OPTIMIZATION OF SEDIMENTATION TANK BY CFD

A DISSERTATION

SUBMITTED IN PARTIAL FULFILMENT OF THE REQUIREMENTS FOR THE AWARD OF THE DEGREE OF

MASTER OF TECHNOLOGY

IN

ENVIRONMENTAL ENGINEERING

Submitted By

Sudheer Pandia

ROLL NO. 2K16/ENE/16

Under The Supervision Of

DR BHARAT JHAMNANI

DEPARTMENT OF ENVIRONMENTAL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

JULY, 2018

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

CANDIDATE’S DECLARATION

I Sudheer pandia, 2K16/ENE/16 student of M.Tech, Environmental Engineering, hereby declare that the project Dissertation title “Optimization of sedimentation tank by CFD” which is submitted by me to the Department of Environmental Engineering, Delhi Technological University, Delhi in partial fulfilment of the requirement for the award of the degree of Master of Technology, is original and not copied from any source without proper citation. This work has not previously formed the basis for the award of any Degree, Diploma Associateship, Fellowship or other similar title or recognition.

Place: Delhi (SUDHEER PANDIA)

Date:

ENVIRONMENTAL ENGINEERING DEPARTMENT

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

CERTIFICATE

I hereby certify that the Project Dissertation titled "Optimization of sedimentation tank by CFD" which is submitted by [Sudheer pandia], Roll No.2K16/ENE/16, [Environmental Engineering Department], Delhi Technological University, Delhi in partial fulfillment of the requirement for the award of the degree of Master of Technology, is a record of the project work carried out by the student under my supervision. To the best of my knowledge this work has not been submitted in part or full for any Degree or Diploma to this University or elsewhere.

Place: Delhi (DR.BHARAT JHAMNANI)

Date: SUPERVISOR

ACKNOWLEDGEMENT

It is a great pleasure to have the opportunity to extend my heartiest gratitude to everybody who helped me throughout this thesis. It is distinct pleasure to express my deep sense of gratitude and indebtedness to my learned supervisors **Dr. Bharat jhamnani** in the Department of Environmental Engineering, Delhi Technological University for their invaluable guidance, encouragement, and patient review. His continuous inspiration has enabled me to complete this major project.

I would also like to take this opportunity to present our sincere regards to my Head of the Department **Prof. S. K. Singh** for his kind support and encouragement. I am thankful to my family members, friends and classmates for their unconditional support and motivation.

SUDHEER PANDIA

List of figures

Figure 1.1 flow chart of waste water treatment plant………………………………10

Figure 1.2 Rectangular sedimentation tank with horizontal flow………………….11

Figure 1.3 Circular sedimentation tank…………………………………………….12

Figure 1.4 Discrete settling…………………………………………………………12

Figure1.5 Flocculent settling………………………………………………………..13

Figure 1.6 Hindered settling ………………………………………………………..13

Figure 1.7 Compression settling…………………………………………………….13

Figure 1.8 Settling velocity in discrete settling……………………………………..14

Figure 3.1 Geometry of sedimentation tank without baffle…………………………26

Figure 3.2 Geometry of sedimentation tank with baffle wall……………………….26

Figure 3.3 Meshing of sedimentation tank without baffle wall……………………...27

Figure 3.4 Meshing at inlet…………………………………………………………..28

Figure 3.5 Meshing of sedimentation tank with a baffle wall………………………..29

Figure 3.6 Graph of grid dependency………………………………………………...30

Figure 4.1 Model validation graph…………………………………………………...31

Figure 4.2 Velocity contour without a baffle wall……………………………………32

Figure 4.3 Velocity contour with a baffle wall……………………………………… 32

Figure 4.4 Pressure contour of sedimentation tank with baffle………………………33

Figure 4.5 Pressure contour of sedimentation tank without baffle…………………..34

Figure 4.6 Path lines without baffle wall…………………………………………….35

Figure 4.7 Path lines with a baffle wall………………………………………………35

Figure 4.8 Particle tracking with baffle wall and residence time…………………..36

Figure 4.9 Particle tracking with baffle wall and velocity magnitude……………...37

Figure 4.10 Particle tracking without baffle wall and residence time………………37

Figure 4.11 Particle tracking without baffle wall and velocity magnitude………….38

Figure 4.12 Streamlines……………………………………………………………..38

Figure 4.13 Graph of Particle tracking with varying diameter………………………39

Figure 4.14 Graph of Efficiency of tank with varying particle diameter…………….39

Figure 4.15 Graph between particle injected and trapped…………………………....40

Table of Contents

Table of Contents 7
ABSTRACT 9
CHAPTER 1 10
INTRODUCTION 10
1.1 Wastewater 10
1.2 Wastewater generation and treatment 10
1.3 Sedimentation tank 12
1.4 Types of settling 13
1.5 Settling velocity in discrete settling 15
1.6 Aims and objectives 17
CHAPTER 2 18
LITERATURE REVIEW 18
2.1 CFD 18
2.2 Application of CFD in waste water 20
2.3 Application of CFD in sedimentation tank 20
2.4 Some important concepts in CFD software 22
CHAPTER 3 24
METHODOLOGY 24
3.1 Mathematical model 24
3.2 Governing equations 25
3.3 Geometry and meshing 26
3.4 Mesh independent study 29
CHAPTER 4 30
RESULTS AND DISCUSSIONS 30
4.1 Model validation 30
4.2 Comparison between velocity contours 31
4.3 Comparison between pressure contours 33

4.4 Path lines 34
4.5 Particle tracking 36
4.6 Streamlines 38
4.7 Particle tracking with varying diameter 39
4.8 Efficiency of tank with varying particle diameter 40
4.9 Graph between particle injected and trapped 41
CHAPTER 5 42
CONCLUSION 42
References 42

ABSTRACT

The aim of this study is to optimize the sedimentation tank with the application of CFD (computational fluid dynamics). In this study, we have used DPM i.e. discrete phase model for simulations and we assume multiphase solid and liquid. In multiphase simulation, we consider the momentum exchange between two phases. In this study, we simulate various contours and particle tracking by Lagrangian approach instead of Euler approach. Our focus is the difference in the results after applying a baffle wall in the sedimentation tank. Although there is a wide range of simulations in CFD but for simplicity, we assume the velocity inlet, particle diameter, dimensions of sedimentation tank and location of baffle is constant. For further investigation we can vary these parameters also but due to the limited power of a computer the simulations will be very complicated but in the simplify manner we can optimize the sedimentation tank in an easy way. Simulations are of two types 2D and 3D; in this study, we consider 3D flow, which is more complicated and better than 2D. There is less investigation available on lagrangian approach or DPM with double-coupled model but in our study, we use this approach and compare the results with or without a baffle wall.

CHAPTER 1

INTRODUCTION

1.1 Wastewater

Wastewater is contaminated water and generated by different sources. It can be surface runoff or sewer inflow or sewer infiltration. The contamination in wastewater depends on the source from which wastewater generated. Wastewater maybe domestic wastewater or industrial wastewater from industries. Wastewater contains different type of pollutants of different nature. Domestic wastewater may produce by flush toilets, kitchens, bathrooms, washing clothes etc. Flushing system produces more wastewater. Wastewater may be carry in a sewer line, which carry only sewage. It can be convey in a combined sewer, which involve storm water and industrial wastewater. Wastewater treats in a wastewater treatment plant and the treated wastewater or effluent is discharge to a river or a lake. If wastewater discharges to the environment without proper treatment, it will cause severe water pollution.

Pollutants should be remove from the wastewater for a good environment and better public health. After the use of water by humans, water becomes contaminated and its treatment is essential if untreated wastewater discharged then these pollutants would adversely affect our environment. Contaminated water may be toxic also adversely affect the aquatic life for example fish kills, foul odours. Waterborne diseases can eliminated by suitable wastewater treatment. Some pollutants are toxic in nature and they adversely affect the aquatic life.

1.2 Wastewater generation and treatment

It has estimated that approximately 48,254 million litre per day (MLD) of wastewater is generate in urban areas. Compare to population increase, demand of freshwater for all uses will become difficult.

Central Pollution Control Board (CPCB) studies shows that there are 816 sewage treatment plants (STPs) in India having capacity of 23277MLD,more than half of total wastewater cannot be treat and that untreated sewage cause's water pollution and adversely affect the ecology. There are a large numbers of treatment plants, which are not working properly. The development process in India is increasing and the rural population, which is lacking of basic infrastructural facility, will have to be give equality in terms of water supply and sanitation.

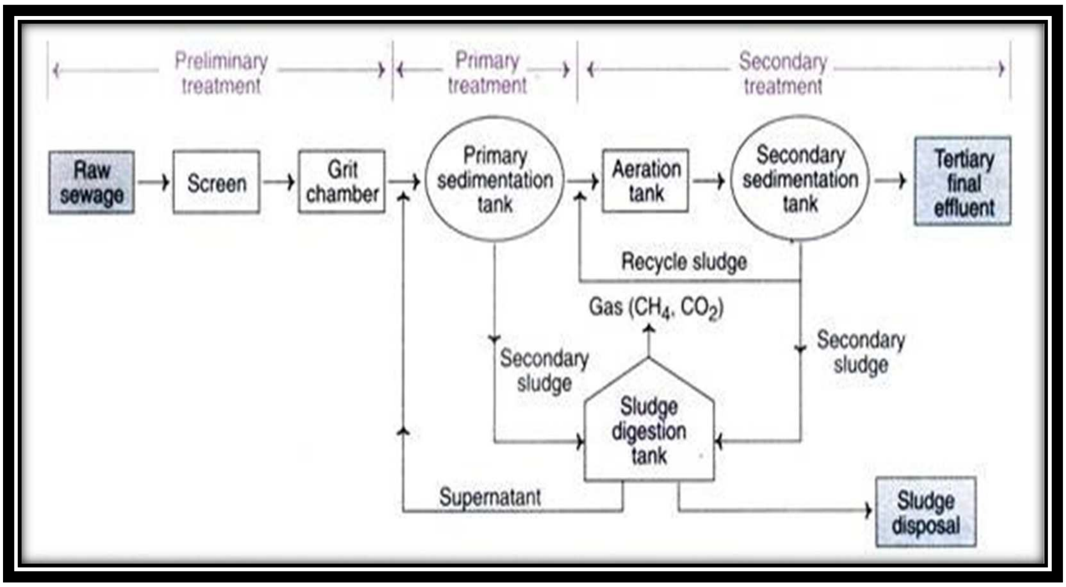


Fig1.1 flow chart of waste water treatment plant

(a) Preliminary treatment

The purpose of preliminary treatment is to remove heavy and bulky materials, these can easily remove by some physical actions like screening and grit chamber.

(b) Primary treatment

In this stage particles removes by the application of sedimentation.

(c) Secondary treatment

In the stage we remove, dissolved solids by the action of biological actions in the presence of oxygen .we can remove biodegradable and colloidal particles also .Secondary treatment has done after the primary treatment. In this stage by the application of biological actions, we stabilize the biodegradable matter. In this process, bacteria use the dissolved solid as food and convert it into biomass.

Preliminary treatment includes the following steps

1.Screening
2.Comminution and Maceration
3.Grit chamber
4.Detritus tank
5.Skimming tank
6.Pre-Aeration

1. **Screening**
The major function of screen in WWTP is to retain the coarser particles.Generally, a screen made of uniform openings. Mainly it used to sustain the coarser particles(pieces of wood, plastics, rags, papers, leaves, roots etc.) from the wastewater.

2. **Macerators:**
These are uses for grinding.

3. **Grit chamber**
It is a physical process or Unit operation similar as screening. In grit chamber, we remove inorganic solids or grits. Main objective of Grit chamber is to remove grit, containing of sand, gravel, or silty particle, or heavy solid materials of higher densities than the biodegradable particles in wastewater.
Grit chamber is similar to sedimentation tanks, designed mainly to remove heavier inorganic particles from the wastewater. Grit chambers are of two types with or without aeration. The organic solids kept in suspension by the application of aeration system.

1.3 Sedimentation tank

Sedimentation tank is very important unit in water and wastewater treatment plants. It works on the principle of gravity. In sedimentation tank if the velocity of flow is low then particles having a tendency to move towards bottom due to gravity may settle down. Therefore, a sludge layer forms in the bottom of sedimentation tank. The efficiency of sedimentation tank depends on the following points.

- Dimensions of the tank
- Inlet velocity
- Flow rate
- Particle diameter
- Particle density etc.

Types of sedimentation tank
Sedimentation tank are of two types

1. Rectangular sedimentation tank

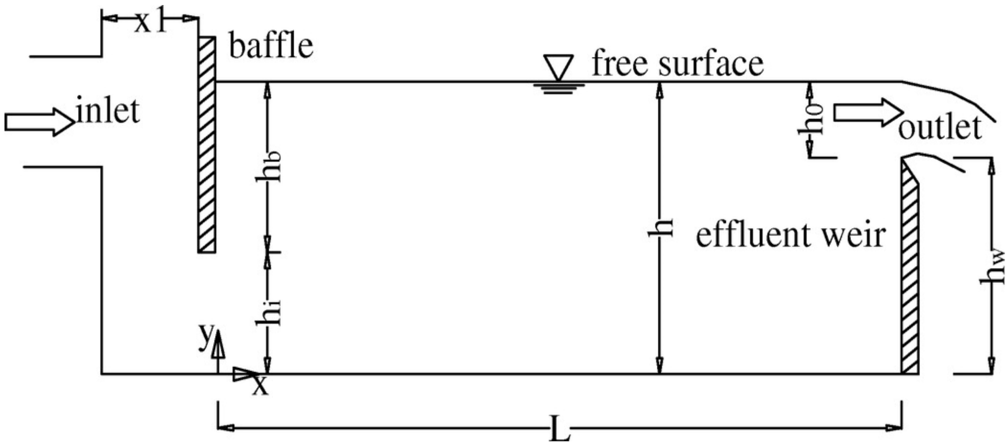


Fig 1.2 Rectangular sedimentation tank with horizontal flow

2. Circular sedimentation tank

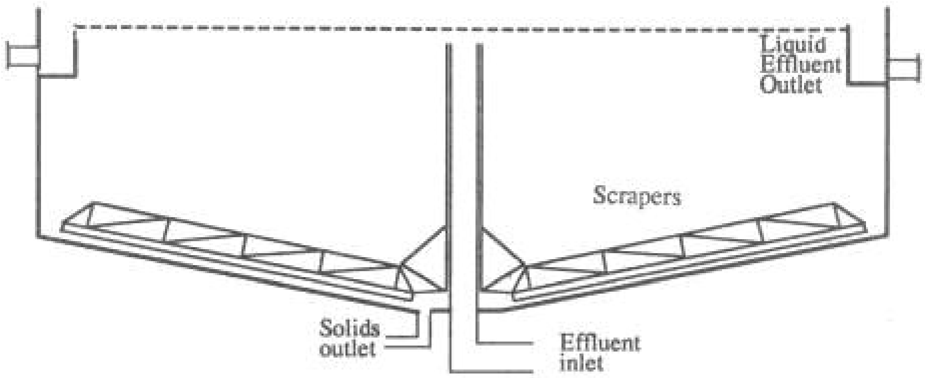


Fig 1.3 Circular sedimentation tank

1.4 Types of settling

There are generally four kind of settling which occurs in a sedimentation tank

1. Discrete
2. Hindered
3. Flocculent
4. Compression

1. Discrete settling

In this type of settling particles, maintain their identity. It occurs in grit chamber. In Discrete settling, particles settle independently and individually. Particle does not affected by the settling of nearby particles. Generally, in CFD simulation, we assume discrete settling and this occurs primarily in wastewater treatment plants.

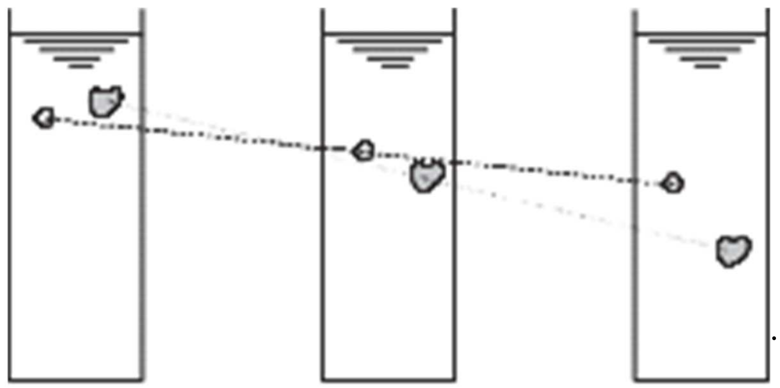


Fig 1.4 discrete settling

2. Flocculent settling

In this type of settling, a particle is affects by the settling of nearby particles. As shown in figure floc forms and these flocs traps the other particle while settle down. Floc formation occurs so settling velocity is gradually increases. It occurs in primary and secondary sedimentation tank.

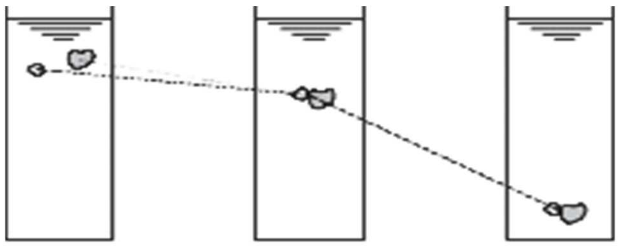


Fig. 1.5

3. Hindered settling

If concentration of solid is so high then a sludge blanket may form, which settle down as a unit mass. There is no relative movement between particles. Therefore, particles acts separately from the fluid. Sludge blanket moves downward and traps the particle due to this trapping. The sludge blanket become heavy and settling velocity will increase. It occurs in secondary sedimentation tanks.

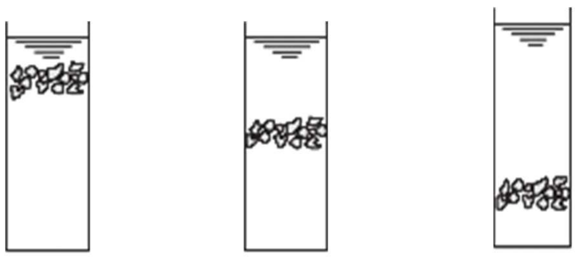
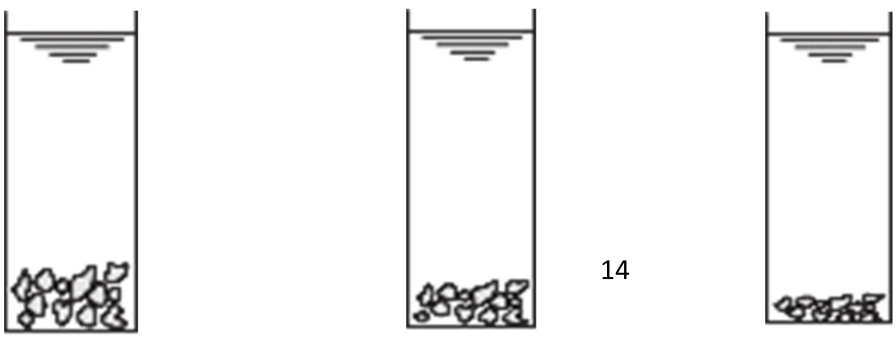


Fig 1.6

4. Compression settling

Compression settling occurs due to the compression of the particles. The upper layers due to their weight continuously compress lower layers.



1.5 Settling velocity in discrete settling

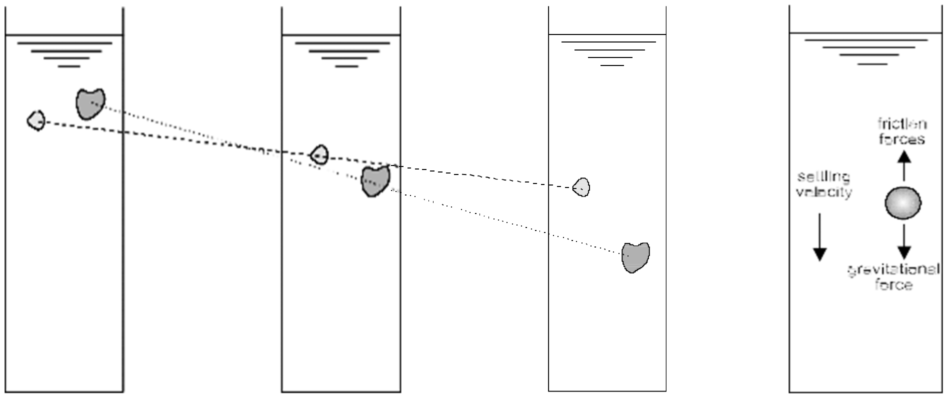


Fig 1.8

The sedimentation of discrete particles has based on the Stokes law. According to this law, the final velocity or the terminal velocity of a particle is constant in a sedimentation tank. It is achieve when the frictional force counter the force due to gravity. This terminal velocity is reach very quickly in liquid. The terminal velocity given by

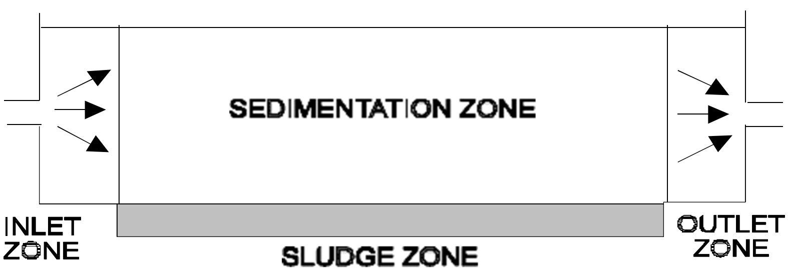
$$V = \frac{d^2(\rho_\omega - \rho_o)r\omega^2}{18\eta}$$

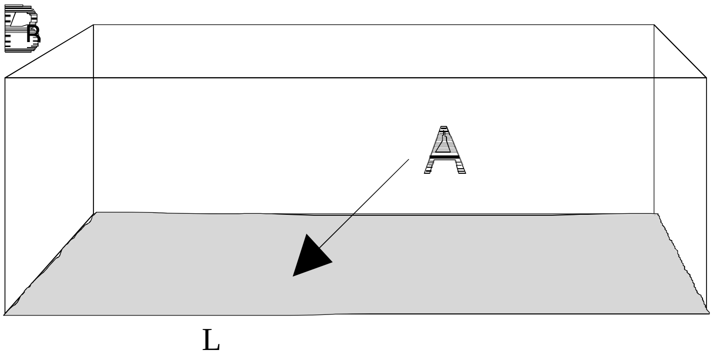
Where
d = diameter of the particle (m)

Density of wastewater does not change significantly due to temperature change. Therefore, we neglect this and standard value of 1000 kg/m3 has adopted. But the variation of viscosity depends on the temperature.

The diameter is proportional to the square root of velocity of particle so by increasing the size of particle terminal velocity increases significantly .so with a higher size of particles removal is so faster.

Different zones in a sedimentation tank





Dimensions in the sedimentation zone

B- Breadth of sedimentation tank

L- Length of sedimentation

H-height of sedimentation

A- Plan area of sedimentation tank

Above figure shows the typical zones of an ideal tank. In sedimentation zone where settling occurs theoretical consideration are applied.

There are some assumptions, which should be consider in a theoretical analysis.

- The distribution of particles should be uniform in inlet zone
- If a particle reach the sludge zone then it is consider as removed.
- If a particle reach the outlet zone then it is consider as not removed
- Detention time in sedimentation tank gives by

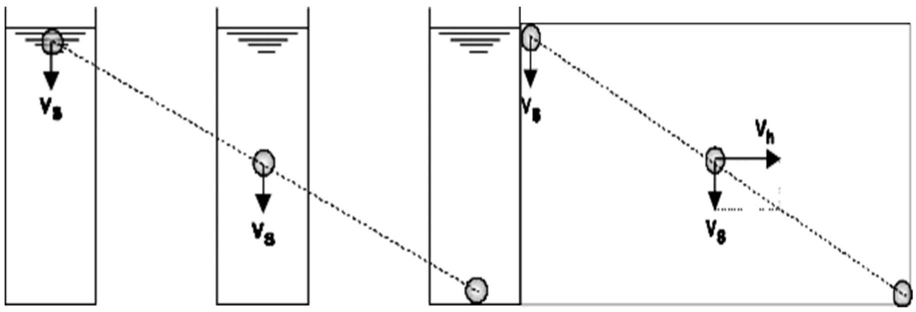
$$t = H/V$$

As we know detention time = volume of tank/ discharge

$$T = \text{Volume/discharge}$$

$$= HA/Q$$

SETTLING COLUMN **HORIZONTAL; FLOW TANK**



Analysis of discrete settling

V=Q/A

If we know the discharge and the settling velocity V of particle that can be obtained from strokes law then we can calculate the required surface area A for removal of particles having settling velocity V or greater than it .

A=Q/V

From the above equation, it can be note that:
• V can be calculate from experiments and also from literature because it's a design parameter
• The discrete particle removal can be influence by only the surface area.
.

1.6 Objectives

- TO develop a three dimensional CFD double phase model that describes flow pattern in sedimentation tanks.
- Evaluate hydraulic performance of the sedimentation tank, particle tracking, velocity and pressure contours, streamlines and efficiency of tank in respect to particle tracking.
- To systematically study hydraulic performance of different sedimentation tanks, in addition to study the hydraulic efficiency of different horizontal sedimentation tanks.
- TO develop a three-dimensional multiphase flow model that describes density current, sludge accumulation and particle separation in sedimentation tanks.
- To compare the simulations of sedimentation tank with or without a baffle wall.

CHAPTER 2

LITERATURE REVIEW

2.1 COMPUTATIONAL FLUID DYNAMICS

One of the earliest contributions of CFD could track back to 1910, Richardson (1910) introduced approximate arithmetical solution by finite differences of physical problems, also application of this solution to stresses in a masonry dam in his paper, his computational work use hand calculations with human computers. Although the calculation extremely slow (only 2000 operations per week), it still provide ideal for numerical research, he thereby is regarded as pioneer of CFD.

After World War 2, with development of semi-conductor technology, computer architecture and electronic engineering, computer power was increased dramatically, computer was employed to solve fluid problems governed by Naiver-Stokes equations, Francis H. Harlow and his group (Khalil 2012) developed a series of numerical methods for unsteady, two dimensional problems. Some of milestone of CFD was developed during this period (from early 1950s to late 1960s), such as Marker and Cell methods developed by Harlow and Welch (1965) in 1965, Fluid-in-cell method proposed by Gentry, Martin and Daly (1966) in 1966 and velocity stream function method provided by Fromm (1963) in 1963.

Three-dimensional models began to appear in late 1960s, with launch of space program, as well as stimulated by cold war, more fluid dynamics solution is required, and CFD appeared in development and manufacture of aerospace and military equipment, such as submarines, helicopters, aircraft and missile. In 1967, Hess and Smith (1967) published the first paper about three-dimensional problem; they proposed the Panel Method, which is performance discretization according to geometry of panels based on requirement of aircraft manufacture. Lifting panel code also described by engineers from major aircraft company, such as Boeing, Douglas and NASA.

In 1970s, report about Finite difference methods for Navier-Stokes equations and Finite element methods for stress analysis appeared, however, finite difference methods based on structured mesh, suitable for problems with rectangular and cubic sharp only, finite element methods require more computer power. To compensate the limitation of finite difference method and finite element method. The CFD group at Imperial College proposed finite volume methods in 1970s. They launched a program which aim at solve simple shear flows and jet flows, from 1970s to 1980s, some algorithm and models which employed by nowadays commercial CFD software was developed, such as the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm, which supply a straight forward solution of the Navier Stokes Momentum Equations. Additionally, the most popular turbulence model, Standard k-ε turbulence model, also

proposed during this time, Launder and Spalding develop the Standard k-ε turbulence model in 1974, this model could describe turbulent flow with high Reynolds numbers. These achievement made fluid dynamic problems suitable for programming and solved by computer, thus create features that differ
CFD with traditional fluid dynamics problems. More two equations turbulent models was developed in late 1970s, application of turbulence flow transfer from simple shear flow to flow with strong swirling, circulation and turbulent chemical reaction in complex geometries. K-ε turbulence model validated in a series of applications with and without swirl. Additionally, besides solve fluid dynamics problems, k-ε turbulence model also testify in turbulent reacting furnace configuration with and without swirl.

CFD was encouraged to apply in a variety of industrial applications in 1980s, in 1985, CFD already commonly used in aero industries, at that time, although assisted by more advanced models, CFD still use simple structured grid and difficult to handle unstructured boundary and wall conditions. Other drawbacks of CFD at that time include slow convergence and numerical diffusion caused by limited computer power, as well as difficult to handle simulation with three-dimensional complex geometries (Anderson 1995).

In early 1980s, commercial CFD codes became available in market, more and more CFD users began to choose commercial CFD software rather than develop their own CFD codes. Commercial CFD software based on a series of very complex non-linear mathematical expressions. These expressions defined the basic governing equations of fluid dynamics, such as continuity equation, Navier-Stokes equation and energy equation, as well as additional models available in commercial CFD software, such as multi-phase models, turbulence models, species transport model etc, assisted by algorithms embedded in commercial CFD software, these equations are solved. Commercial CFD software enable users define geometry of flow field need to simulate, identity physical and chemical condition of fluids, also specific initial and boundary condition, with above condition as input, a converged solution as output with information of the flow field is provide. Output of commercial CFD software can be viewed graphically, such as contour plot, velocity vector, path-line, or numerically, such as output data and x-y plot.

In 1990s, CFD spread from traditional aero industries to non-aero industries, more and more applications of CFD appeared in a variety of industries. CFD is now regards as an essential part of the Computer aided engineering (CAE), have similar function as Finite element analysis (FEA), CAE technology together with Computer aided design (CAD) and Computer aided manufacture (CAF), could completely change traditional industries.

CFD supply a “virtual wind tunnel” on engineer’s desktop. Engineers could simulate fluid flow, equipment performance and reactor hydraulics before really construction work start, all they need is only a computer. CFD can supply fully information of the completely computational zone, with all conditions well controlled and almost without any constrains, with appropriate conceptualization of model geometry, mesh generation and select proper solution method, as well as proper verification and calibration, CFD could supply acceptable accuracy. Nowadays, CFD has become an indispensable part of industries, aerodynamics and hydrodynamics simulation for airfare, car, train, missile, ship and submarine has been popular, in last two decades, CFD also contribution in non-traditional field, such as in-door environment simulation and pollutants transport. In water and wastewater industry, CFD also employed by some institution while still infancy and need further research.

2.2 Application of CFD in waste water

To avoid potential failures and obtain fully understanding about hydraulic mechanism in particle separation processes such as flocculation or sedimentation, CFD could supply new solutions. Originated from aerospace engineering in late 1960s, CFD has employed by engineers from variable fields, included chemical engineering, hydraulic engineering and civil engineering (Anderson 1995).

With the development of computer science, nowadays personal computer is getting greater and greater computational power, CFD is not only limited in academic environment or specialized consultant company, it’s more and more popular in water and wastewater treatment research, technically, CFD is applicable for every water and wastewater treatment processes. This section aim at give a briefly introduction of CFD in varies water and wastewater treatment research, includes sedimentation, flocculation, flotation, biological treatment, disinfection and sludge treatment.

2.3 Application of CFD in sedimentation tank

As one pioneer in numerical simulation of sedimentation tank, Larsen (1977) applied CFD simulation to several sedimentation tanks. Although with simplification and conceptualization, he still shown several major hydraulic phenomena of sedimentation tank, such as “density waterfall. Due to heavier fluid sink into bottom of sedimentation tank soon after entering, bottom current and surface return current. Nowadays, thanks to effort of computer engineers, mathematicians and fluid dynamics scientists, several more advanced models has been developed and available in commercial CFD software, based on these models and today’s high performance computer, we could run advanced simulation which far beyond than 1970’s.

Goula et al (2008) researched influence of baffle on sedimentation tank in potable water treatment by using CFD; a circulation zone is detect in the original tank without baffle. After equipped baffle, the recirculation zone around inlet in original tank decreased, the baffle enhanced setting of particles. Due to effect of baffle, particles around inlet move downwards and reach the bottom of the tank.

Density current, or turbidity current, means when two fluids with different density due to temperature, concentration or salinity confront each other, fluid with higher density sinks and flow along the bottom of fluid with smaller density, Goula et al (2008) studied the influence of temperature variation on density current in sedimentation tank. He found temperature difference between incoming fluid and fluid in tank could leads to density current. Under density current phenomenon, a rising buoyant plume appears in the tank, and changes the direction of the main circular current.

Shahrokhi et al (2012) studied effect of baffles on sedimentation tanks, result show that baffle at optimum location could reduce the circulation zone, kinetic energy and maximum velocity magnitude; uniform velocity vector inside the settling zone from CFD simulation result could indicate better sedimentation effect.

In present time, we are facing water shortage although more than seventy percentage area of earth contains water. The reason behind this shortage is that most of the available water is salty or marine water so it need a proper treatment for make it usable that is so expensive .on the other hand 33% of fresh water or usable water is in the form of ice so that also cannot be used approximately 98% water is salty. So approximately 2.5 billion people that is 33% of total population is not able to get clean and fresh water and this figure will abruptly increase with time. The solution of this problem is to treat the water properly and use it efficiently .water treatment is not an easy process it consists of several stages like mechanical processes and then biological processes also sedimentation and filtration then disinfection so it is a complicated process and ca done in many stages. The simplest process in wastewater treatment is sedimentation .sedimentation remove the particles that is why we use sedimentation after flocculation and coagulation also so we can remove that particles which formed during flocculation and coagulation. By the application of sedimentation, we can remove up to 40% BOD, suspended solids significantly, in this process flocculated particles settled down due to gravity, and water can be carry out by outlet. (Raj Vaidya and Markandey1998). By the use of empirical formulas instead of practical significance in design of sedimentation tank, we are not using the proper hydrodynamic system. Some factors that affects the efficiency and working of sedimentation tank like surface loading rate, size of particles temperature, inlet and outlet conditions, dimensions of sedimentation tank etc. For effective design and operation, we have to consider all the parameters that affects the efficiency of sedimentation tank in designing. In traditional sedimentation tank, we can see only external features so it is call as a black box, we cannot see the internal features. (Metcalf, 2002). From the last decades CFD simulation starts to overcome from the problem of black box and also overcome from expensive ,time consuming and difficult experiments .In CFD simulation the chances of failure is negligible because we do all simulation before designing the tank. (Anderson, 1995).

Our purpose is to know the benefits and application of CFD in wastewater treatment so we can improve efficiency and optimize our plants specifically in the field of sedimentation tank. (Shilton, Glynn et al. 2010). CFD is a powerful tool and its application is widely used in different kinds of industries. .

In water treatment plants sedimentation is very important and common unit. The use of sedimentation is to remove the particles by the action of gravity; a major part of suspended solids is remove.

The efficiency of sedimentation tank depends on the particle settling so it is an important parameter in designing of sedimentation tank. So our purpose in this study to optimize the settling of particles.

Sedimentation is a very common unit in wastewater treatment so many research studies and papers has published on sedimentation tank. Larsen first used the applications of CFD in secondary clarifiers. Shamber and Larock solved the Navier-Strokes equation, the k–e turbulence model equations by using finite volume method and done modelling of secondary clarifier by using transport equation for settling of particles.. Zhou et al.

Stimulate the clarifier model and considered the neutral density and hot water by interrelating energy equation with navier stroke equation.

Imam et al. considered a constant settling velocity of particles in a sedimentation tank that is simply the average of settling velocities of different sizes of particles. Stamou et al. Also have done work on primary sedimentation tank but he used a 2D model, he simply analysed and stimulate some equations like momentum equation and also used equation of concentrations of particles. Adams and Rodi also done work on the same model but their field of work is different, they investigate the effect of inlet arrangements although it is an important parameter and instead of a normal flow, they have done their work on flow through curves. Lyn et al. Investigate the flocculation process and taken six different sizes and velocities. This paper gave us some important results and very important in wastewater treatment.

Goula et al. In addition, done almost same research but his work area is potable water instead of wastewater, his model deals with sedimentation tank for potable water and he investigated the effect of baffles and dependence on temperature at inlet. The drawback of his study is he did not consider the interaction between the solid phase and liquid phase, as we know that interaction affects the whole study. Wang et al. done his work on rectangular sedimentation tank. He simulated the concentration of particles and flow pattern.

Sedimentation is very common and important process and it is use in industries. Kahane et al. have done work in industries to reduce the operating costs. White et al. Also done his work on the flow behaviour of fluid. For improving the operations in thickener, flocculation studies have done by the Farrow et al.

Righetti and Romano , Sbrizzai et al. , Hetsroni , Li et al. , Reeks , these researchers have investigated the interaction between solid and liquid phase but their investigation in not useful for us because their investigation is in different filed ,they were not analyse the sedimentation tank. Although on settling tank there are many researchers done their work, on settling and removal of particles. For potable water, the application of CFD models are not used much and in this field, the work on CFD is limited. The interaction between the solid and liquid phase and changes in the velocity due to that interaction and momentum change has firstly investigated by Roza Tarpagkou in his research paper.

2.4 Some important concepts in CFD software

Simple algorithm

After discretization, a series of algebraic equations need to be solve simultaneously in every control volumes, SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm is the most popular CFD solution procedure, it developed by Patankar and

Spalding (1972) in 1972. For a three-dimensional problem, four equations need to be solve, includes Navier-Stokes equation in three directions and one continuity equation. There are four unknown values in the four governing equations: three velocity components in the Navier-Stokes equation at three directions and the pressure. Pressure term is the most time consuming and complex step in CFD solution procedure, because the pressure does not have its own explicit equation, special techniques have devised to extract the pressure term. Currently SIMPLE algorithm is the best-known technique for solving the pressure term. In order to implement the SIMPLE algorithm, firstly, a guessed pressure field is use to solve the Navier-stokes equations, based on the guessed pressure field, a new velocity can be computed. however, the velocity calculated by the guessed field will not satisfy the continuity equation, so the velocity correction value can be confirmed, based on the velocity correction value, pressure correction also can be determined, the pressure correction then added to the original guessed pressure field, at last, the pressure field updated and remaining unknowns is solved, one iteration is completed. Then in the next iteration above algorithm has repeated.

Residuals

Residuals are the differences in the value between two iterations. As we can see from the SIMPLE algorithm, initially, the solution of Navier-stokes equations based on a guessed pressure field, in each steps, the solution of governing equations based on inexact solution from the previous iteration, through repeated iterations, the solution of governing equations refined. Residuals depend from the models and initialization, because residuals related to mathematical convergence, so that during calculating process, residuals should be monitor in order to evaluate the convergence behaviour.

Convergence criteria

During iterative solution of governing equations, when the residuals decrease to pre-set level, the solution is regard as converged, this pre-set condition for the residuals known as the convergence criteria. The default convergence criteria in Fluent is 10-4, we use this default value as convergence criteria in this study.

Under relaxation

With the governing equations solved iteratively, in each step, the initial value used in current step based on the information from previous iteration, during this process, small difference added to the old values of variable in the previous iteration, generate a new value. When several coupled equations are, solve, only a fraction of computed difference is use rather than use the full computed difference, this process known as under relaxation. the under relaxation factors decided how much fraction of the computed difference is used, the under relaxation factors varies from 0.1 to 1.0, through adjust under relaxation factors, the convergence behaviour of governing equations can be controlled, in general, lower under relaxation factor give stable but slower convergence process.

CHAPTER 3

METHODOLOGY

3.1 Mathematical model

By using Euler–Euler or an Euler–Lagrange approach we can analyse the hydrodynamic behaviour of a sedimentation tank, from the above approaches we can easily analyse the multiphase behaviour in a sedimentation tank. Eulerian applications are widely used in the analysis of sedimentation tank but the major drawback of this approach is it does not consider the individual particle motion as in lagrangian approach. Due to wide range of applications, lagrangian approach is majorly use in multiphase problems. In Lagrangian approach every individual particle is consider so it gives a realistic and a well-defined model by the use of this approach, we can simulate the flow behaviour in multiphase and a realistic way. In multiphase approach particle or solid phase is treated in the lagrangian way and fluid is always treated as continuum phase. In this phase(multiphase) coupling effect may be considered ,due to coupling effect the model will become more difficult but the results are more realistic.in these types of models we assume that the volume of solid phase is much less than the volume of primary phase or liquid phase although high loading rate is allowed. According to De Clercq et al., we cannot apply the lagrangian approach if the volume of secondary phase is high that means that we can apply that approach if volume of secondary phase does not exceeds 10–12%. As mentioned earlier in this approach the particle motion and their paths are individually consider at different intervals of time.

RNG k-ε model

Equation of kinetic energy

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + 2\mu_t E_{ij} E_{ij} - \rho\epsilon$$

Equation of dissipation

$$\frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial(\rho \epsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\epsilon}\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2\epsilon}\rho\frac{\epsilon^2}{k}$$

3.2 Governing equations

By the application of integration on the equation of force, balance of discrete phase trajectory our model works that is lagrangian approach. In this equation, we balance all the forces acting on the particle like inertia force.

$m_p\ (du_p\ /dt) = m_p F_D (u-u_p) + m_f (Du/Dt) + 1/2\ m_f (Du/Dt- du_p /dt) + (m_p - m_f)g + 1/2\ (\pi \rho_p r^2)$

$C_L LV2$

Where:

$F_D\ = 18\mu\ C_D Re/ (24 \rho_p\ d_p^{2)}$

d_p is diameter of particle

Fluid phase: By the application of navier stroke equation, fluid phase is consider as continuum. The following equation accounts the non-conservation of mass and non-conservation of energy and incompressibility also.

$\partial u_i / \partial x_i\ = 0$

$U_j\ \partial U_i / \partial x_j\ = -1/ \rho\ \partial p/ \partial x_i\ + \partial / \partial x_i\ (v(\partial U_i / \partial x_j + \partial U_j / \partial x_i\ ) - u_i{}' u_j{}'$

Turbulence. The RNG k–e turbulence model is show in the following equation

$\partial / \partial X_i\ (\rho\ ku_i) = \partial / \partial X_j (a_k\ \mu_{eff}\ \partial k / \partial X_j) + G_k + G_b - \rho\ \varepsilon$

$\partial / \partial X_i\ (\rho\ \varepsilon u_i) = \partial / \partial X_j (a_\varepsilon \mu_{eff}\ \partial \varepsilon / \partial X_j) + C_{1\varepsilon} \varepsilon /K (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho\ \varepsilon^2 /K - R_\varepsilon$

Momentum exchange: As mentioned above in lagrangian approach we use multiphase or discrete phase with or without coupled effects but we are considering coupling effect also. momentum exchange in multiphase is between the primary and secondary phase that are liquid and solid phase .we will consider the momentum transfer from primary to secondary and vice versa . In single-phase simulations, we does not consider the momentum exchange between the two phases. Multiphase simulations gives better and realistic results than the single phase

This momentum change is compute as:

$F = \sum\ (18\mu CRe\ (u_p - u) / (\rho_p d_b^2 24) + f_{other})\ mp \Delta t$

3.3 Geometry and meshing

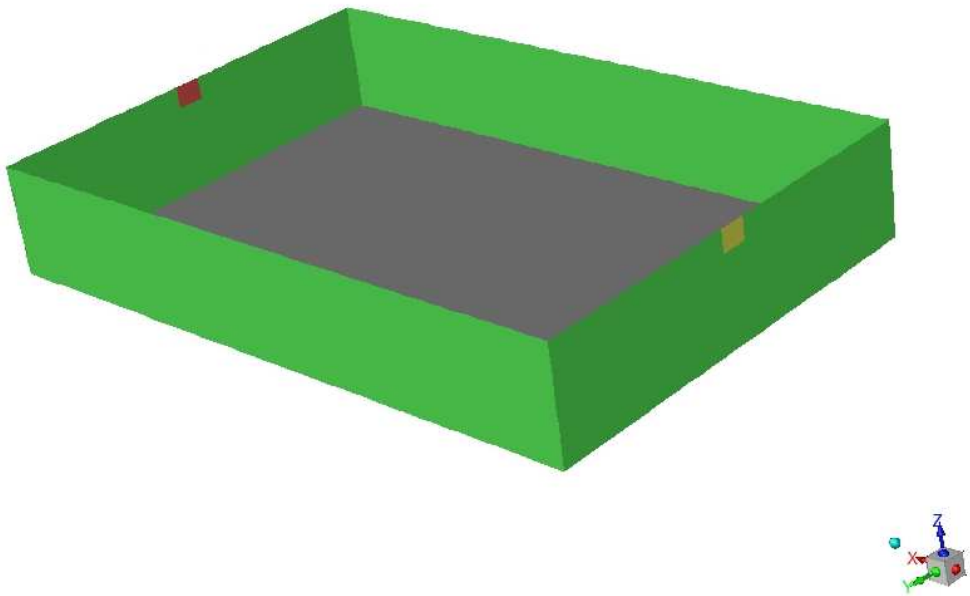


Fig 3.1 Geometry of sedimentation tank without baffle

Geometry details

depth	length	width	Inlet channel	Outlet channel
.8	6	4	.25*.2	.25*.2

In the above figure inlet has shown in yellow colour, outlet has shown in red colour, walls are in green colour, and bottom wall is in grey colour. The noticeable point is, it seems an open tank but in actual case, it is covered. The purpose of showing it uncovered is to show its inner features that can be see easily in the above figure.

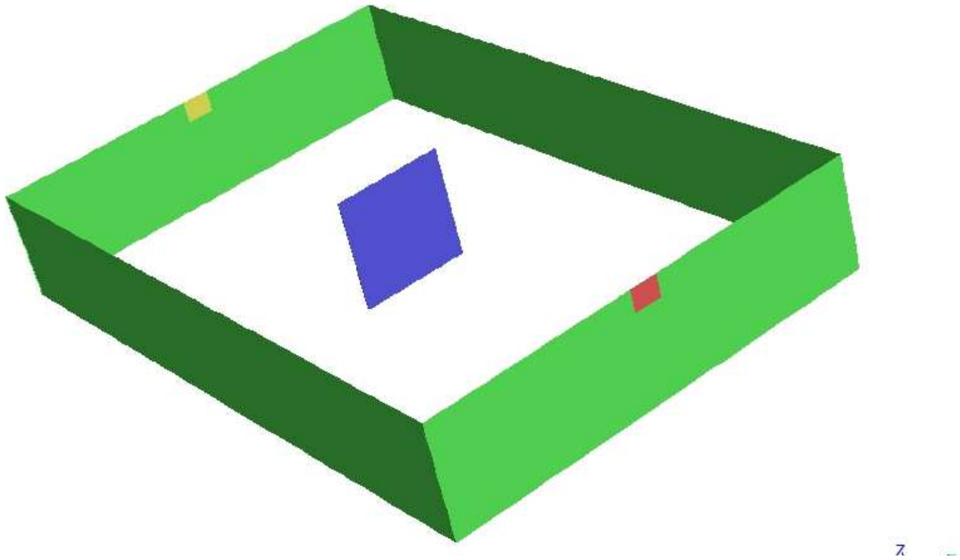


Fig 3.2 Geometry of sedimentation tank with baffle wall

All the dimensions are similar as shown in sedimentation tank without baffle instead of a baffle provided on mid plane across the length shows in figure by blue colour.

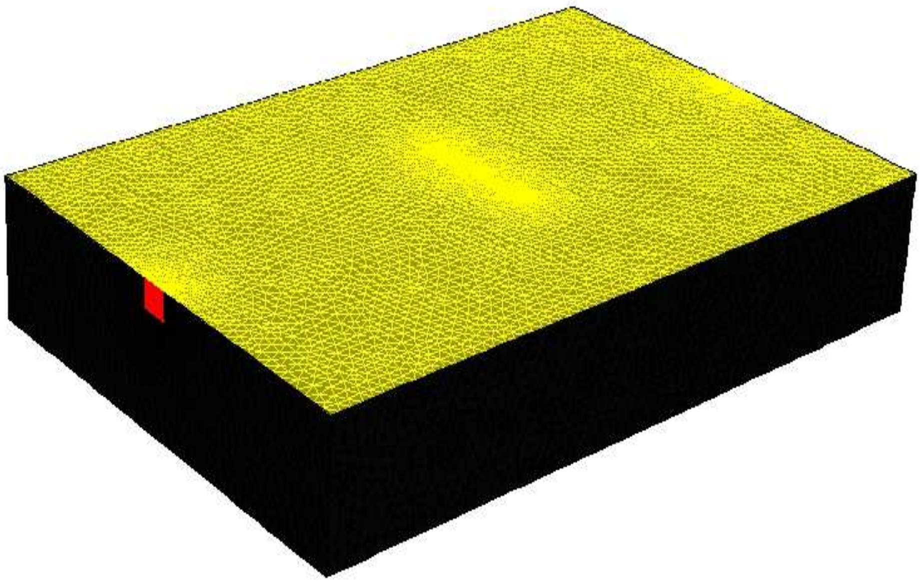


Fig 3.3 Meshing of sedimentation tank without baffle wall

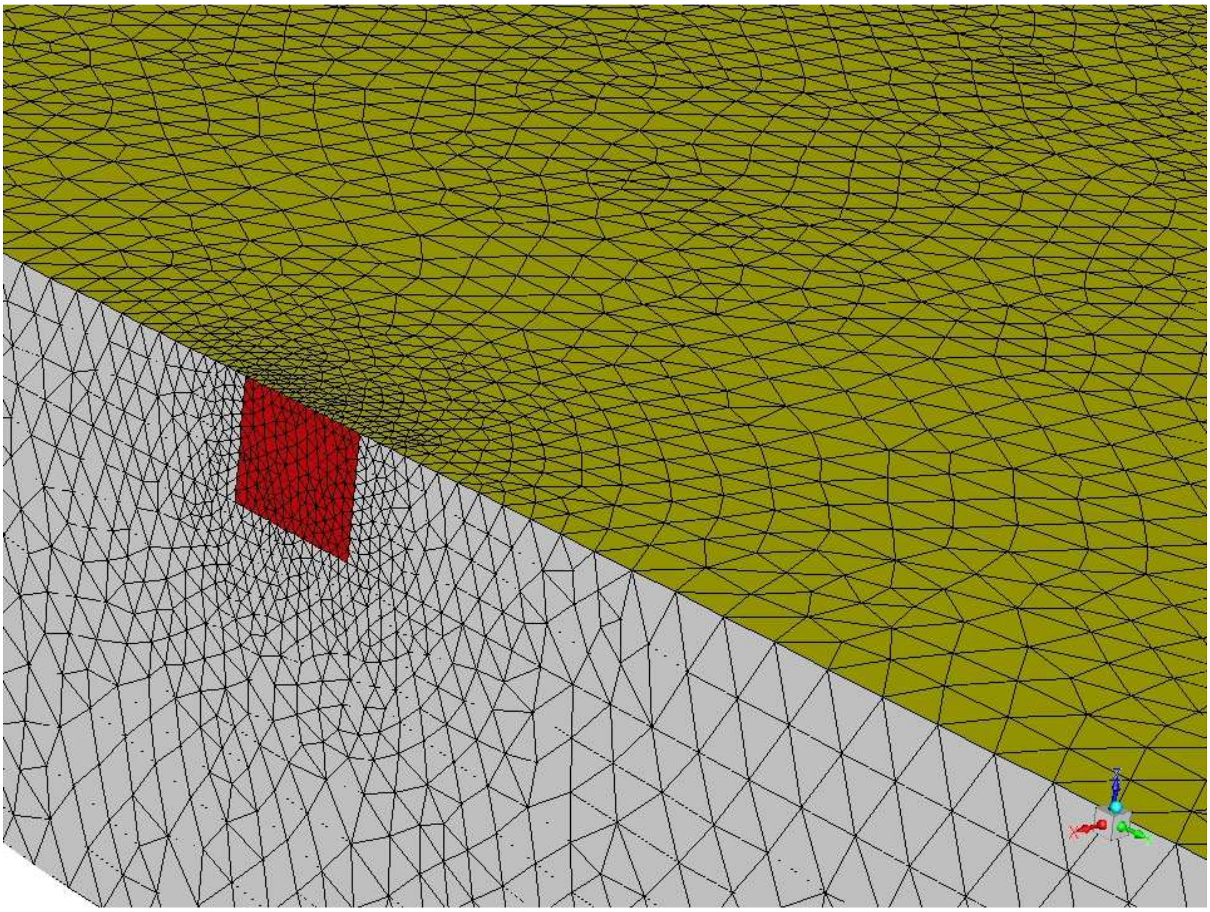


Fig. 3.4 Meshing at inlet

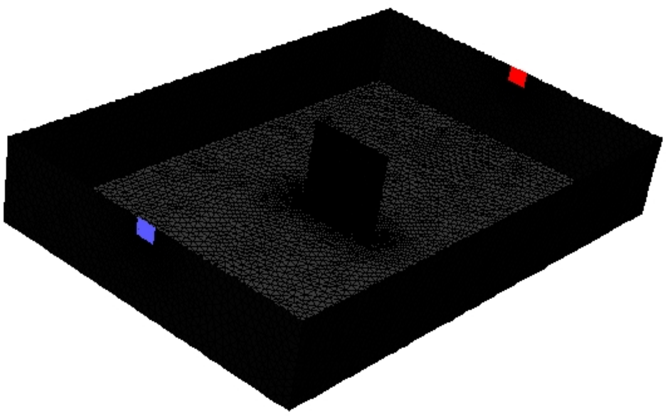


Fig 3.5 Meshing of sedimentation tank with a baffle wall

Meshing details	cells	nodes	faces
Geometry without baffle	554836	199592	1272520
Geometry with baffle	1921912	337036	2122784

3.4 Mesh independent study

The aim of this study is to show that the results of the CFD simulation will not change with our meshing if we have done the meshing perfectly. In our actual, meshing or selected meshing there are 554836 cells but in our finer mesh, there are 322546 cells only. So as shown in the below graph the results will not change with different meshing so our study is mesh independent. It is noticeable that the graph is plot between average velocity on mid plane Vs the distances along x- axis or the length of the sedimentation tank.

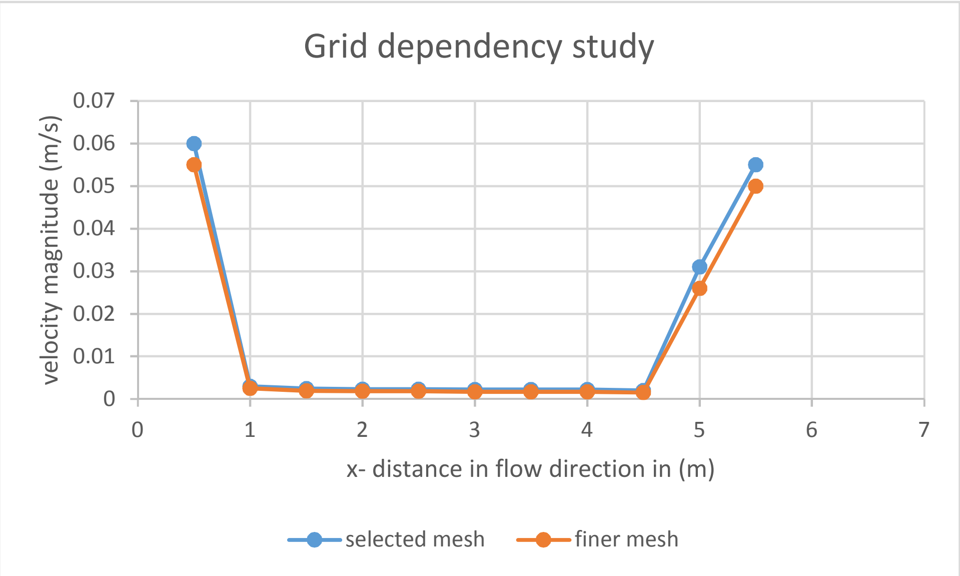


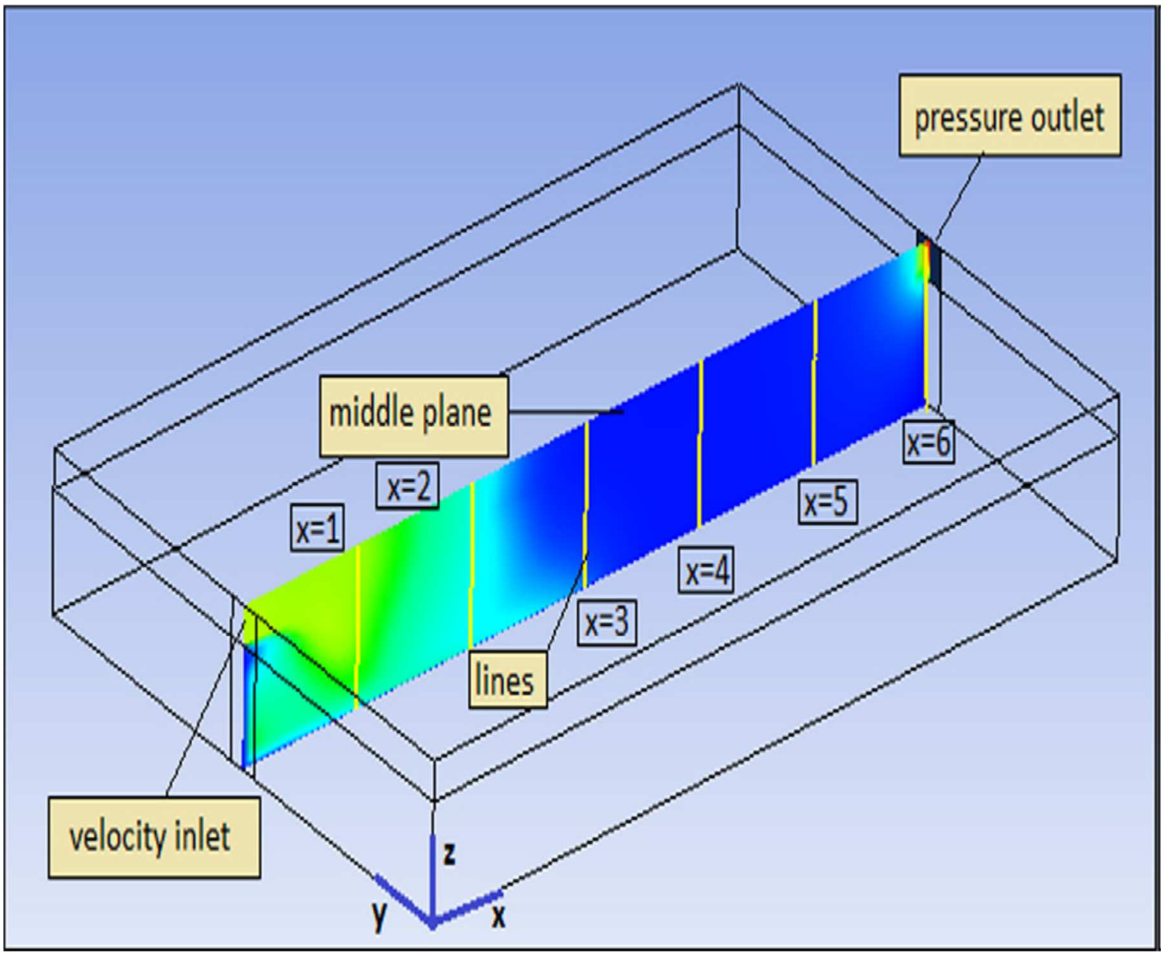
fig 3.6 Graph of grid dependency

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Model validation

For validate the model we make a mid-plan along the length of the sedimentation tank and by simulation we calculate the velocity magnitude at different locations like x= 1, 2, 3, 4, 5, 6 and then draw the results on the graph and compare that results with numerical of Kantoush et al (CCHE2D**).** We can see in the following graph that results are matching.



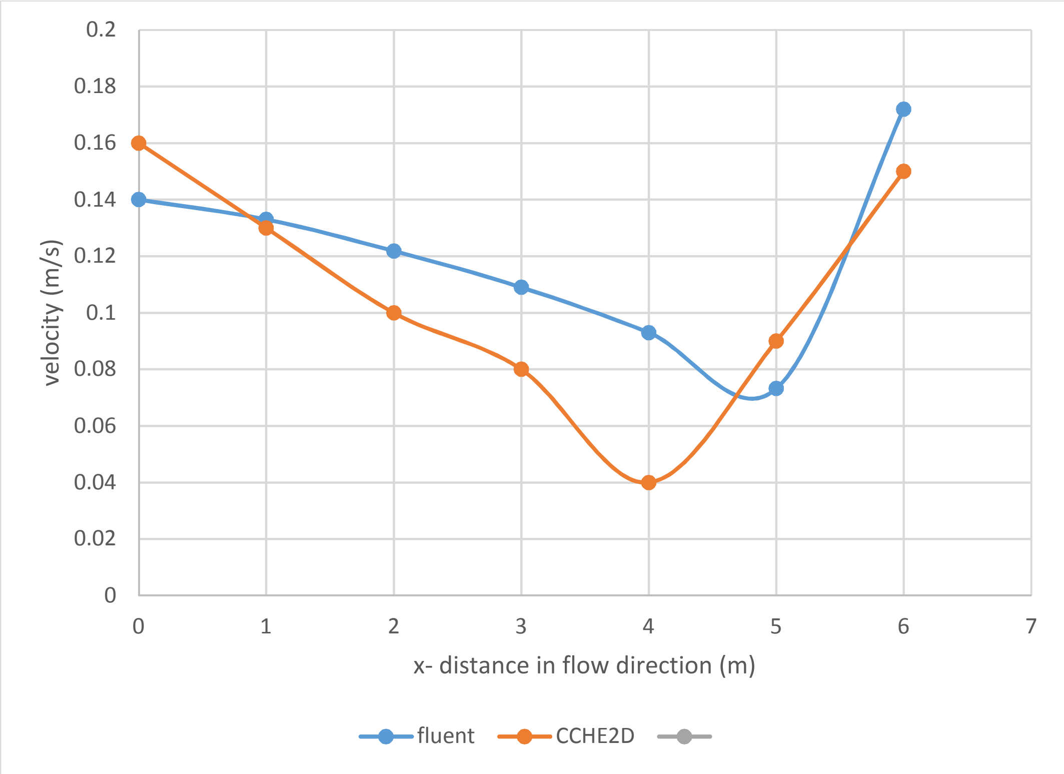


Fig 4.1 MODEL VALIDATION GRAPH

4.2 Comparison between velocity contours

This study can easily compare the simulations of velocity contours of sedimentation tanks with or without baffle wall. As shown in below figures the colourful scale in the left of the figure shows the velocity magnitude and by the help of this scale, we can easily see the variation of velocity in the two profiles. It is noticeable that we are considering the middle plane along the length so all the results that are show below are on the middle plane.

It is easily shown in the below figures that in the tank with baffle ,after the baffle the profile is changed and velocity is decreased that's why more sedimentation takes place in case of a sedimentation tank with baffles.

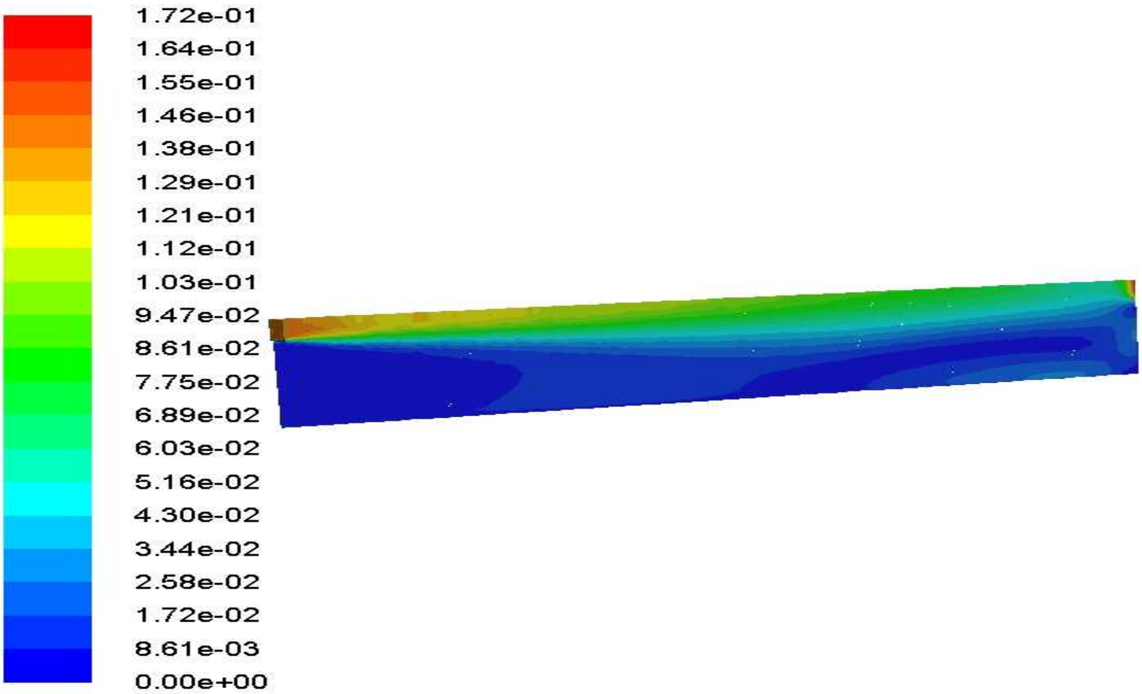


Fig 4.2 Velocity contour without a baffle wall

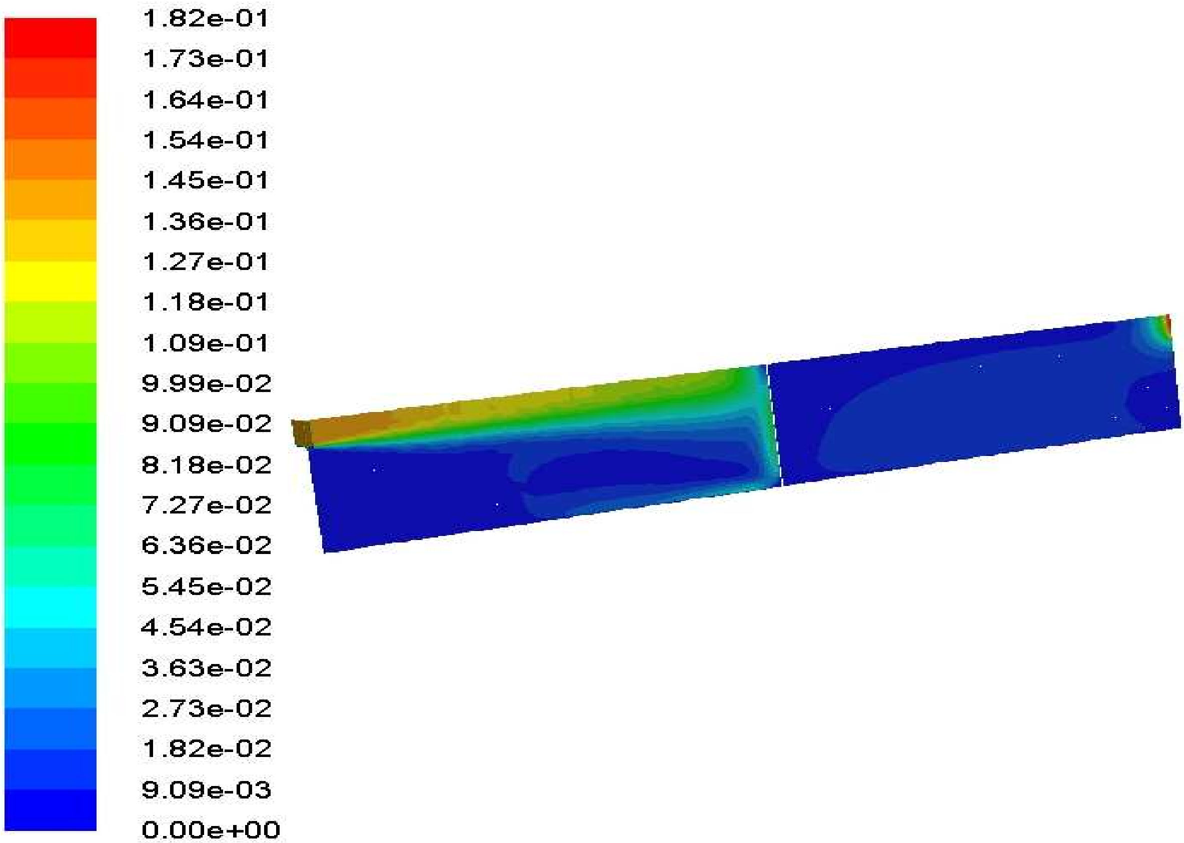


Fig 4.3 Velocity contour with a baffle wall

4.3 Comparison between pressure contours

Although there is a wide application of these pressure contours in designing of the walls of a sedimentation tank. However, we focus on the comparative study or the change in pressure contour by the application of a baffle. At the location of baffle, pressure is higher than the other locations. It is due to the turbulence created at the baffle .

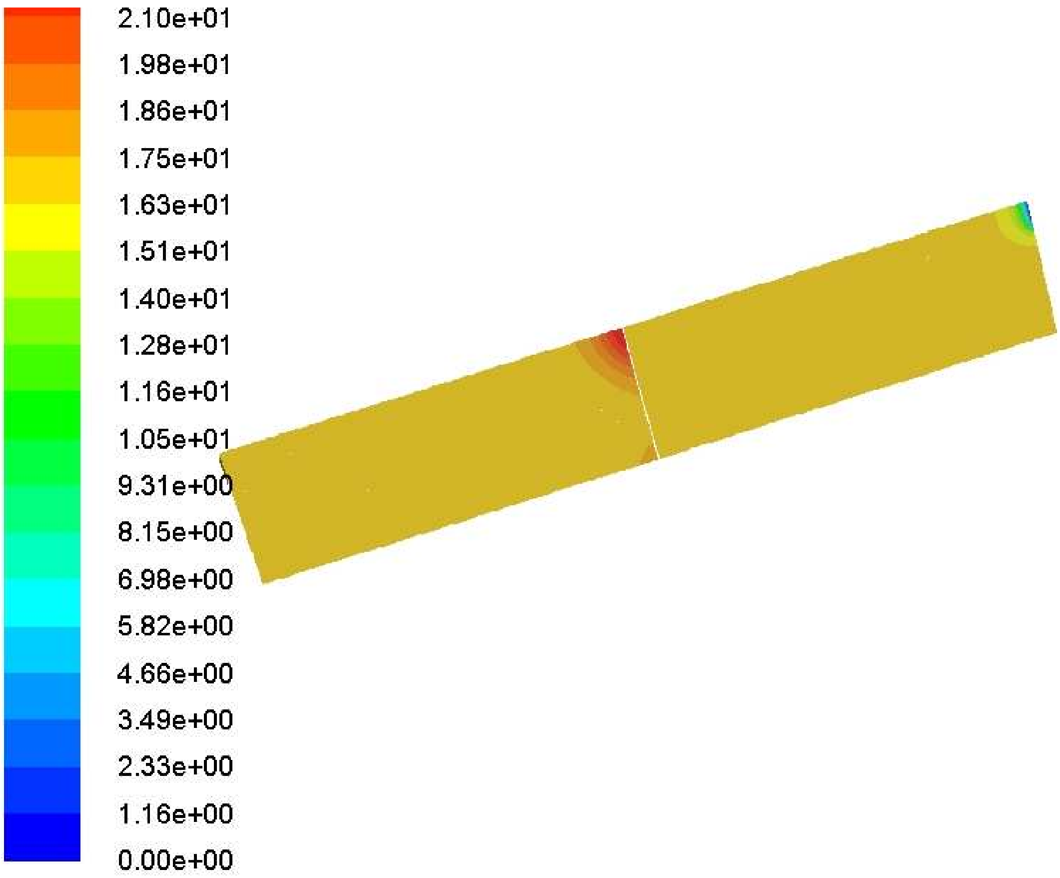


Fig 4.4 Pressure contour of sedimentation tank with baffle

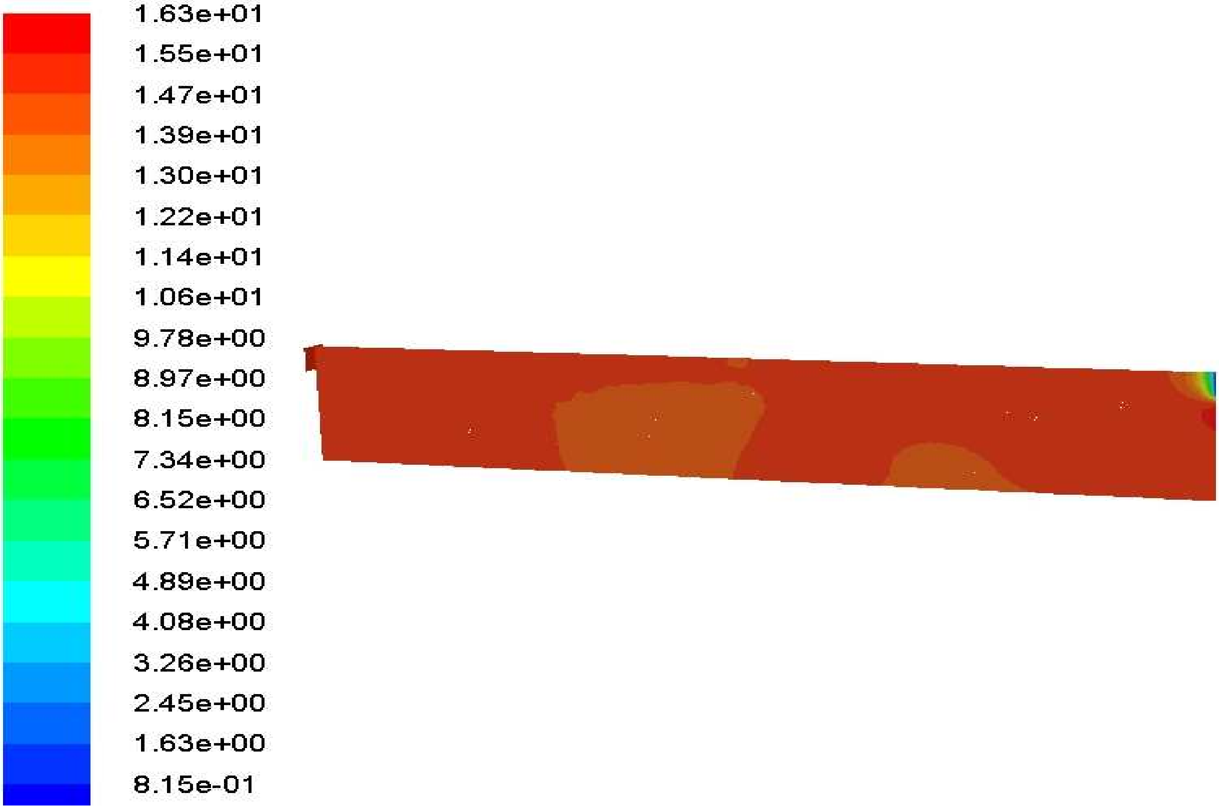


fig 4.5 Pressure contour of sedimentation tank without baffle

4.4 Path lines

Path lines are the trajectories of flow of fluid if we can consider individual elements in the fluid. As shown in the below figure it is clearly shown that in the case of baffle wall higher turbulence will generate due to loss of energy occurs and velocity diminishes by a considerable amount so less velocity leads to higher sedimentation and higher efficiency. Without baffle fluid moves in a defined manner but in case of baffle, the randomness will increase and path lines become congested that means a particle affect the motion of others.

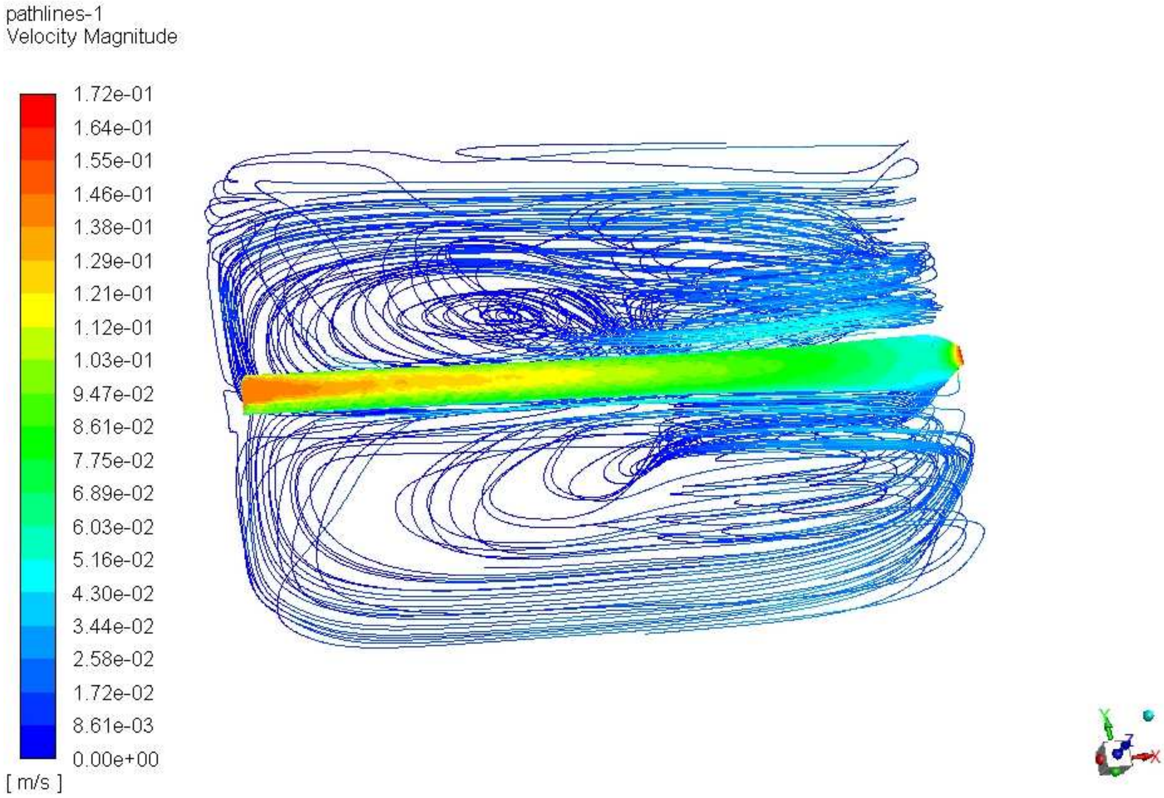


Fig 4.6 Path lines without baffle wall

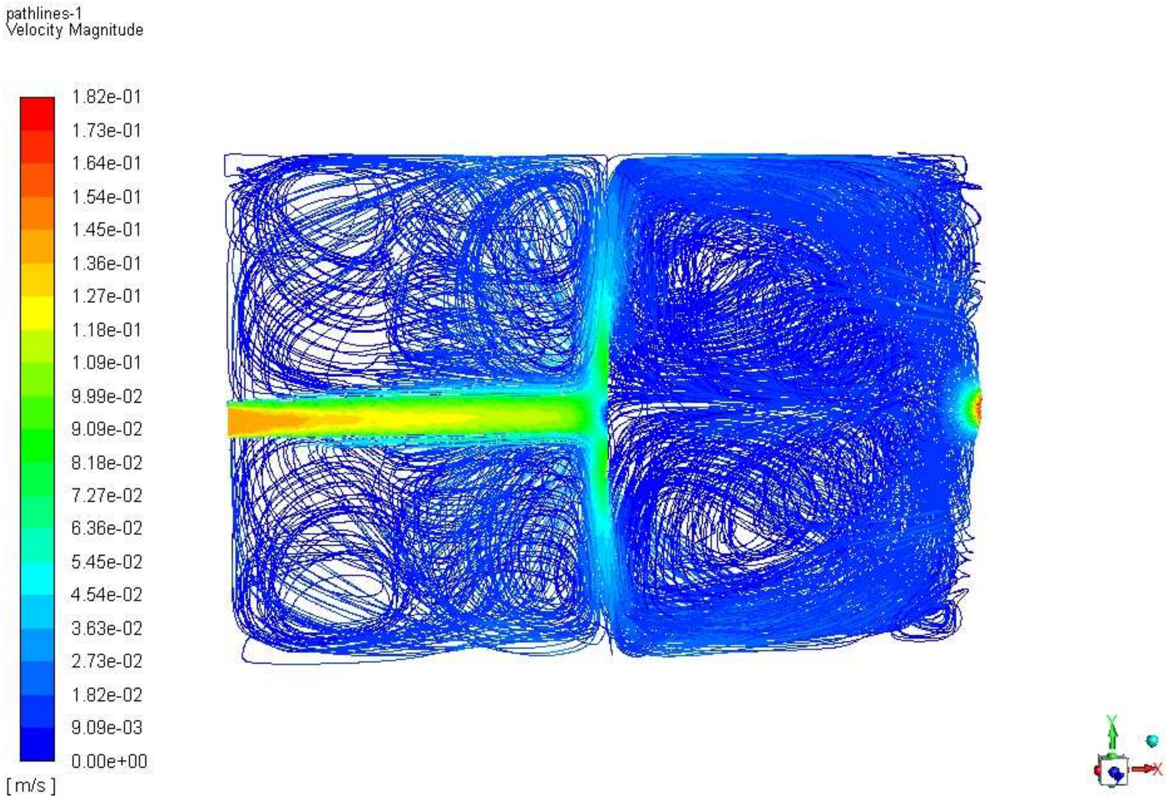


Fig 4.7 Path lines with a baffle wall

4.5 Particle tracking

Our main objective of sedimentation tank is to remove maximum suspended particles or sediments. In the below figures it is clearly understood that by the use of a baffle wall we can increase the efficiency of a sedimentation tank. By the use of a baffle, velocity will decrease significantly and residence time of particles will increase consequently more sedimentation takes place. The results of simulations of particle tracking are such that if we inject 246 particles from inlet in a sedimentation tank. Without baffle wall only 91 particles are trapped and the remaining are escaped but in case of sedimentation tank with baffle wall 216 particles are trapped from 246 and remaining are escaped therefore by the use of a baffle we can increase the efficiency of the particles tracking . We can also check the efficiency of the sedimentation tank with varying the velocity inlet, particle diameter, or by the changing, the dimensions of the sedimentation tank. Therefore, by the use of CFD analysis we can optimize the sedimentation tank. The efficiency is 37% of the sedimentation tank without a baffle wall and with baffle wall, efficiency will increase to 85.37%.

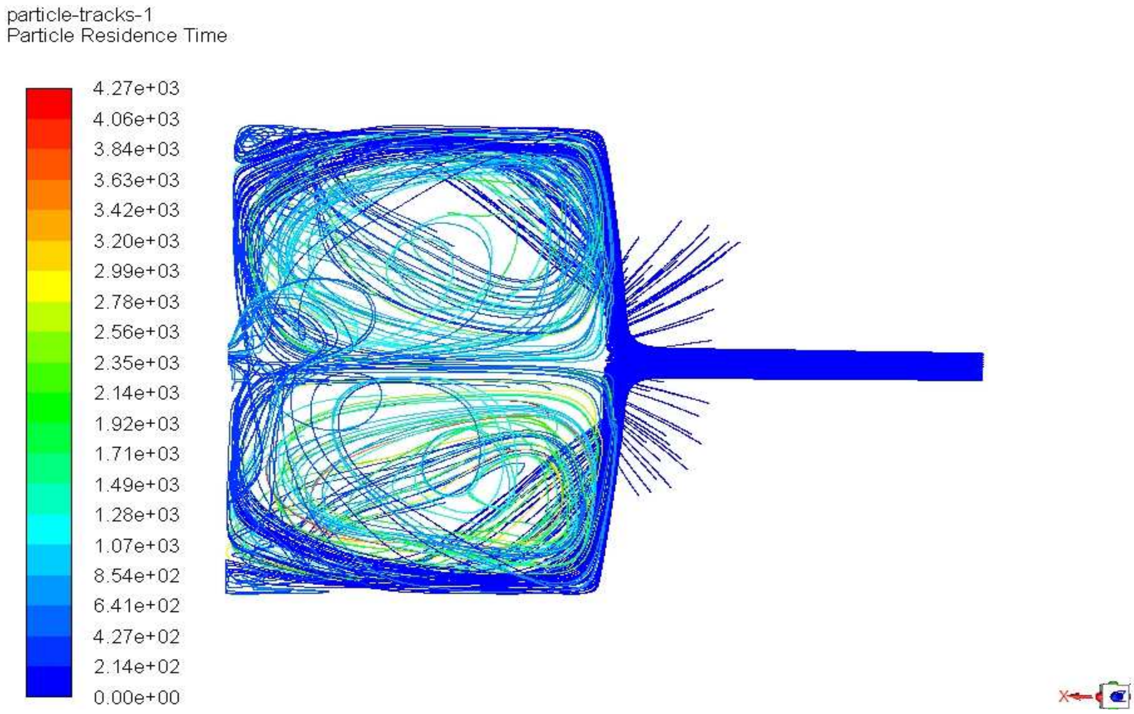


Fig 4.8 Particle tracking with baffle wall

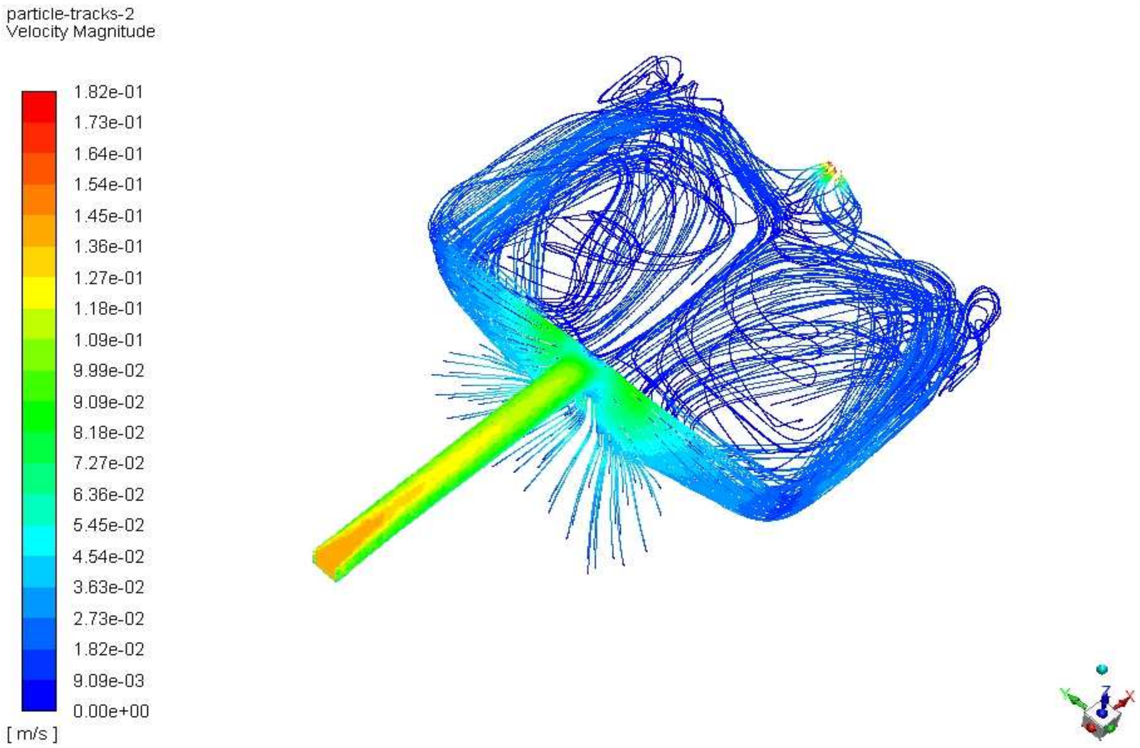


fig 4.9 Particle tracking with baffle wall and velocity magnitude

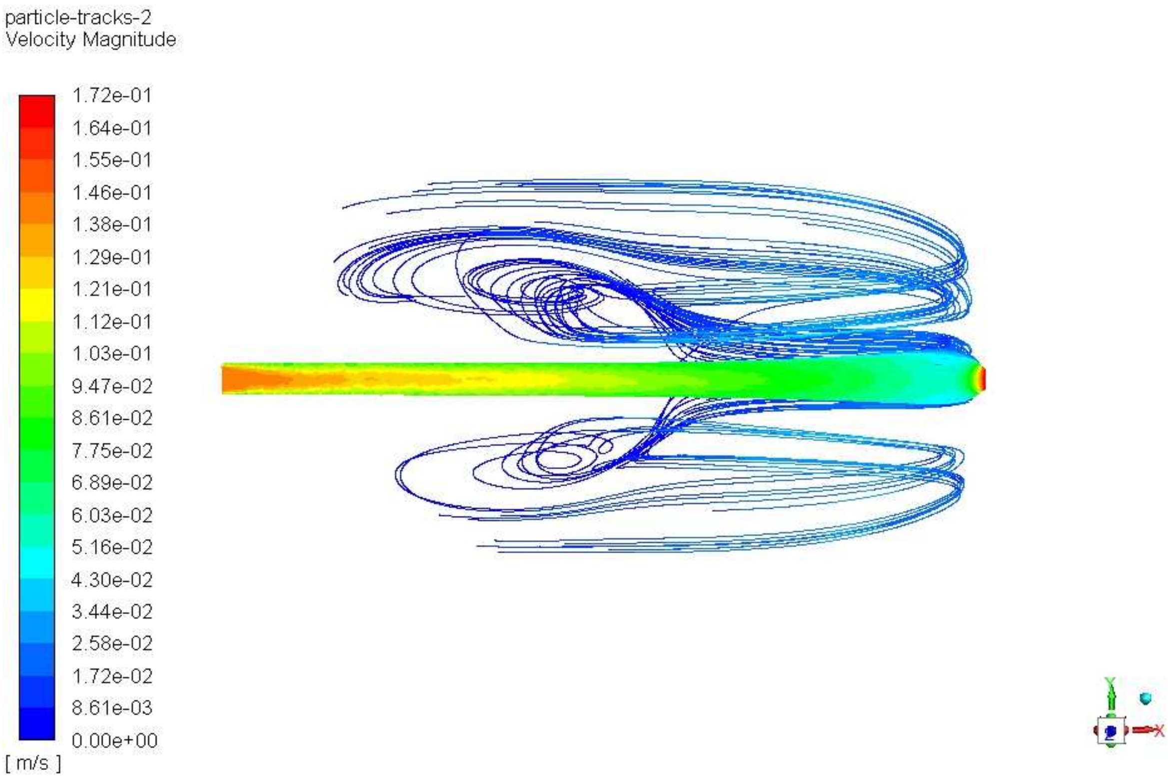


Fig 4.10 Particle tracking with velocity magnitude without a baffle wall

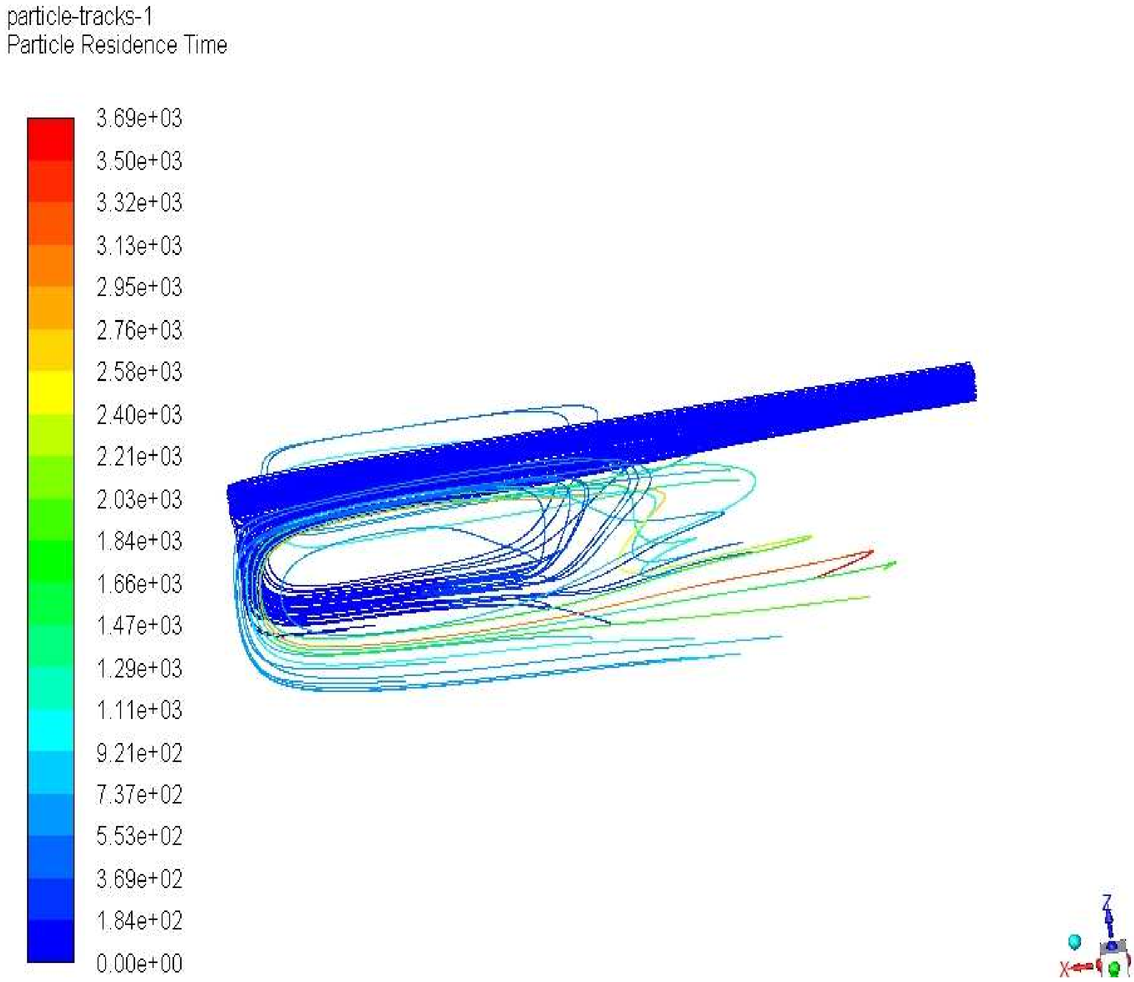


Fig 4.11 Particle tracking with residence time without a baffle wall

4.6 Streamlines

Streamline show the direction of flow of fluid. In case of sedimentation tank with baffle wall the flow of fluid has interrupted. Due to this disturbance, turbulence will generate subsequently velocity of fluid decrease. As we know sedimentation is inversely proportional to the velocity, therefore, particle tracking will rise and efficiency of tank would increase.

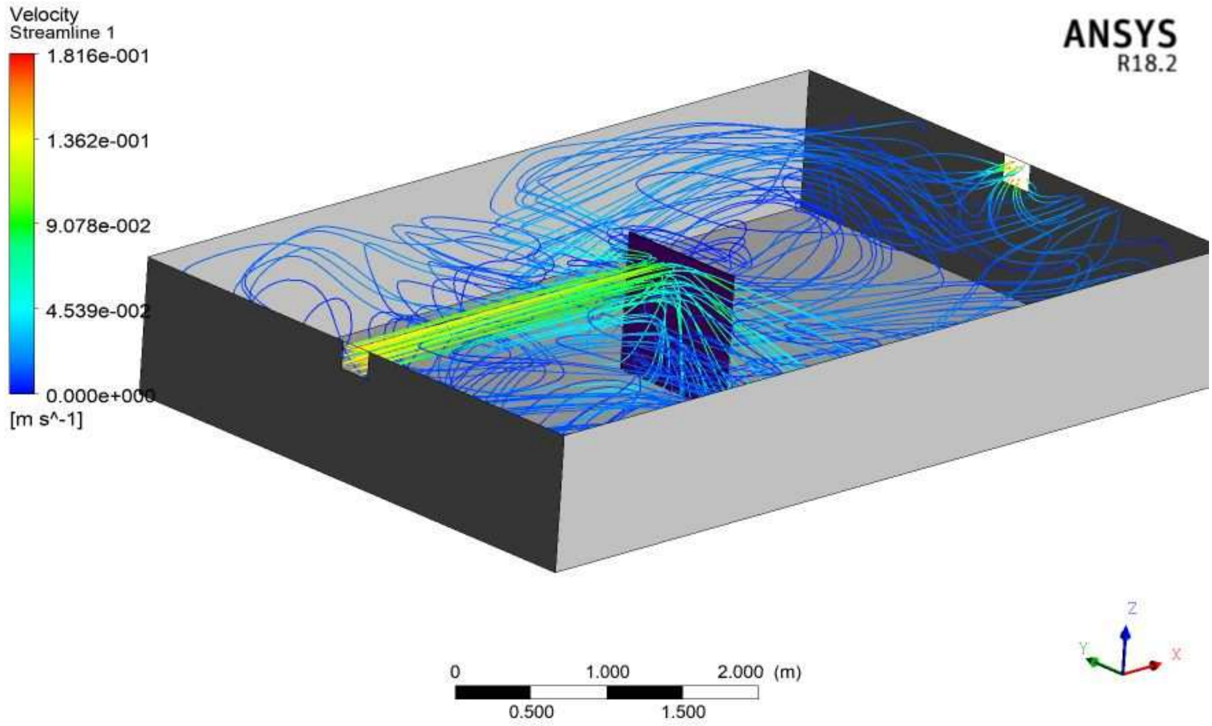


Fig 4.12 streamlines of a sedimentation tank with a baffle wall

4.7 Particle tracking with varying diameter

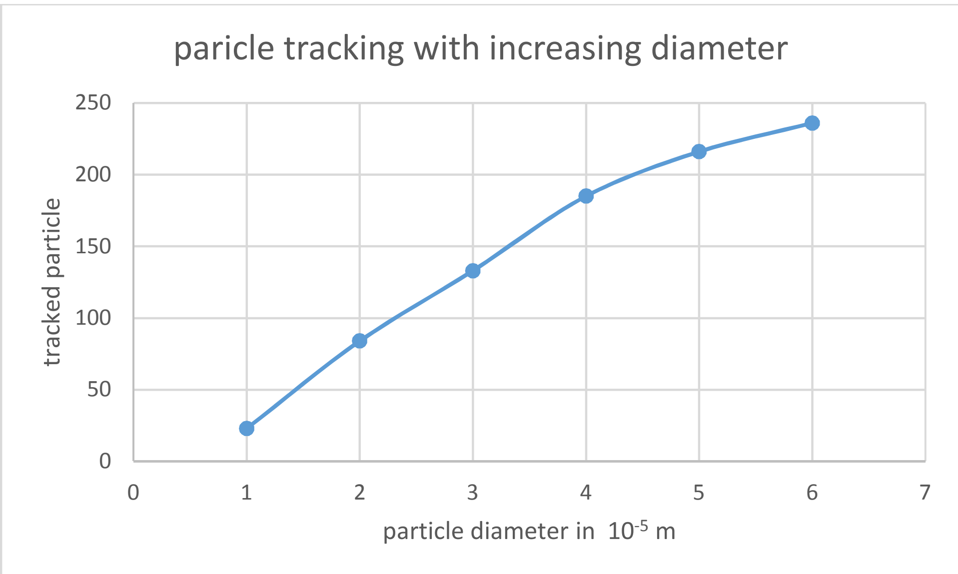


Fig 4.13

As shown in the graph with the increase in particle diameter sedimentation will increase. In the above graph diameter of particles has shown on x-axis and their dimensions are in 10^{-5} m. sedimentation or particle trapping will increase with size due to the effect of gravity. In addition, no. of particles has shown on y-axis.

4.8 Efficiency of tank with varying particle diameter

This graph shows the efficiency with increasing the diameter of the particle. The total injected particles are 246 at inlet . It is very clear that efficiency will increase with diameter and it will go up to 100% if we further increase the diameter of the particle.

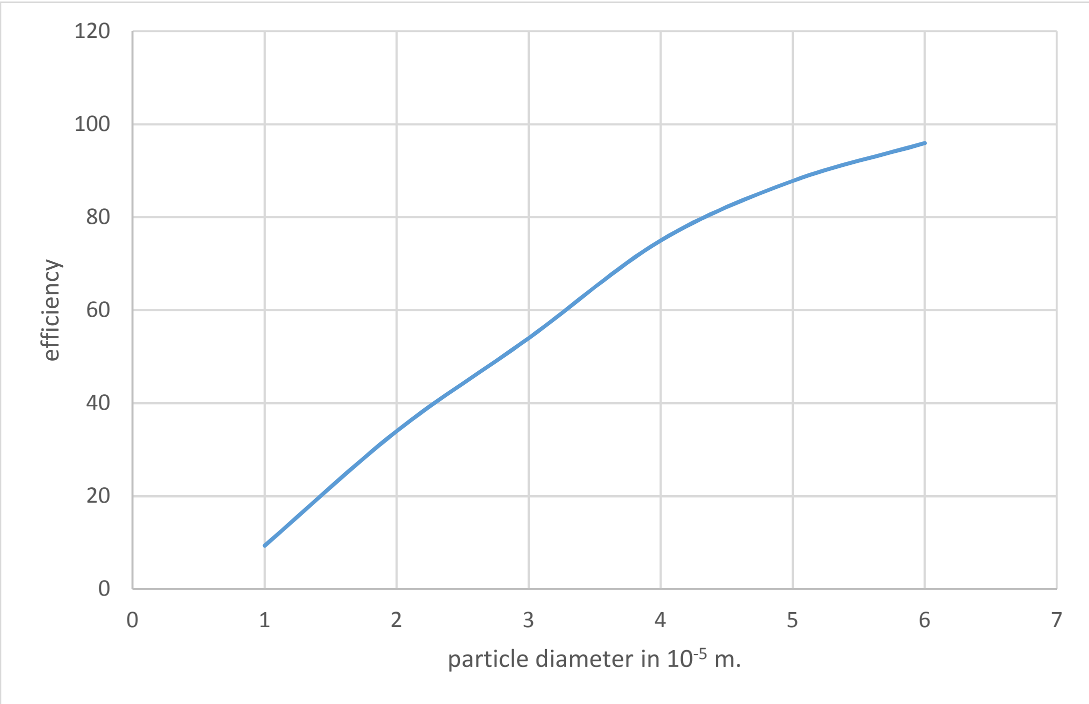


Fig 4.14 Efficiency of tank with varying diameter

4.9 Graph between particle injected and trapped

This graph shows the trapped particles if injected particles are varying. In the below graph two graphs are showing, in which the graph of baffle shows more trapping. However, the graph is not linear and in case of baffle wall the trapped particles will increase with the injected particles. It can be note down by the slopes between two points.in case of the tank without baffle the trapping will almost linear because in the case of baffle turbulence will create and a particle will affect the movement of other particle so particle trapping will increase.

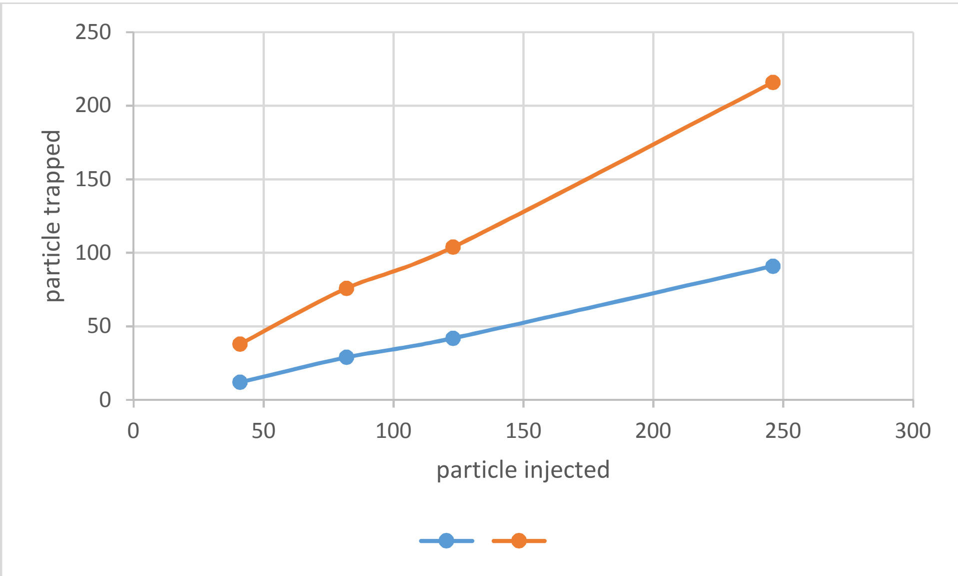


Fig 4.15

CHAPTER 5

CONCLUSION

In order to attain the aims stated above, a good knowledge of sedimentation tank and CFD required. CFD can easily applied in the optimization of sedimentation tank. Particle tracking in sedimentation tank can be achieve for different inlet velocities and for different particle diameters. CFD is an easy and inexpensive tool for optimizing a sedimentation tank. By the use of CFD, we can easily simulate the performance of sedimentation tank.

Baffle can be installed easily and economically without significant influence on the operation. Single phase CFD simulation result shows that baffle re-distribute incoming wastewater on the whole cross-section, reduced surface current and re-circulation regions. By displaying contour of velocity magnitude, and velocity vectors, we found baffle in suitable location and tilted bottom reduces the circulation zone and kinetic energy, create uniform velocity vector inside the sedimentation tank. Therefore, the baffle and tilted bottom improved the hydraulic efficiency of the original sedimentation tank.

In general, CFD provides new insight into the sedimentation tank and design, CFD supply useful information for further research and design. CFD can model internal changes (e.g. baffles, hopper shape and mixer) as well as external changes (inlet velocity, volume fraction and hydraulic load). So that CFD is a powerful tool that can optimize both the operation and geometry of water and wastewater treatment devices. The application of CFD is significant cost savings, because the effects of different geometry models and operation conditions are examined in computer, without implementation of any modification in the real world. The application of CFD can be employed as the first step in the process of optimizing wastewater treatment reactors.

References

Susumu Kavamura, Integrated Design of Water Treatment Facilities, John Wiley & Sons, 1991.

P. Larsen, On the hydraulics of rectangular settling basins, Report No. 1001, Department of Water Research Engineering, Lund Institute of Technology, Lund, Sweden, 1977

D.R. Shamber, B.E. Larock, Numerical analysis of flow in sedimentation basins, J. Hydr. Div, ASCE. 107 (1981) 575–591.

J.A. McCorquodale, E.M. Yuen, Z. Vitasovic, R. Samstag, Numerical simulation of unsteady conditions in clarifiers, Water Poll. Res. J. Can. 26 (1991) 201– 222.

J.A. McCorquodale, S. Zhou, Effects of hydraulic and solids loading on clarifier performance, J. Hydr. Res. 31 (1993) 461–477.

S. Zhou, J.A. McCorquodale, A.M. Godo, Short-circuiting and density interface in primary clarifiers, J. Hydr. Eng. 120 (1994) 1060–1080.

E. Imam, J.A. McCorquodale, J.K. Bewtra, Numerical modelling of sedimentation tanks, J. Hydr. Eng. 109 (1983) 1740–1754.

A.I. Stamou, E.A. Adams, W. Rodi, Numerical modelling of flow and settling in primary rectangular clarifiers, J. Hydr. Res. 27 (1989) 665–682.

E.W. Adams, W. Rodi, Modelling flow and mixing in sedimentation tanks, J. Hydr. Eng. 116 (1990) 895–913.

D.A. Lyn, A. Stamou, W. Rodi, Density currents and shear induced flocculation in sedimentation tanks, J. Hydr. Eng. 118 (1992) 849–867.

A.M. Goula, M. Kostoglou, T.D. Karapantsios, A.I. Zouboulis, A CFD methodology for the design of sedimentation tanks in potable water treatment case study: the influence of a feed flow control baffle, Chem. Eng. J. 140 (2008) 110–121.

X. Wang, L. Yang, Y. Sun, L. Song, M. Zhang, Y. Cao, Three-dimensional simulation on the water flow field and suspended solids concentration in the rectangular sedimentation tank, ASCE 134 (2008) 902–911

R. Kahane, T.V. Nguyen, M.P. Schwarz, CFD modelling of thickeners at Worsley Alumina Pty Ltd, Appl. Math. Model. 26 (2002) 281–296.

R. White, T.V. Nguyen, I. Sutalo, Modelling studies of fluid flow in a thickener feedwell model, Miner. Eng. 16 (2003) 145–150.

J.B. Farrow, P.D. Fawell, R.R.M. Johnston, T.B. Nguyen, M. Rudman, K. Simic, J.D. Swift, Recent developments in techniques and methodologies for improving thickener performance, Chem. Eng. J. 80 (2000) 149–155.

M. Righetti, G.P. Romano, Particle–fluid interactions in a plane near-wall turbulent flow, J. Fluid Mech. 505 (2004) 93–121.

F. Sbrizzai, V. Lavezzo, R. Verzicco, M. Campolo, A. Soldati, Direct numerical simulation of turbulent particle dispersion in an unbaffled stirred-tank reactor, Chem. Eng. Sci. 61 (2006) 2843–2851.

[18] G. Hetsroni, Particles-turbulence interaction, Int. J. Multiphase Flow 15 (1989) 735–746.

C. Li, A. Mosyak, G. Hetsroni, Direct numerical simulation of particle-turbulence interaction, J. Multiphase Flow 25 (1999) 187–200.

M.W. Reeks, The transport of discrete particles in inhomogeneous turbulence, J. Aerosol. Sci. 14 (1983) 729–739.

S.A. Kantoush, E. Bollaert, G. De Cesare, J.L. Boillat, A. Schleiss, Flow Field Investigation in a Rectangular Shallow Reservoir using UVP, LSPIV and numerical model, in: 5th International Symposium on Ultrasonic Doppler Methods for Fluid Mechanics and Fluid Engineering.

B.J. Dewals, S.A. Kantoush, S. Erpicum, M. Pirotton, A.J. Schleiss, Experimental and numerical analysis of flow instabilities in rectangular shallow basins, Environ. Fluid Mech. 8 (2008) 31–54.

J. De Clercq, J. Jacobs, D.J. Kinnear, I. Nopens, R.A. Dierckx, J. Defrancq, P.A. Vanrolleghem, Detailed spatio-temporal solids concentration profiling during batch settling of activated sludge using a radiotracer, Water Res. 39 (2005) 2125–2135.

R. Tarpagkou, A. Pantokratoras / Applied Mathematical Modelling 37 (2013) 3478–3494

A.N. Georgoulas, P.B. Angelidis, T. Panagiotidis, N.E. Kotsovinos, 3D numerical modelling of turbidity currents, Environ. Fluid Mech. 10 (2010) 603–635.

L. Ma, H.D.B. Inghamst, X. Wens, Numerical modelling of the fluid and particle penetration through small sampling cyclones, J. Aerosol Sci. 31 (2000) 1097–1119.

B.F. Armaly, F. Dursts, J.C.F. Pereira, B. Schonung, Experimental and theoretical investigation of backward-facing step flow, J. Fluid Mech. 127 (1983) 473–496.

S.V. Patankar, Numerical Heat Transfer and Fluid Flow, McGraw-Hill Book Company, New York, 1980.

M.P. Schwarz, Improving zinc processing using computational fluid dynamics modelling – successes and opportunities, Miner. Eng. 30 (2012) 12–18.

D. Geropp, H.J. Odenthal, Drag reduction of motor vehicles by active flow control using the Coanda effect, Exp. fluids 28 (2000) 74–85.

C. Tao, Y. Zhang, D.G. Hottinger, J.J. Jiang, Asymmetric airflow and vibration induced by the Coanda effect in a symmetric model of the vocal folds, J. Acoust. Soc. Am. 122 (2007) 2270–2278.

J.W. French, W.G. Guntheroth, An explanation of asymmetric upper extremity blood pressures in supravalvular aortic stenosis: the coanda effect, J. Amer. Heart Ass. 42 (1970) 31–36.

P.R. Owen, Pneumatic transport, J. Fluid Mech. 39 (1969) 407–432.

K. Mohanarangam, D. W. Stephens, CFD modelling of floating and settling phases in settling tanks, in: Seventh International Conference on CFD in the Minerals and Process Industries CSIRO, Melbourne, Australia, 2009.

J.D. Kulick, J.R. Fessler, J.K. Eaton, Particle response and turbulence modification in fully developed channel flow, J. Fluid Mech. 277 (1994) 109–134.

M. Mandø, L. Rosendahl, On the motion of non-spherical particles at high Reynolds number, Powder Technol. 202 (2010) 1–13.