Numerical Simulation of Lake Tap Flow

Jagtap[1] R. N., Patil[2] B. M, Bhosekar[3]V. V. and Sohoni[2] V. S.

[1]M.Tech. Student, [2]Professor, Department of Civil Engineering, BharatiVidyapeeth University College of Engineering Pune, India
[3] Scientist , Central Water & Power Research Station, Khadakwasla, Pune, India

---

ABSTRACT -

Presently, it is emphasized that there is a need of an energy resource that will affect least and will be available and fulfill the increasing demands for energy in a rapidly developing country like India. Hydroelectric power is considered to be the most vital, simple for design, pollution free, emission-free,easy for maintenance and an inexpensive renewable source of energy with zero fuelling cost. Hydropower with lake tapping is increasing in India.

This study deals with assessment of Lake Tap flow in the intake tunnel due to the proposed Lake Tap III at Koyna Hydroelectric Project using CFD techniques and evolution of a suitable geometry of the muck pit considering the lake tap flow to be two-phase(water and boulders) flow.

Keywords: Lake tap flow, two-phase flow, CFD, FLUENT, muck pit, k-ϵ model.

1. INTRODUCTION

1.1 Energy Scenario

Electricity is our part of life and it is very hard to live without electricity. Increase in power potential of a country is considered to be the most important factor among the all. Flourishing power generation industry reflects progress and the prosperity of a country.

The highlights of World Energy Congress (WEC,2013) i.e. Word Energy Scenarios composing energy futures to 2050 include that by 2050, the total primary energy supply would increase between 27 % and 61%. Fossil fuels would be remaining as the dominant energy source supplying between 59 % to 77% of the global primary energy mix, while the share of renewable energy sources in the global energy mix will record the biggest growth to reach between 20% and 30%.

India is the fourth largest consumer of energy in the world after the United States, China and Russia. Population of India is increasing. To meet energy demand for this increasing population there is need for production of energy in maximum quantity. India is having lowest Average Power per capita available among the top ten power consuming countries and significantly below the world's average. As GDP growth accelerates to an ambitious 8 to 10%, the shortage of power will become more severe.

The first major source (57%) of energy in India is coal. Coal is the non-renewable source of energy; it is available in the limited quantity in the nature. Also it produces the environmental pollution by emission of carbon which is harmful to the nature. As it is the largest contributor to the greenhouse effect other sources are the biomass, diesel, gas, Nuclear etc. Due to combustion of diesel used in vehicles, machineries and plants, CO is produced and contributes to the greenhouse effect and badly affects the atmosphere. Biomass is comparatively less harmful for the adverse effect to the environment but very limited in quantity as its resources will not last for longer duration. This emphasizes the need of an energy resource that will affect least and will be available and fulfill the increasing demands for energy of India, a rapidly developing country.

The second major source (19%) of energy in India is Hydropower. Hydropower is by far the most significant renewable resource of electricity exploited to date. Hydro-electric engineering is concerned with the efficient and economic conversion of energy freely available from a supply of water deposited at a suitable head by the action of the cycle of evaporation and rainfall produced by the effect of solar radiation. An essential requirement is, therefore, that the water should be at a suitable height above a lower reference point to where the water could flow and be discharged. The difference in levels between the water and discharge point represents the potential energy that would become available for use. Hydroelectric plants convert this potential energy of water into an electrical output. The process involves flow of water from the source, through the turbine to the turbine outflow (tailrace), which acts as a sink and in the process of conversion use is made of water turbines, of associated civil structures and of rotating electrical machinery.

In hydro-electric plants, the potential energy of water is converted into kinetic energy first passing through the tunnel to the power house. The kinetic energy of the water is converted into mechanical energy in the water turbines.

The mechanical energy of the water turbine is further utilized to run the electric generator.

1.2 Hydropower and Lake Tapping

Hydropower is the best from many points of view. These include simplicity of design, easy maintenance, absence of pollution and zero fuelling cost, as the source is perpetual one and goes to waste if not exploited. It is a pollution free source since it does not contribute to air and water pollution to Green House Effect.

The site of hydro-electric station is selected on the criterion of water availability at economical head. The lakes/reservoirs situated in high hilly area are therefore best suited. Hence, lake tapping is gaining importance in India, particularly, in Maharashtra State. Two lake taps have been carried out (Huddar, 2010) and the third is in process at Koyana Hydroelectric Project. One lake tap is carried out recently at Modak Sagar near Mumbai.

If a dam is not having outlets of sufficient capacity or if some additional outlets are required for power generation etc. then dam should be provided with new/additional outlet system. This outlet system will serve as intake system for the power generation system. This intake system can be a nonconventional intake system by piercing the lake from bottom by blasting the rock plug using dynamites. This is called as Lake Tapping. After the blasting of rock plug, the water in the dam flows by gravitation force and strikes on the blades of turbine which is installed at the outlet of the intake system and power is generated.

The principle in lake tapping is to provide a tunnel under a lake, leaving a short rock plug towards the lake floor below the tapping level. The final plug is then blasted, piercing the lake floor from underneath. This technology is being used in Norway and numerous lake taps have been carried out in Norway alone, apart from beginning waterways for hydropower, drinking- and irrigation water purposes. This method has also been utilized for landing of oil and gas pipes from offshore fields. Important necessities for successful construction of lake tapping are suitable ground conditions. The successful construction of the great number lake taps is based on the fact that the piercing process has been controlled by a highly trained professional having practical experience in these challenging works. Due to the difficulty and risk of damage to existing structures, a thorough and detailed design based on detailed geological data and experience from similar work is a precondition for a successful design.

For lake tapping, studies of proper location, ground conditions, possible flow conditions in the tunnel, geometry of muck pit etc are essential. Attempt is made in this study to assess Lake Tap flow in the proposed intake tunnels at Koyna dam using numerical modeling technique - a CFD software and evolve a suitable geometry of the muck pit. The lake tap flow is considered here to be one as well as two phase flow containing water and boulders.

1.3 Two-phase flow simulation

Water- Boulder flow in tunnel has been a popular mode of transportation in tunnel. Circular underground tunnels are used for long distance transportation of variety of materials in bulk quantities. During the several past decades, the complex computational scheme describing the microscopic processes in the solid- liquid flow and the computing costs has been a drawback to the development and use of commercial software. Ling et al.(2002) investigated the double slurry flow in the pipe for the fully turbulent flow using Eulerian granular multiphase model. Lin and Ebadian (2008) used a simplified three-dimensional algebraic slip mixture model and the RNC k-ϵ turbulent model while focusing on the developing process of volume of fraction, density distributions and mean velocity profiles.

Al Araby et al(2004) performed a numerical study on the single phase combined free and force convection in the entrance region of a horizontal pipe with its wall temperature fixed as a constant value.

Lake tap flow simulations are carried out by Chaudhary (2011) and Deolalikar et al (2001). The present paper describes numerical model study for simulation of Lake Tap flow in Koyna hydroelectric project. The study is mainly focused on the simulation of single phase flow (only water) and two Phase flow (Water-Boulders) for different alternatives and flow conditions and suggestion of suitable geometry of the muck pit.

2. NUMERICAL MODELLING TECHNIQUE

In the present study, FLUENT software (2006) is used to simulate the lake tap flow. It is based
on numerical solution of 3D equations of flow namely, mass conservation i.e. continuity equation, momentum conservation equation i.e. Reynolds Average Navier-Stokes (RANS)equations and the standard $k-\epsilon$ model for simulation of turbulent flows which is a model based on transport equations for the turbulence kinetic energy (k) and its dissipation rate (ϵ). The governing equations are :

A Governing Equations

The continuity equation for the mixture is

$$\frac{\partial}{\partial t}(\alpha * \rho) + \nabla.(\alpha * \rho * \vartheta) = \sum_{p=1}^{n} m$$

Where α denotes the volume fraction of single material, ρ is the density of a material, m is the mass flow rate.

The Navier-Stokes equations are,

$$\rho \frac{Du_i}{Dt} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho F_i$$

$$\rho \frac{De}{Dt} = -p\frac{\partial u_k}{\partial x_k} + \varphi + \frac{\partial}{\partial x_k}\left(k\frac{\partial T}{\partial x_k}\right)$$

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - 2/3 \frac{\partial u_k}{\partial x_k}\delta_{ij}\right)$$

$$\varphi = \tau_{ij}\frac{D\rho}{Dt} + \rho\frac{\partial u_i}{\partial x_i} = 0\frac{\partial u_i}{\partial x_i}$$

where, τ_{ij} is the viscous stress tensor with the bulk density omitted and φ is the dissipation function where u is the fluid velocity, ρ the density of fluid, T the absolute temperature, k the thermal conductivity, e the internal energy, F_i is a body force term, μ is the dynamic viscosity and δ_{ij} is the Kronecker delta.

k-ϵ model is the most commonly used turbulence model when modeling turbulent flow. The transport equation in k-ϵ is given by:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k$$

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(G_k + C_{3\epsilon}G_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon$$

Here, due to the mean velocity gradient generation of turbulence kinetic energy is G_k is the, due to buoyancy the generation of turbulence kinetic energy is G_b, Y_M is the contribution of the fluctuation dilation in compressible turbulence to the overall dissipation rate, $C_{1\epsilon}$, $C_{2\epsilon}$ and $C_{3\epsilon}$ are constants, σ_k and σ_ϵ are the turbulent Prandtl numbers for k and ∈ respectively, S_k and S_ϵ are user defined source terms.

Simple geometry is prepared in the GAMBIT software; a fine meshing is to be done by successive ratio and later the boundary conditions are given for the geometry. This file is exported into fluent software, after exporting it gives the inputs like mass flow rate, volume fraction, size of boulder, density of boulder etc. Two dimensional geometry is used to study the flow in tunnel and for solving the mass and momentum equation. The standard $k\text{-}\epsilon$ turbulence models with standard wall functions are used to solve the problem.

B Boundary Conditions

In the present study, pressure outlet boundary condition and the mass flow inlet boundary condition are imposed on the outlet and the inlet of the tunnel, respectively. In the water-boulder mixture flow the mass flow rate is set to180 m^3/s. The gravitational acceleration is considered as 9.81 m/s^2 in downward direction i.e. in negative 'y' direction and 101326 pa of Operating pressure is considered. The density of water is taken as 998.3 kg/m^3.The size of boulder is given to be 0.3 m.

GAMBIT software is used for mesh generation. To discretize the computation domain non uniform grid system with tetrahedral elements and a multi-block unstructured is used. For discretization schemes of both diffusion and convection terms, central difference and the second order upwind are selected, respectively.

3. STUDY AREA

Koyna river originates from the Sahyadri hill ranges in Maharashtra and flows in a north south direction to Helwak. The Koyna dam is situated at Koynanagar in Patan taluka of Satara district of Maharashtra. The Koyna Hydroelectric Project (KHEP) is situated at Koynanagar in Maharashtra. The project was initiated in late fifties with the construction of 103 m high ruble concrete dam across Koyna river with a gross storage of 2797.40 MCM (98.78 TMC). The hydro power development on this project has taken place in four stages over the past 40 to 50 years. In the stage 4 the Lake tapping techniques are used for the development with two Lake Taps were already completed and third is being planned.

4. MODEL SIMULATIONS

Simulations is carried out considering single phase flow of water considering different values of the angle of the base line of the muck pit with the horizontal (Θ) namely 0^0, 1^0, 2^0, 3^0, 4^0, 5^0, 6^0, 7^0, 8^0 and 9^0 degrees, corresponding to 10 different cases i.e. Case-1 to Case-10 of the angle. Typical results corresponding to the angle 0^0 and 9^0 in the form of flow pattern (Velocity vectors) and contours of velocity is shown in Figs. 1, 2 and 3 , 4 respectively.

From Fig. 1, it is seen that the velocity profile changes at different region of intake tunnel and muck pit. After blasting of rock plug, the water is flow at velocity ranging from 2.9 to 3.4 m/s in intake tunnel of initial stretch of 11 m. But after a stretch of 11 m it can be observed that, due to converging diameter of intake tunnel, the velocity rises suddenly up to 2.3 to 2.9 m/s in the bend portion of intake tunnel. After this water enters into the muck pit and observed that velocity in the muck pit is of the order of 1.2 to 2.3 m/s. It is seen that velocity near the top surface of muck pit is higher, while the velocity at the bottom of muck pit is very less and it ranges from 0.037 to 0.26 m/s. Later on the water is enter into the intake tunnel after the muck pit with higher velocity ranging from 4.0 to 4.5 m/s. Some circulation remains observed near the bottom of upstream portion of the muck pit. Simulations are carried out for the different 10 cases of angles ranging from 0^0 to 9^0. Due to the change in the angle the magnitude of velocities is not significantly change in magnitude of velocities in the upstream intake tunnel and muck pit. But in the downstream intake tunnel particularly at the outlet of the tunnel the magnitudes of velocity are significantly reduced. It can also observed that some circulation is observed near the bottom of downstream portion of the muck pit and reverse flow occurring at the bottom of muck pit, as the angles are increased. This circulation and reverse flow are causing energy loss leading to reduction in the magnitude of velocity at the outlet. Due to change in the value of angles from 0^0 to 9^0 reduction in the outlet velocity is occurring in the range 6% to 47%.

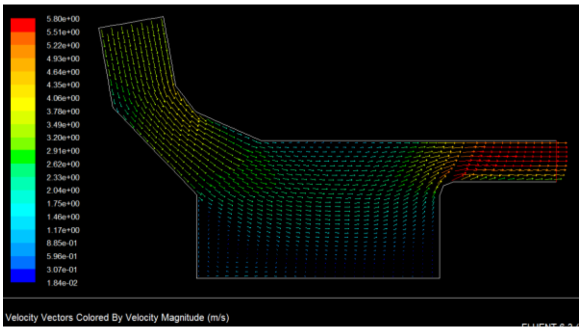

Fig.1: Vector plot for Case-1 with ϴ =0^0

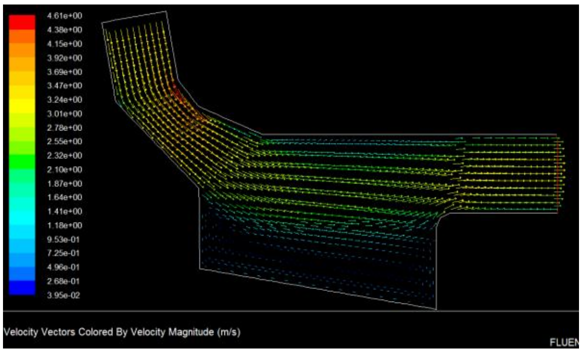

Fig.2: Vector plot for Case-10 with ϴ =9^0

The velocity contour plot are shown in Fig. 3 and 4 shows clearly the Velocity variation in the intake tunnel and muck pit as observed from the Corresponding vector plot in Fig.1 and 2.

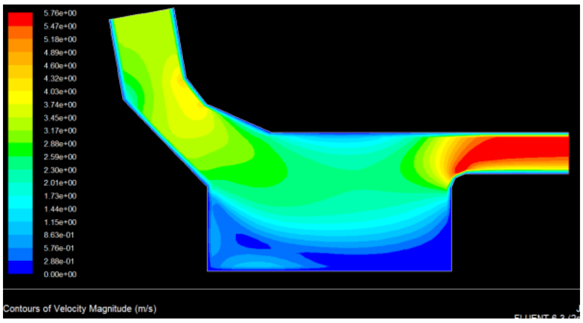

Fig.3: Velocity contour plot for Case-1 with ϴ =0^0

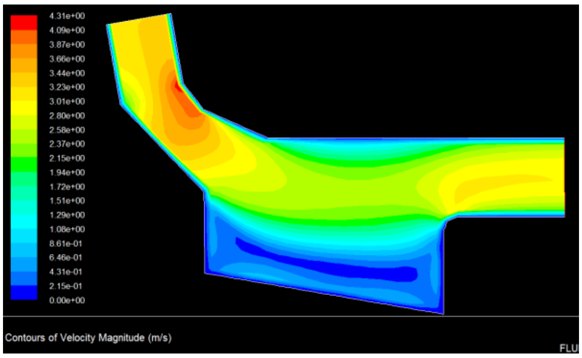

Fig.4: Velocity contour plot for Case-10 with ϴ =9^0

Table 1: Velocity changes in intake tunnel and muck pit

Sr. No.	Angles	Velocity in m/s			
		Upstream side intake tunnel		Muck pit	Downstream Intake tunnel
		In 11 m streach	After 11 m streach		
1.	0^0	2.9 to 3.2	2.3 to 2.9	1.2 to 2.3	4.4 to 5.8
2.	1^0	2.9 to 3.2	2.3 to 2.9	1.2 to 2.3	4.3 to 5.3
3.	2^0	2.9 to 3.2	2.3 to 2.9	1.2 to 2.3	3.9 to 4.9
4.	3^0	2.9 to 3.2	2.3 to 2.9	1.2 to 2.3	4.0 to 4.5
5.	4^0	2.9 to 3.2	2.3 to 2.9	1.2 to 2.3	3.7 to 4.1
6.	5^0	2.9 to 3.2	2.3 to 2.9	1.2 to 2.3	3.4 to 3.9
7.	6^0	2.9 to 3.2	2.3 to 2.9	1.2 to 2.3	3.2 to 3.7
8.	7^0	2.9 to 3.2	2.3 to 2.9	1.2 to 2.3	2.9 to 3.2
9.	8^0	2.9 to 3.2	2.3 to 2.9	1.2 to 2.3	2.7 to 2.9
10.	9^0	2.9 to 3.2	2.3 to 2.9	1.2 to 2.3	2.5 to 2.9

4.1 Flow Simulation – Two Phase (Water-Boulders)

Similarly, simulations are carried out considering two phase flow of water-boulder considering different values of the angles of the base line of the muck pit with the horizontal (Θ) namely 0^0, 1^0, 2^0, 3^0, 4^0, 5^0, 6^0, 7^0, 8^0 and 9^0 degrees, corresponding to 10 different cases i.e. Case-1 to Case-10 of the angle. Typical results corresponding to angle 3 and 9 degrees in the form of contours of volume fraction of boulder are shown in Figs. 5 and 6 respectively.

The volume of boulders collected in the muck pit corresponding to the different angles is tabulated in Table 2 which indicates that due to the change in the angle from 0^0 to 9^0, deposition of boulders in the muck pit is occurring in the range 358.70 m^3 to 363.39 m^3 with maximum deposition being 413.15m^3 for angle 3^0.

As compared to the incoming volume of boulders (423.5m^3) due to the blasting of rock plug, the volume of boulders collected in the muck pit for different angles is varying from about 85% to 98% with the maximum corresponding to the angle 3^0. Thus it can be concluded that the angle 3^0 is the optimum angle for the maximum deposition in the muck pit. The variation of volume collected in the muck pit with respect to the angle is plotted in the Fig. 9 which also indicates the optimum angle 3^0 for the maximum deposition of boulders in the muck pit. The present study confirms the optimum value of the angle, Θ which has been already decided.

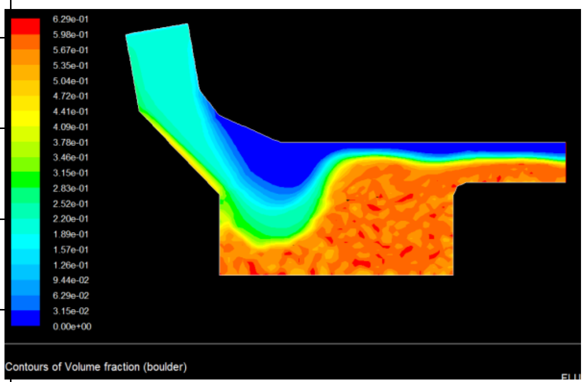

Fig.5: Contours of volume fraction (Boulder) for Case-1 with Θ $=0^0$

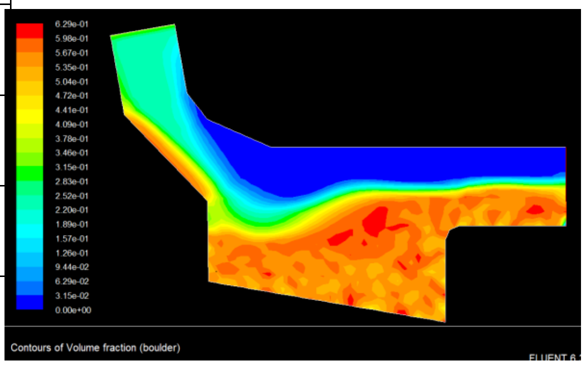

Fig. 6: Contours of volume fraction (Boulder) for Case-10 with Θ $=9^0$

Table 2: Volume of boulders in muck pit

Sr. No.	Angle (Deg.)	Total Volume of Boulder (m^3)		% volume of boulders in muck pit
		Volume of boulders in muck pit (m^3)	Volume of boulders collected in D/S intake tunnel (m^3)	
1.	0	358.70	64.80	84.69
2.	1	383.20	40.30	90.48
3.	2	407.80	15.70	96.05
4.	3	413.15	10.35	97.55
5.	4	403.60	19.90	95.30
6.	5	399.18	24.32	94.25
7.	6	392.01	31.50	92.56
8.	7	381.91	41.59	90.17
9.	8	374.68	48.82	88.47
10.	9	363.39	60.11	85.80

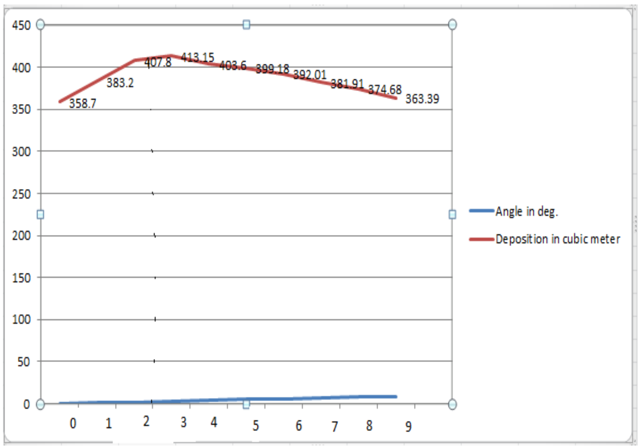

Fig.9 : Variation of volume collected in the muck pit with respect to the angle

5. CONCLUSIONS

This study is carried out to asses Lake Tap flow in the intake tunnel due to the proposed Lake Tap III at Koyna Hydroelectric Project using CFD techniques i.e. EFLUENT software and suitable geometry of the muck pit is evolved considering the lake tap flow to be single phase (water only) flow as well as two-phase(water and boulders) flow. The main conclusions of the study are as follows.

1. Simulations are carried out considering single phase flow of water considering different values of the angle of the base line of the muck pit with the horizontal (ϴ) namely 0, 1, 2, 3, 4, 5, 6, 7, 8 and 9 degrees. The change in the magnitude of velocity in different parts of intake channel and muck pit due to the change in angle of base line of muck pit with horizontal was observed. Due to the change in the angle, there is no significant change in magnitude of velocities in the upstream intake tunnel and muck pit. But in the downstream intake tunnel, particularly, at the outlet of the model the magnitude of velocity is significantly reduced. Thus if the angle is increased then vortex formation is becoming strong and due to this the energy loss takes place and velocity at the outlet is reduced.

2.Simulations are also carried out considering two phase flow of water-boulder considering different values of the angle of the base line of the muck pit with the horizontal (ϴ) namely 0, 1, 2, 3, 4, 5, 6, 7, 8 and 9 degrees which indicates that due to the change in the angle from 0^0 to 9^0 ,deposition of boulders in the muck pit is occurring in the range 358.70 m^3 to 363.39 m^3 with the maximum deposition being 413.15m^3 for angle 3^0.

3. As compared to the incoming volume of boulders (423.5m^3) due to the blasting of rock plug, the volume of boulders collected in the muck pit for different angles is varying from about 85% to 98% with the maximum corresponding to the angle 3^0. Thus it can be concluded that the angle 3^0 is the optimum angle for the maximum deposition in the muck pit.

4. This study indicates two fold effect of the increase in the angle. Due to the increase in the angle eddies are formed and energy is dissipated causing the reduction in the velocity. Secondly, the reduction in velocity in the muck pit causes larger deposition.

5. The present study confirms the optimum value of the angle, ϴ i.e. 3^0 which has been already decided.

ACKNOWLEDGEMENT

The authors express their deepest gratitude to Dr. A. R. Bhalerao, Principal, BharatiVidyapeeth University College of Engineering, Pune, for his support and encouragement.

.

REFERENCES

[1] Al Araby, M. A., Salem M. K. and Mahmoud,M. M. ‘ Laminar Combined Free and Forced Convection Heat Transfer in the Entry Length of a Horizontal Pipe,’ Proceeding of the ASME Heat Transfer/ Fluid Engineering Summer Conference, HT/FED 2004, Vol. 1, Charlotte, 11-15 July 2004.

[2] ANSYS FLUENT, Theory Guide, 2006.

[3] ANSYS FLUENT, User's Guide, 2006.

[4] Chaudhary Hanif, M. and Prashanth Reddy, H 'Mathematical Modeling of Lake Tap flows' *J. Hydraul. Eng.*, 137(5), 611–614, 2011.

[5] Deolalikar P. B., Bhosekar V.V and Sridevi M. I. , 'Lake Tapping –Experience from hydraulic model studies', National Conference on Hydraulics and Water Resources – HYDRO-2001, 2001
.

[6] Huddar S. N., 'Geotechnical aspects of the design of underwater lake tap in Koyna hydroelectric project Sage IV', Water & Energy International, Sep. 2010.

[7] Ling, J, Lin C. X. and Ebadian M.A , ' Numerical Investigation of Double- Species Slurry Flow in a Straight Pipe Entrance,' ASME 2002 International Mechanical Engineering Congress and Exposition, Heat transfer Vol. 4, New Orleans, 17-22 November 2002 .

[8] Lin C. X. and. Ebadian, M. A. 'A Numerical Study of Developing Slurry Flow in the Entrance Region of a Horizontal Pipe,' Computers and Fluids, Vol. 37, No.8, 2008.

[9] World Energy Council, 'Word Energy Scenarios composing energy futures to 2950', World Energy Congress (WEC) was held at Daegu, Korea on 14 October 2013.