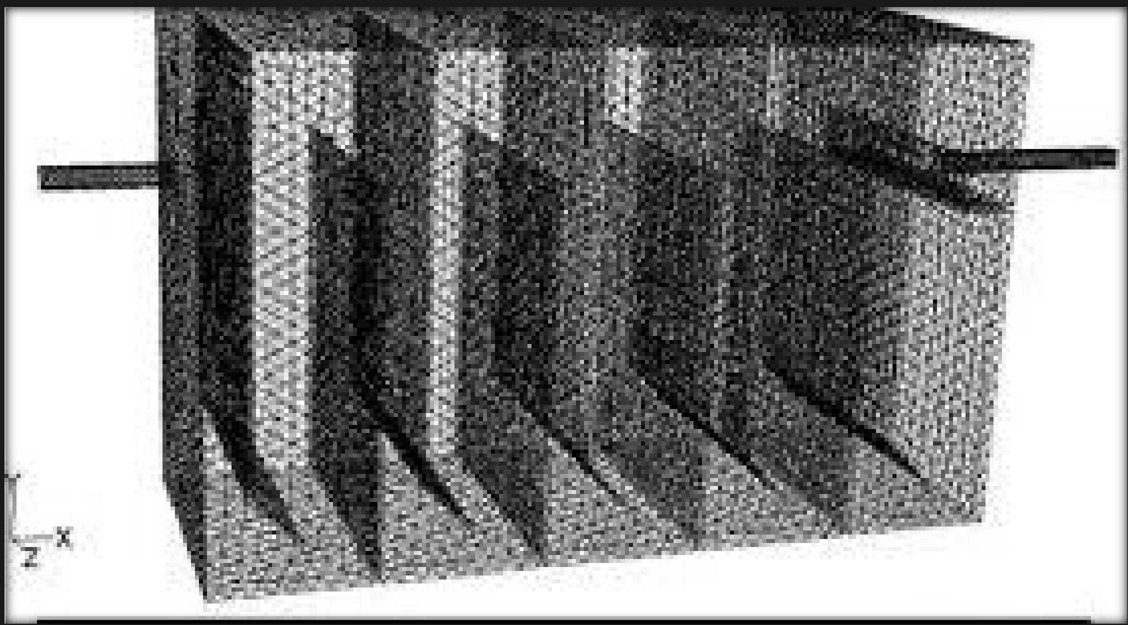
Flow Analysis In Rectangular Baffled Reactor Having Differently Modified Angled Baffles Using ANSYS Fluent (16.0)



Under the guidance of :-

Dr. Rakesh Mehrotra

Associate Professor

Civil Engineering Department

Delhi Technological University, Delhi

Submitted by:-

Anu Gautam

2K14/HFE/02

Civil Engineering Department

Delhi Technological University, Delhi

FLOW ANALYSIS IN RECTANGULAR BAFFLED REACTOR HAVING DIFFERENTLY MODIFIED ANGLED BAFFLES USING ANSYS FLUENT (16.0)

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

MASTER OF TECHNOLOGY

IN

HYDRAULICS & FLOOD ENGINEERING

BY

ANU GAUTAM
(ROLL NO. 2K14/HFE/02)

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-2016

DEPARTMENT OF CIVIL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY, DELHI

CANDIDATE'S DECLARATION

I do hereby declare that the work presented is this thesis entitled **"flow analysis in rectangular baffled reactor having differently modified angled baffles using ANSYS fluent (16.0)"** in the partial fulfilment of the requirement 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 my own work carried out from January 2016 to July 2016 under the supervision of Dr. Rakesh Mehrotra (Associate Professor), Department of Civil Engineering.

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

Anu Gautam

Date:

(2K14/HFE/02)

DEPARTMENT OF CIVIL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY, DELHI

CERTIFICATE

This is to certify that the thesis entitled **"flow analysis in rectangular baffled reactor having differently modified angled baffles using Ansys fluent (16.0)"** is a bonafide record of authentic work carried out by ANU GAUTAM (2K14/HFE/02) under my supervision and guidance for the partial fulfilment of the requirement for the award of Master of Technology degree in Civil Engineering with specialization in HYDRAULICS AND FLOOD ENGINEERING at the DELHI TECHNOLOGICAL UNIVERSITY, DELHI. The results embodied in this thesis have not been submitted to any other Institute for the award of any degree.

Date: **Dr RAKESH MEHROTRA**

ASSOCIATE PROFESSOR

Place: DELHI

Department of Civil Engineering

DELHI TECHNOLOGICAL UNIVERSITY,DELHI

DEPARTMENT OF CIVIL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY, DELHI

ACKNOWLEDGEMENT

First and foremost, thanks and praises to the God, the Almighty, for His showers of blessings throughout my dissertation work to complete the thesis successfully.

I take this moment to express my great gratitude and deep regards to **Dr Rakesh Mehrotra** (Associate Professor, Civil Engineering Department, DTU) for his model guidance, supervising and constant motivation throughout the course of this project work. The blessings, help, and guidance are given by him from time to time shall carry me a long way in life on which I am going to start. I would also like to thank him for his friendship, empathy, and a great sense of humour. I an extremely grateful to, **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 support. They have been kind enough to give their opinions on the project matter; I am deeply obliged to them. They always have been a source of encouragement and have continuously been supporting me with their knowledge base, during the study. I would also like to thanks, **Mr Siddhartha Giri** who taught me about the actual hydrodynamics of baffled reactors which was really helpful in the completion of this project. Several of well-wishers offered their help to me directly or indirectly and I thank full to all of them without whom it would not be possible for me to carry on my work.

DEPARTMENT OF CIVIL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY, DELHI

ABSTRACT

Physical/ biochemical processes are designed for specific detention time. The control of detention time is not difficult in a batch reactor but in a continuous flow reactor, the flow distribution pattern makes it difficult to have flow through time matching with hydraulic detention time.

In the recent past, the determination of flow through time could be done only through tracer studies and visualisation of flow patterns on physical models. The overall conclusions about the flow regimes in the reactors were deciphered through residence time distribution studies. Every modification of the reactors configuration required the construction of a modified physical model and maintenance of desired/ designed detention time.

For efficiency of reactors, it is important to choose the L: B ratio appropriately further the volume of dead zone needs to be kept to a minimum. Effective detention time can also be enhanced through the use of baffles.

In the present study, flow analysis of rectangular baffled reactor through differently modified hanging baffles is studied using computational fluid dynamics software. The geometry of reactor was designed in design modular (platform provided in ANSYS fluent) and simulated flow/velocity and fluid properties were input for the study of a different configuration of rectangular four compartments baffled reactor. Resulting flow patterns, flow regimes, velocity fields and dead spaces under varying conditions were analysed.

Of the FOUR models, the 1st model reactor contains normal hanging angled baffle, 2nd model reactor contains baffles having 15 mm horizontal straightener, 3rd model having baffles having side steps like structure and 4th model contain straight baffles having 15 mm horizontal straightener. In each case flow pattern are considered and dead zones, velocity vector, turbulent kinetic energy, streamlines and velocity contours are analysed. Each model is made run with two different velocity magnitudes i.e. 0.07 cm/s and 0.14 cm/s hence the discharges are 10 litres per hour and 20 litres per hour. Different modifications are done on hanging baffles to minimise dead zones and short-circuiting.

In model 1st and model 2nd streamlines and velocity vectors are observed in the maximum area of the reactor which means the effective volume of the reactor is utilised and hence an increase in efficiency of the reactor, also the up-flow velocity in these two reactors are within specific range. In final results, velocities at three different points in each chamber of reactors are obtained and graphs are plotted for each chamber.

Observed results would be helpful in further future design improvement of the anaerobic

Baffled reactor.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER-1

INTRODUCTION

The hydrodynamics and mixing pattern in all bioreactors have strongly influenced the extent of contact in substrate and bacteria thus enhance the efficiency and working of the reactor. It is one of the aspects which should be studied during operation and designing of bioreactors whether pilot or laboratory purpose for a better understanding of flow pattern inside the reactor. In each case efficiency of mixing is largely depends on the type of flow i.e. whether plug flow or completely mixed flow and effective detention time which can vary with flow regime for same reaction kinetic coefficients. Hence mean residence time distribution for all bioreactors is necessary because along with relationship in treatment efficiency and time, there are also used to calculate expected efficiency for such reactors. In most such basins, ideal pattern is plug flow but this is not possible to achieve in practice. The removal and conversion of organic matter in any reactor depends on two main factors i.e. the degree of performance of the microbiological process and hydrodynamics of reactor. Hence the mixing and transport process affects the overall efficiency of such bioreactors. It is known that mixing is the most important factor that strongly affects the efficiency of bioreactor because it helps to homogenize temperature, content of digester and PH. Even mixing pattern at small scale is desirable to provide better conditions for substrate transport to and from the microbial aggregates. (Pena et al.) ,it is necessary to assess the total of mixing required for uniform distribution of digester contents and to obtain the other needed conditions to improve the reactors performance. Hence, the existence of the large recirculation regions can decrease the tank efficiency. Re-circulation regions or dead zones in these types of reactors create flow mixing problems and hence decrease the hydraulic retention time. Thus, the important objective in designing rectangular baffled reactors is to lower the formation of the re-circulation zone. One applicable method to minimize the volume of the dead zones and increase the efficiency of the digesters is to use a proper baffle configuration (Razmi et al., 2009). In this dissertation, using ANSYS FLUENT (R 16.0), (a fluid analysis tool of COMPUTATIONAL FLUID DYNAMICS software), “flow analysis in a rectangular baffled reactor having differently modified hanging baffles” is done.

In recent past, to design such digesters, physical model studies, and tracer studies were conducted after construction by empirical correlation. With the availability of general purpose codes such as Ansys fluent, cfx etc., computational fluid dynamics has become popular increasingly in every branch of engineering. CFD techniques allow us to simulate the actual conditions happening inside the study model with ease and accuracy that is very difficult to achieve through physical model studies. Velocity and flow distributions patterns are studied in this project which relates with the short-circuiting as well as recirculating 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 modeling setup, it is very difficult to find out the coordinates and velocity of flowing particles and to do that advanced instruments like a stroboscope, Pitot tube, laser anemometry etc. are needed.

Every modification of the reactors configuration required the construction of a modified physical model and maintenance of desired/ designed detention time. For efficiency of reactors, the volume of dead zone needs to be kept minimum. Effective detention time can also be enhanced through the use of baffles. In the present study by the use of computational fluid dynamics flow analysis of rectangular baffled reactor through differently modified hanging baffles is studied to simulate the input conditions and study of different configuration of four compartmentalized reactors for obtaining the resulting flow patterns, flow regimes, velocity fields and dead spaces under varying conditions. as per their requirements with less cost and time. 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.

1.1 **ABR** (Anaerobic Baffled Reactor)

Anaerobic Baffled Reactor is a simplified version of septic tank consists of a number of alternative baffles hanging straight or angled position and standing position. Inside it, both physical and biological treatment is done by microbial decomposition of wastewater. Active biomass needs to be retained in desirable concentrations in all chambers. The waste water allows to enter from the inlet to move downward and then upward. In upward velocity should be less than the settling velocity so that solid particles can settle down in the chamber. After reaching final compartment the treated wastewater is obtained from the outlet. These types of systems are suitable for high strength industry waste water and efficiency increases with higher OL, comprising soluble BOD/non-settable solids having low COD/BOD ratio. The flow in ABR starts forms 5 to 225 per day. HRT is in between 45 to 70 hours. The most important designing criteria are up-flow velocity which must not be greater than 0.055556 cm/s which would 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.

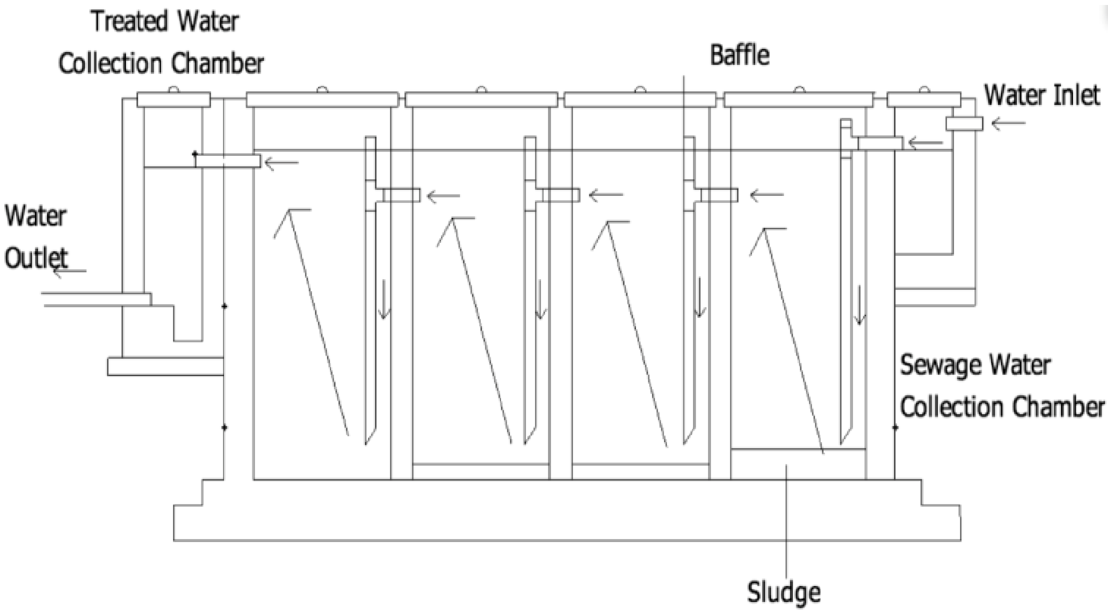


Figure 1.1 Anaerobic baffled reactor

1.2 COMPUTATIONAL FLUID DYNAMICS:

Computational fluid dynamics is software which is used to simulate the flow patterns using boundary conditions, fundamental equations and flow rates to calculate the results that same obtained in the experimental system by use of computer models. Computational fluid dynamics approach make is easier to visualise the effect of different modification done in reactors on flow patterns without carrying out it on large scale

1.2.1 ANSYS FLUENT(16.0)

(An analysis tool available in computational fluid dynamics software)

This software was founded by Mr JOHN A. SWANSON. "ANSYS Fluent" is one of the programs of CFD which is used to solve the fluid related problems. In this tool fluid is used as influent. It has wide physical modelling capability which is needed to flow, model, turbulence, reactions and heat transfer for industrial application ranging from blood flow to semiconductor to bubble column to oil platforms. The approaches involves in CFD are:

1. Finite volume method
2. Finite difference method
3. Finite element method

1.2.2 ADVANTAGES OF CFD:

CFD is used for forecasting of a variety of issues related to flow, density, velocity, temperature etc.

1. It foretells performance before installation
2. Saves time and cost. It is reliable
3. It can execute simulations at great high speed with negligible error

1.2.3 DISADVANTAGES OF CFD:

Sometimes it is unrealistic to expect that the modelling process will give results that are a perfect match for the real case. Therefore it is necessary to know about limitations also.

1. The model will be perfect only when boundary conditions are correct.
2. Models are based on actual geometry hence all the dimensions should be correctly used while using geometry module.

1.3 OBJECTIVE OF DISSERTATION:

The main objectives of the study are as follows:

1) To minimize the dead zones near the chamber, walls and bottom by different modifications in hanging baffles using ANSYS Fluent.
2) To maintain downflow velocity and up-flow velocity within the range.
3) To study flow patterns, flow regimes, velocity fields and dead spaces under varying conditions.

4) To enhance the efficiency of rectangular reactor and to utilize the effective volume of reactor by different modifications in hanging baffles.

1.4 ASSUMPTIONS IN DISSERTATION:

1. Only one phase flow is considered i.e. effects of solids and gasses were not taken into consideration.
2. In the analysis, water was used as the fluid.
3. Biological parameters like organic loading rates were not considered. Only hydraulic loading rate was considered.
4. Whole work was done in ANSYS FLUENT software because actual modelling is more time taking .

1.5 ORGANISATION OF DISSERTATION:

This thesis is subdivided into six chapters. Chapter 1 states the introduction of topic and objective of the study. Chapter 2 presents literature reviews on the topic which I studied. Chapter 3 & chapter 4 includes methodology and numerical data obtained in the project and last chapter i.e. chapter 6 contain the future scope of study and results in the form of pictorial representations and velocity graphs.

CHAPTER-2

LITERATURE REVIEW

Leonardo. Rosa et al. [1] In this study, a numerical study was presented with the aim of optimizing anaerobic sequencing batch reactor for bio-hydrogen production. Computational fluid dynamics techniques were used to provide accurate results for the fluid flow which directly affects the momentum and kinetics of reactions involved. Eulerian -Eulerian approach to describe the flow of the phase is used by using OPEN FOAM CODE. Small baffle plates added horizontally in the reactor to provide higher mixing. What authors found that after the inclusion of baffle plates of length equal to ¼ of the reactors radius, turbulent kinetic energy was increased with very little effect on pressure drop which showed better mixing.

Sheng-nan li et al. [2] In this study authors analysed the hydraulic characteristic between the different structure of two anaerobic baffled reactors. In this study, they used plane folded plate reactor (PFPR) and opposite folded plate reactor (OFPR). they studied residence time distribution (RTD) on PFPR and OFPR under clean and working conditions at the same hydraulic residence time of 4,6,8 and 10 hours to visualize mixing pattern and dead spaces for both reactors. They observed that mixing pattern for both the reactors were within the "intermediate state" i.e. between plug flow and completely mixed flow. However, the mixing pattern of opposite folded plate reactor was closer to plug flow thus, dead spaces of the opposite folded plate reactor were less that of plane folded plate reactor.

<u>Jun-Mei Zhang et al.</u> [3] In this study authors presented the study of potable water service reservoir. They used COMPUTATIONAL FLUID DYNAMICS software STAR-CCM+ (version 5.04) to visualize the effect of baffles configuration on flow pattern. They used Reynolds Time-Averaged Nervier Strokes (RANS) model to simulate the turbulent flow field inside the rectangular tank. They proposed fine individual baffle configuration for the service reservoir. In this study, they showed a dual effect of baffle located at the flow recirculation region. On one hand, it can break up the vortex to shorten the flow path and on the other hand, the velocity magnitude of fluid is reduced after contradictory to one another enhancing the performance of service reservoir acting as a storage tank.

<u>J.Z hang .M.ASCE et al.</u> [4] In this study with the help of ANSYS FLUENT authors analysed on ozone reactions in bio-reactor. In this research, the reactor was made of using normal OPEN FOAM software. Three designs were made using half and quarter width and simulations were done using RANS equations. Their objective was to reduce dead zones region and short-circuiting that makes a model to work less efficient. They also studied losses of energy as well as the performance of baffles.

<u>R.Renuka et al.</u> [5] The aim of this study was to present the influence of hydraulic behaviour in the treatment of sewage (domestic wastewater) using Panelled Anaerobic Baffle-cum Filter Reactor (PABFR). The PABFR has fine compartments of equal sizes in which the first three compartments operated as anaerobic baffled reactor (ABR) followed by anaerobic filters (AF). The combined reactor has great potential for arrangements of baffles inside each compartment. In this study, they determined the hydraulic behaviour of the reactor by means of pulse tracer test and by calculating the residence time distribution curves at different flow rates. They noticed that at high flow rates, the mixing pattern in anaerobic baffle reactor was the completely mixed type with maximum dead zones of 14% and as the flow decreased, the anaerobic baffle reactor's mixed behaviour was intermediate between plug flow and completely mixed flow. On AF , as the flow increased, the dispersion was intermediate between completely mixed and plug flow but when the flow rate was decreased, the reactor become completely plug flow with minimum dead zones ranging 2.7 & 7.4.

P.Dama et al. [6] In this study authors used COMPUTATIONAL FLUID DYNAMICS software to simulate the flow pattern using fundamental equations, boundary conditions, and flow rates to compare results one would obtain on an experimental system. They used ANSYS FLUENT tool, a commercially available CFD program to develop a laboratory scale model (10 litres) of anaerobic baffle reactor that could be further used in the design and scale up to a pilot scale of ABR. They observe that the 45° angle at the bottom of the baffle and down flow to up flow ratio of 1:3 gave the most suitable pattern. They also verified the results by injecting a dye tracer in the laboratory scale anaerobic baffle reactor. The flow pattern observed during the dye tracer was similar to pattern predicted by fluent software.

D.C.Stuckey et al. [7] In this paper the author presented the residence time distribution on both clean and working reactors to investigate the mixing pattern and dead spaces in the reactor. They showed results in which mixing was characterised by a number of theoretical perfectly mixed compartments which correlates closed with the actual number of the compartment in the reactor at low hydraulic residence time (HRT) . There said that there is no direct relation between dead space and HRT because dead zones can be shown to made up partly of biological dead space (due to the presence of biomass) which decreases with HRT and partly of hydrologic dead space (due to flow patterns) which increase with HRT.

Alexander M Mendoza et al. [8] In this research, they used computational fluid dynamics to simulate a 3-Dimensional steady state flow for a particular anaerobic digester in order to visualize the flow patterns. They represented flow and velocity profiles inside the digester to identify dead zones. In this paper the used the geometry of real digester installed in Valencia a wastewater treatment plant in Spain. What they noticed is that the distribution of velocities and streamlines in the geometry played a decisive role, as the inflow nozzle which may determine the occurrence of the dead zone along the flow. They also noticed that the importance of considering mixing pattern when simulation anaerobic digester and designing of bio-reactors.

CHAPTER-3

METHODOLOGY

3.1 NUMERICAL METHOD:

Different models of anaerobic baffled reactor are analysed in ANSYS FLUENT (16.0) software. Generally, there are three steps involved in the numerical simulation of fluid flow in ANSYS FLUENT tool.

(i) Pre-Processing

This is the first step in CFD flow simulation. It helps in making geometry in geometry modular embedded in ANSYS fluent tool available in computational fluid dynamics software. One needs to generate fluid domain and then further meshing is done which means dividing geometry into smaller segments.

(ii) Solver

In solver one can set boundary conditions, fluid material properties, flow physics model. Using Ansys fluent tool, it becomes easy to solve the governing equations related to flow physics problem.

(iii) Post-Processing

In post processing analysis of final results are done in the form of the velocity vector, velocity contour, turbulent kinetic energy, streamlines, volume rendering etc.

3.2 GEOMETRY MAKING:

ANSYS fluent provides the platform in which one can draw geometry in design modular. After geometry meshing and set up of different parameters is done. After setup solver solves governing equations and provides results in the form of streamlines, velocity contours etc.

3.2.1 DESIGN MODULAR:

Geometry for all four models was drawn in design modular which was very time taking process. Since the results depend on the accuracy of geometry, it should be made carefully with actual dimensions. Four models of rectangular tanks having dimensions 420 x 200 mm are drawn. All dimensions are in millimetres. Each model has four chambers and has differently modified baffles. Model 1st contain simple angled baffle. Model 2nd contain baffle with 15 mm horizontal straightener. Model 3rd contains baffles having steps like structure and model 4th contain straight baffles having 15 mm horizontal straightener.

Table 3.1: Dimensions of geometry

Nomenclature of Geometry	Dimensions (mm)
Length of reactor	**420**
the width of the reactor	**200**
Angle of angle baffle	**45**
the length of standing baffle	**140**
the length of hanging baffle	**140**
length to the horizontal straightener	**15**
dimensions of steps in 3rd model	**3x5**
the width of baffles	**10**

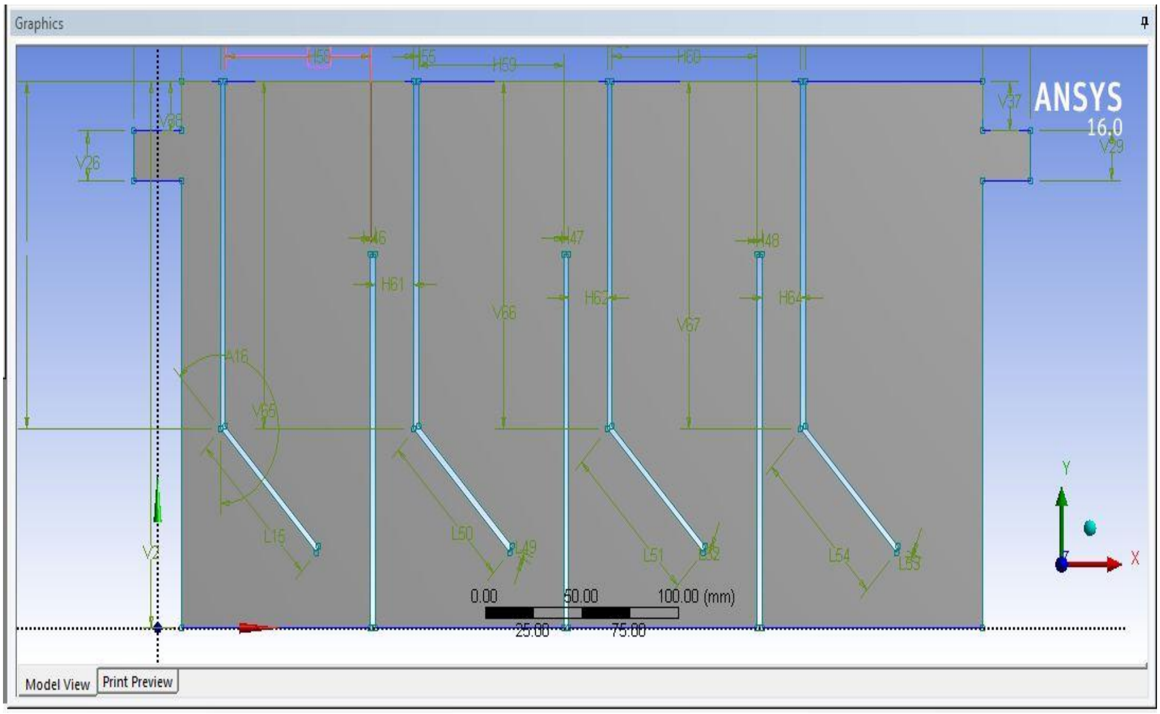


Fig. 3.1 <u>Geometry of Model -1 (reactor with simple angled baffles)</u>

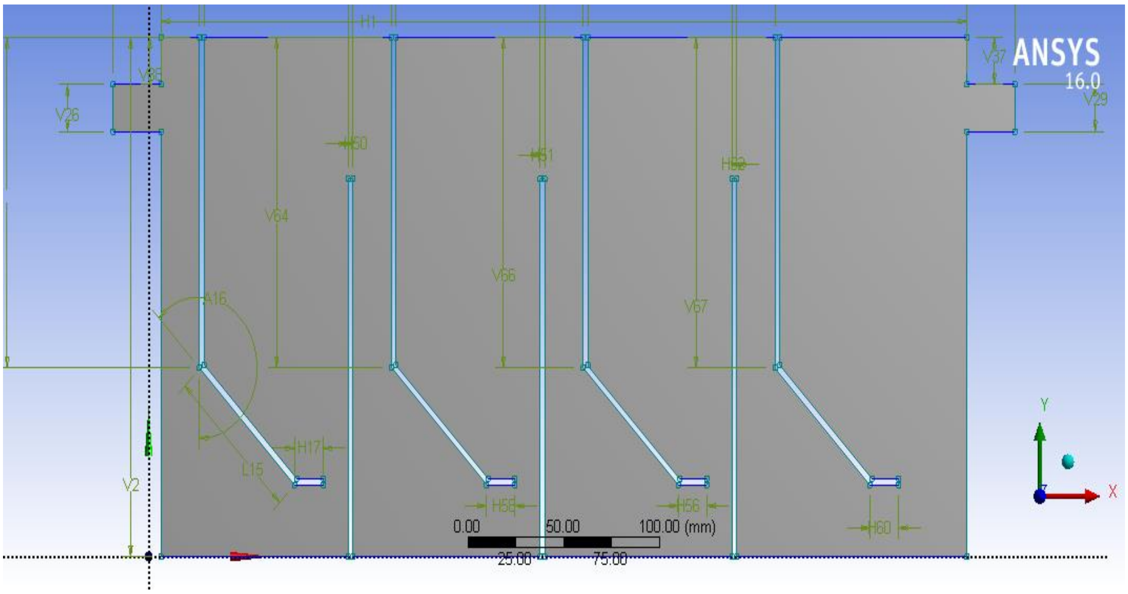


Fig. 3.2 <u>Geometry of Model -2 (reactor with baffles having 15mm horizontal straightener)</u>

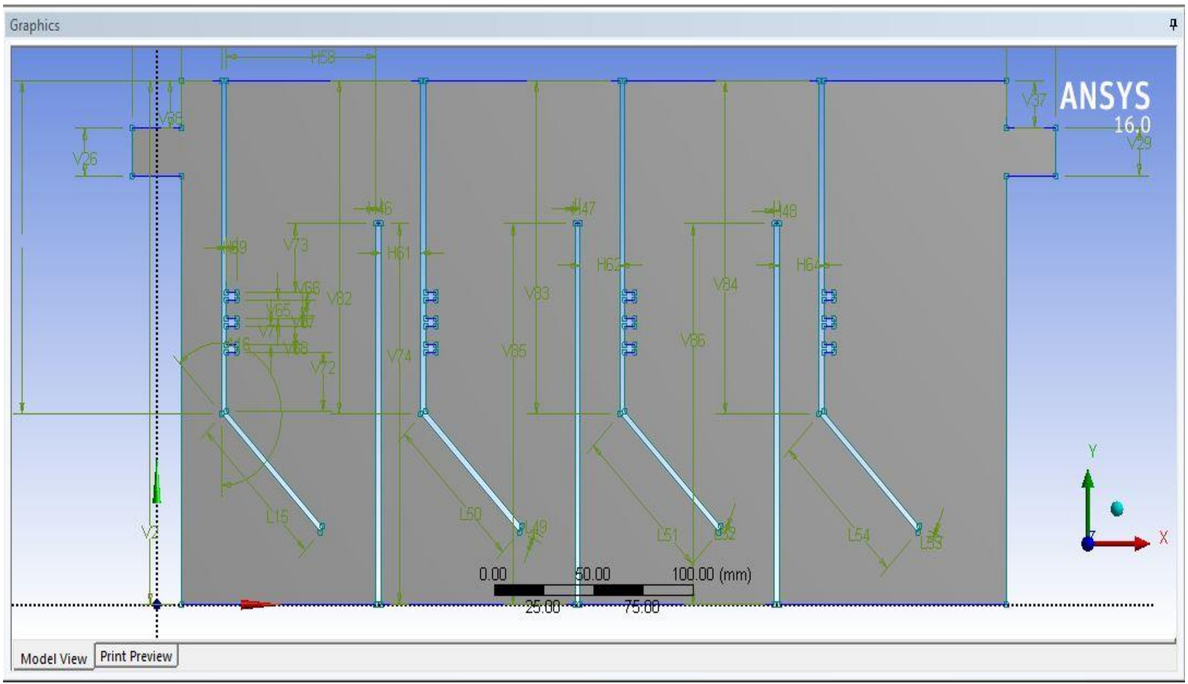


Fig. 3.3 Geometry of Model -3 (reactor having baffles with steps like structure)

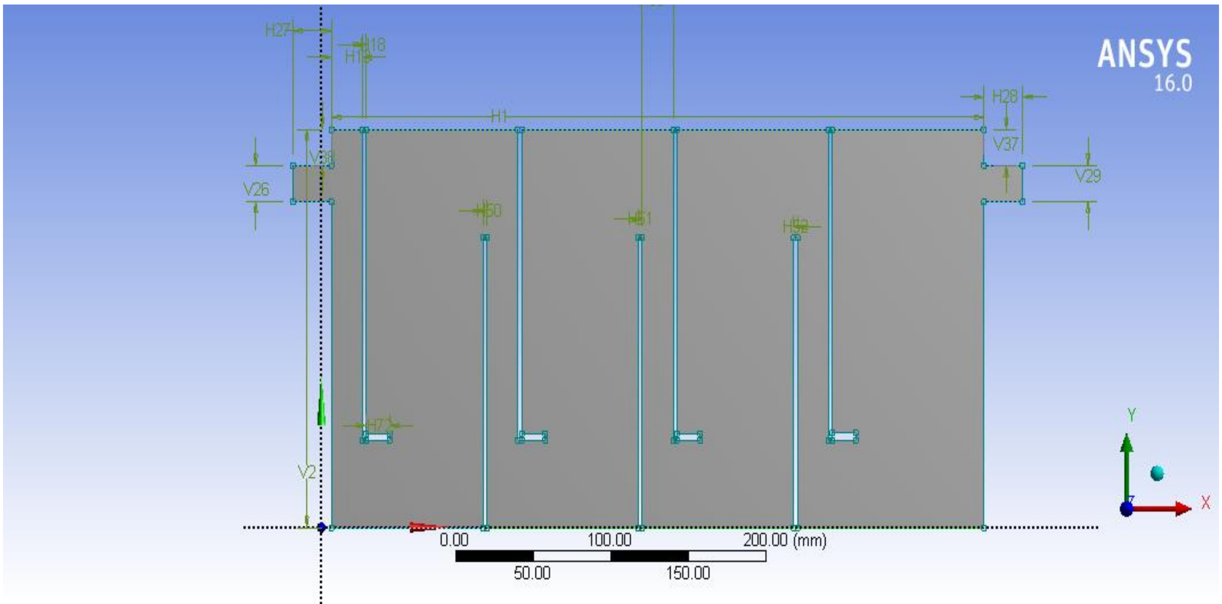


Fig. 3.4 Geometry of Model -4 (reactor with straight baffles having 15mm straightener)

3.2.2 MESHING:

After geometry, the next step is meshing. Meshing is the most important step in all types of simulations. Because of this one can get different results. It means creating a mesh of grid points called nodes. Since the results will be carried out by solving governing equations at each node, mesh parameters should be correctly set up.

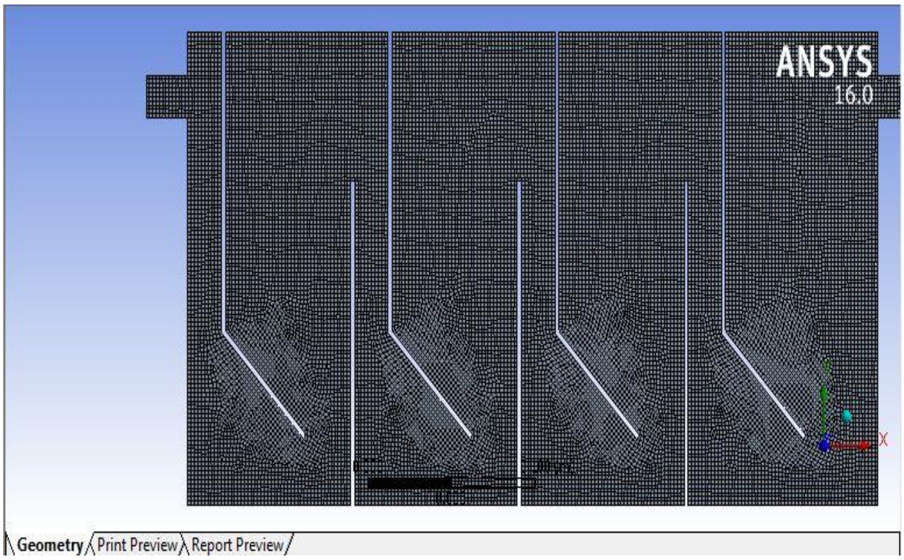


Fig. 3.5 Meshing of Model-1^{s}

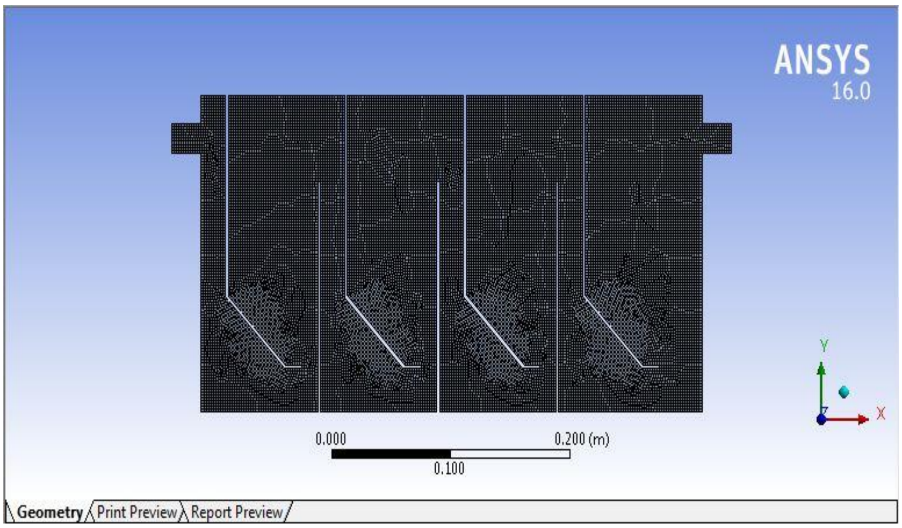


Fig. 3.6 Meshing of Model-2nd

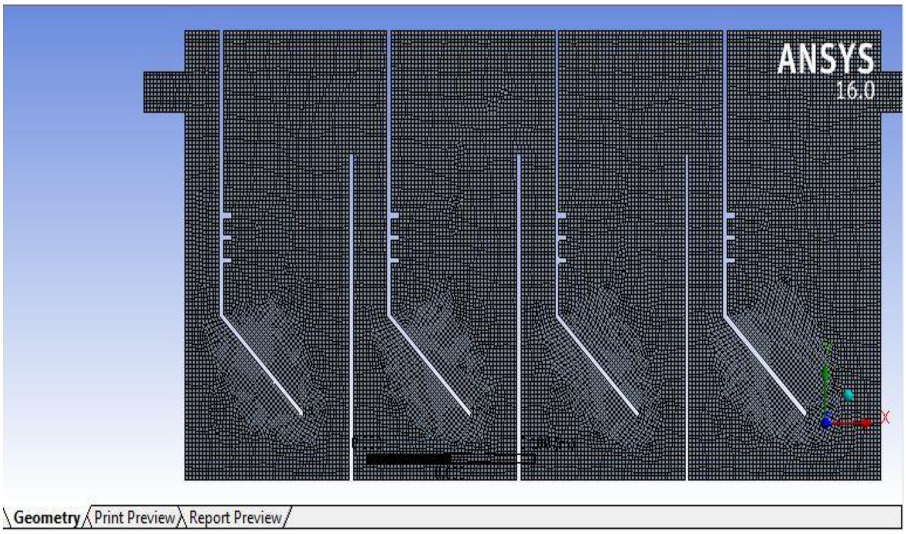


Fig. 3.7 **<u>Meshing of Model 3rd</u>**

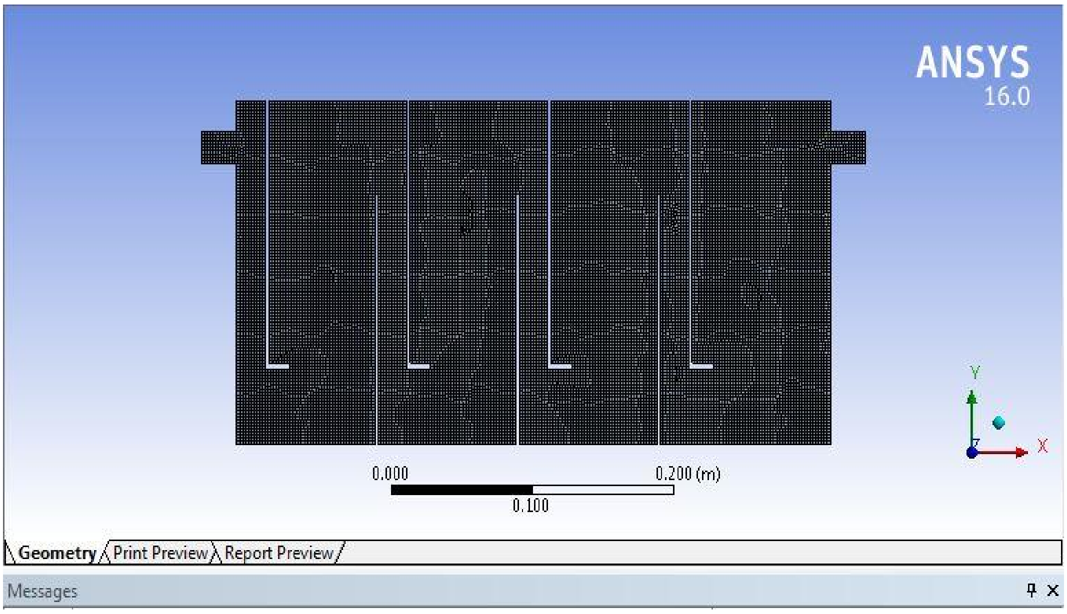


Fig. 3.8 **<u>Meshing of Model-4th</u>**

3.2.3 FLUENT SETUP:

In ANSYS FLUENT tool, there are two solvers available.

- Pressure based solver
- Density based coupled solver

In first one pressure and momentum used as primary variables and in second one pressure velocity coupling algorithms. In this dissertation pressure based solver is used. Pressure based solver is applicable for single phase flows only. In this step model is selected for analysis and fundamental equations had been used by software to initialize conservation of mass, momentum and energy as RANS and k-epsilon equations to find eddy viscosity by representing turbulence characteristic.

3.2.4 Reynolds-Averaged Navier–Stokes Equations (or RANS equations):

Reynolds-Averaged Navier-Strokes equations are time-averaged equations of motion for flow. The target behind the equations is Reynolds decomposition, through which an instantaneous quantity is decomposed into its time-averaged and vary quantities, this idea first given by Osborne Reynolds. The RANS equations are initially used to define the turbulent flows. These equations can be used with conjecture (approximations) based on fact of the properties of flow turbulence to give conjecture time-averaged solutions to the Navier–Stokes equations. For an incompressible Newtonian fluid, equations can be written in as:

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

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

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:

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

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

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 AND BOUNDARY CONDITION: This step is to define material and boundary conditions. The fluid was used as the material. Boundary conditions such as a wall, inlets, and outlets. Boundary conditions inlet was taken as velocity inlet and outlet was taken as pressure outlet. Velocity magnitude was calculated by dividing discharge by area of the inlet. Velocity magnitudes were used as 0.07 cm/s and 0.14 cm/s. free surface was taken as symmetry condition.

3.2.5: SOLUTION:

The last step is to initialize the solution. To initialize a number of iterations were set up to 700 and then the solution is calculated until it is get converged. Since the results highly depend on the convergence of solution so it is necessary to be it converged and convergence of solution also depends on mesh quality. If the mesh is fine, more time will be taken to converge the solution. In this case number of iterations required to converge the solution were always less than 250. Here are the converged solutions of all four models at two different velocity magnitudes.

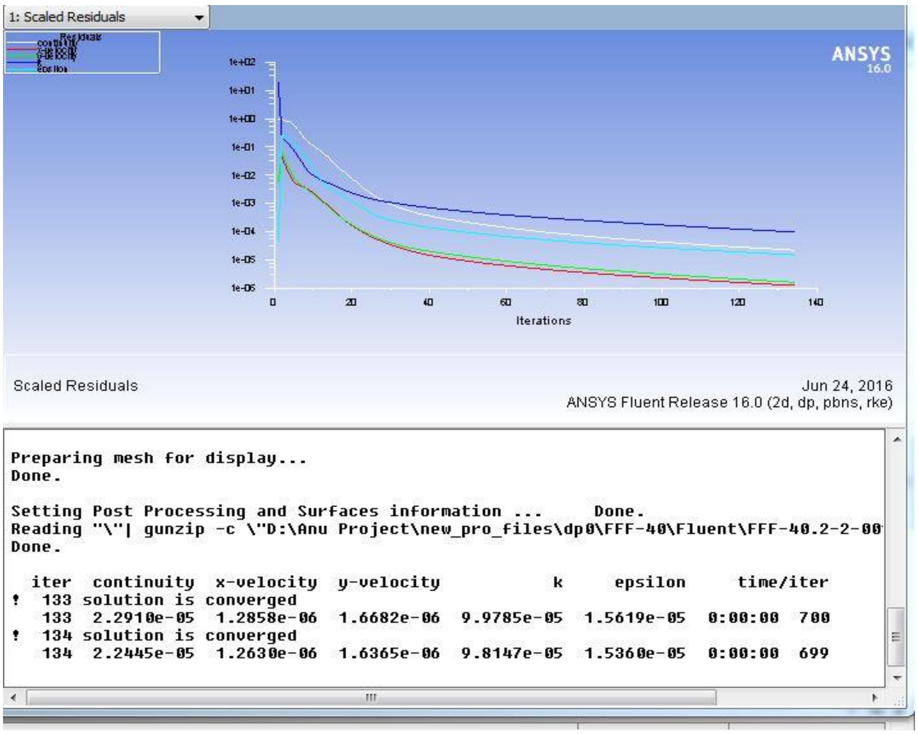


Fig. 3.9 Converged solution of model-1st for 0.07 cm/s

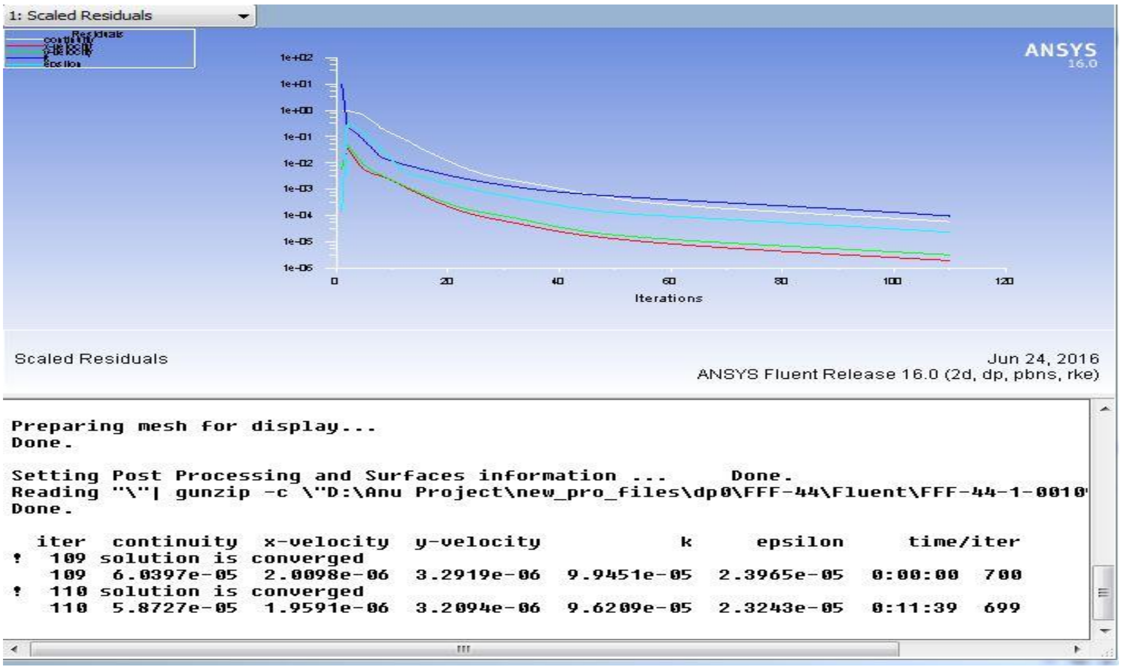


Fig. 3.10 <u>**Converged solution of model-1st for 0.14 cm/s**</u>

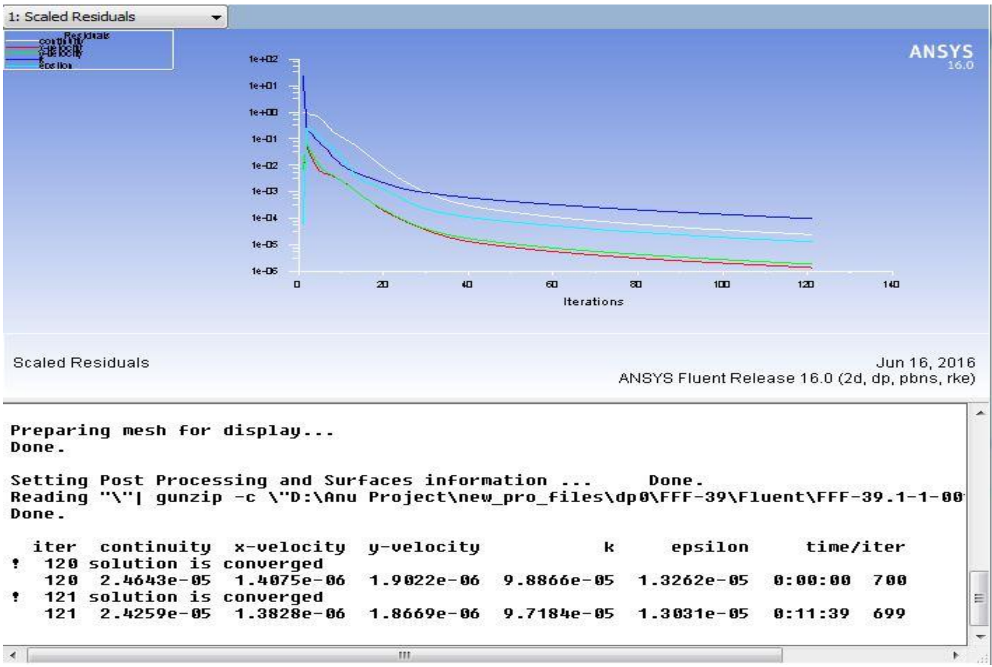


Fig. 3.11 <u>**Converged solution of model-2nd for 0.07 cm/s**</u>

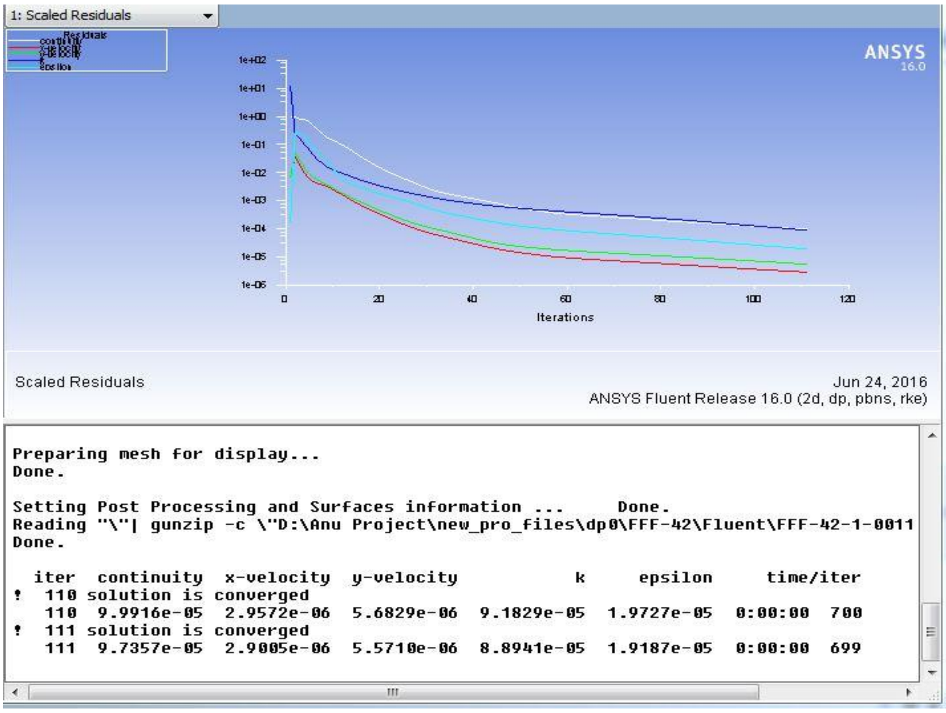


Fig. 3.12 <u>Converged solution of model-2nd for 0.14 cm/s</u>

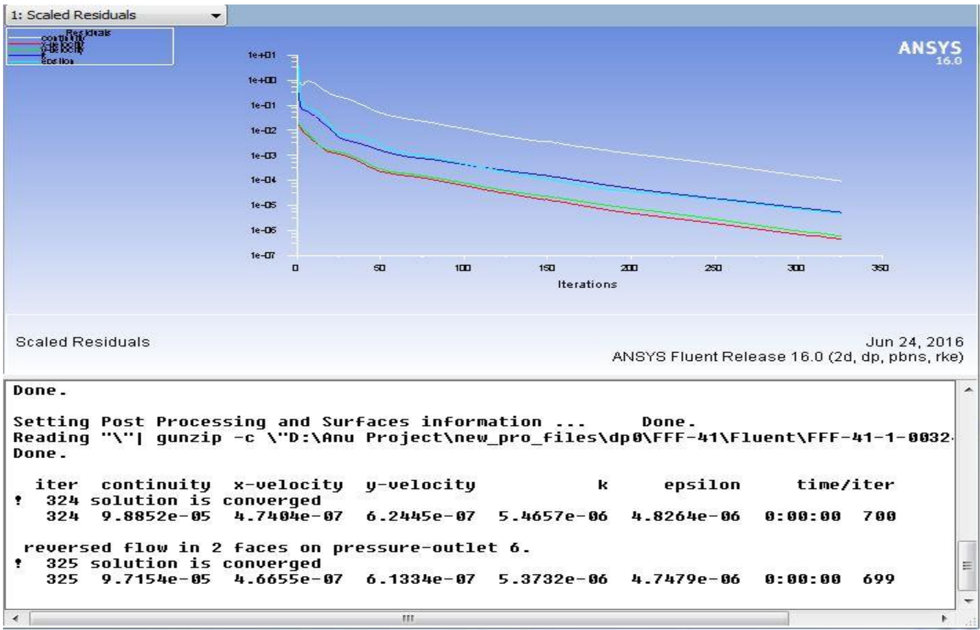


Fig. 3.13 <u>Converged solution of model-3rd for 0.07 cm/s</u>

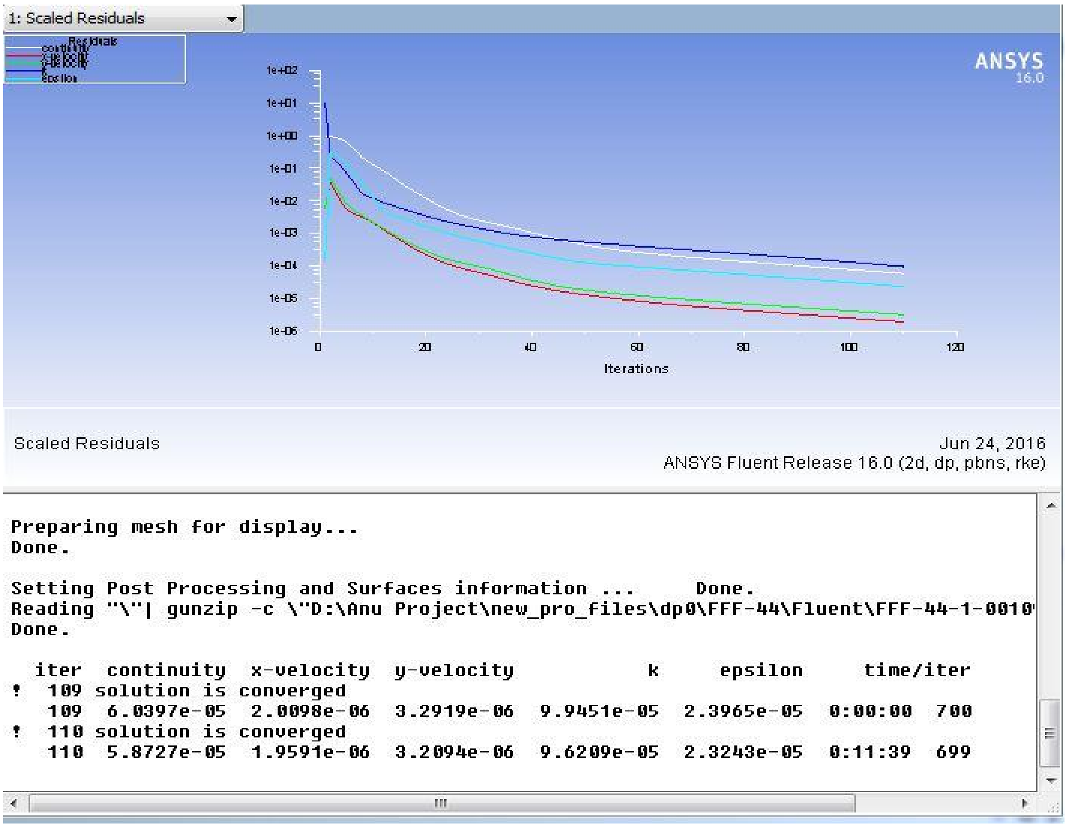


Fig. 3.14 <u>**Converged solution of model-3rd for 0.14 cm/s**</u>

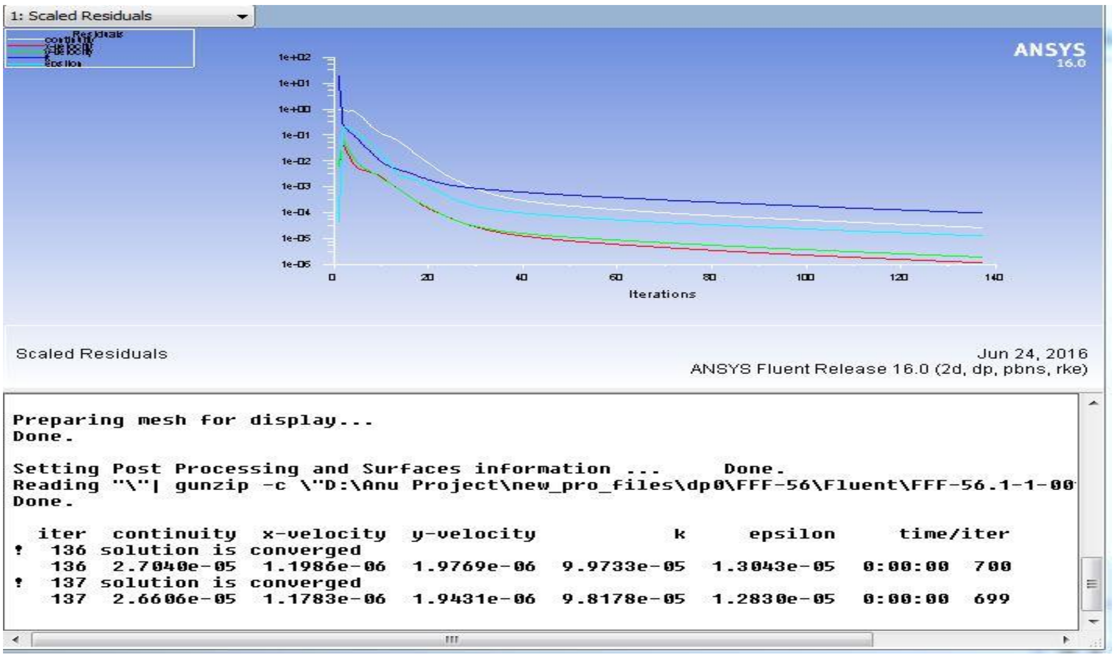


Fig. 3.15 <u>**Converged solution of model-4th for 0.07 cm/s**</u>

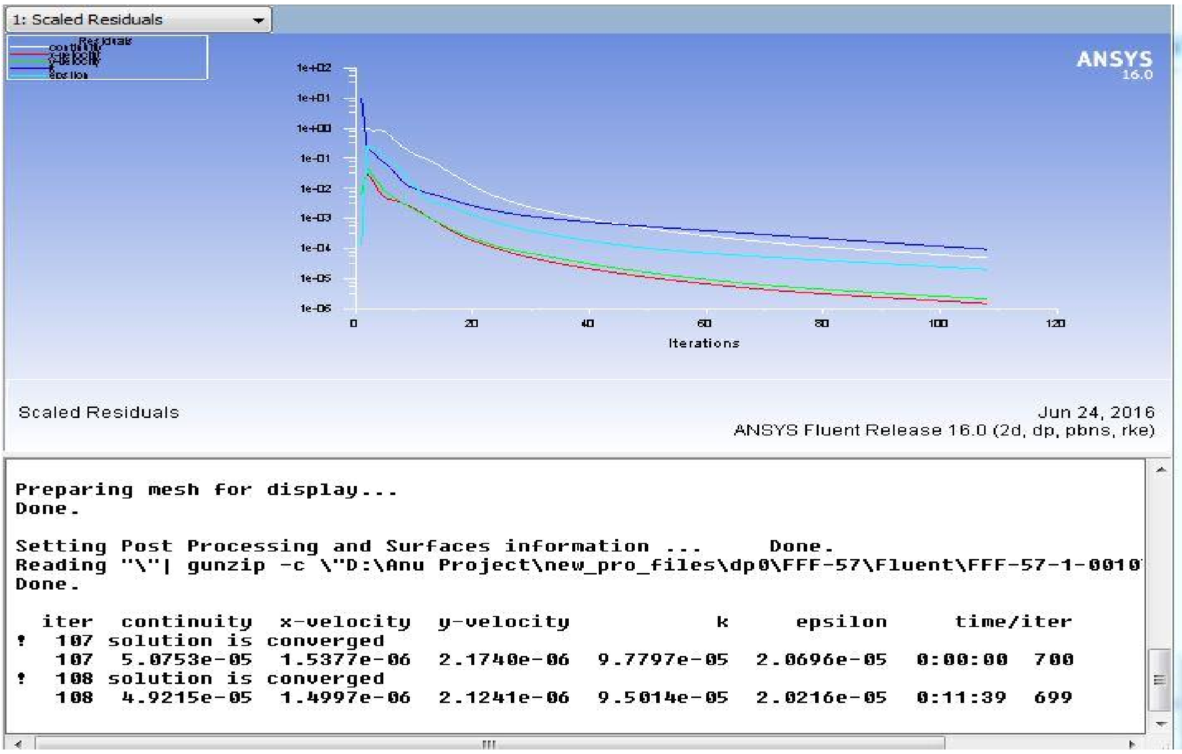


Fig. 3.16 Converged solution of model-4^{th} for 0.14 cm/s

CHAPTER- 4

NUMERICAL DATA

After analysis of results in CFD-POST embedded in Fluent, velocities data in the form of table is presented in this chapter. As there are 4 models made to run in which each model is made to run at two different velocity magnitudes i.e. 0.07 cm/s and 0.14 cm/s. so, there is a total of 8 cases. In each case velocities at three different points i.e. at the bottom of the chamber, at the middle of the chamber and at the upper surface of the chamber are obtained by using Probe option in CFD.

Table 4.1: Velocities at three different points in each chamber for model - 1 (reactor with simple angled baffle) for 10 litres per hour Discharge.

FOR VELOCITY MAGNITUDE 0.07 cm/s		Distance in mm	Velocity in cm/s		
		x-Axis	Near Bottom	At middle	Near upper surface
	CHAMBER-1	0	0	0	0
		20	0.01738	0.05567	0.05385
		40	0.04138	0.01008	0.01131
		60	0.0534	0.03554	0.02850
		80	0.0313	0.02422	0.03604
		100	0	0	0

MODEL -1 **REACTOR WITH SIMPLE ANGLED BAFFLE**	**CHAMBER-2**	100	0	0	0
		120	0.02647	0.01024	0.05154
		140	0.04370	0.02571	0.02511
		160	0.03649	0.03291	0.04086
		180	0.02649	0.02248	0.03485
		200	0	0	0
	CHAMBER-3	200	0	0	0
		220	0.01803	0.05057	0.05556
		240	0.02758	0.04592	0.01342
		260	0.05418	0.02756	0.02620
		280	0.03879	0.01465	0.03388
		300	0	0	0
	CHAMBER-4	300	0	0	0
		320	0.01704	0.01001	0.05350
		340	0.02849	0.01203	0.01249
		360	0.05503	0.02439	0.02008
		380	0.03385	0.02772	0.03336
		400	0	0	0

Table 4.2: Velocities at three different points in each chamber for model - 1 (reactor with simple angled baffle) for 20 litres per hour Discharge.

FOR VELOCITY MAGNITUDE 0.14 cm/s		Distance in mm	Velocity in cm/s		
		x-Axis	Near Bottom	At middle	Near upper surface
MODEL -1 REACTOR WITH SIMPLE ANGLED BAFFLE	CHAMBER-1	0	0	0	0
		20	0.03047	0.01769	0.01895
		40	0.02354	0.02276	0.03215
		60	0.05889	0.04577	0.03560
		80	0.05333	0.05658	0.05164
		100	0	0	0
	CHAMBER-2	100	0	0	0
		120	0.01859	0.01909	0.01485
		140	0.03368	0.05433	0.02162
		160	0.06653	0.05205	0.04005
		180	0.05009	0.05304	0.07442
		200	0	0	0
	CHAMBER-3	200	0	0	0
		220	0.03191	0.0127	0.01683
		240	0.03696	0.01721	0.01751
		260	0.06882	0.03940	0.032386
		280	0.04503	0.06306	0.06076
		300	0	0	0

	CHAMBER-4	300 320 340 360 380 400	0 0.03198 0.07005 0.06405 0.06505 0	0 0.017441 0.01278 0.05307 0.06583 0	0 0.01832 0.08174 0.04359 0.06053 0

Table 4.3: Velocities at three different points in each chamber for model -2 (reactor with 15 mm horizontal plate added baffle) for 10 litres per hour Discharge.

FOR VELOCITY MAGNITUDE 0.07 cm/s		Distance in mm	Velocity in cm/s		
		x-Axis	Near Bottom	At middle	Near upper surface
	CHAMBER-1	0 20 40 60 80 100	0 0.02676 0.07765 0.06781 0.06368 0	0 0.01028 0.01916 0.02821 0.03071 0	0 0.01087 0.01345 0.01915 0.02568 0

MODEL -2 **REACTOR WITH 15 mm HORIZONTAL PLATE ADDED TO BAFFLE**	**CHAMBER-2**	100	0	0	0
		120	0.06448	0.06728	0.05874
		140	0.03743	0.05387	0.06475
		160	0.07301	0.08852	0.02563
		180	0.07112	0.08940	0.02613
		200	0	0	0
	CHAMBER-3	200	0	0	0
		220	0.05877	0.04410	0.09390
		240	0.03131	0.03692	0.01195
		260	0.06961	0.03641	0.07799
		280	0.05560	0.08140	0.07827
		300	0	0	0
	CHAMBER-4	300	0	0	0
		320	0.07690	0.03911	0.05151
		340	0.06671	0.09224	0.08888
		360	0.04394	0.07779	0.06093
		380	0.08394	0.06594	0.06763
		400	0	0	0

Table 4.4: Velocities at three different points in each chamber for model -2 (reactor with 15 mm horizontal plate added baffle) for 20 litres per hour Discharge.

FOR VELOCITY MAGNITUDE 0.14 cm/s		Distance in mm	Velocity in cm/s		
		x-Axis	Near Bottom	At middle	Near upper surface
MODEL -2 REACTOR WITH 15 mm HORIZONTAL PLATE ADDED TO BAFFLE	CHAMBER-1	0	0	0	0
		20	0.03473	0.08766	0.02014
		40	0.03610	0.01457	0.01610
		60	0.06225	0.08218	0.01750
		80	0.01074	0.01369	0.0800
		100	0	0	0
	CHAMBER-2	100	0	0	0
		120	0.02729	0.02001	0.02062
		140	0.07620	0.01508	0.02696
		160	0.08333	0.02535	0.07589
		180	0.01109	0.01847	0.07218
		200	0	0	0
	CHAMBER-3	200	0	0	0
		220	0.02184	0.01487	0.01561
		240	0.04064	0.04634	0.01426
		260	0.09223	0.09548	0.06858
		280	0.01209	0.01392	0.08513
		300	0	0	0

	CHAMBER-4	300	0	0	0
		320	0.05660	0.01468	0.01648
		340	0.06822	0.06364	0.01283
		360	0.07726	0.02873	0.02922
		380	0.01254	0.08761	0.06309
		400	0	0	0

Table 4.5: Velocities at three different points in each chamber for model -3 (reactor having steps like structure on hanging baffles) for 10 litres per hour Discharge.

FOR VELOCITY MAGNITUDE 0.07 cm/s		Distance in mm	Velocity in cm/s		
		x-Axis	Near Bottom	At middle	Near upper surface
	CHAMBER-1	0	0	0	0
		20	0.08307	0.05835	0.07118
		40	0.07486	0.05708	0.02279
		60	0.06944	0.07643	0.05504
		80	0.06253	0.08404	0.08107
		100	0	0	0

<u>MODEL -3</u> **<u>REACTOR HAVING STEPS LIKE STRUCTUREON BABBLES</u>**	**CHAMBER-2**	100	0	0	0
		120	0.07178	0.02055	0.07758
		140	0.01768	0.031198	0.07078
		160	0.08169	0.09142	0.08720
		180	0.02768	0.08619	0.08557
		200	0	0	0
	CHAMBER-3	200	0	0	0
		220	0.08920	0.06183	0.05707
		240	0.02564	0.05482	0.04205
		260	0.03782	0.08127	0.08155
		280	0.07953	0.08911	0.0932
		300	0	0	0
	CHAMBER-4	300	0	0	0
		320	0.08107	0.09860	0.08900
		340	0.06652	0.0800	0.07124
		360	0.04788	0.08251	0.087250
		380	0.04019	0.06921	0.07294
		400	0	0	0

Table 4.6: Velocities at three different points in each chamber for model -3 (reactor having steps like structure on hanging baffles) for 20 litres per hour Discharge.

FOR VELOCITY MAGNITUDE 0.14 cm/s		Distance in mm	Velocity in cm/s		
		x-Axis	Near Bottom	At middle	Near upper surface
MODEL -3 REACTOR HAVING STEPS LIKE STRUCTUREON BABBLES	CHAMBER-1	0	0	0	0
		20	0.09752	0.09116	0.02374
		40	0.02829	0.02362	0.01707
		60	0.08693	0.06924	0.09804
		80	0.04263	0.01737	0.02136
		100	0	0	0
	CHAMBER-2	100	0	0	0
		120	0.02068	0.01183	0.01645
		140	0.04753	0.01721	0.04197
		160	0.06239	0.01735	0.05427
		180	0.06871	0.01856	0.01935
		200	0	0	0
	CHAMBER-3	200	0	0	0
		220	0.02541	0.01143	0.01158
		240	0.02816	0.05752	0.04714
		260	0.08008	0.03180	0.06795
		280	0.05884	0.018852	0.01704
		300	0	0	0

	CHAMBER-4	300	0	0	0
		320	0.05195	0.010060	0.03858
		340	0.06833	0.04042	0.04182
		360	0.07237	0.01562	0.054431
		380	0.07232	0.01372	0.01284
		400	0	0	0

Table 4.7: Velocities at three different points in each chamber for model -4 (reactor having straight baffle with 15mm horizontal straightener) for 10 litres per hour Discharge.

FOR VELOCITY MAGNITUDE 0.07 cm/s		Distance in mm	Velocity in cm/s		
		x-Axis	Near Bottom	At middle	Near upper surface
	CHAMBER-1	0	0	0	0
		20	0.01844	0.05937	0.03583
		40	0.02900	0.01089	0.0137
		60	0.02702	0.02627	0.02346
		80	0.01866	0.02427	0.02111
		100	0	0	0
	CHAMBER-2	100	0	0	0
		120	0.01239	0.05370	0.02563
		140	0.02133	0.01321	0.01557
		160	0.03087	0.02780	0.02607
		180	0.02434	0.02528	0.02025
		200	0	0	0

MODEL -4 **REACTOR HAVING STRAIGHT BAFFLES WITH 15 mm STRAIGHTENER**	**CHAMBER-3**	200 220 240 260 280 300	0 0.01127 0.02809 0.02896 0.01567 0	0 0.05035 0.01450 0.03193 0.0197 0	0 0.05403 0.01823 0.02536 0.04290 0
	CHAMBER-4	300 320 340 360 380 400	0 0.01882 0.02274 0.03199 0.01217 0	0 0.04566 0.01282 0.02570 0.01385 0	0 0.05989 0.01116 0.02422 0.06115 0

Table 4.8: Velocities at three different points in each chamber for model -4 (reactor having straight baffles with 15mm horizontal straightener) for 20 litres per hour Discharge.

FOR VELOCITY MAGNITUDE **0.14 cm/s**		**Distance in mm**	**Velocity in** **cm/s**		
		x-Axis	**Near Bottom**	**At middle**	**Near upper surface**
	CHAMBER-1	0 20 40 60 80 100	0 0.04499 0.09589 0.06189 0.05256 0	0 0.01749 0.02254 0.06454 0.04005 0	0 0.04322 0.02757 0.05461 0.01048 0

	CHAMBER-2	100	0	0	0
		120	0.04977	0.01035	0.01062
		140	0.08603	0.03428	0.03413
		160	0.07503	0.07728	0.06513
		180	0.03911	0.03763	0.07599
		200	0	0	0
MODEL -4 **REACTOR HAVING STRAIGHT BAFFLES WITH 15 mm STRAIGHTENER**	**CHAMBER-3**	200	0	0	0
		220	0.04368	0.01107	0.01325
		240	0.07176	0.01207	0.01909
		260	0.07409	0.06757	0.04514
		280	0.03519	0.04701	0.05365
		300	0	0	0
	CHAMBER-4	300	0	0	0
		320	0.02679	0.06984	0.01880
		340	0.07649	0.02253	0.03363
		360	0.07043	0.06333	0.03003
		380	0.04365	0.06219	0.06229
		400	0	0	0

CHAPTER-5

RESULTS AND DISCUSSION

In this chapter results in the form of velocity vectors, streamlines, velocity contours, turbulent kinetic energy and dead zones for all four models are presented for two different velocity magnitudes.

DEADZONES: from below the pictorial representations of results, we can see that **dead zones are minimum in model 1st** (reactor with a simple angled baffles**) and model 4th** (reactors having baffles with 15 mm horizontal straighteners). Also, model 3rd shows the worst result in the case of dead zones which means there is a recirculating zone creating inside the tank. But as the velocity magnitude is increased dead zones are also increasing in all four models. Model 1st and 4th obviously having a minimum in that case also.

STREAMLINES: In the case of streamlines also, in model 1st and in model 4th **streamlines are covering a maximum area** of the chamber which maximum effective volume of the reactor is used by flowing flow. Since flow in these two models showing the most blended path and hence showing minimum dead zones and short circuiting near baffles. But in model 2nd and in model 3rd streamlines covering a minimum area of the reactor and hence dead zones and short circuiting are increasing in these two chambers.

VELOCITY VECTORS: In this case also model 1st and model 4th are more appropriate in regard to up-flow velocity which should be less than the settling velocity in the chamber. As we can see that in model 1st and in model 4th velocity **vectors at the right of the chamber are very light in nature** which means the flow is intermediate flow i.e. flow is in between plug flow and completely mixed flow. But in other two models i.e. in 2nd and 3rd, velocity vectors in right of each chamber are slightly dark which means up-flow velocity is greater than the specific value.

VELOCITY CONTOURS: Velocity contours are showing different velocity at different points. By obtaining velocity at three points in each chamber, graphs were plotted for all four models showing up flow velocity.

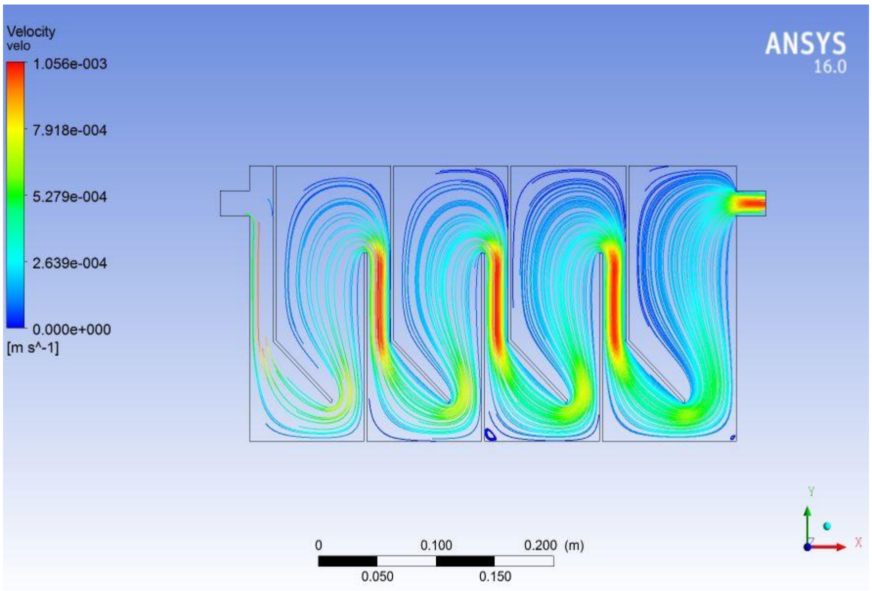


Fig. 5.1 <u>Dead zones for model-1 at 0.07 cm/s</u>

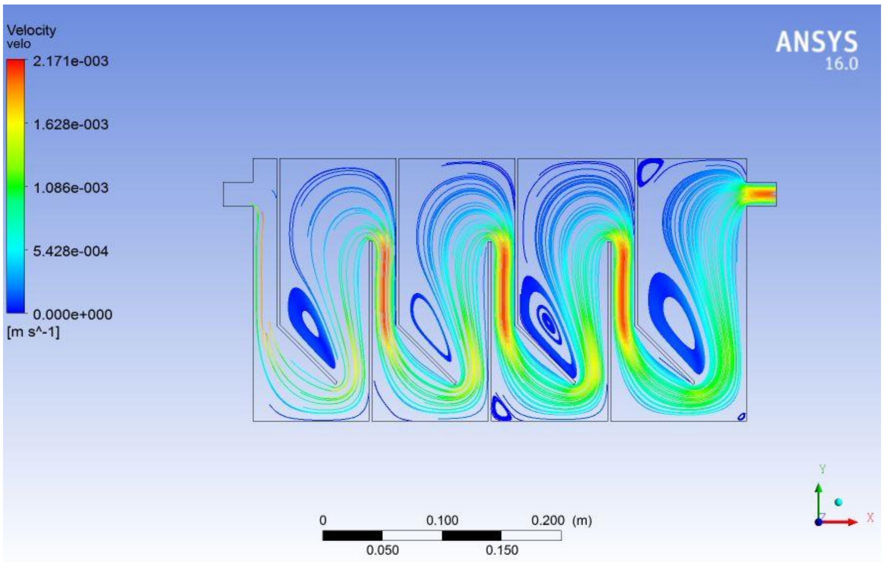


Fig. 5.2 <u>Dead zones for model-1 at 0.14 cm/s</u>

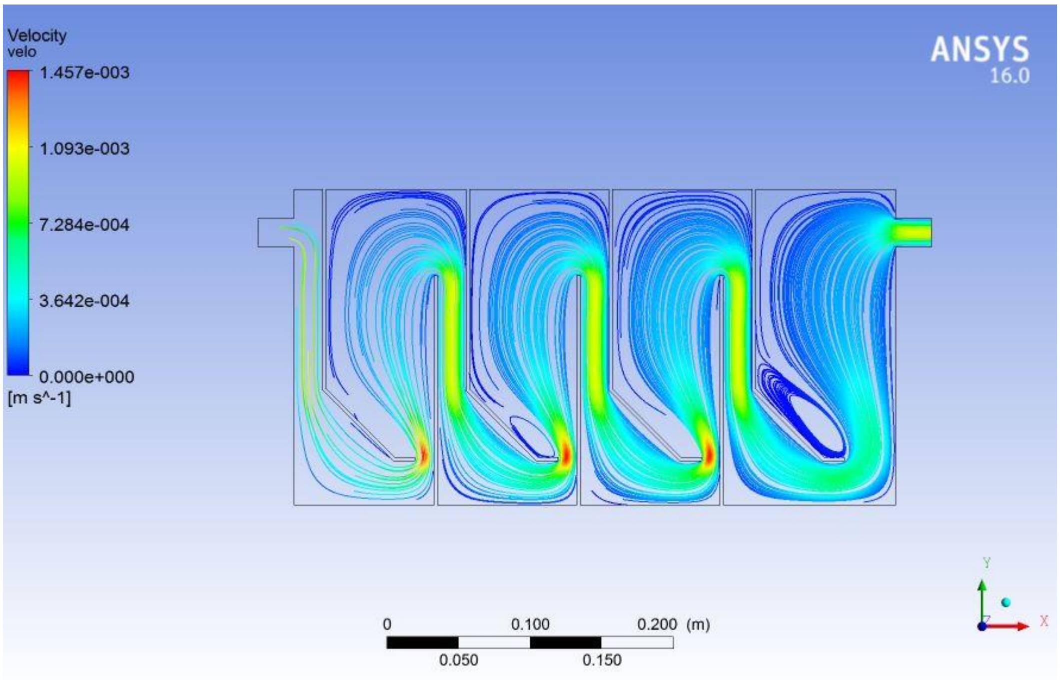


Fig. 5.3 <u>Dead zones for model-2 at 0.07 cm/s</u>

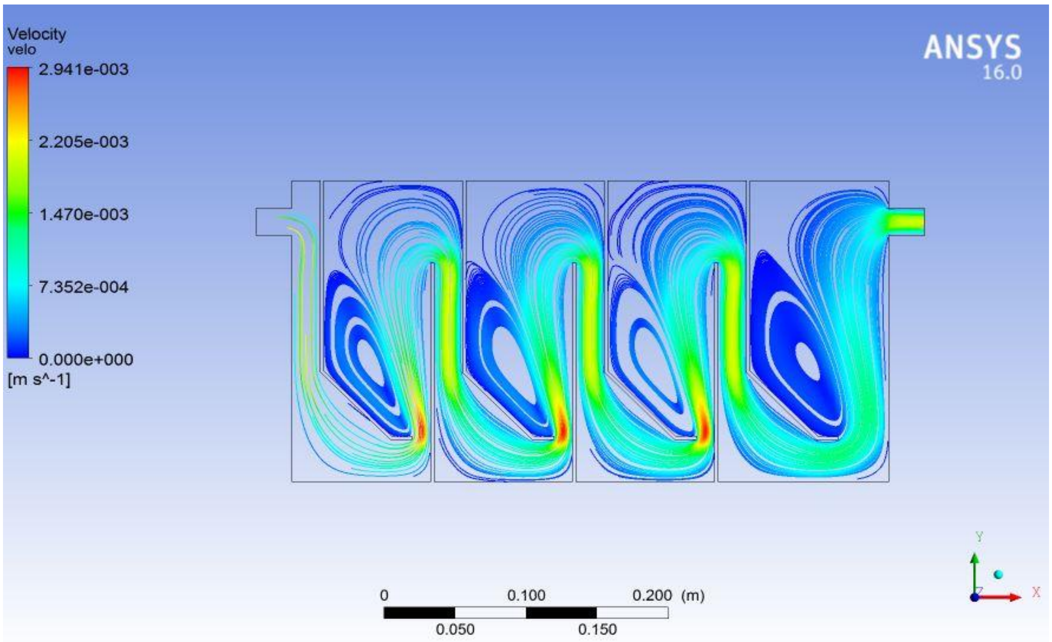


Fig. 5.4 <u>Dead zones for model-2 at 0.14 cm/s</u>

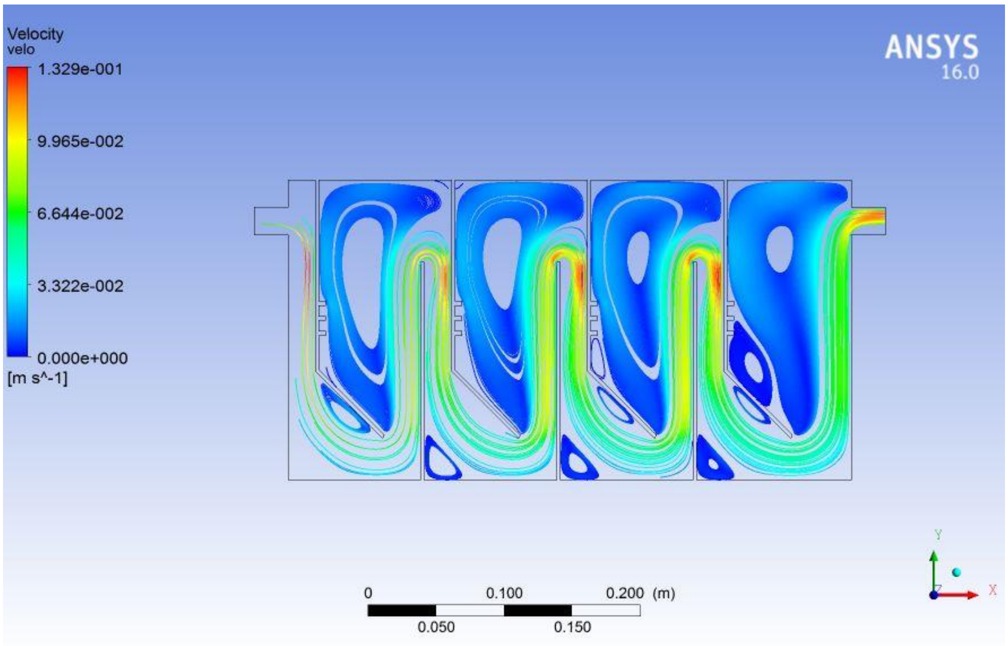


Fig. 5.5 **Dead zones for model-3 at 0.07 cm/s**

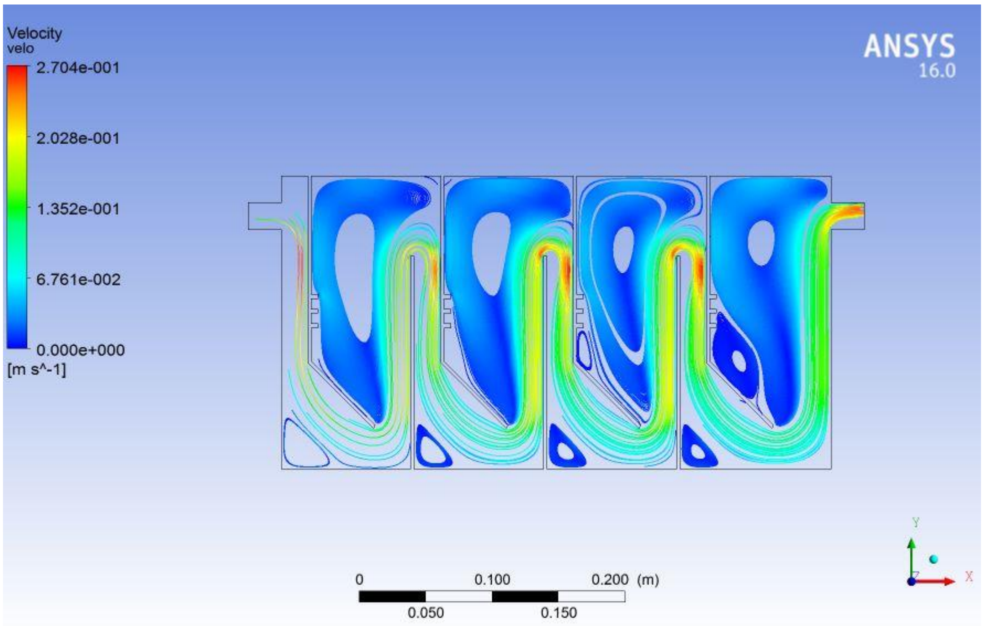


Fig. 5.6 **Dead zones for model-3 at 0.14 cm/s**

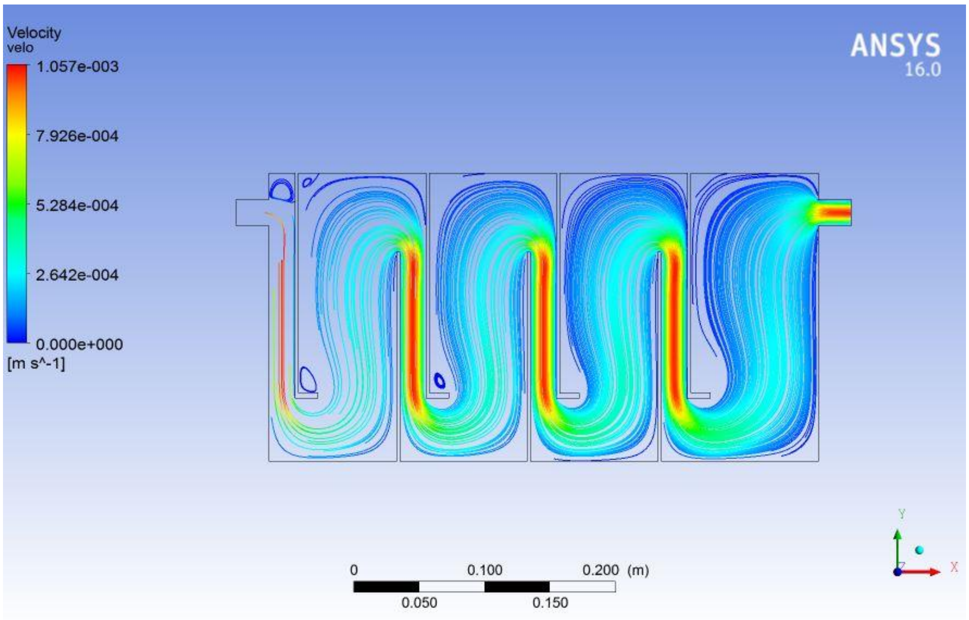


Fig. 5.7 <u>Dead zones for model-4 at 0.07 cm/s</u>

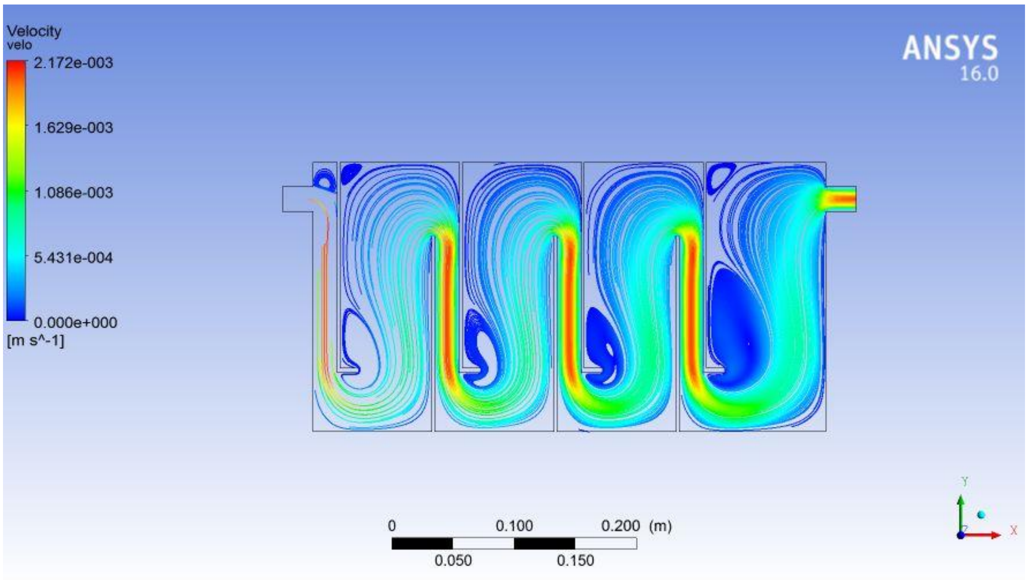


Fig. 5.8 <u>Dead zones for model-4 at 0.14 cm/s</u>

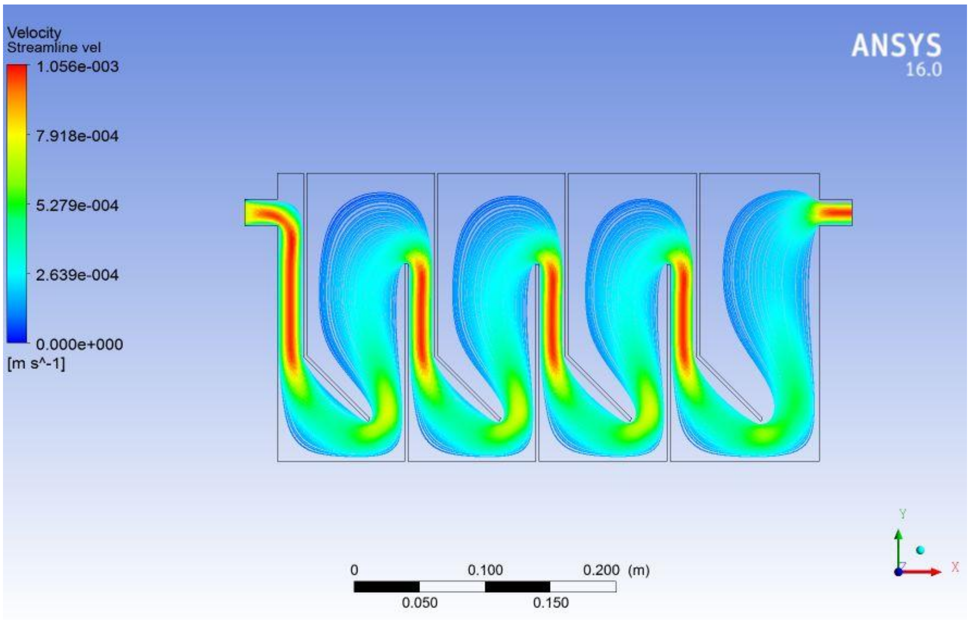


Fig. 5.9 **<u>Streamlines for model-1 at 0.07 cm/s</u>**

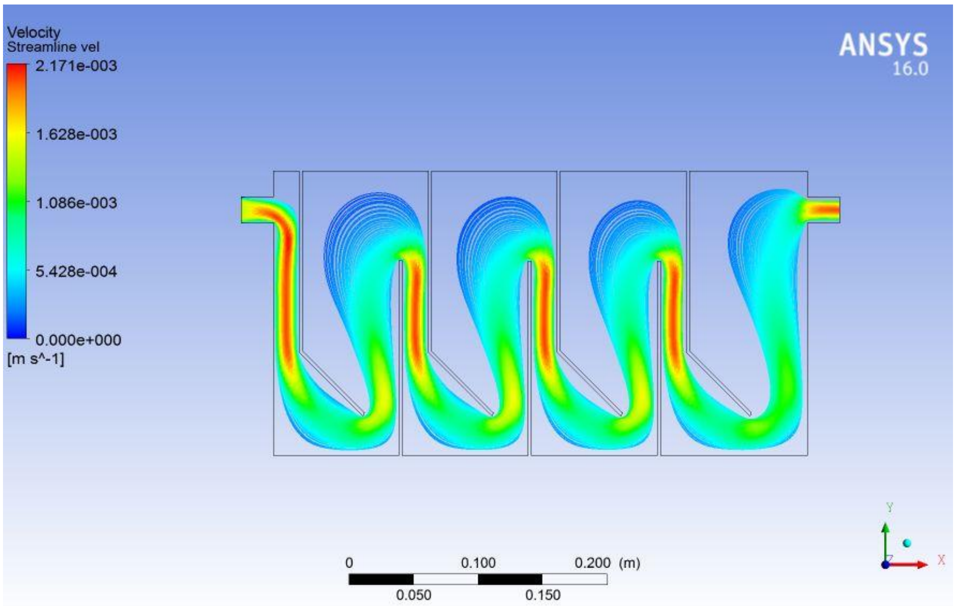


Fig. 5.10 **<u>Streamlines for model-1 at 0.14 cm/s</u>**

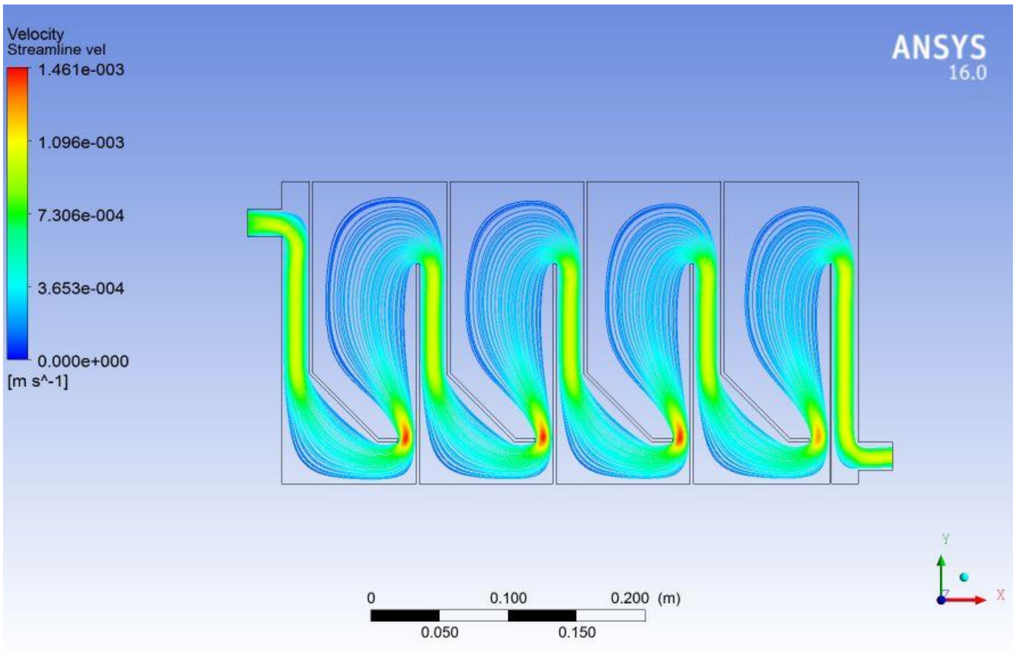


Fig. 5.11 Streamlines for model-2 at 0.07 cm/s

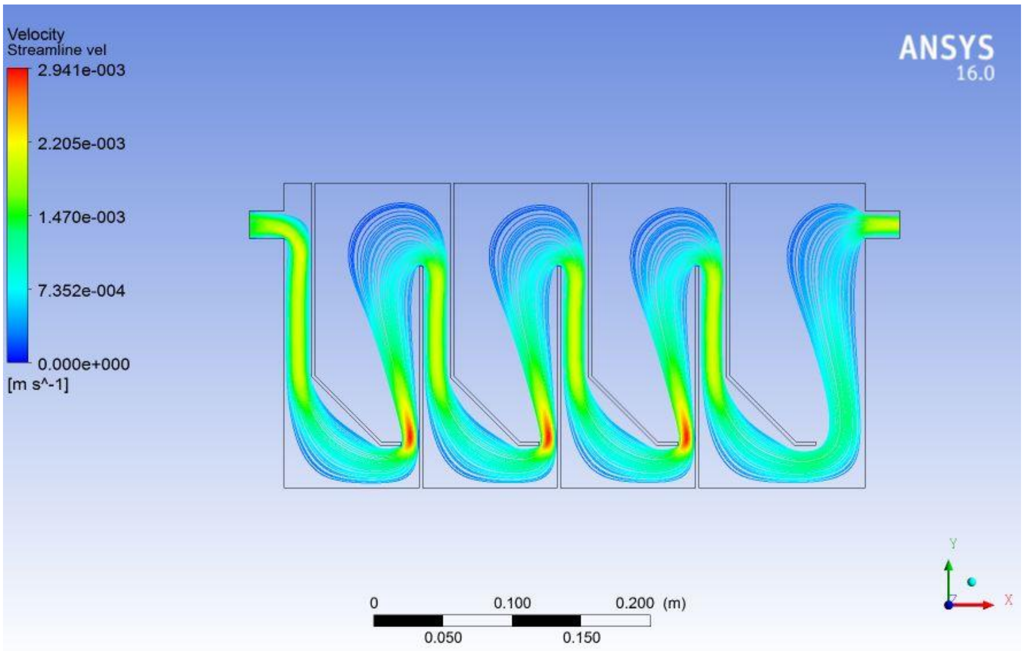


Fig. 5.12 Streamlines for model-2 at 0.14 cm/s

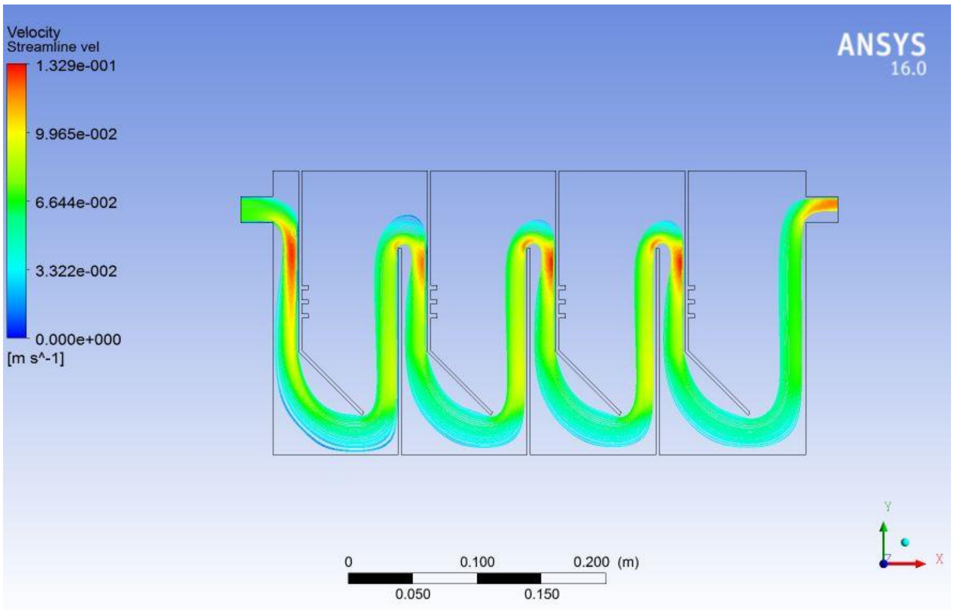


Fig. 5.13 <u>**Streamlines for model-3 at 0.07 cm/s**</u>

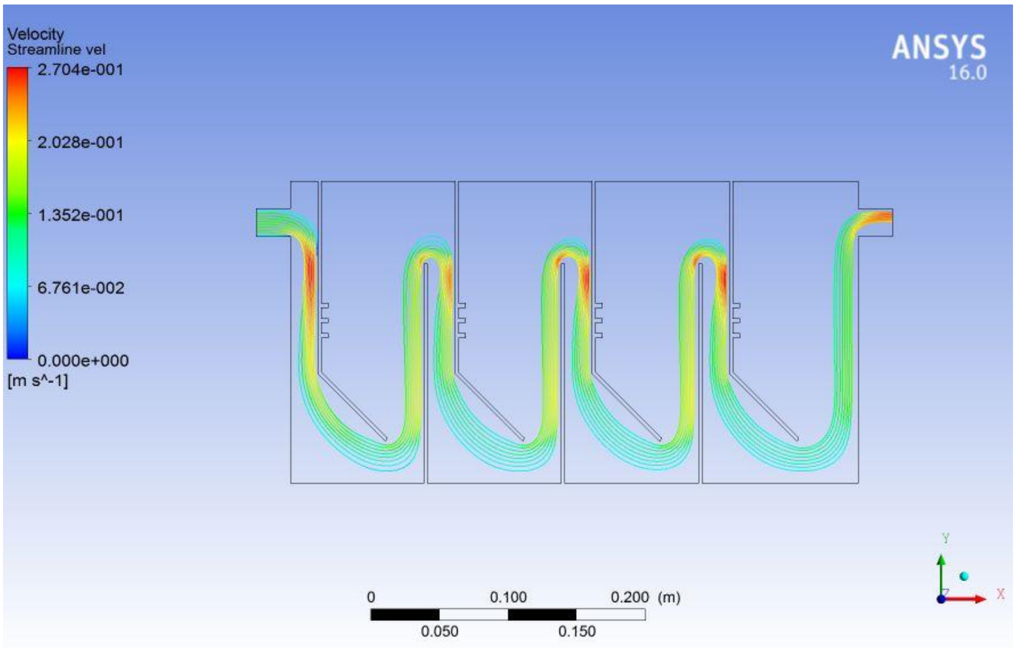


Fig. 5.14 <u>**Streamlines for model-3 at 0.14 cm/s**</u>

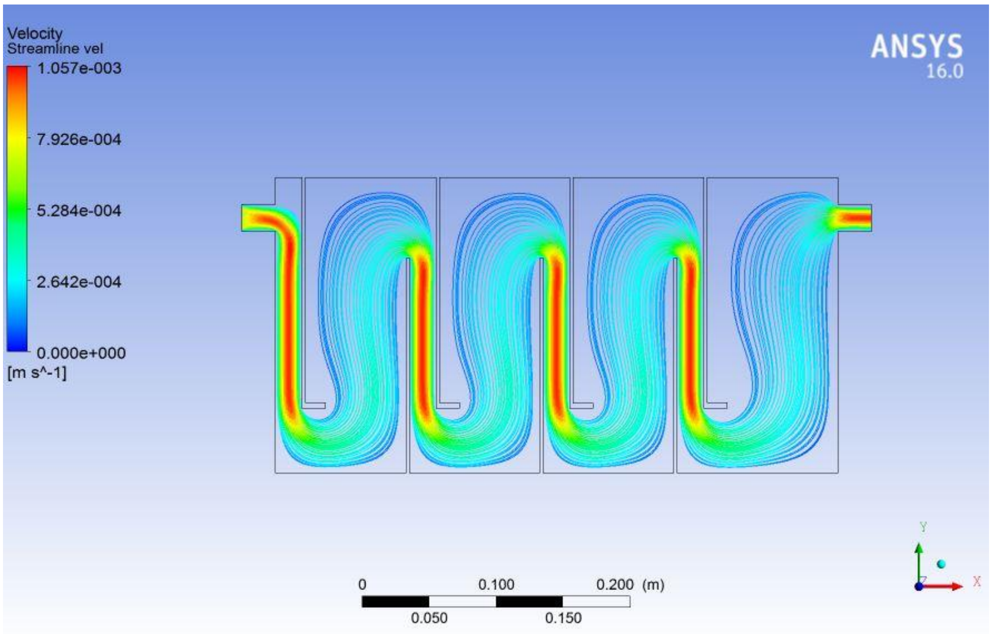


Fig. 5.15 Streamlines for model-4 at 0.07 cm/s

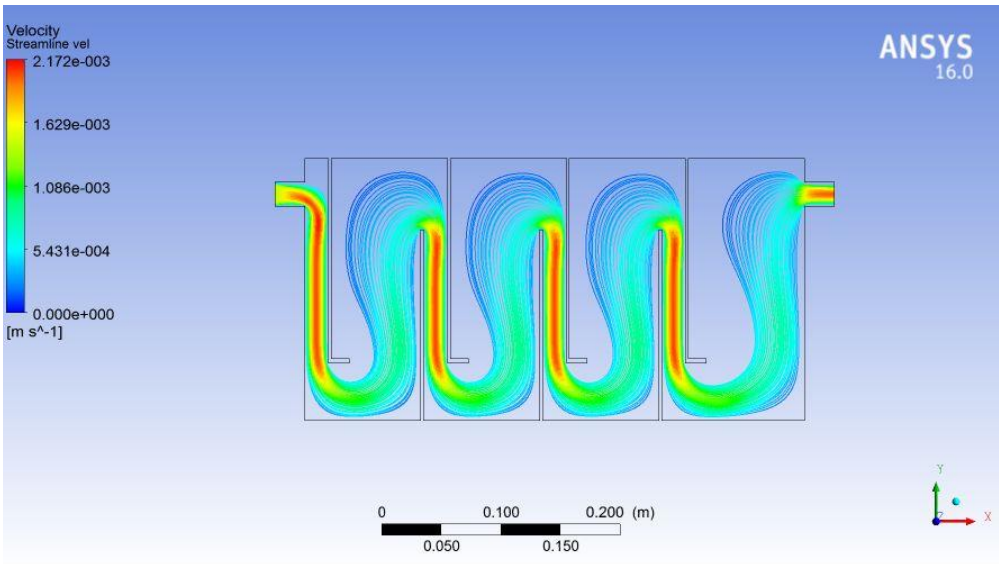


Fig. 5.16 Streamlines for model-4 at 0.14 cm/s

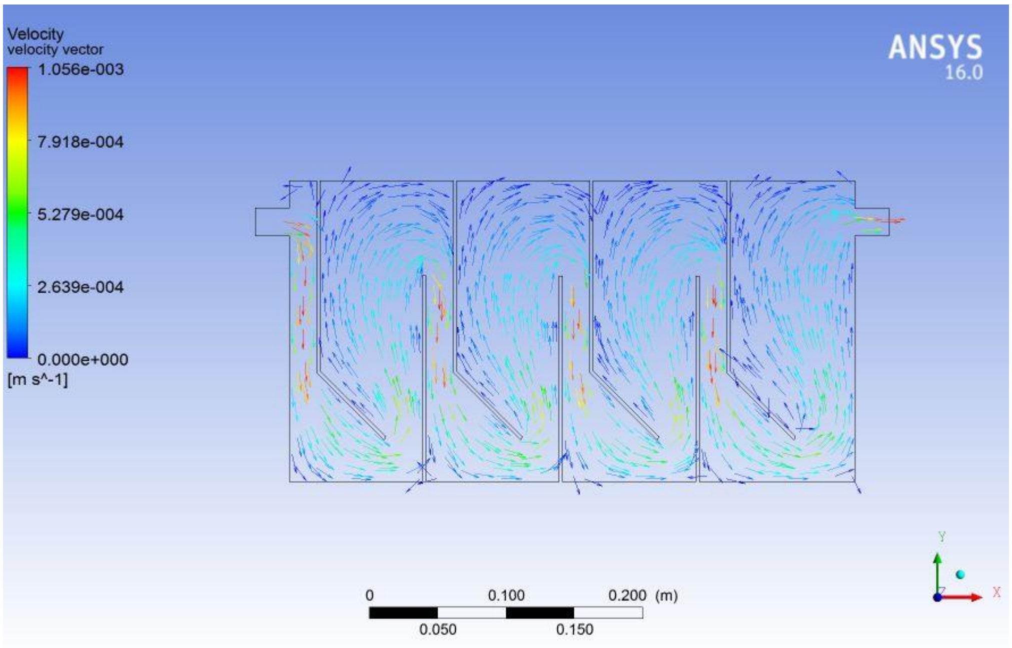


Fig 5.17 **Velocity vectors for model-1 at 0.07 cm/s**

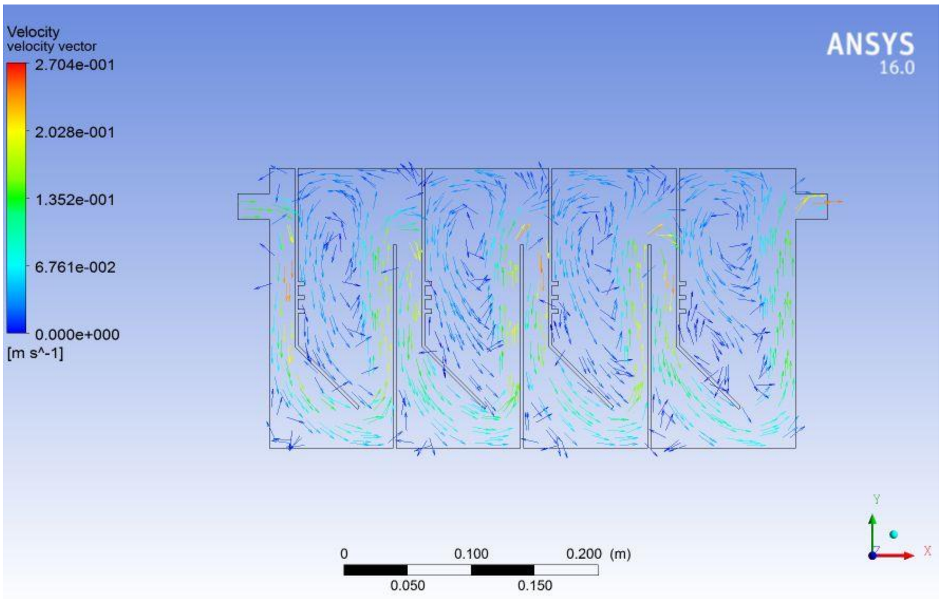


Fig. 5.18 **Velocity vectors for model-1 at 0.14 cm/s**

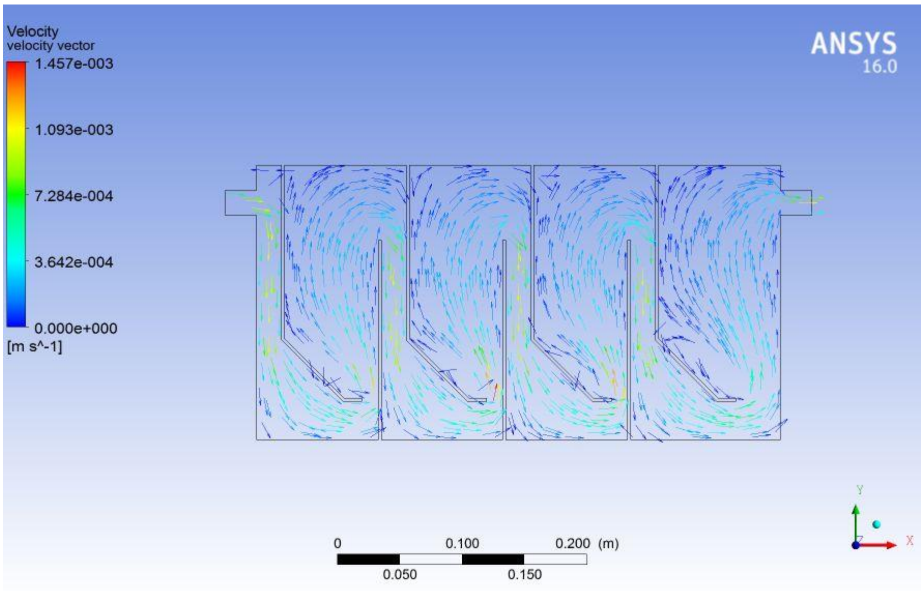


Fig. 5.19 Velocity vectors for model-2 at 0.07 cm/s

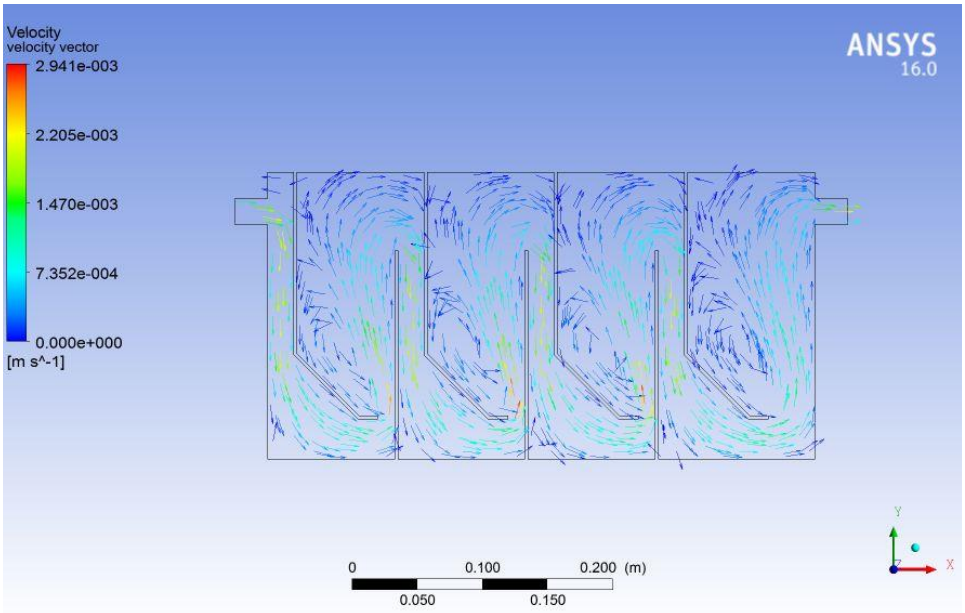


Fig. 5.20 Velocity vectors for model-2 at 0.14 cm/s

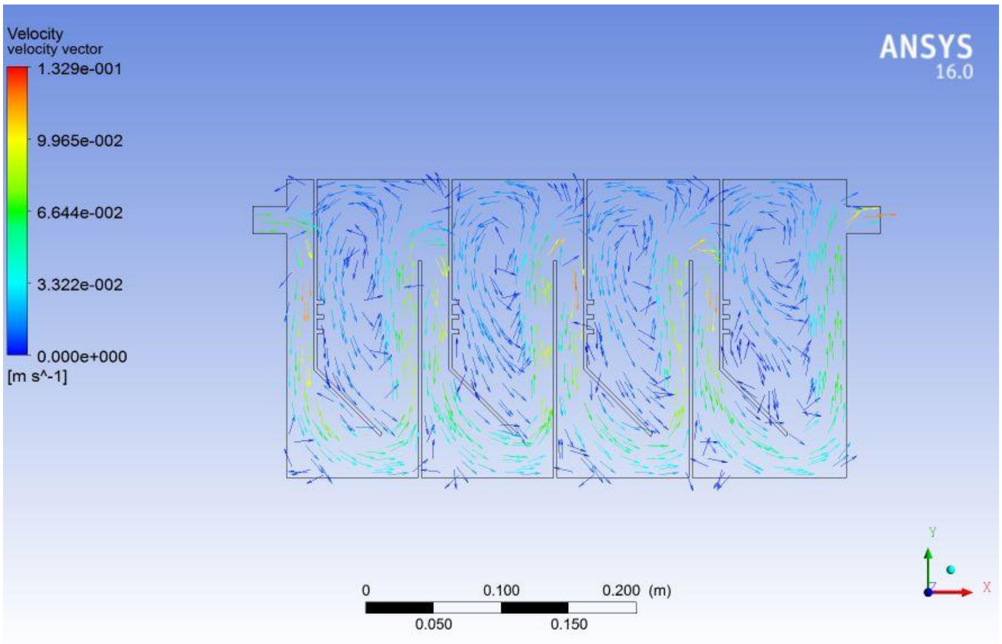


Fig. 5.21 Velocity vectors for model-3 at 0.07 cm/s

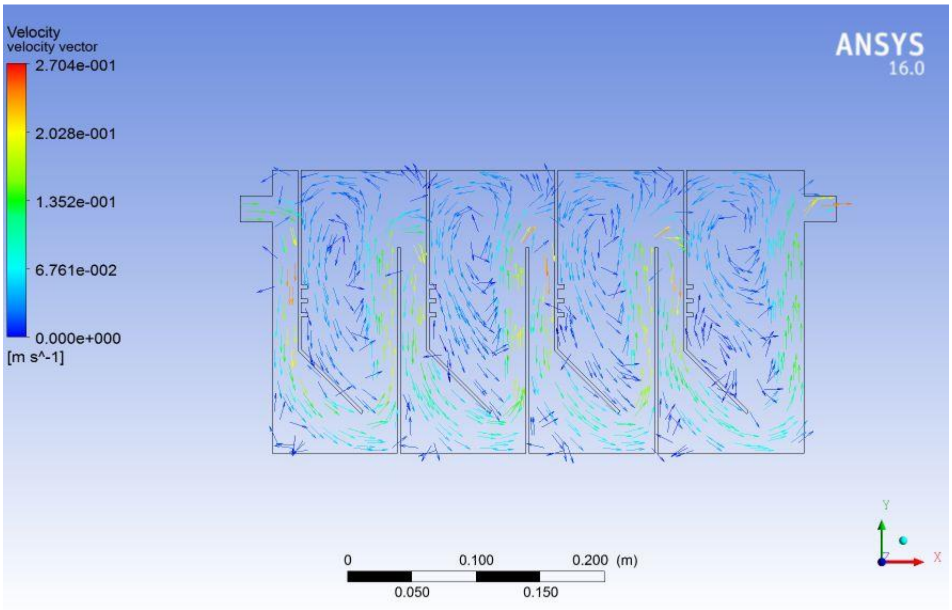


Fig. 5.22 Velocity vectors for model-3 at 0.14 cm/s

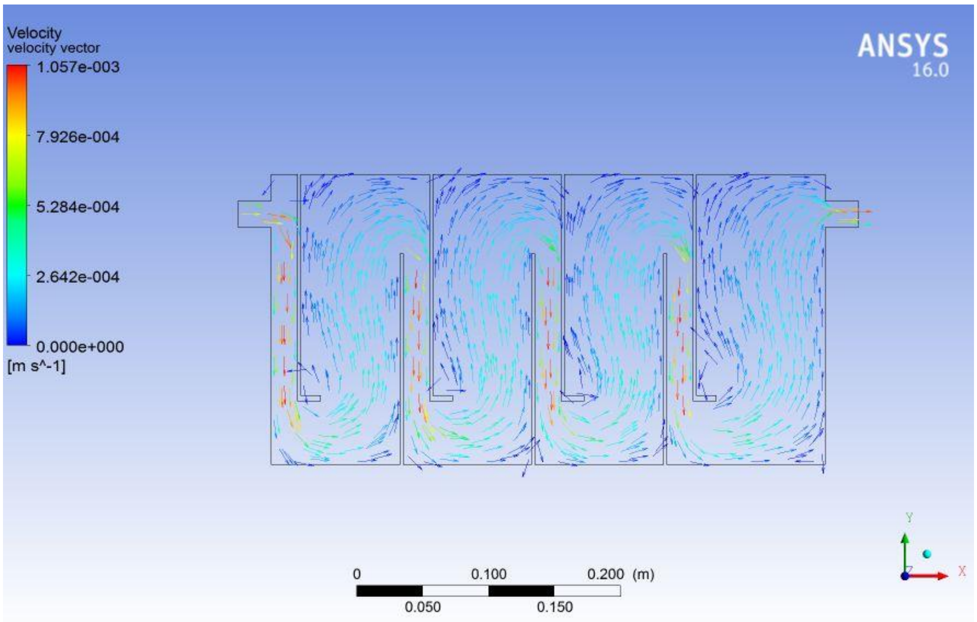


Fig. 5.23 <u>Velocity vectors for model-4 at 0.07 cm/s</u>

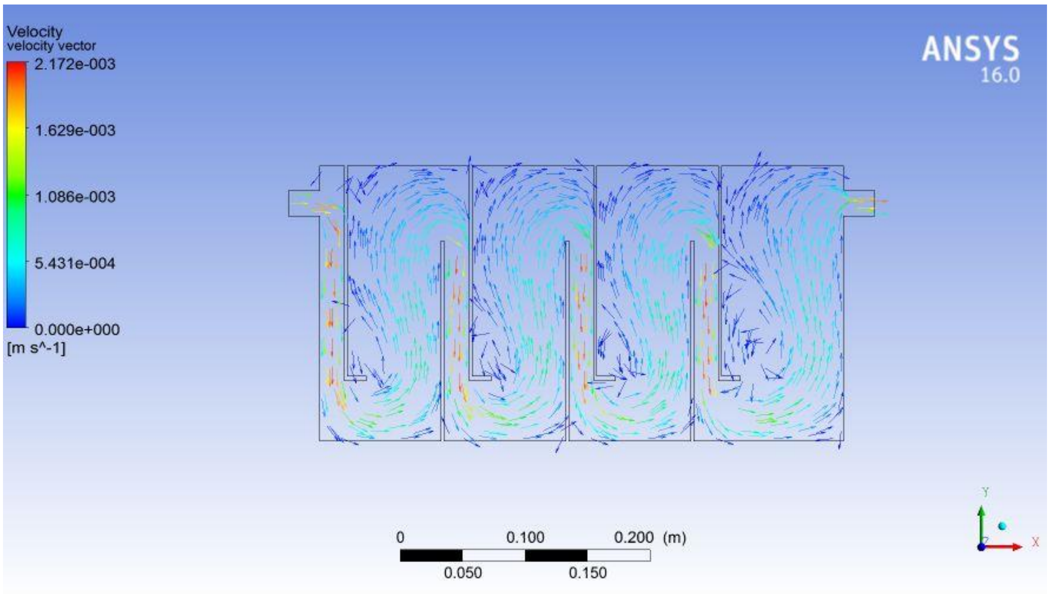


Fig. 5.24 <u>Velocity vectors for model-4 at 0.14 cm/s</u>

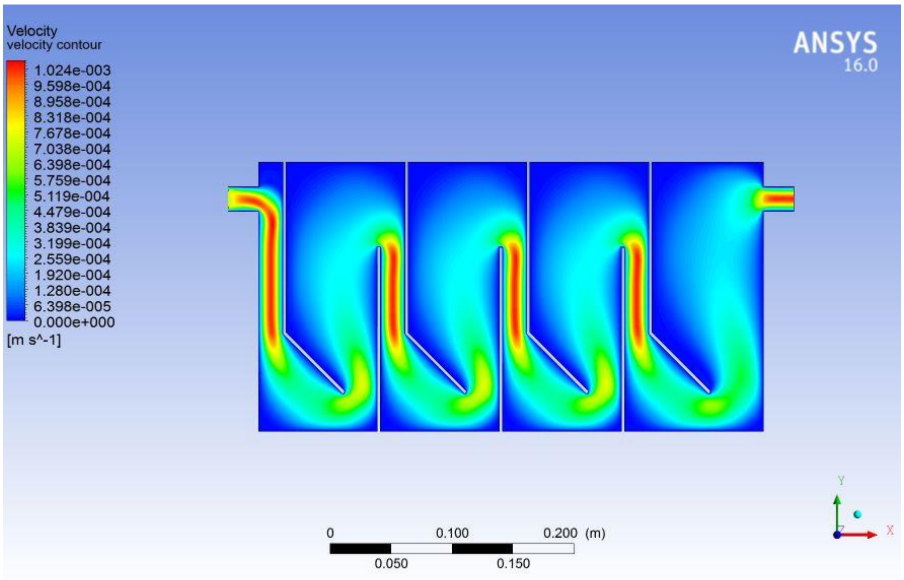


Fig. 5.25 **<u>Velocity contours for model-1 at 0.07 cm/s</u>**

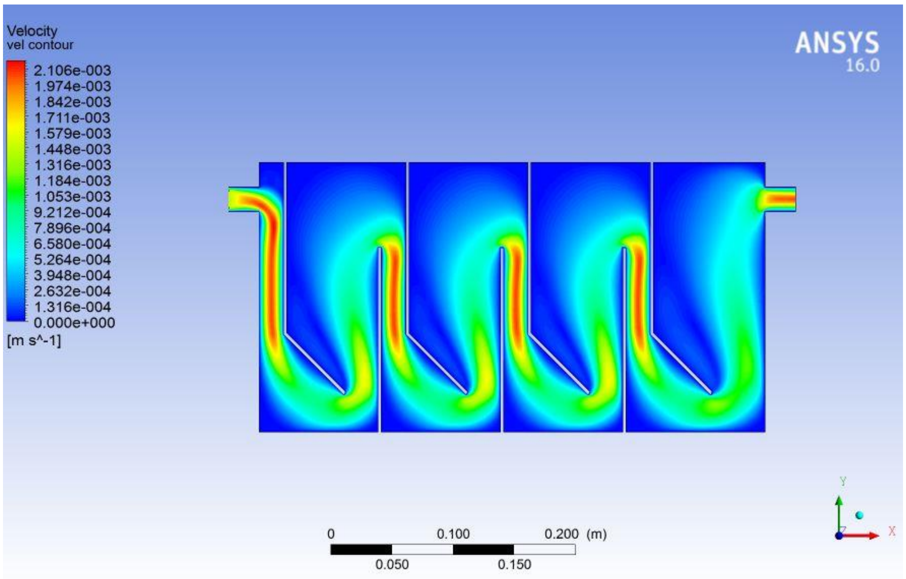


Fig. 5.26 **<u>Velocity contours for model-1 at 0.14 cm/s</u>**

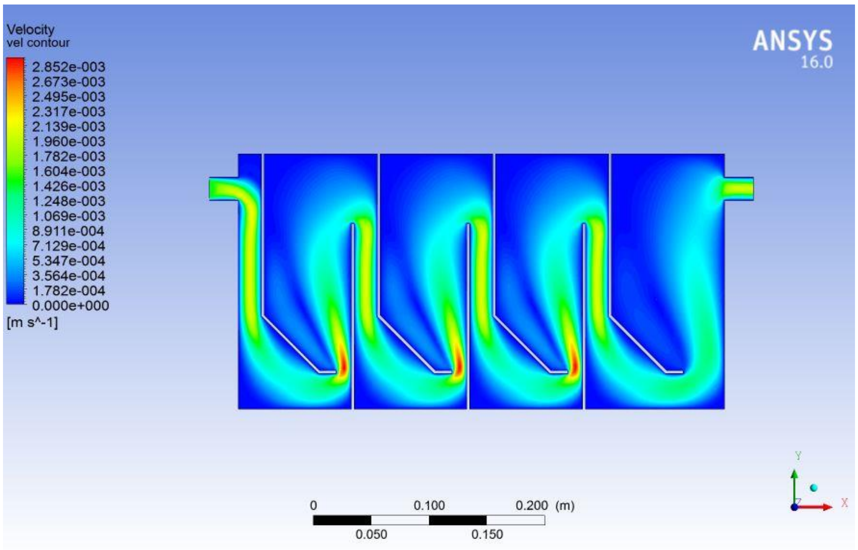


Fig. 5.27 Velocity contours for model-2 at 0.07 cm/s

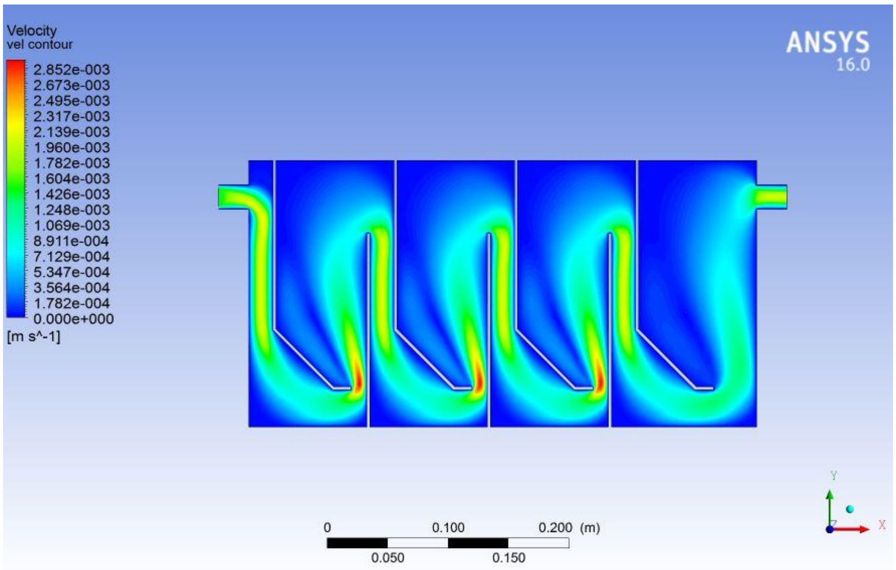


Fig. 5.28 Velocity contours for model-2 at 0.14 cm/s

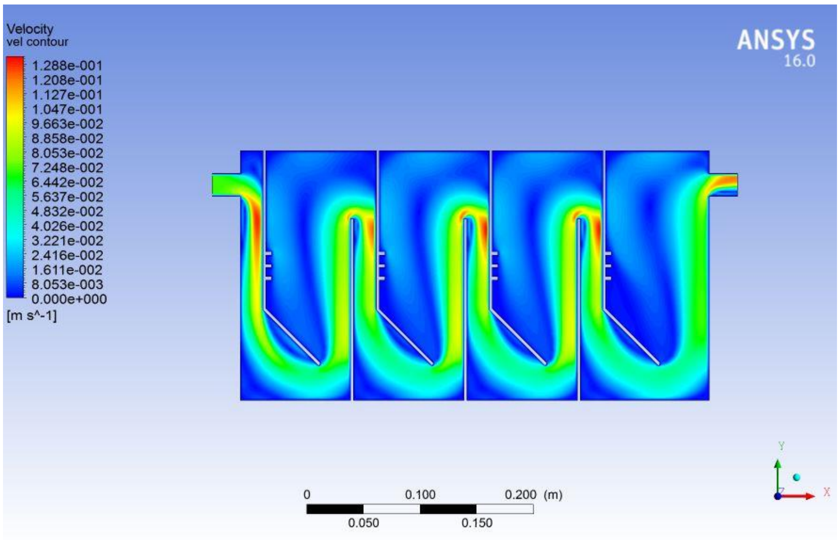


Fig. 5.29 **Velocity contours for model-3 at 0.07 cm/s**

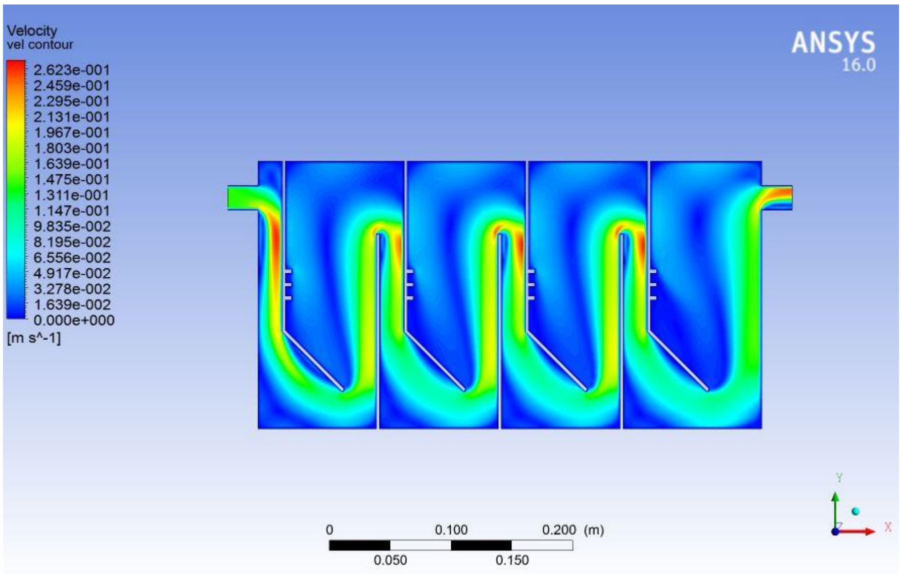


Fig. 5.30 **Velocity contours for model-3 at 0.14 cm/s**

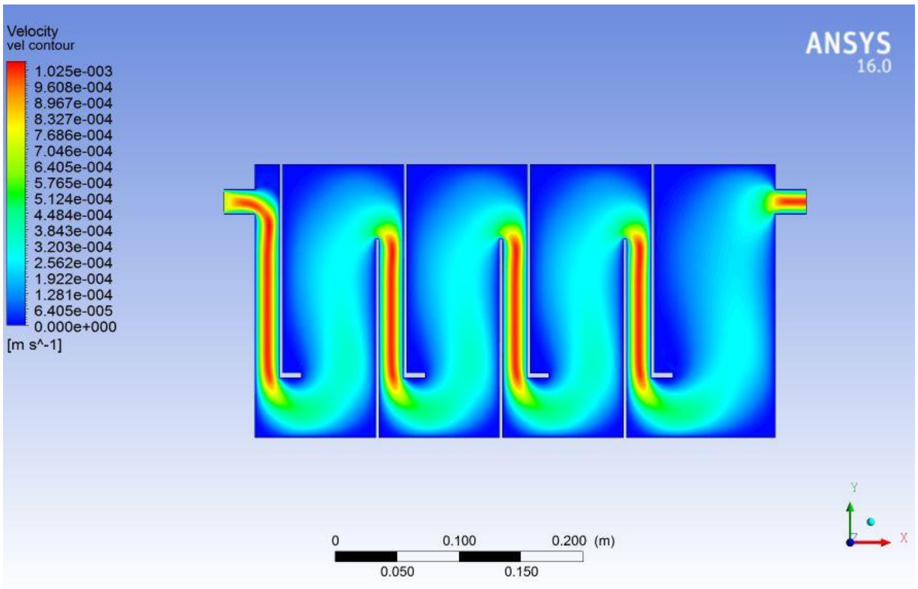


Fig. 5.31 <u>Velocity contours for model-4 at 0.07 cm/s</u>

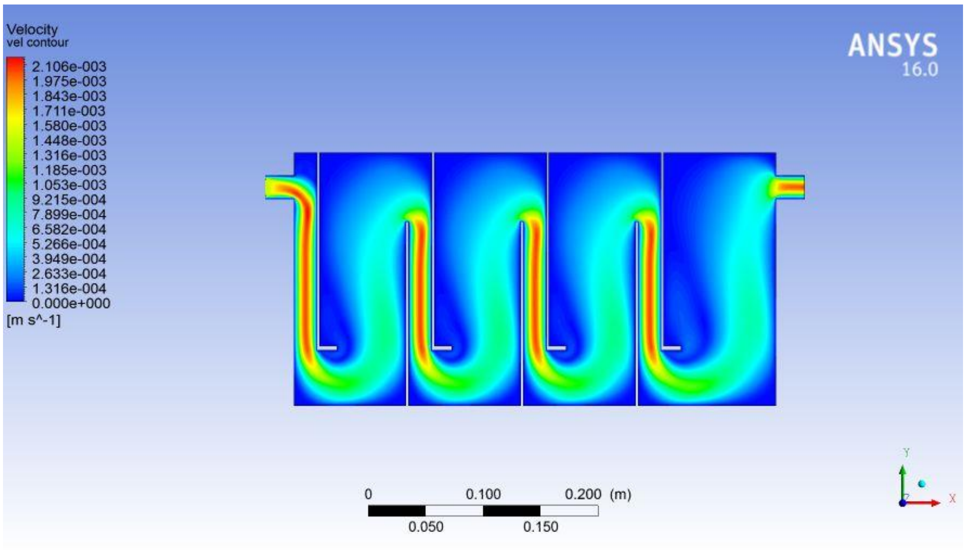
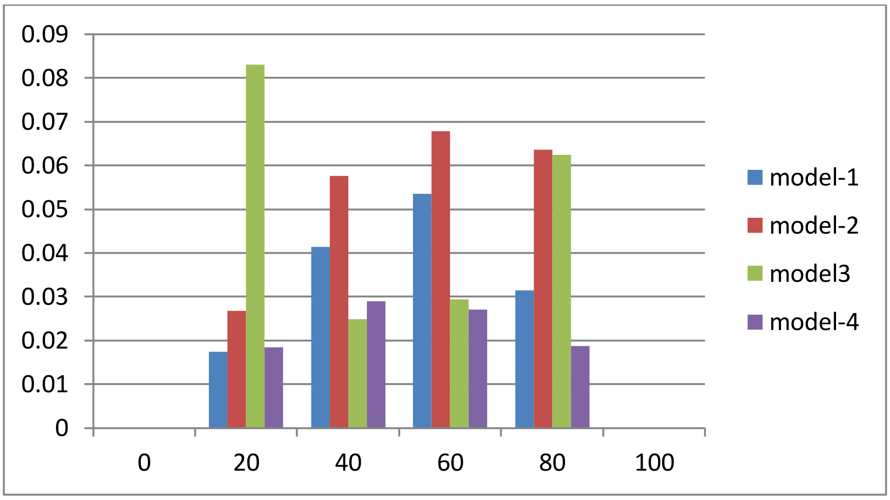
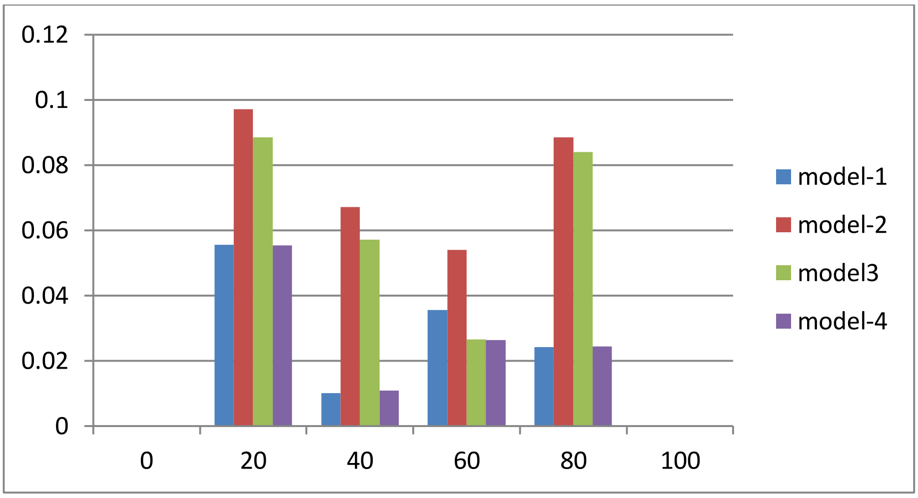


Fig. 5.32 <u>Velocity contours for model-4 at 0.14 cm/s</u>

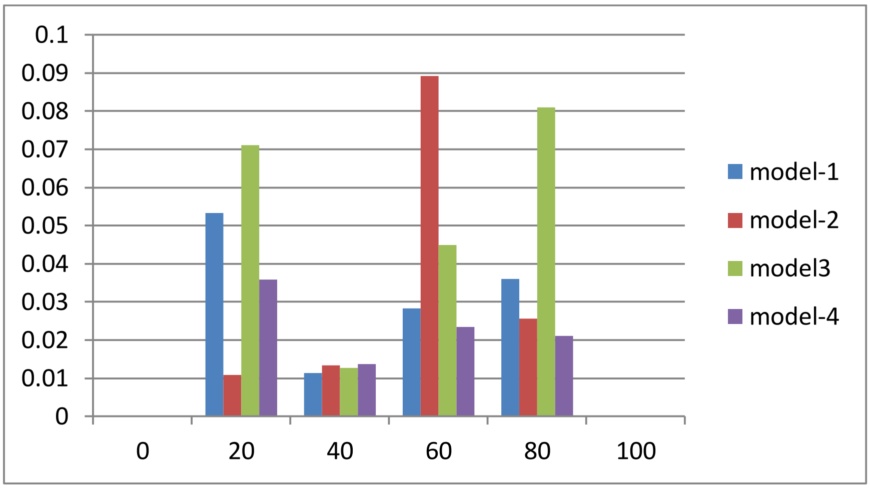
Graphs plotted for all four models showing up-flow velocity at 0.07 velocity magnitude. From the graphs, we can see that in model 1st and model 4th, up-flow velocity in each chamber is within the specific range i.e. 0.0555 cm/s. hence these two models are more appropriate than other two because according to previous studies up-flow velocity should be less than the specific range (settling velocity) so that should particles should not wash out of the reactor.



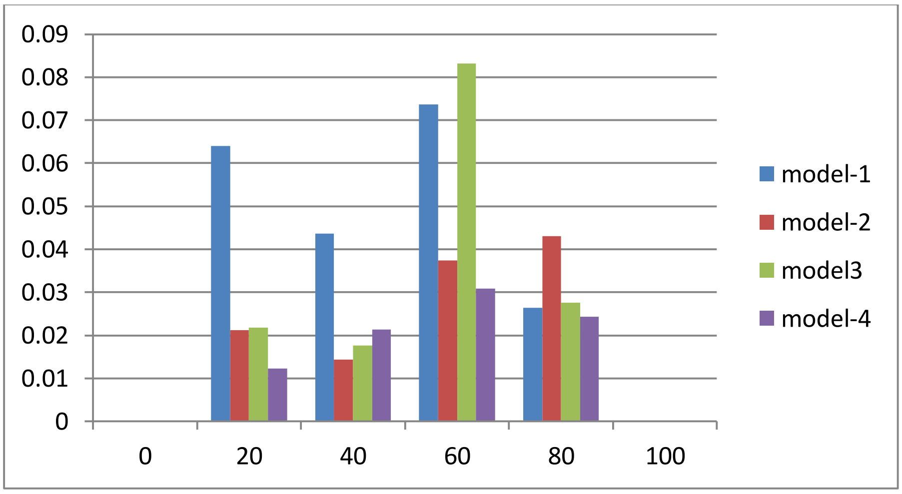
Graph-1 **<u>plotted for 0.07cm/s at bottom of chamber 1</u>**



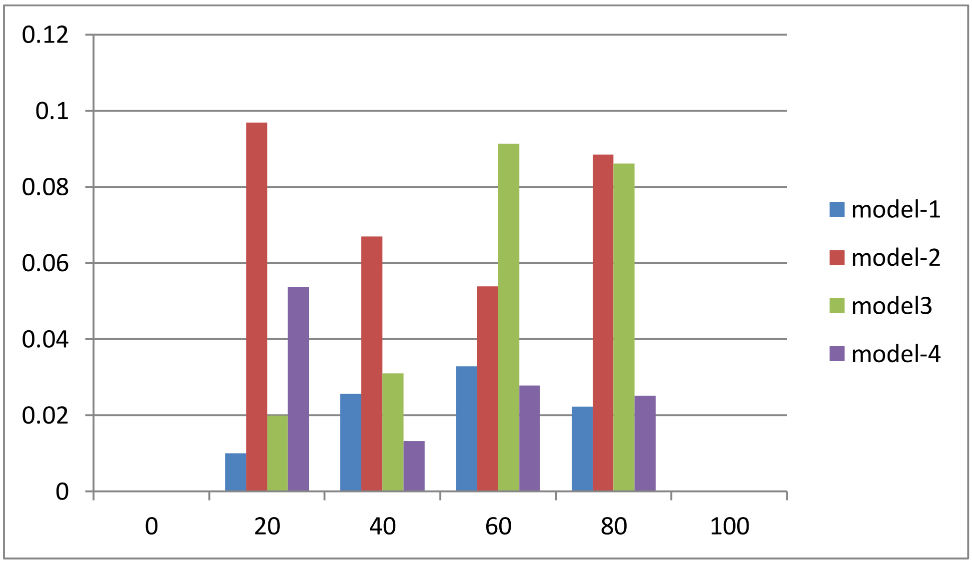
Graph-2 **<u>plotted for 0.07cm/s at mid of chamber 1</u>**



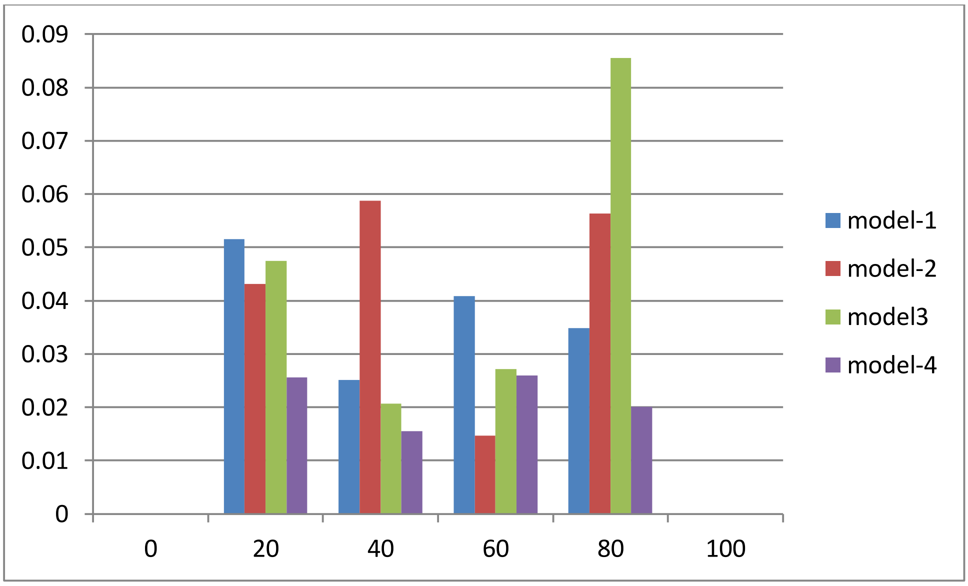
Graph-3 **<u>plotted for 0.07cm/s near upper surface of chamber 1</u>**



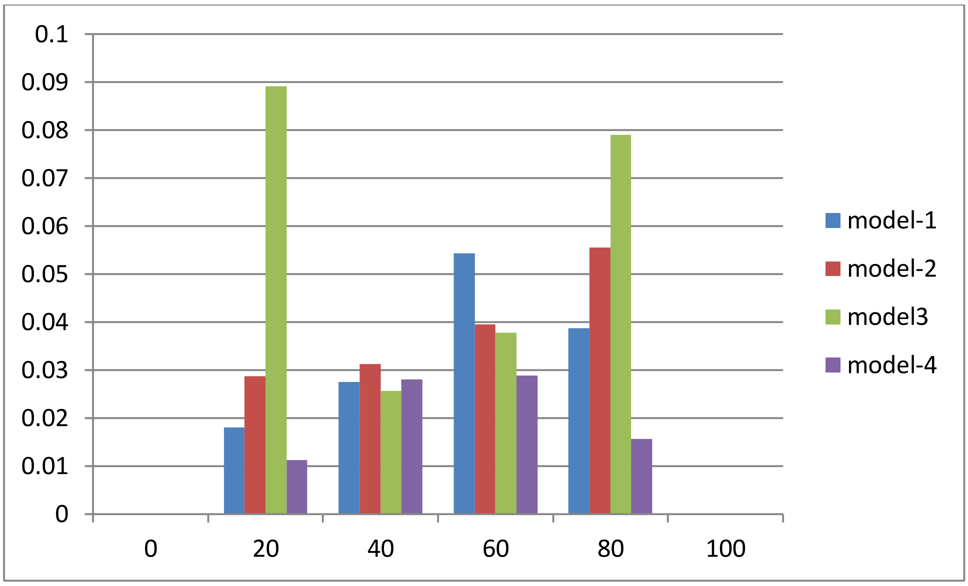
Graph-4 **<u>plotted for 0.07cm/s at bottom of chamber 2</u>**



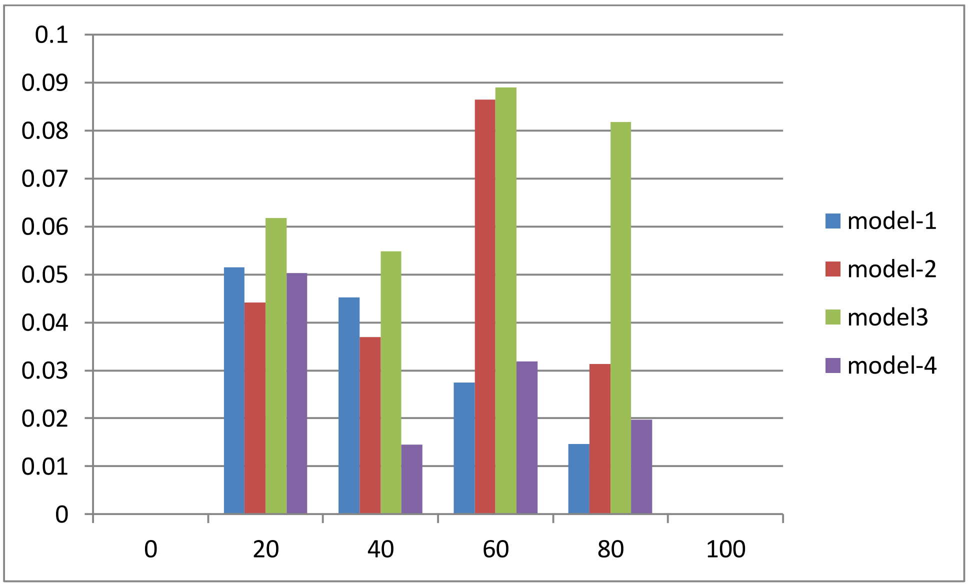
Graph-5 plotted for 0.07cm/s at mid of chamber 2



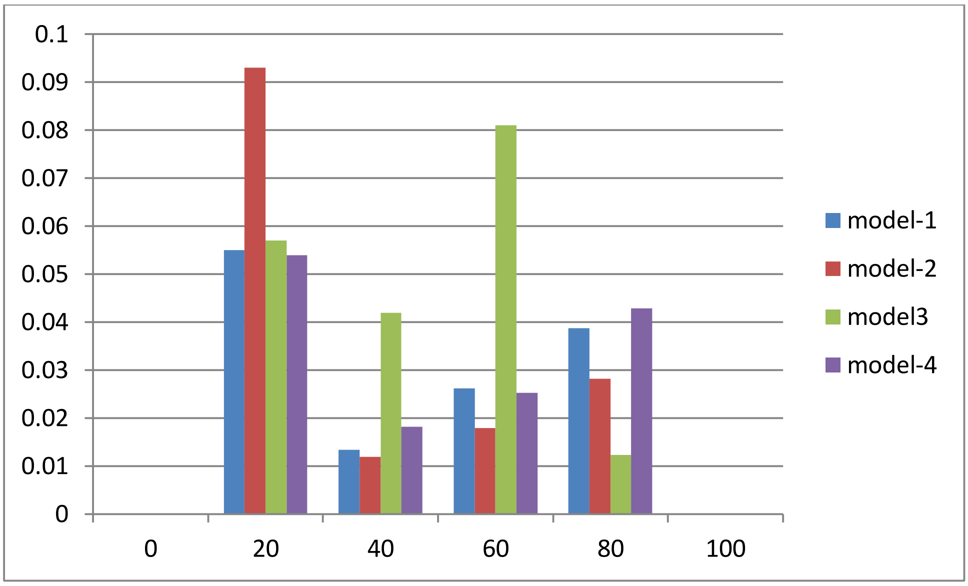
Graph-6 plotted for 0.07cm/s near upper surface of chamber 2



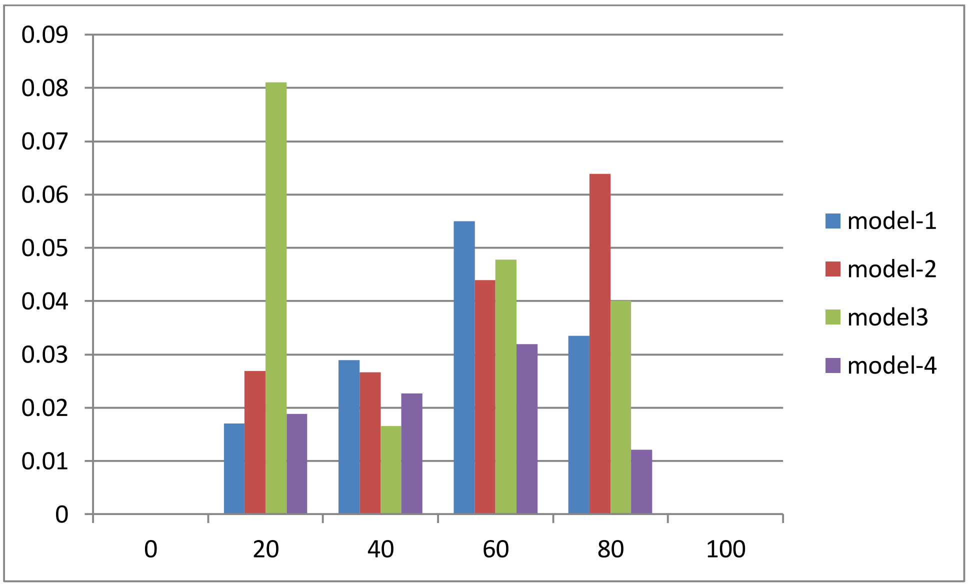
Graph-7 plotted for 0.07cm/s at bottom of chamber 3



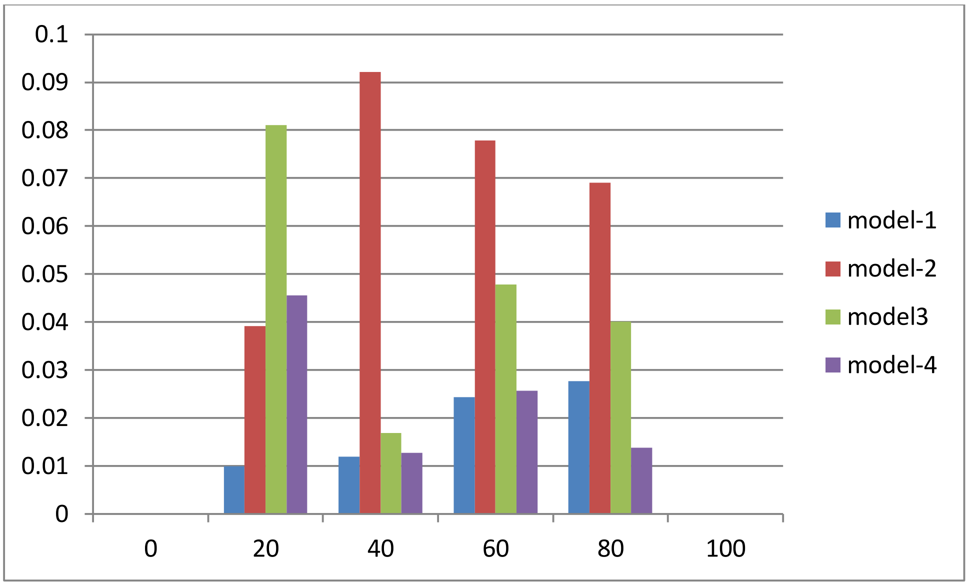
Graph-8 plotted for 0.07cm/s at mid of chamber 3



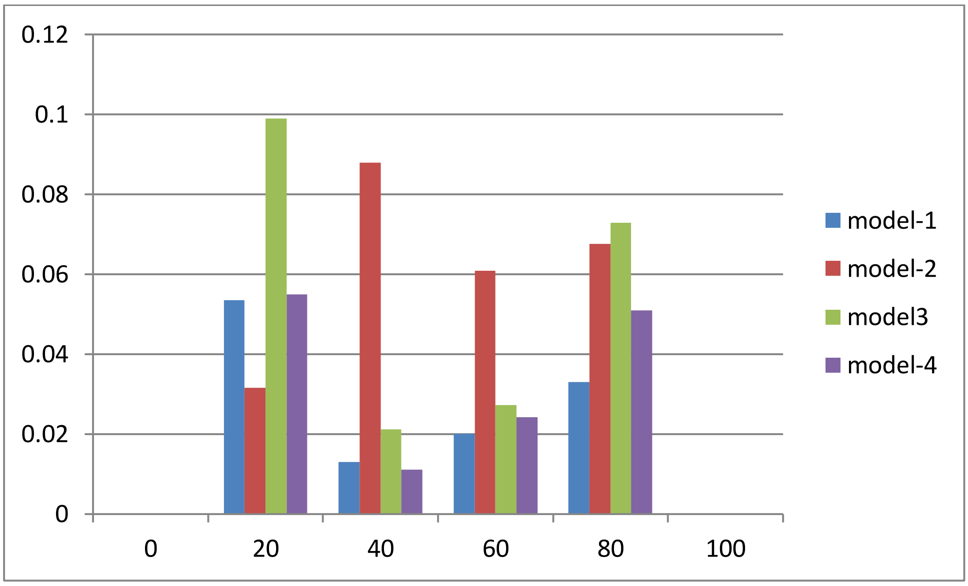
Graph-9 plotted for 0.07cm/s near upper surface of chamber 3



Graph-10 plotted for 0.07cm/s at bottom of chamber 4



Graph-11 plotted for 0.07cm/s at mid of chamber 4



Graph-12 plotted for 0.07cm/s near upper surface of chamber 4

CHAPTER-6

CONCLUSION

1. To minimize the dead zones and short circuiting in bio-reactors such an anaerobic baffled reactors, one should use appropriate configuration and structure of baffles. For maximum efficiency of reactor, angle of hanging baffles should be 45º.

2. Model 1^{st} and model 4^{th} gives the better results i.e. better mixing and minimum dead zones and short circuiting. Although by increasing velocity magnitude dead zones were increasing in all models but minimum in model 1^{st} and model 4^{th}.

3. Model-1st and model-4th can be used as full-scale plan project as these two are giving better results.

4. Numerical approaches were carried out to observe the effects of differently modified baffles on the flow field. Using CFD and VOF methods, we can develop a numerical simulation of flow in the anaerobic baffled reactor through the ANSYS fluent.

5. From the results, it is concluded that if we use baffles with a horizontal straightener in anaerobic baffled reactors then it will give better results in respect of dead zones and better mixing.

6. It is observed that by increasing velocity magnitude, dead zones were also increasing in all cases.

7. Number of baffles can also be increased for getting better results but for that one needs to change the dimensions of geometry.

8. From graph plotted it in concluded that in model 1st and model 4th, up flow velocity is within the specific range i.e. 0.0555 cm/s. hence modification done in these two models showing better results than other two.

9. It was also concluded that if the hanging baffles placed at the centre of each chambers then results will be opposite in respect of present study but that would we inappropriate since the down flow to up flow ratio of 1:3 gives the most suitable pattern(P. Dama et al.;2000)

Future scope of study: Present study can be useful in further future improvement in the design of full-scale anaerobic baffled reactors. By changing reactors dimensions, number of baffles can also be increased in model-1 and model-2 and shall be develop on large scale.

REFERENCES

1. P Dama, J Bell, C J Brouckaert, C A Buckley and D C Stuckey1 **"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.

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

3. M. Shahrokhi, F. Rostami, M.A. Md Said, S. Syafalni; (2001); **Numerical Investigation of Baffle Effect on the Flow in a Rectangular Primary Sedimentation** Tank; World Academy of Science, Engineering and Technology International Journal of Environmental, Chemical, Ecological, Geological and Geophysical Engineering Vol:5, No:10, 2011.

4. Alexandra Martínez Mendoza , Tatiana Montoya Martínez , Vicente Fajardo Montañana , P. Amparo López Jiménez; 2011; **Modeling flow inside an anaerobic digester by CFD techniques**. international journal of energy and environment Volume 2, Issue 6, 2011 pp.963-974 Journal homepage: www.IJEE.IEEFoundation.org.

5. Leonardo M. da Rosa, Letícia Pederivaa, Guilherme Z. Maurinaa, Lademir L. Beala, Ana P. Torresb and Maira Sousab; VOL. 38, 2014; **CFD Analysis of the Effect of Baffle Plates on the Fluid Flow in an Anaerobic Sequencing Batch Reactor.**

6. R. Renuka , S. Mariraj Mohan ,S. Amal Raj; 2015**; Hydrodynamic behaviour and its effects on the treatment performance of panelled anaerobic baffle-cum filter reactor**. Int. J. Environ. Sci. Technol. (2016) 13:307–318 .

7. 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.

8. Wei-Kang Qi, Toshimasa Hojo, Yu-You Li; Graduate School of Environmental Studies, Tohoku University, 6-6-06 Aza-Aoba, Aramaki, Aoba-Ku, Sendai 980-8579, Japan; **Hydraulic characteristics simulation of an innovative self-agitation anaerobic baffled reactor (SA-ABR)**

9. Mohit yadav; Delhi technological university;2015; **Computational fluid dynamics analysis of 3D multi baffled rectangular chamber using ANSYS fluent.**