# SIMULATION OF STATIC PRESSURE, VELOCITY MAGNITUTE AND SAND VOLUME FRACTION IN AGITATED SLURRIES USING CFD TECHNIQUE

**A Project Report**

**Submitted by**

**S.Karthikeyan 71205402004**

K C T LIBRARY Coimbatore - 641 006

P-2074

*in partial fulfillment for the award of the degree*
of

## Master of Engineering
in
## CAD/CAM

**DEPARTMENT OF MECHANICAL ENGINEERING**

**KUMARAGURU COLLEGE OF TECHNOLOGY**

**COIMBATORE – 641 006**

ANNA UNIVERSITY: CHENNAI 600 025

**JUNE– 2007**

# ANNA UNIVERSITY:: CHENNAI 600 025

## BONAFIDE CERTIFICATE

Certified that this project report entitled "**Simulation of Static pressure, Velocity magnitute and Sand volume fraction in Agitated slurries using CFD technique**" is the bonafide work of

Mr.S.Karthikeyan - Register No. 71205402004

Who carried out the project work under my supervision.

C Sivanandan

Signature of the HOD

Signature of the supervisor
31/5/2007

4/7/07

**Internal Examiner**

4/7/07

**External Examiner**

**DEPARTMENT OF MECHANICAL ENGINEERING**

**KUMARAGURU COLLEGE OF TECHNOLOGY**

**COIMBATORE 641 006**

# Contemporary Approaches in Design and Manufacturing

CADAM '07

# CADAM '07

## A.C.College of Engineering and Technology
## Karaikudi - 04, Tamil Nadu.

26th & 27th April 2007

### Certificate of Recognition

This is to certify that Mr. / ~~Ms.~~ /~~Mrs.~~ S. KARTHIKEYAN

m KUMARAGURU COLLEGE OF TECHNOLOGY , COIMBATORE has presented

paper entitled SIMULATION OF STATIC PRESSURE AND VELOCITY MAGNITU IN AGITATED SLURRIE USING FLUENT SOFTWARE in the National Conference

DAM '07 organized by Department of Mechanical Engineering, A.C. College of Engineering & Technology, araikudi - 04.

-Authors :

| | | |
|---|---|---|
| :A.Syed Abu Thaheer | Prof. K.R.Viswanathan | Dr.R.Sundararajan |
| Co-Ordinator | Chairman | Principal |

REVEAL THE UNREVEALED

# ABSTRACT

The condition of slurry exists in process industries including the chemical processing industry, minerals, pulp and paper, waste water treatment and almost every process sector. The empherical approach to develop mathematical model bears some crucial drawbacks, local information of the system was difficult to obtain and pilot plant tests were time consuming and expensive. The objective of this study was to develop a computational fluid dynamics model of slurries in agitated vessel for finding the static pressure, velocity Magnitude and Sand volume fraction. The slurry taken for this work was water liquid with sand. Three dimensional flow field and solid concentration distribution in solid –liquid baffled stirred vessels were simulated using fluent software. A two-fluid approach and the single phase K-epsilon model were employed. The recirculation loop below the impeller and above the bottom is predicted .Conceptual sketching of the actual experimental model, selection of solver and initiating the flow conditions were important criteria in multi-phase model.

## ஆய்வுச் சுருக்கம்

சேறு போன்ற நிலை சுத்திகரிப்பு தொழிற்சாலைகளில் நிலவுகிறது. வேதியியல் தொழிற்சாலை, தாது, கூழ் மற்றும் காகித தொழிற்சாலை நீர் பதப்படுத்தும் ஆலை போன்ற இடங்களில் எல்லாம் சேறு போன்ற நிலை காணப்படுகிறது. கணித மாதிரியை உருவாக்கக்கூடிய இந்த முயற்சியில் சில செயற் சிரமங்கள் நிலவுகின்றது. இதுபோன்ற தொழிற்சாலைகளில் நிலவும் சூழ்நிலையை பற்றிய தகவல்கள் கிடைப்பது மிகவும் அரிதாக உள்ளது மற்றும் சிறிய அளவிலான மாதிரியை உருவாக்க பொருளாதார மற்றும் கால அவகாசம் அதிக அளவில் தேவைப்படும்.

இந்த ஆராய்ச்சியின் நோக்கமானது கலக்கப்பட்ட பாத்திரத்துடன் கூடிய சேறுக்கான நிலை அழுத்தம், திசைவேக அளவு மற்றும் மண்ணின் அடர்த்தி மதிப்பு இவைகளை கணித திரவ இயக்க மாதிரியை உருவாக்கி கண்டறிதல். இந்த ஆராய்ச்சிக்கு எடுத்துக்கொண்ட சகதியானது திரவ தண்ணீருடன் கூடிய மணல். முப்பரிமான நகரும் திரவ மற்றும் திட அடர்த்தி பரப்புதலை கலக்கப்பட்ட பாத்திரத்துடன் கூடிய பாவனையை, "பூலுயன்ட்" மென் பொருளை கொண்டு உண்மையில் உள்ளதை போன்றே உருவாக்கப்பட்டது.

இரண்டு திரவ மாதிரி மற்றும் ஒரு நிலை கே.எபிசிலான் மாதிரி கணக்கில் எடுத்துக்கொள்ளப்பட்டது. கலக்கியின் கீழ் மற்றும் மேல் உள்ள மறு கலவை சுழற்சியை கண்டறியப்பட்டது. ஆராய்ச்சி மாதிரியின் எண்ணக்கரு வரைபடம் வரைதல், தீர்வுகாணும் முறையை கண்டறிதல் மற்றும் செயற்க்கையாக திரவ இயக்கத்தை உருவாக்குதல் ஆகியவை பலநிலை மாதிரியை உருவாக்குதளில் உள்ள முக்கியமான வேலைகள் ஆகும்.

# ACKNOWLEDGEMENT

The Author takes this opportunity to express his deep sense of gratitude to the following hearts for encouraging us to take up this project

The Author owes his immense gratitude and thanks to his guide **Shri.Dr.V. Velmurugan** Asst.Professor, Department of Mechanical Engineering, Kumaraguru College of Technology, for his Valuable guidance, co-operation, constructive criticism and encouragement for making this project a successful one.

The Author expresses his deep gratitude to **Shri.Dr.Joseph V. Thanikal**, Principal, and Kumaraguru College of Technology for patronizing him, besides providing all assistance.

The Author is grateful to **Shri Dr.N.Gunesekaran**, Head of the Department, Department of Mechanical Engineering for his Valuable suggestions and Timely help towards his project

The author also express his sincere thanks to Shri **Dr. P.Palanisamy and Shri P.kannan** , Assistant professors, Mechanical Engineering Department for their valuable suggestions and encouragement and timely help towards this project.

# CONTENTS


| Title | | Page No. |
|---|---|---|
| Certificate | | ii |
| Abstract | | iv |
| Acknowledgement | | vi |
| Table of Contents | | vii |
| List of Tables | | ix |
| List of Figures | | x |
| List of Symbols | | xi |
| **CHAPTER 1** | **INTRODUCTION** | 1 |
| 1.1 | Solids suspension | 2 |
| 1.2 | Software used | 2 |
| 1.3 | The core activities in this project | 3 |
| 1.4 | Scope of the project | 3 |
| 1.5 | Limitations | 3 |
| **CHAPTER 2** | **LITERATURE SURVEY** | 4 |
| **CHAPTER 3** | **COMPUTATIONAL FLUID DYNAMICS** | 12 |
| 3.1 | Working definition of CFD | 12 |
| 3.2 | The need for CFD | 12 |
| 3.3 | Brief introduction to CFD | 13 |
| 3.4 | The benefits of CFD | 14 |
| 3.4.1 | Insight | 14 |
| 3.4.2 | Foresight | 15 |
| 3.4.3 | Efficiency | 15 |
| 3.5 | Advantages of CFD over experimental technique | 15 |
| 3.5.1 | Low cost | 15 |
| 3.5.2 | Speed | 15 |
| 3.5.3 | Complete information | 15 |
| 3.5.4 | Realism of flow conditions | 16 |
| 3.6 | Disadvantages of CFD | 16 |
| 3.6.1 | Simulation | 17 |

| | | |
|---|---|---|
| 3.7 | Fluent's technology advantage in CFD | 17 |
| 3.7.1 | Accuracy | 17 |
| 3.7.2 | Ease of use | 18 |
| 3.7.3 | Speed | 18 |
| 3.7.4 | Powerful visualization | 18 |
| 3.8 | The CFD process | 18 |
| 3.8.1 | Preprocessing | 19 |
| 3.8.2 | Solving | 19 |
| 3.8.3 | Postprocessing | 20 |
| 3.9 | Boundary conditions | 21 |
| **CHAPTER 4** | **SIMULATION** | 22 |
| 4.1 | Problem definition and concept sketching | 22 |
| 4.2 | Preprocessing | 23 |
| 4.2.1 | Preprocessing | 23 |
| 4.2.2 | Grid | 23 |
| 4.2.3 | Models | 25 |
| 4.2.4 | Materials | 25 |
| 4.2.5 | Phases | 26 |
| 4.2.6 | Boundary conditions | 26 |
| 4.3 | Solution | 28 |
| 4.3.1 | Fix zone | 28 |
| 4.3.2 | Initial impeller velocities for water | 29 |
| 4.3.3 | Initial impeller velocities for sand | 30 |
| 4.3.4 | Initial steled sand bed | 31 |
| 4.3.5 | After 1 sec | 32 |
| 4.4 | Post processing | 38 |
| **CHAPTER 5** | **RESULTS AND DISCUSSIONS** | 46 |
| **CHAPTER 6** | **CONCLUSION** | 47 |
| | **REFERENCES** | |

# LIST OF TABLES


| Table | Title | Page No. |
|---|---|---|
| 4.1 | Impeller profile specifications-1 | 27 |
| 4.2 | Impeller profile specifications-2 | 27 |

# LIST OF FIGURES


| Figure | Title | Page No. |
|---|---|---|
| 4.1 | Concept sketch | 22 |
| 4.1 | Half grid view | 24 |
| 4.3 | Full grid upright view | 24 |
| 4.4 | Fix-zone | 28 |
| 4.5 | Primary water velocity Magnitude | 29 |
| 4.6 | Initial Sand Velocity Magnitude | 30 |
| 4.7 | Initial settled sand bed | 31 |
| 4.8 | Sand velocity at fix- zone – After 1 sec | 32 |
| 4.9 | Water velocity at fix- zone – After 1 sec | 33 |
| 4.10 | Sand velocity colored by Water velocity magnitude – After 1 sec | 34 |
| 4.11 | Sand velocity colored by sand velocity magnitude – After 1 sec | 35 |
| 4.12 | Water velocity colored by Water velocity magnitude – After 1 sec | 36 |
| 4.13 | Volume fraction of sand – After 1 sec | 37 |
| 4.14 | Velocity Vectors for Sand - After 20sec | 38 |
| 4.15 | Velocity Vectors for Water - After 20sec | 39 |
| 4.16 | Velocity Vectors for Water - After 20sec | 40 |
| 4.17 | Path lines colored by sand velocity magnitude - After 20sec | 41 |
| 4.18 | Path Lines colored by Water Velocity Magnitude - After 20sec | 42 |
| 4.19 | Sand Volume Fraction - After 20sec | 43 |
| 4.20 | Static Pressure - After 20 Sec | 44 |
| 4.21 | Scaled Residuals | 45 |

# LIST OF SYMBOLS & ABBREVATIONS

| | | |
|---|---|---|
| Re | - | Reynolds number |
| $\rho$ | - | Density ($kg/m^3$) |
| V | - | Velocity (m/s) |
| No | - | Nusselt number |
| Pu | - | Prantyl number |
| T | - | Temperature ( $°C$ ) |
| CFD | - | Computational Fluid Dynamics |
| CAD | - | Computer Aided Design |
| GUI | - | Graphical user Interface |

# CHAPTER 1

# INTRODUCTION

# CHAPTER 1

# INTRODUCTION

Mixing tanks were used to maintain solid particles or droplets of heavy fluids in suspension. Mixing may be required to enhance reaction during chemical processing or to prevent sedimentation. In this project the Eulerian multiphase model solves momentum equations for each phases were allowed to mix in any proportion.

The slurry was a non Newtonian fluid which didn't have constant viscosity. In that the viscosity changes with the applied strain rate. In a strict sense, a fluid was any state of matter that was not a solid, and a solid was a state of matter that has a unique stress free state. A conceptually simpler definition was that a fluid was capable of attaining the shape of its container and retaining that shape for all time in the absence of external forces. Therefore fluids encompass a wide variety of states of matter including gases and liquids as well as many more esoteric states. (For example, plasmas, liquid crystals, and foams)

Among the Various industrial unit operations involved with multi - phase systems, agitation of solid-Liquid systems was quite commonly encountered such as catalytic reactions, leaching, polymerization, etc. Despite their widespread use, the complex 3-D recirculation and Turbulent flow in the vessel makes designing and optimizing the reactor usually resorted to pilot plant tests and the empirical formulation, even for single phase applications.

The system was difficult to obtain and pilot plant tests were time –consuming and expensive. With the development of fluid dynamics (CFD) techniques, numerical method was more popularly adopted to simulate flow field in stirred vessels. Most of works reported were on single phase flow in stirred vessels. When a second phase was introduced, the treatment became drastically complex. The main difficulty in simulating the flow field in multi-phase stirred vessels was the accurate representation of the impeller action, the inter-phase interaction and the turbulent quantities.

A number of investigations had been published on the fluid dynamic properties of solid –liquid systems in stirred vessels for achieving empirical information. Most Researchers were focused on the distribution of solid particles (Baladi et al., 1981; Barris and Baldi, 1987) and the criteria of suspension (Chudacek, 1986; Zwietring, 1958).

## 1.1 SOLIDS SUSPENSION:

- Mechanical Agitation was widely used in process industry operations involving Solid –Liquid flows
- The typical process requirement for was for the solid phase to be suspended (Dissolution, reaction, feed uniformity).
- The was understanding the fluid dynamics and relating this knowledge to design
- The CFD model can provide insight to both the multiphase transport and design parameters.

In this project the slurry conditions and the flow field of slurries within the vessel were simulated by considering this as Eulerian multiphase K-epsilon model and segregated solver was used to take the time dependent solutions.

## 1.2 SOFTWARE USED:

- Modeling –Auto cad 2004
- Meshing - Gambit ( A mesh tool for Fluent 6.0 )
- Solver- Fluent 6.0

## 1.3 THE CORE ACTIVITIES IN THIS PROJECT ARE LISTED BELOW:

- Concept Sketching
- Using the granular Eulerian multiphase model
- Specifying fixed velocity with a user defined- function to simulate an impeller.
- Set boundary conditions for internal flow
- Modeling and meshing
- Solving unsteady state transient problem

## 1.4 SCOPE OF THE PROJECT:

- Best method of analyzing the slurry conditions
- Time and cost optimum method than Pilot Projects

## 1.5 LIMITATIONS:

- The simulation result would be with the error of 15%
- No Validation part concentrated due to lack of experimental data.

Reason being learning of the software and collection of dates were practically involving long lead Time. Owing to time constraints to complete the project: it was not included in the Project. However on the continuous basis the work is under progress for validating the simulation results with the experimental values and finding the Heat Transfer coefficient to slurries, for my conference paper work.

# CHAPTER 2

# PROBLEM DEFINITION

# CHAPTER 2
# REVIEW OF LITERATURE

Agitation increases the heat Transfer rate in nucleate boiling.Degree of agitation plays an important role. An Ordinary degree of agitation causes only a negligible change in the maximum heat flux. Violent agitation causes an observable effect. The natural agitation resulting from the boiling action itself results in a high degree of Turbulence is of the same order of magnitude it will scarcely affect the heat transfer Coefficient.

Heat transfer of Newtonian fluids can be easily found out from the standard tables. The simple definition of Newtonian fluid is fluids in which viscosity is constant and independent of shear rate but is highly dependent on temperature; also refer to absolute viscosity. Examples of Newtonian fluids are water, syrup, mineral oils, lacquers, and solvents.

A Newtonian fluid is a fluid in which shear stress is linearly proportional to the velocity gradient in the direction perpendicular to the plane of shear. The constant of proportionality is known as the viscosity. For a Newtonian fluid, the viscosity by definition depends only on temperature and pressure, and also the chemical composition of the fluid if the fluid is not a pure substance. If the fluid is incompressible and viscosity is constant across the fluid,

A non-Newtonian fluid is in which the changes with the applied strain rate. As a result, non-Newtonian fluids may not have a well-defined viscosity.

Non-Newtonian fluid can be defined as a fluid that departs from the classic linear Newtonian relation between stress and shear rate. In a strict sense, a fluid is any state of matter that is not a solid, and a solid is a state of matter that has a unique stress-free state. A conceptually simpler definition is that a fluid is capable of attaining the shape of its container and retaining that shape for all time in the absence of external forces. Therefore, fluids encompass a wide variety of states of matter including gases and liquids as well as many more esoteric states (for example, plasmas, liquid crystals, and foams).

Heat transfer to non- Newtonian fluids can not be found out easily. Slurry is a non Newtonian Fluid. Jacketed vessels with agitator for heating and cooling drew much importance in process Industries including chemical Industries, minerals, Pulp and paper, wastewater treatment and almost every process sector.

In general agitation increases heat Transfer rate. The rate of heat transfer to or from an agitated liquid mass in a vessel is a function of the physical properties liquid and of the heating or cooling medium, the vessel geometry, and the degree of agitation.

When N is rotational speed in rpm, d is the stirrer diameter in m, $\rho$ is fluid density in $kg/m^3$ and $\mu$ is the viscosity in $Ns/m^2$, the presence or absence of turbulence in an impeller-stirred vessel can be correlated with an impeller Reynolds number as

$$\mathrm{Re} \approx \frac{d^2 N\rho}{\mu}$$

Flow in the tank is turbulent when Re > 10,000. Thus viscosity is not a valid indication of the type of flow to be expected. Between Reynolds number of 10,000 and approximately 10 is a transition range in which is the flow is turbulent at the impeller and Turbulent at the impeller and laminar in remote parts of the vessel. When Re, 10, flows is laminar only.

In batch operations, it is often necessary to calculate the time '$\tau$' needed to heat or cool the contents of a jacketed vessel from temperature t 1 to t 2 in simplified form, the relevant equations are as follows

For Heating

$$\tau \approx \left[\frac{MsCp}{UA}\right] \ln\left[\frac{T - t1}{T - t2}\right]$$

For Cooling

$$\tau \approx \left[\frac{MsCp}{UA}\right] \ln\left[\frac{t1 - T}{t2 - T}\right]$$

Where T is a jacket temperature, 'Ms' is the mass of slurry in the vessel and Cp is the specific heat of the slurry, in both of the equations assumed that the jacket temperature is constant. These equations can also be used in instances where the difference between the jacket inlet and outlet temperature is not greater than 10 % of the LMTD between the average temperature of the jacket and the temperature of the vessels contents. In such instances, the average value can be taken for jacket temperature. In the design of experiments series of structured tests are designed in which planned changes are made to the input variables of a process or system the effects of these changes on a defined output are assessed. This is very useful technique when the numbers of variables are more, which are kept at different levels. There were five variables each at 3 levels has been considered for investigation. As per probability rate 243 chances of experiments have to be conducted. It will consume more time and cost. Therefore, the experimental trials have been reduced to 27, as per Taguchi Experimental Techniques. Each trial was conducted thrice to check the repeatability and the average of the three values have been taken for analysis. They concluded that the higher the agitation rate the heat transfer will be more .Among the five variables considered stirrer speed and slurry concentration have significant effect on vessel side heat transfer coefficients and the remaining variables have almost no significant effect. The pitched blade Turbine provides higher level turbulence, which's turn resulted in higher values heat transfer coefficient when compared with flat blade turbine agitator.

Harry W. Heinlein, et al (1972), Heat transfer to liquids in agitated vessels is a common practice in the chemical process indusries. Non-Newtonian fluid behavior is often encountered in these industries and since the consistency of these non-Newtonian fluids is usually quite high, their processing frequently occurs in the laminar or transition flow regimes. The research reported in this paper concerned with heat transfer at low Reynolds numbers to non Newtonian, time –independent fluids in jacketed vessels. The impellers use I this work were different sized anchor agitators and the fluids used were various aqueous Carbopol solutions and slurries of precipitated chalk in water. The rheological behavior of the Carbopol solutions allowed them to be classified as pseudo plastic fluids conforming to the law model

$$\tau = K\gamma^{n}$$

The chalk slurries behaved according to the Bingham plastic model for fluids exhibiting a yield stress.

The research is concerned with the heat transfer at low Reynolds numbers to non-Newtonian, time dependent fluids in jacketed vessels. The impellers used in this work were different sized anchor agitators and the fluids used were various aqueous Carbopol solutions and the slurries of precipitated chalk in water.

Joseph A. Martane, et al (1971), Heat transfer to non Newtonian Bingham plastic slurries in a jacketed, agitated vessel was studied using an anchor and a flat blade disk turbine as the impellers. An effective Viscosity based on the average shear rate existing in the agitated vessel as determined from the studies on the rates of viscous dissipation of energy in the agitation of non –Newtonian fluids was used to correlate the data. Data were taken for both impellers over a prantyl number range of 2 to 700 and for a volume concentration of solids up to 13.6 %.For the Reynolds number range of 200 to 80000 data for the turbine impeller were correlated by

$$Nu = 0.536\, N_{Re}^{2.3} N_{pr}^{1.3} N_{Visc}^{.14} \left[ \frac{\rho}{\rho} - 1 \right]^{0.065}$$

For the Reynolds number range of 300 to 95000 data for the anchor impeller were correlated. In heat transfer with the systems involving non-Newtonian fluids which has been occasioned by the impeller increasing industries of materials whose theological behavior cannot be characterized by Newtonian relationships. Non-Newtonian fluid behavior is frequently encountered in such process industries as the manufacturing and processing of petroleum, paints, plastics, nuclear fuels, and foods... Two types of impellers were investigated, a flat blade disk turbine and an Anchor. Palvlushenko and Gluz (1966a.b) used dimensional analysis to obtain the appropriate dimensional analysis.

Bakker, et al (1998), Large –scale bio oxidation of gold is usually carried on in continuous is carried on in continuous stirred tank reactors. Attaining homogeneous slurries is a difficult task, as solids tend to stratify in the tank. The objective of this work was to determine the optimal conditions of agitation in a CSTR so to obtain the best solids Suspension. The experiments were performed in a 5 liters glass tank operated with 3 liters of 6%w/v slurry. The impellers (Pitched blade turbine or marine propeller) were placed at heights of 6.7 to 13.4 cm from the bottom and operated at 370 to 1040 rpm, with specific aeration rates of 0.3 to 3.7 vvm.

A statistical experimental design was used to which allowed the deviation of a model representing response surfaces of the exit and mean solids concentration as a function the impeller type, impeller distance from the bottom and aeration and agitation rates. During the experiments no solids were deposited on the bottom and the solids

Concentration near the bottom was always higher than that of the top region. At the optimal conditions for the each type of impeller, the marine propeller required agitation rates about 15 to 22 % higher than the pitched blade turbine.Neverthless it is concluded that the marine helix is preferable because it requires less power and produces a more homogeneous suspension.

Ferreira C, et al (2001), Ideal Solutions with equal molar concentrations have freezing Point. The properties that determine heat and mass transfer processes encountered in a secondary cooling cycle are heat and mass transfer processes encountered in a secondary cooling cycle are however determined by the mass fraction of solutes, generally for aqueous, the more freezing point depressant added the less efficient heat and mass transfer properties. Therefore substances with low molecular weight are expected to result in more efficient ice slurries.

By Calculating ice slurry properties, heat transfer and pressure drop, it was investigated if ice slurries of low molecular weight additives result in efficient ice slurries. For the substances considered, it was found that the molecular weight is a good indication for the slurry efficiency, but this not decisive under all conditions. Ice Slurries of three particular substances were found to have most promising properties: Sodium Chloride, Lithium Chloride and Potassium format.

Substances like antifreeze proteins depress the freezing point of a solution actively by interfering in the ice crystal formation. These substances have been suggested for freezing point depression in slurries too, but can only be used to control ice crystal size or super cooling effects, and not to depress the freezing point because of the extra temperature difference introduced with thermal hysteresis...

The molecular weight of freezing point depressing substances gives a good indication of the efficiency of ice slurry in a secondary cooling system when compared to similar substances with higher molecular weight. It is however decisive under all conditions. In the three pairs of freezing point depressants that were compared for the thermo physical properties, heat transfer and pressure drop, the effects were stronger at low temperatures. This is explained with the influence of the apparent heat capacity, which is relatively less important at low temperatures.

Molecules with low molecular weight tend to be more efficient for ice slurries, because the thermo physical properties of the ice slurries decrease when more solute is added, and additives of low molecular weight require low mass fractions added to depress the freezing point. From this research it followed that lithium chloride; sodium chloride and potassium format are advantageous freezing point depressants. The performance of an ice slurry in a secondary cooling system is not only determined by the freezing point depression and the ice fraction but for a great part also by mass and heat transfer properties as heat capacity,viscosity,density and thermal conductivity. For aqueous solutions these properties as heat capacity, viscosity, density and thermal conductivity. For aqueous solutions these properties become less efficient when more solute is added. Mostly this is not in a proportional way to the molar fractions is approximately the case with freezing point depression. An equal mole fraction as is approximately the case with the freezing point depression. An equal mole fraction as is approximately the aces with freezing point depression. Solutes with a low molecular weight therefore probably result in more efficient ice slurries compared to similar solute will result in equal freezing points but nit in an equal viscosity of the solution.

Rao M, et al (1976), Aqueous polymer solutions of sodium carboxymenthyl cellulose (SCMC) and sodium alginate (SA) have been studied in a turbine –agitated vessel for standard and nonstandard vessel configurations with agitator diameter, depth of agitation, helix diameter, and coiled tube outside diameter as parameters. The jacket- and coil –side heat transfer results are correlated. There has been increasing interest in the mechanically aided heat transfer to non- Newtonian fluids which are often encountered in processing Industries. There are many situations where such fluids have to be heated or cooled in an agitated vessel and non- Newtonian characteristics can be of considerable significance. Voluminous literature has been built up on heat transfer to Newtonian and non-Newtonian fluids in an agitated vessel which is confirming to standard vessel configuration. Many times, in industry, one has to make use of available agitated vessel with its accessories, which are not confirming to standard vessel configuration. Under these circumstances it becomes very difficult to know beforehand the heat transfer performance of such nonstandard vessel configuration to put into operation with a fair degree of confidence the heat transfer characteristics of the vessel for which agitator diameter, and the coiled tube outside diameter deviate from the values pertaining to the standard vessel configuration using water and different pseudo plastic liquids of equal importance.

Zai-sha Mao, et al (2003), Three dimensional flow field and solid concentration distribution in solid-liquid stirred vessels are numerically simulated using an improved inner-outer iterative procedure. A two-fluid approach and the single K-epsilon model are employed. The recirculation loop below the impeller and above the bottom is predicted. Agreement of simulation results with experimental data is satisfactory for both mean velocities of phases and solid concentration in two cases with the average solid concentration.

The empherical approach bears some crucial drawbacks. Local information of the system is difficult to obtain and pilot plant tests are time consuming and costly... Most of works reported single phase flow in stirred vessels .When a second phase is introduced; the mathematical treatment becomes drastically complex. The main difficulty in simulating the flow field in stirred vessels. Is the accurate representation of impeller action, the inter phase interaction and the turbulent quantities. Gosman et al (1992) calculated that the flow in the Solid-liquid stirred tanks with the Ruston impeller region treated as a block box hence the experimental data had to be imposed on the surface swept by the impeller blades as the boundary conditions. The oblivious shortcoming of such approach is that the experimental data in this region is crucially needed for initiating the simulation and it is not applicable to novel operation conditions without experimental measurements. Micale et al (2000) applied the inner –outer iterative procedure developed by Brucato et al (1998) to simulate the floe in a solid liquid stirred vessel, this method does not need experimental data as the impeller region boundary conditions. The information's on the surfaces of the 'inner' and 'outer' domains were averaged over the azimuthal direction thus some important features for the flow in the stirred vessel generated by the periodical rotation of the impeller were ignored. Wang and Mao (2002) improved the inner-outer iterative procedure by combining it with a snapshot approach proposed and applied it simulate the flow in single –phase and gas liquid stirred vessels. The computational results agreed well with the experimental data.

Derrick Kersershaw, et al (1999), Large, mechanically agitated pump box is used between mixers, for mixing oil sand and water to produce slurry and a pipeline for conveying the slurry being discharged into the pump box from the mixer; it is screened to reject large solids. The oversize is directed to an impactor where it is communicated product is screened again prior to being discharged into the pump box from the mixer ,it is directed to an impactor where it is comminuted and the comminuted product is screened again prior to being into the pump box. The pump box is designed to increase the residence time of the slurry in the pump box and to separate the slurry into two phases, the suspended slurry and the larger lumps that cannot be suspended. The larger lumps that settle in the pump box are recycled to the impactor for communication.

K C T LIBRARY Coimbatore - 641 006.
P- 2074

# CHAPTER 3

# COMPUTATIONAL FLUID DYNAMICS

# CHAPTER 3

# COMPUTATIONAL FLUID DYNAMICS

## 3.1 WORKING DEFINITION OF COMPUTATIONAL FLUID DYNAMICS:

First, let's break down the words:

- Computational - having to do with mathematics, computing
- Fluid Dynamics - the dynamics of things that flow

This is CFD a computational technology that enables us to study the dynamics of things that flow. Using CFD we can build a computational model that represents a system or device that we want to study. Then we apply the fluid flow physics and chemistry to this virtual prototype, and the software will output a prediction of the fluid dynamics and related physical phenomena. Therefore, CFD is a sophisticated computationally-based design and analysis technique.CFD software gives us the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling. Using CFD software, we can build a 'virtual prototype' of the system or device that we wish to analyze and then apply real-world physics and chemistry to the model, and the software will provide us with images and data, which predict the performance of that design.

## 3.2 THE NEED FOR CFD:

Applying the fundamental laws of mechanics to a fluid gives the governing equations for a fluid. The conservation of mass is

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \overline{V}) = 0$$

And the conservation of momentum equation is

$$\rho\frac{\partial\overline{V}}{\partial t}+\rho(\overline{V}.\nabla)\overline{V}=-\nabla p+\rho\overline{g}+\nabla.\tau_{i,j}$$

These equations along with the conservation of energy equation form a set of coupled, non liners partial differential equations. It is possible to solve these equations analytically for most engineering Problems.

However, it is possible to obtain approximate computer –based solutions to the governing equations for a variety of engineering problems .This is the subject matter of Computational Fluid Dynamics (CFD)

## 3.3 BRIEF INTRODUCTION TO CFD:

Computational fluid dynamics is the art of replacing the governing partial differential equations of fluid flow (The Navier-Stokes Equations) with numbers, and advancing these numbers in space and/or time to obtain a final numerical description of the complete flow of interest.

The end product of CFD is indeed a collection of numbers, in contrast to a closed-form analytical solution. However, in the long run the objective of most engineering analyses, closed form or otherwise, is a quantitative description of the problem, i.e. numbers. The starting point for any description of a flow field is the solution of the Navier-Stokes equations, which can be described as the variation of the three components of velocity (u,v,w) in both space and space, coupled to the pressure gradients in x,y and z. This describes a flow which is steady, compressible, three-dimensional and viscous. To analyse incompressible viscous flow, the N-S equations and the continuity equation are sufficient, for compressible flow, the energy equation is also needed.

The CFD method involves the discretisation of the fluid domain into cells, with each cell having a grid point at its vertex. This series of grid points and cells make up the finite-difference mesh.

The mesh covers the whole fluid area and associated boundaries, inlets and outlets. These boundaries are described as boundary conditions and show how the flow is contained inside the domain. Of course these boundaries can be inlets, outlets, walls or lines of symmetry, depending on the description of the flow problem. The boundary conditions are specified in terms of velocities, pressures and temperatures. The fluid's physical properties (viscosity, density etc) are applied to flow domain.

The solution method depends on the type of problem being modeled and can vary between laminar, turbulent, compressible, transient and two-phase flow. It can be either an explicit or implicit technique, or semi-explicit scheme.

The final unchanging state is called the convergence of the iterations. The converged solution is actually the correct solution of the nonlinear equations. It is possible that successive iterations would not ever converge to a solution. The computed result may steadily drift or oscillate with increasing amplitude. This is known as divergence.

## 3.4 THE BENEFITS OF CFD:

There are three compelling reasons to use this CFD software is insight, foresight and efficiency.

### 3.4.1 Insight:

If we have a device or system design which is difficult to prototype or test through experimentation, CFD enables us to virtually crawl inside our design and see how it performs. There are many phenomena that can witness through CFD, which wouldn't be visible through any other means.CFD gives, us a deeper insight into our designs.

### 3.4.2 Foresight:

Because is a tool for predicting what will happen under a given set of circumstances, it can quickly answer many 'what if?' questions. If we provide a set of boundary conditions, and the software gives us outcomes. In a short time, we can predict how our design will perform, and test many variations until we arrive at an optimal result. All of this can be done before physical prototyping and testing.

### 3.4.3 Efficiency:

The foresight we gain from CFD helps us to design better and faster, save money, meet environmental regulations and ensure industry compliance.CFD analysis leads to shorter design cycles and our products get to market faster. In addition, equipment improvements are built and installed with minimal downtime.CFD is a tool for compressing the design and development cycle allowing for rapid prototyping.

## 3.5 ADVANTAGES OF CFD OVER EXPERIMENTAL TECHNIQUES:

### 3.5.1 Low Cost:

Computational prediction costs many orders of magnitude lower than complicated experimental investigation. This factor becomes more important as the physical situation under study becomes larger and more complicated. Also, as the price of most items is increasing, the cost of computing power is continually reducing.

### 3.5.2 Speed:

The study of the implications of many different configurations can be assessed in a shorter time frame than the experimental investigation.

### 3.5.3 Complete Information:

A computer solution of a problem gives detailed and complete information. It can provide the values of all the relevant variables (such as velocity, pressure, temperature, powder concentration, turbulence intensity) throughout the domain of interest. There is no interference due to measuring probes, and few areas are inaccessible. If needs be, computational techniques, can be used to supplement experimental study.

### 3.5.4 Realism of Flow Conditions:

In simulation techniques, realistic models can be achieved. There is no need for scaling and cold-flow models. Computer models can easily vary between large and small dimensions, low or high temperatures, fast or slow processes, toxic or flammable substances. Due to this vast variation in parameters, ideal situations can easily be achieved, thus removing the interference from irrelevant effects. Experimental investigation is rarely an idealization.

## 3.6 DISADVANTAGES OF CFD OVER EXPERIMENTAL TECHNIQUES:

Computational techniques rely totally on the validity of the analytical models in the coding. If these are wrong, or the understanding of the physical phenomena is not yet available, simulation techniques can not be used. Experimental investigation on the other hand will still produce a result. Some examples where a mathematical description is till lacking, includes complex turbulent flows, certain non-Newtonian fluids and some combustion processes. Even where the mathematical description is correct, some small scale effects, like small scale turbulence, if they are to be modeled for every location in the computational domain, will provide prohibitively expensive in computer power.

### 3.6.1 Simulation

Simulation is defined as "Imitation of reality". The experimental set up will be simulated by using Simulation Software 'Fluent 5/6'.The input variables which we are choosing to the problem and boundary conditions will play the key role in the simulation of the system.

The discrimination of the equation, the commercial CFD codes the Finite Volume method. When we are using FLUENT or another finite volume code, it's useful to remind ourselves that the code is finding a solution such that the mass, momentum, energy and other relevant quantities are being conserved for each cell.

## 3.7 FLUENT'S TECHNOLOGY ADVANTAGE IN CFD:

### 3.7.1 Accuracy:

The accuracy of a CFD analysis is driven largely by his quality of the physical models used and the degree to which the actual geometry is captured. Among commercial CFD software products, FLUENT has the largest array of industrially tested capabilities - some 1,000 physical models. These models are remarkably robust, with associated features to accelerate convergence every time. The models allow for simulations involving:

- Incompressible or compressible flow and heat transfer
- Species transport
- Chemical mixing and reaction (combustion)
- Rotating reference frames
- Radiation and conjugate heat transfer
- Dispersed phase trajectories
- Multiphase flows
- Lumped parameter models (for complex system analysis)
- Aero acoustics
- Dynamic meshing
- Turbulence models (using high- and low-Re and LES models)

### 3.7.2 Ease-of-Use:

Our interactive software allows us to make changes to the analysis at any time during the setup, solution, or post processing phase. This saves time and enables us to refine our designs efficiently. The intuitive interface because FLUENT uses unstructured, hybrid modeling technology, models can be built that conform to arbitrary geometric shapes and other complex surfaces. As a result, our CFD model will have accuracy it needs, where it is needed.

Makes learning easy. Smart panels show only the modeling options that are appropriate for the problem setup at hand. CAD geometries are easily imported and adapted for CFD solutions.

### 3.7.3 Speed:

Solver enhancements and numerical algorithms that decrease the time to solution are included in every new release of our software. Our mature, robust, and flexible parallel processing capability enables you to solve bigger problems faster, and has been proven on the widest possible variety of platforms in the industry.

### 3.7.4 Powerful Visualizations:

FLUENT's post processing provides several levels of reporting, so we can satisfy the needs and interests of all audiences. Quantitative data analysis can be as rigorous as we require. High resolution images and animations allow us to communicate our results with impact. Numerous data export options are available for integration with structural analysis and other CAE software programs.

## 3.8 THE CFD PROCESS:

There are essentially three stages to every CFD simulation process: preprocessing, solving and post processing.

### 3.8.1 Preprocessing:

This is the first step in building and analyzing a flow model. It includes building the model within a computer-aided design (CAD) package, creating and applying a suitable computational mesh, and entering the flow boundary conditions and fluid materials properties.

CAD geometries are easily imported and adapted for CFD solutions in GAMBIT, Fluent's own preprocessor. 3D solid modeling options in GAMBIT allow for straightforward geometry construction as well as high quality geometry translation. Among a wide range of geometry tools, Boolean operators provide a simple way of getting from a CAD solid to a fluid domain. A state-of-the-art set of cleanup and conditioning tools prepares the model for meshing. Gambit's unique curvature and proximity based "size function" produces a correct and smooth CFD

C type mesh throughout the model. Together with our boundary layer technology, a number of volumetric meshing schemes produce the right mesh for our application. Parametric variations are also inherent to the process.

Fluent's solvers also couple with leading tools ANSA, Harpoon, Sculptor and YAMS, extending your capability to effectively create the mesh you need.

### 3.8.2 Solving:

The solver does the flow calculations and produces the results. We provide four general-purpose products. Flow Izard is the first general-purpose rapid flow modeling tool for design and process engineers built by Fluent. POLYFLOW (and FIDAP) are also used in a wide range of fields, with emphasis on the materials processing industries.

The FLUENT code has extensive interactivity, so we can make changes to the analysis at any time during the process. This saves the time and enables us to refine our designs more efficiently. Our graphical user interface (GUI) is intuitive, which helps to shorten the learning curve and make the modeling process faster. It is also easy to customize physics and interface functions to our specific needs. In addition, FLUENT's adaptive and dynamic mesh capability is unique among vendors and works with a wide range of physical models. This capability makes it possible and simple to model complex moving objects in relation to flow.

We provide the broadest range of rigorous physical models that have been validated against industrial scale applications, so we can accurately simulate real-world conditions, including:

- Multiphase flows
- Reacting flows
- Rotating equipment
- Moving and deforming objects
- Turbulence
- Radiation
- Acoustics, and
- Dynamic meshing

The FLUENT solver has repeatedly proven to be fast and reliable for a wide range of applications. The speed to solution is faster because our suite of software enables us to stay within one interface from geometry building through the solution process, to post processing and final output. FLUENT's performance has been tried and proven on a variety of multi-platform clusters. Our parallel computing capability is flexible and it will enable us to solve larger problems faster.

### 3.8.3 Post processing:

This is the final step in analysis, and it involves the organization and interpretation of the predicted flow data and the production of images and animations.

All of Fluent's software products include full post processing capabilities. Our post processing tools enable us to provide several levels of reporting, so we can satisfy the needs and interests of all the stakeholders in our design process. Quantitative data analysis can be as sophisticated as we require. High-resolution images and animations help us to tell our story in a quick and impact manner.

Fluent's data exports to such as Ensight, Field view and Tec Plot as well as to VRML formats. In addition, FLUENT CFD solutions are easily coupled with structural codes such as ABAQUS, MSC and ANSYS, as well as to other engineering process simulation tools.

## 3.9 BOUNDARY CONDITIONS:

FLUENT like other CFD codes, offers a variety of boundary condition options like Pressure inlet, Pressure outlet, Velocity outlet etc. It is very important that the proper boundary conditions in order to have a well defined problem. A Single wrong boundary Condition can give us totally wrong result.

# CHAPTER 4

# SIMULATION

# CHAPTER 4

# SIMULATION

## 4.1 PROBLEM DEFINITION AND CONCEPT SKETCHING:

The set up involves the transient startup of an impeller –driven mixing tank. The primary phase is water; while the secondary phase consists of sand particles with a 120 micron diameter. The sand initially settled at the bottom of the tank, to a level just above the impeller. The domain is modeled as 2D axis metric.

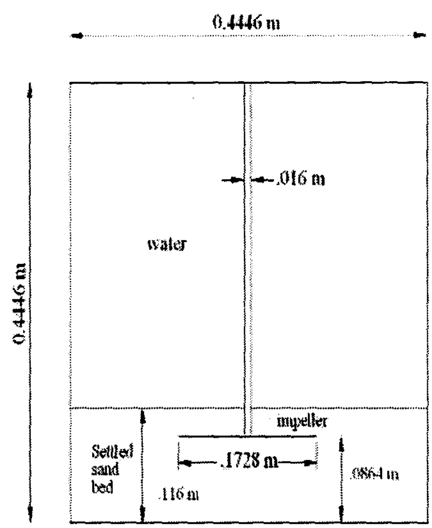


**Fig.4.1. Concept Sketch**

The fixed values option is used to simulate the impeller. Experimental data are used to represent the Time –averaged velocity and Turbulence Values at the impeller Location. This approach avoids the need to model the impeller itself. These experimental data are provided in a user defined function.

## 4.2 PREPROCESSING:

This was the first step in building and analyzing a flow model. It includes building the model within a computer-aided design (CAD) package, creating and applying a suitable computational mesh, and entering the flow boundary conditions and fluid materials properties.

CAD geometries were easily imported and adapted for CFD solutions in GAMBIT, Fluent's own preprocessor. 3D solid modeling options in GAMBIT allow for straightforward geometry construction as well as high quality geometry translation. Among a wide range of geometry tools, Boolean operators provide a simple way of getting from a CAD solid to a fluid domain. A state-of-the-art set of cleanup and conditioning tools prepares the model for meshing. Gambit's unique curvature and proximity based "size function" produced a correct and smooth CFD

### 4.2.1 Preparation:

- Copied the files mix tank / mix tank. msh and mix tank/ fix.c from the FLUENT web site to working directory.
- Started the 2D version of the FLUENT. Drawn the half portion of the concept sketch. Since the diagram was axisymmetric.

### 4.2.2 Grid:

- Read the grid file mix tank.msh and it reported its progress in the console window.
- Checked the grid.
- Fluent performed various checks on the mesh and would report the progress. We have to pay attention to the reported minimum volume and make sure that it was a positive number.
- Displayed the grid using default settings.

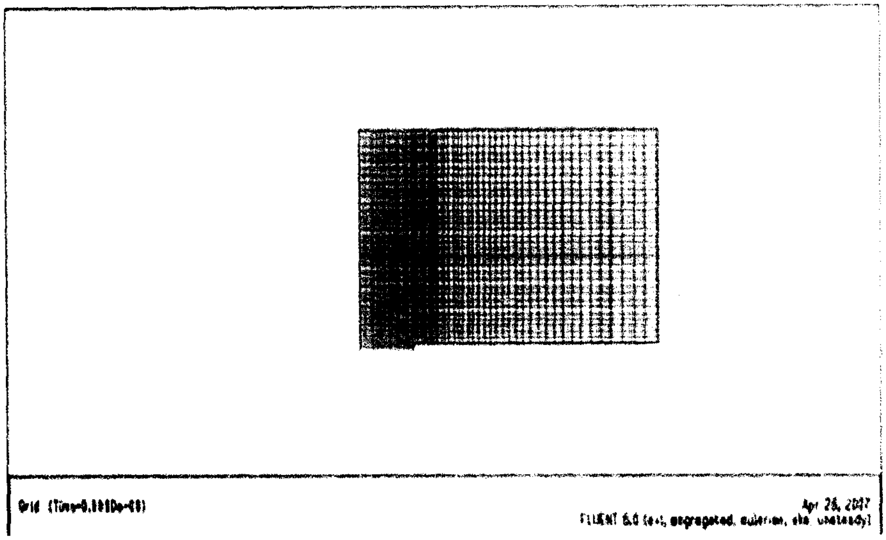

**Fig.4.2 Half grid view**

❖ Manipulated the grid display to show the full tank upright. Selected the mirror plan axis. Using the auto scale and camera parameters, the upright view was displayed in the screen.

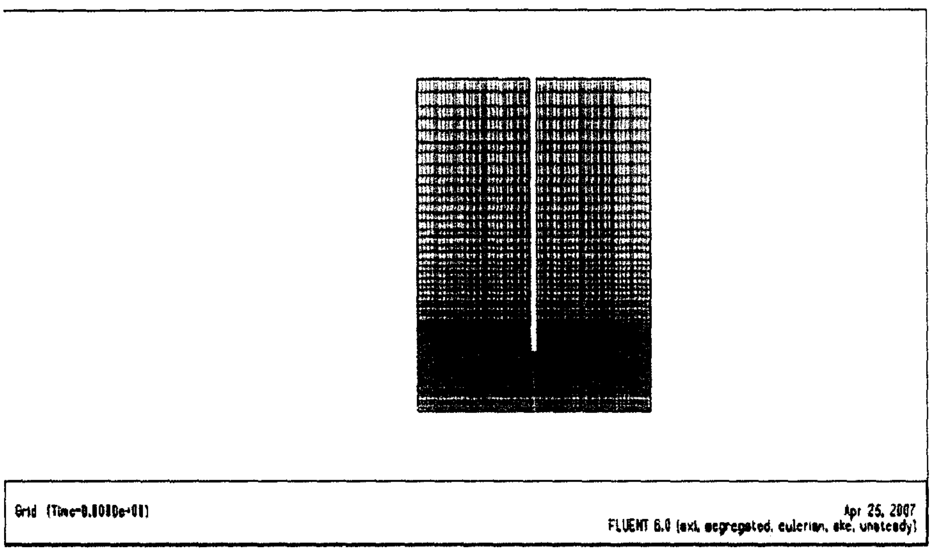

**Fig.4.3 Full grid upright view**

### 4.2.3 Models:

- Specified transient, axisymmetric model under the solver window and retained default segregated solver.
- Under space selected Axisymmetric.
- Under time selected Unsteady.
- In the multiphase model selected Eulerian as the model and kept the default settings.
- Under viscous model selected k- epsilon as the model.
- Kept the default selection of standard wall function under Near-Wall Treatment.
- Under k-epsilon multiphase model selected dispersed model since it was applicable in the case there was one primary continuous phase and the material density ratio of the phases was about 2.5. Further more the stokes number was much < 1 therefore the partial kinetic energy would not depart significantly from that of the liquid.
- Under operating conditions set the gravitational acceleration in the x-direction to $-9.81\ m/s^2$.

### 4.2.4 Materials:

- Under data base materials liquid water was selected as a primary phase.
- Created a new material called sand under the material dialogue box with the density of 2500 $kg/m^3$., which was the secondary phase.

### 4.2.5 Phases:

- Defined water has primary phase.
- Under secondary phase specified sand with the diameter of 120 microns.
- Under granular viscosity selected syamlal-obrien.
- Under granular bulk viscosity selected lun-et-al, with the packing limit of 0.6.
- Specified the drag law to be used for computing momentum transfer by clicking the interaction button in the phases panel.
- Under the phase interaction panel selected gidaspow in the drag co-efficient drop-down list.

### 4.2.6 Boundary Conditions:

To this problem, there were no conditions to be specified on the outer boundaries. Within the Domain, there were here fluid Zones, representing the impeller region, the region where the sand is initially located, and the rest of the tank. There were no conditions to be specified in the latter two zones, so set conditions for the Zone representing the impeller.

The UDF was used to specify the fixed velocities that simulate the impeller. The Values of the Time Averaged impeller velocity components and the turbulence quantizes were based on experimental measurement. The variations of these values may be expressed as a radius and imposed as polynomials.

Variable $= A_1 + A_2 r + A_3 r^2 + \ldots\ldots\ldots\ldots\ldots\ldots$

**4.1 Impeller profile specifications - 1**

| Variable | $A_1$ | $A_2$ | $A_3$ |
|---|---|---|---|
| u velocity | -7.135e-2 | 54.304 | -3.1345+3 |
| v velocity | 3.1131e-2 | -10.313 | 9.5558+2 |
| kinetic energy | 2.2723e-2 | 6.7989 | -424.18 |
| Dissipation | -6.5619e-2 | 88.845 | -5.3731+3 |

**4.2 Impeller profile specifications - 2**

| Variable | $A_4$ | $A_5$ | $A_6$ |
|---|---|---|---|
| u velocity | 4.55 | -1.9664e+5 | _ |
| v velocity | 3.1131e-2 | -1.1856e+5 | _ |
| kinetic energy | 2.2723e-2 | -7.7251e+4 | 1.8410e+5 |
| Dissipation | -6.5619e-2 | -9.1202e+5 | 1.9567e+6 |

The order of Polynomial used were depends upon the behavior of the function being fitted. For this, the polynomial and C code were imported from Fluent Website .The conditions for fluid zone representing the impeller were set. The conditions for the water and sand specified separately. There were no conditions to be specified for the mixture was acceptable.

- Complied the UDF, fix.c using the interpreted UDF panel.
- Set the condition for the fluid zone respecting the impeller. Set the condition of water and sand separately there were no condition specified for the mixtures
- Set the condition on fix-zone for the water.
- Set the condition on fix-zone for the sand.

## 4.3 SOLUTION:

- Set the solution parameters, Under Relaxation Factors set pressure to 0.5 Momentum 0.2 and Turbulent viscosity to 0.8.
- Under Discretization, kept the default settings.
- Enabled the process of residuals during the calculations.
- Initialized the solution using the default initial values.
- Patch the initial sand bed configurations.
- Set the time stepping parameters, time step size to 0.005 .Under iteration set max.itterations per time step to 40.
- Saved the initial case &data files.
- Created a zone surface for fix zone to display the initial velocities in the fluid zone.

### 4.3.1 FIX- ZONE:

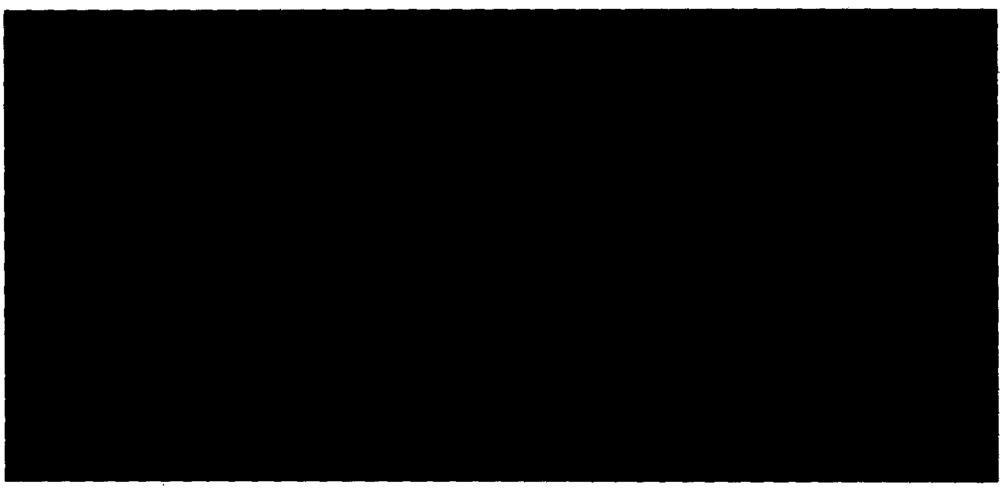

**Fig 4.4 Fix- Zone**

## 4.3.2 INITIAL IMPELLER VELOCITIES FOR WATER

Fluent displays the water velocity fixes at the impeller location.

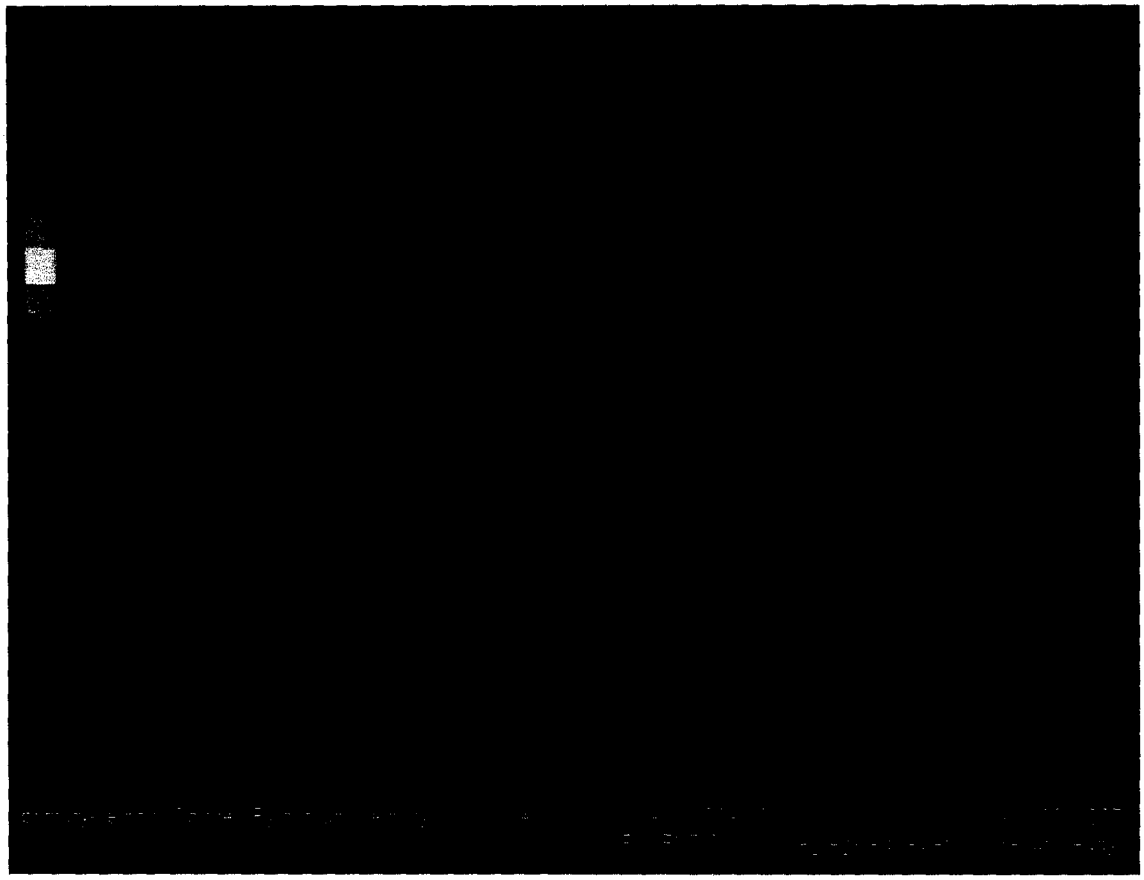

**Fig.4.5 Primary water velocity Magnitude**

## 4.3.3 INITIAL IMPELLER VELOCITIES FOR SAND

Fluent display the sand velocity fixes at the impeller location.

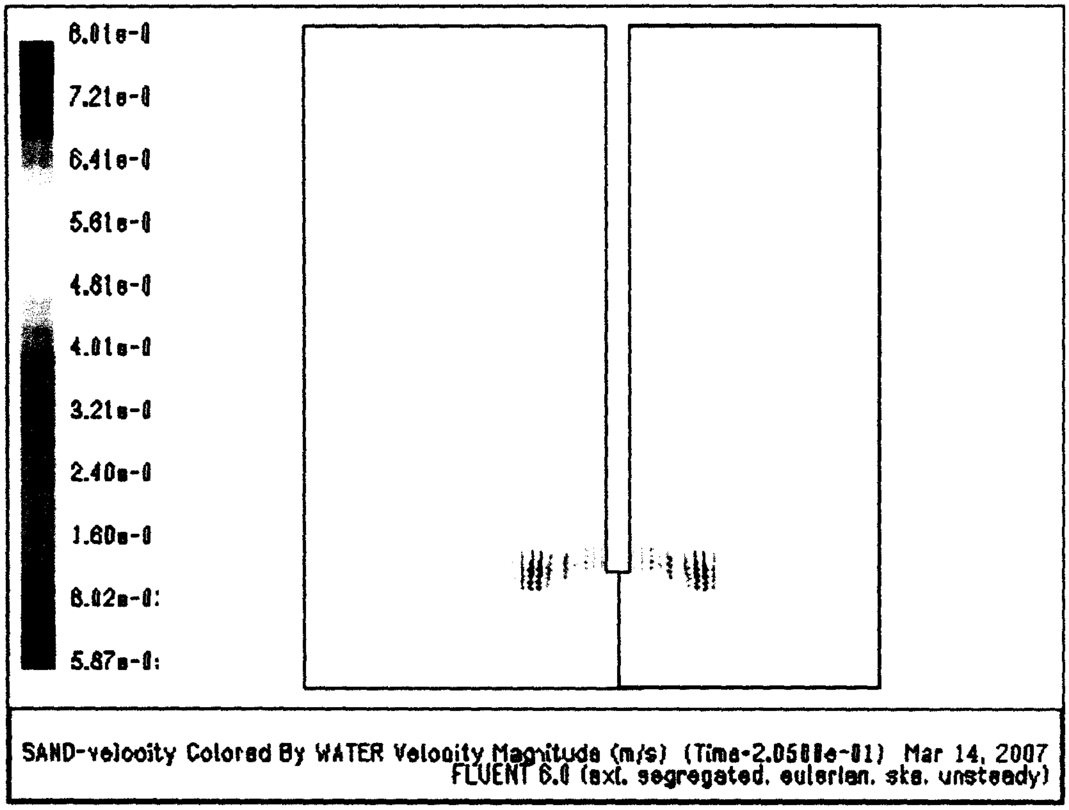


**Fig.4.6 Initial Sand Velocity Magnitude**

### 4.3.4 INTIAL STELED SAND BED

**Fig.4.7 Initial settled sand bed**

### 4.3.5 AFTER 1SEC

Run the calculation for 1 second with the number of time steps to 200.

**Fig.4.8 Sand velocity at fix- zone – After 1 sec**

**Fig.4.9. Water velocity at fix- zone – After 1 sec**

**Fig.4.10. Sand velocity colored by Water velocity magnitude – After 1 sec**

**Fig 4.11 Sand velocity colored by sand velocity magnitude – After 1 sec**

**Fig.4.12. Water velocity colored by Water velocity magnitude – After 1 sec**

Fig.4.13 Volume fraction of sand – After 1 sec

## 4.4 POST PROCESSING

Now examined the progress of the sand and water in the mixing tank after total of twenty seconds the mixing tank had nearly, but not quite reached a steady flow solution.

**Fig.4.14 Velocity Vectors for Sand - After 20sec**

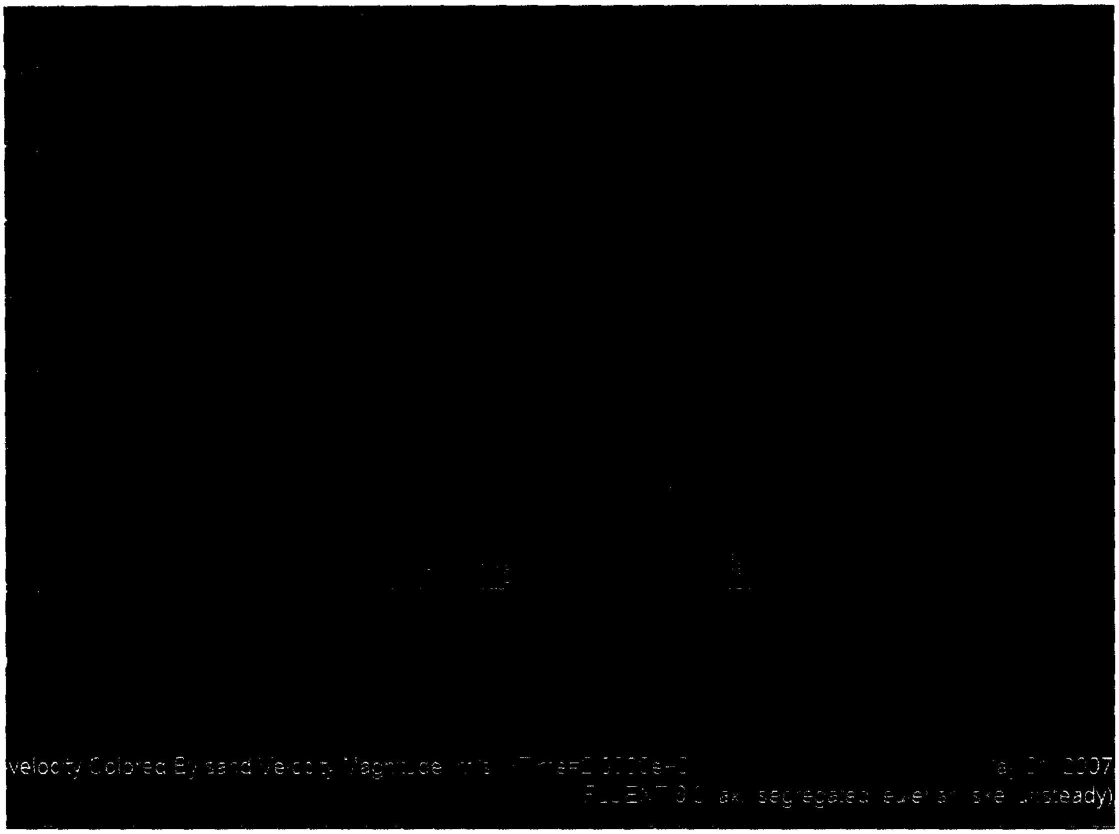

**Fig.4.15 Velocity Vectors for Water - After 20sec**

**Fig.4.16 Velocity Vectors for Water - After 20sec**

Fig.4.17 Path lines colored by sand velocity magnitude - After 20sec

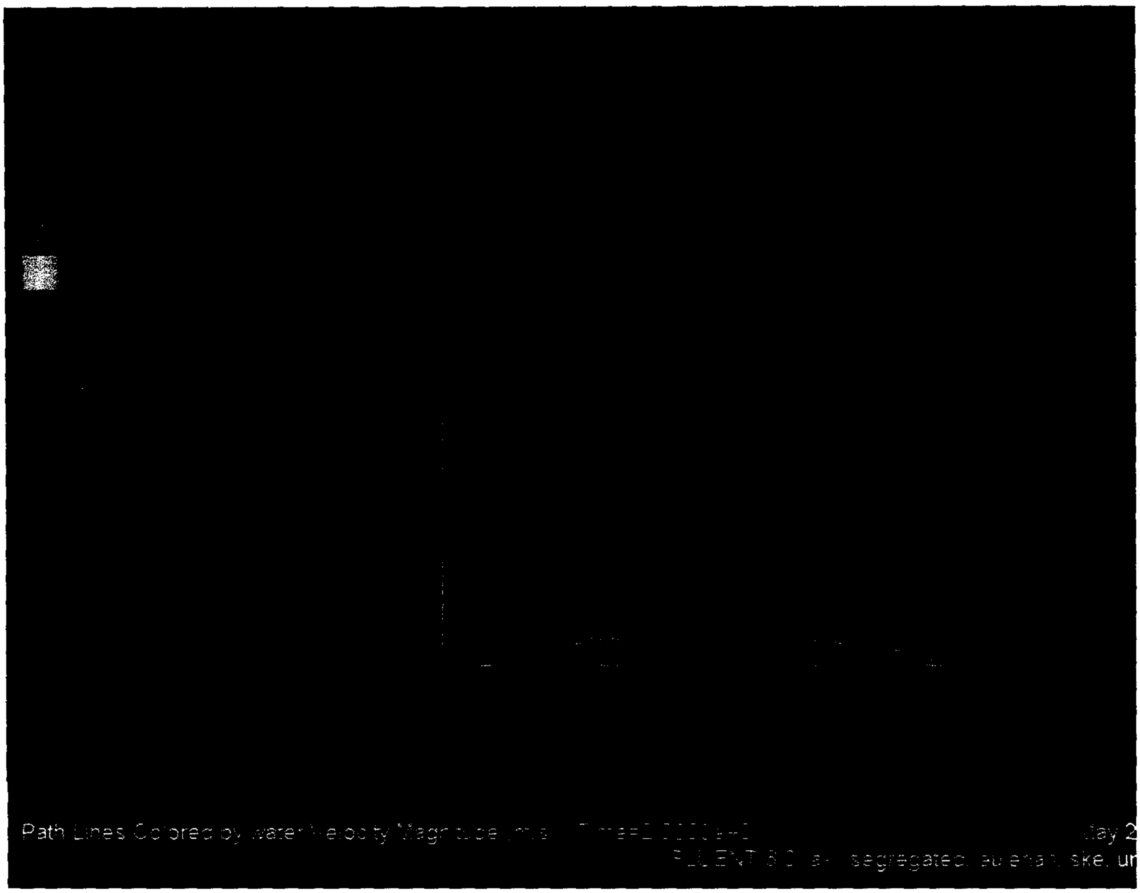

**Fig.4.18 Path Lines colored by Water Velocity Magnitude - After 20sec**

**Fig.4.19 Sand Volume Fraction - After 20sec**

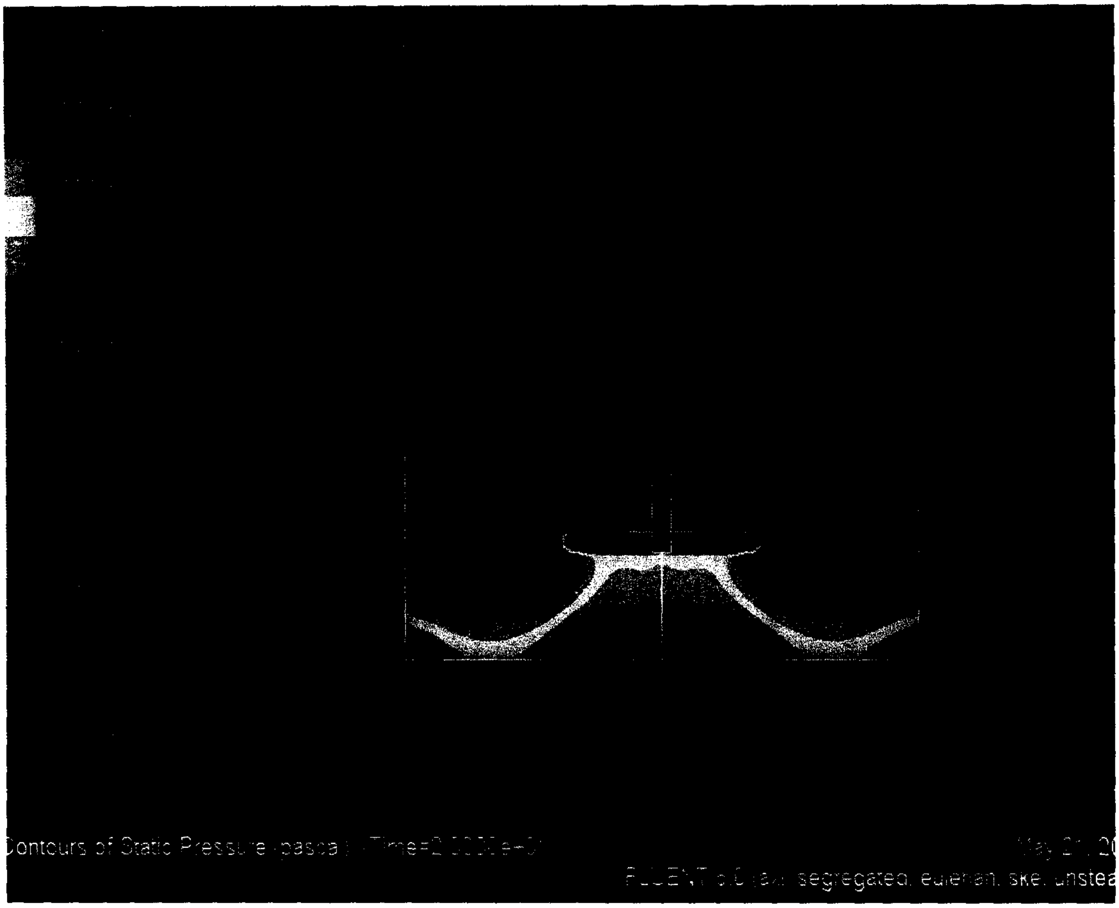

**Fig.4.20 Static Pressure - After 20 Sec**

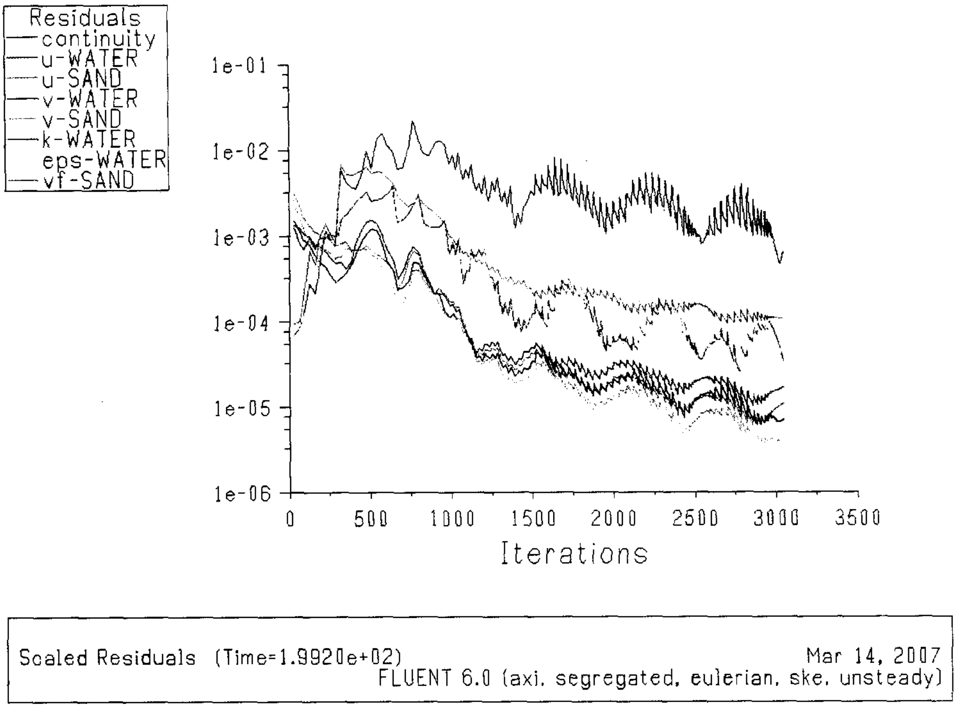


**Fig.4.21 Scaled Residuals**

# CHAPTER 5

# RESULTS AND DISCUSSIONS

# CHAPTER 5

# RESULTS AND DISCUSSIONS

The following are the inferences made from the simulation reports

**AFTER 1SEC**

- ❖ The circulation of sand around the impeller was significant but no sand vectors were plotted in the upper part of the tank, where the sand was not present as shown in (Fig.4.10).
- ❖ The water velocity shows that the circulation was confined to the region near the impeller, and had not enough time to develop in the upper portion of the tank, as shown in (Fig.4.12).
- ❖ The contours of sand volume fraction shows that the action of the impeller draws clear fluid from above the originally settled bed and mixes it into the sand .To compensate, the sand bed was lifted up slightly. The maximum sand volume fraction had increased as result of settling underneath the impeller and near the outer radius of the tank as shown in (Fig.4.13).

**AFTER 20SEC**

- ❖ In the post processing stage, the mixing tank had nearly, but not quite reached a steady flow solution.
- ❖ In the sand velocity vector, the sand suspended much high in the mixing tank, but it did not reach the upper portion of the tank. Because the water velocity in this region was not sufficient to overcome the gravity force on the sand particles as shown in (Fig.4.14).
- ❖ In the water velocity vector, the circulation of water was very strong in the bottom portion of the tank as shown in (Fig.4.15 & Fig.4.16) the contours of sand volume fraction after 20 seconds of operation was shown in (Fig.4.17).
- ❖ The Pressure field represents the hydrostatic pressure except for some slight deviations due to the flow of the impeller near the bottom of the tank as shown in (Fig.4.20).

# CHAPTER 6

# CONCLUSION

# CHAPTER 6

# CONCLUSION

In a nutshell, the highlights of the project work are briefed hereunder.

- This project aimed to simulate the slurry conditions, in an agitated vessel
- Velocity magnitude, static pressure , and sand volume fraction of the slurry were simulated
- In the simulation process, the slurry considered were the mixture of sand and water
- To model the experimental setup AutoCAD 2004 was used
- To simulate the flow field , Fluent 6.0 simulation software used
- At the end of the assignment , I have learnt to

    - Use the granular Eulerian multiphase model
    - Specify fixed velocities with a user defined function (UDF) to simulate the impeller
    - Set boundary conditions for internal flow.
    - Calculate a solution using the segregated solver
    - Solve a time –accurate transient problem

**It can be satisfactorily concluded that**

- The slurry condition in agitated vessel were simulated.

- CFD can be used to model solid suspension in agitated vessels .Process Design and analysis can be rapidly performed to scale geometry, modify tank geometry, baffling and to evaluate agitator performance rapidly.

- The predicted results for the solid concentration distribution in the tank shows the present procedure can be applied to the system with high solid concentration up to 20 %.

- This simulation imitates the reality and gives the realistic view for the further analysis.

# REFERENCES

## REFERENCES


1. Bonila,C.F., Cervi,A., Colven,T.J., Wang,S.J.,1959, "Heat transfer to Slurries in Pipe, Chalk and water in turbulent Flow", Chemical Engineering Progress Symbosium Series No.5, pp. 127-134.
2. Pandey,K.S., Pandey,S.K., and Sachdev,P.L.,1978, " Reduction of Nickel Oxide by Hydrogen" Indian Journal of Engineers , Volume 18, pp.17-20.
3. Pandey,S.K.,1978, "Heat Transfer Studies of agitated liquids"., Journal of Chemical Engineering World, Volume 13, pp. 47-50.
4. Pandey,S.K.,1978, "Programmed for Air Quality control", Journal of Chemicals and Petrochemicals, Volume 9, pp.29-32.
5. Pandey,S.K., and Sharma,P.R.,1982, "Nucleate Pool Boiling of Binary Liquid Mixtures at V Sub atmospheric Pressures" , Paper presented and published in the proceedings of the sixth National heat and Mass Transfer Conference I.I.T. Madras, pp. 36.
6. Sinha,S., 1970 "Phosphate Fertilizers in India" , Paper published in CHESS Technical Journal , Published by chemical Engg, Students Society, University of roorkee, Volume 5, pp.312-319
7. Kesavilambi,N., Pandy,S.K., Subramaniam,P., and Arunachalam,V.R.,1984, "Entrainment Losses in Enhanced settling by Application of Buoyant Particles", paper was presented and published in the proceedings of the 6th National Conference on Fluid Mechanics and Fluid Power, Regional Engineering College, Trichy, pp. 12.
8. Pandey,S.K.,1985, "Heat Transfer Studies in agitated liquid Mixtures" , Paper was presented at 37th Annual Session , Indian Institute of Chemical Engineers at New Delhi, pp. 17-20.
9. Pandey,S.K., and Varshney,B.S.,1985 " Agitation and Heat Transfer in Two Phase Liquid Mixtures" ,Paper was presented and published in the proceedings of the Eighth National , Heat and Mass transfer conference at college of Eng, Andhra University Visakapatnam , pp. 29-31.

10. David S. Dickey., Richard W. Hicks.,1976, "Fundamentals of agitation", Journal of Chemical Engineering, pp.93-100.
11. Harnett,J.P., Roshenow,W.M., "Handbook of Heat transfer", McGraw Hill,1999.
12. Perry,R.H., Chilton,C.H., "Chemical Engineer's Handbook", McGraw Hill,1973.
13. Phillip J. Ross, "Taguchi Techniques for quality Engineering", McGraw Hill, 1968.
14. Dream,R.F.,1999, "Heat Transfer in Agitated Jacketed Vessels", journal of Chemical Engineering, volume 33, pp. 112-119.
15. Myers K.J and Bakker, "Solids suspension with up-pumping pitched –Blade and high Efficiency impellers", Canadian Journal of Chemical Engineering, 1998.volume 64, pp. 634-713.
16. Olds hue J.Y, "Fluid Mixing Technology", McGraw Hill, 1998.