COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS OF RECTANGULAR MULTI BAFFLED CHAMBER USING ANSYS FLUENT

A Dissertation submitted in partial fulfillment of the requirement for the Award of degree of

MASTER OF TECHNOLOGY

IN

HYDRAULICS & FLOOD ENGINEERING

BY

MOHIT YADAV
(ROLL NO. 2K13/HFE/11)

Under the Guidance of

Dr. RAKESH MEHROTRA

Associate Professor
Department of Civil Engineering
Delhi Technological University
Delhi

DELHI TECHNOLOGICAL UNIVERSITY
(FORMERLY DELHI COLLEGE OF ENGINEERING)
DELHI - 110042
July-2015

CANDIDATE'S DECLARATION

I do hereby certify that the work presented is the report entitled "**Computational fluid dynamics(CFD) Analysis of Rectangular Multi Baffled Chamber using ANSYS Fluent**" in the partial fulfillment of the requirements for the award of the degree of "Master of Technology" in Hydraulics & Flood engineering submitted in the Department of Civil Engineering, Delhi Technological University, is an authentic record of our own work carried out from January 2015 to July 2015 under the supervision of Dr. Rakesh Mehrotra (Associate Professor), Department of Civil Engineering.

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

Mohit Yadav

Date:30/7/15 (2K13/HFE/11)

CERTIFICATE

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

Dr. Rakesh Mehrotra
(Associate Professor)
Department of Civil Engineering
Delhi Technological University

ACKNOWLEDGEMENT

I take this opportunity to express my profound gratitude and deep regards to Dr. Rakesh Mehrotra (Associate Professor, Civil Engineering Department, DTU) for his exemplary guidance, monitoring and constant encouragement throughout the course of this project work. The blessing, help and guidance given by him from time to time shall carry me a long way in life on which I am going to embark.

I would also like to thank Dr. Nirendra Dev (Head of Department, Civil Engineering Department, DTU) for extending his support and guidance.

Professors and faculties of the Department of Civil Engineering, DTU, have always extended their full co-operation and help. They have been kind enough to give their opinions on the project matter; I am deeply obliged to them. They have been a source of encouragement and have continuously been supporting me with their knowledge base, during the study. Several of well-wishers extended their help to me directly or indirectly and we are grateful to all of them without whom it would have been impossible for me to carry on my work.

ABSTRACT

In this Dissertation, using ANSYS Fluent software (R14.5) computational fluid dynamics (CFD) analysis of 3D Rectangular Baffled Chamber is done. The Model consists of combination of vertical standing as well as hanging angled baffles. Flow patterns are taken into consideration. The dead zones and stream lines together with velocity vectors are analyzed of chamber by changing number of baffles in terms of spacing and the discharge coming into the chamber. In a finite volume, the numbers of chambers form 4 to 6 are increased by reducing the spacing in terms of width between them and discharge from 10 liters per hour to 50 liters per hour is varied to value of less, equals and more than the capacity of chamber. The combination of stream lines and Velocity vectors covering maximum area of chamber is presented which will increase the (baffling performance) efficiency as well as the effective volume utilized. In results, the velocity graph at locations i.e. near bottom, mid depth and near surface of models containing different number of chambers is shown. The Results obtained can be used in further improvement of ABR design.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

Chapter-1

INTRODUCTION

Hydrodynamics is one of the aspects which should be investigated during operation and design of reactors whether full/pilot or laboratory scale for understanding the fluid flow distribution. In all cases, the degree of treatment depends largely on the flow regime (plug flow/completely mixed flow or intermediate flow dispersion) and effective detention time (could vary with the flow regime for same reaction kinetic coefficients). Therefore, estimation of mean residence time and residence time distribution for such basins is important, because, along with relationships between treatment process efficiency and time, these are used to calculate expected efficiencies for specific basin designs. The ideal residence pattern in most treatment basins would be plug flow, but this is impossible to achieve in practice. Dispersion caused by unsteady flow rates, wind, inlet-outlet effects, and shear stresses at the sides and bottom cause some parcels of fluid to exit earlier than *T (theoretical detention time)* and some to exit later. These deviations from plug flow vary continuously as conditions, such as sediment buildup/suspended solid concentration, wind effect and gas evolution etc.

There are also "dead zones" within the basins, in which velocities toward the outlet are considerably less than average, and in which eddy recirculation currents exist. In a dead zone the rate of exchange of water between the dead zone and the main advective flow zone is much slower than for an internally-mixed zone. Parcels of water which enter dead zones have very long residence times in them and a high percentage of suspended solids are removed there. However, the presence of dead zones adversely affects the overall treatment efficiency of a basin, because the dead zone volume is unavailable to the main flow, thus reducing the mean residence time for most of the inflow. Dead zones tend to occur in shallow or hydraulically rough areas, corners, and areas behind baffles or obstructions that are sheltered from the mixing effects of advective flow or wind. These zones are not part of the volume through which water flows; thus, the **effective** reactor/basin volume is less than the total volume, and the mean residence time (t) is less than T. The hydraulic efficiency (*t/T)* partially describes this departure from ideal flow. In order to achieve a particular value of *t* and therefore to achieve a specified treatment efficiency, the theoretical residence time (*T)* must be made larger than *t* by a factor equal to the hydraulic efficiency correction factor (HECF) which is the reciprocal of the hydraulic efficiency. If an engineer can design a basin to increase the hydraulic efficiency, given a particular value of *t*, the value of *T and* thus the basin volume and cost, can be reduced.

Historically, Reactors have been designed by empirical co-relation, physical model studies or tracer studies conducted after constructing the reactor. With the availability of computation facilities and improvement in modeling software, efficient alternatives in the form of computational fluid dynamics software have become available. CFD approach allows us to simulate the real conditions happening inside the study model with ease and accuracy that is very difficult to achieve through physical model studies. Velocity and flow distributions are studied which relates with the short circuiting as well as dead zones that tend to develop in chambers. The velocity of fluid particles at each and every point can be known (comprising real data), whereas in the physical modelling setup, it is very difficult to find out the coordinates and velocity of flowing particles and to do that advanced instruments like stroboscope, Pitot tube, laser anemometry etc. are needed.

In case of baffled reactors design, various alterations of configuration may be required to come up with best optimal design. In physical modelling the exercise will become very cumbersome and time taking to actually check different configurations for residence time distribution, velocity and flow distribution to arrive at the best choices. In case of alteration, the entire set up is changed to begin with starting but with CFD alterations can be done at every stage without being doing work from scratch. The availability of data is in the records which can be seen whenever it is needed. The results got in CFD analysis can be easily applied to reactors, contact tanks and the effects can be seen in animation. Therefore, CFD allows researchers to set up model as per their requirements with less cost and time.

The path of flow of water can be increased by using number of baffles at straight or some angled positions. Mainly in waste water treatment as well as in potable storage water reservoir, the water is allowed to move but to perform biological reactions flow has to be confined. Therefore, the term Baffle comes into consideration. **Baffles** are also medium by which mixing of flow in fluid system as well as in heat exchangers can be done. They can be provided in reactors, contactors and in service reservoir. In reactors like **ABR** (Anaerobic Baffled Reactor), these are one of the main designing as well as defining component upon which the whole working of reactor depends. In Ozone contactors and service reservoir of water, baffles serve to be achieving better water quality as without baffles they act as a storage tank in which water quality is not as high as in mixed or moving flow. Different Baffles cut are shown below:

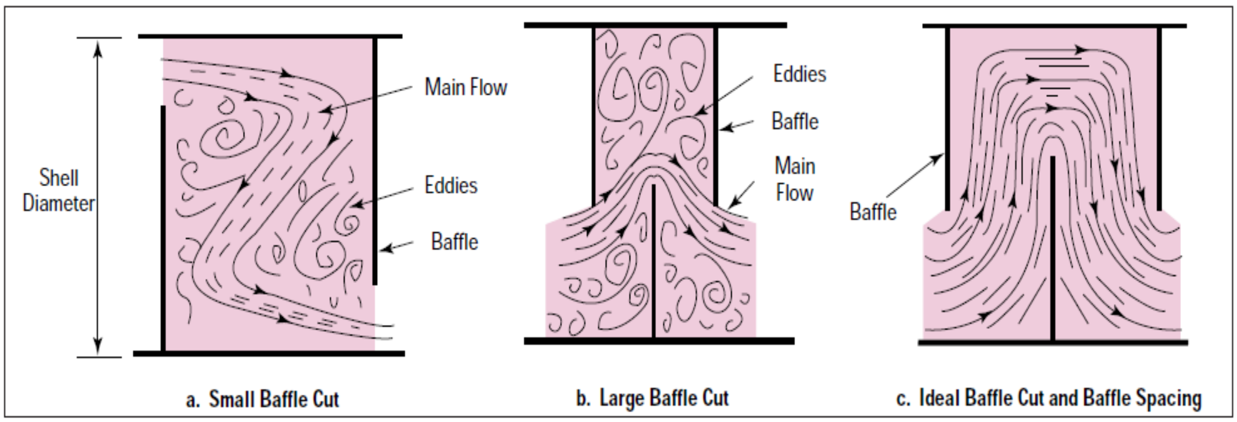


Fig 1.1 **Different baffle Cut**

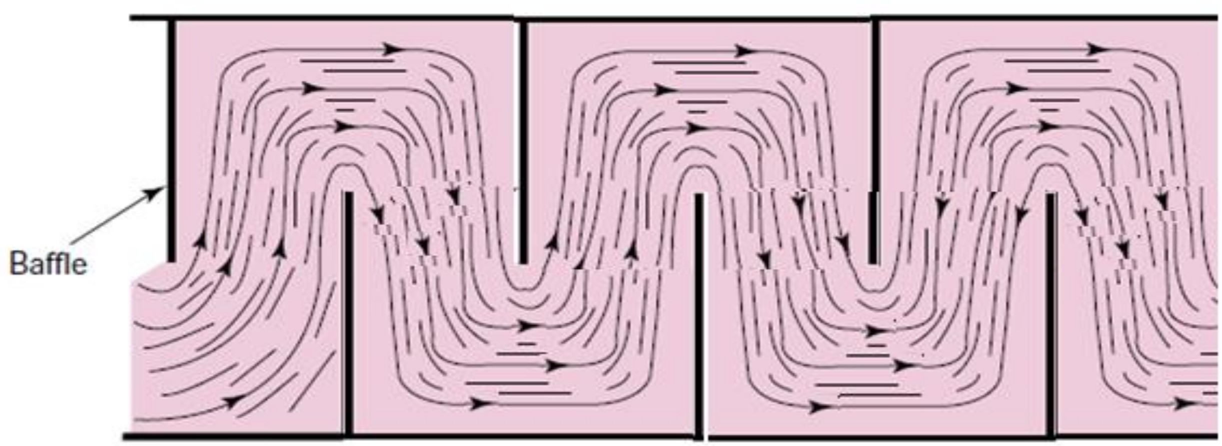


Fig 1.2 **Ideal Baffle Cut**

1.1 TYPES of Baffle:

1) **Over and under Baffle system**

This system comprises of a weirs and sluices combination. Water entering in each compartment rises over weir to a sufficient level which will be distributed over the width by orifice of several means.

2) **End Around Baffle system**

Water entering in chambers one by one through successive bends. Series of bends are provided in order to make whole baffle system. Water move around every bend in such a manner that whole system volume is used for flow of water. The below figure showing top view of End Around system.

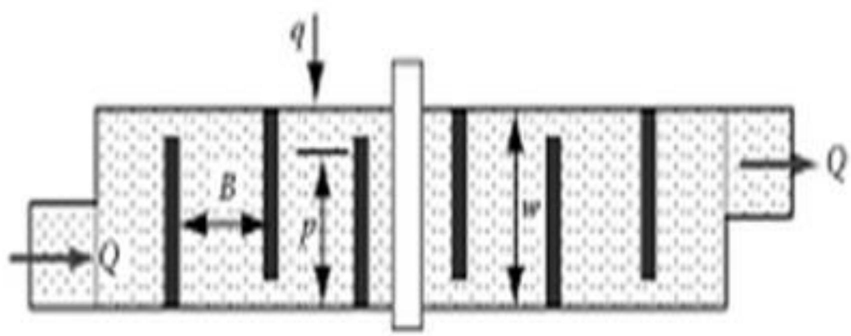


Fig 1.3 **End-Around Baffle System**

These Baffle designs can be used in various places such as ABR, Potable water reservoir working as a storage tank, ozone contactors and many more places where fluid has to flow in confined manner. The main focus is on the baffle arrangement particularly in case of ABR and learning the different **flow patterns** occur when the spacing of chambers is decreased by increasing number of baffles in same width of model.

1.2 ABR (Anaerobic Baffled Reactor)

Anaerobic Baffled Reactor is a secondary waste water treatment system which is a modified version of septic tank which consists of a number of baffles hanging straight or at some angled position and standing position. In this, both physical as well as biological treatment is done by microbial decomposition of waste water organics. Active biomass is required to be retained/maintained in desirable concentrations in the various chambers. The liquid i.e. waste water entering from inlet is allowed to move downward and then upward mode. In upward movement the solid particles will tend to settle down if the up flow velocity is less than settling velocity of the particles in the flow chamber. The top head space in the reactor chambers is provided for gas formed by biological activity in the chambers. After reaching last chamber the treated waste water (effluent) is obtained from outlet. ABR systems are suitable for higher organic loading rate, comprising soluble BOD/non-settable solids having low COD/BOD ratio. The range of flow in ABR starts form 2 to 200 m^3 per day. The Hydraulic Retention Time (HRT) is in between 48 to 72 hours. The main designing criteria are the upward velocity flow which must not be greater than the 2m/h (which is necessary for solids to settle down) to prevent wash out of biological solids. The chambers can be separated by vertical pipes or Baffles. In each chamber accessibility is necessary for the maintenance.

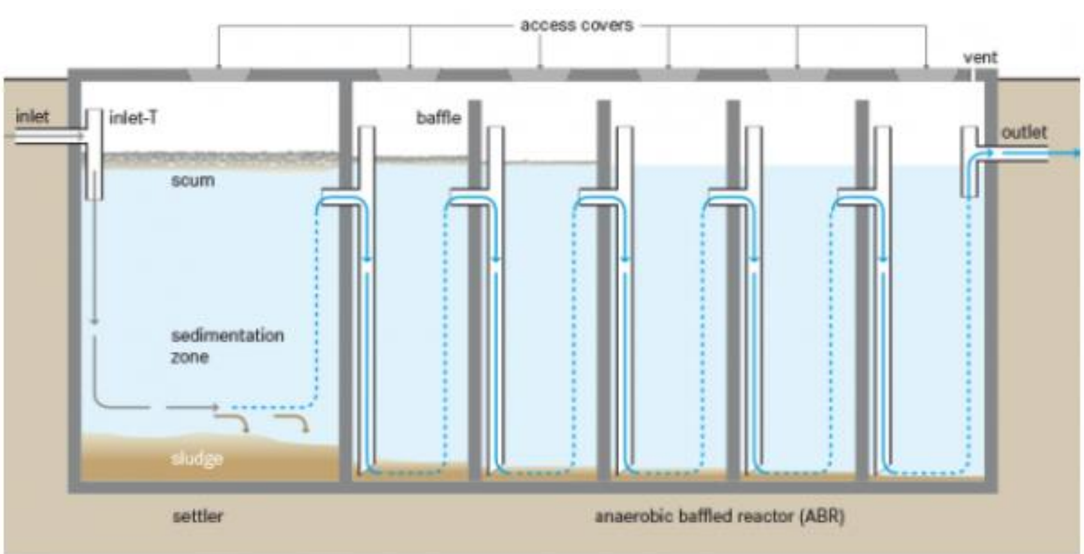


Fig 1.4 ABR (Anaerobic Baffled Reactor)

The following 3 processes occur in ABR are:

I. **Hydrolysis -** It is the first stage in which particulate matter is converted into soluble compounds that can be hydrolysed into the stiffener compounds by the acid formers.
II. **Fermentation** (acidogenesis) **-** In this stage, the soluble compounds from the previous stage are converted into acids and alcohols of lower molecular weight by action of acid formers.
III. **Methanogenesis -** The acid & alcohols produced are converted into the methane by the action of methane formers.

By these processes, COD is removed serially. Multiple chambers are divided into up flow and down flow zones. The up flow zones are the main reactor zones in which biological treatment is taking place and simultaneously the settling of the solids is taking place and methane gas and carbon dioxide formed. In first compartment settling down of highest percentage of solids takes place and decomposition of solids occurs and the products flow to second chamber. Second chamber will support microorganisms according to the nature of substrate available. Similarly third and fourth chambers will also have predominance of microorganisms matching with the substrate. Each chamber starting from first will have predominance of hydrolysing acid formers or methanogen depending upon the stage. The first stages will have hydrolysing; next stage will be of acid formers and last is methane formers. The ABR functions more efficiently because it is microbially multi phasic reactor and every chamber performs a specialized function.

1.3 Assumptions in the dissertation

1. One phase flow is considered that is effect of solids and gas are not considered.
2. Water is used as fluid in the analysis.
3. The other biological treatment parameters like organic loading rate, waste water characteristics etc. are also not considered.
4. Physical modelling requires more precision, time and cost, using CFD approach analysis can be done with ease. Therefore, in this study whole work is done using **ANSYS Fluent (R14.5)** software and the CFD analysis of chamber is done using it.

1.4 ANSYS Fluent

ANSYS Software allows us to do engineering simulation of fluid dynamics, Structural mechanics, Electromagnetics, Hydrodynamics, Multiphysics and many more. Here we are talking about Fluent as it deals with the fluid Dynamics of system. The company was found by Mr John A. Swanson. Computational Fluid Dynamics (CFD) is a computer based mathematical tool. The emerging interest on the use of CFD based simulation by engineers has long been analyzed in various fields of engineering. The basic principle in the application of CFD is to determine fluid flow in-detail by solving a system of non-linear governing equations over the region of interest, after applying specified boundary conditions. The CFD based simulation confides on combined numerical accuracy, modeling precision and computational cost.

Using ANSYS CFD, the system of fluid flow can be virtually simulated using a computer. One can start analysing by making a mathematical model of physical system. The CFD approach comprises of 3 methods Finite Difference Method, Finite Volume Method and Finite Element method.

1.4.1 Advantages of CFD

I. It lowers the cost of simulation and the geometry can be changed as many as times till the accurate result is obtained.
II. It can perform simulations at much high speed with error margin neglible.
III. It can make the model to work at real conditions which is very difficult to make in experimental models.

1.4.2 Limitations of CFD

I. The model results are totally based on physical model so it should be made correctly.
II. The accuracy of model is perfect till the given initial and boundary conditions are good.

1.5 <u>Objective of Dissertation</u>

The objective of this study is:

I. To verify the dead zones tend to develop near the chamber walls, surface and baffle position by doing flow analysis obtained from the ANSYS Fluent.
II. To maintain the downward and upward velocity flow within the specified range.
III. To identify the maximum velocity after the inlet in the chamber nearby baffle end.
IV. To make the stream lines and vector of flow in proper manner utilising the total volume capacity of chamber.

1.6 <u>Organisation of Dissertation</u>

The report is subdivided into 6 chapters. *Chapter 1* states the importance of the matter and aim of the study. *Chapter 2* presents the review of the literature available on this topic. The methodology and numerical data involved in this dissertation work is discussed in *Chapter 3* and *Chapter 4* respectively. The results and the discussions are enlisted in *Chapter 5*. Finally in *Chapter 6,* conclusions and scope of future work are discussed.

Chapter-2

LITERATURE REVIEW

Liaqat A. Khan et al. [1] A three-dimensional (3D) computational fluid dynamics (CFD) model of a contact tank is presented. STAR (CD) Computational Dynamics, 3D CFD software, was used in this study. The software solves the fundamental equations for fluid flow, conservation of mass, and momentum, known as the Reynolds-averaged Navier-Stokes equations, as well as equations representing turbulence characteristics to determine eddy viscosity and mixing coefficients by a second-order accurate finite volume method. The objective is to demonstrate that CFD models can simulate both the FTC and the 3D velocity field quite well. The physical model was 200 cm long and 94 cm wide, and was divided into eight chambers by baffles. The flow rate into the contact tank was 1.17 l / s and the corresponding mean water depth was 53.6 cm. The results were validated against velocity and tracer concentration data from a 1:8 scale physical model.

J. Zhang, M.ASCE et al. [2] with the help of CFD, the analysis of ozone reactors can be done. Reactor was made using Open FOAM software. Three designs are made using normal, half and quarter width. Simulation is done using RANS equation. The objective is to reduce dead zones regions and short circuiting that make the model works less efficient. RTD of tracer is also shown. Losses of energy as well as performance of baffles are studied together.

Jun-Mei Zhang et al. [3] In this study the potable water service reservoir is presented. CFD software Star-CCM+ (version 5.04) is used in this study to investigate the effects of baffle configurations on the flow pattern, chlorine concentration, and mean age distributions of service. To simulate the turbulent flow field inside the tank, the RANS (Reynolds-time-averaged Navier-Stokes) model is employed. Five individual baffle configurations proposed for the service reservoir. The results of this study show a dual effect of the baffles located at the flow recirculation region. On one hand, it can break up the vortex to shorten the flow path. On the other hand, the velocity magnitude of the fluid is reduced after flowing past the baffle, because of the impact and viscous forces induced by the baffle. These two effects are contradictory to one another in enhancing the performance of the service reservoir acting as a storage tank, because short flow path and high velocity magnitude is preferred to achieve better water quality. The overall effect of baffling is found to be counter-productive.

Dongjin Kim et al. [4] A 3D multi chamber model of different chamber width are numerically analysed which serves for function of ozone contactors. The approach is numerical as well as using software SSIIM model. The models of normal and half width are analysed. In this study, simulations were performed applying two different numerical modeling approaches i.e. LES and RANS. The results from the RANS model are used only for comparison purposes.. LES results are found to be more satisfactory as they are near to accurate. The tracer transportation is also done.

Daniel Kevin Peplinski et al. [5] This study has been done to evaluate the predictive capabilities of computational fluid dynamics (CFD) models of disinfection contactor hydraulics under model input uncertainty. The study consists of modeling the transport of a chemical tracer in a full-scale reactor and predicting the effluent residence time distribution (RTD) curve. An uncertainty analysis using Monte Carlo probabilistic techniques was used to determine the sensitivity of the effluent RTD to uncertainty in the influent turbulent kinetic energy constant, the turbulent Schmidt number, the wall roughness height, the influent turbulent length scale, and the turbulence model selection. The study determines the tracer is being transported in full scale chemical reactor which will predict the RTD curve of effluent. The prediction of RTD curve of effluent in unbaffled and baffled reactor is compared. The constant kinetic energy for the turbulent influent length scale evaluates change in Morril index. The variations in RTD of effluent increases by decreasing hydraulic efficiency of contactor.

Chris Brouckaert et al. [6] A working model of ABR is constructed based on a 10 litres CFD model demonstrated. The model is created in PreBFC (a preprocessor used for FLUENT). The model was made to work at South Africa. Hanging straight baffle which are angled at $45^0$from bottom is there in model. Results are compared of 1:1 to 1:3 baffle arrangements. The vectors of both arrangements are shown. Settling tests as well as tracer test are done. The advantage of using a dye tracer is that visual observations allow for a better understanding of the flow pattern. The flow patterns observed during the dye tracer tests were compared to the velocity vector projections and velocity contour plots obtained from the CFD model. The result is made that 1:3 with 45^0angle in baffle made the most accurate close to real model which provides higher settling of solids at the bottom.

Chapter-3

METHODOLOGY

3.1 Numerical method

The Rectangular multi baffled chamber has been analysed in ANSYS FLUENT. The process of the numerical simulation of fluid flow generally involves three steps:

(a) Pre-Processing

- Geometry set-up and Discretization of domain
- Defining the flow condition (laminar, turbulent etc.)
- Defining the boundary condition and initial condition

(b) Solver

- The equation emphasizes over and over till desirable level of accuracy is attained.

(c) Post processing

- Results are analysed.

3.2 Making a MODEL

ANSYS provides platform for designing, meshing, solving and then the results can be visually seen in graph, contours, vector, streamlines, volume rendering and particle track.

3.2.1 Geometry Setup

ANSYS Workbench has its own geometry making software called Design Modular embedded in it. So the making of model starts with its geometry which is most time consuming and the percentage of accuracy is associated with it. The dimensions of chamber are $42 \times 22 \times 20$cm. Three models are made of same dimensions with four, five and six chambers in it by reducing the width of chamber i.e. In Model-I (four chambers) width of chamber is 10cm. In Model-II (five chambers) width of chamber is 8cm. In Model-II (Six chambers) width of chamber is 6.5cm.

Table 3.1: Nomenclature of geometries

Dimension	Value (cm)
Length of Model	42
Width of Model	22
Depth of Model	20
Inlet width	2
Outlet Width	2
Length of Standing Baffle	16
Length of hanging baffle	14
Length of angled baffle	7.5
Width of all baffle	.25
Angle of Baffle	45^0

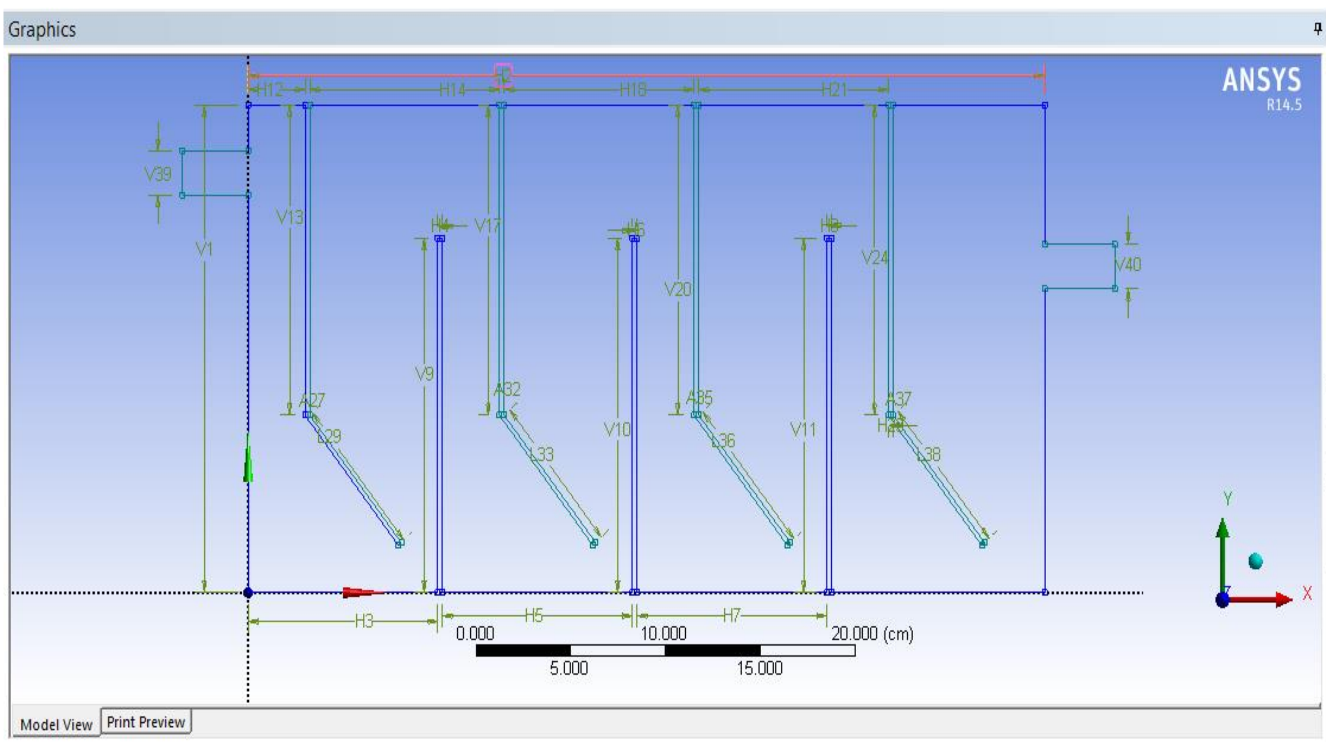


Fig3.1 Geometry of 4 MODEL-I

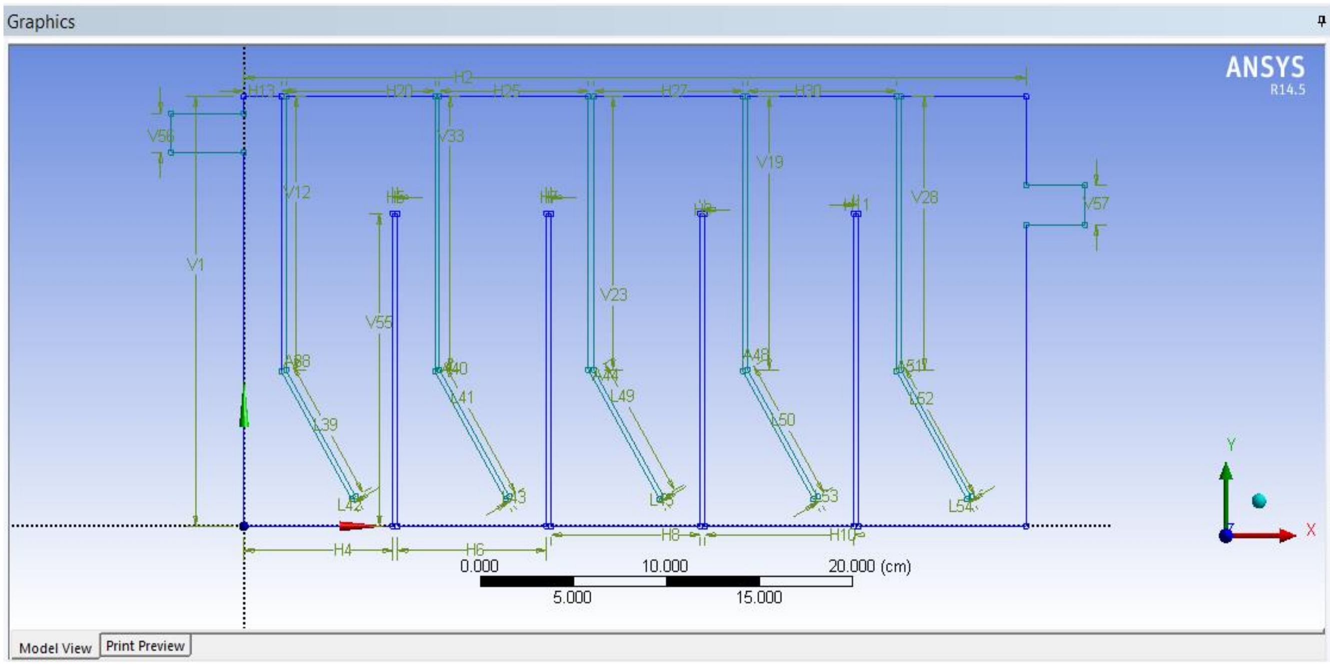


Fig3.2 Geometry of MODEL-II

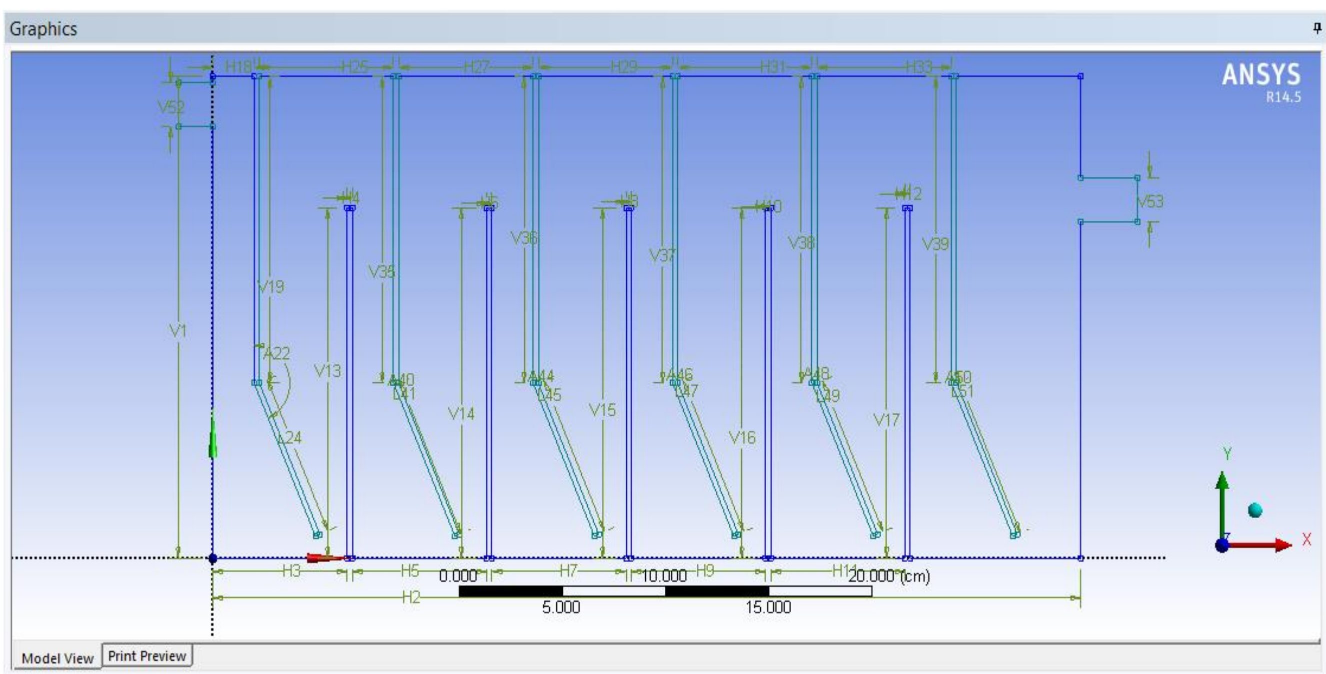


Fig3.3 Geometry of MODEL-III

3.2.2 Meshing

ANSYS workbench has its own meshing software. The meshing of all 3 models is done separately. All the models are tetrahedral medium meshed.

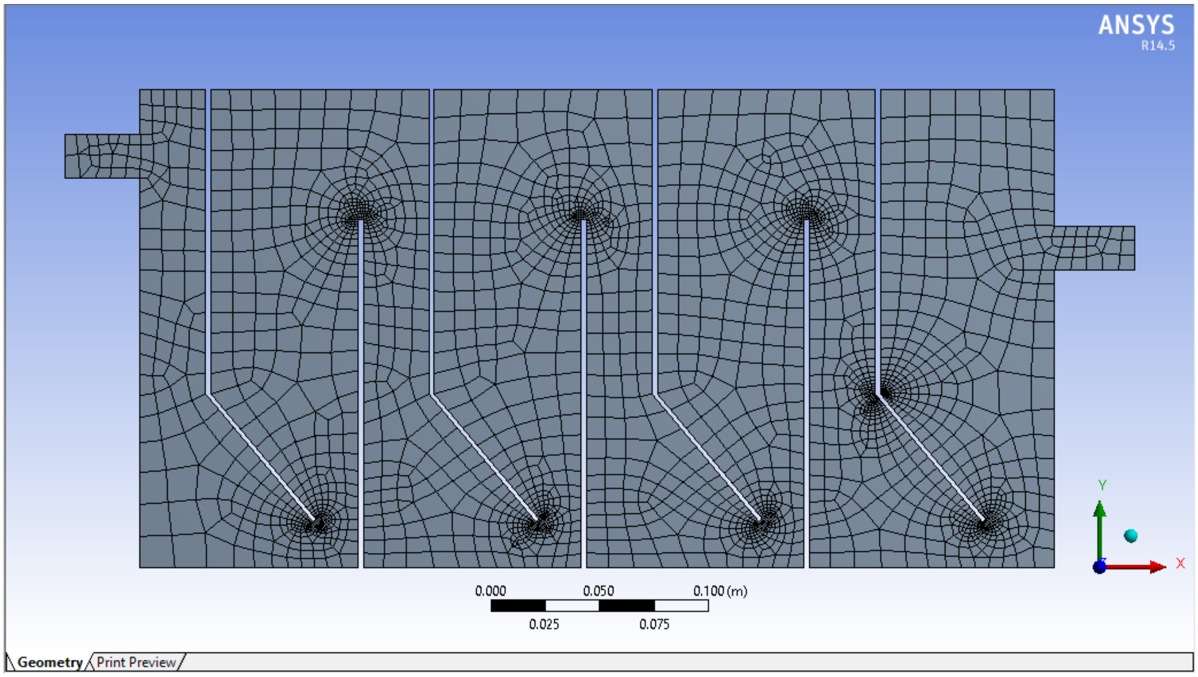


Fig3.4 Meshing of MODEL-I

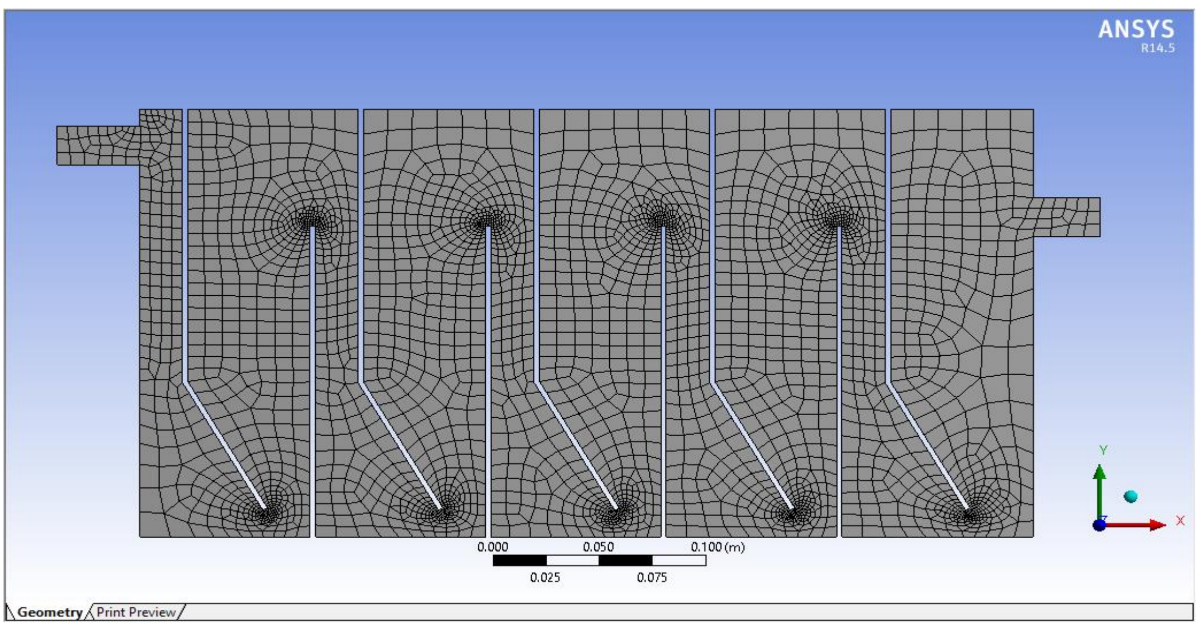


Fig3.5 Meshing of MODEL-II

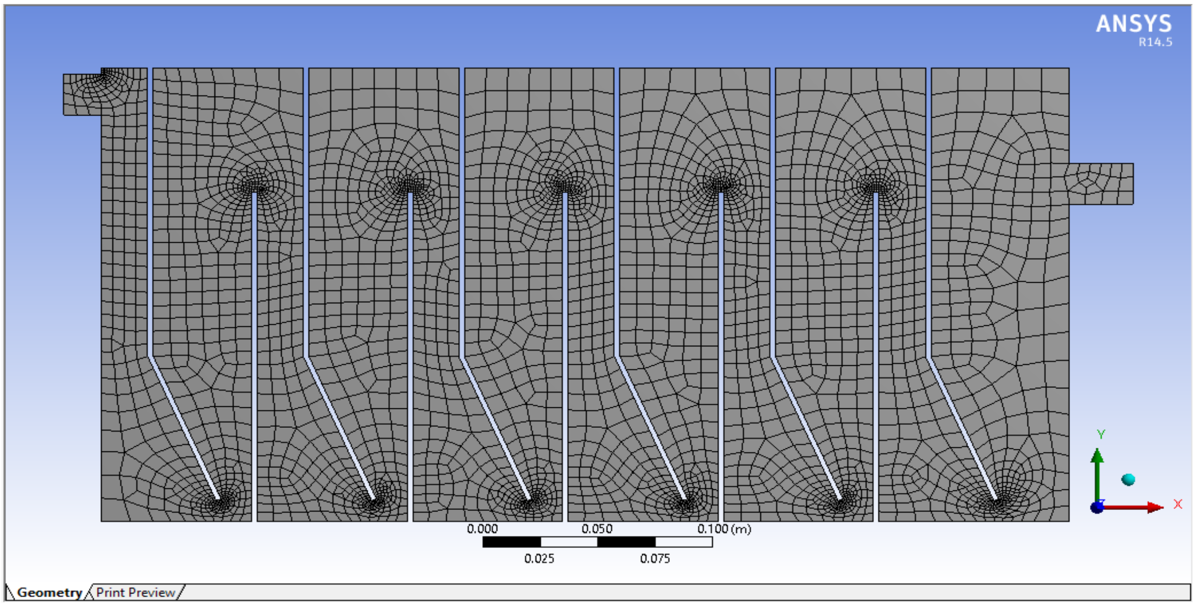


Fig3.6 Meshing of MODEL-III

3.2.3 Fluent setup

The first step is to define gravity in negative y-direction. The flow occurs along the x direction. The next step is to select the type of model to be used in the analysis. The software initialize fundamental equation for flowing fluid, conservation of mass and momentum known as Reynolds-averaged navier-stokes equation as well as (k-epsilon) equation for representing turbulence characteristics to determine eddy viscosity by finite volume method.

In RANS, equation consists of Reynolds-Averaged continuity equation and incompressible Navier-Stokes equation:

$$.\frac{\partial\langle u_i\rangle}{\partial x_i} = 0 \qquad \text{............Eq.3.1}$$

$$\frac{\partial\langle u_i\rangle}{\partial_t} + \langle u_i\rangle\frac{\partial\langle u_i\rangle}{\partial x_j} = -\frac{1}{p}\frac{\partial\langle p\rangle}{\partial x_i} + v\frac{\partial^2\langle u_i\rangle}{\partial x_j^2} - \frac{1}{p}\frac{\partial\langle u_i' u_j'\rangle}{\partial x_j} + f_i \qquad \text{............Eq.3.2}$$

Where bracket denotes Reynolds-averaging, vector u_i is velocity, vector x_i is position, t is time, p is pressure, ∂ is density and v=kinematic viscosity.

The turbulence kinetic energy (k) and its dissipation rate (ε) are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial t}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \qquad \text{..Eq.3.3}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial t}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial k}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \text{ ..Eq.3.4}$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}, C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε respectively. S_k and S_ε are user-defined source terms.

The turbulent (or eddy) viscosity, μ_t, is computed by combining k and ε as :

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \text{…………Eq.3.5}$$

Where C_μ is a constant.

The model constants $C_{1\varepsilon}, C_{2\varepsilon}$, $C_{3\varepsilon}$, σ_k and σ_ε have the following default values

$$C_{1\varepsilon} = 1.44,, C_{2\varepsilon} = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3$$

These default values have been determined from experiments for fundamental turbulent flows including frequently encountered shear flows like boundary layers, mixing layers and jets as well as for decaying isotropic grid turbulence.

Materials: Next step in Fluent is to define materials. The cell zone condition for the Model Chamber has to solid and for model space is liquid.

Boundary conditions: the boundary conditions are as follows:

Inlet- The inlet was taken as velocity-inlet. For each discharge value, velocity is calculated by dividing it with the area of inlet. Velocities taken are .07, .14 and .35 cm/s.

Outlet- The outlet was taken as pressure-outlet.

Free surface- It was taken as symmetry condition.

Bed and side walls- The Model surface, standing and hanging baffles are all set to non-slip surfaces. All surfaces are considered to be hydro dynamically smooth.

<u>Solution:</u> The next step is to initialize the solution. Then no. of iterations is set to 250 and the solution is calculated until it is **converged.** It is necessary for the solution to get converged. The no. of iterations required were always less than 120. The finer the mesh, the more time it takes to calculate the solution. After the analysis, the results can be viewed under the CFX-POST which is embedded in ANSYS workbench. Figures showing iterations value for solution to be converged of Model-I, II and III.

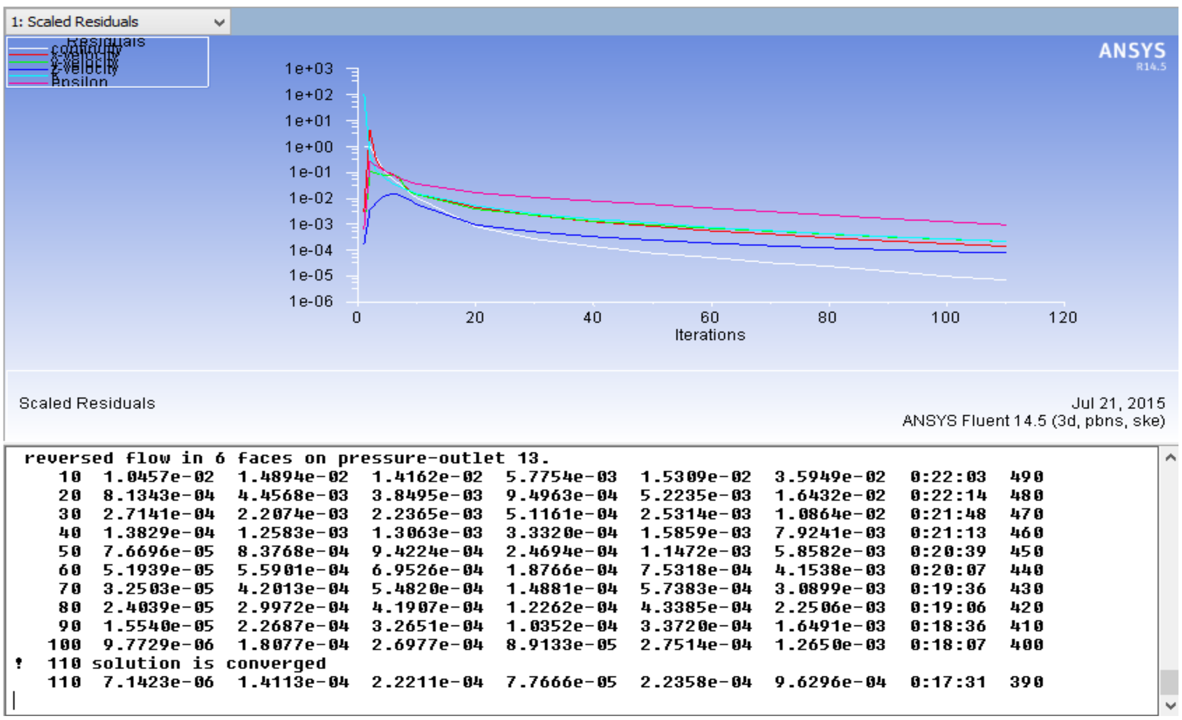


Fig.3.7 Window in Fluent after solution convergence of MODEL-I

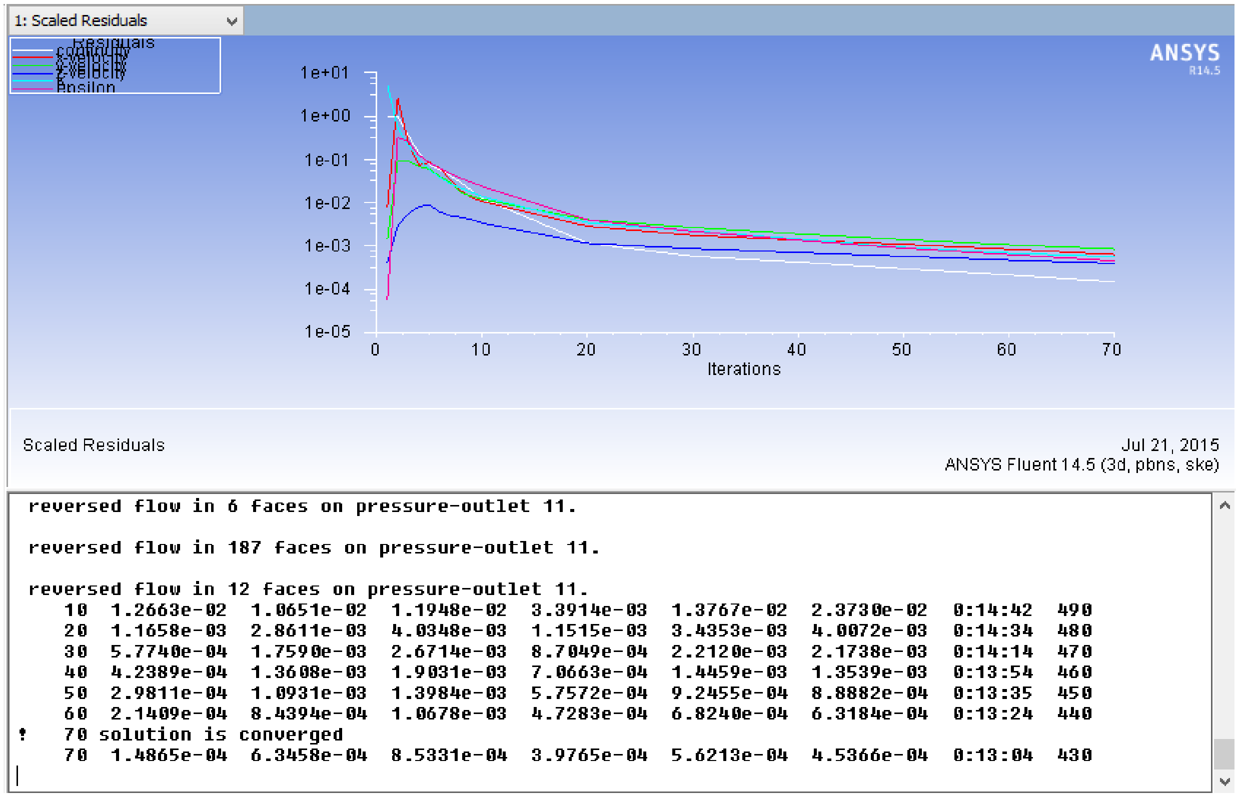


Fig.3.8 Window in Fluent after solution convergence of MODEL-II

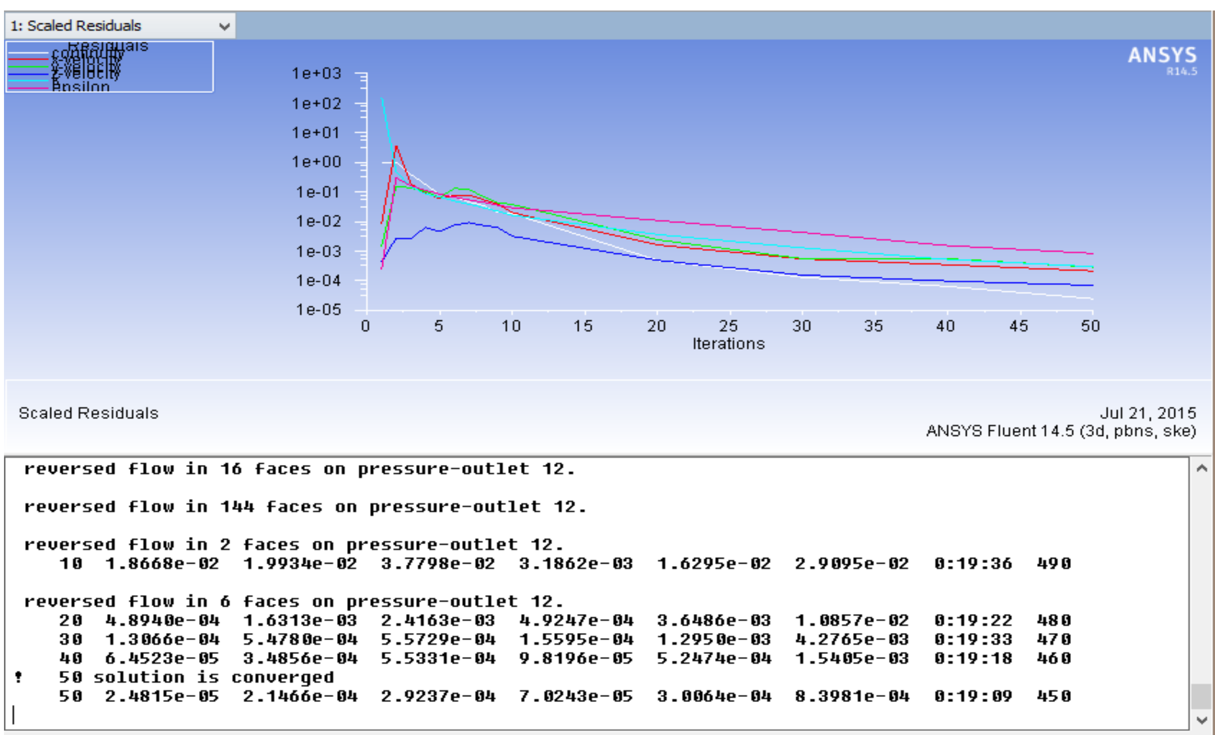


Fig.3.9 Window in Fluent after solution convergence of MODEL-III

Chapter-4

NUMERICAL DATA

This chapter contains the data obtained from the Fluent after calculation is done. There are total number of 9 cases are made to run which comprises of three models having different chamber configuration which are analysed by three different discharge of value **10**, **20** and **50** litres per hour. In each case, the velocity **near bottom, at mid depth and near surface** are obtained using point technique in every chamber.

The following table contains the values of velocity calculated from three different discharges of three different models at three locations i.e. near bottom, at mid depth and near surface from the fluent software. The velocities are taken along x-direction keeping y-direction changing for different levels and z component to be constant in mid means half of model depth which is 10 cm.

Table 4.1: Discharge of 10 litres per hour in MODEL-I (Four Chambers)

For DISCHARGE 10 litres per hour		Distance in cm x-direction	Velocity in cm/s Near Bottom	Mid Depth	Near Surface
MODEL-I (Four Chambers)	Chamber-1	0	0	0	0
		2	0.01867	0.05789	0.02874
		4	0.03892	0.01143	0.01144
		6	0.05753	0.02991	0.01616
		8	0.05925	0.02739	0.02421
		10	0	0	0
	Chamber-2	10	0	0	0
		12	0.01536	0.06211	0.02258
		14	0.03365	0.02081	0.01078
		16	0.05448	0.02947	0.0161
		18	0.05224	0.02108	0.0245
		20	0	0	0
	Chamber-3	20	0	0	0
		22	0.01401	0.06741	0.02399
		24	0.03335	0.01793	0.01424
		26	0.05448	0.02727	0.02399

		28	0.05447	0.03227	0.03003
		30	0	0	0
	Chamber-4	30	0	0	0
		32	0.01114	0.06413	0.0273
		34	0.03047	0.01754	0
		36	0.05186	0.0297	0
		38	0.05987	0.02513	0
		40	0	0	0

Table 4.2: Discharge of 20 litres per hour in MODEL-I (Four Chambers)

For DISCHARGE 20 litres per hour		Distance in cm x-direction	Velocity in cm/s Near Bottom	Mid Depth	Near Surface
MODEL-I (Four Chambers)	Chamber-1	0	0	0	0
		2	0.01998	0.1154	0.05782
		4	0.04964	0.02193	0.01076
		6	0.1039	0.05841	0.03064
		8	0.1633	0.05546	0.04617
		10	0	0	0
	Chamber-2	10	0	0	0
		12	0.01575	0.1234	0.04414
		14	0.04645	0.01495	0.03296
		16	0.09456	0.0569	0.04992
		18	0.1665	0.05902	0.05934
		20	0	0	0
	Chamber-3	20	0	0	0
		22	0.01384	0.1336	0.04661
		24	0.0428	0.03355	0.02786
		26	0.09023	0.06322	0.04374
		28	0.168	0.02071	0.05674
		30	0	0	0
	Chamber-4	30	0	0	0
		32	0.01012	0.1275	0.0528
		34	0.03646	0.03534	0
		36	0.08073	0.05532	0
		38	0.1646	0.04812	0
		40	0	0	0

Table 4.3: Discharge of 50 litres per hour in MODEL-I (Four Chambers)

For DISCHARGE 50 litres per hour		Distance in cm x-direction	Velocity in cm/s Near Bottom	Mid Depth	Near Surface
MODEL-I (Four Chambers)	Chamber-1	0	0	0	0
		2	0.05056	0.2857	0.1459
		4	0.1217	0.05396	0.02749
		6	0.25	0.1407	0.06959
		8	0.39	0.1392	0.1055
		10	0	0	0.1366
	Chamber-2	10	0	0	0
		12	0.0399	0.3055	0.1014
		14	0.1173	0.04079	0.03458
		16	0.2316	0.1219	0.06593
		18	0.4023	0.1289	0.1029
		20	0	0	0.1272
	Chamber-3	20	0	0	0
		22	0.03439	0.3288	0.105
		24	0.1068	0.05996	0.04154
		26	0.2204	0.1183	0.06325
		28	0.404	0.1137	0.1018
		30	0	0	0
	Chamber-4	30	0	0	0
		32	0	0.1392	0.1193
		34	0.02421	0.1289	0.01513
		36	0.09062	0.1545	0.02037
		38	0.1983	0.119	0.1667
		40	0.3954	0	0

Table 4.4: Discharge of 10 litres per hour in MODEL-II (Five Chambers)

For DISCHARGE 10 litres per hour		Distance in cm x-direction	Velocity in cm/s Near Bottom	Mid Depth	Near Surface
MODEL-II (Five Chambers)	Chamber-1	0	0	0	0
		1.6	0.01924	0.05877	0.6101
		3.2	0.04323	0.02178	0.01025
		4.8	0.08906	0.03341	0.01425
		6.4	0.1625	0.02933	0.01878
		8	0	0	0
	Chamber-2	8	0	0	0
		9.6	0.01236	0.07918	0.01473
		11.2	0.03761	0.01296	0.01071
		12.8	0.07797	0.02961	0.01675
		14.4	0.1348	0.03533	0.02117
		16	0	0	0
	Chamber-3	16	0	0	0
		17.6	0.01042	0.09025	0.02335
		19.2	0.0316	0.01465	0.1982
		20.8	0.06686	0.02758	0.01425
		22.4	0.1402	0.03242	0.01758
		24	0	0	0
	Chamber-4	24	0	0	0
		25.6	0.01092	0.08595	0.01968
		27.2	0.02777	0.01032	0.01293
		28.8	0.06016	0.09757	0.01311
		30.4	0.1402	0.02789	0.01761
		32	0	0	0
	Chamber-5	32	0	0	0
		33.6	0.0143	0.06359	0.02091
		35.2	0.02399	0.01574	0.2108
		36.8	0.05739	0.02693	0
		38.4	0.1155	0.02975	0
		40	0	0	0

Table 4.5: Discharge of20 litres per hour in MODEL-II (Five Chambers)

For DISCHARGE 20 litres per hour		Distance in cm x-direction	Velocity in cm/s Near Bottom	Mid Depth	Near Surface
MODEL-II (Five Chambers)	Chamber-1	0	0	0	0
		1.6	0.03793	0.1178	0.1255
		3.2	0.08521	0.04187	0.01161
		4.8	0.1672	0.06511	0.025
		6.4	0.2599	0.5786	0.03188
		8	0	0	0
	Chamber-2	8	0	0	0
		9.6	0.02237	0.1587	0.0397
		11.2	0.07297	0.02112	0.01434
		12.8	0.1462	0.06228	0.02907
		14.4	0.2735	0.0766	0.03673
		16	0	0	0
	Chamber-3	16	0	0	0
		17.6	0.01659	0.1778	0.04377
		19.2	0.06255	0.01738	0.0158
		20.8	0.1269	0.06283	0.246
		22.4	0.2773	0.07171	0.0336
		24	0	0	0
	Chamber-4	24	0	0	0
		25.6	0.01062	0.169	0.03978
		27.2	0.05373	0.01999	0.04064
		28.8	0.1154	0.05759	0.01293
		30.4	0.2527	0.07297	0.0348
		32	0	0	0
	Chamber-5	32	0	0	0
		33.6	0.04575	0.1529	0.0421
		35.2	0.1095	0.02409	0.04555
		36.8	0.2154	0.0616	0.01862
		38.4	0.2299	0.06016	0.02144
		40	0.1155	0	0

Table 4.6: Discharge of 50 litres per hour in MODEL-II (Five Chambers)

For DISCHARGE 50 litres per hour		Distance in cm x-direction	Velocity in cm/s Near Bottom	Mid Depth	Near Surface
MODEL-II (Five Chambers)	Chamber-1	0	0	0	0
		1.6	0.07543	0.2962	0.3238
		3.2	0.2009	0.04588	0.01824
		4.8	0.3873	0.1281	0.05337
		6.4	0.6509	0.2791	0.07997
		8	0	0	0
	Chamber-2	8	0	0	0
		9.6	0.4137	0.3968	0.07974
		11.2	0.1719	0.05898	0.01375
		12.8	0.3417	0.1131	0.05208
		14.4	0.674	0.2742	0.07544
		16	0	0	0
	Chamber-3	16	0	0	0
		17.6	0.04017	0.4336	0.09548
		19.2	0.1535	0.02462	0.02386
		20.8	0.3062	0.1004	0.04981
		22.4	0.6564	0.2496	0.07186
		24	0	0	0
	Chamber-4	24	0	0	0
		25.6	0.03229	0.4108	0.09546
		27.2	0.1215	0.04607	0.01578
		28.8	0.2757	0.0725	0.03952
		30.4	0.5844	0.2271	0.06764
		32	0	0	0.8163
	Chamber-5	32	0	0	0
		33.6	0.01979	0.3697	0.1027
		35.2	0.1257	0.08714	0.0116
		36.8	0.2811	0.09817	0.03131
		38.4	0.5173	0.2392	0.4858
		40	0	0	0

Table 4.7: Discharge of 10 litres per hour in MODEL-III (Six Chambers)

For DISCHARGE 10 litres per hour		Distance in cm x-direction	Velocity in cm/s Near Bottom	Mid Depth	Near Surface
MODEL-III (Six Chambers)	Chamber-1	0	0	0	0
		1.3	0.02191	0.08426	0.0823
		2.6	0.0514	0.0141	0.01396
		3.9	0.1028	0.04705	0.01661
		5.2	0.09501	0.04312	0.02129
		6.5	0	0	0
	Chamber-2	6.5	0	0	0
		7.8	0.01383	0.09638	0.01879
		9.1	0.04152	0.02044	0.01162
		10.4	0.08762	0.04461	0.02182
		11.7	0.01935	0.4696	0.02387
		13	0	0	0
	Chamber-3	14.3	0	0	0
		15.6	0.01018	0.0889	0.01982
		16.9	0.03402	0.01	0.01272
		18.2	0.07782	0.03787	0.02073
		19.5	0.11	0.04791	0.02422
		20.8	0	0	0
	Chamber-4	20.8	0	0	0
		22.1	0.01544	0.07799	0.02216
		23.4	0.04433	0.06779	0.01083
		24.7	0.09665	0.02414	0.02002
		26	0.03786	0.04888	0.2468
		27.3	0	0	0
	Chamber-5	27.3	0	0	0
		28.6	0.01137	0.07438	0.0241
		29.9	0.03824	0.0843	0.01398
		31.2	0.0856	0.01408	0.01888
		32.5	0.0103	0.4674	0.024
		33.8	0	0	0

	Chamber-6	33.8	0	0	0
		35.1	0.01715	0.03701	0.02488
		36.4	0.03129	0.09651	0.01359
		37.7	0.07387	0.01053	0
		39.0	0.1213	0.02592	0
		40.3	0	0	0

Table 4.8: Discharge of 20 litres per hour in MODEL-III (Six Chambers)

For DISCHARGE 20 litres per hour		Distance in cm	Velocity in cm/s		
		x-direction	Near Bottom	Mid Depth	Near Surface
MODEL-III (Six Chambers)	Chamber-1	0	0	0	0
		1.3	0.04397	0.168	0.1653
		2.6	0.1017	0.02844	0.01312
		3.9	0.2027	0.09387	0.03098
		5.2	0.1878	0.08587	0.04008
		6.5	0	0	0
	Chamber-2	6.5	0	0	0
		7.8	0.02772	0.1914	0.0364
		9.1	0.08257	0.04138	0.01081
		10.4	0.1728	0.08941	0.02904
		11.7	0.227	0.09373	0.04273
		13	0	0	0
	Chamber-3	14.3	0	0	0
		15.6	0.02036	0.177	0.03919
		16.9	0.06769	0.02024	0.01759
		18.2	0.153	0.08056	0.04102
		19.5	0.2252	0.09589	0.04758
		20.8	0	0	0
	Chamber-4	20.8	0	0	0
		22.1	0.01279	0.1724	0.04376
		23.4	0.05707	0.06633	0.0211
		24.7	0.1376	0.1004	0.03976
		26	0.2404	0.06051	0.04857
		27.3	0	0	0

	Chamber-5	27.3	0	0	0
		28.6	0.01188	0.1486	0.04753
		29.9	0.0471	0.05275	0.01737
		31.2	0.1186	0.0984	0.03664
		32.5	0.2233	0.07201	0.04678
		33.8	0	0	0
	Chamber-6	33.8	0	0	0
		35.1	0.01067	0.1723	0.04832
		36.4	0.03745	0.0342	0.1153
		37.7	0.0102	0.05993	0.1158
		39.0	0.2049	0.07024	0
		40.3	0	0	0

Table 4.9: Discharge of 50 litres per hour in MODEL-III (Six Chambers)

For DISCHARGE 50 litres per hour		Distance in cm	Velocity in cm/s		
		x-direction	Near Bottom	Mid Depth	Near Surface
MODEL-III (Six Chambers)	Chamber-1	0	0	0	0
		1.3	0.1107	0.481	0.4188
		2.6	0.2486	0.07247	0.01916
		3.9	0.4885	0.2324	0.06185
		5.2	0.4566	0.2106	0.07604
		6.5	0	0	0.08934
	Chamber-2	6.5	0	0	0
		7.8	0.06901	0.4727	0.07984
		9.1	0.203	0.1572	0.02398
		10.4	0.4168	0.2222	0.06678
		11.7	0.5524	0.2296	0.08932
		13	0	0	0
	Chamber-3	14.3	0	0	0
		15.6	0.05046	0.4397	0.08933
		16.9	0.167	0.1483	0.05596
		18.2	0.3681	0.2014	0.09184
		19.5	0.5944	0.2378	0.1026
		20.8	0	0	0

	Chamber-4	20.8 22.1 23.4 24.7 26 27.3	0 0.04543 0.1417 0.3331 0.6056 0	0 0.3884 0.1672 0.2473 0.1506 0	0 0.09887 0.04725 0.08535 0.09989 0
	Chamber-5	27.3 28.6 29.9 31.2 32.5 33.8	0 0.04164 0.1173 0.2885 0.5476 0	0 0.371 0.1497 0.2435 0.1787 0	0 1041 0.03901 0.08345 0.1075 0
	Chamber-6	33.8 35.1 36.4 37.7 39.0 40.3	0 0.4495 0.1334 0.3502 0.5623 0	0 0.4275 0.09915 0.1666 0.1588 0	0 0.1114 0.01472 0.033 0.0448 0

Fluent also gives the pictorial presentation in form of Volume Rendering, Stream Lines, Vectors and Contours. The figures below show these features in Model-I, II and III.

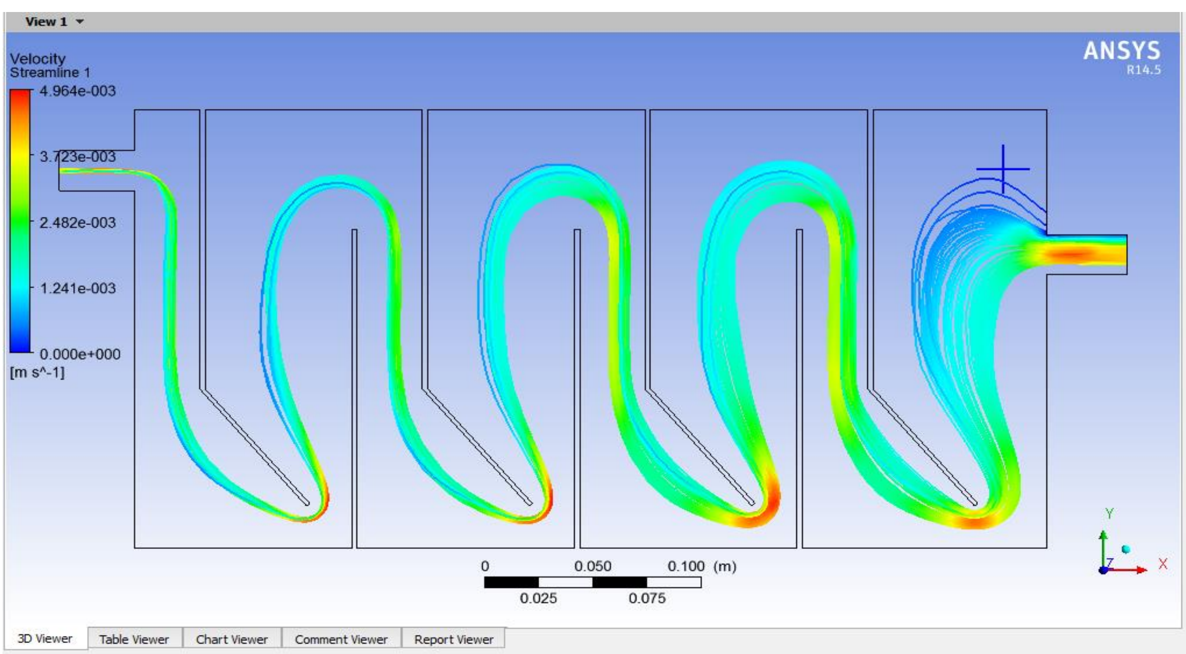


Fig 4.1 Stream lines in MODEL-I

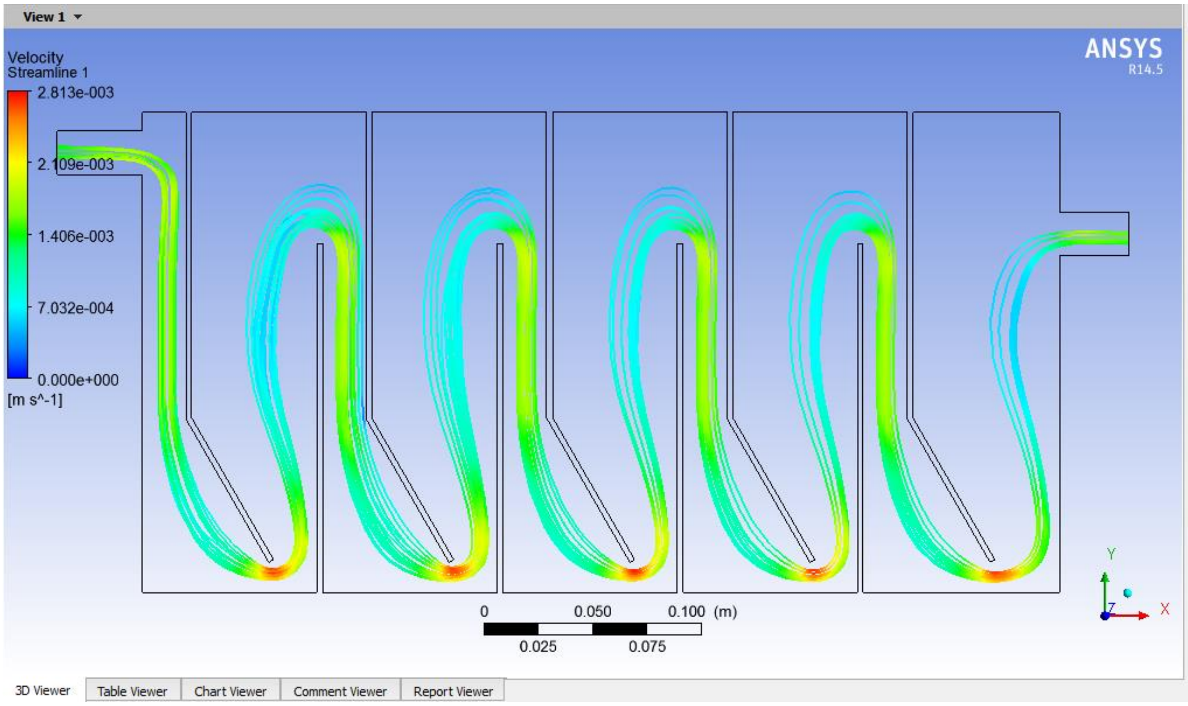


Fig 4.2 Stream lines in MODEL-II

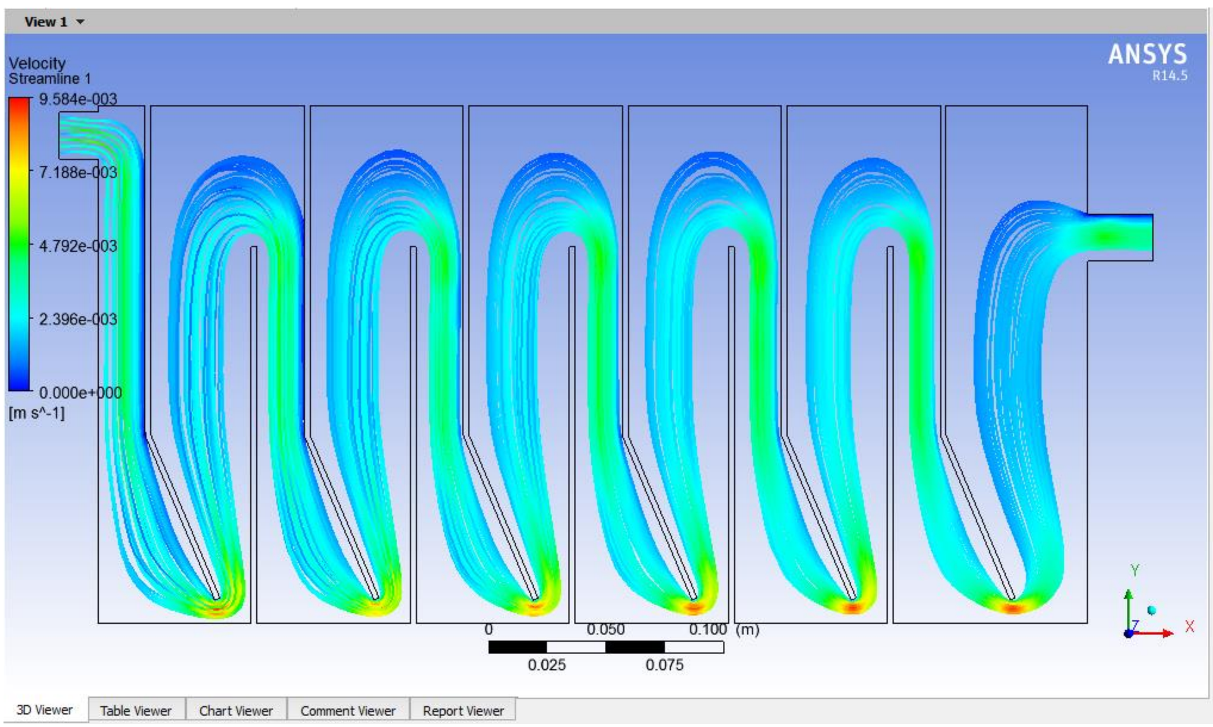


Fig 4.3 Stream lines in MODEL-III

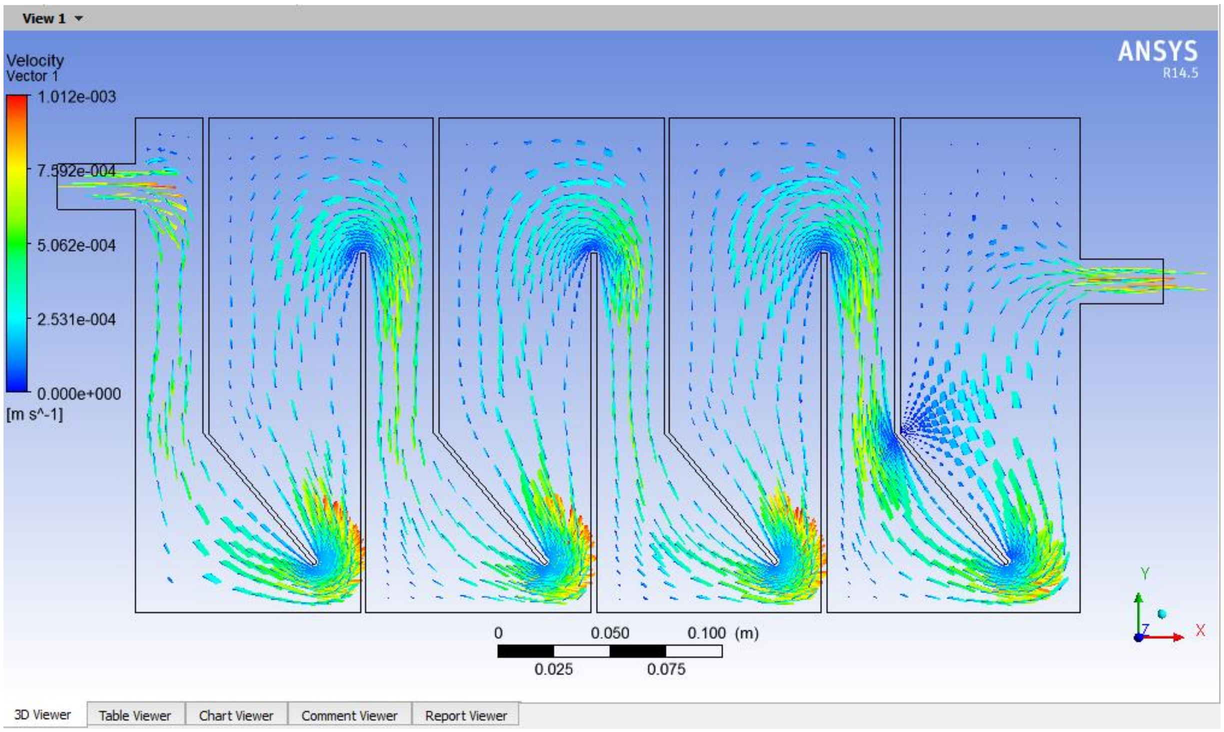


Fig 4.4 Vectors in MODEL-I

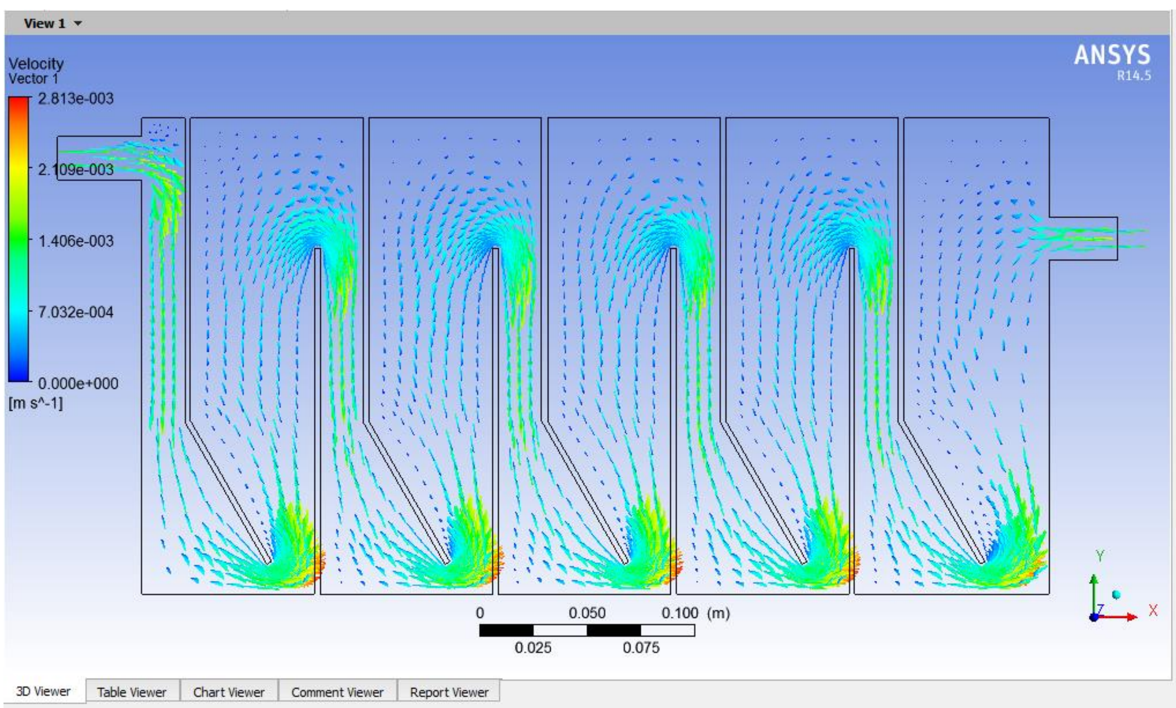


Fig 4.5 Vectors in MODEL-II

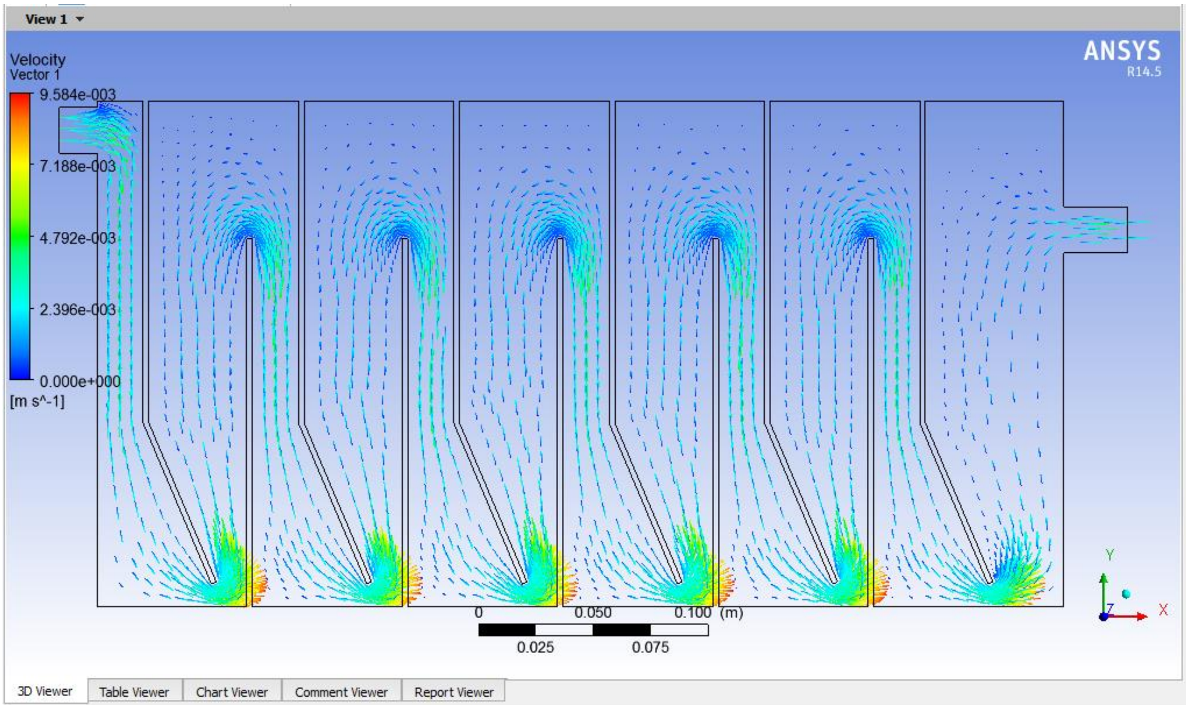


Fig 4.6 Vectors in MODEL-III

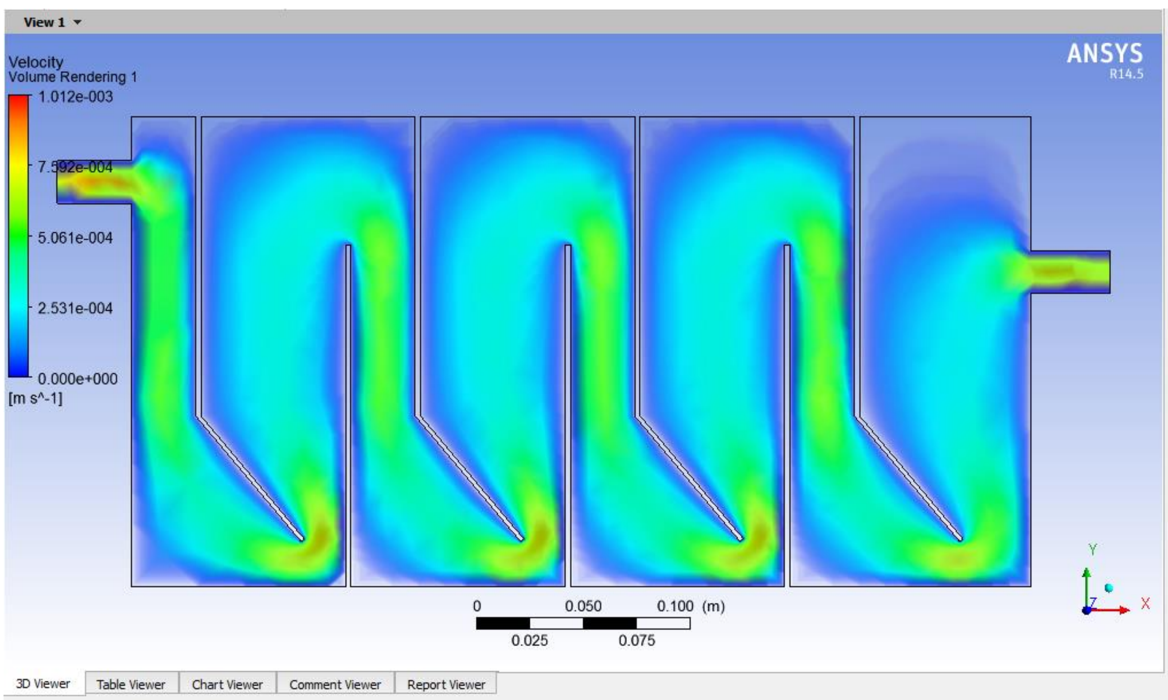


Fig 4.7 Volume Rendering in MODEL-I

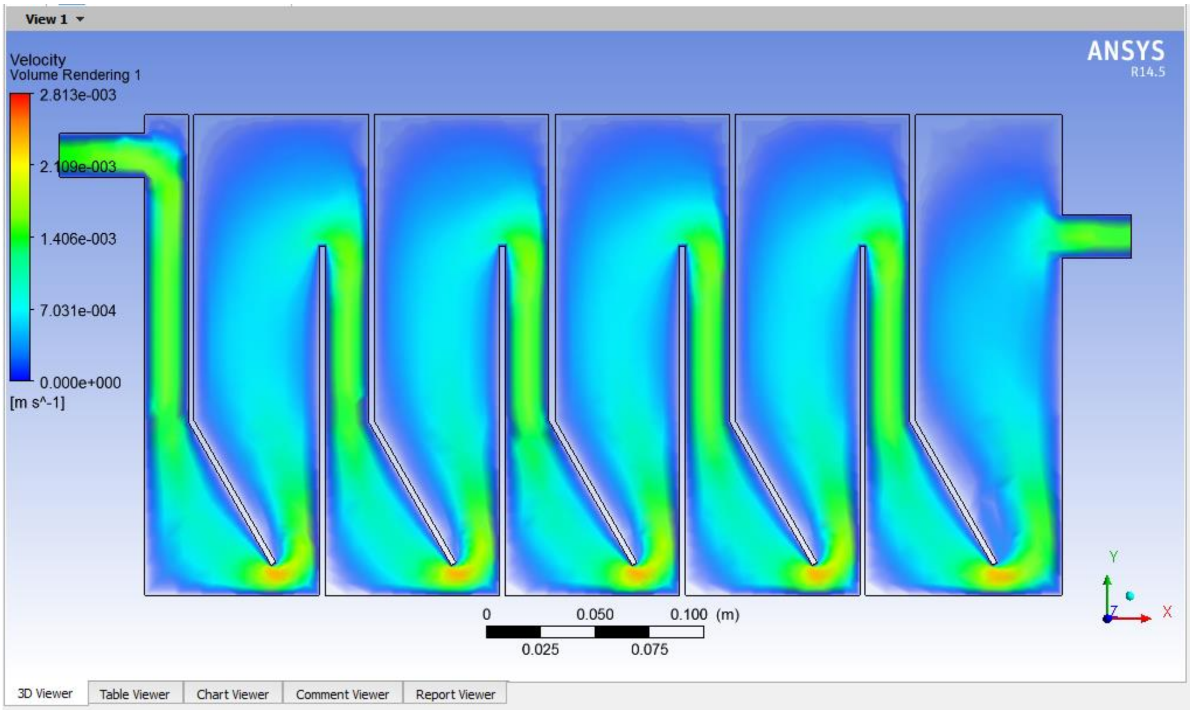


Fig 4.8 Volume Rendering in MODEL-II

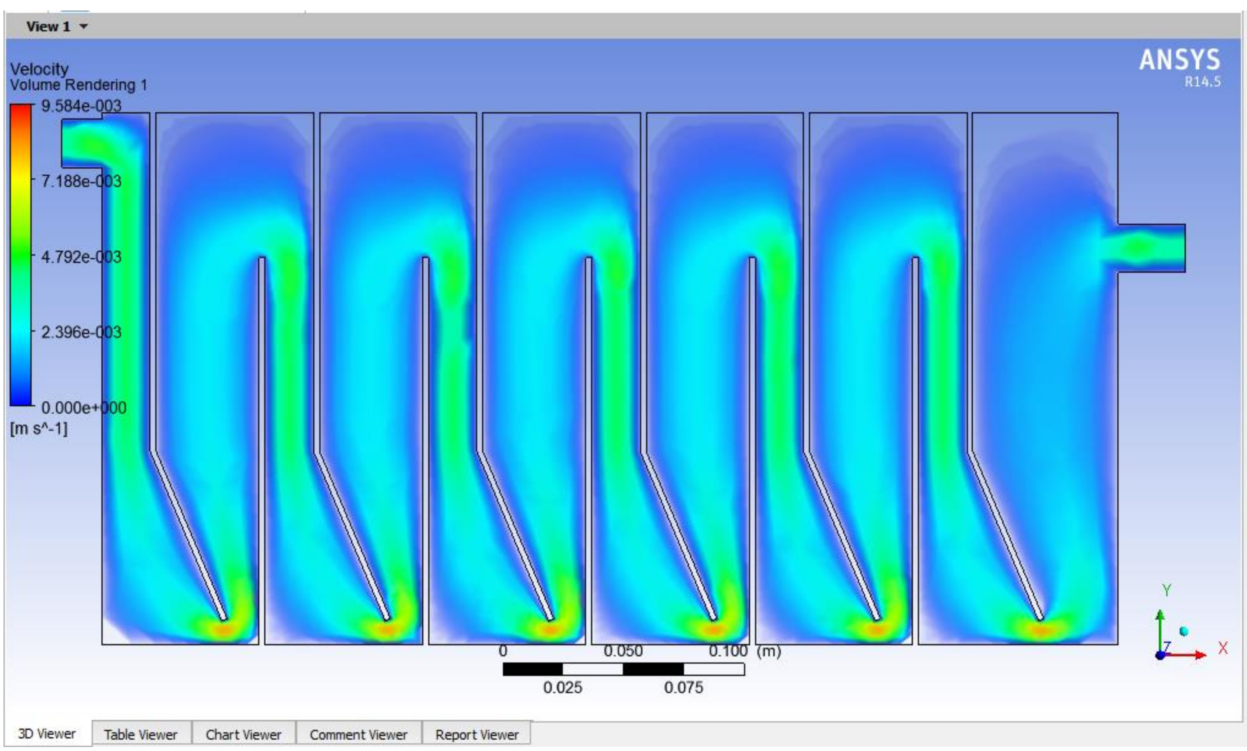


Fig 4.9 Volume Rendering in MODEL-III

Chapter-5

RESULTS AND DISCUSSIONS

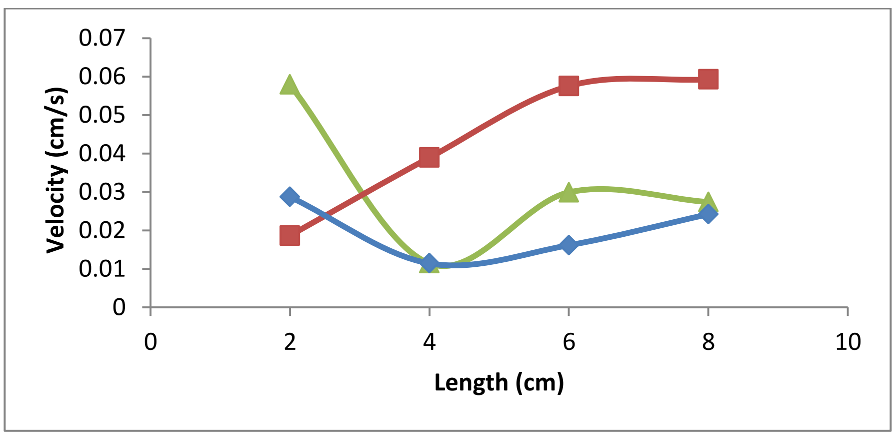
Results

Graphs are plotted on basis of velocity obtained in MODEL I, II and III at near bottom, mid depth and near surface. For each Chamber in a model the velocity of all three locations are merged on a single graph. Figures below are showing graphs, which are taken at three different discharge.

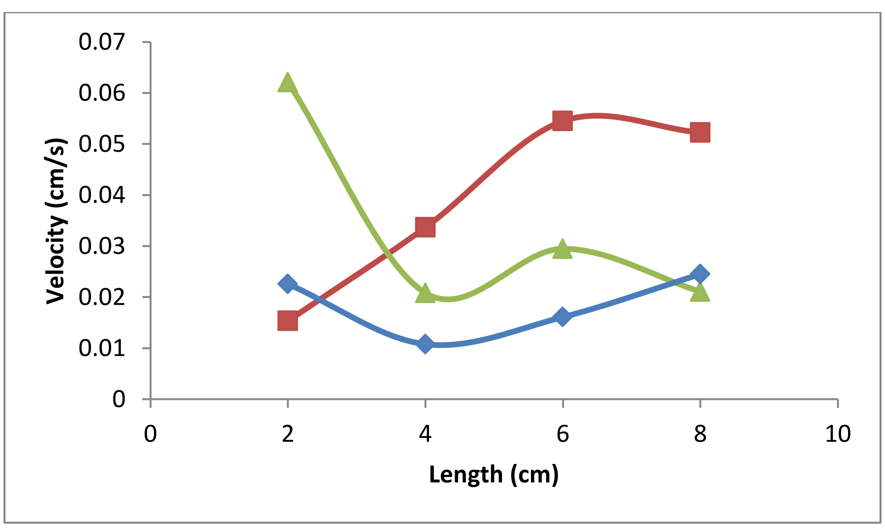
1. For Discharge 10 litres per hour

   1.1 MODEL-I

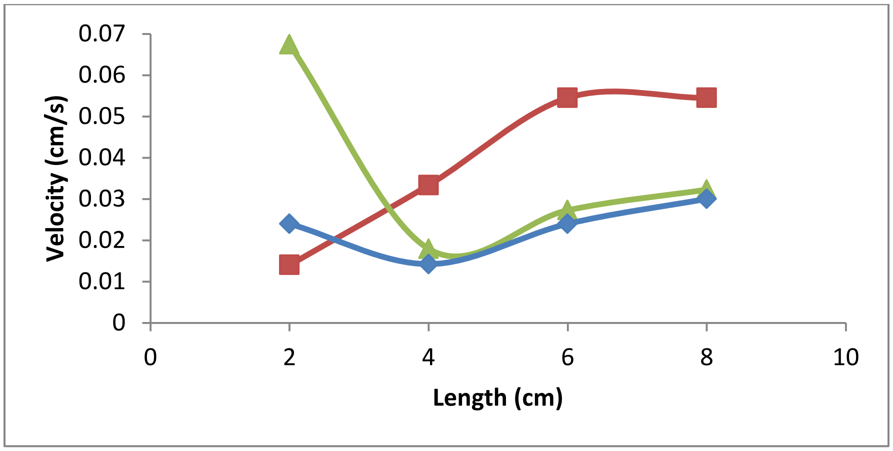
   (a) First Chamber



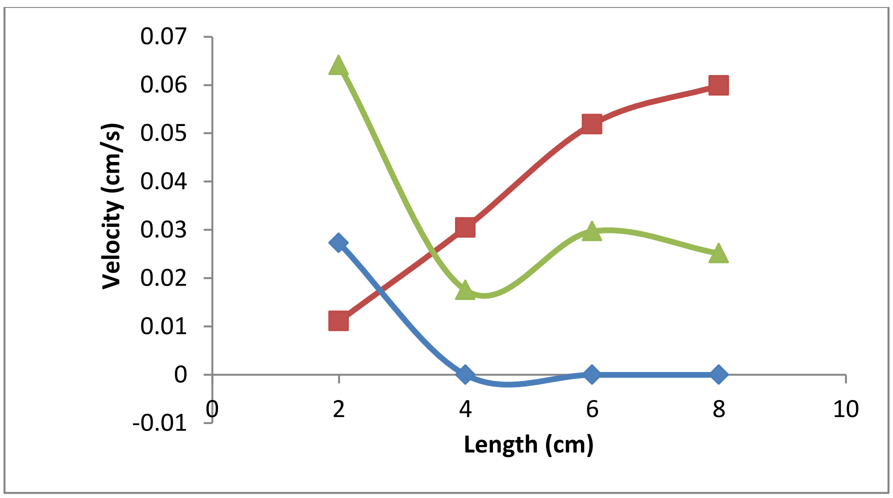
   (b) Second Chamber



(c) Third Chamber

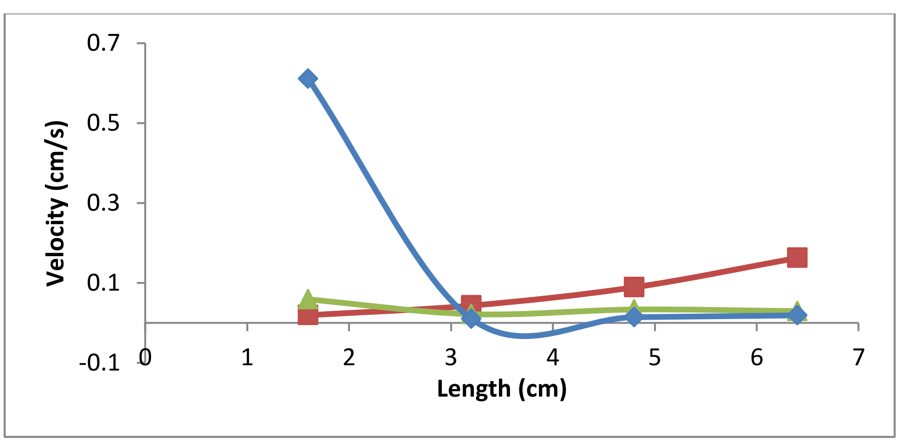


(d) Fourth Chamber

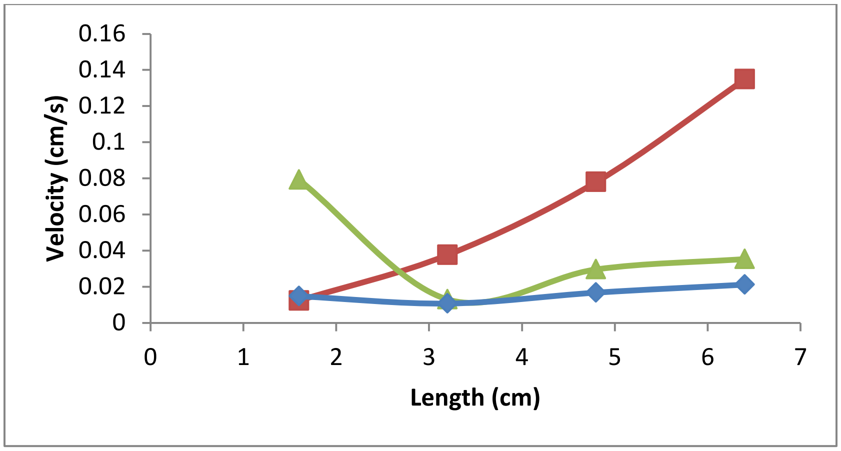


1.2 MODEL-II

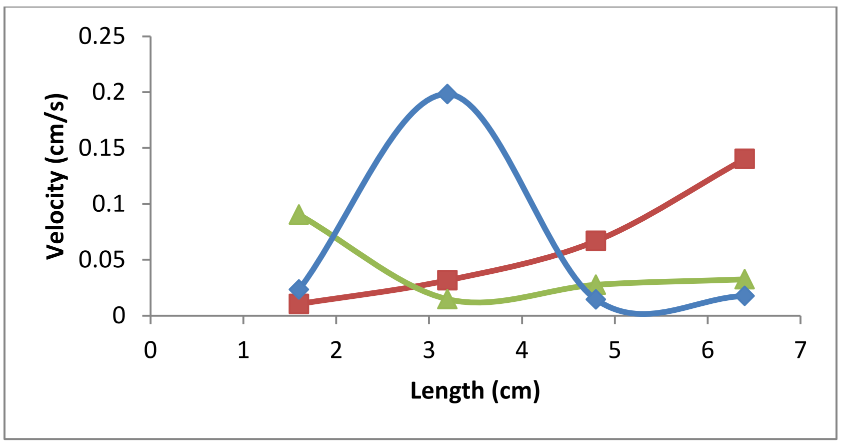
(a) First Chamber



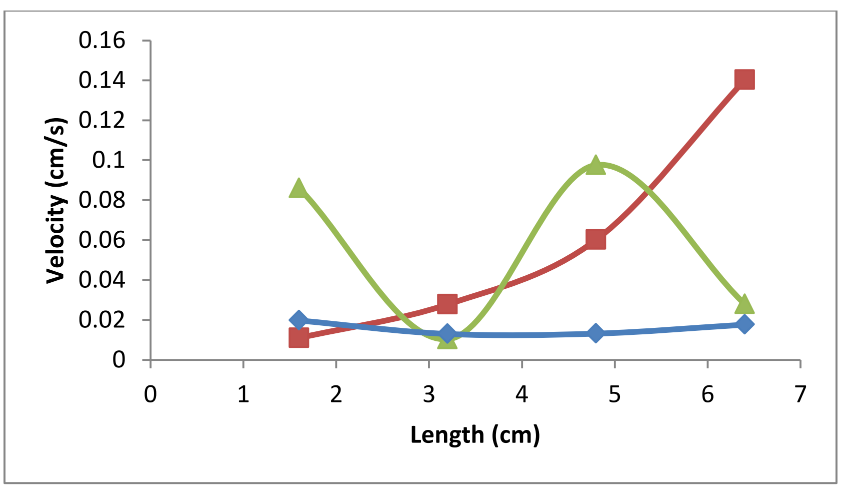
(b) Second Chamber



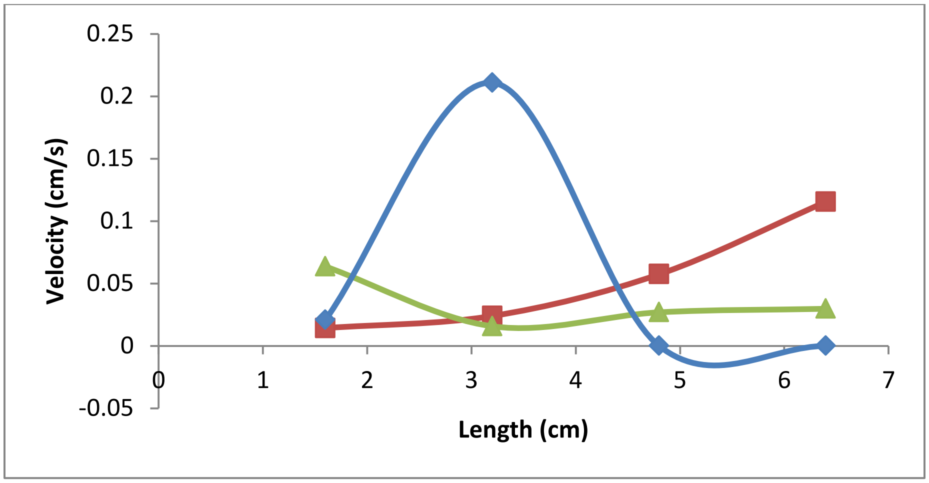
(c) Third Chamber



(d) Fourth Chamber

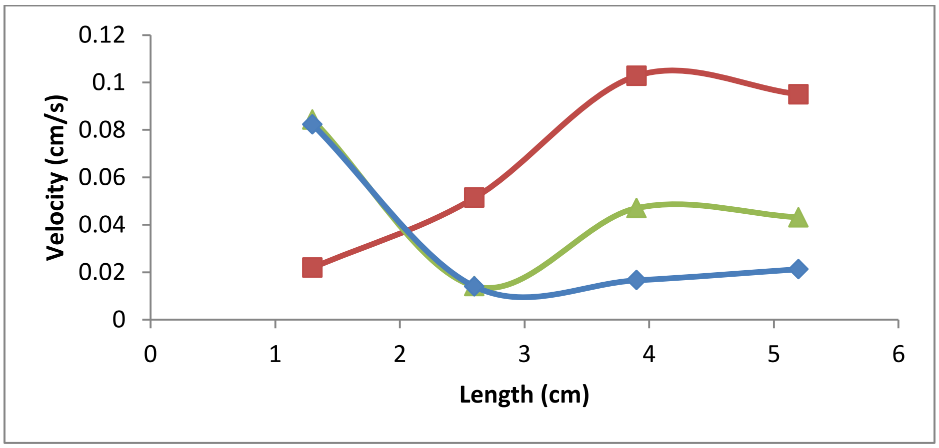


(e) Fifth Chamber

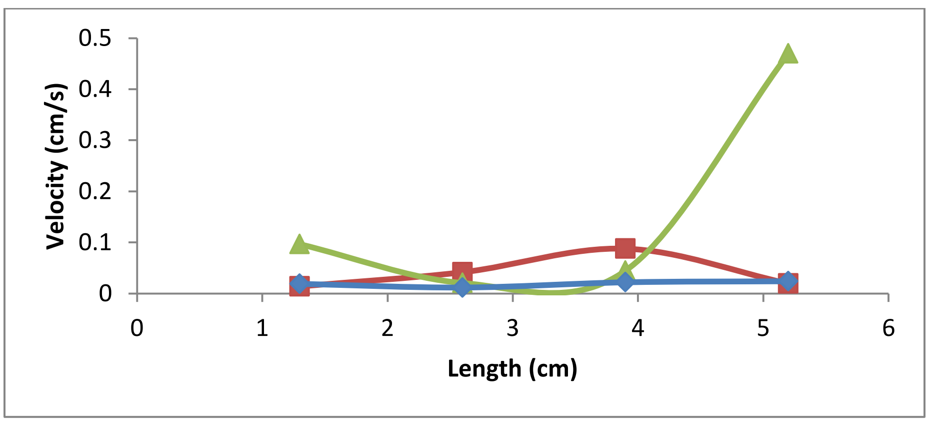


1.3 MODEL-III

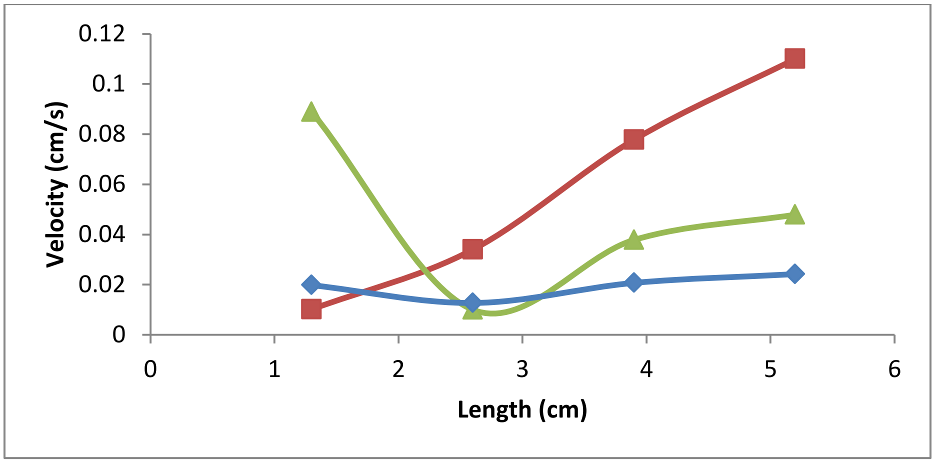
(a) First Chamber



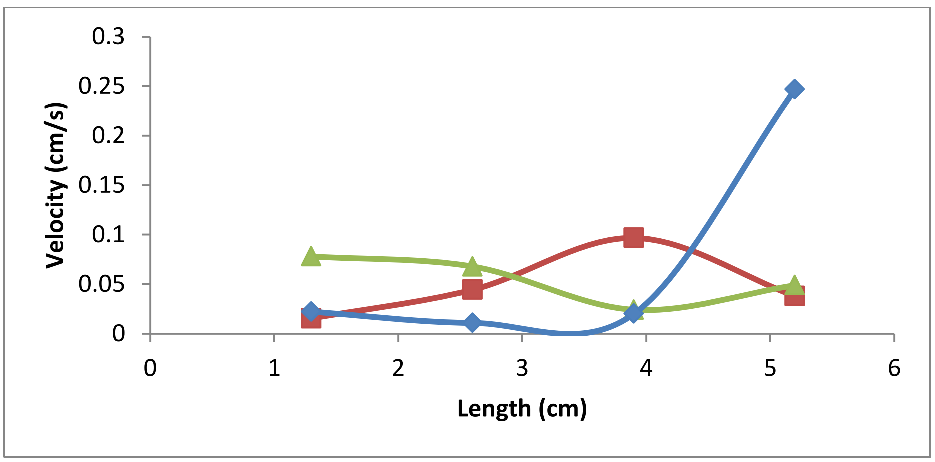
(b) Second Chamber



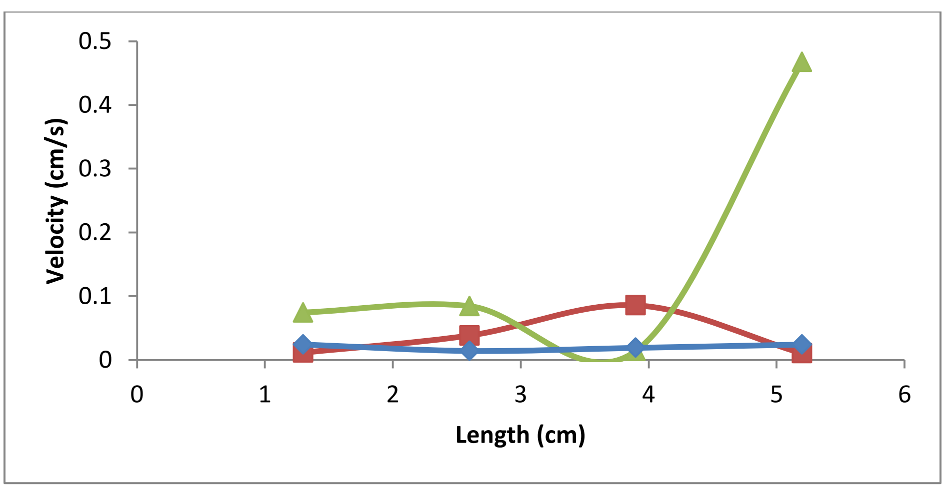
(c) Third Chamber



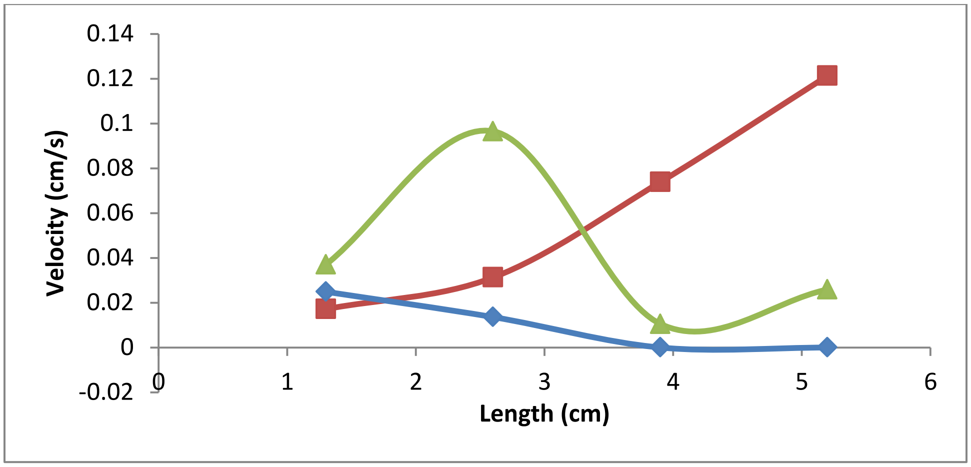
(d) Fourth Chamber



(e) Fifth Chamber



(f) Sixth Chamber



For remaining two Discharges, the entire graphs can be plotted like shown above.

From the graph it is seen that the water incoming from inlet has its certain velocity which is increased when it falls downward and nearby baffle tip it has the maximum velocity. When it moves upwards the velocity is optimum and within the range i.e. 2m/h. Near surface as well as near bottom the velocity is found to be minimum because of the wall effect.

Discussions

Each model I, II and III have 4, 5 and 6 chambers respectively. The volume of every model is of finite nature to a value of $18480 cm^3$. Therefore in increase of number of baffles provided, the chamber width shortens. The model-I is having standing baffles placed at a distance of 10cm. The model-II is having standing baffles placed at a distance of 8cm. The model-III is having standing baffles placed at a distance of 6.5cm. As we move from model I to III, the number of baffles increases therefore the flow get more blended path and hence short circuiting and dead zones decreases.

In Model-I, the dead zones stretch can be seen in corners as well as nearby surface. In Model-II, the dead zones show their effect as stream lines are narrow in nature. In Model-III, the dead zones are seems to be negligible, flow is covering the maximum width of chamber and short circuiting is not present. These characteristics indicate that the Model-III is more efficient in comparison to model I and II. Figures are provided below showing stream lines in each case.

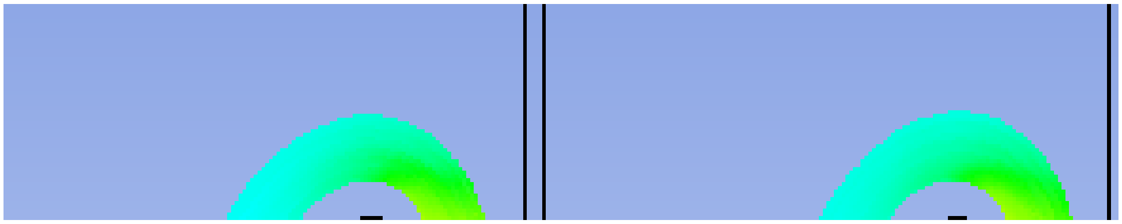


Fig 5.1 Stream lines in Model-I

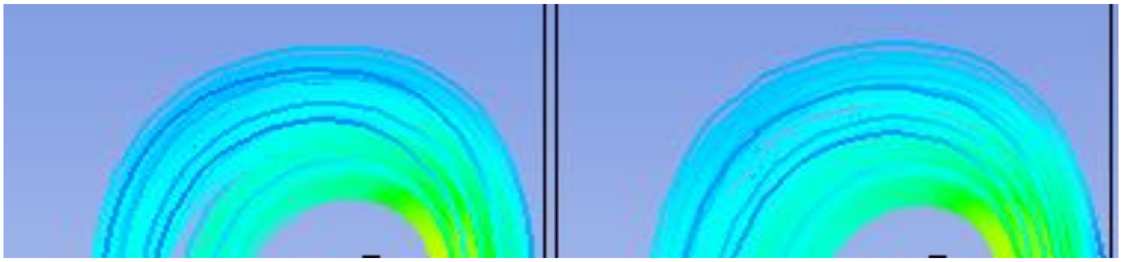


Fig 5.2 Stream lines in Model-II

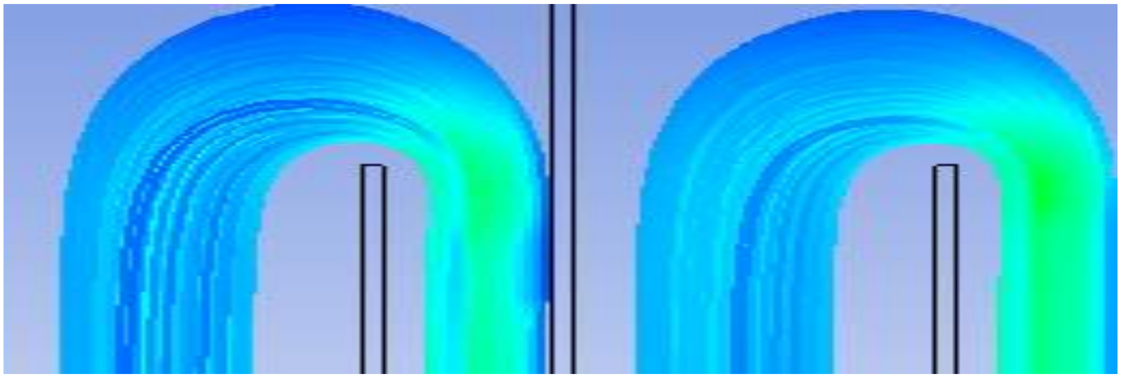


Fig 5.3 Stream lines in Model-III

In case of Velocity vector, the vector lines of Model-I show the dotted lines are dark which implies that the upward velocity to the right of baffle is high which is not good for the biological reactions taking place and the upward velocity should be less than the settling velocity of solids.

The vector lines of Model-II show the lines are slightly lighter than the Model-I and upward velocity is more which can create turbulence.

The vector lines in Model-III are seen to be very light in nature which implies that this kind of upward velocity lines are preferred which is less than the settling velocity of solids but will not create turbulence and take solids with them. Figures showing velocity vector lines in chamber of each model I, II and III.

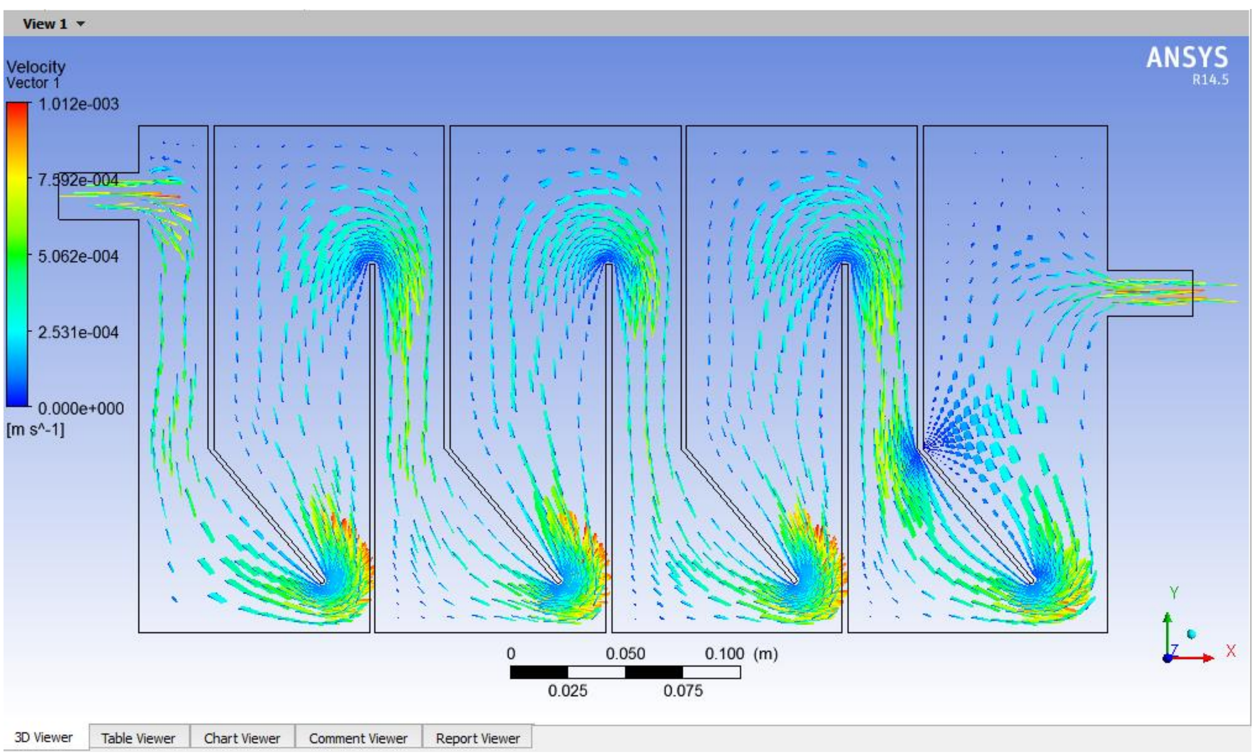


Fig 5.4 Velocity Vector in Model-I

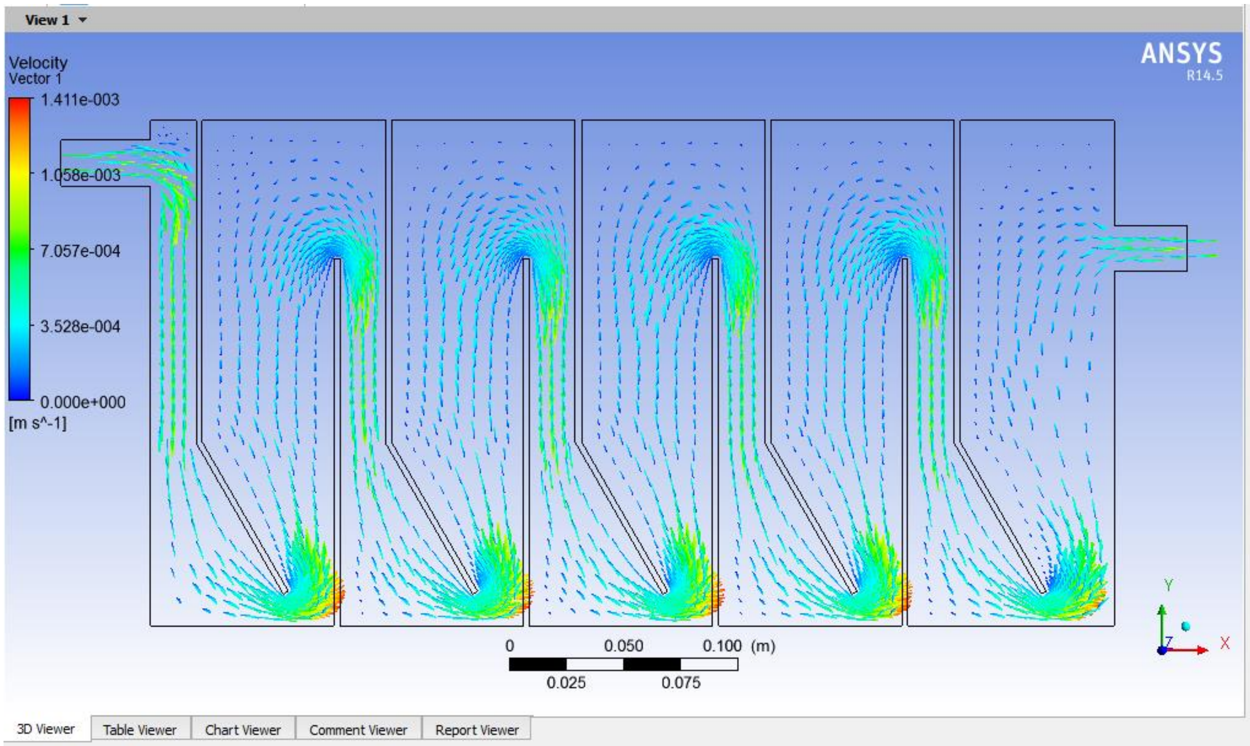


Fig 5.5 Velocity Vector in Model-II

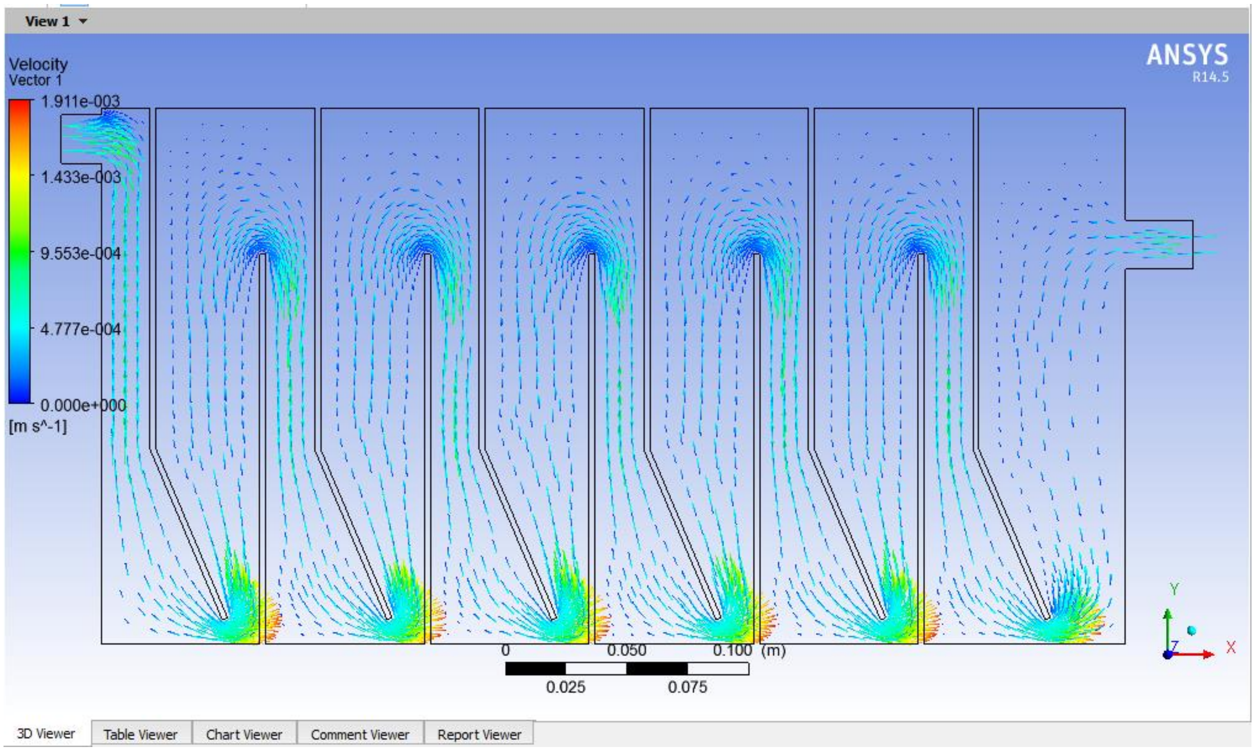


Fig 5.6 Velocity Vector in Model-III

Chapter-6

CONCLUSIONS

1. Baffle cut or placement of baffles plays vital role in the flow of fluid. Small and large must be avoided as they can create eddy formation in corners. Angled baffle gives maximum output when placed at 45^0.
2. The flow patterns observed in each Model I, II and III. By increasing number of baffles, the fluid flow starts flowing in confined manner.
3. The short circuiting is seen if the baffles are placed too far apart from each other. So to avoid dead zones, Stream lines should flow covering the whole volume of chamber.
4. The most governing point in designing of any chamber, reactor etc. is that the upward velocity should be limited so that the biological reactions or contact takes place without wash out. In Model-III, the upward velocity is efficient which can be seen in table 4.7, 4.8 and 4.9.
5. With the help of ANSYS Fluent, it provides platform for hassle free designing. The Boundary conditions plays important role as model accuracy is based on it.
6. In previous experiments, the velocity graphs are made along depth of flow which gives results in linear expression. In this study, the graphs are plotted in the direction of flow therefore the change in velocity is sudden due to placement of hanging baffle or wall etc. so the graphs are not linear in nature.

Future scope of study

In further studies, plug flow that is real 3 phase flow (solid, liquid and gas) can be considered to achieve better results. The baffles used in Models can be used of perforated type with the corners having circular edges. Baffling performance may be improved by using different materials and increasing/decreasing the length of angled baffle. Tracer study and RTD (Residence Time Distribution) of tracer can be done. Various models in ANSYS Fluent such as LES model etc. can be used.

REFRENCES

[1] Khan, Liaqat A., Edward A. Wicklein, and E. C. Teixeira. "Validation of a three-dimensional computational fluid dynamics model of a contact tank." *Journal of Hydraulic Engineering* 132.7 (2006): 741-746.

[2] Zhang, J., A. E. Tejada-Martínez, and Q. Zhang. "Hydraulic Efficiency in RANS of the Flow in Multichambered Contactors." *Journal of Hydraulic Engineering*139.11 (2013): 1150-1157.

[3] Dama, P., et al. "Computational fluid dynamics: application to the design of the anaerobic baffled reactor." *WISA 2000 Biennial Conference, Sun City*. South Africa: Presented at the WISA 2000 Biennial Conference, Sun City, 2000.

[4] Zhang, Jun-Mei, et al. "Effects of baffle configurations on the performance of a potable water service reservoir." *Journal of Environmental Engineering* 138.5 (2011): 578-587.

[5] Peplinski, Daniel Kevin, and Joel J. Ducoste. "Modeling of disinfection contactor hydraulics under uncertainty." *Journal of environmental engineering*128.11 (2002): 1056-1067.

[6] Kim, Dongjin, et al. "Large eddy simulation of flow and tracer transport in multichamber ozone contactors." *Journal of Environmental Engineering* 136.1 (2009): 22-31.

[7] Anderson J D Jr; (1995);Computational Fluid Dynamics: The basics with applications; McGraw- Hill; New York.

[8] Bachmann A, Beard V L and McCarty P L; (1985); Performance characteristics of the anaerobic baffled reactor; *Wat. Sci. & Tech*; **19** (1); 99 - 106.

[9] Barber W and Stuckey D C; (1999) The use of the anaerobic baffled reactor (ABR) for wastewater treatment: A review; *Wat. Research*; **33** (7); 1559 - 1578.

[10] Grobicki A and Stuckey D C; (1992); Hydrodynamic characteristics of the anaerobic baffled reactor; *Wat. Sci. & Tech*; **26** (3); 371 – 378

[11] Thackston, Edward L., F. Douglas Shields Jr, and Paul R. Schroeder. "Residence time distributions of shallow basins." *Journal of Environmental Engineering* (1987).