = Hydraulic mean depth, m

Δx = Change in distance between two sections, m
n = Manning's coefficient

The type of channel bed slope was estimated by estimating normal depth and critical depth as detailed in Table 2.

Table 2 Assessment of Type of Bed Slope

Discharge	**Normal Depth**	**Critical depth**	**Type of Channel Slope**	**Alternate Depth**
Q	**y_n**	**y_c**		**y_2'**
cumec	**m**	**m**		**m**
0.0118	0.063	0.03843	$y_n > y_c$ Mild Slope	0.1397
0.0113	0.061	0.03734		0.1291
0.0107	0.059	0.03601		0.1170
0.0104	0.058	0.03533		0.1112
0.0125	0.0585	0.03994		0.1755
0.0113	0.0550	0.03734		0.1456
0.0102	0.0515	0.03488		0.1207
0.0099	0.0505	0.03419		0.1144
0.0130	0.0565	0.04100		0.1755
0.0113	0.0515	0.03668		0.1456
0.0102	0.0484	0.03488		0.1207
0.0098	0.047	0.03396		0.1123

The M_3 profile starts theoretically from the upstream channel bottom, at either a vertical-angle slope or an acute angle, depending on the type of uniform-flow formula used, and the flow terminates with a hydraulic jump at the downstream end. This type of profile usually occurs when a supercritical flow enters a mild channel or when channel bottom slope changes from steep to mild. The beginning of the profile, although cannot be defined precisely by the theory, depends on the initial flow velocity. The flow profile will begin farther downstream, when the flow velocity is high.

In order to assess the type of bed slopes, normal depths as well as critical depths were computed for different flow rates adopted in the experimentation. Hence, to know the type of surface profile that is likely to develop, alternate depths corresponding to the pre jump depths (y_2') were estimated from the specific energy equation. Upon, comparing the values of y_2' with y_3 (post jump depth), it was concluded that M_3 profile is likely to exist for the flow conditions considered in the present investigation. The water surface profile as obtained by using eq. (1) to (3) is detailed in Fig. 4 for a bed slope of 1:1200 [3].

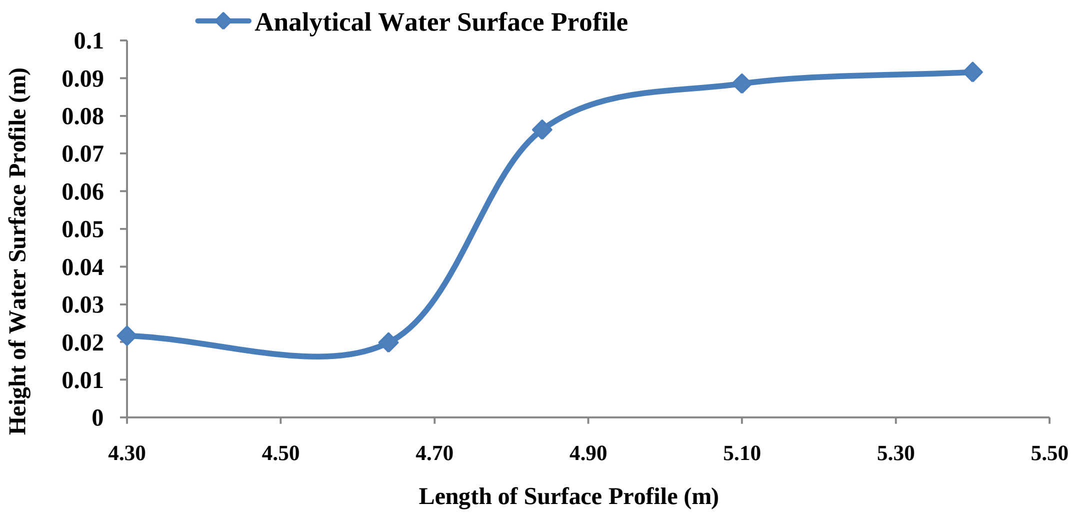

Fig. 4 analytical water surface profile (1:1200)

Numerical Simulation:

The commercial numerical modeling technique ANSYS – CFD (Fluent) uses a Finite Volume approach for solving Reynolds Averaged Navier Stoke equations. This software package is adopted in the present study to plot the water surface profile. As a first step in simulation, the geometry (2D and 3D) of the experimental setup was created and is shown in Fig. 5. Then, the meshing for the geometry was generated in ANSYS – Workbench for the channel geometry with the same gate openings of 2.50 cm and 2.30 cm. The mesh generated as per the fluent data base contains 2750 nodes and 2491 elements. The mesh is finer under the gate considering number of cell gaps as 15 and coarser towards the walls of the flume. The following boundary conditions were considered: velocity was defined at the inlet, no slip condition was applied along the walls, and velocity and pressure were computed at the outlet. The mesh generated in the workbench of ANSYS was imported for the simulation process.

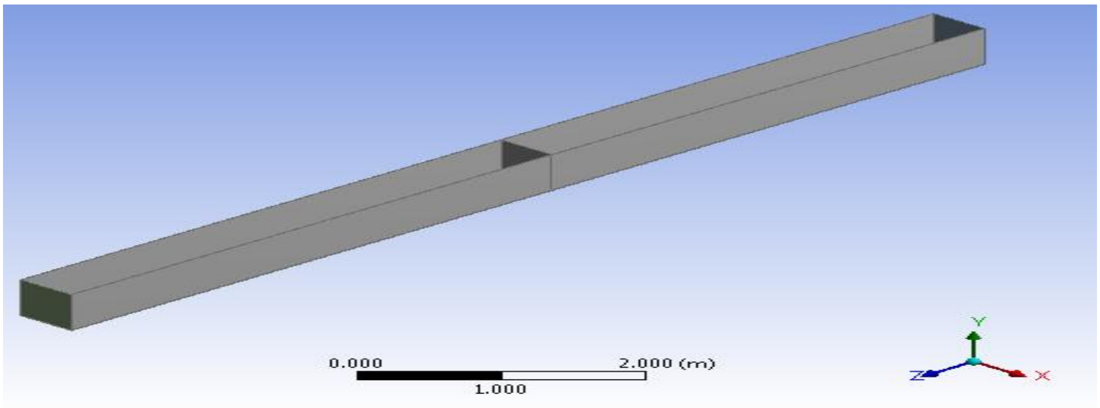

Fig. 5 3D - geometry of channel

The Volume of Fluid (VOF) method delivers the shape and location of constant pressure at free surface boundary. For the two dimensional steady state incompressible flow the Reynolds-averaged Navier-Stokes equations are given below [10]. The water surface profile was evaluated by means of User Defined Function written in C-Programming. The results were estimated by solving the basic governing equation of two-equation model of k-ε as highlighted in eq. (4) and k-ω in eq. (5) on each control volume of the flow region [4].

$$\left.\begin{aligned}\frac{\partial k}{\partial t}+U_j\frac{\partial k}{\partial x_j}&=\tau_{ij}\frac{\partial U_i}{\partial x_j}-\varepsilon+\frac{\partial}{\partial x_j}\left[\left(v+v_T/\sigma_k\right)\frac{\partial k}{\partial x_j}\right]\\ \frac{\partial \varepsilon}{\partial t}+U_j\frac{\partial \varepsilon}{\partial x_j}&=C_{e1}\frac{\varepsilon}{k}\tau_{il}\frac{\partial U_i}{\partial x_j}-C_{e2}\frac{\varepsilon^2}{k}+\frac{\partial}{\partial x_j}\left[\left(v+v_T/\sigma_\varepsilon\right)\frac{\partial k}{\partial x_j}\right]\end{aligned}\right\} \quad \ldots\ldots\ldots\ldots\ldots\ldots (4)$$

$$\left.\begin{aligned}\frac{\partial k}{\partial t}+U_j\frac{\partial k}{\partial x_j}&=\tau_{ij}\frac{\partial U_i}{\partial x_j}-\beta^* k\omega+\frac{\partial}{\partial x_j}\left[\left(v+\sigma^*\frac{k}{\omega}\right)\frac{\partial k}{\partial x_j}\right]\\ \frac{\partial \omega}{\partial t}+U_j\frac{\partial \omega}{\partial x_j}&=\alpha\frac{\omega}{k}\tau_{ij}\frac{\partial U_i}{\partial x_j}-\beta\omega^2+\frac{\sigma_d}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}\left[\left(v+\sigma\frac{k}{\omega}\right)\frac{\partial \omega}{\partial x_j}\right]\end{aligned}\right\} \quad \ldots\ldots\ldots\ldots\ldots\ldots (5)$$

The first term on the LHS of eq. (4) represents the rate of change of k or ε and the second term explains the transport of k or ε by convection. While the first term on the RHS symbolizes the rate of production of k or ε, the second term demonstrates the rate of destruction of k or ε and the third term illustrates the transport of k or ε by diffusion. The production and the dissipation terms of eq. (5) are formed from the production and dissipation terms of the turbulent kinetic energy scaled by ε/k and multiplied by empirically determined constants and wall damping functions (C_{e1} and C_{e2}). An additional damping function must be included for the eddy viscosity in the k-ε equation by near walls so that k and ε will have the proper behavior in the near region.

The Closure coefficients and auxiliary relations are given below: C_{e1} = 1.44, C_{e2} = 1.92, σ_k = 1.0, σ_ε = 1.3, $\omega = \varepsilon/(C_\mu k)$.

where, k - Kinetic energy of turbulent fluctuations per unit mass, U_i - Mean velocity in tensor notation, v_T - Kinematic eddy viscosity, v - Kinematic molecular viscosity, τ_{ij} - Specific Reynolds stress tensor

The Semi-Implicit Method for Pressure – Linked Equations (SIMPLE) algorithm, was adopted which is designed exclusively for turbulence simulations. Each trial run was solved for various equations such as Continuity, X - Velocity, Y - Velocity, k – equation and ε – equation leading to convergence. At each time step, a maximum of 10 iterations were taken to achieve the convergence. All the trials were observed to converge after 4500 iterations. The computation time ranges from 20.0 s to 30.0 s. The run time of 3.0 h was required to establish and stabilize the flow field for each velocity. The convergence criterion of 10^{-6} was adopted in the present investigation. The contours of multiphases viz., water and air, velocity profiles were plotted using

Graphics and Animations in fluent. The variations in multiphase analysis obtained from the simulations for two different phases of water (Red color) with a value of 1.0 and air (Blue color) assigned with the zero value are shown in Fig. 6. The water surface profile was also extracted from the UDF programme as represented in Fig. 7.

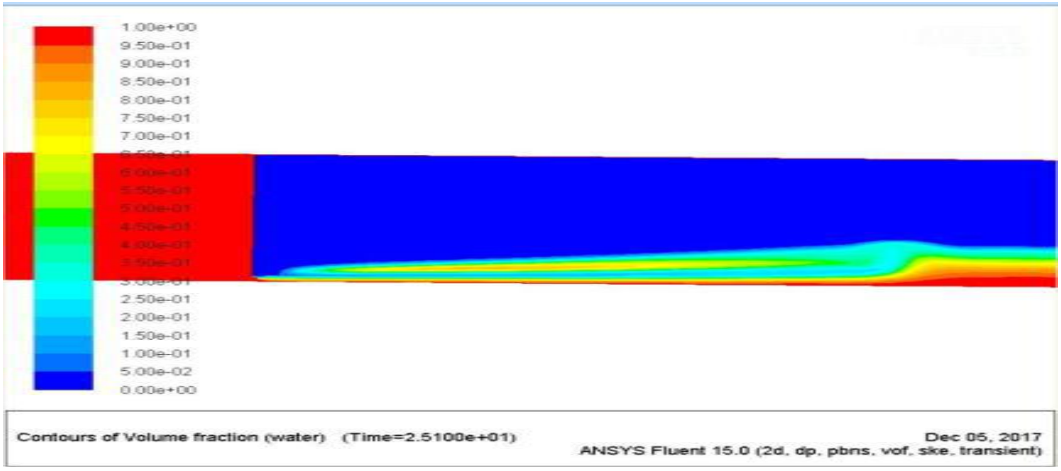

Fig. **6 contour of phases for k-ε turbulence model**

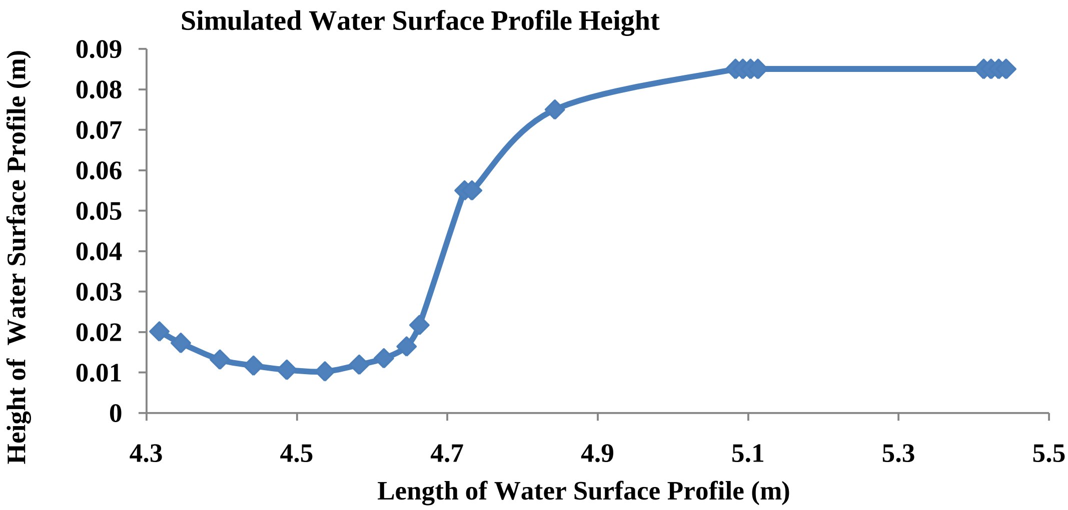

Fig. **7 simulated water surface profile for 1:1200, y_G = 0.025m**

III. RESULTS AND DISCUSSIONS

The following conclusions are drawn from the present experimental, analytical, and simulated analysis

1. The water surface profiles for various velocities as obtained from experimental analysis were also analyzed in Fluent. The values were obtained using a User-Defined Function, by the interpretation and execution of the macro generated mainly based on the multiphase analysis.
2. The results obtained from the fluent solver simulations with multiphase model for both k-ε and k-ω are in corroboration with the analytical estimations and experimental results.
3. The water surface profile plotted from analytical calculations, experimental observations and simulation results were found to be in fair agreement with each other for various slopes and gate opening as detailed in the Fig. 8(a) to 8(c).

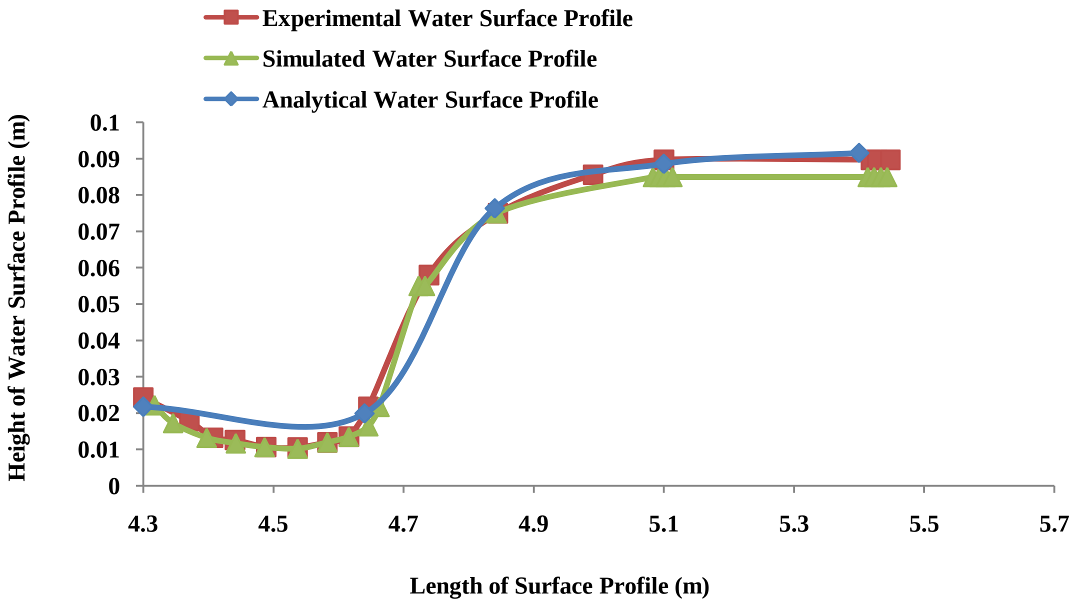

Fig. 8(a) comparative water surface profile for 1:1200, y_G = 0.025m

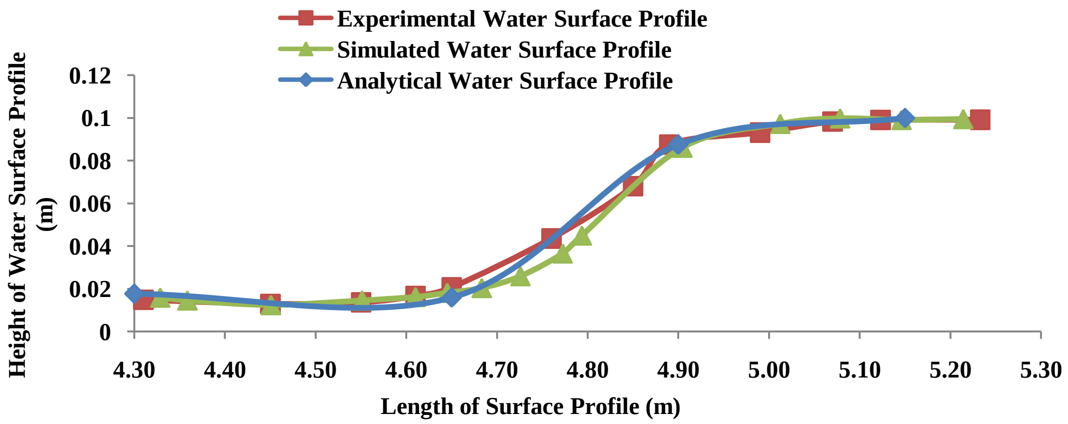

Fig. 8(b) comparative water surface profile for 1:857, y_G = 0.023m

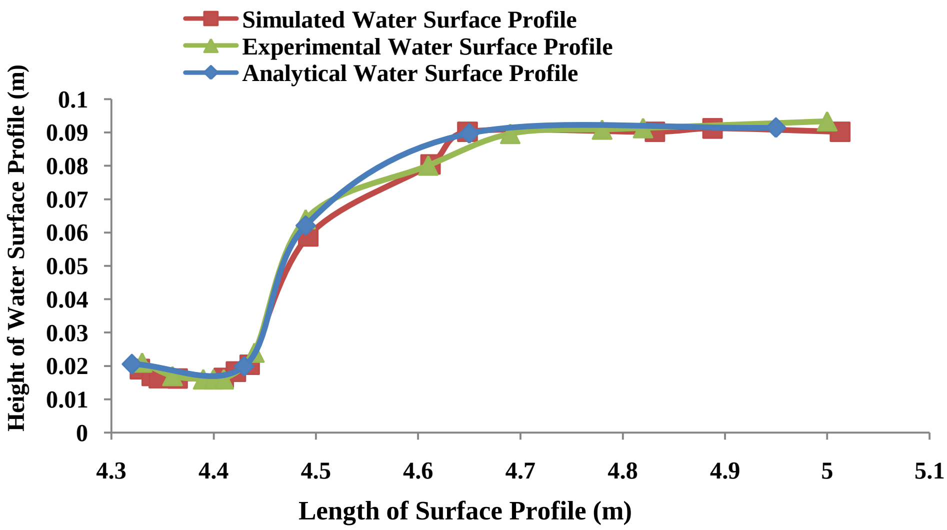

Fig. 8(c) comparative water surface profile for 1:705.8, y_G = 0.023m

REFERENCES

[1]. M. Hanif Chaudhry M., Applied Hydraulic Transients, (Van Nostrand Reinhold Company, New York, 1979), 332 – 363.
[2]. Chow Ven Te, Open Channel Hydraulics, McGraw-Hill Book Company, New York, 1959, 217 – 270.
[3]. Henderson, F.M., Open Channel Flow, Mc Millan Publishing Company, New York, 1967, 68 – 209.
[4]. Anderson John D. Jr., Computational Fluid Dynamics – The basics with applications, McGraw Hill Education (India) Pvt. Ltd., New Delhi, 2012, 3 – 94.

Prasanna S V S N D L" Reproduction of Water Surface Profile Using Ansys – Cfd" International Journal of Engineering Science Invention (IJESI), Vol. 08, No. 05, 2019, PP 43-50