ANALYSIS OF FLOW OVER A STEPPED SPILLWAY USING ANSYS FLUENT 16.0

A dissertation submitted in partial fulfillment for
the requirement to award the Degree of

MASTER OF TECHNOLOGY
IN
HYDRAULICS AND WATER RESOURCES ENGINEERING

Under the guidance of

Dr. Rakesh Kumar
Professor
Department of Civil Engineering
Delhi Technological University

By

CHETAN ARORA
(2K14/HFE/05)

Department of Civil Engineering
Delhi Technological University
New Delhi-110042
June, 2016

CANDIDATE'S DECLARATION

I do hereby certify that the work presented is the report entitled **"Analysis of flow over a stepped spillway using Ansys Fluent 16.0**" in the partial fulfillment of the requirements for the award of the degree of "Master of Technology" in "Hydraulics & water resources engineering" submitted in the Department of Civil Engineering, Delhi Technological University, is an authentic record of my own work carried out from January 2015 to July 2015 under the supervision of Dr. Rakesh Kumar (Professor), Department of Civil Engineering.

I have not submitted the matter embodied in the report for the award of any other degree or diploma.

Chetan Arora

Date: 1/8/15 (2K14/HFE/05)

CERTIFICATE

This is to certify that above statement made by the candidate is correct to best of my knowledge.

Dr. Rakesh Kumar
(Professor)
Department of civil engineering
Delhi technological university

ACKNOWLEDGEMENT

I take this opportunity to express my sincere regards and deep gratitude to Dr. Rakesh Kumar(professor, civil engineering department, DTU) for his consistent guidance, monitoring and constant encouragement throughout the course of this project work. The help, blessing and guidance by him shall carry me a long way in life on which I am going to embark.

Also, I express my gratitude to Dr. Nirender Dev (Head of Department, civil engineering department, DTU) for extending his support and guidance.

Professors and faculties of the civil engineering department, DTU have always been very supportive and cooperative. They have been always present for their kind opinions and suggestions regarding this project work and therefore I am deeply obliged to them.

Last but not the least, I would like to thank my family and friends who encouraged me to bring this work to a successful close.

ABSTRACT

In this dissertation, analysis of flow over a stepped spillway is done. The CFD analysis will concentrate on the energy dissipation caused by the stepped spillway and flow transition from the nappe regime to skimming regime with the variation in discharge for a constant step size. The change in the regime will alter the efficiency of energy dissipation done by the stepped spillway. After studying the flow regimes for different flow discharges and for a constant slope, effect of variation of number of step size (height and length) on the energy dissipation is analyzed.

Broadly the project is divided into 3 segments

1) For a fixed step size and a constant slope, variation of flow profile by varying discharge per unit width is studied. This study will tell change in mechanism of energy dissipation
2) For a constant discharge per unit width and a constant slope, variation of the number of steps i.e. changing the height of each steps and varying the number of steps keeping the total height of spillway being the same is analyzed.
3) Comparison of efficiency of stepped and non-stepped spillway

The project entails study of the relationship between the critical depth of flow and energy dissipation.

The importance of this work is associated with the stability of downstream bed channel. The aim is to reduce the total energy of water overflowing through spillway to a considerable extent such that there is no scouring due to the energy of flowing water on the downstream river bed.

LIST OF FIGURES

LIST OF TABLES

CONTENTS

CHAPTER 1

INTRODUCTION

1.1 Spillway and its requirement

Spillway is a structure which is constructed on the downstream side of the dam so as to discharge the excess flood to the tail race level of the channel. Generally water is impounded on the upstream side of the dam which can be diverted for beneficial use for example hydropower generation. But from practical and economic considerations it is necessary to pass down the excess water safely on the downstream side of dam so as to avoid overtopping or damage to the dam. This is done with the help of spillways.

When water is discharged from spillway it flows from a high elevation point where it is having considerably large amount of energy to tail race level. When water impinges on the river bed it may have sufficient kinetic energy so as to cause scouring of the bed channel.

1.1.1 **Problem associated with scouring**

Scouring is a phenomenon where the river bed channel gets eroded and its equilibrium is disturbed leading to instability in the side slope and bed slope of the channel. This leads to degrading condition in the river channel. This instability is undesirable as it changes the river flowing condition, river alignment and amount of silt carried by the channel. Degrading condition on the downstream side will lead to aggrading condition at some other point of river channel where this excess of silt will be deposited. This will eventually lead to meandering condition in the river channel and ultimately affect the design purposes of other hydraulic structures. This is the reason why scouring need to be restricted on the downstream side of the dam or reservoir.

For this purpose, energy dissipating devices are used at the toe of the spillway.

1.2 **Various types of spillways:**

1 Free over fall spillway

Water drops freely down the crest for instance in case of an arch dam. In the case of freely falling, water is sufficiently aerated or ventilated to avoid pulsating, jerky or varied jet of flow. There is a possibility of scour hole on the downstream unprotected beds the volume and depth of which can be correlated with the variation of discharge , drop height and tail water depth.

2 Overflow spillways

This type of spillway has s shaped crest. At low discharge, the profile follows the shape of lower nappe of a freely flowing weir transmitting the discharge at at higher discharges, there is a possibility of detachment of water profile from surface and negative pressure may built up thus disturbing its efficiency

3 Chute spillways

Here through an open channel, the discharge is conveyed from the ponding structure to downstream level of river. They are mostly used for embankment dam, for instance Tehri dam. The advantage with these spillways is that their design is simple and can be constructed on a number of foundation materials varying from clay to rocks

4 Side channel spillway

Discharge from the side channel may be diverted into a closed conduit that may run under pressure, or into a chute channel or into an open channel. The side channel spillway is expensive and moreover it’s not hydraulically efficient but it becomes advantageous in the cases where it is required to limit the afflux with the help of a long overflow crest. It is also desirable in the cases where the abutments are steeper.

5 Shaft spillways

In this type of spillway, water enters from a horizontal lip, then it falls through a vertical or inclined shaft and thereafter it flows to downstream channel of river through a tunnel or conduit.

In this spillway type, piers or guide vanes are required to minimize the effect of vortex in the reservoir. This is desired in cases where there are narrow gorges and steep rise abutments.

6 Tunnel spillways

When a closed passage is used to transmit the flood water around a dam via the neighboring hill sides then this type of spillway can be termed as tunnel or conduit spillway. They are generally suited in case of valleys having large width because in these cases, the application of such type of spillway would enable it to be constructed beneath dam in a close proximity to the stream bed

7 Stepped spillways

Stepped spillway consists of series of steps which initiates at the crest of spillway and propagates downstream. The stepped spillways were being used commonly since ancient epoch because of considerable impact of steps on the energy dissipation caused by this spillway but in the recent years, with the advent of new technologies like roller compact concrete and new manufacturing techniques for the concrete, innovations in the use of admixtures; the stepped spillways are more in use.

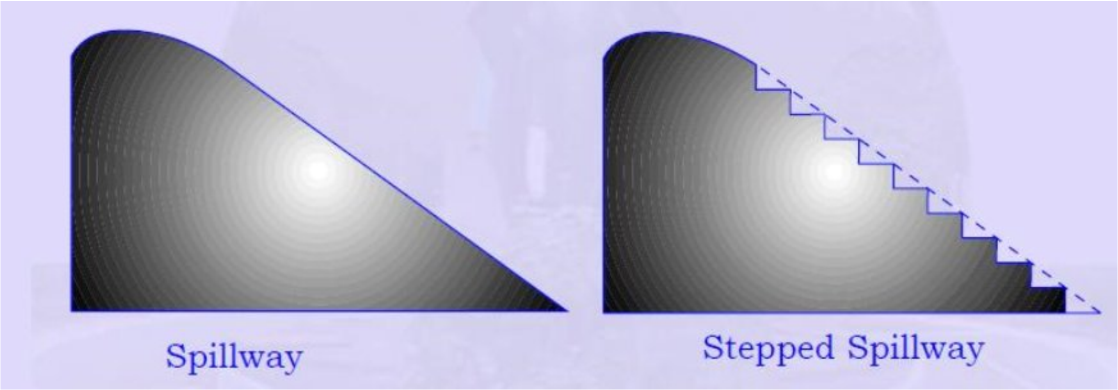


Figure 1.1(source: NPTEL Lectures)

Generally, roller buckets or trajectory buckets or hydraulic jump type stilling basins are used for dissipating the energy but when the discharge is high then these devices are not much efficient.

The advantage of the stepped spillway over other spillways is that no additional energy dissipating devices are required. The cascading series of hydraulic jumps and the impact of jet on the steps are sufficient enough to reduce the energy of the flowing water to a level such that downstream channel is free of risk of scouring.

The only limitation of stepped spillway is that at very high discharges, there may be a possibility that water jet is not aerated for some distance on the downstream portion of spillway where low pressure may build up and lead to cavitation damage

1.3 Flow classification over a stepped spillway

The idea of stepped spillway was evolved in 1892-1906 in new croton dam. For the very first time, the use of stepped spillway in intermediate erodible river reaches was done by Lombardi and Marquenent. These reaches were having a sufficient slope that causes hydraulic jump at the base of each fall. Basically there are three types of flow possible over a stepped spillway namely

Nappe flow

Skimming flow

Partial nappe flow (transition flow)

For low discharges the flow is observed to be nappe flow, for intermediate discharges flow is observed to be partial nappe flow and for high discharges the flow is observed to be skimming flow.

1) Nappe flow

The flow down rush over the steps, in a series of jumps falling from one step to another in a thin layer that adheres to the visible side of each step. There may be a number of modes of energy dissipation viz. splitting of jet in the air or pound of jet on the steps, transfusing on the steps in either the presence or absence of partial hydraulic jump on each step. For the occurrence of nappe flow, step height should be large enough

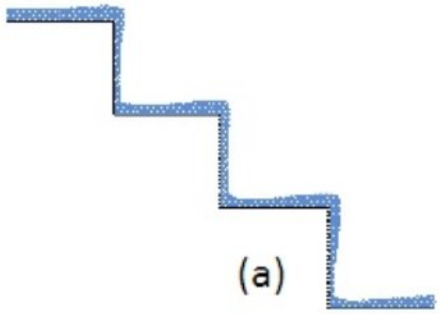


figure 1.2 (nappe flow)

2) Partial nappe flow

In partial nappe flow, the nappe does not completely bear upon the step surface and it dissipates with an appreciable amount of turbulence and the down the length of spillway flow is supercritical. For a given geometrical model, with an increase in the discharge, there will be an intermediate flow profile between nappe and skimming flows. This is attributed by splash of circulating water and often associated with a very small cavity and considerable water spray and deviation of water jet right away downstream to the stagnation point. Downstream the water spray portion there is deceleration of supercritical flow up to the downstream edge of steps. The intermediate flow pattern exudes considerable longitudinal variation in the properties of flow on the each step. It does not depict the coherent emergence of skimming flows.

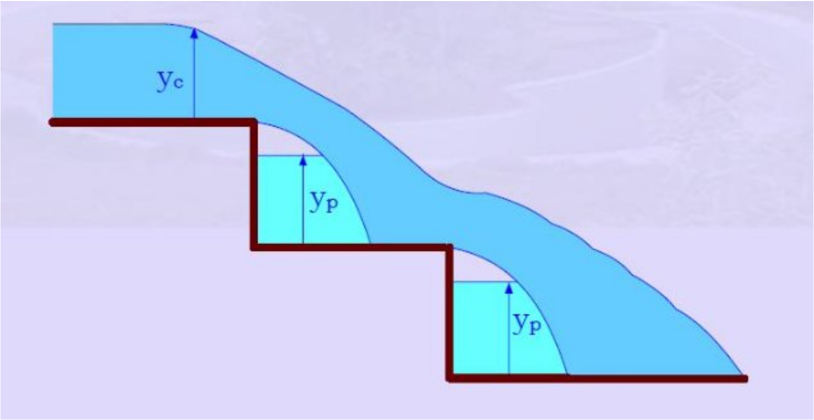


figure 1.3(partial nappe flow) (source: NPTEL lectures)

3) Skimming flow

Here, the discharge flows down the steps as a coherent course topping or skimming over the steps and padded by recirculating flow entrapped between them. The outer sides of the steps behave as a false (pseudo)bottom over which skimming flow occurs. Under this, the recirculating vortices are created and the transmission of shear stresses from the edges over which water is flowing is the reason for their prevalence.

The flow is lucid and clear and has a shiny appearance at the upstream end and also there is no air entrainment. Down the few steps, the flow is attributed by entrainment of air like self-aerated flow through an invert spillway. In skimming flows, a stable vortex develops and overlying flow moves down the spillway cushioned by these vortices which acts as a rigid boundary surface for the skim type flow whether air entrainment occurs or not. There is an incessant interchange of flow between the top layer and vortices developed on steps. The flow circulates in the vortex for a short period of time and then return to the original flow to continue down the facing spillway. Similarly when aeration occurs; the bubbles of air get in and rotate along with vortex flow.

Change from one type of flow to another is bit by bit i.e. creeping and continued leading to the simultaneous appearance of both nappe and skimming flow in certain flow regions varying with space and time.

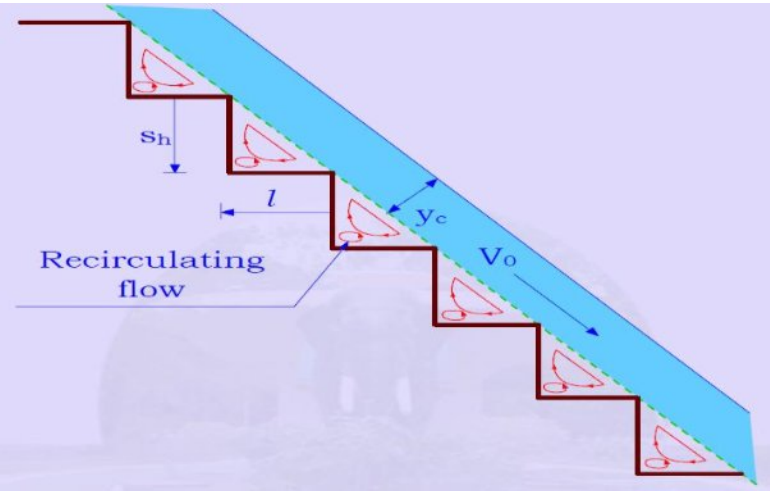


Figure 1.4 (skimming flow) (source: NPTEL lectures)

1.4 ANSYS Fluent

ANSYS is an advanced software which enables one to do all the engineering related simulations of problems related to fluid dynamics, chemical engineering, environmental engineering, hydrodynamics, metaphysics, electromagnetics, structural mechanics and so on. In this project we have particularly used fluent because we are concerned with the system's fluid dynamics. The foundation of company was laid by Mr. John A Swanson. Computational fluid dynamics (CFD) is a mathematical tool based on computer programming. The growing interest in the field of CFD based simulations has been widely used by engineers in all those areas where experimental or numerical analysis becomes cumbersome. Determination of movement of fluid in detail by solving a system set of nonlinear governing equations after the use of specified boundary conditions over the ambit of interest is the basic principle which is used in the analysis of CFD problems. The simulations based on CFD are contingent upon combined numerical accuracy, cost of computations and precision of modeling.

Using ANSYS CFD, virtually, the system of fluid flow can be simulated using computer analysis. Analysis can be started by first of all creating a mathematical model of physics problem associated. The CFD method of solving entails 3 approaches:

- Finite difference method
- Finite element method
- Finite volume method

In ANSYS fluent for the analysis of flow over stepped spillway, we have used finite volume method.

Advantages of CFD

1. Real time simulation conditions can be achieved which are difficult to maintain in the experimental analysis
2. High speed performance
3. The simulation cost is minimized and optimized
4. Multiple changes in geometry can be done until the accurate results are obtained
5. Margin of error is negligible if done carefully

Limitations of CFD

1. Physical modeling should be done meticulously because all the analysis and resuts are completely based on it
2. The model will give accurate results only if boundary conditions are accurately specified.

1.5 Objective of dissertation

The main objective of this study is:

1 To analyze the energy dissipation downstream the stepped spillway when the parameters such as step height or discharge are varied for a constant slope angle

2 Study of change of flow pattern from nappe flow to skimming flow and verify it with the equations given in the previous literature work.

3 Study of relationship between the critical depth of flow and associated energy dissipation on downstream side.

4 Comparison of efficiency of stepped and non-stepped spillway

1.6 Organization of dissertation

The report is fragmented into 7 chapters.

- Chapter 1 elaborates the aim and the significance of this study done.
- Chapter 2 entails the literature review done with the associated topic
- Chapter 3 and chapter 4 present the methodology and numerical data used in the study respectively.
- Chapter 5 includes the analysis of energy dissipation of a non-stepped spillway
- Chapter 6 presents the results and discussions
- Chapter 7 includes the conclusion and future scope associated with the work done in the present study

CHAPTER 2

LITERATURE REVIEW

2.1 George C. Christodoulou et al. (1993) they examined energy dissipation of stepped spillway experimentally in the laboratory. The flow considered was skimming and step height to width ratio is kept 0.7 with a moderate number of steps. They concluded that the most significant parameters which influence the energy dissipation is ratio of critical depth of flow available over the spillway crest to step height (Yc/H) and number of steps (N) .they concluded the energy dissipation was more when Yc/H value is near to unity and the energy dissipation goes on decreasing with its increasing value. They conducted the experiments at applied hydraulics laboratory of the national technical university of Athens. The slope angle used was 55 degree with respect to horizontal. The spillway face was divided into 7 steps on the curved part and 8 steps on the straight part with length of each step 1.75 cm and height of each step as 2.5cm and l/h ratio as 0.7. Spillway was made of wood and it was epoxy coated to avoid warping. It was placed in a laboratory flume having length of 10m and width 0.5m. The bed slope of flume is 0.04 and at the downstream side supercritical flow is maintained without formation of hydraulic jump.

2.2 M.R. Chamani and N. rajatraman. (1994) they further carried out and extended the experimental work of Horner and Essery (1969) who had indicated that energy dissipation of jet flow regime on a single step is mainly because of mixing of jet with the recirculating back water and the quantified energy dissipation occurs due to partial hydraulic jump formation on the deflected jet. They analyzed experimentally the relative loss of energy on each step. They demonstrated the variation of ΔE/E with Yc. They took the ratio of critical height to step height less than 0.8. They introduced the concept of a factor α which accounts for relative loss on each step and using the results of Horner they evaluated the value of α. They eventually concluded that α is a dependent function of Yc/H and H/L where L is the length of each step, Yc is the critical depth of flow and H is the step height. They also concluded when the value of Yc/H exceeds 0.8 then the energy losses becomes insignificant. Assuming that skimming flow occurs when the jet leaving the step has a slope is equal to that of stepped spillway when it impinges on the water pool behind it on the next step; an equation has been developed to predict the incipient value of Yc/H. Earlier chanson had proposed the empirical equation for the onset of skimming flow

$Yc/H = 1.057- 0.465\ (H/L)$ (equation 2.1)

Cross check with this equation was done too in this study.

Chanson presumed that the skimming flow initiates when the air pocket under the falling jet disappears and as a consequence, the depth of pool Yp under it is equal to step height H. This criterion gives rise to equation

$$\left[\frac{y_c}{h}\right] = \mathsf{F}_b^{2/3}\,\frac{\sqrt{1+\frac{1}{\mathsf{F}_b^2}}}{\sqrt{1+2\mathsf{F}_b^2\left[1+\frac{1}{\mathsf{F}_b^2}\right]^{3/2}\left[1-\frac{\cos\theta_b}{\sqrt{1+\frac{1}{\mathsf{F}_b^2}}}\right]}}$$

(equation 2.2)

Where F_b Froude number at the brink of step and θ_b angle of jet leaving the step with horizontal

2.3 Robert M Boes and Willi H Hager. (2003) They conducted an experiment on a big model of flume using fiber optical instrumentation and indicated that the onset of skimming flow is a function of chute angle, critical depth and step height. Homogeneous mixture depths that determine the height of chute sidewalls a uniform equivalent clear water depths are described in terms of a roughness Froude number containing unit discharge, chute angle and step height. The spillway length needed to attain uniform flow is expressed as function of critical depth and chute angle. The flow resistance of stepped spillway is considerably larger than for the smooth chutes due to Marco roughness of steps. The friction factor for uniformly aerated flow is of the order of 0.1 for typical gravity dam and embankment dam slopes where the effect of relative roughness is rather small.

2.4 Saman abbasi and amir kamanbedast. (2012) Their topic was the investigation of effect of changes in dimension and hydraulics of stepped spillway in order to study energy dissipation.

The software used was CFX and they carried out the comparison with work of rajatram and salmasi and found out the percentage error in the results. They also found out the effect of variation of number of steps, step height and discharge.

2.5 Hamid reza vosoughifar and azam dolatshah (2013). Their study was associated with the development of computational dynamics fluid code known as V flow using MATLAB done for a 2 dimensional modeling of unsteady flow over stepped spillway. V Flow can be associated with GAMBIT software and modeling can be done for varying spillway geometries using linear mesh elements. The volume of fluid method was used for solving the governing equations. Moreover laminar flow over the spillway was considered. The implicit time approximation, power law scheme and gauss siedel method and simple algorithm were used for the purpose of discretization. The simulation of the experimental model using both v flow and fluent software was carried out for the purpose of validation. Both results obtained were in good agreement with each other.

2.6 Moussa rassaei, segheh rahbar (2014). They analyzed the stepped spillway in order to maximize the energy dissipation. Their method of study was based on computer software. They had used ANSYS fluent software. The regular mesh was used for the analysis purpose. In order to solve the governing Navier stokes equation the finite volume method is used and in order to access the turbulence effect, k-€ turbulence model was used. They compared their results with the work done by the other researchers who did same work experimentally and validated their results. Their results were better than most of the other researchers due to incorporation of turbulence model in the latest version of ANSYS fluent

CHAPTER 3

METHODOLOGY

Design Study about stepped spillway

↓

Theory on computational fluid dynamics

↓

FLUENT software analysis

↓

Results in the form of tables and graphs

↓

Conclusion

3.1 FLOW PARAMETERS

In open channel flows, the critical depth of flow is obtained as per equation

$\mathbf{Y_c = (q^2/g)^{1/3}}$ (equation 3.1)

where q is discharge per unit width and g is acceleration due to gravity

The different discharge rates considered are 0.0126, 0.0195 and 0.0285 m^3/s

The width of model considered is 0.5m. Hence the different discharges per unit width are 0.025, 0.039 and 0.057 m^2/s

From above equation we can find that different critical depth of flow is 0.04, 0.054 and 0.069m

For software input is required in mass flow rates. Therefore different mass flow rates are 12.36, 19.5 and 28.5 kg/s

The equation for the onset of skimming flow is given by **Yc/H = 1.057- 0.465 (H/L)**

(source: Chamani, M.R. and Rajaratnam, N, Jet flow on stepped spillways, Journal of Hydraulic ASCE, Engineering. Vol.120, No.2, 254-259)

Where H is height of step and L is length of steps.

For 6 no. of steps, step height is 0.166m and step length is 0.2875m

For 12 no. of steps, step height is 0.083m and step length is 0.1437m

As per the equation we find that for all the cases considered with 6 no. of steps, flow regime will be nappe because calculated for all the flow ratio is less than the value calculated from equation mentioned above and in cases with 12 no. of steps, flow is skimming in the case when mass flow rate is 28.5kg/s

3.2 Designing steps

In this thesis, the analysis of stepped spillway with variant of discharge and step size has been done using ANSYS fluent. The fluid flow simulation analysis is divided into 3 parts

1 pre processing

- Creating the geometry and confining the geometry in a domain in which flow will be analyzed and discretization of geometry by meshing is done
- Defining the various boundary conditions pertaining to the flow of water over the steps
- Defining the flow parameters (mass flow rate and operating pressure conditions)

2 Solver

The Navier stokes equations of continuity and momentum are solved till desired accuracy is attained

3 post processing

Analysis of results is carried out

3.3 Making a model

ANSYS fluent enables us design, mesh, solve and virtually see the results exuded in the form of velocity vector, volume fraction of both the fluids, steam lines and energy vectors

3.3.1 Geometry set up

The geometry for the stepped spillway has been created in the design modular of ANSYS workbench.in this thesis, a total of 6 models has been created as there are 3 variants of discharge and 2 variants of step size. The creation of geometry in design modular is a very cautious task because the overlapping of any two nodes or a small gap left between any two nodes disables the geometry to be generated. The dimensioning of the model is as per the scaled ratio only so as to compare it with the previous work done

Slope of steps is 60 degree and angle of entry of water over the steps is kept 15 degrees with the horizontal

While creating the geometry, water and air both the fluids have been given their respective inlets. This is necessary in open channel flows as without defining the interface between the two different fluids computations cannot be carried out.

Figure 3.1 and 3.2 below show the geometry of the structure with 6 steps and 12 steps respectively along with the boundary conditions assigned.

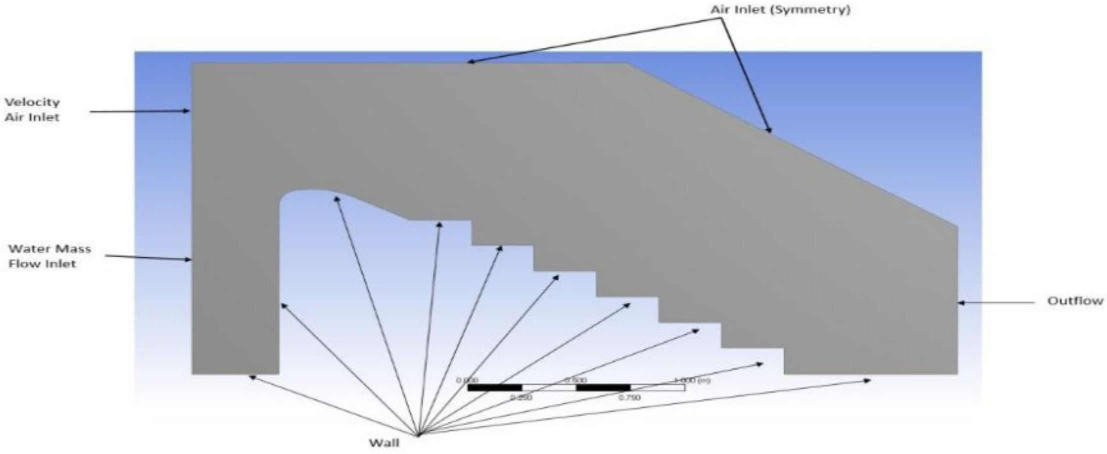


Figure 3.1 Geometrical model with 6 steps

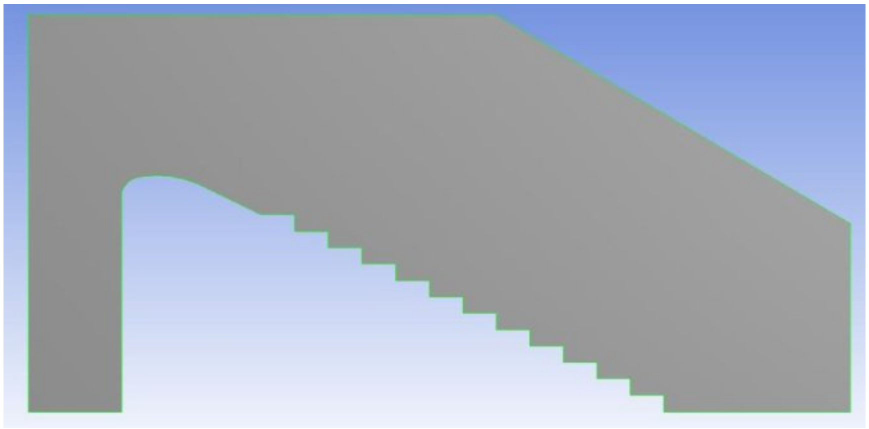


Figure 3.2 Geometrical model with 12 steps

3.3.2 Meshing

Meshing of the structure means fragmentation or discretization of the whole fluid domain so that analysis of the flow in the domain region can be carried out block by block. Instead of considering the whole domain, analysis will proceed from one cell to another. Meshing is required so that solver is able to solve for numerous equations as the complex geometrical model is now divided into finite number of standard shapes. Figure 3.3 and 3.4 shows the fine meshing for both the cases

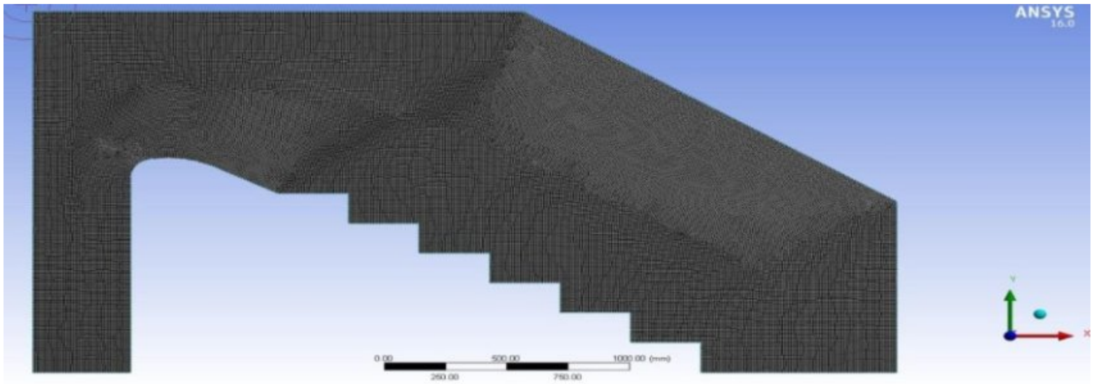


Figure 3.3 Meshing of model with 6 steps

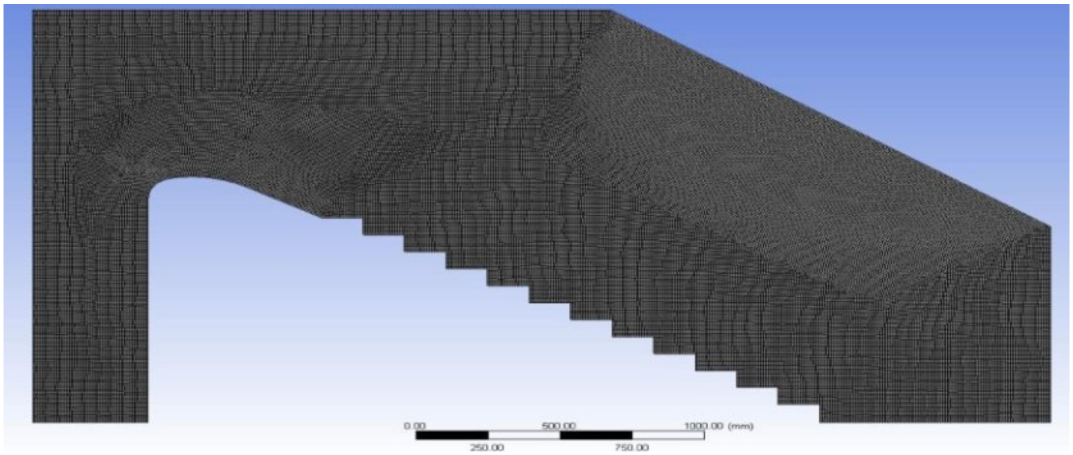


Figure 3.4 meshing of model with 12 steps

3.3.3 Fluent set up

After meshing step, for a particular model, first and foremost step is defining of gravity in negative y direction. The flow over the spillway includes movement in both X and Y direction. The governing equations for computing the results are Reynolds averaged Navier stoke equation which incorporates the conservation of mass and conservation of momentum principle. Volume of fluid model is used for defining the intermixing of 2 different fluids and for accessing the effect of turbulence k € model has been used.

3.3.4 Materials

Define 2 materials of fluid: water and air

3.3.5 The boundary conditions

- **Inlet**

  The inlet was further divided into two – air inlet and water inlet. First of all, for the whole domain the inlet fluid assigned was air. Then using the technique of patching, up to a height of water over the crest of spillway the fraction of air for the region is given zero. This automatically fills that portion of the geometry with the water creating the 2 phase inlet. Inlet for the air is pressure inlet. The inlet for the water is mass flow rate inlet since from the previous studies; we have the value of discharge

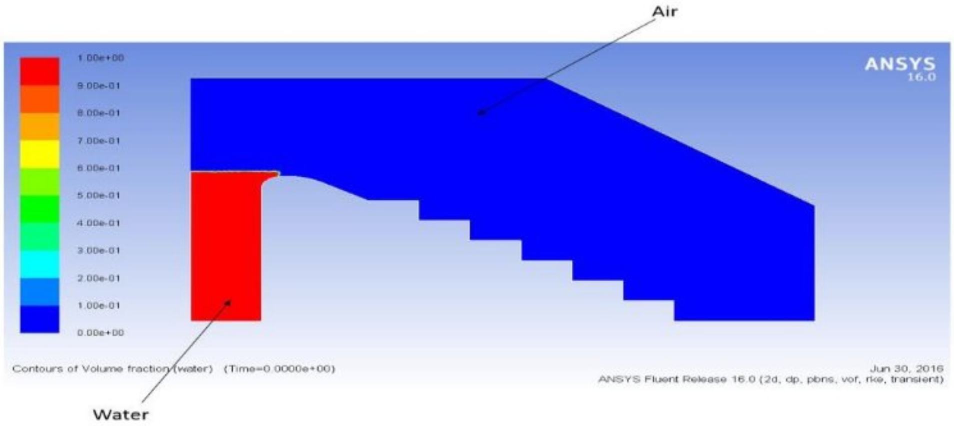


Figure 3.5 inlet description

- **Symmetric boundary conditions**

  If there is no flow across the boundary and velocities normal to the boundary are assigned to be zero. Moreover, the scalar flux from the boundary is zero. In the cases where symmetric boundary conditions are used, the values of attributes immediate to the domain of solution are taken as values at the nearest node within the domain.

- **Wall boundary conditions**

  Also known as no slip boundary condition, this boundary condition is the most common boundary that is used in the confined fluid flow problems .this is the most appropriate condition for the flow velocity components at the wall. Out of both the components of velocity, the normal velocity component can be initialized with zero and tangential velocity component is initialized with the velocity of the wall.

- **Outlet**

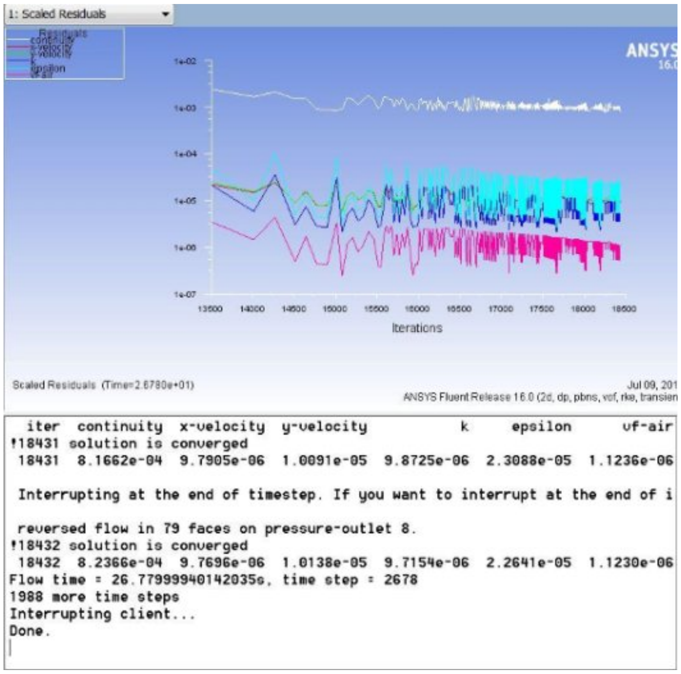
At the downstream end of the spillway i.e. At outlet, the pressure boundary condition is assigned

3.3.6 Solution

The next step is to initialize the solution. The no. of iterations is set to be and the solution is computed until it is converged. It is necessary for a solution to get converged. After the analysis, the results can be viewed under CFX-POST which is embedded in ANSYS workbench.

Upwind scheme is the simplest and one of the most stable techniques of discretization characterized by a feature that it is more dissipative according to the flow. It makes use of the value upstream in the domain region to find out the property on the boundaries of these cells and then makes their use to ascertain the values at the center of each cell**.** This way the chain is propagated from one end of the domain to another end**. Momentum spatial discretization or second order upwind** scheme is used for the discretization part and 1e-06 or the default value of 1e-03 may be taken as absolute criteria for convergence.

Figure showing iterations value for the solution to be converged for various models

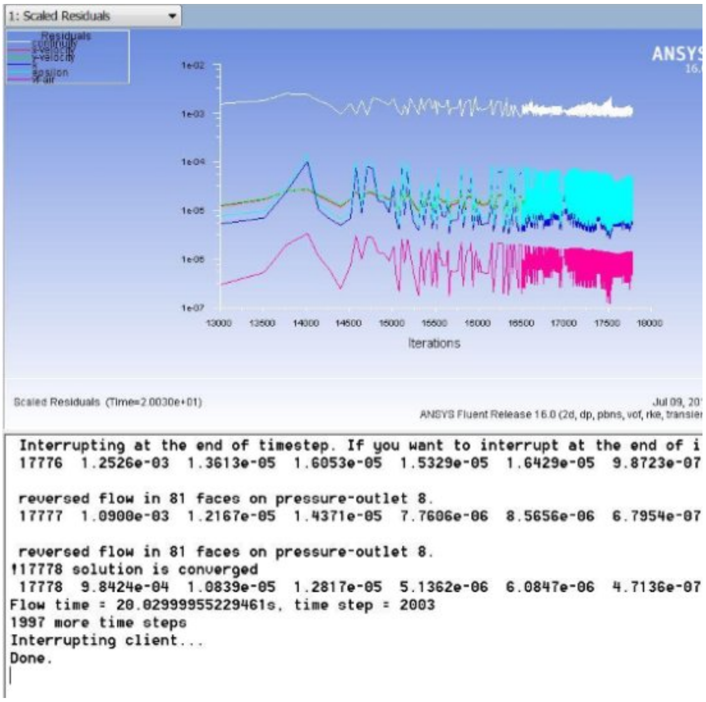


```
  iter  continuity  x-velocity  y-velocity           k     epsilon      vf-air
!18431 solution is converged
 18431  8.1662e-04  9.7905e-06  1.0091e-05  9.8725e-06  2.3088e-05  1.1236e-06

 Interrupting at the end of timestep. If you want to interrupt at the end of i

 reversed flow in 79 faces on pressure-outlet 8.
!18432 solution is converged
 18432  8.2366e-04  9.7696e-06  1.0138e-05  9.7154e-06  2.2641e-05  1.1230e-06
Flow time = 26.77999940142035s, time step = 2678
1988 more time steps
Interrupting client...
Done.
```

Figure 3.6 Convergence of results in case of mass flow rate 12. 36kg/s with 6 steps



```
 Interrupting at the end of timestep. If you want to interrupt at the end of i
 17776  1.2526e-03  1.3613e-05  1.6053e-05  1.5329e-05  1.6429e-05  9.8723e-07

 reversed flow in 81 faces on pressure-outlet 8.
 17777  1.0900e-03  1.2167e-05  1.4371e-05  7.7606e-06  8.5656e-06  6.7954e-07

 reversed flow in 81 faces on pressure-outlet 8.
!17778 solution is converged
 17778  9.8424e-04  1.0839e-05  1.2817e-05  5.1362e-06  6.0847e-06  4.7136e-07
Flow time = 20.02999955229461s, time step = 2003
1997 more time steps
Interrupting client...
Done.
```

FIGURE 3.7 Convergence of results in case of mass flow rate 19.5 kg/s with 12 steps

3.4 THE GOVERNING EQUATIONS FOR THE FLOW

Fluctuating velocity field is one of the main properties of turbulent flow. These fluctuations results in mixing of the transported vector quantities for example energy and momentum. The instantaneous governing equations are time averaged to detach the small scales and as a consequence, a set of less expensive equations containing fewer additional unknown variables is obtained. These unknown (turbulence) variables are computed in terms of modeled variables in turbulent models.

This method of time averaging is called Reynolds's averaging. When this is completed, then solution variables in the instantaneous Navier stokes equation are fragmented into mean (time averaged) and fluctuation components (Reynolds's decomposition). For the velocity components total velocity is written as

$$u_i \equiv \overline{u_i} + u_i'$$

Where $\overline{u_i}$ and u_i' are the average and fluctuating components respectively

Simulations of compressible flow were computed by solving the **unsteady Reynolds's averaged Navier stokes equations** which are expressed in the form:of set of two equations namely continuity equation and momentum equation.

1) **Continuity equation**

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

(equation 3.3)

For the study we have taken incompressible flow so 1st term will reduce to zero

2) Momentum equation

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

(equation 3.4)

Source of these equations: https://en.wikipedia.org/wiki/Navier%E2%80%93Stokes_equations

It is a Simplified form in which the term related to compressibility is ignored

Where t is time

U_i , U_j are the velocity component

X_i , X_j are the coordinate component

ρ is density

μ is dynamic viscosity

μ_t is the turbulent viscosity

The corrected pressure p' is obtained from the following expression

$$p' = p + \frac{2\rho k}{3}$$

(equation 3.5)

Where p is the pressure

K is the kinetic energy

The turbulence model which is incorporated in this study is k-€ realizable model. Ever since it was developed in 1972, its utility in industrial flow simulations has demonstrated its feasibility, robustness, economy and reasonable accuracy for a large range of turbulent flows. The model is a semi empirical model based on modeled transport equations for the turbulence kinetic energy k and its dissipation rate €. The turbulence kinetic energy k and its dissipation rate € are obtained from the following set of equations:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + G + \rho\varepsilon$$

(equation 3.6)

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{K}G - C_{2\varepsilon}\rho\frac{\varepsilon^2}{K}$$

(equation 3.7)

In which μ_t is obtained by the following relationship

$$\mu_t = \rho c_\mu \frac{k^2}{\varepsilon}$$

(equation 3.8)

C_μ=0.09 is an experimental constant

σ_k =1

$\sigma_{€}$ =1.3

$C_{1€}$ =1.44

$C_{2€}$ = 1.92

G is the generation turbulence kinetic energy

$$G = \mu_T = \left\{\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right\}\frac{\partial u_i}{\partial x_j}$$

(equation 3.8)

Source: https://en.wikipedia.org/wiki/K-epsilon_turbulence_model#Applications

3.5 VOLUME OF FLUID (VOF) MODEL IN FLUENT

The VOF method is based on the presumption that the two fluids are not interpenetrating. A variable is given to each phase (fluid) which considers that how much percentage of the computational cell is considered by that fluid phase. This variable is called volume fraction of that phase. These volume fractions sum up to unity in each computational cell

Denoting α_q as the volume fraction of the q_{th} face three possibilities for a given phase can be noted:

$\alpha_q = 0$ the cell is empty of the q_{th} face

$\alpha_q = 1$ the cell is fully filled with the q_{th} face

$0 < \alpha_q < 1$ the cell is partially filled with that phase and partly with some another

The presence of component phase in each cell determines the properties appearing in the transport equations. In a system with n phases the property p in each cell is given by

$$p = \sum_{q=1}^{n} \alpha_q p_q$$

(equation 3.9)

The momentum equation is solved throughout the domain and the phase shares the resulting velocity field. The momentum equation stated below is described in a slightly different way from mentioned above depending upon the volume fractions of all the phases.

$$\frac{\partial}{\partial t}(\rho\vec{u}) + \nabla\cdot(\rho\vec{u}\vec{u}) = -\nabla p + \nabla\cdot\left[\mu\left(\nabla\vec{u} + \nabla\vec{u}^{T}\right)\right] + \rho\vec{g} + \vec{F}$$

(equation 3.10)

Here ρ is the density, μ is the dynamic viscosity, p is the static pressure and finally $\rho\vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces respectively.

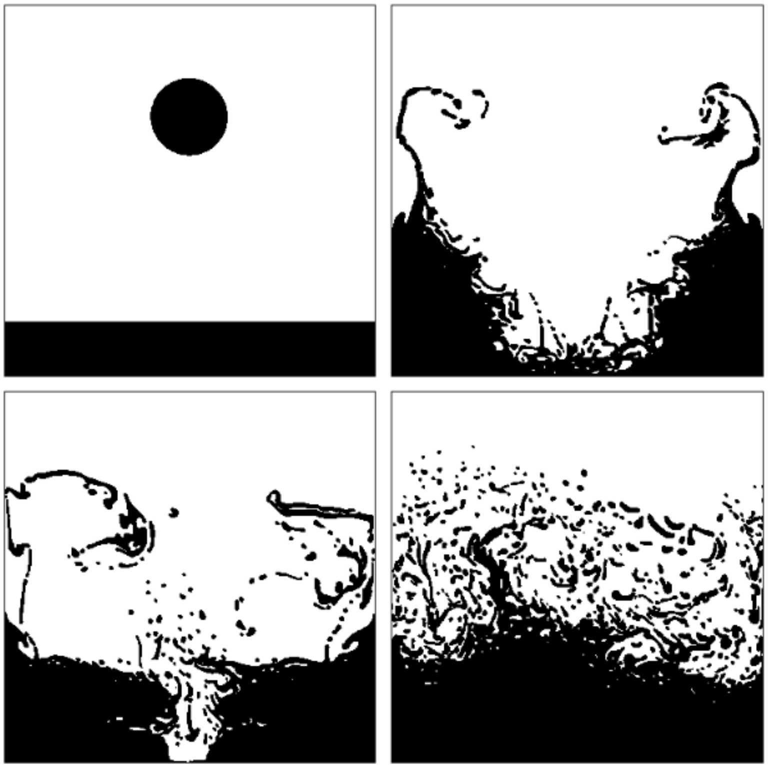


Figure 3.8 volume of fluid model

Source: https://en.wikipedia.org/wiki/Volume_of_fluid_method

3.6 K epsilon model

K-epsilon (k-ε) turbulence model is one of the most widely used turbulence model applicable in all the software of CFD. The simulations of flow characteristics for turbulent flow regime are carried out by this model. It is a basically a two equation model which depicts a general idea of turbulence in the fluid flow with the help of two transport equations (Partial differential equations). The early stimulus for the K-epsilon model was not only to enhance the mixing length model but also to find the substitute for numerically assigning scales of turbulent length in tranquil to complex flows.

- The determination of the energy in the turbulence of the flow is done by first transported variable called turbulent kinetic energy (k).
- The determination rate of dissipation of the turbulent kinetic energy is done by the second transport variable which is turbulent dissipation (ε).

The well-established model which is adequate enough for resolving over the boundary layer is standard k−ε model. With the advent of time and emerging researches an improvement over the standard k−ε model is developed which is realizable k−ε model. It is a comparatively new development and varies from the standard k−ε model in certain aspects. . A new formula for the turbulent viscosity is encountered in realizable k−ε model and different transport equation for the energy dissipation rate, ε, which is referenced from an exact equation for the transport of the mean-square vorticity fluctuation. The word "realizable” indicates that the model is in agreement with certain mathematical constraints applicable on the Reynolds stresses, also in agreement with the physical nature of turbulent flows. It introduces a Variable Cμ instead of constant. The advantage of the realizable k-ε model is that it gives better quality predictions of the movement rate of water jets. It also depicts better results for flows involving vortices, boundary layers with adverse pressure gradients, flow recirculation and boundary layer separation. Both the standard k-ε model and the RNG k-ε model are not realizable. In every aspect of comparison, virtually, Realizable k-ε model has a superior ability to analyze the mean flow of the complex fluid structure interaction problems.

3.7 Energy dissipation over a stepped spillway

The dissipation of turbulent kinetic energy (the energy arising from the turbulent eddies in a fluid flow) is the rate at which there occurs absorption of turbulent energy by the breakdown of eddies into smaller and smaller eddies till the time it is ultimately converted into heat by viscous forces. It is expressed in kinetic energy per unit mass per second (m^2/s^3)

When flow occurs over a stepped spillway, then potential energy decreases and the kinetic energy is also restricted due to iterative impinging action on steps which otherwise in the absence of steps would have gone on increasing.

$$\frac{\Delta H}{H_{max}} = 1 - \frac{0.54*\left(\frac{d_c}{h}\right)^{0.275} + 1.715*\left(\frac{d_c}{h}\right)^{-0.55}}{\frac{2}{3}+\frac{H_{dam}}{d_c}}$$

(equation3.11)

Where H_{dam} = dam crest head level above the heel (m)

H_{max} = total head

y_c= critical flow depth

ΔH= head loss

h= step height

3.8 Sample results of only 1 flow model when mass flow rate is 12.36 kg/s and no. of steps are 6

3.8.1 Air volume fraction- as two different inlets are provided for water and air initially they are unmixed but as the flow proceeds, the two different fluids gets intermixed leading to a 2 phase system.

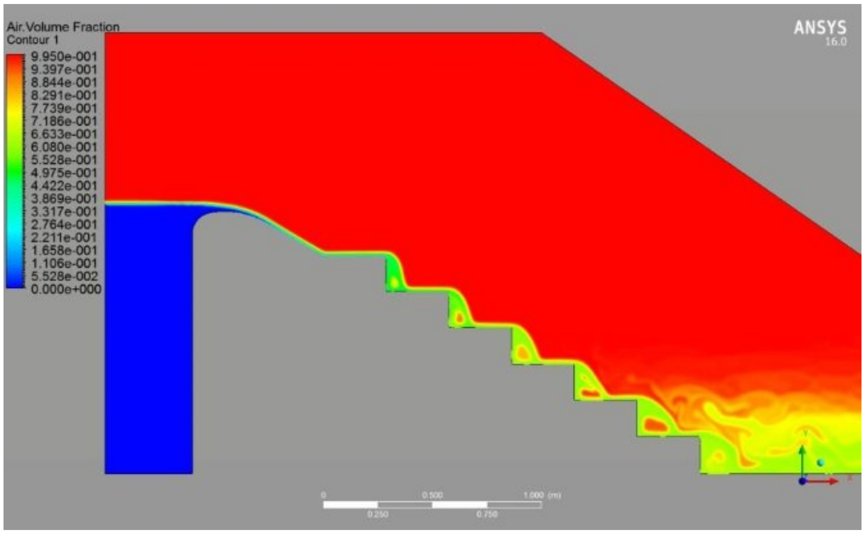


Figure 3.9 Air volume fraction

3.8.2 **Velocity contours**

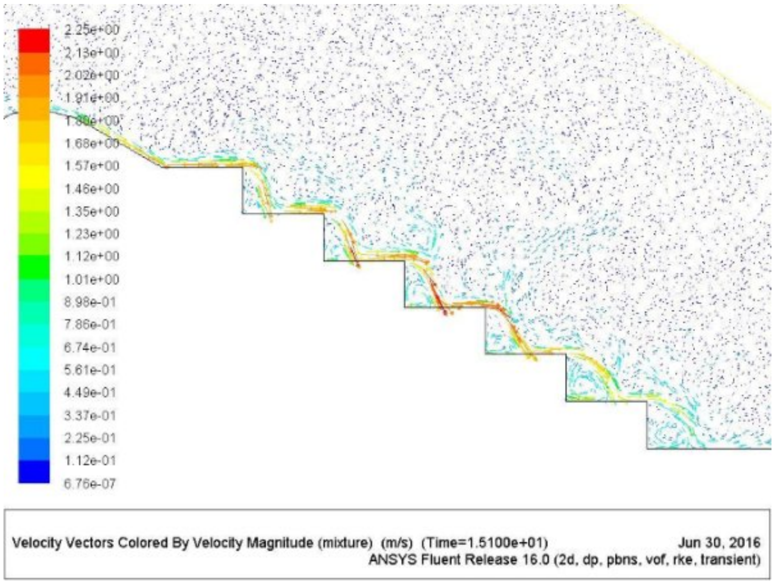


Figure 3.10 Velocity contours

3.8.3 **Turbulent kinetic energy-** this is the kinetic energy generated due to turbulence or eddies only. They are expressed in joule/kg. kg here is the weight of fluctuations in turbulence

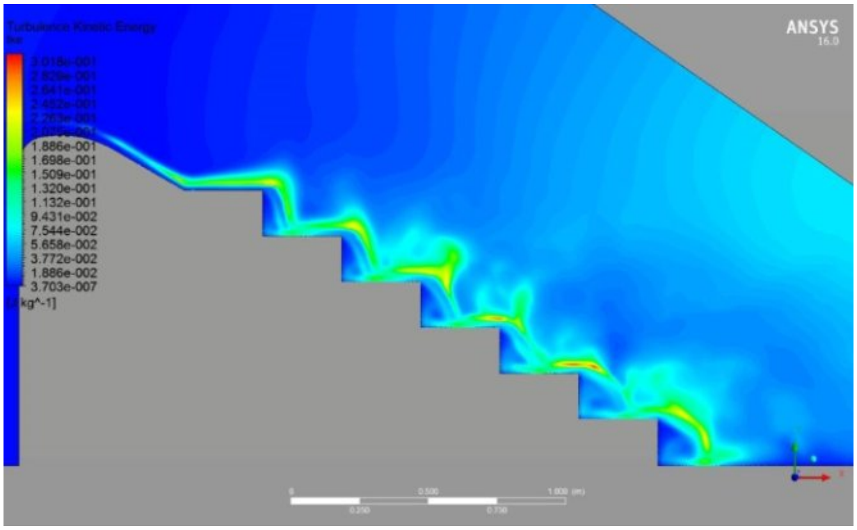


Figure 3.11 Turbulent kinetic energy

3.8.4 **Turbulent eddy dissipation** – this signifies the rate of dissipation of kinetic energy

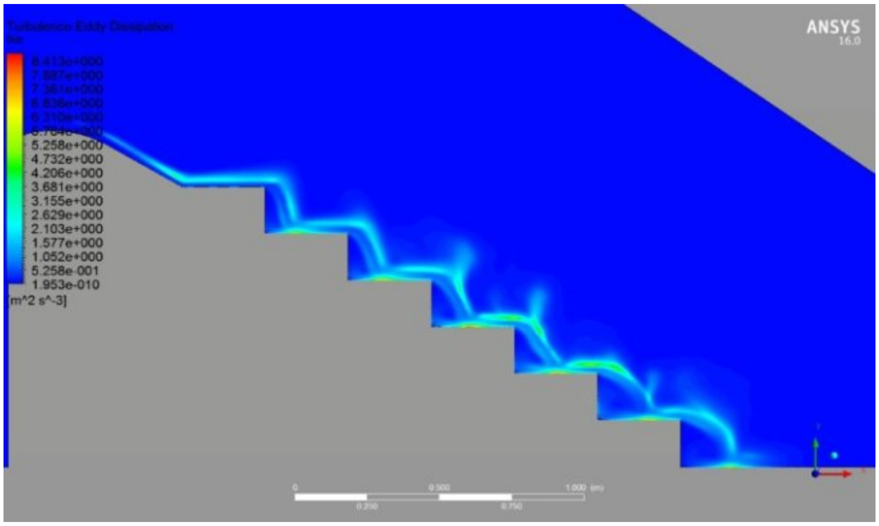


Figure 3.12 Turbulent eddy dissipation

CHAPTER 4

NUMERICAL DATA

This chapter consists of data obtained From Fluent after the calculations are over. There are total of 6 cases which are run in the software. The variants in mass flow rates are 12.36 kg/s, 19.5 kg/s and 28.5 kg/s and the variants of steps are with step heights 0.166m(6 steps for total height) and 0.0833m (12 steps for total height). The maximum eddies dissipation rate, velocity, pressure and turbulent kinetic energy at each step is calculated.

Numerical data on each step in quantitative form as obtained in ansys fluent:

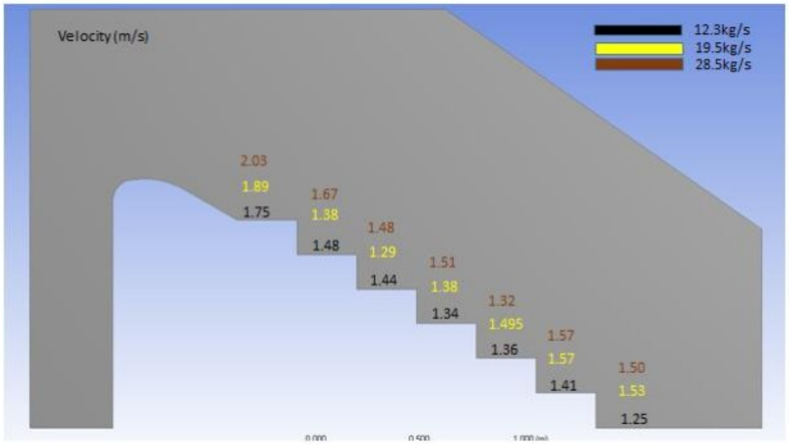


FIGURE 4.1 velocity magnitude on each step for different discharges for 6 steps

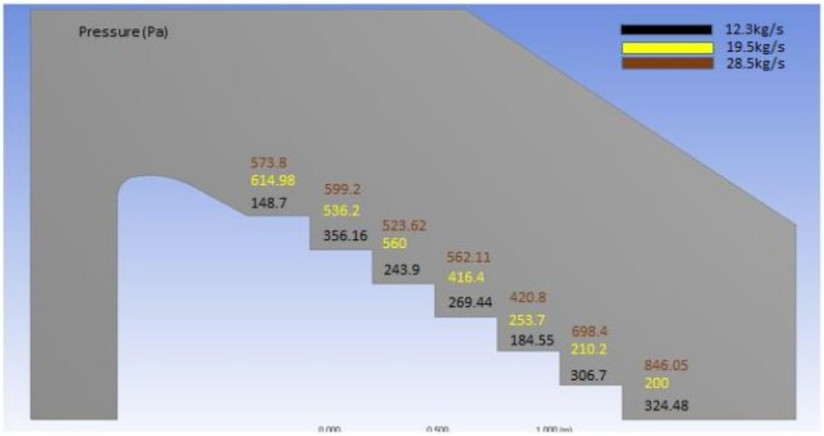


FIGURE 4.2 pressure magnitude on each step for different discharges for 6 steps

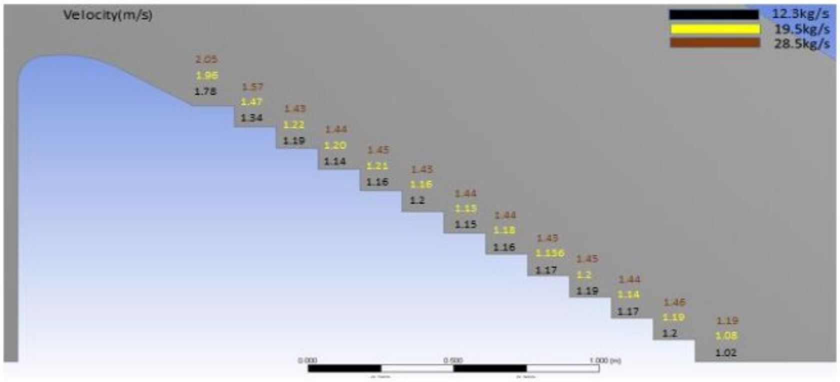


FIGURE 4.3 velocity magnitude on each step for different discharges for 12 steps

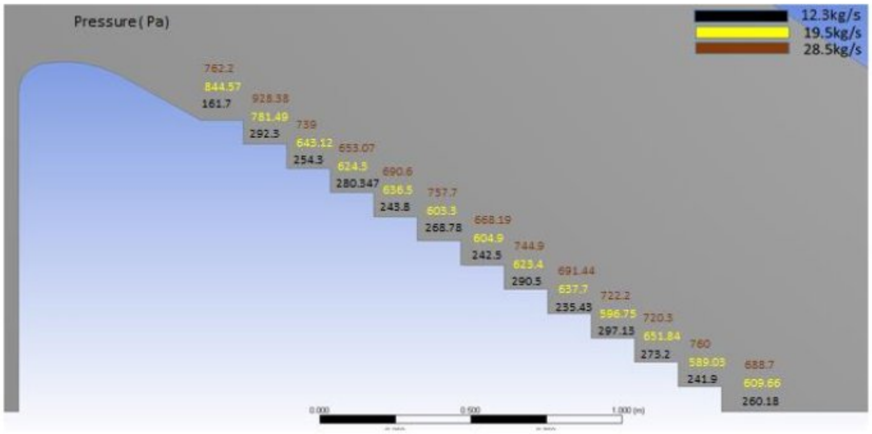


FIGURE 4.4 pressure magnitude on each step for different discharges for 12 steps

The results are tabulated for different cases:

1. COMPUTATIONS OF ENERGY DISSIPATION ON EACH STEP

Jet Impinging on step no.	Height from bottom	Eddy dissipation in units $m^2/s3$
1	0.83	2.4
2	0.664	3.4
3	0.498	4.79
4	0.332	4.68
5	0.166	2.6
6	0	2.07

Table 4.1 Mass flow rate is 28.5 kg/s and step height 0.166m

Jet impinging on step no.	Height from bottom	Eddy dissipation in units m^2/s3
1	0.83	0.90128
2	0.664	1.1866
3	0.498	5.2402
4	0.332	2.02018
5	0.166	1.78
6	0	1.6018

Table 4.2 Mass flow rate is 19.5 kg/s and step height 0.166m

Jet impinging on step no.	Height from bottom	Eddy dissipation in units m^2/s3
1	0.83	1.76
2	0.664	2.168
3	0.498	2.019
4	0.332	2.15
5	0.166	2.074
6	0	2.01343

Table 4.3 Mass flow rate is 12.36 kg/s and step height 0.166m

Jet impinging on step no.	Height from bottom	Eddy dissipation in units m^2/s3
1	0.9136	0.789
2	0.8306	0.99
3	0.7476	1.24
4	0.6646	1.45
5	0.5816	1.45
6	0.4986	1.51
7	0.4156	1.45
8	0.3326	1.44
9	0.2496	1.52
10	0.166	1.48
11	0.0836	1.69
12	0	1.73

Table 4.4 Mass flow rate is 12.36 kg/s and step height 0.0833m

Jet impinging on step no.	Height from bottom (meter)	Eddy dissipation in units m^2/s3
1	0.9136	1.08
2	0.8306	1.289
3	0.7476	1.36
4	0.6646	1.42
5	0.5816	1.41
6	0.4986	1.59
7	0.4156	1.43
8	0.3326	1.45
9	0.2496	1.50
10	0.166	1.69
11	0.0836	1.73
12	0	1.69

Table 4.5 Mass flow rate is 19.5 kg/s and step height 0.0833m

Jet impinging on step no.	Height from bottom (meter)	Eddy dissipation in units m^2/s3
1	0.9136	1.38
2	0.8306	1.43
3	0.7476	1.63
4	0.6646	1.93
5	0.5816	2.25
6	0.4986	2.16
7	0.4156	1.96
8	0.3326	2.0
9	0.2496	2.2
10	0.166	2.26
11	0.0836	2.46
12	0	2.55

Table 4.6 Mass flow rate is 28.5 kg/s and step height 0.0833m

2 COMPUTATIONS OF TOTAL ENERGY HEAD

TABLE 4.7- Mass flow rate is 12.36 kg/s and step height 0.166m

Step number	**Datum head**	**Pressure head**	**Kinetic head**	**Total head**
1	0.8306	0.114	0.162	1.1366
2	0.6646	0.088	0.1056	0.8582
3	0.4986	0.109	0.0915	0.6991
4	0.3326	0.0966	0.094	0.5232
5	0.166	0.0928	0.1013	0.3601
6	0	0.06	0.0796	0.136

Initial energy =1.24

Final energy = 0.135

Loss of energy =89.2%

TABLE 4.8- Mass flow rate is 19.5 kg/s and step height 0.166m

Step number	**Datum head**	**Pressure head**	**Kinetic head**	**Total head**
1	0.8306	0.0546	0.088	0.9822
2	0.6646	0.0571	0.0848	0.8065
3	0.4986	0.04244	0.097	0.6380
4	0.3326	0.0258	0.1139	0.4723
5	0.166	0.0214	0.125	0.3124
6	0	0.02038	0.124	0.1444

Initial energy =1.254

Final energy = 0.1444

Loss of energy =88%

TABLE 4.9- Mass flow rate is 28.5 kg/s and step height 0.166m

Step number	**Datum head**	**Pressure head**	**Kinetic head**	**Total head**
1	0.8306	0.06	0.142	1.0326
2	0.6646	0..05	0.112	0.8266
3	0.4986	0.0573	0.116	0.672
4	0.3326	0..0428	0.088	0.4634
5	0.166	0.0712	0.1256	0.3628
6	0	0.08624	0.1146	0.2008

Initial energy =1.269

Final energy = 0.2008

Loss of energy =84.17%

TABLE 4.10- Mass flow rate is 12.36 kg/s and step height 0.0833m

Step number	Datum head	Pressure head	Kinetic head	Total head
1	0.9136	0.03	0.0915	1.0351
2	0.8306	0.026	0.07	0.926
3	0.7476	0.028	0.0662	0.8418
4	0.6646	0.0248	0.0685	0.758
5	0.5816	0.0274	0.0734	0.6824
6	0.4986	0.0247	0.0674	0.5907
7	0.4156	0.0296	0.0685	0.5137
8	0.3326	0.024	0.0697	0.426
9	0.2496	0.030	0.0721	0.3517
10	0.1666	0.0278	0.069	0.2634
11	0.0836	0.02465	0.0733	0.18
12	0	0.0265	0.053	0.0795

Initial energy =1.24

Final energy = 0.0795

Loss of energy =93.58%

TABLE 4.11- Mass flow rate is 19.5 kg/s and step height 0.0833m

Step number	Datum head	Pressure head	Kinetic head	Total head
1	0.9136	0.079	0.110	1.1026
2	0.8306	0.0655	0.075	0.9711
3	0.7476	0.0636	0.0733	0.8845
4	0.6646	0.0648	0.074	0.8034
5	0.5816	0.0614	0.068	0.711
6	0.4986	0.06144	0.065	0.625
7	0.4156	0.0635	0.0709	0.55
8	0.3326	0.0650	0.0657	0.4633
9	0.2496	0.0608	0.0733	0.3837
10	0.1666	0.0664	0.0662	0.299
11	0.0836	0.0600	0.0721	0.2157
12	0	0.06214	0.059	0.1214

Initial energy =1.254

Final energy = 0.1214

Loss of energy =90.3%

TABLE 4.12- Mass flow rate is 28.5 kg/s and step height 0.0833m

Step number	**Datum head**	**Pressure head**	**Kinetic head**	**Total head**
1	0.9136	0.0946	0.1256	1.1338
2	0.8306	0.0753	0.1042	1.0101
3	0.7476	0.066	0.1056	0.9156
4	0.6646	0.0704	0.10716	0.8422
5	0.5816	0.077	0.1042	0.7628
6	0.4986	0.068	0.1056	0.6722
7	0.4156	0.0759	0.1054	0.5975
8	0.3326	0.0704	0.1042	0.5072
9	0.2496	0.0735	0.107	0.4303
10	0.1666	0.0734	0.1056	0.345
11	0.0836	0.0774	0.1086	0.2696
12	0	0.0702	0.072	0.1422

Initial energy =1.269

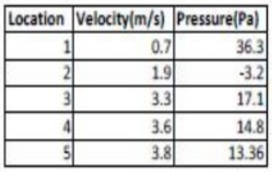
Final energy = 0.1422

Loss of energy =88.87%

CHAPTER 5

ANALYSIS OF NON STEPPED SPILLWAY

In this chapter, a non-stepped spillway of same model dimension is considered and the energy dissipation or efficiency of the spillway is calculated.



Location	Velocity(m/s)	Pressure(Pa)
1	0.7	36.3
2	1.9	-3.2
3	3.3	17.1
4	3.6	14.8
5	3.8	13.36

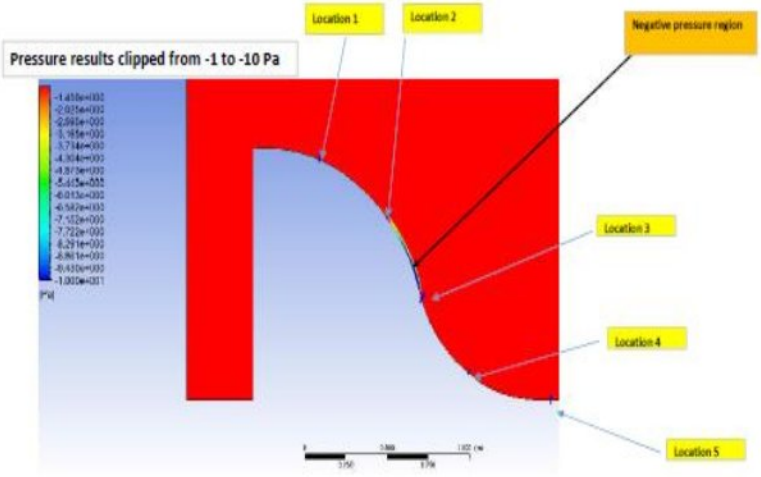


Figure 5.1 Sample model analysis of a non-stepped spillway when the mass flow rate is 12.36 kg/s

ANALYSIS OF NON STEPPED SPILLWAY

Case 1 Mass flow rate 12.36 kg/s

Initial total energy head at crest= 1.24+0.025= 1.265 m

Final energy head at the toe= 0.73 m

Energy loss= 42.29%

Case 2 Mass flow rate is 19.5 kg/s

Initial total energy head at crest= 1.254+0.028= 1.282

Final energy head at the toe= 0.782

Energy loss= 39.15%

Case 3 Mass flow rate is 28.5 kg/s

Initial total energy head at crest= 1.269+0.027= 1.296

Final energy head at the toe= 0.819

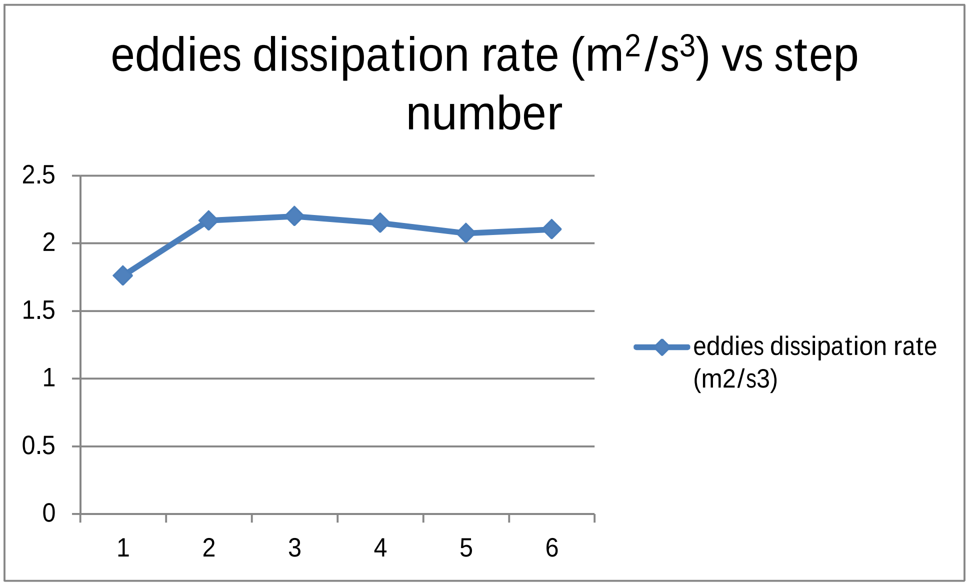
Energy loss= 36.8%

Remark:

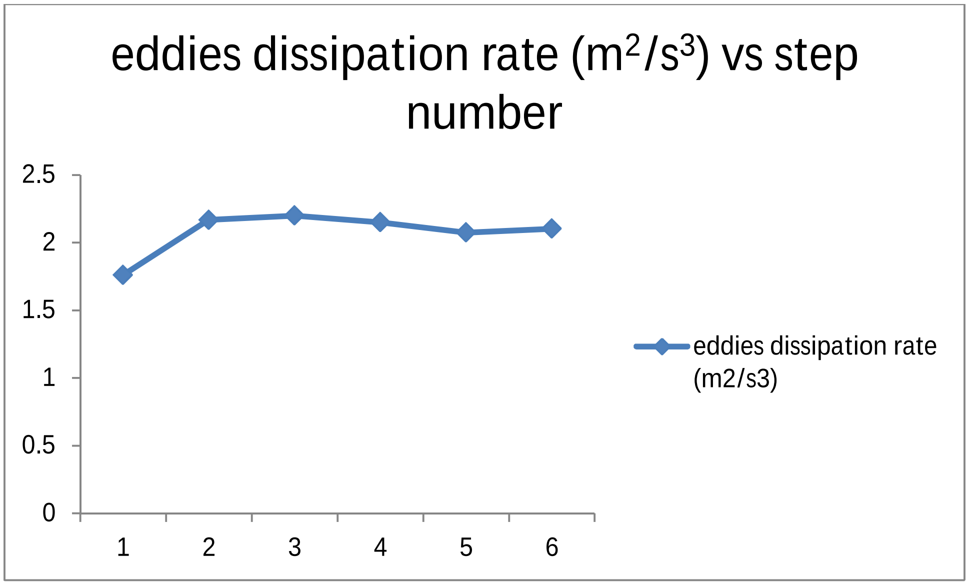
The energy dissipation or efficiency in case of non-stepped spillway is very less as compared to those in stepped spillways for all the three discharge rates.

CHAPTER 6

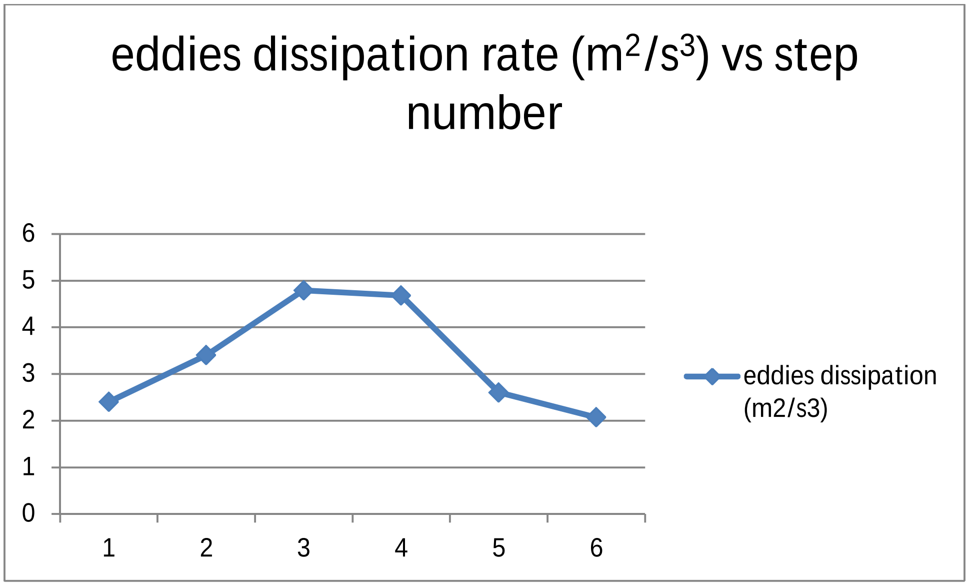
RESULTS AND DISCUSSIONS



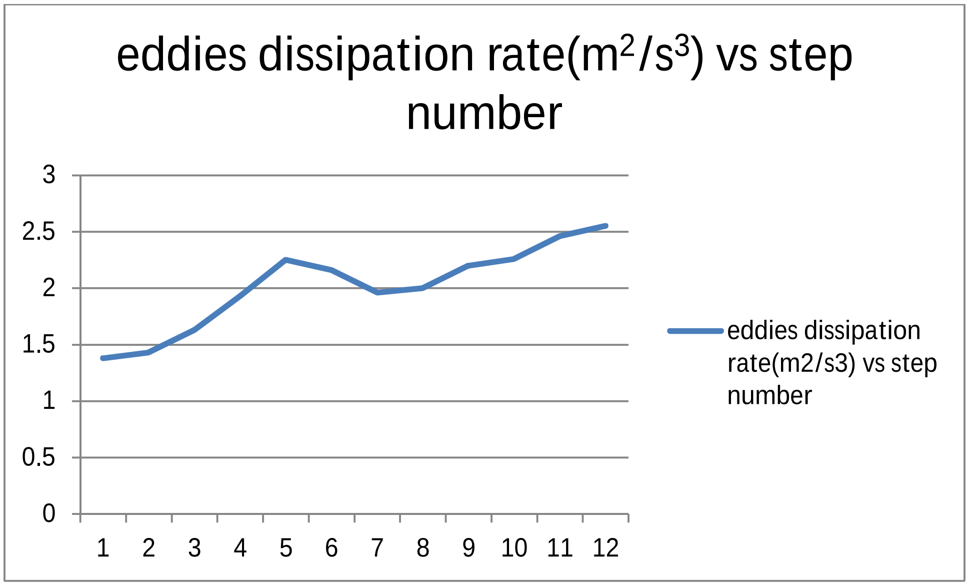
GRAPH 6.1 Variation of eddies dissipation on each step for mass flow rate 12.36Kg/S



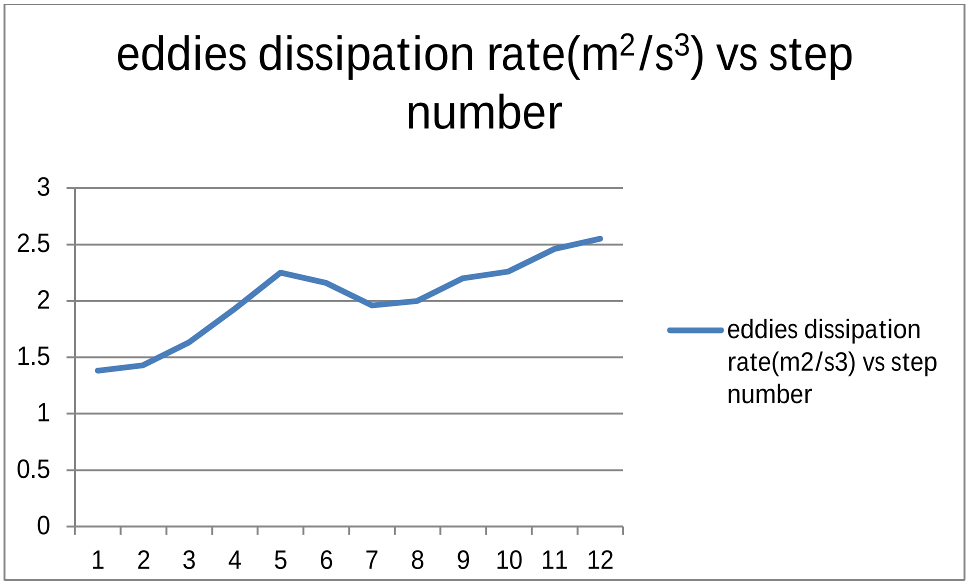
GRAPH 6.2 Variation of Eddies Dissipation On Each Step For Mass Flow Rate 19.5 Kg/S



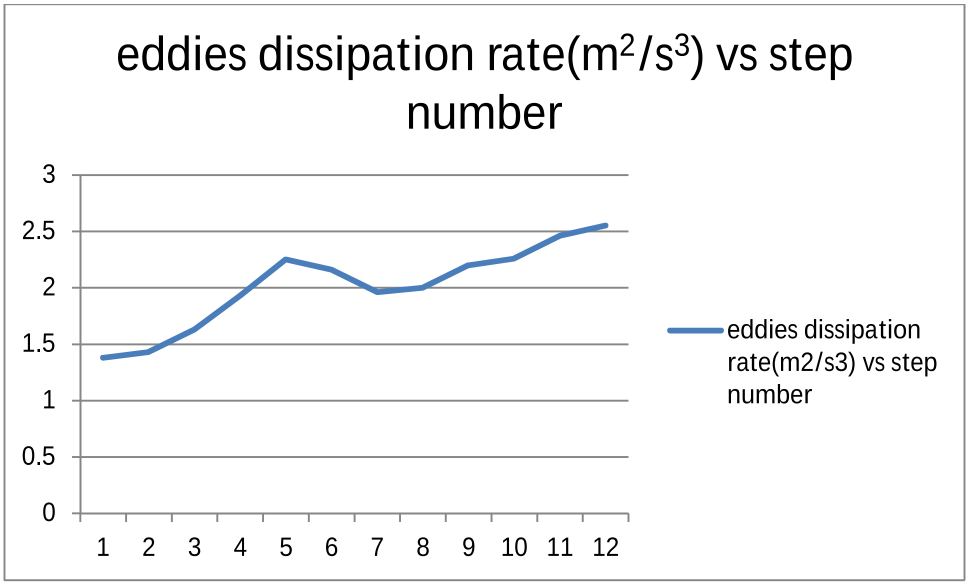
GRAPH 6.3 Variation of Eddies Dissipation on Each Step For Mass Flow Rate 28.5 Kg/S



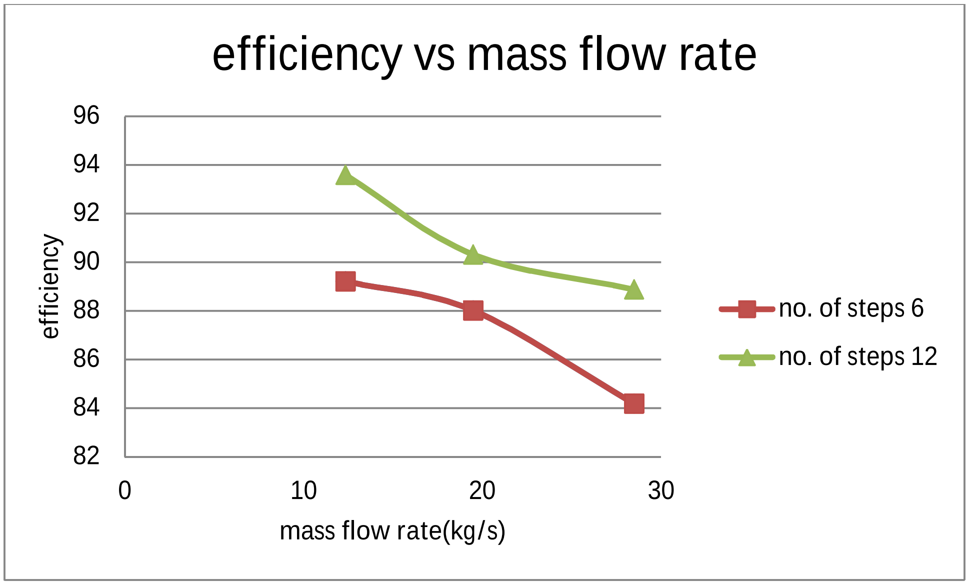
GRAPH 6.4 Variation of Eddies Dissipation On Each Step For Mass Flow Rate 12.36 Kg/S



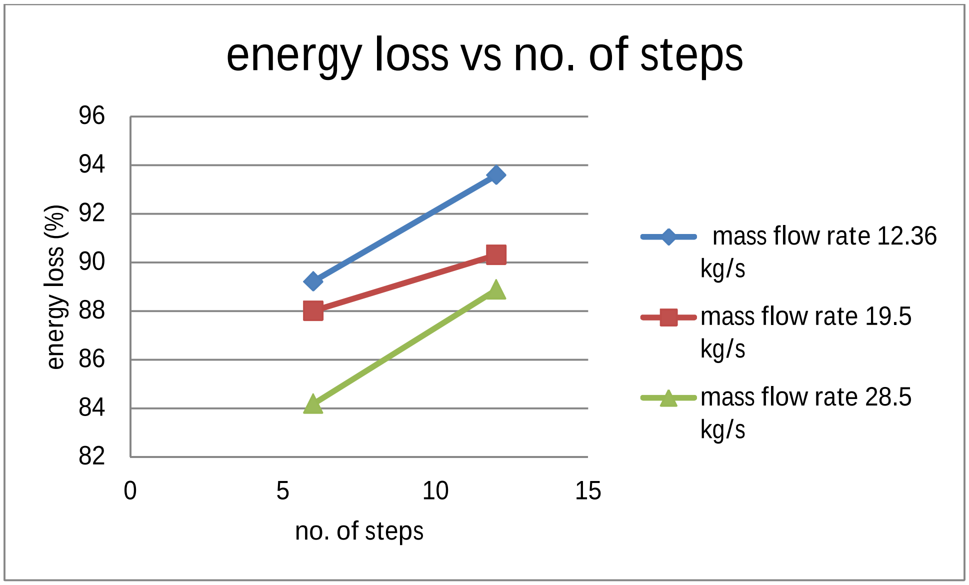
GRAPH 6.5 Variation Of Eddies Dissipation On Each Step For Mass Flow Rate 19.5 Kg/S



GRAPH 6.6 Variation of Eddies Dissipation On Each Step For Mass Flow Rate 28.5 Kg/S



GRAPH 6.7 Variation of efficiency of spillway and mass flow rate for both the cases with step height 0.166m (6 steps) and 0.0833m (12 steps)



GRAPH 6.8 variation of energy loss with no. of steps for all the mass flow rate

DISCUSSIONS

1) In all the cases with 6 steps, flow is nappe since critical depth is less than the critical onset depth for skimming flow2) From observation we find that eddy dissipation is maximum at any intermediate step. It first increases, reaches the maximum eddy dissipation rate and then decreases. However no general trend is observed between eddy dissipation with variation in discharge.

3) In the cases with 12 steps, we observed that the maximum rate of eddy dissipation almost occurs at the last step unlike the case with 6 steps where maximum rate of eddy dissipation occurred at some intermediate steps.

4) Hence we can observe that when the step height is more, then the maximum rate of energy dissipation is attained earlier but when the step height is less then maximum rate of energy dissipation occurs near the toe of the spillway.

5) Energy dissipation of a stepped spillway is much more than that of a non- stepped spillway as computed in chapter 5.

6) There is no danger of cavitation anywhere in the stepped spillway for all the mass flow rates. The pressure magnitude at any point is more than the vapour pressure of water as observed from the pressure contours.

CHAPTER 7

CONCLUSIONS

7.1 From the above computations, we can conclude the following things:

1 The energy dissipation decreases with increase in discharge for a constant step height.

2 In nappe flow the energy dissipation is more as compared to that in skimming flows

3 The energy dissipation decreases with decrease in the no. of steps (or increase in step height) for a constant total height of dam and a constant discharge

4 when the no. of steps are less (step height is more) then the maximum eddies dissipation rate occurs earlier than that in the case when no. of steps are more (step height is less)

5 energy dissipation or efficiency of a stepped spillway is much more than that of non-stepped spillway.

7.2 Comparison of energy dissipation computed from manual approach with help of fluent software and that from numerical formula

Equation 3.11 is used for numerical formula approach.

Critical depth (m)	Energy loss from Manual approach	Energy loss from Numerical approach	% error
0.04	89.2%	86.5%	3.0%
0.054	88%	84.37%	4.124%
0.069	84.17%	82.25%	2.28%

Table 7.1 comparison of energy dissipation when there are 6 no. of steps

Critical depth (m)	Energy loss from Manual approach	Energy loss from Numerical approach	% error
0.04	93.58%	90.21%	3.6%
0.054	90.3%	88.414%	2.08%
0.069	88.87%	86.64%	2.51%

Table 7.2 comparison of energy dissipation when there are 12 no. of steps

7.3 Future scope of study

In further studies,

a) The effect of variation of downstream slope on the energy dissipation can be studied.
b) Energy dissipation also depends upon the climatic slope of entry of water on the steps. Effect of different entry angles on the efficiency of spillway can be considered.
c) 3 dimensional model analysis can also be done and results can be compared with 2 dimensional model analysis.
d) In this thesis, slope of spillway is kept constant. Therefore the step size is kept constant. The step size can be kept non uniform as we have observed that rate of energy dissipation is not same on each step. It may lead to economical design.

CHAPTER 8

REFERENCES AND RESOURCES

- Christodoulou, G.C, (1993) Energy dissipation on stepped spillway, Journal of Hydraulic Engineering, ASCE, *Vol.119, No.5, 644-649.*
- Chamani, M.R. and Rajaratnam(1994), N, Jet flow on stepped spillways, Journal of Hydraulic ASCE, Engineering. Vol.120, No.2, 254-259
- Chamani, M.R. Rajaratnam, N, (1994) Onset of skimming flow on stepped spillways, Journal of Hydraulic Engineering. ASCE, Vol.125, No.9, 969-971
- Chen, Q, Dai, G Liu H (2002) volume of fluid model for turbulence numerical simulation of stepped spillway overflow ASCE journal of hydraulic engineering. Vol.128 p 683-688
- Boes, R.M. Hager, H.W. (2003), Hydraulic design of stepped spillways, Journal of Hydraulic Engineering. ASCE, Vol.129, No.9, 671- 679
- Chatila, J. Tabbara, M, Computational simulation of flow over stepped spillways, Journal of Computers and Structures, vol. 126,No.83, 2215-2224.
- Tabbara, M, M. Chatila, J. Awwad, R, Computational simulation of flow over stepped spillways, Journal of Computers and Structures, No.83, 2215-2224.
- Bjorn Margeisson, computational modeling of flow over a spillway
- NPTEL lecture notes from IIT Kanpur(Department of civil engineering) (image source of various types of flow over stepped spillway)
- Naderi Rad. a, Evaluation of energy dissipation in stepped spillway variety simple, sloping edges and by using numerical method by software (FLUENT), MS, University of Shiraz, 1386.
- Barani, GA Rahmana, MB Bagheri H (2005) optimization of stepped spillway dimension and investigation of flow energy dissipation over a physical model. Journal of applied sciences vol.5, no. 5, p 878-882

- Gongalez, C.A. Chanson, H (2007). Hydraulic design of stepped spillways and downstream energy dissipators for embankment dams. Dam engineering vol.17, no.4, p 223-244

- Naderi rad, I Teimuri, M (2010). An investigation of flow energy dissipation in simple stepped spillways by numerical model. European journal of scientific research, vol. 45, no. 4, p 544-553

- Hamid reza vosoughifar, alam dolatshah (2013). Evaluation of fluid flow over stepped spillways using finite volume method as a novel approach- strojniski vestnil, journal of mechanical engineering no. 301-310

- Moussa Rassaei, Sedigheh Rahbar(2014), numerical flow model stepped spillways in order to maximize energy dissipation using FLUENT software, IOSR journal of engineering, vol.4, 2250-3021

- ANSYS FLUENT (2014) user guide, ANSYS Inc. Lebanon