FEM analysis of water surface profile using VOF Method

Avdhesh Kumar Sharma¹, Prashant Kumar Dixit², Shashank Srivastava³

¹Research Scholar, Department of Mechanical Engineering, Institute of Engineering & Technology, GLA University Mathura, UP, India -281406
²,³Department of Mechanical Engineering, Institute of Engineering & Technology, GLA University Mathura, UP, India -281406
corresponding author’s e-mail: sharma.avdhesh7979@gmail.com

Abstract. The prediction of water surface elevation in open channels is important for evaluation and determination of the heights of side wall of structures in open channel systems. At present empirical equations are basically used for the purpose. In the early half of the 19th century experimental approach was mainly used but it had the drawbacks of laborious data collection and instrument operation limitation. The 3D flow behavior or some complex turbulent structure which is the nature of any open channel flow, cannot be effectively captured through experiments, so in these circumstances, computational approach can be adopted to overcome these obstacles. In comparison to experimental studies computational approach is repeatable, can simulate at full scale, can generate the flow taking all the data points into consideration. The motive of the present work is to use the simple geometry of the rectangular broad crested weir to test the commercial CFD software ANSYS-CFX in order to test its feasibility for implementation in more complex open channel flows. The experimental results of Hager & Schwalt’s are validated and then the analysis has been done for weirs with different slopes.

Keywords: Computational Fluid Dynamics (CFD), VOF, Volume Fraction, Weir.

1. Introduction

These In order to evaluate and determine the side wall heights of structures in an open channel system the prediction of water surface elevation in open channels is quite important but the dependence on empirical equations is still in vogue and are given preference over any other method for the mentioned purpose. The physical model tests (investigating different flow conditions and geometrical setups) and numerical modeling (for free surface in open channel) may be the prominent methods for assessment of the effect of flows on water levels. In comparison to experimental studies the computational approach is repeatable; it can simulate at full scale and can generate the flow taking all the data points into consideration [1-3].

The study of free surface flow in channels play an important role in hydropower engineering, river modeling, irrigation engineering, flood mitigation etc. The prediction of free surface profile is play an important role in the design of various hydraulic structures such as measuring flumes, spillways, hydropower canals etc. The flow in open channel is affected by following parameters such as shape and the slope of the channel and the depth and velocity of fluid. The Interface capturing (makes use of advection of a function that represents the interface for simulating interface behavior) and tracking method (simulates the behavior of interface by deforming the elements) are the methods to carry out the
analysis of free surface flow modeling in CFD [4,5]. Although later method simulates the interface behavior more accurately as compared to earlier one but it requires regeneration of elements depending upon the fluctuations on interface which makes the method unstable. This instability can be avoided by increasing the number of elements; however, this increases the computational load. As compared to this the interface capturing method gives fairly accurate results without deforming the interface elements [6]. The Volume of Fluid (VOF) method for capturing the free surface variation is prominently in vogue because of the simplicity in coding (since interface elements need not to be regenerated) and stable calculations (even when there are fluctuations in interface).

In 1981, Hirt C. W. And Nichols B. D. [7] came up with volume of fluid (VOF) technique for the Dynamics of Free Boundaries as a simple and efficient means for numerically treating free boundaries embedded in a calculation mesh of Eulerian or Arbitrary Lagrangian-Eulerian cells. It is particularly useful because it uses a minimum of stored information, treats intersecting free boundaries automatically, and can be readily extended to 3D calculations. In 1999 Gueyffier Denis [8] introduced three-dimensional methods for interface calculations that can deal with topological changes, describe a numerical scheme, built from a volume of-fluid interface tracking technique that uses a piecewise-linear interface calculation in each cell [9, 10]. In 2007 Hargreaves D. M. [11] describes the validation of Computational Fluid Dynamics (CFD) for modeling free surface flows over common hydraulic structures. Different CFD simulations were compared against an existing set of experimental results for the free surface flow over a broad-crested weir which was given by the Hager and Schwalt (1991). He fixed the upstream and downstream waterheights in the CFD simulation and reproduces the analytical free surface profiles, velocity profiles and pressure profiles and discharges over the weir for different discharge rates. The use of turbulence model is very important for good results, he used different turbulence model for different cases. In 2010 Gandhi B. K.[12] investigated the different real flow conditions to determine the velocity profiles in both the directions in which field ideal flow condition rarely exists. A commercial CFD code ‘Fluent’ was used for numerically models of various situations. He investigated and discussed the effects of bed slope, convergence/divergence of channel width and upstream bend on velocity profile. It was observed that the actual velocity profile differs from the normal profile due to non-existence of ideal flow conditions. A lot more specialist have likewise seen the impact of utilizing nano-fluid and nano-composites in water surface profile [13-18].

2. Numerical Simulation

Initially results gave validation to the 3D simulations results of Hargreaves’s study on flow overbroad crested weir. Then the 3D simulation of the experimental work of Goodarzi for flow over broad crested weir with sloping weir surface was taken into consideration for four different cases, i.e. 10°, 30°, 60°, 90°. The dimension of the 3D open channel with the weir slope 90° which had been taken by Hargreaves for his simulation in FLUENT 6.2. Channel is 7m long, 0.5m wide & height of channel is 0.8m. The weir is at distance of 1.2m from inlet & 7m long, 0.4m wide & height of weir is 0.5m & the dimension of basic model of weir which was used by Goodarzi in his initial work was is 12m long, 0.25m wide & height of channel is 0.6m. The weir which is at a distance of 1.2m from inlet is 0.6m long, 0.25m wide & height of weir is 0.25m. Figure 1, shows the isometric view of Hargreave’s & Goodarzi’s model.

The flow domain was meshed using unstructured tetrahedral mesh. The mesh quality was checked and refined by using various quality parameters i.e. aspect ratio, skewness etc. The total no. of nodes and elements for Hargreaves model were 401628 and 2376592. The generated mesh is finer than that used by Hargreaves in his simulation model. The meshing of the model is shown in Fig. 2.
3. Mesh Generation

For simulating the experimental results of Goodarzi’s the mesh refined at the bottom of surface of the channel & weir surface for all the variants. The no. of elements in the different variants ranged from 0907007 to 2741765 while, the no. of nodes from 163453 to 464548. The meshing of weir with upstream slope 90°is shown by Figure 2. Simulation of free surface flows usually requires defining boundary and initial conditions to set up appropriate pressure and volume fraction fields. An inlet boundary where the volume fraction above the free surface is 1 for air and 0 for water and below the free surface is 0 for air and 1 for water. A pressure specified outlet boundary, where the pressure above the free surface is constant and the pressure below the free surface is a hydrostatic distribution. This requires the knowledge of the approximate height of the fluid at the outlet. Generally, an analytical solution for 1D flow over weir is used to determine the value for downstream height. The simulation is not sensitive to the exact outlet fluid height, so an approximation is sufficient.
4. Boundary Conditions

For inlet condition the velocity is taken 0.23 m/s which and for all other simulation the velocity is taken 0.35 m/s which were used in the experiment conducted by Goodarzi. For outlet condition pressure at the outlet of the channel was kept at static pressure, which had been calculated by the expressions used. Outlet boundary condition was taken by review papers and tutorials at the top surface of channel the relative pressure was taken 0 bars.

5. Result & Discussion

The 3D simulation has been performed for keeping the same upstream energy head and tail water level as taken by Hargreaves et.al for this simulation. The simulation was run until the variation in discharge at inlet and outlet sections became negligible. At the converged condition the inlet discharge was observed to be 68.57 kg/s while the outlet discharge was 68.67 kg/s making the error as 0.14%, while Hargreaves et.al had reported a validation of 10-20% in his study. The discharge reported in Hargreaves study for his study 3D simulation was 68.37 kg/s. Hence the variation in discharge values of present study as compared to that of Hargreaves et.al is 0.3% at inlet and 0.7% at outlet which is less than 1% hence the results can be considered to have been validated. Hargreaves et.al [19] has not given the water surface profile for his 3D simulation. However, the water surface profile obtained for the various 2D simulations performed by him which are in agreement of Hager & Schwalt experiment results are given in Figure 3. In this figure curve number 11 corresponds to the 2D simulation performed for the same geometry as used for his 3D simulation model. The water surface profile obtained from the present simulation study is shown in figure 4. It is seen clearly that the shape of water surface profile is similar to that obtained for the 2D simulations model of Hargreaves et al although the magnitudes differ. This is probably because of difference in 2D and 3D simulation. Hence as the discharge and shape of the predicted profile for the present model agree very well with that of Hargreaves result, hence the simulation model can be considered to be validated.

![Figure 3. Iso-surface profile for standard weir](image)

At standard weir (upstream slope 90°) - The 3D variation of flow profile over the weir obtained by plotting the iso-surface at 0.5 volume fraction of water is given in Fig 4 and the corresponding water
surface profile is shown in Figure 5. This water surface profile obtained from numerical simulation is in good agreement with that obtained by Goodarzi et al in their experimental study. Hence the simulation is validated. It is observed that the water surface profile has a very steep slope near the upstream end of the weir crest which reduces gradually towards the downstream end. This is probably due to the large circulation zone at the upstream end of the weir.

![Figure 4. Iso-surface profile for standard weir](image1)

![Figure 5. Water surface profile for standard weir](image2)

The streamlines pattern for flow over the weir (Figure 5) shows a large zone of stagnation with circulating flow at very low velocity just upstream of the weir at the base of the wall. On the downstream side of the weir the nappe is seen clearly as a zone of separation. The pressure contour also shows as a negative pressure zone at the downstream edge at the beginning of the nappe. These zones of circulating flows are created due to the failure of streamlines to follow the sharp and abrupt changes in geometry due to the vertical weir walls [20].
Effect of upstream slope on streamline pattern - Analysis of simulation results shows that decreasing the upstream slope reduces the stagnation of flow at the base of the upstream wall and makes the flow more uniform. At 60° minute separation of flow is seen at the starting of the slope, although circulation of flow is not observed. For angles below 45°, there is neither a stagnation zone nor separation of flow is seen at the starting of the slope, although separation of flow at the base of the upstream wall and hence the flow becomes progressing more uniform. The best flow conditions are seen at 10° slope. The separation zone on the weir crest at inlet edge is also not seen for weirs with sloping upstream face. The variation of water surface profiles is shown in. It is observed that as the slope of the upstream surface is reduced the steepness of the water surface profiles at the weir entrance also reduces with it. The decreased curvature of approaching flow tends to bring the profiles closer to horizontal streamlines above the weir crest [21].

Figure 6. Streamline profile for standard broad crested weir

Figure 7. Streamline pattern with upstream slope 10°
Effect of Upstream Slope on Velocity Profile - Analysis of the profile shows that for all the slopes the magnitude of velocity at the entrance edge of the weir crest is almost same, i.e. nearly equal to 0.4 m/s. At the downstream edge the magnitude of velocity reduces gradually from 2 m/s for 90° to about 0.9 m/s for 10°. This is because at higher slopes the water surface profiles are steeper causing the velocity to accelerate along the weir crest, this increases the magnitude. Considering the variation across the depth of flow it is observed that near the entrance of the crest the magnitudes of velocity are higher at the free surface as compared to that at the crest surface. However, this difference reduces as the slope is decreased. At the downstream end of the crest it is observed that the velocity profile shows a lesser value at the free surface as compared to that at the crest level. Moreover, the profiles show a wavy nature this is probably due to the presence of nappe and low velocity zone at the exit edge of weir.

6. Conclusion

The proposed study presents and evaluates a new analytical method which can be present work is aimed at testing the free surface modelling capabilities of the commercial software Ansys CFX-14.0. The flow over a rectangular broad crested weir was in fair agreement with experimental results validated the model with more complex geometries and flow configurations. The results of the study are discussed as follows:

1. The simulation results for free surface flow over rectangular broad crested weir are found to be in good agreement with Hargreaves et.al simulation work as;

2. The variation in discharge values of present study as compared to that of Hargreaves et.al is 0.3% at inlet and 0.7% at outlet which is less than 1%.

3. The water surface profile obtained was similar in shape to that of Hargreaves in his 2D simulation.

4. Water surface profile obtained for a standard weir (90°) is in agreement with that of obtained by Goodarzi et.al in their experimental study.

5. Analysis of flow pattern over a 90° weir shows large zone of stagnation with circulating flow just upstream of the weir wall and zone of separation on the downstream of the weir wall. The flow separates at the upstream edge of the weir crest and reattaches on the crest after a certain distance creating a small zone of separation.

6. The separation zone on the weir crest at the inlet edge is not taken into consideration for weirs with sloping upstream face.

7. At 60° & 45° minute separation of flow is seen at the base of the upstream wall but circulation is absent.

8. For angles below 45° there is neither the stagnation zone nor separation of flow at the base of the weir wall. Hence the flow becomes progressively more uniform with reducing upstream slopes.

9. As the slope of the upstream surface is reduced the steepness of the water surface profiles at the weir entrance also reduces. The decreased curvature of approaching flow tends to bring the profiles closer to horizontal streamlines above the weir crest.
10. The magnitude of velocity at the entrance edge of the weir crest is almost same (nearly equal to 0.4 m/s) for all the upstream slope values of the weir. The magnitude of velocity at the downstream edge reduces gradually from 2 m/s for 90° to about 0.9

References

[1] Cozzolino L., Morte R. Della 2014 A broad-crested weir boundary condition in finite volume shallow-water numerical models 12th International Conference on Computing and Control for the Water Industry, CCWI2013

[2] Csaba H, Kullmann L 2007 Numerical study on the free-surface channel flow over a bottom obstacle Journal of Computational and Applied Mechanics 8 57-70

[3] Feurich R, Niels R and Olisen B 2012 Finding free surface of supercritical flow numerical investigation Enng application of computational fluid mechanics 6 307-315

[4] Kumar A, Sharma K and Dixit A R A review on the mechanical and thermal properties of graphene and graphene-based polymer nanocomposites: understanding of modelling and MD simulation Molecular Simulation 46 136-154

[5] Saraswat M, Chauhan N R, Sharma V K, Shukla R K and Sharma K 2020 Assessment of Biomass Potential in Engine Emission Reduction Journal of Scientific and Industrial Research 79 77-80

[6] Singh PK, Sharma K, Kumar A and Shukla M 2016 Effects of functionalization on the mechanical properties of multiwalled carbon nanotubes: A molecular dynamics approach Journal of Composite Materials 51 671-680

[7] Hirt C.W. and Nichols B D 1981 VOF method for dynamics of free boundaries Journal of computational physics 39 201-225

[8] Gueyffier D, Jie L and Nadim A 1998 Volume-of-Fluid Interface Tracking with Smoothed Surface Stress Methods for 3D flow Modélisation en Mécname, CNRS URA 229, Université Pierre et Marie Curie, 4 place Jussieu, 75005 Paris, France.

[9] Gupta M, Sharma K and Saraswat M 2020 Heat Transfer and Pressure Drop Investigation in a Circular Tube by the use of Various Kinds of Inserts Journal of Scientific and Industrial Research 79 160-163

[10] Kumar A, Sharma K and Dixit A R 2020 Carbon nanotube-and graphene-reinforced multiphase polymeric composites: review on their properties and applications Journal of Materials Science 55 2682–2724

[11] Hargreaves D. M 2007 Validation Of The Volume Of Fluid Method For Free surface Engineering Applications of Computational Fluid Mechanics I 136-146

[12] Gandhi B. K 2002 Investigation of Flow Profile in Open Channels using CFD IGHEM-2010 AHEC IIT Roorkee

[13] Verma, S.K., Tiwari, A.K., Tiwari, S., Chauhan, D.S., 2018, “Performance analysis of hybrid nanofluids in flat plate solar collector as an advanced working fluid”, Solar Energy 167 231-241.

[14] Singh P. K.; Sharma K.; Kumar A.; Shukla M., 2017, “Effects of functionalization on the mechanical properties of multiwalled carbon nanotubes: A molecular dynamics approach”, Journal of composite materials 51(5).

[15] Singh P.K.; Sharma K., 2018 Molecular Dynamics Simulation of Glass Transition Behavior of Polymer based Nano-composites JSIR 77 (10) 592-595.

[16] Singh R.K., Sharma A.K., Dixit A.R., Mandal A., Tiwari A.K., 2017 Experimental investigation of thermal conductivity and specific heat of nanoparticles mixed cutting fluids Materials today: Proceeding 4(8) 8587-8596.

[17] Sharma A.K., Tiwari A.K., Dixit A.R., Singh R.K., 2017 Investigation into Performance of SiO2 Nanoparticle Based Cutting Fluid in Machining Process Materials today: Proceeding 4(2) 133-141.

[18] Yadav A.; Kumar A.; Singh P.K.; Sharma K., 2018 Glass transition temperature of functionalized graphene epoxy composites using molecular dynamics simulation Integrated Ferroelectrics 186(1) 106-114.

[19] Kumar A, Sharma K and Dixit A R 2019 A review of the mechanical and thermal properties of
graphene and its hybrid polymer nanocomposites for structural applications. *Journal of materials science* **54** 5992-6026

[20] Singh RP, Sharma K and Tiwari A K 2020 An Experimental Investigation of Thermal Conductivity of TiO2 Nanofluid: Proposing a New Correlation *Journal of Scientific and Industrial Research* **79** 160-163

[21] Yadav A, Kumar A, Singh P K and Sharma K 2018 Glass transition temperature of functionalized graphene epoxy composites using molecular dynamics simulation *186* 106-114.