International Journal of Mechanical Engineering and Technology (IJMET)
Volume 9, Issue 12, December 2018, pp. 470–486, Article ID: IJMET_09_12_050
Available online at http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=9&IType=12
ISSN Print: 0976-6340 and ISSN Online: 0976-6359

NUMERICAL SIMULATION OF TWO –PHASE FLOW WITH HYDRAULIC JUMP

Eman Shaker Hussein
Department of Mechanical Engineering, University of Al-Qadisiyah, Iraq

AtheerSaadHashim
College of Water Resources Engineering- Al-Qasim Green University, Iraq

ABSTRACT

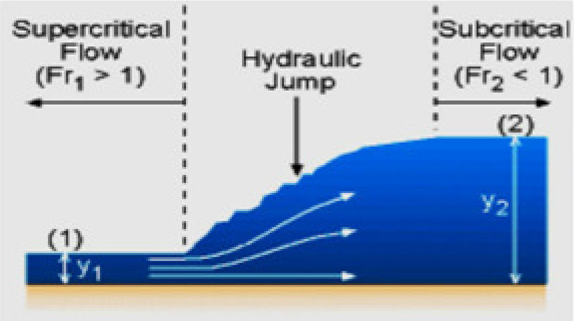
The present study tries to analyze the water channel junction, their upstream and downstream derived hydraulic jump due to an abrupt variation in the liquid flow phenomenon from the Supercritical to Subcritical stage, and that transforms the water into a gaseous form without any indication or visible phase changeover. A hydraulic jump condition explained as a significant loss of head, manifesting the available energy to act, scour and generate turbulence. The Triple Point phase position coexisting in equilibrium is created at the three diverse phases interaction point, where liquid, solid, and gaseous states of water coexist in a stable equilibrium. The water depth measurements for the simulation of the CFD was developed and further equated with experimental results and theoretical assumptions. The Reynolds-averaged Navier-Stoke called RANS, derives the turbulence model, and various hydraulic jumps applied to the turbulent flow explained. The numerical simulation results concluding design provide very supportive conclusion, concerning the physical type of model, definitely showing the CFD ability to simulate the complicated condition of fluid phenomenon. In the methodology, highly common modelling and application is shown in the water flow equations in the open-channel that analyses the Hydraulic Jump Process.

Key words: Hydraulic jump, Turbulence, CFD, Open channel, RANS.

Cite this Article: Eman Shaker Hussein and AtheerSaadHashim, Numerical Simulation of Two –Phase Flow with Hydraulic Jump, International Journal of Mechanical Engineering and Technology, 9(12), 2018, pp. 470–486.
http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=9&IType=12

1. INTRODUCTION

The Hydraulic jump is a sudden variation in the liquid flow phenomenon signifying the abrupt fluid flow change indicating to transfer of Supercritical flow pattern to another Subcritical arrangement, that transforms the liquid into the formation of gaseous state form without any clear phase transformation.



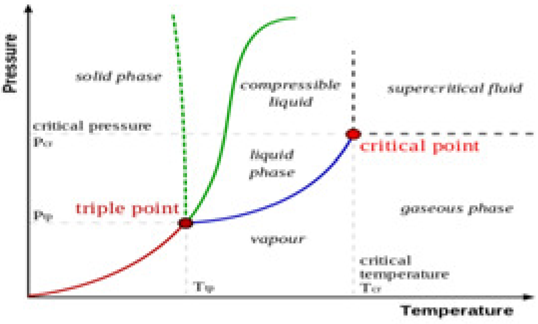


Figure 1: Fluid Flow Variations (Henderson, 1966).**Figure 2**: Supercritical Fluid and Critical Point (Bernhard, 2013).

The Triple Point phase position is the intersecting point of three diverse phases coexisting in equilibrium. In the diagram of water, the triple point corresponds to one single pressure and the temperature point at which liquid, solid, and gaseous states of water coexist in a steady equilibrium, usually at 611.657 Pa (Pascal) partial vapor pressure and 273.16 K (temperature).

$$1\ \mathrm{Pa} = 1\ \frac{\mathrm{N}}{\mathrm{m}^2} = 1\ \frac{\mathrm{kg}}{\mathrm{m}\cdot\mathrm{s}^2} = \frac{\mathrm{J}}{\mathrm{m}^3}$$

Where,
N = Newton; m = Meter, kg= Kilogram, **s =** Seconds; J = Joule .

Due to a rapid hydraulic jump, there is a transition of flow system in the natural river, stream, channel, coastal water, resulting in a complicated flow structure with high turbulence, and entrapment of air. The three patterns of fluid flow from Supercritical state to Subcritical position, and thereafter critical flow configurations always observed in any open channel flow categories, as they always remain in any one of these three specified categories.

- The Supercritical fluid flow usually happens in the higher fluid velocity, in any shallow space flow, within the critical deepness and it particularly happens when the sloping channel is too steep, and it is shown by Froude No > 1.
- The Subcritical condition of fluid flow happens in the reduced velocity, at the deep-rooted flow, normally happens over the depth of critical stage, specifically in the sloping and shallow condition of channel, where the Froude No is < 1.
- The Critical condition of fluid flow bifurcates Supercritical from Subcritical condition of flow (Seyedpouyan, 2017).

The specific energy parameter defines all these three concepts and is defined as the sum of Potential and Kinetic Energy per unit weight of liquid flowing in relation to the bottom of the channel, showing:$E = y + V^2/2g$**,** where, E = specific energy, feet-pound/pound, y = flow depth above bottom of channel bottom in feet, g = gravity acceleration = 32.2 ft/sec^2, V = average velocity of flow, ft/Sec, = Q/A, Q = Volumetric rate of flow rate, CFT, A = Area, cross-section of flow, ft^2.

For the incompressible liquid flow, the Navier-Stokes equations of the liquid movement and its continuous equation is possible to solve simultaneously, while taking the boundary requirements, to get the correct solution. But, the equation of Navier-Stokes remains non-linear, therefore, it becomes complicated to derive it analytically, when Re value is higher, the viscosity status can be easily neglected for the flow condition when the quick flow of the stream takes place surrounding the wing. But, for the Re intermediate value measures are not possible to reduce due to the presence of the force of inertia, which is almost same as the viscosity. Therefore, the best alternative is to get the needed results numerically (Harada and Li, 2017).

In the case of hydraulic jump, the straight and horizontal channel maintaining Froude Supercritical number between 2·0 to 4·0 was numerically simulated applying Reynolds averaged and Navier–Stokes equation. The turbulence was measured by using k–ϵ closing equation, where the mixed flow was used to prevail over the free moving boundary problem.(Chippada et al., 2014).

1. Literature review of Numerical Simulation of two-phase flow with hydraulic Jump

The hydraulic jump instant transformation is mainly because of high fluid flow velocity from the open channel condition towards the Subcritical flow specifications. Such flow properties and its characteristics are explained by using continuity momentum measures. Concerning this research study, we have considered the latest innovations performed in the hydraulic jump turbulent fluid flow development that includes the development of Undular and Irregular tidal bore upstream as well as downstream due to various obstacles, resulting in non-break hydraulic jump, showing the positive stream rush, and air bubble entrapment with the rolling stage in the state of the hydraulic-jump. This clearly indicates that the hydraulic jump creates a considerable turbulence in the flow of fluid. (Chanson, 2009).

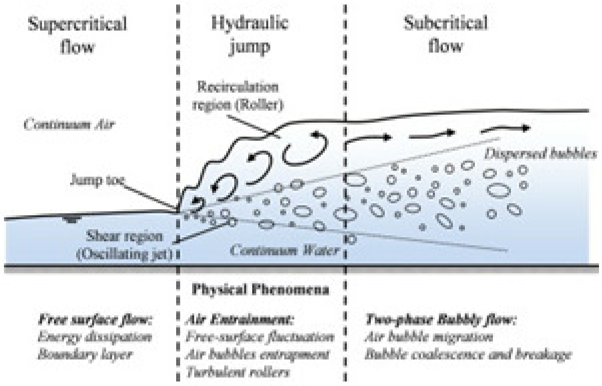


Figure 3 Correct phenomena of Hydraulic Jump (Chanson, 2009).

Initially, the concept of Hydraulic Jump was created by Leonardo da Vinci and offered for acceptance somewhere in the sixteenth century. Nearly two centuries afterward, further studies were initiated regarding the macroscopic stages of Hydraulic Jump, which was not specifically related to the microscopic Hydraulic Jump of internal flow (Beddhu et al, 1996). (Rouse et al. 1958) also commenced the experimental activities to draft the hydraulic jump structure of the turbulence, and processed a good contribution to these subject experiments, studies, and research. Helped by other academicians, Rouse and the team further managed to go deep into the subject of the field velocity structure using three Froude numbers; 2; 4; & 6 regarding the hydraulic jump free model realized in the atmospheric state by using an anemometer, hot wired. Thereafter, this specific air model was criticized for not including simulating every feature concerned with the hydraulic jump. Based on that, they took the hydraulic jump to be their main study that also concerned with the jet plane wall turbulence under unexpected and dire conditions due to wavering pressure pitch and gradients. This was done by investigating the study of Froude nine super critical numbers ranging from 2.57 to 9.67. Along with that, many different methods were tried by using analytical approach to assess the characteristics of inner jump, which was attempted by (Peyret& Taylor 1983). The turbulent flow production and flow energy dissipation using the low, ranging Froude numbers was further purposed by (Liu et al. 2004).

The knowledge of this new phenomena, occurrence, and observable facts of hydraulic jump was not completely realized, and hence, the research continued to enhance the complexities, structure, and construct of this complicated fluid flow knowledge and air entrapment awareness regarding the distribution status in the case of the hydraulic jump. Earlier, these experimental and computational studies presumably went through the scale problem, where, the setup dimensions utilized were happened to be very unrealistic and practically small. The fluid volume method helped track the free surface. The normal turbulence model k-ε provided turbulence closure (Seyedpouyan A. 2017).

The fluid transitory flow from the pumps is usually assessed as incompressible, while the velocity of fluid is considerably smaller, at the time it is related to the speed and sound of fluid. Therefore, the equation directing the incompressible flow, adiabatic motion measure of fluid having the differential, partial equation to illustrated by the continuing process, mass conservation and also the momentum of Navier-Stokes. This kind of equations is discredited and indicating the geometry of the flow-domain in the pump.(Bulten, 2006).

2. Reynolds Setup & Run; Navier-Stokes average simulations

A specific hydraulic jump occurs in the case of fluid flow specifically in the open channel when the fluid moves at a very high velocity, and then, abruptly alters its flow to a slow condition in the moving zone. When the flow of fluid reduces, an abrupt rise in the fluid depth happens, and in that condition, it converts part of the kinetic to potential energy, discarding part of the energy entirely. This type of fluid flow when explored by applying the water channel to measure the water depth, simulation of the CFD was formed, which was equated with experimental amounts using the theoretical prediction methods. Many researchers provided suitable attention to the fluid dynamics of CFD - Computation in the previous many years. The CFD contemporary code, its application and the application of commercial software have provided valuable support at every level of the project, providing the correct experimental investigation and analysis to be processed specifically in the final project stage. But, the computer speed, fluid flow complexity, format and shape, meshing method, flow pattern, and many different concerned factors remain as the main constraints while getting the accurate results. To obtain the simulated fluid flow near the stationary or moving objects, the separation pattern of fluid flow, its simulation combustion, some hypothetical model needs to be formed to evaluate the turbulent flow features. In this case Reynolds-averaged &Navier-Stoke (RANS) turbulence model is particularly used. The solution verification, its Code, and the model validation are similarly confronts the forming and after verification of codes, they indicate; 1) Assessment of the code accuracy observed when compared with its previous result; 2) The quantification of code and its numerical accuracy predicted as solutions to justify codes; 3) By evaluating and establishing the accurately processed work, indicating the realistic flow and that can be evaluated by findings and comparing with those of experimental results, to authenticate the code .

Generally, the attributes of all the associated errors of numerical approximation related to simulations are considered as the verifying methods, as per (Oberkampf et al., 2002). Or else, the verifying process and related numerical approximation figures basically referred to the code so as to solve the chosen equations correctly, without any errors, when using physically correct validation to address the probable model values (Roy &Oberkampf, 2011). They observed two concepts of verification. The initial stage is to justify the code process, to make sure the extent of possibility, that should be bug-free, and it provides the precise and methodical outcome with refined mesh to advance due to the accuracy in practical sequences.

The further process is to justify the solution, where, he process is verified so as to qualify every observed numerical value transpired on the numerical simulation system. Therefore, the possible numerical errors, in case observed, can be round-off as errors, known as iterative faults, and other several mistakes identified when operating with the programming of computers (Bulten, 2006). Hence, the accurate procedures to apply and identify the code can be:

- The Correct solution and Procedure: This approach is to readily compare every numerical result with explanation by providing correct answers to be managed by correct equations and methods (He & Sato, 2001).
- The procedure verifies and develops the concerned solutions is another approach to attain the proper solution to the accepted format of particular equation rather than locating the primary equations to obtain the precise solution (Germano, 1986).
- By submitting the standard solution, that were earlier located and also verified (He & Sato, 2001).

The complicated fluid flow pattern is developed by the two phases turbulent spillway flow applying the numerical system and it was investigated, so as to assess and expand the simulation results comparing with the results procured by using the numerical procedure using a large scale objective model to ensure precise simulation flow in the complicated situation of multi-phase flow. Hence, the ultimate numerical modeling results were validated and compared with the experimental results and measured. Thereafter, it was trying to ascertain the precise CFD model application in the physical modeling studies as the prime and complementary pertinent tool.

The preliminary boundary position was very important and the crucial stage of the numerical model. Hence, complete precaution was needed in the formation of the boundary position and by this way, the practical event could be satisfactorily symbolized. The boundary condition accuracy is vital and clearly indicates the position of physical process occurrence for the particular conditions of fluid flow. The Boundary condition signifies the reach and array of variable flow and its related gradient on the domain of computational boundary fluid flow condition.

The Conditional state of Upstream Boundary can further be configured and arranged as a set depending on the water reservoir inlet condition of fluid flow. However, when the inward fluid flow discharge together with its velocity having unidentified, such conditions remain too far away in relation to the requisite spillways to restrain the reflection consequences.

The Downstream Boundary condition has to be properly positioned and that depends on the stretch to the validity of the requisite domain. This is because, the normal flow of fluid on the sloping downstream side of this spillway remains in Supercritical conditions. However, the downstream area needs to be chosen for the spillway at the downstream end due to the condition that the ski-jump jet and the hydraulic jump are altogether formed (Gurav, 2015).

There are some identified preceding conditions to commence the solution, and it can be gathered by initially assuming different parameters to obtain the fluid flow field results. By clearly and precisely predicting the generated free surface profile and the finding the fluid velocity, the proper results can be processed and measured to meet the computations.

The studies of the hydraulic model based on the original designs were performed to obtain:

a. The spillway discharge capacity;

b. Pressure Profile;

c. The Profile of water surface;

d. Suitability and Correctness of the geometrical profiles.

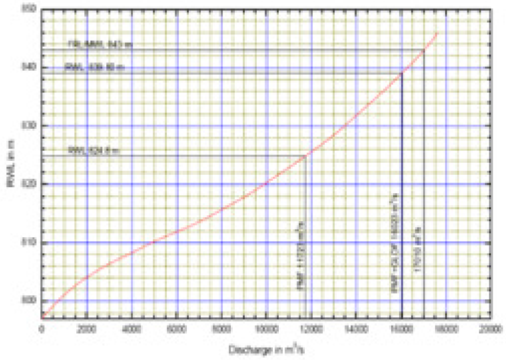


Figure 4: Discharging Capacity of Spillway (Gurav, 2015).

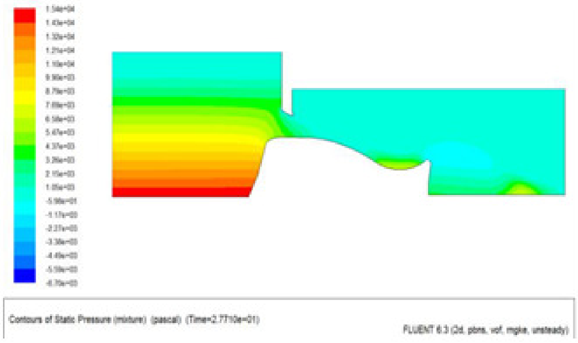


Figure 5: Pressure contour of spillway when operating Q = 16025 m3/second (Bhajantri, et al., 2006).

The Spillway Pressures Profile helps to compare is more vital to assess the potential of the Cavitation in the fluid flow. The numerical and hydraulic modelling Results when superimposed, the normal trend and the magnitude of the work provided a proper understanding with the physical model data of the study and observed condition. Little variations were observed due to probably local geometry of mesh and errors located in the measures of physical model.(Savage, & Johnson, 2006).

3. Literature Review on RANS applied in turbulent flow condition

Every research study tries to investigate the pattern of fluid flow in relation to its stepped spillway position and the commence by using laboratory experimental methods applying scaled modeling methods by the use of analytical and numerical approach. For an example, as stated by (Degoutte et al. 1992), the energy dissipated is excessive in the jet flow condition and is further than what is observed because of skimming flow. Additional verifications indicate that two different types of jet flow exist.

This initial fully recognized and formulated hydraulic jumps, while several other methods partially developed hydraulics jumps, as stated by (Pfister and Hager 2011) as they provided a complete visually observed procedure taking the help of high-speed processing camera to compute air inception and air intensity close to pseudo-bottom position in this specific step spillway model. (Felder & Chanson 2011) thereafter experimented using the practical investigation procedure in this study, understanding the slopping format usual steps of simple chute and thereafter performed several tests depending on five different configurations. The results of this test were subsequently evaluated and also compared using several flow resistance methods understanding the flow pattern, by using energy dissipating methods.

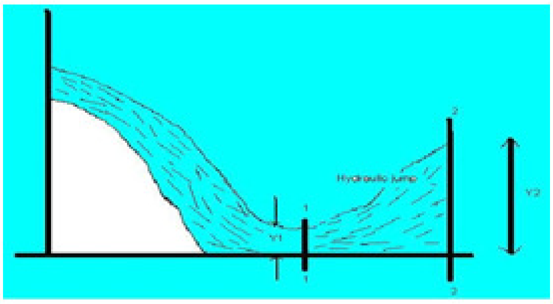


Figure 6: Y1 = flow depth of sub critical side, Y2 = flow depth after hydraulic jump

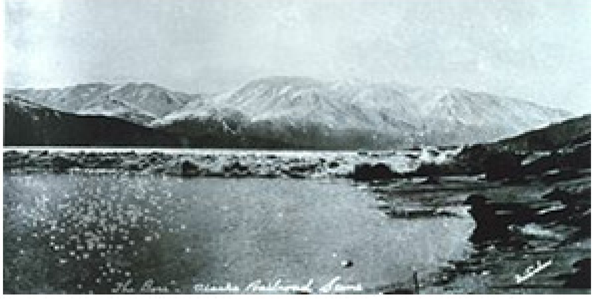


Figure 7 shows the A tidal bore indicates turbulence in shock-wave type condition in the front (Lighthill, 1978).

The turbulent characteristics and submerged condition of hydraulic jumps were numerically investigated by the means of the conventional model of *k*-ε turbulence. The VOF - fractional fluid volume concept was engaged to track the pathway of moving state of the free surface. These Numerical assumptions included hydrodynamic pressures, surface profiles, turbulence intensity, shear stresses, mean velocities, optimum horizontal velocity along with friction coefficient of the channel bed. The resulting computational measures were offered for Froude numbers starting with 3.3 to 8.3, while the submergence factors ranged from 0.25 to 0.86. Thereafter, the final results were judged against the experimental data available (Ma et al., 2010).

4. Open Channel fluid Flow Hydraulic Jump

The flow transition from the Supercritical condition to the fluid flow Subcritical state is the main and critical position of the entire Hydraulic Jump occurrence. The abruptly reducing the average velocity of fluid and largely unpredicted fluid depth generation during the flow on its pathway surface zone, there are losses to a great extent in the mechanical energy, and this kind of phenomena happens in Hydraulic Jump.

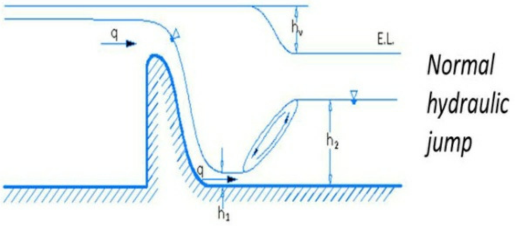


Figure 8 Normal Hydraulic Jump (Bulten, 2006).

hv = the energy loss in the fluid flow process, and it is a particular energy concept, which is not applicable in this particular case, and also not applicable entirely. As a result, in the case of Hydraulic Jump, the concept of energy loss due to specific energy is not entirely understood (Bulten, 2006).

(Salmasi 2010) studied the effects of the aerator kind to oppose any kind of damage happen because of the Cavitation developed in the stepped section of the Chute, an inclined passage or trough in the Hydraulic large scale Power Plant. This was done by applying the advanced technological application by using high functioning computers to observe the developments and progress in the traditional CFD software, called Computational Fluid Dynamic. The complementary analytical tool was developed to solve all the hurdles to assess the pattern of the fluid flow. That has now become very practicable (Rotunno& Bryan, 2018). It was further examined that the hydraulic jumps generated at the particular stepped spillways and assessed analytically, physically, and numerically. They thereon developed another kind of model designed on the concept of different prototype by applying typical hydraulic formula. Under the circumstances, the development model was of a large size and scale, which was applied in the experimental investigation to study the conduct by evaluating the resemblance to showed between every stage of the hydraulic jump. It was prepared by dropping the hydraulic jump length, while optimizing the discharge unit width The numerical Two 2D - Dimensional model of Reynolds Average Navier Stoke, called RANS was used for the calculations, which was useful to obtain the pressure and velocity of flow, and further to differentiate and find various concerned hydraulic forces developed on baffles and sills. Thereon, they established to discuss every result obtained by their elaborate study on the subject to deal with non-aerated zones of the glided water flow in the region of sharp and severe stepped spillway. Such particular process was concerned with all the experimental amenities offered based on a large-scale model. The advantages of actual CFD software also were explored in this program and was noted to scrutinize the flow pattern problems. By applying the FVM process using the FVM - Finite Volume Method together with the meshing grid of Voronoi, they measured the available equations, by which, they valuated the step spillway pattern of flow, specifically the velocity vectors, streamlines, static and dynamic pressures developed together with the all types of pressures generated at each step (Bernhard, 2013).

2. CONCLUSION WITH RECOMMENDATIONS

1. Numerical Simulation, two-phase hydraulic Jump

The Practical models developed for this study are very important and play a vital role in finalizing the most effective hydraulic designs. It turns too problematic and also time consuming while assessing to accommodate various changes and also to conduct various tests for making modifications to get the results. Hence, the results of numerical simulation of finally approved designs will offer an encouraging finding with respect to the practical kind of model. It surely shows the ability of CFD to simulate this type of complicated fluid observable fact (Si-Ying, et al., 2012).

In multi-phase position of the numerical direct method, the two dissimilar fluid interfaces must be examined to obtain the conclusive results concerning the inter-facial acute features. They must be liberated to deform, move, break-up, and coalesce just like any different interface structure behaves. In this manner, several multi-phase fluid flow applying numerical methods can be processed considering its above mentioned fundamentals, as they remain as the major challenge to evaluate all the systems of multi-phase patterns.

2. Recommendation

Even though such research study drives many alterations of multi-phase procedures in the analysis of Hydraulic Jump, the research has indicated a considerable development in the previous one decade. The application along with its relevance of all the methods are classified and controlled, particularly with respect to result patterns, which are yet uncovered. It is possible to perform an extensive research and study to conjure up all the possible methods to simulate many advanced flow patterns which should be regulated with systems we need, along with innumerable bubble formations in the fluid, based on the flow pattern of the fluid and related computational sources and their availability (Tryggvason*et al.* 2005). Several ordinary problems may generate concerning the engineering structures, as they are normally encountered in such experimental analysis. They are:

- The dissipation of energy during the film and bubble formation because of random and natural fluid flow characteristics;
- By considering many sites for the experiments, by locating to assess the formations of Hydraulic Jump at different intensities, various scales, of varied nature based on the pattern of flow on various surfaces, its gravitational pull, the nature of the stream, generated pressure difference in the flow at various stages due to material surface roughness and properties, also the overall operating condition of the system and its formation;
- The scale, structure, and the nature of water or liquid bubbles, droplets, and generated film, though all of them vary in nature, are important for the further study and different scale length;
- All the contact angles of fluid along with the surface slip condition needed, should be appropriately determined;

- While considering numerically, the most understandable and clear description of the obstruction must be determined having the overall details of the larger mesh size forces transacting within the required resolution for the computational time to obtain the correct solution. Plenty of efforts are needed to generate advanced techniques to diagnose, and solve all the Hydraulic Jump drawbacks similar to those of Adaptive Grid Meshing. However, with deep experimental studies, this kind of complex issues, specifically those problems faced during the sequences of phase changes and those involving complexities at the industrial scale should be further researched, experimented and solved (Lee et al., 2012).

3. METHODOLOGY, RESULTS, CFD MODELING, AND GOVERNING EQUATIONS

The highly common modelling and application is shown in the below equations in the water flow in the open-channel that specifically deals and analyses the Process of Hydraulic Jump. The rise of water level always occurs while transforming the Rapid unstable or otherwise, a Supercritical water flow that converts into a tranquil and stable Subcritical flow, also known as Hydraulic Jump, and it manifests itself as the standing wave. At this specific place, the process of hydraulic jump takes place and a plenty of flowing liquid energy gets dissipated, particularly into the heat energy. Hence, the hydraulic jump becomes and works as a Dissipator and Removal of surplus water energy. Beyond the state of hydraulic jump, the flow of water goes into further and higher depth, and hence, it flows with less velocity (Stein, 2018).

Free Surface Flow governing equations with appropriate boundary position was explained by using the Cartesian coordinating system CFD software. The transfer of heat was assumed to be negligible and steady fluid flow condition was assumed in all cases. The bed slope perpendicular direction needs more accuracy while simulating free surface so as to capture velocity gradient properly.

The scientific method pertaining to CFD - Computational Fluid Dynamics, can be used to predict mass transfer, fluid flow, chemical reactions, heat transfer, and related phenomena applying mathematical equations governing these processes. The equations of Navier-Stoke are solved to get fluid mechanic results. In the case of turbulent fluid flow, the turbulence model is required to be used. The equations can be solved numerically and the errors can be reduced by certain measures.

The STM - Standard step method has become the important part of the computational technique now applied to estimate a two-dimensional water surface in the open channel with progressively varying the water flow initially under the steady condition. The combination of the momentum, energy, and continuity calculating methods and equations are used to understand the depth of water knowing the channel slope, friction slope and channel geometry, when the flow rate is given. In the practical application, such techniques are extensively used by using computer programming developed by the engineers of the US Army, HEC - Hydrologic Engineering Center (USACE. Manual).

In the case of the open channel water flow equation of energy can be used for computation is a simplified Bernoulli Equation, that considers the elevation head, pressure head, and velocity head. These head and energy considerations are all synonymous and identical in the case of Fluid Dynamics. In the case of open channels, certain parameters are assumed and that the change in the atmospheric pressure due to hydraulic jump is negligible, hence, the Pressure Head applied in the Bernoulli Equation can be neglected. The equation is given as:

$$p + \frac{1}{2}\rho V^2 + \rho g h = constant$$

where p is the pressure, ρ is the density, V is the velocity, h is elevation, and g is the gravitational acceleration

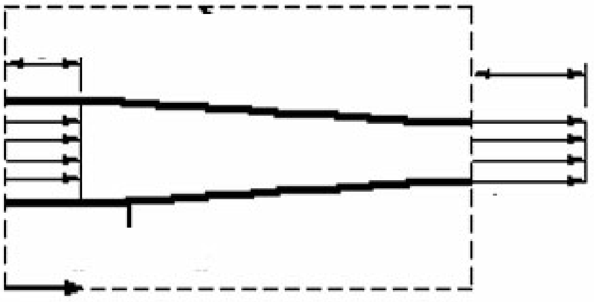
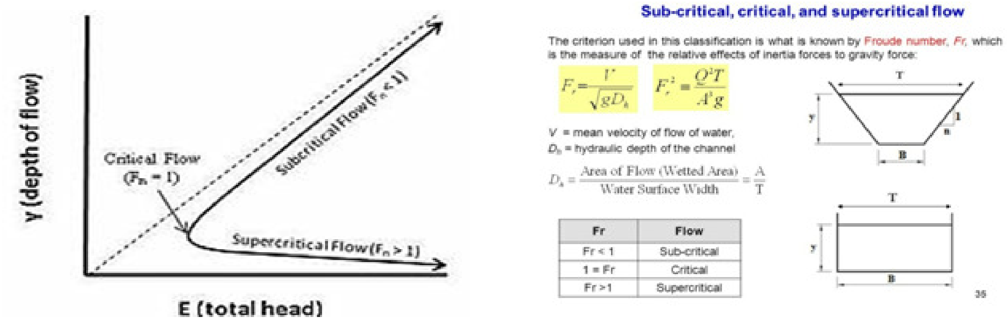


Figure 9: Flow format



Sub-critical, critical, and supercritical flow

The criterion used in this classification is what is known by Froude number, *Fr*, which is the measure of the relative effects of inertia forces to gravity force:

$$F_r = \frac{V}{\sqrt{gD_h}} \qquad F_r^2 = \frac{Q^2 T}{A^3 g}$$

V = mean velocity of flow of water,
D_h = hydraulic depth of the channel

$$D_h = \frac{\text{Area of Flow (Wetted Area)}}{\text{Water Surface Width}} = \frac{A}{T}$$

Fr	Flow
Fr < 1	Sub-critical
1 = Fr	Critical
Fr >1	Supercritical

Figure 10: Flow Conditions- Subcritical; Critical; & Supercritical

The depth of fluid flow is corresponding to the Froude Number, as the depth becomes greater than that of the critical position of depth, it becomes a Subcritical, and has the Froude Number below 1, when the depth is below the critical depth, it becomes the Supercritical flow, when the Froude Number becomes more than 1.

Under the steady flow condition of fluid, when the flood waves are not formed, in the case of flow in the open channel, it is bifurcated into further three sub-sections flow types: gradually varying fluid flow, uniform fluid flow, rapidly and varying fluid flow.

The uniformity of flow described as the situation when the depth of flow depth remains the same, without any change even after flowing certain distance down the channel, that occurs in the smooth flowing conditions in the channel without any flow change, channel geometrical change, alterations in roughness or the slope of the channel. During the uniform fluid flow, Yn depth remains analogous to its terminal velocity in free fall of the object, when the frictional forces and gravity force remain in balance (Moglen, 2013). The depth of the flow is measured by Manning Equation. Gradually varied flow occurs because of alterations in flow depth and also the alterations in very small flow distance. In such cases, the hydrostatic relationship generates a uniform and uniform flow that applies. For example, in the case of the backwater just behind the structure of in-stream conditions like weir, sluice gate, dam and so on, when the contraction takes place while the fluid flow continues in the channel, and in the case when certain changes in slope occur during the flow region. In such circumstances, due to a minor slope change, the flow instantly varies and the flow depth change per flow distance change is significant. In such cases, the hydrostatic relationship does not go proportional with analytical solutions, when the rule of flow continuity and momentum employed. For examples, when the abrupt and large slope is realized during the flow, like in the spillway, due to the abrupt state of constriction and flow expansion, the hydraulic jump occurs.

1. Hydraulic Jump

Hydraulic Jump is formed when the Supercritical flow converts into Subcritical. The Froude Number F1 is a dimension less number defines the characteristics of flow as Critical, Subcritical, or Supercritical. Upstream Depth = Y1, Downstream Depth = Y2, The following equation is for smooth horizontal and rectangular channel.

$$\frac{Y2}{Y1} = 1/2\left(\sqrt{1+8F1^2}-1\right)$$

The boundary conditions are

(a) The Inlet average velocity of each stage;

(b) The Static pressure Outlet; the distribution phase of hydrostatic water along with the air pressure values and reference;

(c) The condition of non slip wall possessing the rough wall; covered with the slope of bed on every wall;

(d) Opening of Static pressure for reference use; forced on domain cover. The Figure mentioned below shows every boundary domain flow condition (Munson, et al., 2002).

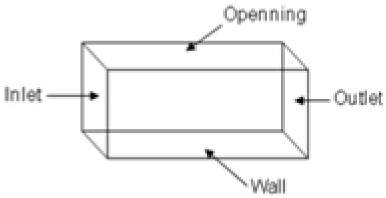


Figure 11: The Geometry of Channel Boundary condition

Case 1: An open channel Flow in rectangular section, of the Subcritical condition of flow regime.

If we take the case of the open channel having a rectangular cross sectional area, which is 1000 meter long, 10 meter wide, where the coefficient of Manning = 0.03. The outflow rate = 20 m^3/s, while in the Subcritical outflow and inflow regime, and bed slope = *S0* (*x*) can be given by the equation:

$$S_0(x) = \left[1-\frac{4}{gy(x)^3}\right]y'(x)+0{,}36\frac{[2y(x)+10]^{4/3}}{[10y(x)]^{10/3}}$$

Here the depth of free surface in this analytical solution depth together with derivatives can be shown as *y* (*x*); *y'*(*x*),

$$y(x)=\left(\frac{4}{g}\right)^{1/3}\left\{1+\frac{1}{2}\exp\left[-16\left(\frac{x}{1000}-\frac{1}{2}\right)^2\right]\right\}$$

$$y'(x)=-\left(\frac{4}{g}\right)^{1/3}\frac{2}{125}\left(\frac{x}{1000}-\frac{1}{2}\right)\exp\left[-16\left(\frac{x}{1000}-\frac{1}{2}\right)^2\right]$$

Case 2: Flow in rectangular open channel in the Subcritical and also Supercritical condition of flow regime. Equation ABC & Equation NMO.

The simulation was done on a rectangular open channel having 1000 meter length, 10 meter width, where the Manning coefficient was = 0.02, and outflow rate = 20 m^3/s, and the Inflow regime was Subcritical, Outflow regime was Supercritical; Slope of Bed was = *S0*(*x*) can be shown by the equation:

$$S_0(x) = \left[1 - \frac{4}{gy(x)^3}\right]y'(x) + 0{,}16\frac{[2y(x)+10]^{4/3}}{[10y(x)]^{10/3}}$$

The free type surface depth with analytical solution first derivatives are shown by *y*(*x*); *y'*(*x*)

$$y(x) = \begin{cases} \left(\frac{4}{g}\right)^{1/3}\left\{1 - \frac{1}{3}\tanh\left[3\left(\frac{x}{1000} - \frac{1}{2}\right)\right]\right\} & 0 \le x \le 500 \\ \left(\frac{4}{g}\right)^{1/3}\left\{1 - \frac{1}{6}\tanh\left[6\left(\frac{x}{1000} - \frac{1}{2}\right)\right]\right\} & 500 < x \le 1000 \end{cases}$$

$$y'(x) = \begin{cases} -\left(\frac{4}{g}\right)^{1/3}\frac{1}{1000}\sec h^2\left[3\left(\frac{x}{1000} - \frac{1}{2}\right)\right] & 0 \le x \le 500 \\ -\left(\frac{4}{g}\right)^{1/3}\frac{1}{1000}\sec h^2\left[6\left(\frac{x}{1000} - \frac{1}{2}\right)\right] & 500 < x \le 1000 \end{cases}$$

Equations XYZ and Equation QRS

Case 3: In Hydraulic Jump, there is a transition of fluid flow to Subcritical Regime from Supercritical regime as shown below. Once the CFD software validation gets completed for several flow cases in the open channels, there will be alterations in cross sections and bed slopes, while the different Froude Numbers can be built by numerical models to give validation to commercial software for hydraulic jump. Also, the experimental laboratory data can be used to validate the software assessed prediction.

The hydraulic jump boundary conditions can change due to changes in the profile of velocity and that will also change the previously explained parameters besides at the stage of inflow. The condition of hydraulic jump taking place was at the Numerical predictions at the place, besides the depth of water, at the upstream and also downstream places being extremely sensitive to the flow velocity at the inlet domain. Hence, many tests were conducted after altering the velocity change profile so as to establish proper boundaries of inflow conditions.

Considering three inlet velocity profile tests were conducted along with the constant profile of velocity, parabolic profile of velocity and the Couette Profile of Velocity:

$$u(y) = \frac{3\overline{V}}{h^2}\left(hy - \frac{y^2}{2}\right)$$

$$u(y) = \frac{6\overline{V}}{h^2}\left(hy - y^2\right)$$

The effect can be seen in the below figure due to the velocity effect on numerical predictions of free surface where F1 = 2.30

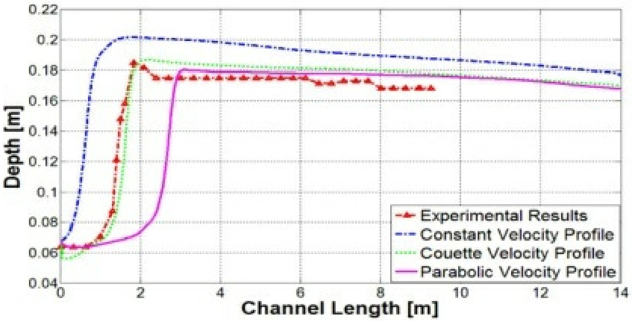


Figure 12: This is the numerical predicted chart for comparison of surface free position for three different velocity profiles when the F1 Inlet = 2.30

The numerical result profile obtained shows very sensitive to the inflow velocity of the fluid boundary conditions. Further, the experimental tests were conducted in the open channel tank as indicated in the below figure, where the length of the open channel is 14 meters, height is 0.915 meters, width is 0.46 meters, and the coefficient of Manning for simulation = 0.008 having an equivalent 0.00017 meter roughness. While, the tank considered for the experiments was about 0.465 meters wide, 1.55 meters in height, 0.45 meters in length. By altering the gate position provided different results of flow and consequently, flows having various Froude numbers were gathered (Hernández, 2006).

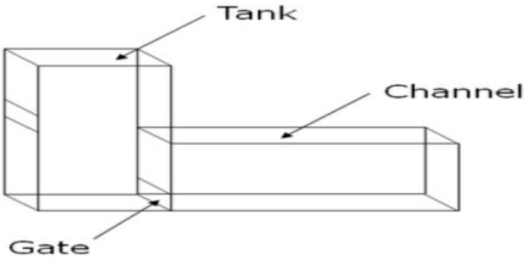


Figure 13: Geometry of Tank for simulating Hydraulic Jump.

Two Froude numbers 2.30; 4.23 were taken for calculations to obtain specific values of various parameters as shown in the below table.

Case	Froude number	Opening Gate High (m)	Velocity (m/s)	Flow Rate (m^3/s)
4a	2.3	0.0064	1.826	0.05376
4b	4.23	0.043	2.737	0.05414

2. Results

The following graph indicates the flow in the open channel having an irregular bed slope. Hence, the comparison was made to validate the software results of Subcritical and Supercritical regimes of different fluid flows.

The below two charts will indicate the chart of comparison position between analytically obtained solution from the equation ABC of the pure Subcritical flow and equation XYZ, besides the transition regime occurred within Subcritical towards the Supercritical conditions of flow. This numerical result obtained demonstrate an excellent result of free surface definition when compared with the analytical results. The two below shown curves are tabulated in the similar positions, which indicate that the applied software accurately produce the free surface results.

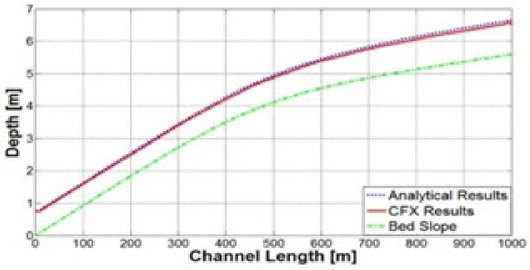


Figure 14: of Case 1: Numerical results comparison with analytical solution for Subcritical Rectangular Cross Sectional Flow Regime.

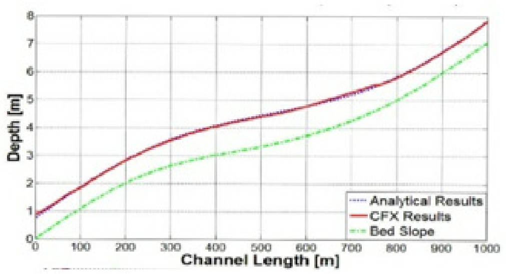


Figure 15: of Case 2: Numerical results compared with analytical solution for Subcritical flow transition to Supercritical flow regime in the Rectangular Cross Sectional Flow.

2. The Case Study of CFD Modulation

The science of Hydraulic Jumps in open channel is now regularly applied to understand the dissipating energy, and thus, they are extensively researched and studied by hydraulic engineers globally specifically by processing laboratory scale experiments. In the hydraulic jump, the surface of water rises abruptly to an unexpected level depending upon several flow parameters, and that cause an extreme agitation. The sort of re-circulation roller appears near the water surface and it intensely mixes with air. It starts from the high value of Fr1- Froude number. The Energy gets dissipated thereon, which is the most significant performance of water and very vital section of high water dam structures. These stilling basins have been extensively used to decrease the destructive the water destructive energy, by sending the water down the spillways in the high level water dams (Yazdi&Rostami, 2007).

The stilling position of the basin is regarded as a very strong hydraulic structure and that dissipates the water flow energy. The stilling basin dimensions and its geometry have tremendous effect on the structure and pattern of flow regions, which are highly influential to understand its hydraulic performance in the entire hydraulic jump system.

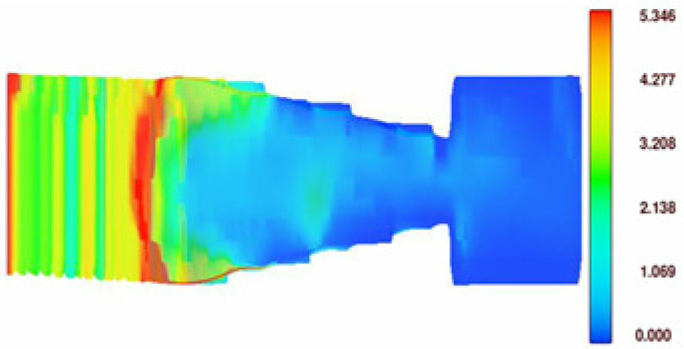


Figure 16: Turbulent dissipation of energy for 830 m3/second flow, at 12.5 degree convergent wall (Babaali et al., 2015).

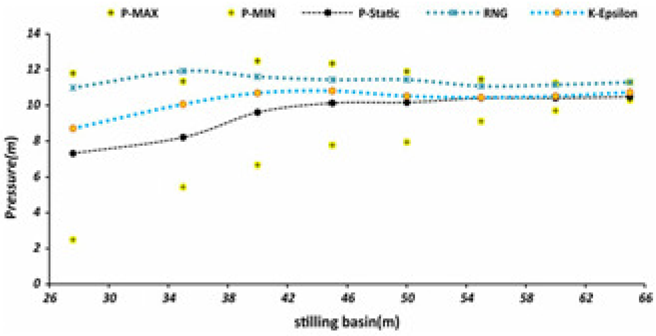


Figure 17: Comparing Physical and Numerical model of discharge pressure of 300 (m3/s)

The RNG model is useful for more applications than that of k-ε standard model. Specifically, the models of RNG are considered to be giving very accurate and desirable results describing the turbulence flow of low-intensity and other flows with strong shear region. In this particular RNG model, C1ε is taken as 1.42 and in the second case, the C2ε is taken as 1.68 (Isfahani&Brethour, 2009).

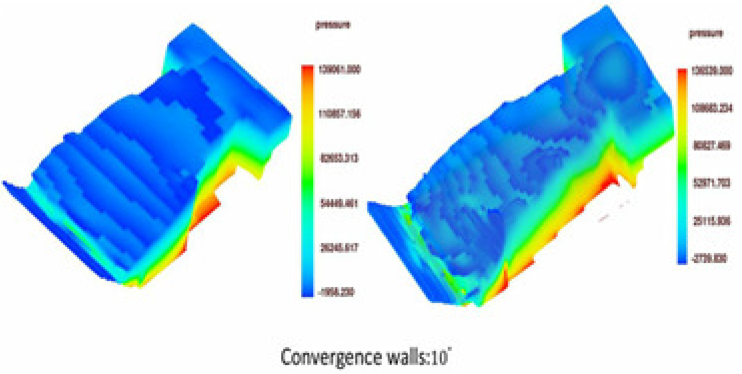


Figure 18: Pressure distribution chart, water discharge of 300 m3/second. The left side is k-ε model, and the right side of the RNG model (Babaali et al., 2015).

REFERENCES

[1] Bhajantri, M. R, T I Eldho, and P B Deolalikar, 2006. “Hydrodynamic modelling of flow over a spillway using a two-dimensional finite volume-based numerical model”- Sadhana Vol. 31, Part 6, December 2006, pp. 743–754.

[2] Beddhu M. L. Taylor, & D. Whitfield: 1996, “Strong conservative form of the incompressible Navier-Stokes equations in a rotating frame with a solution procedure”, Journal of computational physics 128, 427–437.

[3] Bernhard, 2013.“ Boundary between Supercritical fluid and gaseous phase”, Published: Aug 23, 2013.

[4] Bulten, N.: 2006, “Numerical analysis of a water-jet propulsion system”. Ph.D. thesis, Division of Mechanical Engineering, Eindhoven University of Technology, The Netherlands.

[5] Babaali, H. Shamsai, A. &Vosoughifar H. R. 2015. “Computational Modeling of the Hydraulic Jump”, Published: Dec 25, 2015, Journal for science and engineering, 40(2).

[6] Chanson, H. 2009.“Current knowledge in Hydraulic Jumps and related phenomena”, S survey of Experimental results”, European Journal of Mechanics B/Fluids, Vol 28. No. 2. PP. 191-210. doi:10.1016/j.euromechflu.2008.06.004

[7] Chippada S. Ramaswamy B. & Wheeler M. F. 2014.“Numerical simulation of hydraulic jump”, Published: April 30, 2014.

[8] Degoutte, G., Peyras, L., Royet, P. 1992.“Discussion of skimming flow in stepped spillways by Rajaratnam.ASCE Journal of Hydraulic Engineering, vol. 118, no. 1, p. 111-114., DOI:10.1061/(ASCE)0733-9429 (1992)118:1 (111.2).

[9] Felder, S., Chanson, H. (2011). “Energy dissipation down a stepped spillway with non-uniform step heights”,Journal of Hydraulic Engineering, vol. 137, no. 11, p. 1543-1548, DOI:10.1061/(ASCE)HY.1943-7900.0000455.

[10] Germano, M. 1986.“ A proposal for a redefinition of the turbulent stresses in the filtered Navier-Stokes equations”,Phys. Fluids **29** (7), 2323–2324 (1986).

[11] Gurav N. V. 2015.“ Physical and numerical modeling of an orifice spillway”, International Journal of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 3, Issue-10, Oct.-2015.

[12] Harada S. & Li s. S. 2017.“Modelling hydraulic jump using the bubbly two-phase flow method”, . Published: October 11, 2017,Environmental Fluid Mechanics, Volume 18, Issue 2, pp 335–356.

[13] Henderson, F. M. 1966. “Open Channel Flow”, MacMillan Publishing Co., Inc., New York, NY.

[14] He, L. & K. Sato: 2001, “Numerical solution of incompressible unsteady flows in turbo-machinery”. J. Of fluids engineering 123, 680–685.

[15] Hernández F., 2006, BSc in Mechanical Engineering Dissertation Project. Universidad Simón Bolívar - Venezuela.

[16] Isfahani, A. H. G.; Brethour, J. M. 2009. “On the implementation of two-equation turbulence models in FLOW-3D”. Flow Science. FSI-09-TN86 (2009) .

[17] Lee, Moon Soo; Aute, Vikrant; Riaz, Amir; and Radermacher, Reinhard, 2012. "A Review on Direct Two-Phase, Phase Change Flow Simulation Methods and their Applications" (2012). *International Refrigeration and Air Conditioning Conference.* Paper 1289.

[18] Lighthill, James, 1978. “Waves in Fluids”. Cambridge: Cambridge University Press. ISBN 0-521-29233-6.

[19] Liu et al. 2004.“ Turbulence Structure of Hydraulic Jumps of Low Froude Numbers”, Journal of Hydraulic Engineering 130 (6) · June 2004, DOI: 10.1061/(ASCE)0733-9429 (2004)130:6 (511) .

[20] Ma, F., Hou, Y. &Prinos P. 2010. “Numerical calculation of submerged hydraulic jumps”, Journal of Hydraulic Research, Volume 39, 2010 - Issue 5, Published online: 09 Aug 2010.

[21] Moglen, G. 2013. "Lecture Notes from CEE 4324/5894: Open Channel Flow, Virginia Tech". Retrieved April 24, 2013.

[22] Munson, B., Young, D., Okiishit, T., 2002,“ Fundamentals of Fluid Mechanics”, John Willey & Sons, USA.

[23] Oberkampf W. L., M. Sindir, and A. Conlisk, 2002. “Guide for the verification and validation of computational fluid dynamics simulations,” AIAA Journal, January, 2002.

[24] Peyret, R. and T. Taylor: 1983, “Computational methods for fluid flow”. Berlin: Springer.

[25] Pfister, M., Hager, W.H. 2011. Self-entrainment of air on stepped spillways. *International Journal of Multiphase Flow*, vol. 37, no. 2, p. 99-107, DOI:10.1016/j.ijmultiphaseflow.2010.10.007.

[26] Pineda S. F. Blanco, A. &Solorzano L. R. 2009. “CFD Software Applications for Transcritical Free Surface Flow”, Published: August 2009.

[27] Rouse, H., Siao, T.T. &Nagaratnam, S. 1958. “Turbulence characteristics of the hydraulic jump”, Journal of the Hydraulic Division, ASCE, vol. 84, no. HY1, pp. 1528-1-30, 1958

[28] Rotunno R. & Bryan G. 2018. "Numerical Simulations of Two-Layer Flow past Topography". Part I: The Leeside Hydraulic Jump,American Meteorological Society. Published: April 11, 2018,

[29] Roy C. J. and W. L. Oberkampf, 2011. "A comprehensive framework for verification, validation, and uncertainty quantification in scientific computing," Computer Methods in Applied Mechanics and Engineering, vol. 200, pp. 2131-2144, June 2011.

[30] Salmasi, F. 2010." An artificial neural network (ANN) for hydraulics of flow on stepped chutes". *European Journal of Scientific Research*, vol. 145, no. 3, p. 450-457.

[31] Savage, B. & Johnson, M. 2006. "Physical and numerical comparison of flow over ogee spillway in presence of tail water.", Journal of Hydraulic Engineering, 132 (12), 1353-1357.

[32] Sabaagh-Yazdi, S.R.; Rostami, F. 2007. "Turbulent modeling effects onfinite volume solution of three dimensional aerated hydraulic jumps using volume of fluid". In: 12th WSEAS International Conference on APPLIED MATHEMATICS. Cairo. Egypt. 29–31 December27, 2007.

[33] Stein J. 2018. "Hydraulic Jump, Open channel hydraulics for engineers", Chapter 3.

[34] Seyedpouyan A. 2017."Numerical study of two-phase turbulent flow in hydraulic jumps". Masters thesis, Concordia, University. Spectrum Research Repository,Published: April 6, 2017.

[35] Si-Ying, W., Dong-mei, H., Cai-huan, W. 2012."Aerator of Stepped Chute in Murum Hydropower Station". Procedia Engineering, vol. 28, p. 803-807, DOI:10.1016/j.proeng.2012.01.813.

[36] Tryggvason, G., Esmaeeli, a. & Al-Rawahi, N., 2005," Direct numerical simulations of flows with phase change", Computers & Structures., vol. 83, no. 6-7: p. 445-453.