Scientific Journal Impact Factor (SJIF): 5.278
International Journal of Modern Trends in Engineering and Research
ISSN (ONLINE) : 2349 - 9745
ISSN (PRINT) : 2393 - 8161

CFD ANALYSIS OF OGEE SPILLWAY HYDRUALICS

Dolon Banerjee[1] and Dr. Bharat Jhamnani[2]
[1]*M.Tech Student, Department of Civil Engineering, Delhi Technological University*
[2]*Professor, Department of Civil Engineering, Delhi Technological University*

Abstract— This piece of work aims to use CFD modelling to validate the phenomenon of the flow over a ogee spillway model in laboratory. Since this simulation consists of the pure interaction of the water flowing through the spillway and the air present in the surrounding, hence we use Volume of Fluid (VOF) to simulate the free surface of the flow which is having the two phases: Air and Water. Then we have used the most versatile turbulence model "k- ε" for the simulation of the flow turbulence. For CFD modelling, a commercially available Computational Fluid Dynamics (CFD) package, Ansys-Fluent, was used. In order to get the exact geometry of the physical model, the use of Reynolds-Averaged Navier-Stokes(RANS) equations in combination with the realizable "k-ε" eddy-viscosity closure model was adopted. The procedure of CFD model development and the theory related to those model parameter selection are also discussed in this work briefly. We have simulated the pressure readings in steady state and fully hydrodynamic stages. Besides this the level of water along the spillway profile and the water surcharge at the crest produced by numerical models are also presented here. Finally, we have carried out a validation process of the results of CFD model in 2D and 3D by comparing them with the results obtained from physical models. From this comparison it was proved that CFD modelling of the hence experimented ogee spillway was in very good accordance with the test data and only slight differences were revealed between them. Hence CFD modelling was accurately able to model the two phase system along with all the real life conditions originated during the test.
Keywords— Ogee Spillway, CFD-Ansys Fluent, VOF(Volume of Fluid), "k-ε" turbulence model, Water Surcharge

I. INTRODUCTION

In recent times numerical modeling techniques have been rapidly evolved and they are now developed to such an extent that they now can be used for many different studies and hence these are now preferred as a standard design tool in many engineering disciplines. Even though they can be used for a wide range of problems but their basic remains the same. First of all there is a problem whose solution is required. Now this problem is denoted as a set of Partial Differential Equations (PDE). After getting the PDEs, we use Finite Volume Analysis or Finite Element Analysis to change the PDEs into a set of Algebraic Equations. Now the solution to these algebraic equations is calculated using either an Iterative Method or by using Matrix Method but in both these cases the solution is often very computationally intensive and hence we use the Numerical Models to solve these without the tedious solving procedure. Apart from saving time and energy these numerical models are less pricey than physical models because they do not require any space or material or construction and they can be easily customized to any design changes made later on in the process. All we require for performing simulations is a computer, any simulation software and the engineering knowledge to interpret the results obtained. Also Flow 3D has the benefit that it can ignore the air contiguous to the flowing water by using Volume of Fluid(VOF) method invented by Hirt and Nichols[4]. In this work we have derived the physical model experiment data from a previously performed Laboratory tests and used it to validate our CFD modelling as we being the Hydraulics Engineers are interested in CFD and its ability of simulating different flow conditions as found by Gessler[3]. Even though CFD is mostly utilized for modelling flow in many different areas but in this study we will be focusing on the use of CFD to model the flow of water through an ogee spillway.

Spillway is a hydraulic structure catered at the dam to discharge floodwater that cannot be held safely in the reservoir. To get a hydraulically efficient spillway we need to check on its Discharge, Water Surcharge, Pressures generated and quantify them into safe limit. These parameters further are dependent on type of spillway. As we know there are several types of spillways but the ogee-crested spillways are the most common because they have the ability to release excess water from upstream to downstream proficiently when properly designed and implemented[5]. These ogee type spillways have larger water holding capacity, higher hydraulic agreements, and are easily adaptable to all type of dams.

Kjellesvig[7] carried out the numerical modelling of flow over a spillway in 2D and 3D for several geometries and found that the discharge and pressure were in good consensus with physical test results. Johnson and Savage[6] estimated the discharge and crest pressures over an uncontrolled ogee spillway using a FLOW-3D. Then those results were compared to physical model results data and was found to be within limit of physical model study and the USBR data up to $0.1H_d$ to $0.7H_d$. But for heads higher than this, up to 1.2 times H_d, the Flow-3D results showed slight over-prediction in discharge, however, the deviation remained within 1% of the physical model results. Azmoudeh and Kamanbedast[1] simulated the best location of aeration system to cease cavitation and found it to be in accordance with physical model. Further Bhajantri[2] also conducted the formation and development of a 2-D free surface flow mathematical model for flow over a spillway and found good agreement with physical test results. Zhenwei[9] studied about the flow over a spillway using CFD and chose hexahedral grid with “k-ε” model and again got fine similarity in both model results .

II. NUMERICAL METHOD

In the field of hydraulic engineering, computational modelling of spillway flows has been very rapidly gaining popularity but a validation with a physical model test results is required to ensure competency. Several approaches have been established, including modelling in 1, 2 or 3 dimensions which use various equations and discretization methods. Here we will be dealing with 2-D and 3-D simulations only.

2.1 Setup physics

The physical modeling referred here consisted of one ogee spillway installed, in a glass flume of 1.25 m high, 0.50 m wide and 22 m in length. The maximum flow rate obtained in the laboratory from a constant head tank was 130 l/s. The laboratory Setup is shown in Figure 1 below. The spillway was a 1:30 size model of a large dam.

The Model Dimensions along with other hydraulic parameters are tabulated in Table 1 and Table 2

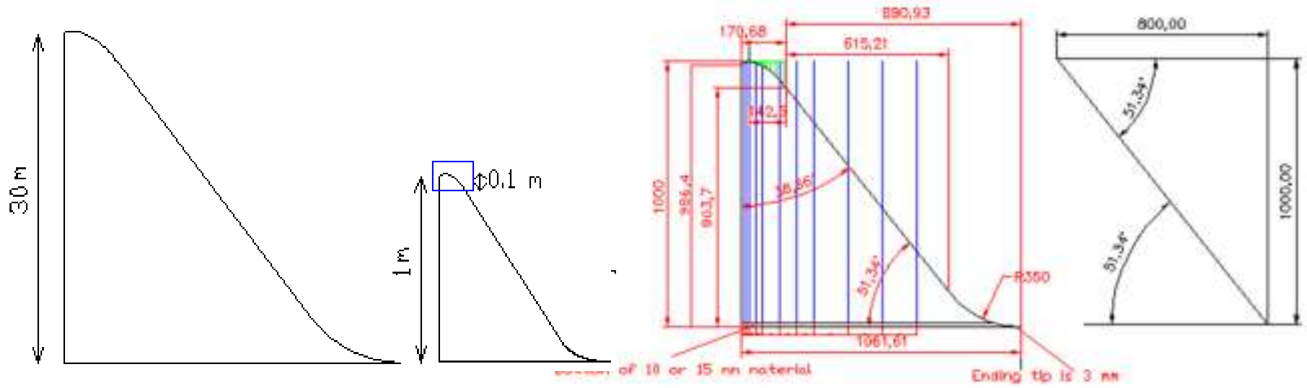

Figure 1: Physical model geometry of 1:30 scale model

Table 1: Model dimensions

Model Type	Spillway Approach Depth	Crest Depth	Radii	
1:30 scaled	1.00	0.50	0.06(R_1)	0.03(R_2)

Table 2: Design Surcharge, coefficient of discharge and design discharge for model

Model Type	Design Surcharge(H_o) in m	Coefficient of Discharge(C_o)	Design Discharge(m^3/s)
1:30 model	0.20	2.10	0.045

2.2 Numerical setup & meshing

The laboratory setup shown above was designed in the CFD simulation package and the 2-D and 3-D geometry were made as shown below:

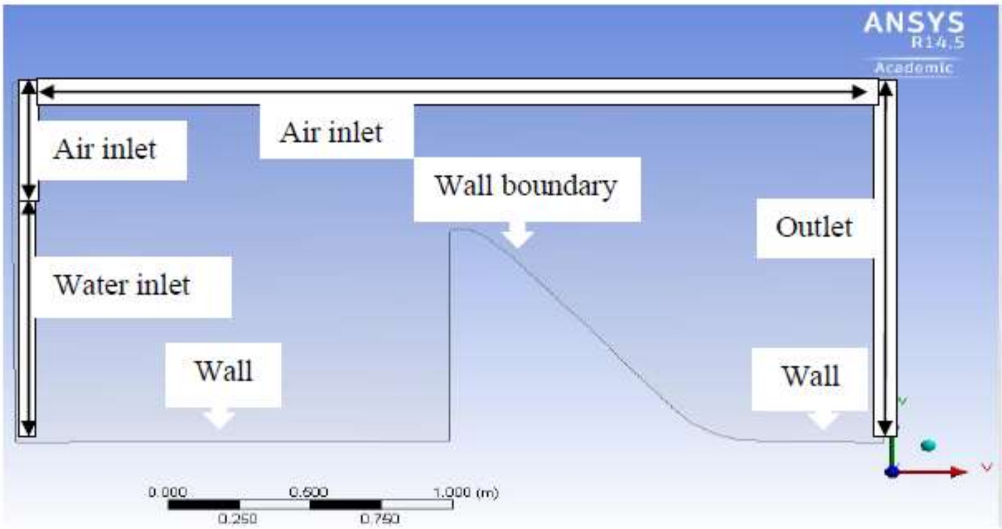

Figure 2: Model geometry constructed in 2-D with boundary labels

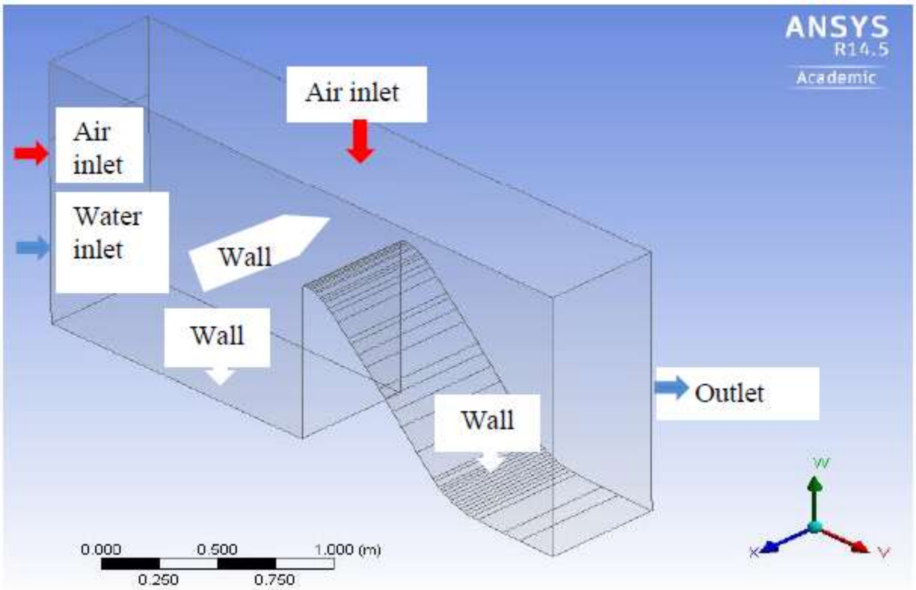

Figure 3: Model geometry constructed in 3-D with boundary labels

Now for the meshing part due care is required as this part decides the accuracy of the simulation. The correctness of a CFD solution is influenced by the number of cells contained by the mesh i.e. the finer is the meshing, the better is the solution accuracy. A number of trials with smaller and non-overlapping computational cells were performed to obtain the most suitable mesh so that a better solution accuracy could be achieved. The areas with highest concerns were meshed with the finest cells while others were relatively coarsely meshed. Triangular meshing was used in this case because of its superiority in accuracy of results. Also at the spillway downstream refinement was done. Figure 4 and Figure 5 show the meshing in 2-D and 3-D respectively.

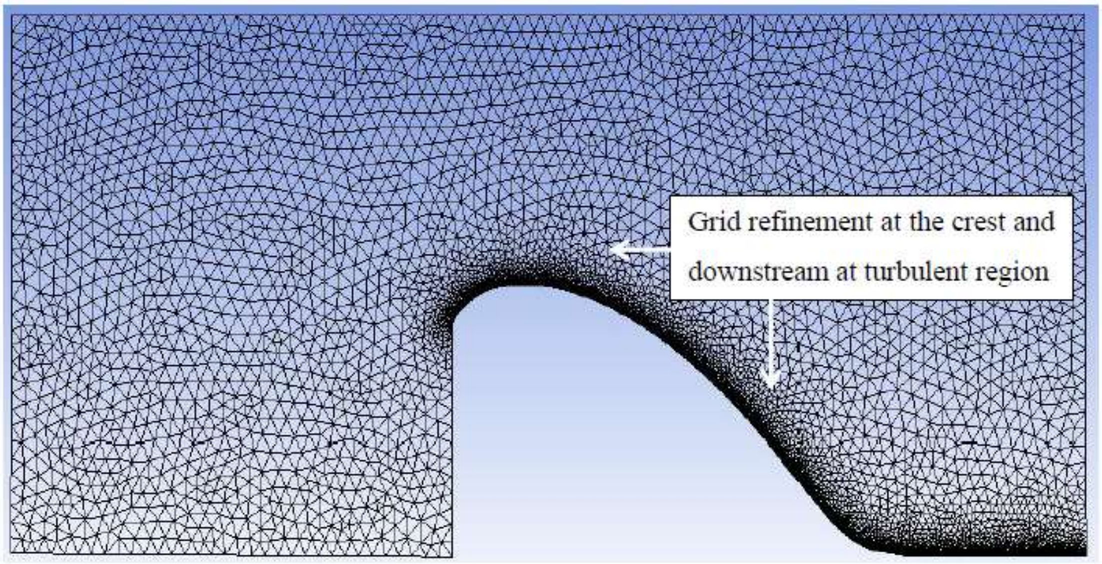

Figure 4: Meshing geometry of 2D model

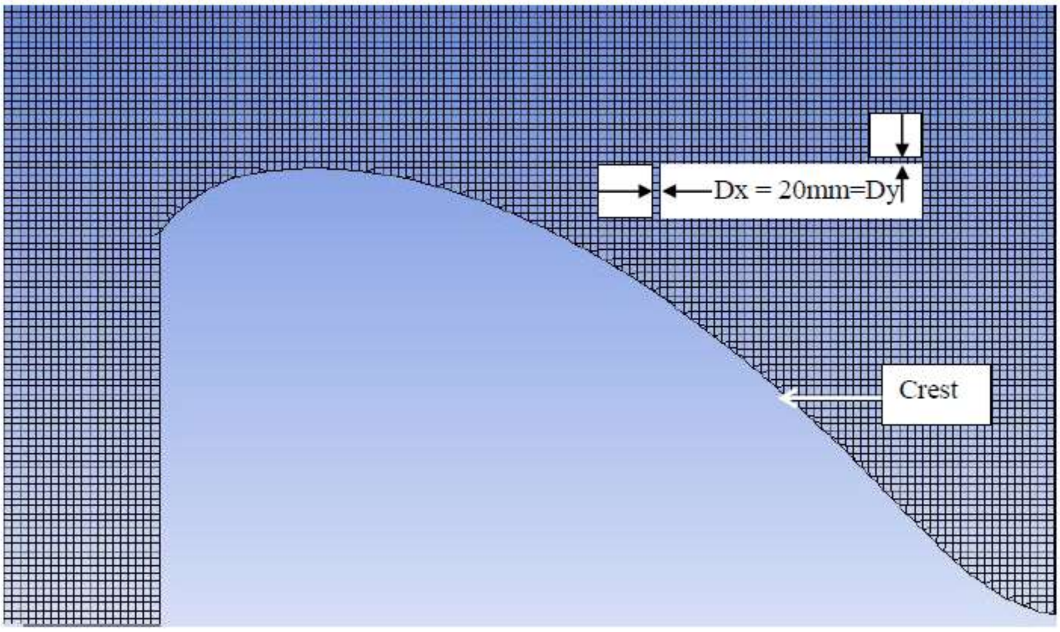

Figure 3-5: Meshing geometry of 3D model

2.3 Solver

For this work the realisable "$k-\varepsilon$" model was used. This model was used because of its latest turbulent viscosity formulation and latest transport equation for the Kinetic Energy (K.E) dissipation rate as studied by Singh[8]. For carrying out the multiphase modelling we have used the VOF (Volume of Fluid) in which the two Eulerian Phases were Air as Primary Phase and Water as Secondary Phase.

2.4 Boundary conditions

This is another important aspect of simulation as it controls the whole phenomenon going on in the geometry and directly affects the outcomes of the process. Since walls and internal faces have direct interaction with the water so they are also given appropriate boundary conditions.

A. Inlet

The inlet is defined at the upstream side of the spillway with pressure boundary as Atmospheric Pressure and since there is water entering at bottom and air at the top so to simulate this situation Velocity-Inlet was given at inlet.

B. Oulet

At the outlet to get the atmospheric condition, outlet boundary was defined and the flow was allowed to flow back into system.

C. Walls

For walls a no-slip condition with stationary throughout the test was used.

After defining these conditions initialization of the simulation was done under steady condition till the solution showed convergence. Further 2D and 3D tests with discharge of 130 l/s was conducted for both steady and fully hydrodynamic stages. In steady state stage, the convergence is monitored while for the fully hydrodynamic condition the test was performed for a longer time i.e. 5 minutes to get the fully developed flow with less fluctuations but reading showed that after a short time interval there was no fluctuations in pressure so finally the time for this was fixed at 20 seconds.

2.5 Simulation outcomes

A. Density contours and pathlines

In this work we have used Density contours and Pathlines to show the flow pattern of water through the spillway. As shown below, water (dark blue color) and air (red color) has a density of 998.2 kg/m^3 and a density of 1.225 kg/m^3 respectively while the yellowish color in between represents the interface of water and air and also signifies the aeration effect.

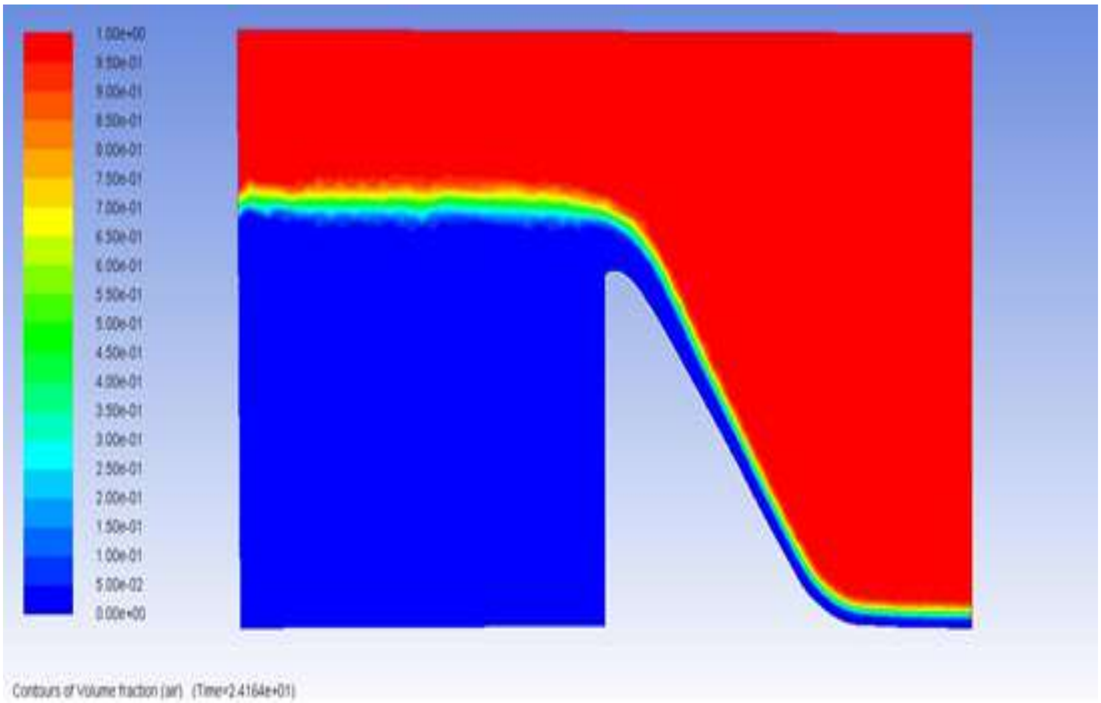

Figure 6: Density contours simulated by CFD model for 130 l/s

For better understanding of flow aeration Path line diagram can be used. Here the red band on the water surface shows the aeration which signifies that the aeration of the flowing water continuously increases as it flows through the spillway profile.

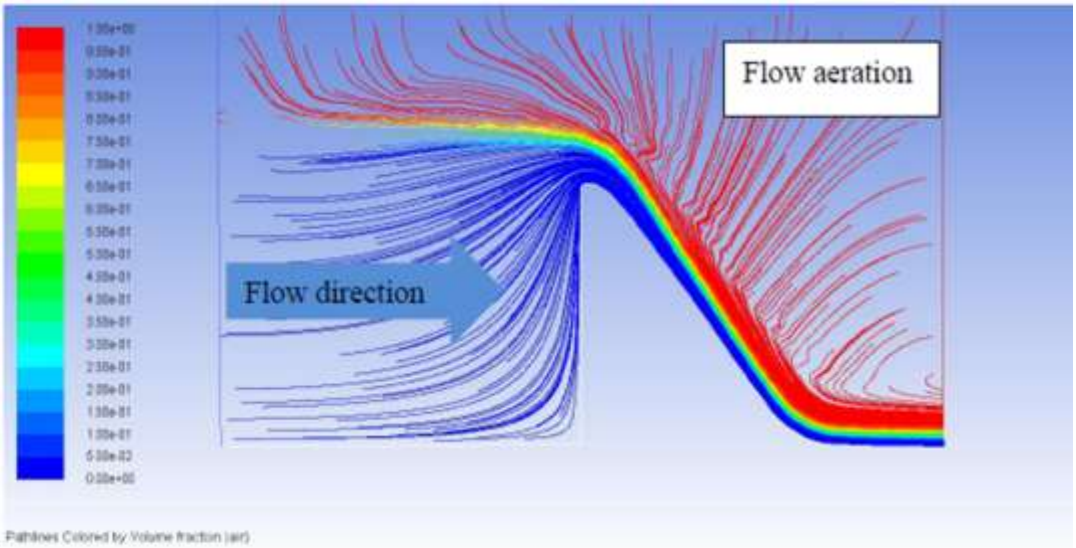

Figure 7: Simulated path lines of particles by volume fraction of air

B. Water surcharge

The water surface profile and the surcharge over the crest were also measured by the help of multiphase modelling. Both these parameters were measured at the centerline of spillway to avoid effect of spillway boundaries. The steady and fully hydrodynamic flows showed nearly the same water surface profile. Also it was noticed that the at crest the upstream side had higher water level than downstream side hence formation and thickening of boundary layer took place there along with increase in velocity.

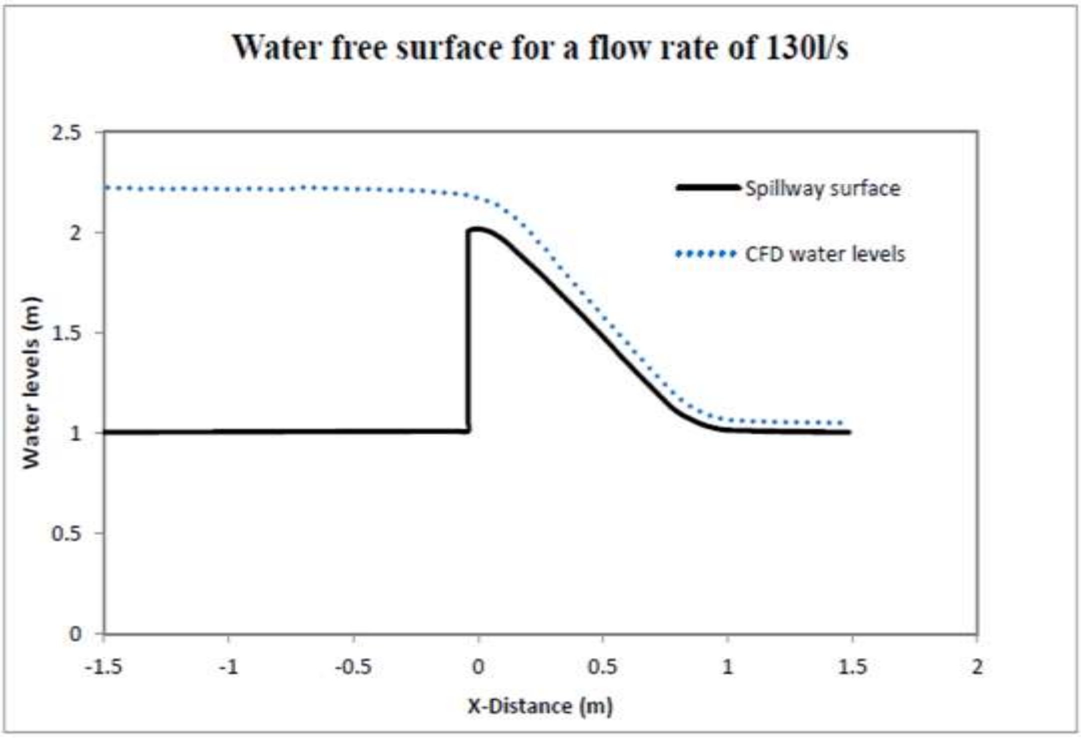

Figure 9: Water free surface over spillway model

C. Pressure results

Pressure results of both steady and fully hydrodynamic case were measured for both 2D and 3D cases. To get these in case of steady state 2500 iterations were used while for fully hydrodynamic case 20 second time was used.

Table 3: CFD simulated steady state pressure results for 2D model

Flow Rate(l/s)	Sensor 1	Sensor 2	Sensor 3	Sensor 4	Sensor 5	Sensor 6	Sensor 7
	CFD modelling: Average Sensor pressure (m)						
130	-0.118	-0.034	0.009	0.041	0.017	-0.031	0.036

The sensors were installed at the physical model along the profile of the spillway starting from the crest and then downward.

Table 4: CFD simulated fully hydrodynamic pressure results for 2D model

Flow Rate(l/s)	Sensor 1	Sensor 2	Sensor 3	Sensor 4	Sensor 5	Sensor 6	Sensor 7
	CFD modelling: Average Sensor pressure (m)						
130	-0.120	-0.034	0.009	0.040	0.017	-0.029	0.036

Table 5: CFD simulated steady state pressure results for 3D model

Flow Rate(l/s)	Sensor 1	Sensor 2	Sensor 3	Sensor 4	Sensor 5	Sensor 6	Sensor 7
	CFD modelling: Average Sensor pressure (m)						
130	-0.072	-0.038	0.012	0.046	0.030	-0.004	0.044

Table 6: CFD simulated fully hydrodynamic pressure results for 3D model

Flow Rate(l/s)	Sensor 1	Sensor 2	Sensor 3	Sensor 4	Sensor 5	Sensor 6	Sensor 7
	CFD modelling: Average Sensor pressure (m)						
130	-0.073	-0.035	0.012	0.047	0.031	-0.003	0.044

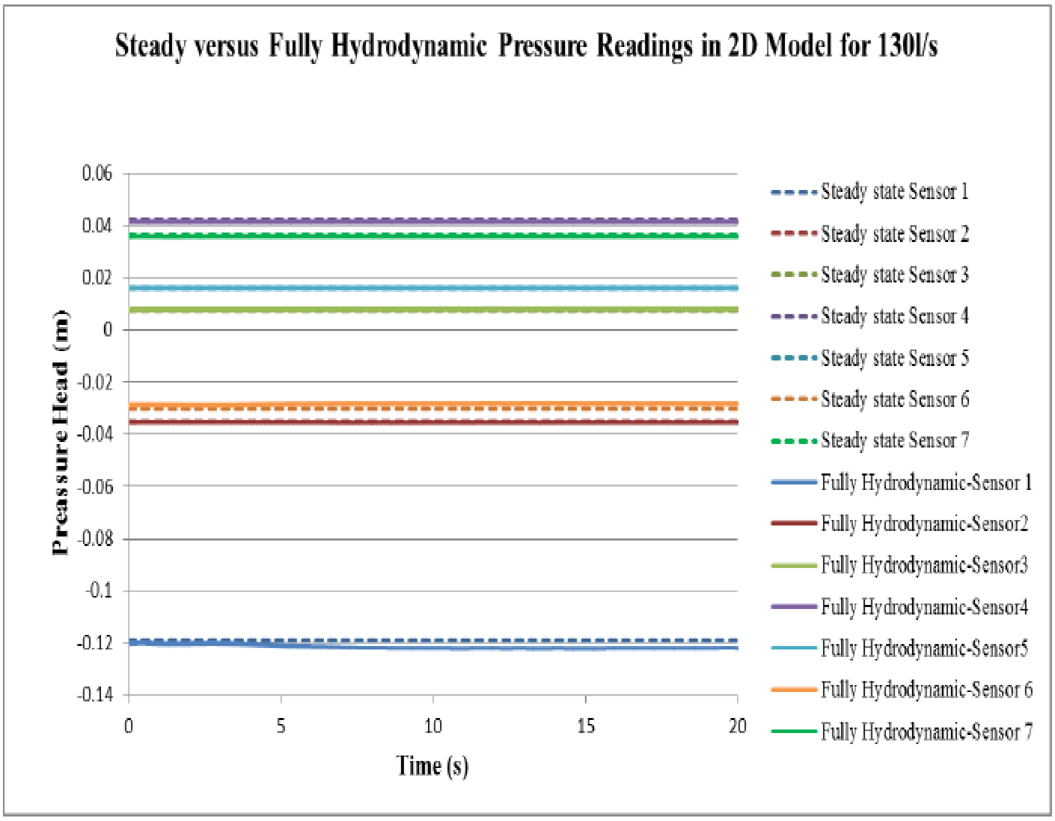

Figure 11: Comparison between 2D steady and fully hydrodynamic state models

Volume: 5, Issue: 07, [July– 2018] ISSN (Online):2349–9745 ; ISSN (Print):2393-8161

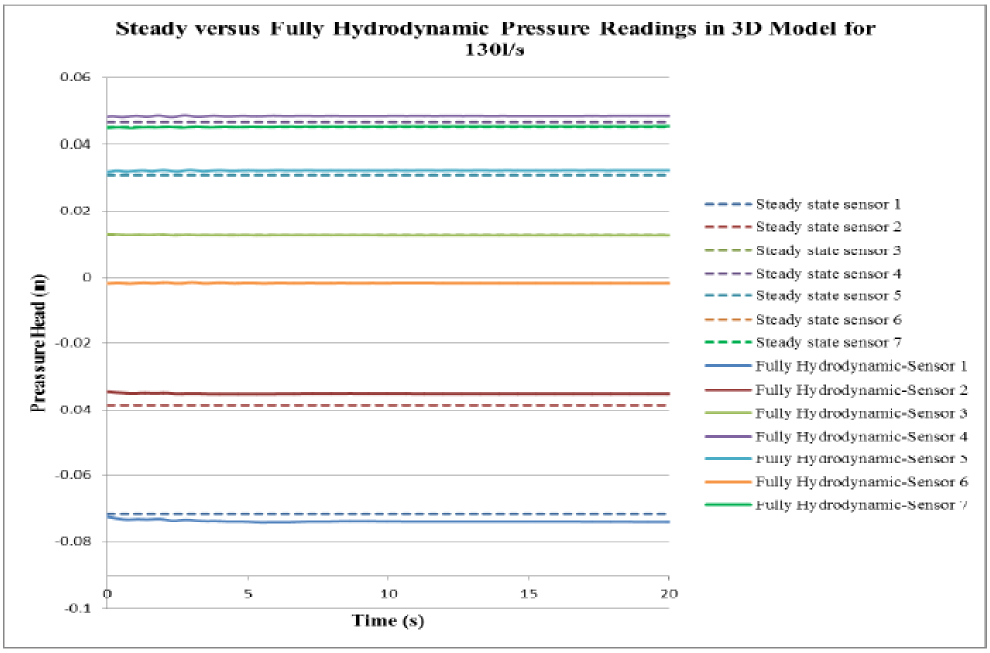

Figure 12: Comparison between 3D steady and fully hydrodynamic state models

III. VALIDATION OF CFD SIMULATION OUTCOMES

3.1 Water surcharge

For this section we will be comparing the Water Surface profile along the spillway profile and the Water surcharge upstream of the crest. To get the graphical representation of the water surcharge we have considered a number of diccharges and from the Figure 13 we can quantify that maximum deviation of surcharge is seen at highest discharge i.e. 130 l/s and is of 14 mm only hence validating the great similarity between both the model readings.

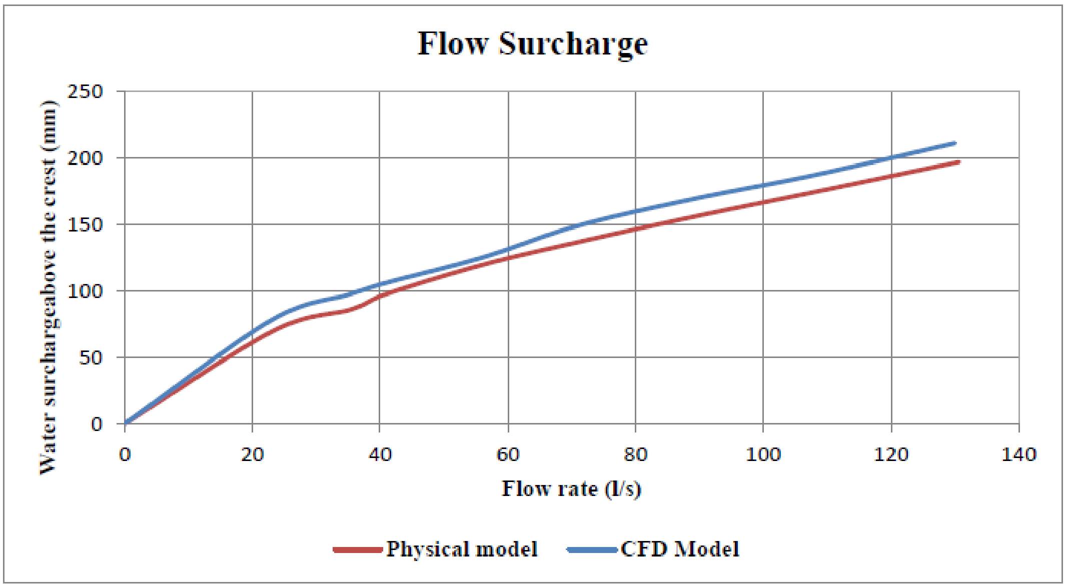

Figure 13: Water Surcharge of CFD Model and Physical Model

Secondly we look into the water surface profile and as shown in Figure 14, we can clearly see that the water free surface simulated by CFD models are in good agreement with those measured in physical modelling hence it can be noted that the CFD models can simulate the surcharge of flow for ogee spillways consistently.

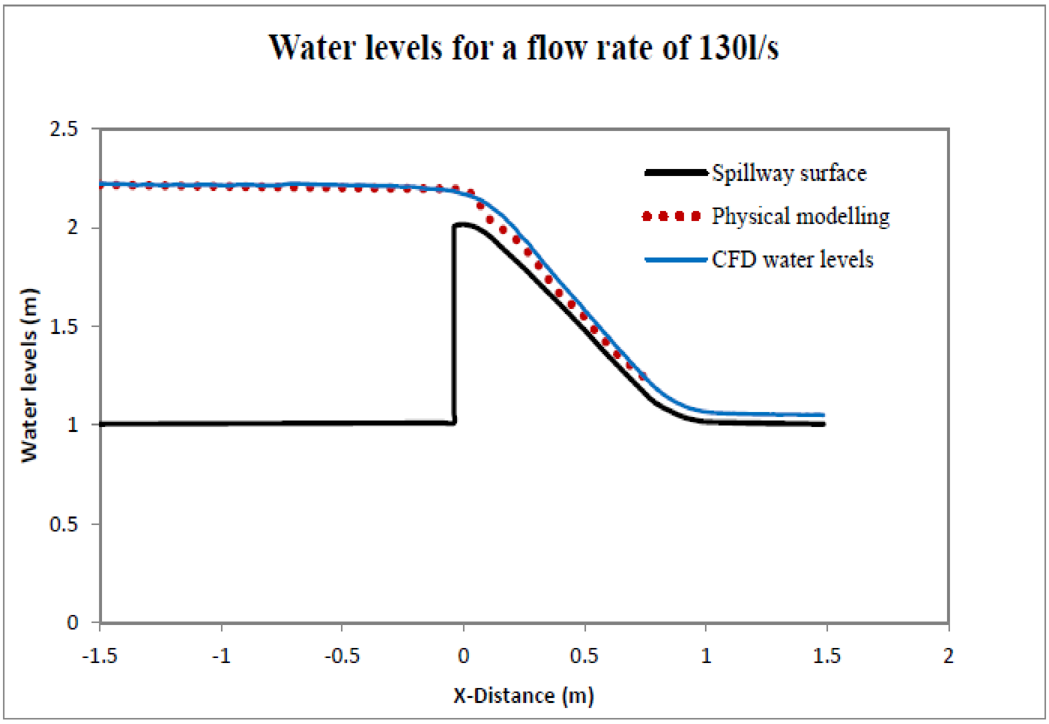

Figure 14: Free water surface of CFD Model and Physical Model

3.2 Pressure results

As already stated the steady state and fully hydrodynamic state have very similar pressure results. So here we will compare the 2D and 3D results of pressure with the Physical Model results.

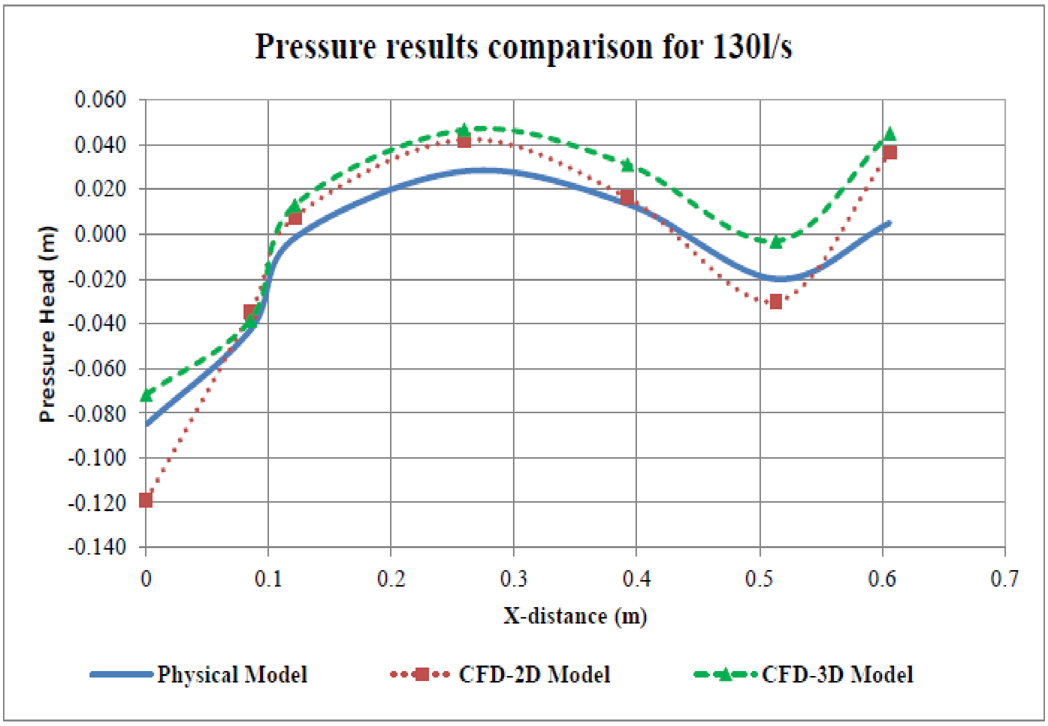

Figure 15: Comparison of 2D, 3D and Physical pressure results

As seen from Figure 15, there is very minute difference between the 2D Model and Physical Model and the 3D Model has larger deviation but the magnitude of this difference is very low and so it can be said that the CFD pressure results confirm to Physical Model results.

Volume: 5, Issue: 07, [July– 2018] **ISSN (Online):2349–9745 ; ISSN (Print):2393-8161**

IV. CONCLUSION

The flow over an ogee spillway was simulated in this work and compared with the pre obtained physical test results. We here compared mainly the Pressure results and the Water Surcharge results. After the comparison we saw that the in case of pressure, a Negative pressure was obtained when the discharge was greater than Design discharge with a flow separation over the crest and a Positive pressure was obtained for discharge less than Design discharge. Also in case of Water Surcharge and Water Surface Profile we saw a minimal difference of 14 mm in their profile obtained from CFD and Physical tests.

Hence the meshing done in 2D case i.e. Triangular meshing with refinement at lower part proved to be a very successful tool for simulating the phenomenon. Finally we can say that the Multiphase modelling chosen here which was Volume of Fluid (VOF) was able to show the Water surcharge and Water surface profile accurately and also was able to simulate the Pressure with high conformity to the Physical test data.

REFERENCES

[1] Azmoudeh, M.H.E., Kamanbedast, A.A., (2013), Determine The Appropriate Location Of Aerator System On Gotvandolidam’s Spillway Using Flow 3D,American-Eurasian J. Agric. & Environ. Sci., 13 (3): Page 378-383. J. Breckling, Ed., The Analysis of Directional Time Series: Applications to Wind Speed and Direction, ser. Lecture Notes in Statistics. Berlin, Germany: Springer, 1989, vol. 61.

[2] Bhajantri, M. E., Deolalikar, P. (2006). Hydrodynamic modelling of flow over a spillway using a two-dimensional finite volume-based numerical model. Sadhana, Page 743-754.

[3] Gessler, D. (2005). CFD Modelling of Spillway performance, EWRI 2005: Impacts of global climate change in Proceedings of the World Water and Environmental Resources Congress. Anchorage, Alaska: American Society of Civil Engineers.

[4] Hirt, C. W., & Nicholas, B. (1981). Volume of Fluid Method (VOF) for the Dynamics of Free Boundaries. J.Comp. Physics, Vol. 39, Page 201-205.

[5] ICOLD. (2001). Tailings Dams- Risk of dangerous occurrences, lessons learnt from practical experiences. ICOLD Comittee on Tailings Dam and Waste Lagoons (p. Bulletin 121). Paris: United Nations Environmental Programme (UNEP), Division of Technology, Industry and Economics (DTIE) and International Commission on Large Dams (ICOLD).

[6] Johnson, M., & Savage, M. B. (2006). Physical And Numerical Comparison of Flow Over Ogee Spillway in the Presence of Tailwater. Journal of Hydraulic Engineering, Vol. 132, No.12.

[7] Kjellesvig, H. (1996). Numerical Modelling of Flow over a Spillway. Hydroinformatics- 96.

[8] Singh, C., & Zhou, F. (1999). Simulation of Free Surface Flow over Spillway. Journal of hydraulic Engineering 125(9),Page 959-967.

[9] Zhenwei, M., Zhiyan, Z., Tao, Z., (2012), Numerical Simulation Of 3-D Flow Field Of Spillway Based On VOF Method, 2012 International Conference On Modern Hydraulic Engineering, SciVerse ScienceDirect, Procedia Engineering 28 (2012) Page 808-812.