

**NUMERICAL EVALUATION OF FLOW THROUGH A CENTRIFUGAL
SLURRY PUMP HANDLING FLY ASH USING CFD**

Thesis submitted in partial fulfillment of the requirement
for the award of degree of

Master of Engineering

In

CAD/CAM & Robotics

**SUKHDEEP SINGH
(801081030)**

Under the supervision of

Mr. SATISH KUMAR

Assistant Professor

Department of Mechanical Engineering.

Thapar University, Patiala



**MECHANICAL ENGINEERING DEPARTMENT
THAPAR UNIVERSITY
PATIALA – 147004
JULY-2012**


DECLARATION

I hereby declare that the work which is being presented in the dissertation work entitled, "NUMERICAL EVALUATION OF FLOW THROUGH A CENTRIFUGAL SLURRY PUMP HANDLING BOTTOM ASH & FLY ASH USING CFD.", in partial fulfillment of the requirement for the award of degree of Master of Engineering in Mechanical Engineering with specialization in CAD/CAM & ROBOTICS submitted in Mechanical Engineering Department of Thapar University, Patiala, is an authentic record of my own work carried out under the supervision of Mr. Satish Kumar (Assistant Prof. MED) refers other researcher's works which are duly listed in the reference section.


The matter presented in this thesis has not been submitted for the award of any other degree of this or any other university.



Sukhdeep Singh

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


Mr. Satish Kumar
Assistant Professor, MED
Thapar University, Patiala

Counter signed by


Dr. Ajay Batish
Professor & head
Mechanical Engineering Department
Thapar University, Patiala


Dr. S.K. Mohapatra
Dean of Academic Affairs
Thapar University, Patiala

ACKNOWLEDGEMENT

I would like to show my gratitude and indebtedness to my thesis guide Mr. Satish Kumar, Assistant Professor, MED Thapar University, Patiala for his invaluable guidance and suggestions which inspired me to submit thesis report in the present form.

I am highly grateful and thankful to other faculty members of Mechanical Department, TU, Patiala for their intellectual support who sincerely devoted their time and service for every activity and task that boosted my self esteem and taught me to be more responsible for my own good and as for others.

It is a great pleasure to thank the authorities of Thapar University, Patiala for providing this opportunity to carry out my thesis work successfully.

I owe sincere and earnest thankfulness to my family members who enthused, encouraged and fully supported me for every trial that comes my way. They did not supported just financially, but also morally and spiritually.

I want to express my appreciation to all the people who have given their heart whelming full support in making this compilation a magnificent experience.

Last but not the least, To God the father of all, I thank for the strength that keeps me standing and for the hope that keeps me believing that this affiliation would be possible and more interesting.

SUKHDEEP SINGH

ABSTRACT

Centrifugal pumps are used for transportation of solids and liquids over short to medium distance through the pipelines. A centrifugal pump designed to handle the liquids is normally single stage, end suction type having radial or mixed flow configuration of blades.

Centrifugal pump radial-flow type is the most common in slurry service. A conventional centrifugal pump is designed to handle clear liquids, bottom ash and fly ash slurry. The modifications incorporated in the pump include enlargement of flow passages to accommodate bigger solid particles, robust impeller with smaller number of vanes, special seals and proper material of construction to ensure longer life. These have to be operated with relatively wide clearance at impeller-casing contacts to minimize choking and localized wear. These modifications increase the hydraulic losses in the pump and deteriorate the pump performance. Computational fluid dynamics (CFD) is being increasingly applied in the design of the centrifugal pumps. 3-D numerical computational fluid dynamics tool can be used for simulation of the flow field characteristics inside the turbo machinery. Numerical simulation makes it possible to visualize the flow condition inside a centrifugal slurry pump, and provides the valuable hydraulic design information of the centrifugal slurry pumps. The present study is concerned with the evaluation of the performance characteristics of a centrifugal slurry pump when handling clear water and numerical simulation with different concentration namely 10%, 20%, 30%, 40% and 50% of bottom ash and fly ash slurry at different speeds namely 1000rpm, 1150rpm, 1300rpm and 1450rpm. The pump characteristics of centrifugal slurry pumps are influenced by various particle parameters such as concentration and flow velocity of slurries. Slurry flow through centrifugal slurry pump has been modeled using the Mixture approach with Multiphase model. A mixture property based $k-\varepsilon$ turbulence model has been used using turbulence. A numerical model of an impeller and casing has been generated and the complex internal pressure and velocity distribution are investigated by using the CFX commercial computational code.

TABLE OF CONTENTS

TITLE	PAGE NO.
Declaration	i
Acknowledgement	ii
Abstract	iii
Contents	iv
List of figures	vii
List of tables	viii
Chapter 1: Introduction	
1.1 Thermal Power Plant	1
1.2 Ash Disposal System	1
1.3 Classification of Ash Handling System	3
1.3.1 Mechanical Handling System	3
1.3.2 Hydraulic Conveying System	4
1.3.3 Pneumatic Conveying System	7
1.3.4 Steam Jet System	9
1.4 Ash Disposal Using Centrifugal Pump	9
1.4.1 Working Principle of Centrifugal Pump	9
1.4.2 Centrifugal Slurry Pump	11
1.4.2 Applications of Centrifugal Slurry Pump	13
1.5 Motivation of the present work	21
Chapter 2 Literature review	22
Chapter 3 Experimental Investigation of Centrifugal Slurry Pump	29
3.1 Experimental Set-Up	29
3.1.1 Mixing Tank	29
3.1.2 Measuring Tank	31
3.1.3 Water Flush by Pump	32
3.1.4 Centrifugal Slurry Pump with VFD	32
3.1.5 Variable Frequency Drive (VFD)	32
3.1.6 Electro Magnetic Flow Meter	33
3.1.7 Density Monitoring Instrument	33
3.1.8 Pressure Transducers	34
3.1.9 Technical Specifications	34

3.2 Performance Characteristics of Centrifugal Slurry Pump	35
3.2.1 Calculation of Head	35
3.2.2 Correction of Head and Discharge	36
3.2.3 Measurement of Slurry Concentration	36
3.2.4 Power Output	36
3.2.5 Efficiency of Pump	37
3.2.6 Head Ratio	37
3.2.7 Efficiency Ratio (ER)	37
3.2.8 Power Ratio (PR)	37
3.3 Performance curves of water	38
Chapter 4 Computational Fluid Dynamics (CFD)	39
4.1 Introduction to Computational Fluid Dynamics (CFD)	39
4.2 History of CFD	39
4.3 Experiments Vs. Simulations	40
4.4 CFD Methodology	41
4.4.1 Initial design	41
4.4.2 Geometry generation	41
4.4.3 Mesh generation	42
4.4.4 Pre-processing	42
4.4.4.1 Boundary Conditions	42
4.4.4.2 Turbulence models	43
a) k - ϵ turbulence model	44
b) The k - ω model	44
(c) The Shear Stress Transport (SST) model	44
4.4.4.3 Solver	45
a) Finite Difference Method	45
4.5 Multiphase Flow	46
4.5.1 Eulerian-Eulerian Model	46
4.5.2 Inhomogeneous Multiphase Flow	47
4.5.3 Homogeneous Multiphase Flow	47
4.5.4 Multicomponent Multiphase Flow	47
4.5.5 Volume Fraction	47
4.5.6 Free Surface Flow	48
4.5.7 Surface Tension	48
4.5.8 Maximum Packing	48
4.6 Post-Processing	48
4.7 Modeling of the Pump Components	49
4.7.1 Impeller	49
4.7.2 Casing	50
4.7.3 Follower plate, Frame and Flange	50
4.7.4 Inlet passage	50

4.8 Preparation of CAD model and Mesh generation	50
4.9 Boundary Conditions	53
4.10 Performance curves of fly ash	56
4.11 Pressure Distribution of fly ash	58
4.12 Validation of Results	60
Chapter 5 Conclusion and Future Scope	62
References	63

FIGURE NO.	DESCRIPTION	PAGE NO.
Figure 1.1	General lay out of a thermal power plant	1
Figure 1.2	General Outline of Generation of Ash	3
Figure 1.3	Mechanical Handling Systems	4
Figure 1.4	Hydraulic Conveying Systems	5
Figure 1.5	Low Pressure Handling Systems	6
Figure 1.6	High Pressure Handling Systems	6
Figure 1.7	Pneumatic System	8
Figure 1.8	Flow of Fluid in a Centrifugal Pump	10
Figure 1.9	Installation of Centrifugal Slurry Pump	12
Figure 3.1	Schematic diagram of pilot plant test loop	29
Figure 3.2	Photographic view of Centrifugal slurry pump set-up	30
Figure 3.3	Performance curve between head v/s discharge	38
Figure 3.4	Performance curve between efficiency v/s discharge	38
Figure 4.1	Isometric view of geometry of outlet casing	51
Figure 4.2	Isometric view of geometry of impeller	51
Figure 4.3	CAD model of inlet passage	52
Figure 4.4	Assembled view of centrifugal pump	52
Figure 4.5	Meshed model of centrifugal pump	52
Figure 4.6	Centrifugal pump with boundary conditions applied	55
Figure 4.7	Performance curve between of Fly Ash at 1450 rpm	56
Figure 4.8	Performance curve between of Fly Ash at 1300 rpm	56
Figure 4.9	Performance curve between of Fly Ash at 1150 rpm	56
Figure 4.10	Performance curve between of Fly Ash at 1000 rpm	57
Figure 4.11	Pressure & Velocity distribution of Fly Ash at 10%	58
Figure 4.12	Pressure & Velocity distribution of Fly Ash at 20%	58
Figure 4.13	Pressure & Velocity distribution of Fly Ash at 30%	59
Figure 4.14	Pressure & Velocity distribution of Fly Ash at 40%	59
Figure 4.15	Pressure & Velocity distribution of Fly Ash at 50%	59
Figure 4.16	Numerical-Experimental result of head /s discharge	60
Figure 4.17	Numerical-Experimental result of efficiency v/s discharge	61

TABLE NO.	DESCRIPTION	PAGE NO.
Table 3.1	Manufacturer's data for the pumps Used in the investigation	31
Table 4.1	Number of Elements	53
Table 4.2	Properties of Fly Ash	53

1.1 THERMAL POWER PLANT

A Thermal power station is a power plant in which the prime mover is steam driven. Water is heated by burning the coal, turns into steam and spins a steam turbine which drives an electrical generator. After it passes through the turbine, the steam is condensed in a condenser and recycled to where it was heated. The greatest variation in the design of thermal power stations is due to the different fuel sources. Some prefer to use the term energy center because such facilities convert forms of heat energy into electricity. Some thermal power plants also deliver heat energy for industrial purposes, for district heating, or for desalination of water as well as delivering electrical power. A large part of human CO₂ emissions comes from fossil fueled thermal power plants; efforts to reduce these outputs are various and widespread. Steam can be produced by using fuels such as coal, oil or gas, depending on the availability of fuel in the site.

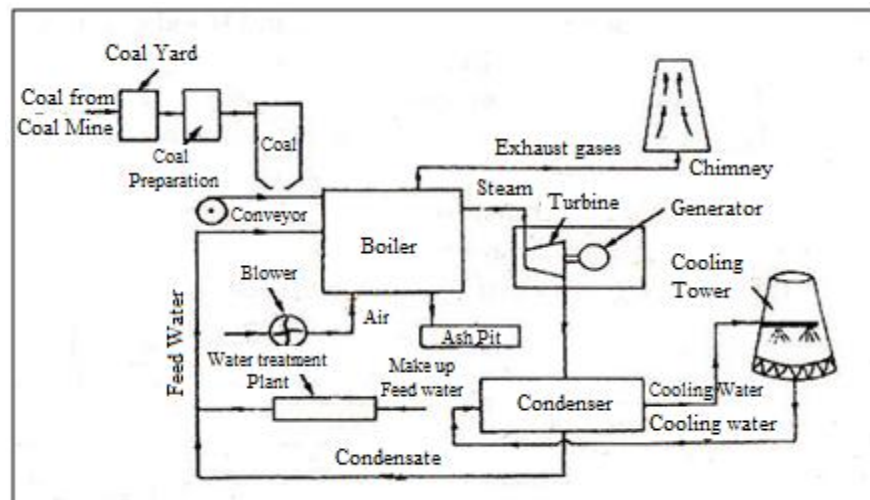


Figure 1.1 General lay out of a thermal power plant

1.2 ASH DISPOSAL SYSTEM

All the available coals have some percentage of ash. When the coal is burnt in a boiler furnace, about 10 to 20% of quantity of coal used results in ash. In the modern large steam power plants

where huge amounts of coal are used, the amount of ash may be go up to many thousands tonnes of ash per year. Theoretically whole of the ash from the furnace should get deposited in the ash hoppers, but actually from 5 to 40% of it leaves with the outgoing gases. Ash handling comprises the following operations:

1. Removing the ash from the furnace ash hoppers.
2. Conveying this ash to a fill or storage by means of conveyors.
3. Disposal of the stored ash.

To handle huge amounts of ash per day, mechanical means are employed. The advantage of mechanical handling over manual handling has been already enumerated under the topic of coal handling. The ash handling and disposal system can work continuously or intermittently. The following are some of the places where the ash can disposed off.

- Where seaborne coal is used, barges may take the ash to sea for disposal into a watery grave.
- Disused queries within reasonable distance of the power station may be used for dumping the ash into evacuated land.
- Building contractors may use it to fill the low lying areas.
- Wasteland sites may be reserved for the disposal of the ash.
- Deep ponds may be constructed and the ash can be dumped into these ponds and when they are completely filled, they may be covered with soil and seeded with grass.

Ash handling is major and difficult problem due to the following difficulties encountered in its handling and disposal:-

- Ash is dusty and so irritating and annoying in handling.
- It is hot when it comes out of the boiler furnace.
- It is abrasive and wears out the containers.
- Poisonous gases are produced.
- Corrosive acids are produced in water.
- It forms clinkers by fusing together in lumps.

The following points should be kept in view while handling and disposing ash:

- Locate the ash plant on the leeward side of the power station to avoid blowing in and

drawing into the buildings of the dry ash.

- If the ash is cold and cannot be disintegrated, it may be better to crush it before passing it further or otherwise it might choke the sluiceway.
- In case of pulverized fuel firing, 60 to 80% of ash is in the form of dust and fly ash; the plant should be designed accordingly.

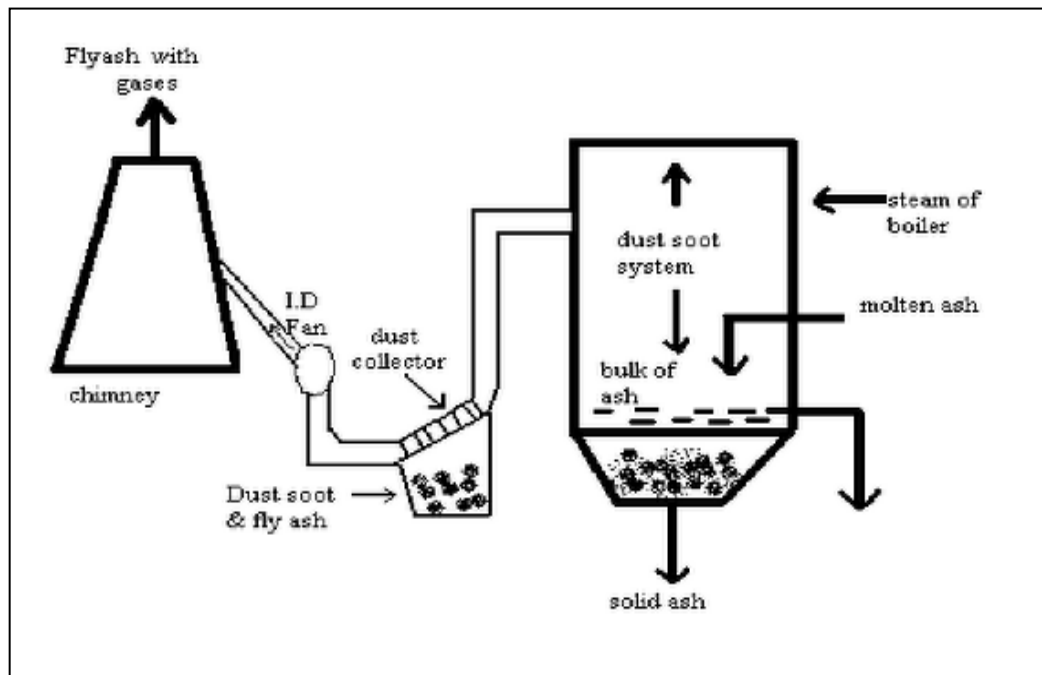


Figure 1.2 General Outline of Generation of Ash

1.3 CLASSIFICATION OF ASH HANDLING SYSTEM

The modern ash handling systems are mainly classified into four groups:-

1.3.1 Mechanical Handling System

This system is generally employed for low capacity power plants using coal as fuel. The hot ash released from the boiler furnaces is made to fall over the belt conveyor after cooling it through water seal. This cooled ash is transported to an ash bunker through the belt conveyor. From ash bunker the ash is removed to the dumping site through trucks.

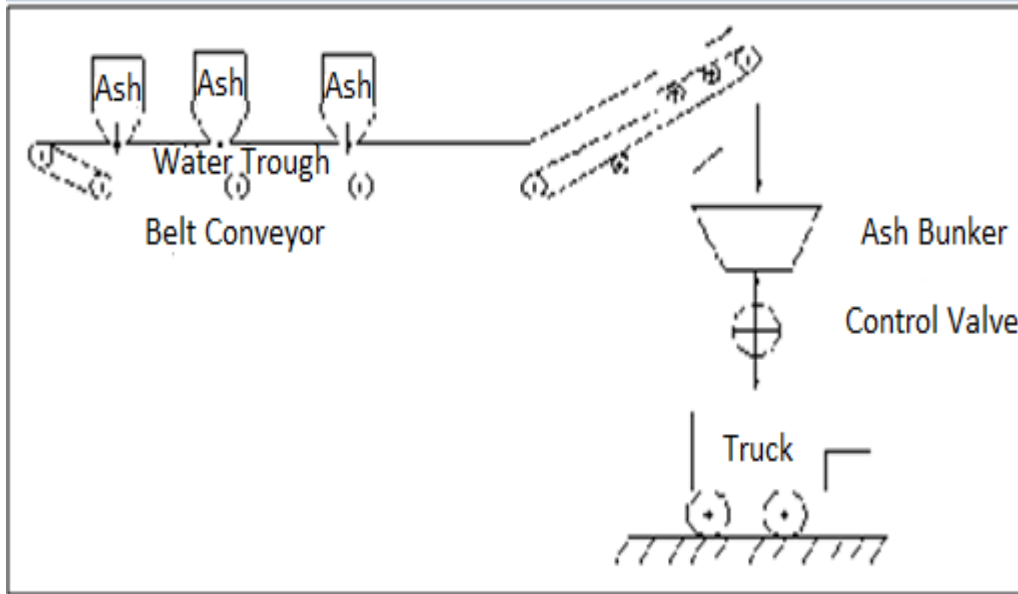


Figure 1.3 Mechanical Handling System

1.3.2 Hydraulic Conveying System

In this system ash is carried with the flow of water with high velocity through a channel and finally dumped in the sump. The Hydraulic Conveying System is a typical bottom ash handling system adopted by many domestic coal-fired power plants. This system has basically two variations. One is the Dewatering Bin Recovery system with a dewatering bin and the other is the Hydraulic Conveying System without a dewatering bin to convey ash directly to an ash dump.

Bottom ash which is cooled and stored temporarily at a clinker hopper is discharged regularly, crushed by a crusher, conveyed to a dewatering bin by a jet pulsion pump, and carried to an ash dump by truck or conveyor after dewatering.

Wastewater from the dewatering bin goes into an ash sedimentation tank or an ash sedimentation pond for the sedimentation of ash particles and water without ash particles is supplied to the JPP by an ash handling pump. The ash particles which settle in the ash sedimentation tank and the water tank are returned to the dewatering bin by a sludge return pump.

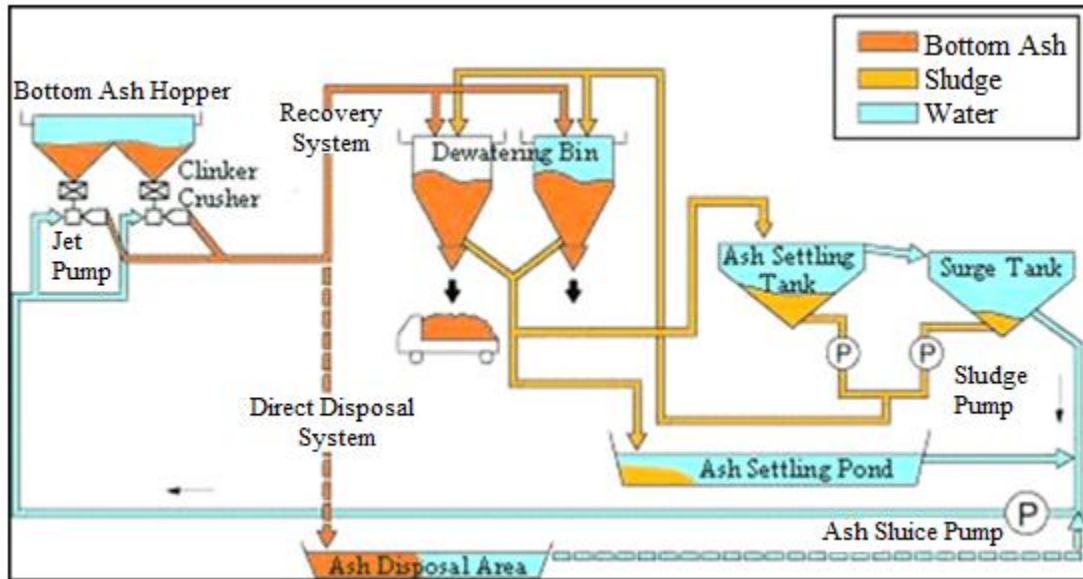


Figure 1.4 Hydraulic Conveying System

Features of Hydraulic system:-

1. This system is a conveying system by water circulation.
2. Most maintenance work can be done without stopping the system due to its intermittent handling.
3. This system can handle mill pyrite along with bottom ash.

The Hydraulic system is subdivided as follows.

(a) Low pressure system. (b) High Pressure system.

(a) Low Pressure System. In this system a trough or drain is provided below the boilers and water is made to flow through the trough. The ash directly falls into the troughs and is carried by water to sumps. In the sump the ash and water are made to pass through a screen so that water is separated from ash; this water is pumped back to the trough for reuse and ash is removed to the dumping yard. The ash carrying capacity of this system is 50 tones/hour and distance covered is 500 meters.

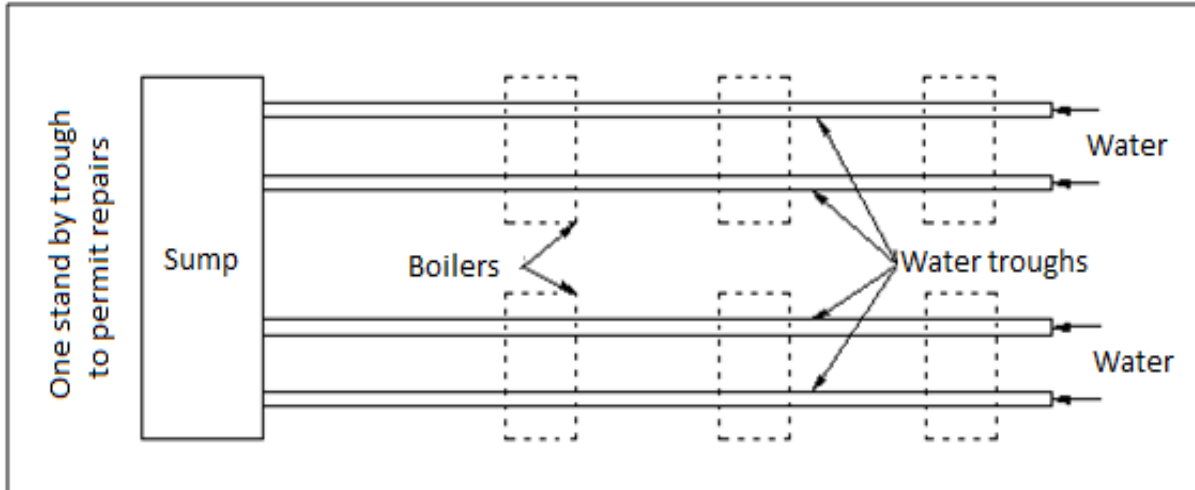


Figure 1.5 Low Pressure Handling System

(b) High Pressure System. The hoppers below the boilers are fitted with water nozzles at the top and on the sides. The top nozzles quench the ash while the side one provides the driving force for the ash. The cooled ash is carried to the sump through the trough. The water is again separated from ash and recalculated.

The ash carrying capacity of this system is as large as 120 tones per hour and the distance covered is as large as 100 meters.

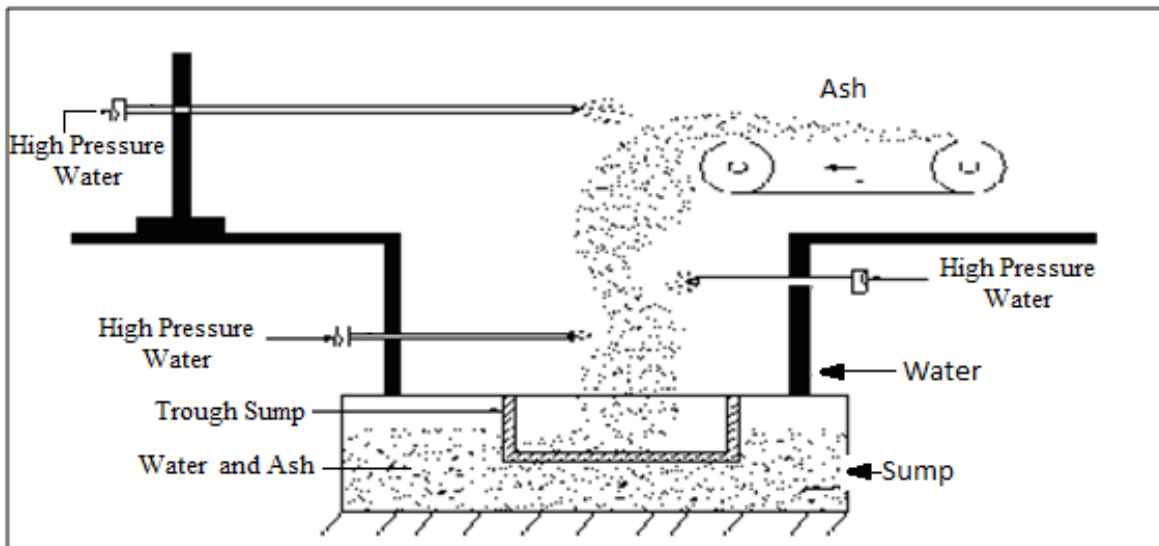


Figure 1.6 High Pressure Handling System

Advantages of Hydraulic System:

- The system is clean and healthy.
- It can also be used to handle stream of molten ash.
- Working parts do not come into contact with the ash.
- It is dustless and totally closed.
- It can discharge the ash at a considerable distance (1000 m) from the power plant.
- The unhealthy aspects or ordinary ash basement work is eliminated
- Its ash carrying capacity is considerably large, hence suitable for large thermal power plants.

1.3.3 Pneumatic Conveying System

Fly ash in each hopper goes through a pneumatic conveyor and is fed into a pressurized stream produced by a pneumatic blower and blown into a fly ash silo directly for storage. An exhaust bag filter is installed on the fly ash silo to purify air for the pneumatic conveying system. After storage in the silo, fly ash is conveyed like the vacuum conveying system.

The pneumatic conveying system conveys fly ash in an air stream at positive pressures and is suitable for a large capacity conveying for a long distance. This system is also suitable when fly ash is distributed to multiple storage places. This system can handle abrasive ash as well as fine dusty materials such as fly ash and soot. It is preferable for the boiler plants from which ash and soot must be transported some far off distance for final disposal. The exhauster provided at the discharge end creates a high velocity stream which picks up ash and dust from all discharge points and then these are carried in the conveyor pipe to the point of delivery. Large ash particles are generally crushed to small sizes through mobile crushing units which are fed from the furnace ash hopper and discharge into the conveyor pipe which terminates into a separator at the delivery end.

The separator working on the cyclone principle removes dust and ash which pass out into the ash hopper at the bottom while clean air is discharged from the top. The exhauster may be mechanical or it may use steam jet or water jet for its operation. When a mechanical exhauster is used it is usually essential to use a filter or washer to ensure that the requirements are less. Steam exhauster may be used in small and medium size stations. Where large quantities of

water are easily and cheaply available water exhauster is preferred. The ash carrying capacity of this system varies from 25 to 15 tones per hour.

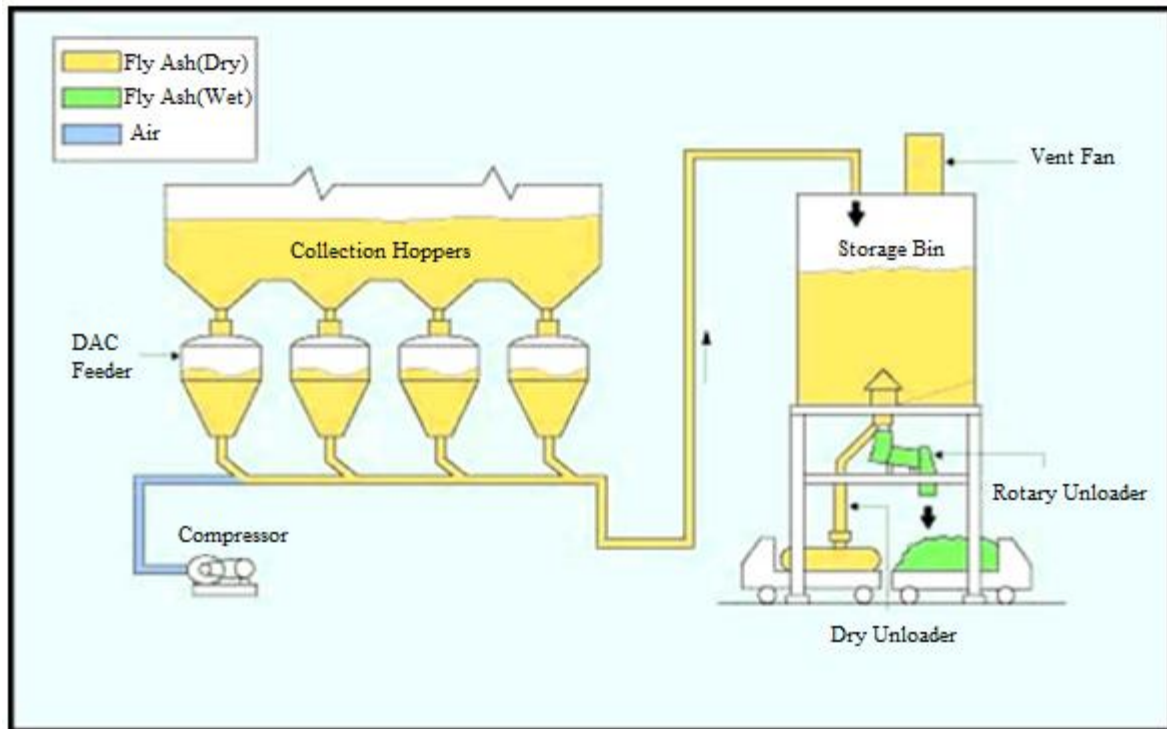


Figure 1.7 Pneumatic System

Advantages:

- No spillage and re-handling.
- High flexibility.
- There is no change of ash freezing or sticking in the storage bin and material can be discharged freely by gravity.
- The dustless operation is possible as the materials are handled totally in an enclosed conduit.
- The cost of plant per ton of ash discharged is less comparison to other system.

Disadvantages:

- There is a large amount of wear in the pipe work necessitating high maintenance charges.
- More noisy that other systems.

1.3.4 Steam Jet System

In this case steam at sufficiently high velocity is passed through a pipe and dry solid materials of considerable size are carried along with it. In a high pressure steam jet system a jet of high pressure steam is passed in the direction of ash travel through a conveying pipe in which the ash from the boiler ash hopper is fed. The ash is deposited in the ash hopper. The system can remove economically the ash through a horizontal distance of 200 m and through a vertical distance of 30 m.

Principle Requirements of a good Ash Handling Plant

1. The plant should be able to operate with minimum personal attention and should be able to handle large clinkers as well as dust and soot.
2. Precautions should be taken against abrasiveness of ash.
3. It should be able to handle both wet and dry ash and operate with little noise and keep the dust menace to minimum.
4. Disposal of ash from the plant site should be speedy.
5. The plant should have adequate capacity to deal with the ultimate station capacity.

1.4 ASH DISPOSAL USING CENTRIFUGAL PUMP

The selection of a pumping system for any slurry transportation system is governed more by the practical considerations rather than purely on economical considerations of maximum efficiency. However, discharge pressure and the abrasivity are the two key factors for the selection of a pump.

1.4.1 Working Principle of Centrifugal Pump

The centrifugal pump works on the principle of forced vortex flow, which means that when a certain mass of liquid is rotated by an external flow, the rise in pressure head of the rotating liquid takes place. The rise in pressure head at any point of the rotating liquid is proportional to the square of tangential velocity of the liquid at that point. Thus at the outlet of the impeller where the radius is more, the rise in pressure head will be more and the liquid will be discharged at the outlet with high pressure head. Due to high-pressure head, the liquid can be lifted to a high level. Figure 1.9 shows flow of fluid in a centrifugal pump.

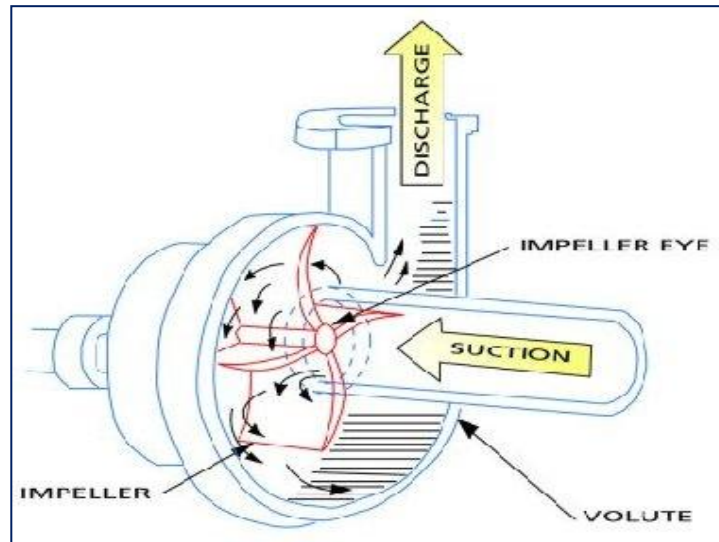


Figure 1.8 Flow of Fluid in a Centrifugal Pump

Advantages of using Centrifugal Pumps for Transportation of Slurry

- 1) Simplicity of design
- 2) Easier installation
- 3) Low maintenance
- 4) Lower weight
- 5) Handles suspensions and slurry easily

Centrifugal pumps are prevalent for many applications in the industrial or other sectors. Nevertheless, their design and performance prediction process is still a difficult task, mainly due to the great number of free geometric parameters, the effect of which cannot be directly predicted. The flow inside the impeller of the centrifugal pump is extremely difficult to predict because of the separation and recirculation of the flow at the inlet and exit.

The curvature of the blades has great influence on the flow structure. The flow structure became more complex with the addition of the solid particles in the fluid. This gives rise to another field of multiphase fluid system. These types of pump, called centrifugal slurry pumps, are extremely used in pipeline transportation system, mining industries, petroleum industries, sewage and sump services etc.

Centrifugal pumps are best suited for short distances and for in-plant slurry pipe line systems.

Though the discharge pressure of centrifugal pumps is relatively low, they can also be used for moderate pressure requirements when used in series. The centrifugal pumps are used for over 97% of all short distance slurry pipelines. The design of a centrifugal pump for slurry handling system needs special consideration to ensure that the flow passage are such as to offer no restriction to the passage of solids. The abrasivity of solids cause wears in the pumps. A vertical split casing design is necessary to provide ready access to the wearing parts for replacement

For a given duty, centrifugal pumps are usually cheaper, occupy less space and have lower maintenance costs than positive displacement types, and can handle much larger solids.

Centrifugal pumps are used in a variety of applications, such as, water supply and irrigation, power generating utilities, flood control, sewage handling and treatment, process industries, transporting liquid-solid mixtures. The list can go on and on. Because of their wide spectrum of application, centrifugal pump are manufactured in a variety of shapes and with a variety of performance characteristics.

1.4.2 CENTRIFUGAL SLURRY PUMP

There are a number of different pump types used in the pumping of slurries. Positive displacement and special effect types are used but by far the most common type of slurry pump is the centrifugal pump. The centrifugal slurry pump utilizes the centrifugal force generated by a rotating impeller to impart kinetic energy to the slurry. The selection process for centrifugal slurry pumps needs to include consideration for impeller size and design for solids passage, appropriate shaft seal possibilities and optimum, long life material selections. These basics need to be considered by the application engineer who will select the liquid end parts to withstand wear caused by the abrasive, erosive and/or corrosive attack on the wetted materials.

Slurry pump are widely used to transport corrosive/abrasive and high concentration slurry in many industries such as: Gold, Silver, iron ore, Lead & Zinc, Mineral sands, Tin, Steel, Titanium, Molybdenum, Copper, Electric Utilities, Oil Shale, Water & Sewage Utilities, Building areas, Coal, Sand & Gravel, Tobacco, Agriculture (hog, poultry, dairy manure), etc

Range of Applications of a Slurry Pump Slurry pumps are used widely throughout the beneficiation section of the mining industry where most plants use wet separation systems. These systems usually move large volumes of slurry through the process. Slurry pumps are also widely used for the disposal of ash from fossil fuel power plants. Other areas where slurry pumps are used include the manufacture of fertilizers, land reclamation, mining by dredges, and the long distance transportation of coal and minerals. Increased global focus on the environment and energy consumption will certainly generate much wider uses for slurry pumping in years to come.



Figure 1.9 a)



Figure 1.9 b)

Figure 1.9 a) and 1.9 b) Installation of Centrifugal Slurry Pump

Determining the when to use a slurry style centrifugal pump can be a challenging decision. Often the cost of a slurry pump is many times that of a standard water pump and this can make the decision to use a slurry pump very difficult. One problem in selecting a pump type is determining whether or not the fluid to be pumped is actually a slurry. We can define a slurry as any fluid which contains more solids than that of potable water. Now, this does not mean that a slurry pump must be used for every application with a trace amount of solids, but at least a slurry pump should be considered.

Slurry pumping in its simplest form can be divided into three categories: the light, medium and heavy slurry. In general, light slurries are slurries that are not intended to carry solids. The presence of the solids occurs more by accident than design. On the other hand, heavy slurries are slurries that are designed to transport material from one location to another. Very often the

carrying fluid in a heavy slurry is just a necessary evil in helping to transport the desired material. The medium slurry is one that falls somewhere in between. Generally, the Percent solids in a medium slurry will range from 5% to 20% by weight.

After a determination has been made as to whether or not you are dealing with a heavy, medium, or light slurry, it is then time to match a pump to the application. Below is a general listing of the different characteristics of a light, medium, and heavy slurry.

Light Slurry Characteristics:

- Presence of solids is primarily by accident
- Solids Size < 200 microns
- Non-settling slurry
- The slurry specific gravity < 1.05
- Less than 5% solids by weight

Medium Slurry Characteristics:

- Solids size 200 microns to 1/4 inch (6.4mm)
- Settling or non-settling slurry
- The slurry specific gravity < 1.15
- 5% to 20% solids by weight

Heavy Slurry Characteristics:

- Slurry's main purpose is to transport material
- Solids > 1/4 inch (6.4mm)
- Settling or non-settling slurry
- The slurry specific gravity > 1.15
- Greater than 20% solids by weight

1.4.2 Applications of Centrifugal Slurry Pump are divided in two parts:

Selection of a Slurry Pump - by duty

In this section we are selecting the optimal Slurry Pump simply against the Specified pump duty.

Selecting against duty means selecting pumps considering parameters like:

- Solids (size, shape, density etc.)
- Head (maximum, high or low)
- Capacity (maximum and minimum)
- Liquid (corrosive, thixotropic, frothy)

This guide is strictly based on technical performance reflecting various Solid/Liquid parameters.

Selection of a Slurry Pump - by industrial application

This section is more of a practical guide, based on experience by day to day applications, working in very different industrial environments.

a) How to pump

- Wood chips
- Mill scales
- Mineral tailings
- Leaching residue
- Industrial waste

b) How to feed

- a hydro cyclone
- a pressure filter
- a tube press
- a flotation machine

The guide is structured according to practical experience from hydraulic transportation of solids in the following industrial segments:

- Minerals (Metallic & Industrial)
- Construction
- Coal
- Waste & Recycling
- Power & FGD
- Pulp & Paper
- Metallurgy

- Chemical
- Mining & tunnelling

Selection of a Slurry Pump - by duty

A.) By Solids

- Coarse particles: Everything larger than 5 mm is considered to be coarse. Use metal pumps only. Do not use rubber pumps. Upper practical limit in particle size is normally 50 mm. Limitation is the impact on the impeller. Particle size must not exceed 1/3 of the pipe diameter.
- Fine particles: If the particles are sharp - use rubber. If particles are not sharp - use rubber or metal.
- Sharp (abrasive) particles: If sizes are below 5 mm - use rubber lined pumps. If particles are above 5 mm - use metal lined pumps.
- High percent solids: You have to be careful if the percent solids are getting close to 40% by volume. Above 50% the slurry is impossible to handle with centrifugal pumps. Only vertical tank pumps are able to handle applications with really high percent solids.
- Low percent solids: Choose the lightest and most cost effective pumps.
- Fibrous particles: The problem is blocking of particles and air blocking. Use induced flow impellers (Vortex).
- Slurry with no fine size particles: When all fine particles are removed from the slurry the solid settling rate can be critical and can call for severe de-rating of the pump performance.
- Pumping efficiency goes down for all pump types when used for this duty.
- Mixing: Tank pumps are excellent as mixers for liquid and fine, dry solids. When mixing water and solids look up the correct ratio between liquid and solids.

B.) By Head

- High head: Normally metal pump applications are due to the high peripheral speed on the impeller. If your application requires rubber lined pumps, "in series" pumping may be needed. Maximum head on hard metal pump 125 m. Maximum

head on rubber impeller 45 m. Note! High rates of wear occur at high speeds on centrifugal pumps.

- b. Varying head at constant flow: Use a multi-speed drive or a variable (frequency control) drive.
- c. Varying flow at constant head: Use variable (frequency control) drives.
- d. High suction lift: Metal pumps are preferred due to risk of rubber lining collapse on high suction lifts. Maximum practical suction lifts 5 - 8 m depending on Slurry S.G. Pumps are not self-priming, i.e. you need a priming device. The pump and suction pipe must be filled with liquid before starting.
- e. High flow: Use parallel pump installations. Risk of Cavitations.
- f. Low flow: Compare to Best Efficiency Point. At low flows rubber linings can overheat. Use metal. Be careful with VS pumps if heads are high and flow is low. Open vertical pumps have no problems.
- g. Duty: Fluctuating flow: Use horizontal pumps with variable speed drive or fixed speed vertical pumps.

C.) By liquid type

- a. Fragile slurries: Use induced flow impellers (fully recessed). Both metal and rubber pumps can be used. Both horizontal and vertical pumps can be used.
- b. Hydrocarbon slurries (oil and reagents contaminated): Natural rubber must not be used. Synthetic rubber can be used. Use metallic pumps or wear parts in polyurethane.
- c. High temperature (greater than 100o C) slurries: Temperature limit for natural rubber is 60° C. Check with your Pump Sales Support Group for pumps available with Synthetic rubber parts. Practical limit for operating temperature is 135° C. Above this temperature the bearings can be over-heated!
- d. Frothy slurries: Use a froth pump of vertical design.
- e. Hazardous slurries: This case must be referred back to your Pump Sales Support Group. Shaft sealing is critical from an explosion point of view. Normally closed pump systems are used.
- f. Corrosive slurries (low pH): For acidic duties use rubber or elastomers. For metal pumps with chrome iron parts the acid limit is pH 2.5. Seawater slurries (containing

chlorides) must have a rubber pump. CuSO₄ (used in flotation circuits) is extremely corrosive, use rubber pumps.

- g. High viscosity fluids (Newtonian): When viscosity increases up to 5 x viscosity of water, the pumping gets critical. With this restriction basically any pump in our range can be used, if properly sized.
- h. High viscosity fluids (non-Newtonian): These applications are very tricky and should be referred back to your Pump Sales Support Group.

Selection of a Slurry Pump - by industrial application

This selection guide is based on practical experience from various Slurry Pump applications within the following industrial segments:

- Metallic and industrial minerals
- Construction
- Coal
- Waste & recycling

A.) Industrial Segment: Metallic & industrial minerals

Application: Pumps for grinding circuits:

- Power & FGD
- Paper coating material (Calcium carbonate etc.)
- Metallurgy
- Chemical
- Mining & tunneling

For particles sizes below 5 mm use rubber. If possible mix flows containing coarse and fine particles together for better slurry stability.

- a. Pumps for froth: The VF range is specially designed for froth pumping. Be cautious when heads are greater than 15 m
- b. Pumps for floor sumps: Use VS Sump Pumps with metallic wear parts, since there is often a risk of oversize waste material coming into floor sumps. The VS pumps can also be equipped with agitator. If rubber must be used, put a strainer in front of the suction or around the pump.

- c. Pumps for tailing: Depending on particle size either rubber or metal pumps can be used. For installations with long distance pumping requirements use pumps in series.
- d. Pumps for hydro cyclone feed: For sharp classification use horizontal pumps of the X or H or VASA HD ranges. For dewatering cyclones use VT Tank Pumps.
- e. Pumps for pressure filter feed: High head needed with variable speed control (alternatively two-speed drive). Avoid rubber due to low flow heat build up
- f. Pumps for tube press feed: Small flow and high head, use metal pumps of the HM range. One pump can feed many tubes from a slurry distribution ring.
- g. Pumps for leaching: For acidic duties use rubber or elastomer. For metal pumps with chrome iron parts the acid limit is pH 2.5.
- h. Pumps for dense media (heavy media): High inlet head and high percent solids in combination with low discharge head can cause expeller seal leakage problems.
- i. Pumps for general purpose (mineral): Horizontal pumps of the MM and MR ranges are ideal for normal duty in mineral process circuits. If the wear is extreme, use the X and H ranges. Rubber is normally preferred in "hard rock" concentrators. For special applications use the vertical pumps.

B.) Industrial Segment: Construction

- a. Pumps for wash water (sand and gravel): Normally, vertical pumps of the VS and VT ranges are used. The M range horizontal pump is also suitable.
- b. Pumps for sand transportation: Horizontal pumps with rubber lining are preferred.
- c. Pumps for tunnel dewatering: Do not use rubber due to oil in slurry. As front pumps use drainage pumps. For the first transportation stage the VS Vertical Sump Pump is normally used. For horizontal distant pumping use HM range. For cuttings from full face boring use HM and MM ranges. For small tunnels (micro bore) use small HM range.
- d. Drainage pumps: For lighter duties use the M range of horizontal pumps. Diesel driven pump can be practical but these are not our standard version?

C.) Industrial Segment: Coal

- a. Pumps for coal washing: Generally metal pumps are used because of risk for oversized tramp material.

- b. Pumps for froth (coal) Use the VF Vertical Froth Pump.
- c. Pumps for dense media (coal): High inlet head and high percent solids in combination with low discharge head can cause expeller seal leakage problems.
- d. Pumps for coal/water mixtures: Use horizontal pumps of the M and H ranges.

D.) Industrial Segment: Waste & recycling

- a. Pumps for effluent handling: Light-duty application. Use either horizontal or vertical pumps. Metal pumps are the first selection.
- b. Hydraulic transportation of light waste: Use horizontal pumps with Vortex induced flow impellers.
- c. Pumps for soil treatment: The VT Vertical Tank Pumps are recommended for mobile and semi-mobile plants (no leaking seal and easy to transport and install).
- d. Pumps for general purpose (coal): The coal industry does not normally use rubber pumps.

E.) Industrial Segment: Power & FGD

- a. Pumps for FGD reactor feed (lime): Normally mineral applications use X, H and M ranges, all with rubber and/or metal parts. For slurry with more than 20000 ppm chloride iron use 30% CR. Rubber for high chloride concentrations.
- b. Pumps for FGD reactor discharge (gypsum): See lime pumps above for wear parts selection.
- c. Bottom ash pumping: Metal pumps are preferred due to temperature and particle size.
- d. Fly ash pumping: Metal is normally used due to risk of oil contamination. If pH is so low that rubber has to be used, look out for any oil or other chemicals or temperature too high, then natural rubber cannot be used.

F.) Industrial Segments: Pulp & Paper

- a. Pumps for liquors: Rubber is not to be recommended on black liquors due to risk of turpentine.
- b. Pumps for lime and caustic muds: These applications are normally of high temperature, therefore metal parts are recommended.

- c. Pumps for reject pulp (containing sand): Normally light duty, but metal parts are recommended. Normally we are competing with stainless steel pumps.
- d. Pumps for solids from debarking: An extra long VS Vertical Sump Pump has been developed for sand and bark. Use metal parts and induced flow impeller (Vortex).
- e. Pumps for hydraulic transportation of wood chips: Use horizontal pumps of the H and M ranges with induced flow (Vortex) impellers.
- f. Pumps for paper filler and coating slurries: No rubber allowed due to colour contamination.
- g. Floor spillage pump: Use a VS Vertical Sump Pump. Sometimes stainless steel parts are required due to low pH.
- h. Pumps for Mill Scale transportation: First choice is a VS Vertical Sump Pump with induced flow impeller and metal wear parts. For heavy mill scale handling can VS with agitator be useful.
- i. Pumps for slag transportation: Same considerations as for “Mill Scale” above.
- j. Pumps for wet scrubber effluents: Normally we recommend horizontal pumps of the M range or vertical pumps of the VS range. If pH is very low, use rubber. If pH is very low and temperature is very high, use stainless steel or synthetic rubber parts.
- k. Pumps for iron powder transportation: See dense media pumps above. Pumps for machine tool cuttings: No rubber parts can be used due to oil. Use vertical pumps of the VS range or horizontal pumps of the M range.

G.) Industrial Segment: Chemical

- a. Pumps for acid slurries: First recommendation is horizontal pumps with rubber or stainless parts. For extremely abrasive slurries use horizontal pumps of the HR range.
- b. Pumps for brines: Very corrosive applications. Can also be abrasive (crystals). Polyurethane can be used to avoid crystallization on pump parts.
- c. Pumps for caustics: Easy application. Both rubber and metal pumps can be used.

H.) Industrial Segments: Mining

- a. Pumps for hydraulic tailings back filling (with or without cement): Watch out for deslimed tailings since that increases the de-rating factor!
- b. Pumps for mine water (with solids): Normal recommendation is horizontal pumps of the HM range (in series if required). Watch out for corrosion with low pH mine drainage water

1.5 MOTIVATION OF THE PRESENT WORK

The objective of the work is to model and numerically investigate the flow field inside the centrifugal slurry pump using commercial CFD code CFX. This is because great improvement on centrifugal pump design must be achieved by CFD analysis of inner flow inside a centrifugal pump and following application of its results in pump design processes. By using simulation result to calculate or predict the performance of a centrifugal pump to replace or reduce the experiments in the process of pump design, a great deal of labour and facility will be saved, as well as its shortening design cycle.

The basic design of centrifugal slurry pumps are meant for handling solid liquid mixture differ in many aspects as compared to that of conventional pumps used for clear liquids. The performance of the centrifugal slurry pumps plays a key role in the successful operation of the hydraulic transportation system. For any application, pump is selected mostly on the basis of its performance characteristics at range of operating parameters. There are various methods available for prediction of performance such as loss analysis approach, artificial neural network approach and various numerical methods. This chapter deals with the literature review of experimental and numerical evaluation of performance characteristics of centrifugal slurry pump handling different types of slurry

Roco et al. (1986) have experimentally evaluated the head-capacity characteristics of few centrifugal slurry pumps of different geometrical configurations handling Silica Sand slurries. They reported that different head losses in the pump vary differently with the solid properties and flow velocities, and hence the effect of suspended solids on individual losses has to be evaluated separately to obtain the performance of the pump in solid-liquid mixture flows.

Roco et al. (1989) have used a two colour Laser Doppler Velocimeter (LDV) to measure the velocity distribution of liquid and solid particles leaving the impeller for very dilute suspension ($C_v=0.002$) of 0.08 mm glass beads. From the velocity distribution at the impeller exit and in the casing, they found that the solids were generally leading the liquid in radial direction and lagging in circumferential direction. They have calculated the particle slip velocity at the impeller-volute interface and at different cross sections of the casing to evaluate the fall in the pump efficiency due to presence of solid particles. They have further shown that the flow periodicity in the pump can be used to evaluate the energy dissipation and erosion wear while handling slurries. They proposed a correlation between velocity distribution and the overall pump performance.

Goto (1992) presented a comparison between the measured and computed exit-flow field of a mixed flow impeller with various tip clearances, for both the shrouded and unshrouded impeller. They confirmed the applicability of the incompressible version of the three-dimensional Navier-Stokes equation code developed by for a mixed-flow centrifugal pump. He found good agreement between measured and computed results for impellers selected for this study.

Walker et al. (1993) have investigated the effect of impeller geometry on the performance of centrifugal slurry pump with sand slurries of two sizes. They found that except for the vane shape, other vane parameters do not have any significant effect on either the head or the efficiency.

Kaupert et al. (1996) have studied pump off-design performance using the commercial software CFX-TASCflow. They observed reverse flow in the impeller shroud region at small flow rates.

Miner, S. M2 [1997] has calculated numerically the flow field and pressure field within the rotor of an axial flow pump. Velocity and pressure profiles were developed on both sides of the impeller. It is observed that the value of tangential velocity increases from the centre line to the outer radius. The axial velocity profile shifts towards the outer radius because of the presence of nose on the hub. The use of coarse and fine mesh does not show significant difference in the values, thus even coarser mesh can be used.

Jude and Covshi (1998) have produced a theoretical analysis and a numerical procedure to determine the pressure distributions along the blade. The flow through a two-dimensional centrifugal impeller with blades of arbitrary geometry was investigated using a combination of conformal mapping with a boundary element method.

Das et al. (1998) presented a computer-aided design method for centrifugal pump impellers with blades of single curvature. Besides computing basic parameters like blade inlet angle, impeller eye and outlet diameters etc. the program has been utilized to study the effect of blade shape, blade outlet angle and number of blades on hydraulic loss. The results obtained are in good agreement with the general observations of the pump designers.

Pagalthivarthi et al. (1998) presented a 2-dimensional compute program based on Galerkin finite element method to solve two dimensional turbulent flow in a centrifugal slurry pump. A mixing length model was used for turbulent stresses. Through this program they found 2.5% difference in inflow and outflow and predicted the recirculation in casing when flow rate exceeds the design flow.

Majidi, K. et al [2000] have observed that static pressure was not distributed uniformly at the outlet of the impeller which results in the radial thrust. There was secondary flow in volute and circular casings of centrifugal pumps. The maximum relative velocity occurred at the periphery of the impeller. The analysis shows that the curvature of the casings creates pressure gradients that cause vortices at cross-sectional planes of the casings.

Miner (2000) has used a commercial CFD code FLOTRAN to study flow pattern in the impeller of axial flow pump and a mixed flow pump. The standard k- ϵ model was used. He has taken fine and coarse meshes for his study. He found that the computational results obtained from coarse meshing closely match with the experimental results and proposed that it can be used to predict the performance.

Sun and Kamoto (2001) have studied pump off-design performance using the commercial software FLUENT. They also predicted reverse flow in the impeller shroud region at small flow rates. They validated the predicted results of the head-flow curves, diffuser inlet pressure distribution and impeller radial forces by comparing with the experimental data over the entire flow range. They observed back flow at small flow rates, however no back flow was observed at higher flow rate.

Oh and Kim [2001] developed a design optimization code for mixed flow pump to determine the geometric and fluid dynamic variables under appropriate design constraints. Optimization problem has been formulated with a nonlinear objective function to minimize the fluid dynamics losses.

Tamm et al. (2001) have used a steady CFD-model for a test-pump called “MRF”-model in FLUENT. The main purpose was to investigate the applicability and limits of this model. The results were compared with detailed measurements and a conventional performance prediction method for centrifugal pump. The conventional approach is based on different loss

– models. A software code developed at their institute was used to determine the different losses and efficiencies. The results of simulation did not shown good agreement with the experimental results.

Benra (2001) presented the economic development of a centrifugal pump impeller of medium specific speed on the basis of given performance parameters for volume flow, head and rotational speed. The commercially available flow simulation program CFX-TASCflow was used for solving Navier-Stokes-equation solutions inside the fluid flow space. The impeller geometry data were transferred to the simulation software with the aid of another commercially available blade generation tool CFX-BladeGen. The CFX-TurboGrid tool was used for grid generation. Based on parametric study, they attempted optimize the impeller geometry.

Zhou et al. (2003) have carried out three-dimensional simulation of internal flow in the pump using commercial CFD code-CFX. Three different types of centrifugal pumps have been chosen. One pump has four straight blades and the other two have six twisted blades. Standard $k-\varepsilon$ (two-equation) turbulence model was used. In the calculation, the finite-volume method and an unstructured grid system have been used for the solution procedure of the discretized governing equations for this problem. They found that the predicted head-flow curve over entire flow range for pumps with twisted blade better than pump with straight blade.

Rudramoorthy et al. (2003) presented CFD analysis of high-speed single stage submersible pump impeller by using commercial CFD package CFX-TURBOGRID and TASC_FLOW. The analysis was done for three different speeds, 10800, 9000 and 8500 rpm and the head developed by the impeller was determined. They calculated that pump developed the 80, 60 and 55m head when it is rotate with 10800, 9000, 8500 rpm respectively for a constant discharge rate of 2 liter per second.

J. H. Horlock et al. [2005] after the development of gas turbine, many empirical design were used for example in obtaining the fluid deflection using the deviation from the blade angle in the assumption of zero radial velocity and in expression for clearance loss. The validity of some of this rules and the basic fluid mechanics behind them is examined by use

of modern idea and CFD code and concluded that the interaction of leakage flow, cooling flow and cavity flow with the main gas path flow has usually been neglected in past but its importance is starting to be realize. Such flow can interact with the main stream flow to produce shear layers which contains vorticity and hence generate additional secondary flow and losses.

Robert Rey et al [2005] studied a 3D-CFD simulation of the impeller and volute of a centrifugal pump has been performed using CFX Codes. The pump has a specific speed of 32 (metric units) and an outside impeller diameter of 400 mm. First, a 3D flow simulation for the impeller with a structured grid is presented. A sensitivity analysis regarding quality and turbulence models were also performed. The final impeller model obtained was used for a 3D Quasi-unsteady flow simulation of the impeller volute stage. A procedure for designing the volute, the nonstructural grid generation in the volute, and the interface flow passage between the impeller and volute are discussed. This flow simulation was carried out for several impeller blades and volute tongue relative positions. As a results, Velocity and pressure field were calculated for different flow rates, Akkowing to obtain the radial trust on the pump shaft.

Cheah et al [2007] has performed study on the numerical simulation of centrifugal pump performance by using commercial code CFX. Computation is carried out with a multiple rotating reference frame (MRF model) approach because the impeller flow field is in a rotating frame whereas the intake section and volute casing are in stationary frame. The meshes of three computational domains, the intake section, impeller, and volute casing and intermediate plates are generated separately. Unstructured mesh with tetrahedral element is used in all three computational domains of centrifugal slurry pump.

John S. Anagnostopoulos [2009] A numerical methodology is developed to simulate the turbulent flow in a 2-Dimensional centrifugal pump impeller and to compute the characteristics performance curve of the entire pump. The flow domain is discretized with a polar, Cartesian mesh and the Reynolds-averaged Navier-Stokes (RANS) equations are solved with the control volume approach and the K-E turbulence model. Advanced numerical techniques for adaptive grid refinement and for the treatment of grid cells that do not fir the irregular boundaries are implemented in order to achieve a fully automated grid construction

for an impeller design, as well as to produce results of adequate precision and accuracy. After estimating the additional hydraulic losses in the casing and the inlet and outlet sections of the pump, the performance of the pump can be predicted using numerical results from impeller section only. The graduations of various energy losses coefficients involved in the model is carried out for a commercial pump, for which there are available measurements. The predicted overall efficiency curve of the pump was found to agree very well with the corresponding experimental data. Finally, a numerical optimization algorithm based on the unconstrained gradient approach is developed and combined with the evolution software in order to find the impeller geometry that maximizes the pump efficiency, using as free design variables the blade angles at the leading and the trailing edge. The result verified that the optimization process can converge very fast and to reasonable optimal values.

Behrouz et al [2010] have carried out study on three-dimensional simulation of turbulent fluid flow to predict velocity and pressure fields for a centrifugal pump. A commercial CFD code was used to solve the governing equations of the flow field. They applied three known turbulence models standard $k - \epsilon$, RNG and RSM, and found that standard $k - \epsilon$ gives better results. The effect of numbers of blades on the efficiency of pump was carried out. The number of blades was changed from 5 to 7 and they observed the impeller with 7 blades has the highest head coefficient.

Lamloumi Hedi et al. (2010) In this study, the viscous Navier-Stokes equations are used to simulate the flow inside the vaneless impeller and volute. The detailed flow structures for different volute tongue geometries are studied in detail. The numerical calculations are compared with experimental data and good agreement is found. The results also suggest direction for further investigations in volute design A solution method is developed to obtain three-dimensional velocity and pressure distribution within a centrifugal pump. The method is based on solving fully elliptic partial differential equations for the conservation of mass and momentum

Pankaj K. Gupta et al (2011) have evaluated the performance of Dense slurry flow through centrifugal pump casing modeled using the Eulerian- Eulerian approach with Eulerian multiphase model in FLUENT 6.1®. First order upwinding is considered for the discretization of momentum, k and ϵ terms. SIMPLE algorithm has been applied for dealing

with pressure- velocity coupling. A mixture property based k- ϵ turbulence model has been used for modeling turbulence. Results are validated first against mesh independence and experiments for a particular set of operational and geometric conditions. Parametric analysis is then performed to determine the effect on important physical quantities viz. solid velocities, solid concentration and solid stresses near the wall with various operational geometric conditions of the pump.

EXPERIMENTAL INVESTIGATION OF CENTRIFUGAL SLURRY PUMP

3.1 EXPERIMENTAL SET-UP

The schematic diagram of experimental set-up for performance evaluation of centrifugal slurry pump is shown in Fig.3.1 and its photographic view is shown in Fig. 3.2. The slurry for the tests were prepared in the hopper shaped mixing tank having a suitable stirring arrangement for keeping the slurry well mixed during experimentation. The height of the tank is 1.55 m and has square cross shape at top (1m x1m).

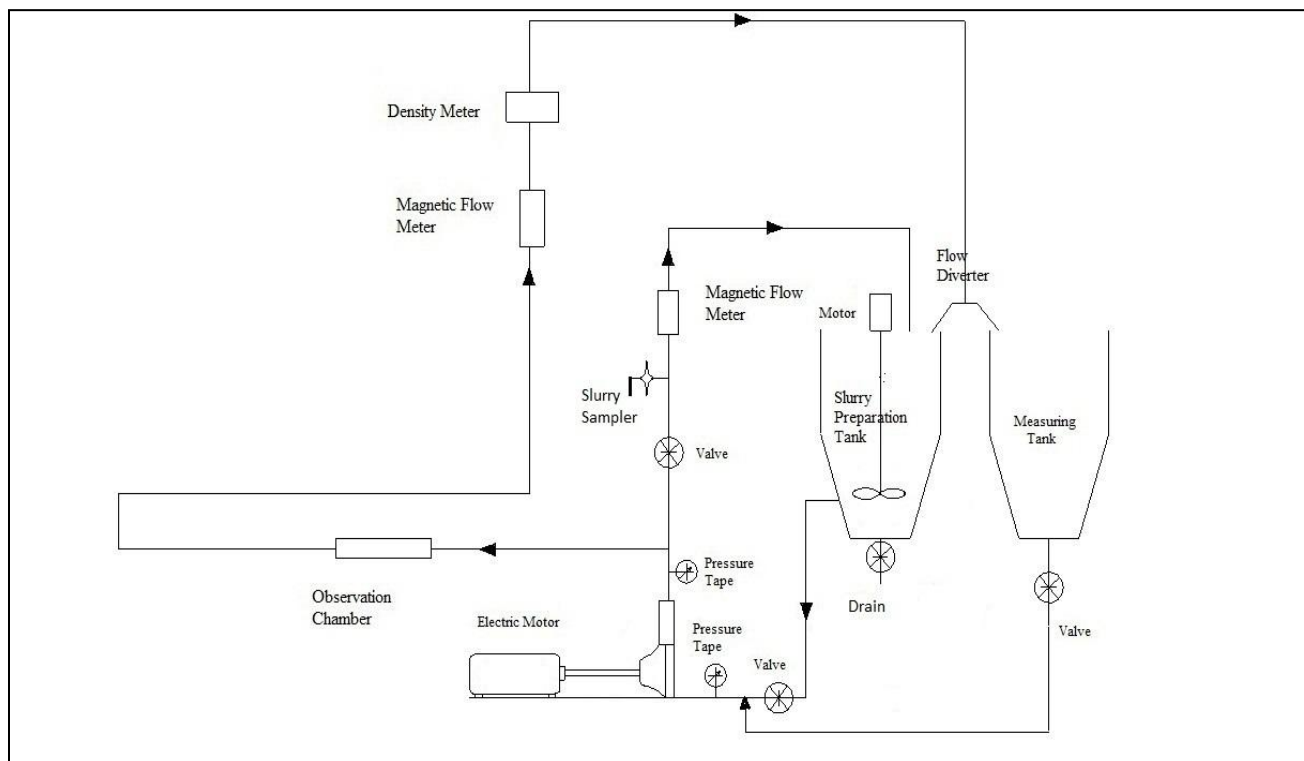


Figure 3.1 Schematic diagram of pilot plant test loop

3.1.1 Mixing tank: It is provided near the measuring tank for mixing the slurry. The return line of pump delivery is connected back into mixing tank for recirculation of water. Both tanks have provided drain plug at bottom side for tank for cleaning purpose. For continuous monitoring of flow rate, pre-calibrated electromagnetic flow meter (type: Magmaster, Make: ABB Limited, Faridabad, India) is installed in vertical pipe section of test loop as shown in Fig.3.1.



Figure 3.2 Photo graphic view of Centrifugal slurry pump set-up

In test loop, density meter is provided for slurry density measurement but in case of clear water it wasn't used. Pressure taps are provided at above separator and separators are installed in suction and delivery side of centrifugal slurry pump. Purpose of separator is avoided to pass sand in pressure transmitter. The suction and delivery pressure is measured by SMAR pressure transmitter (Suction side: LD 290 model and delivery side LD301 model).

Pump is driven by 7.5 kW, 373-456 V, 14.5 A variable speed induction motor (Type: 3 phase square cage, Make: M/s Bharat Bijlee Ltd, Thane, India). For the variation of motor speed, frequency modulator is used. The pump speeds were measured using digital electronic tachometer (Type: Digital photo type tachometer, Make: Systems limited). The manufacturer's specification for centrifugal slurry pump (WILFLEY model, Make: Hindustan Dorr-Oliver Ltd. Bombay) used in present investigation are given in Table 3.1

Table 3.1 manufacturer's data for the pumps used in the investigation

Sr.No.	Specification	50M WILFLY pump
Impeller Details		
1.	Type	Closed
2.	Material	Ni-hard
3.	No. of vanes	5
Casing details		
1.	Type	Volute
2.	Material	Ni-hard
Suction flange size (mm)		100
Delivery flange size (mm)		50
Non-dimensional Specific speed		0.06957
Rated speed (rpm)		1450
Shut- off head (mwc)		20.3
Best Efficiency Point		
1.	Total head (m)	15.1
2.	Discharge rate (lps)	58.1
3.	Pump efficiency	46.0
4.	Pump input power (kW)	4.86

3.1.2 Measuring tank: The slurry is drawn from the mixing tank by the pump and returned after circulation through the test loop to mixing tank either directly or through the measuring tank. The rate of flow is ascertained by measuring the rise in the level of the slurry in measuring tank collected over a known interval of time. The capacity of measuring tank has been chosen so that maximum flow rate can be measured with reasonable time duration. Let the maximum flow rate is 10 lps and time for collection is 20 seconds, then the capacity of measuring tank would be 200L

The detailed of measuring tank are as follows:

- Capacity : 0.242 m³
- Total height of tank : 1045m

- Trapezoidal section : 0.85m
- Square section : 0.6m
- Upper surface area : 0.85 m x 0.85m
- Lower surface area : 0.23 m x 0.23m
- Material Thickness : SS 202 sheet 3mm

3.13 Water Flush by Pump:

Clear water from flush tank is pumped when the slurry pipe line gets blocked with settled slurries. For this purpose one 2 HP centrifugal water pump has been selected and procured.

3.1.4 Centrifugal Slurry Pump with VFD:

The parameters that should be considered while selecting pump for any slurry application are; the head and discharge requirements, type of slurry to be handled, maximum solid concentrations and initial and operating costs.

The settling velocity for abrasive slurry having solid concentration up to 60 % in nun NB pipe is around 1.5 m/s. So the average velocity of flow for pilot plant test p has been chosen as 4 m/s well above the settling velocity.

3.1.5 Variable Frequency Drive (VFD):

Operating speed plays predominant role in the performance of slurry pumps. When duty point changes or the original specification is found to be inappropriate then one has to change the operating speed of the slurry pump. There are alternatives by which we can change the speed of slurry pump to meet variable duty points.

One is by trimming the impeller, but unlike water and chemical pumps, it is difficult to trim the hard metal slurry pump impeller. Other option is, belt drives where speed adjustments are done by altering the pulley sizes. However it often more economical then VFD but here still accurate control lacks.

The introduction of variable speed drives, on slurry pump duties has offset many of the problems of incorrect selection, duty change or variable head. Particularly when pumping slurries, a small change in slurry concentration or flow rate can lead to a large change in head which presents no problems to a pump with a variable speed drive.

Pump wear is also compensated with accurate speed control, which is a further advantage of VFD. Hence with VFD, process performance is maintained and pump life can be extended with overall process cost savings. A motor of 10 HP is chosen to direct coupled to pump.

3.1.6 Electro-Magnetic Flow Meter: For continuous monitoring of flow rate pre-calibrated electro-magnetic flow meter is installed as shown in the figure as letter F.

The specifications of electro-magnetic flow meter are as follows:

- | | | |
|-----------------------------------|---|----------------------------------------------------------------------|
| a. Pipe diameter | : | 50 mm NB |
| b. Range of discharge rate | : | minimum 0-15 lps |
| c. Accuracy | : | $\pm 0.5\%$ |
| d. Application | : | abrasive slurry flow measurement |
| e. Maximum operating pressure | : | 5 bar |
| f. Maximum liquid temperature | : | 60 \cdot C |
| g. Max. Specific gravity of fluid | : | 2.2 |
| h. Display | : | 3 1/2 digit digital display for discharge and velocity |
| i. Repeatability | : | Teflon or Ni- Chrome coated to work in abrasive slurry environments. |

3.1.7 Density Monitoring Instrument: The concentration of the slurry' flowing through pipe loop is monitored by using a Nucleonic density meter installed in the vertical section of the pipe loop near the discharge end as shown in the figure. Alternatively, a density sampler is also provided to collect the slurry samples. In this method the specific gravity of the sample is determined and then average efflux concentration is evaluated g the standard correlation between the slurry specific gravity and solid concentration.

The specification details for nucleonic density meter are as follows;

- | | | |
|--------------------------|---|----------------------------------------|
| a. Detector | : | Scintillation counter, stainless steel |
| b. Nominal pipe diameter | : | 50 mm |
| c. Power supply | : | 230/115 V AC
18 to 32 V DC |
| d. Operation temperature | : | 0 to + 50 \cdot C |
| e. Display | : | LCD-display |

3.1.8 Pressure Transducers: The separators for each pressure taps is provided in the loop to allow only water in the pressure measuring devices to measure the variation across the loop, for this, three sets of pressure transducers are provided. One pressure gauge transmitter at pump outlet, one vacuum gauge transmitter at pump inlet and one pressure differential in the pipe loop. The specifications of gauges are as follows;

1. Type: Gauge
 - Calibrated range: 0-250 kPa
 - Type: Differential
 - Calibrated range: +20 to -20 kPa
2. Type: Vacuum
 - Calibrated range: 0-100 kPa

3.1.9 TECHNICAL SPECIFICATIONS:

- 1. Basic Accuracy : +/- 0.075%
- 2. Process temp. limit : -10 to 70°C
- 3. Output signal : 4-20 mA with digital display
- 4. Power supply : 12V/24V
- 5. Integral digital display : LCD
- 6. Humidity : 95% condensing
- 7. Sensor Housing : Stainless steel
- 8. Process Connection : 12.5 mm NPT/stainless steel

These pressure transmitters are based on a capacitive principle that provides reliable operation and high performance. At suction side, absolute pressure transmitter is used, similarly at delivery side, gauge pressure transmitter is used.

Both pressure transmitters are calibrated by using fluke 718 100G pressure calibrator. In calibrator, pressure is set zero according to pressure transmitter lowest reading value. Pressure in calibrator is increased by pumping simultaneously and readings of pressure transmitter are noted down. Actual pressure for calculation is considered according to calibration curve of both pressure curve of both pressure transmitters.

3.2 PERFORMANCE CHARACTERISTICS OF CENTRIFUGAL SLURRY PUMP

Centrifugal slurry pump are being used extensively for the transportation of solid through pipeline over short to medium distance depending upon the requirement of head discharge ,these pump have essentially two main components namely ,the rotating element comprising of the shaft and impeller and the stationary element consisting of the volute casing enclosing the impeller .the efficiency of a well design conventional centrifugal pump can be as high 92%.the conventional centrifugal pump design has to suitably modify to handling slurry for ensuring proper flow of solid liquid mixture within minimum erosion .the modification incorporated in the pump including enlargement of flow passages to accommodate figure solid particles robust impeller with smaller number of vanes, special seals and proper material of construction to insure longer life. A centrifugal slurry pump in general is single stage and suction type with modified volute type collector.

The performance characteristics of these pumps are much lower as compared to the conventional pumps. The deterioration in the performance could be attributed to the modifications incorporated and the addition of solids particles in the carrier liquid. It is difficult and time consuming to evaluate the performance characteristics of these pumps with all types if slurry, to overcome these problems, the performance of these pumps is evaluated first with clear water and then performance with slurries is estimated by applying suitable correction factors depending on the slurry properties.

1. Calculation of Head (H_{Total}):

$$H = \left(\frac{P_d}{\rho g} + Z_d \right) - \left(-\frac{P_s}{\rho g} - Z_s \right) + \frac{V_d^2}{2g} - \frac{V_s^2}{2g}$$

Where,

V_d = velocity of fluid in delivery pipe (m/s)

V_s = velocity of fluid in suction pipe (m/s)

Z_d = delivery head (meters)

Z_s = Suction lift (meters)

= Delivery pressure gauge reading delivery side (m)

$$\left(\frac{P_s}{\rho g}\right) = \text{Vacuum gauge reading at suction side (m)}$$

2. Correction of head ($H_{1450rpm}$) and discharge ($Q_{1450rpm}$) at rated speed (1450 rpm):

Affinity laws are based on a set of three non-dimensional parameters which enable to evaluation of performance of the pump at any speed from the results obtained during a test for rated speed of 1450 rpm, the head, discharge and power is to be determined as below:

$$H_{1450rpm} = H_n \times \frac{1450}{N^2}$$

$$Q_{1450rpm} = Q_n \times \frac{1450}{N}$$

$$P_{1450rpm} = P_n \times \frac{1450^3}{N^3}$$

Where, N- speed observed during measurement (rpm)

3. Measurement of slurry concentration (C_w):

$$C_w = \frac{\rho_s(\rho_m - \rho_l)}{\rho_m(\rho_s - \rho_l)}$$

Where

C_w =solid concentration, % by weight

ρ_s = mass density of solid material, kg/m³

ρ_l = mass density of liquid, kg/m³

ρ_m = mass density of solid- liquid mixture , kg/m³

4. Power output:

$$\rho_{out} = \rho gQH$$

Where

H =Total head (m)

ρ = mass density of water, kg/m³

g = gravitational acceleration m/sec²

Q = flow rate, m³/s

P_{out} = Out put power, kW

5. Efficiency of pump:

$$\eta_{pump} = \frac{P_{out}}{P_{in}} \times 100$$

Where

P_{in} = Input power of pump

P_{in} = $T \times \omega$

T = Torque of pump shaft, N-m

ω = Rotational speed of the shaft, rad/s

The torque and rotational speed of the pump shaft will be measured by installing a torque sensor at the coupling of pump and motor shaft.

6. Head ratio (HR):

It is the ratio of the head developed with slurry in meters of slurry column to that water in meters of water column (mwc) with other parameters being same.

Alternatively, it is also expressed in terms of head reduction factor as,

$$\text{Head reduction factor} = 1 - \text{HR}$$

7 Efficiency ratio (ER):

It is the ratio of the pump efficiency with slurry to that with water at the same discharge and speed.

Alternatively, it is also expressed in terms of efficiency reduction factor as,

$$\text{Efficiency reduction factor} = 1 - \text{ER}$$

8 Power ratio (PR):

It is defined as the ratio of the power consumed by the pump for slurry to that with water at the same discharge and speed.

Centrifugal slurry pumps are best suited for short and in-plant slurry transportation systems. Over 97% of short distance pipelines use these pumps. These are invariably known as “work-

.horse” of in plant pumping system, for long distance pumping, additional pump station at appropriate distance along the pipe route need to be provided of the same could be achieved by series pumping, these are also widely used for transporting homogeneous slurry in mineral processing plant. These pumps have also been reported to be most efficient and economical for handling coarse and abrasive slurries.

3.3 PERFORMANCE CURVES OF WATER AT VARIOUS SPEEDS:

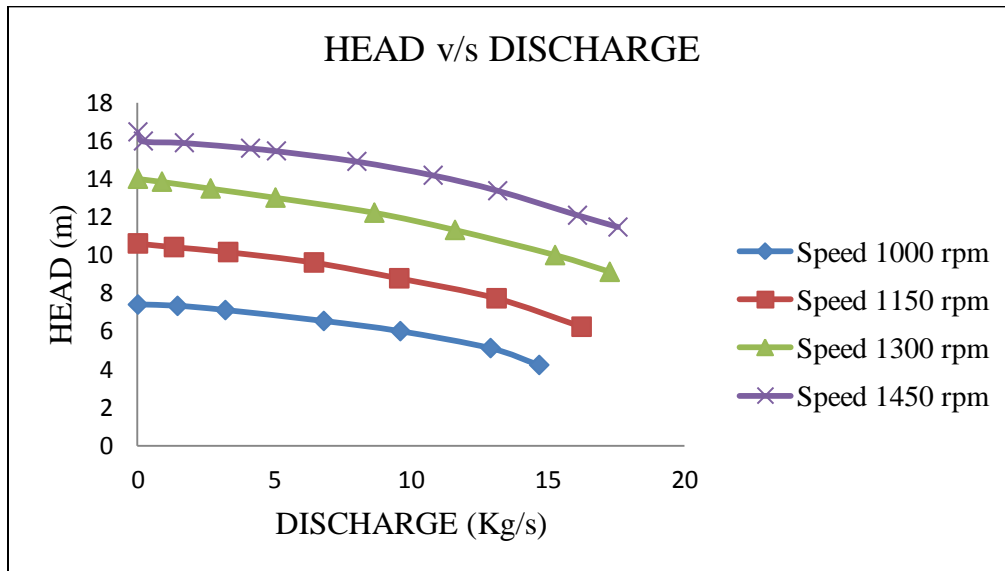


Figure 3.3 Performance curve between Head v/s Discharge of water

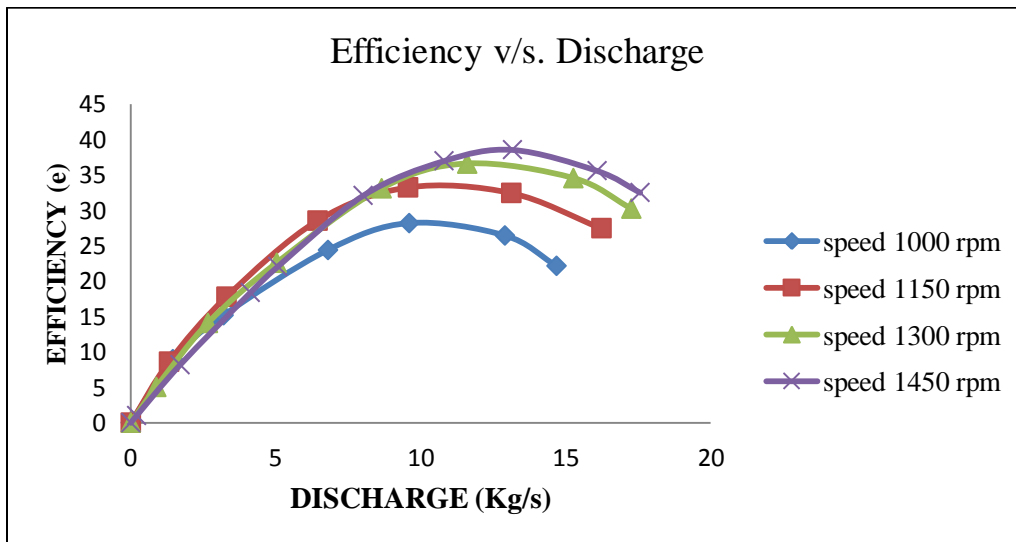


Figure 3.4 Performance curve between Efficiency v/s Discharge of water

INTRODUCTION TO COMPUTATIONAL FLUID DYNAMICS (CFD)

4.1 INTRODUCTION TO CFD

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Fluid (gas or liquid) flows are governed by partial differential equations which represent conservation laws for mass, momentum & energy. Computation fluid dynamics is the art of replacing such PDE systems by a set of algebraic equations which can be solved using digital computers. CFD has become an indispensable tool in the design, development, evaluation and refinement of new industrial equipment and processes. The use of CFD reduces the development cost of new products and cuts the time to market of these products. CFD has applications in

1. Aerospace,
2. Automotive Design,
3. Electronics,
4. Food, Chemical, and Material Processing,
5. Power Generation

Computational Fluid Dynamics (CFD) has become an integral part of the engineering design cycle. Whether it is used to design industrial mixing tanks, aerospace components, or used to simulate transient flow phenomenon in pumps and turbines. CFD analyses reduce development time and increase the reliability of prototype designs. CFD simulations can be used to model fluid flows over a wide range of physical scales. The fundamental basis of any CFD problem is the Navier-Stokes equations, which define any single-phase fluid flow. It works by solving the equations of fluid flow (in a special form) over a region of interest, with specified (known) conditions on the boundary of that region.

4.2 HISTORY OF CFD

Computers have been used to solve fluid flow problems for many years. Numerous programs have been written to solve either specific problems, or specific classes of problems. From the

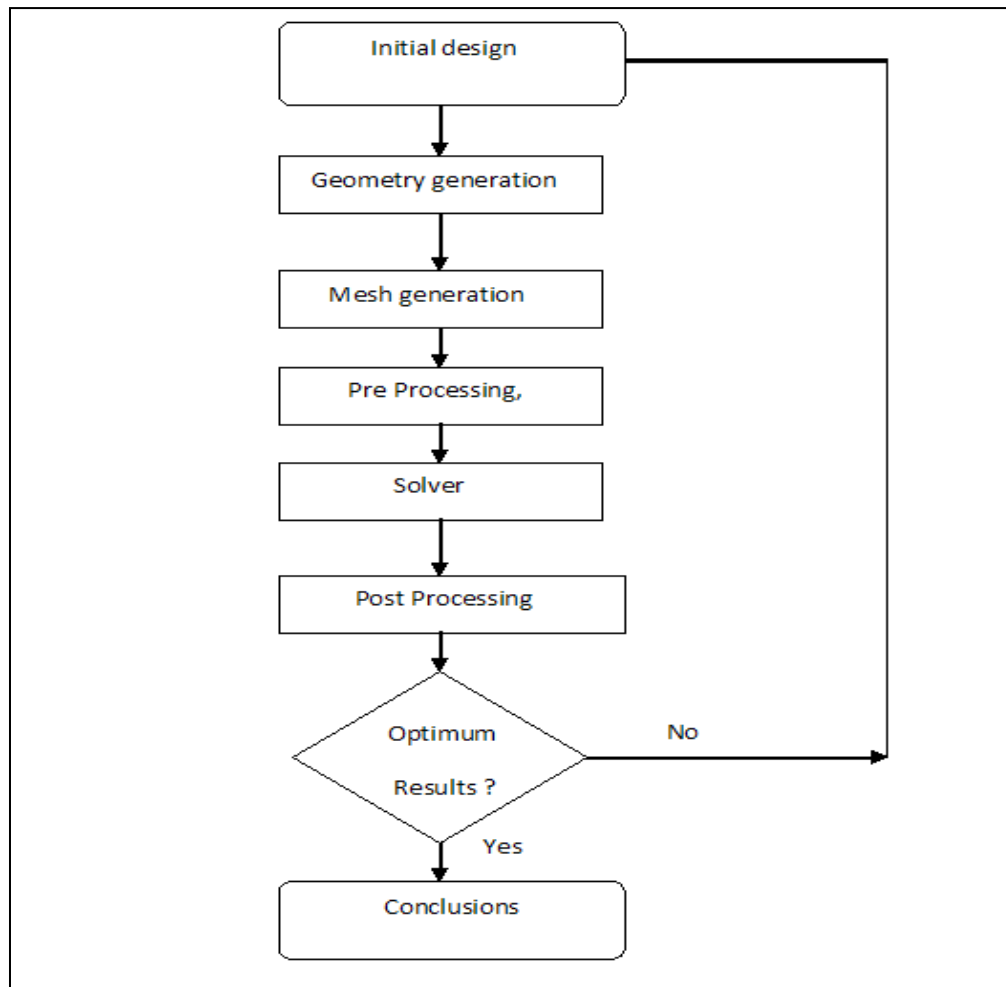
mid-1970's, the complex mathematics required to generalize the algorithms began to be understood, and general purpose CFD solvers were developed. These began to appear in the early 1980's and required what were then very powerful computers, as well as an in-depth knowledge of fluid dynamics, and large amounts of time to set up simulations. Consequently, CFD is a tool used almost exclusively in research. Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analyzing results much less labour intensive, reducing time and, hence, cost. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time. As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world. CFD provides a cost-effective and accurate alternative to scale model testing with variations on the simulation being performed quickly offering obvious advantages.

4.3 EXPERIMENTS VS. SIMULATIONS

CFD gives an insight into flow patterns that are difficult, expensive or impossible to study using traditional (experimental) techniques

Experiments	Simulations
<p>Quantitative description of flow phenomena using measurements</p> <ul style="list-style-type: none"> • For one quantity at a time • At a limited number of point and time instants • For a laboratory-scale method • For a limited number of problems and operating conditions <p>Error sources: measurement errors, flow disturbance by the probes</p>	<p>Quantitative prediction of flow phenomena using CFD software</p> <ul style="list-style-type: none"> • For all desired quantities • With high resolution in space and time • For all actual domain • For virtually any problem and realistic operating condition <p>Error sources: modelling, discretization, iteration, implementation</p>

4.4 CFD METHODOLOGY



4.4.1 Initial design

In order to obtain better design in CFD, following procedure is applied so that fluid flow can easily be modelled. Initial design of the model is a planning decision and the geometry is generated depending on these initial design considerations, using either CFD modelling tools or other Design tools.

4.4.2 Geometry generation

The first task to accomplish in a numerical flow simulation is the definition of the geometry, followed by the grid generation. This step is the most important step for the study of an isolated impeller assuming an axis symmetric flow simplifies the domain to a single blade passage.

The geometry of design needs to be created from the initial design. Any modelling software can be used for modelling and then shifted to other simulation software for analysis purposes.

4.4.3 Mesh generation

Mesh generation (Gridding) is the process of subdividing a region to be modelled into a set of small control volumes. Associated with each control volume there will be one or more values of the dependent flow variables (e.g., velocity, pressure, temperature, etc.) Usually these represent some type of locally averaged values. Numerical algorithms representing approximations to the conservation laws of mass, momentum and energy are then used to compute these variables in each control volume.

Meshing is a method to define and break up the model into small elements. In general, a finite element model is defined by a mesh network, which is made up of the geometric arrangement of elements and nodes. Nodes represent points at which features such as displacements are calculated. Elements are bounded by sets of nodes, and define localized mass and stiffness properties of the model. Elements are also defined by mesh numbers, which allow references to be made to corresponding deflections, stresses, pressures, temperatures at specific model locations. The traditional method of mesh generation is block-structure (multi-block) mesh generation. The block-structure approach is simple and efficient technique of mesh generation.

4.4.4 Pre-processing

4.4.4.1 Boundary condition

The first step in Pre-processing is setting up the boundary conditions. Boundary condition will be different for each type of problem. In Cartesian and cylindrical-polar coordinates, the location of boundary features (inlets, outlets, blockages, etc) can be linked to named 'objects' defined during the grid-generation procedure. This obviates the need to enter the coordinates twice: once when defining the grid, and again when specifying boundary conditions. If an 'object' is subsequently repositioned or re-sized, then the boundary condition is also changed automatically. If an object is deleted, any associated boundary conditions will also be deleted without further instructions from the user. If a new 'object' is created by copying an existing one, the boundary conditions are not automatically copied, but a new boundary condition may be linked to the new object.

In low speed and incompressible flows, disturbances introduced at an out flow boundary can have an effect on the entire computational region. As a general rule, a physically meaningful boundary condition, such as a specified pressure condition, should be used at out flow boundaries whenever possible. Generally, a pressure condition cannot be used at a boundary where velocities are also specified, because velocities are influenced by pressure gradients. The only exception is when pressures are necessary to specify the fluid properties, e.g., density crossing a boundary through an equation of state.

The inlet condition for velocity and temperature can be specified using profile of grid. The turbulent kinetic energy (k) and its dissipation rate can be calculated from the value of turbulence intensity specified in the inlet.

4.4.4.2 Turbulence models

Turbulence consists of fluctuations in the flow field in time and space. It is a complex process, mainly because it is three dimensional, unsteady and consists of many scales. It can have a significant effect on the characteristics of the flow. Turbulence occurs when the inertia forces in the fluid become significant compared to viscous forces, and is characterized by a high Reynolds Number. In general, the Navier-Stokes equations describe both laminar and turbulent flows without the need for additional information. However, turbulent flows at realistic Reynolds numbers span a large range of turbulent length and time scales, and would generally involve length scales much smaller than the smallest finite volume mesh, which can be practically used in a numerical analysis. The Direct Numerical Simulation (DNS) of these flows would require computing power which is many orders of magnitude higher than available in the foreseeable future. Turbulence models are used to predict the effects of turbulence in fluid flow without resolving all scales of the smallest turbulent fluctuations. A number of models have been developed that can be used to approximate turbulence based on the Reynolds Averaged Navier-Stokes (RANS) equations. Some have very specific applications, while others can be applied to a wider class of flows with a reasonable degree of confidence. One of the main problems in turbulence modelling is the accurate prediction of flow separation from a smooth surface. Standard two-equation turbulence models often fail to predict the onset and the amount of flow separation under adverse pressure gradient conditions. This is an important phenomenon in many technical applications, particularly for airplane aerodynamics since the characteristics of a plane

are controlled by the flow separation from the wing. In general, turbulence models based on the equation predict the onset of separation too late and under-predict the amount of separation later on. This is problematic; as this behaviour gives an overly optimistic performance characteristic for an airfoil. The prediction is therefore not on the conservative side from an engineering standpoint. The models developed to solve this problem have shown a significantly more accurate prediction of separation in a number of test cases and in industrial applications. Separation prediction is important in many technical applications both for internal and external flows.

a) *k-ε* turbulence model

One of the most prominent turbulence models, the *k-ε* (k-epsilon) model, has been implemented in most general purpose CFD codes and is considered the industry standard model. It has proven to be stable and numerically robust and has a well established regime of predictive capability. For general purpose simulations, the model offers a good compromise in terms of accuracy and robustness. Within ANSYS CFX, the *k-ε* turbulence model uses the scalable wall-function approach to improve robustness and accuracy when the near-wall mesh is very fine. The scalable wall functions allow solution on arbitrarily fine near wall grids, which is a significant improvement over standard wall functions. While standard two-equation models, provide good predictions for many flows of engineering interest *k* is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. *ε* is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate), and has dimensions of per unit time. The *k-ε* model introduces two new variables into the system of equations.

The continuity equation is then:

$$\partial\rho/\partial\tau + \nabla\cdot(\rho \mathbf{v}) = 0$$

And the momentum equation becomes

$$\partial(\rho \mathbf{v})/\partial\tau + \nabla\cdot(\rho \mathbf{v}\mathbf{v}) - \nabla\cdot(\mu_{\epsilon\phi\phi} \nabla\mathbf{v}) = -\nabla p + \nabla\cdot(\mu_{\epsilon\phi\phi} \nabla\mathbf{v})^T + \beta$$

Where

β = Sum of body forces,

$\mu_{\epsilon\phi\phi}$ = Effective viscosity accounting for turbulence,

p = Modified pressure

(b) The *k-ω* model

(c) The Shear Stress Transport (SST) model

4.4.4.3 Solver

The different types of solvers which are used in computational fluid dynamics are

1. Finite Difference Method
2. Finite Volume Method (Control Volume Method)

Finite Difference Method

A finite difference method (FDM) discretization is based upon the differential form of the PDE to be solved. Each derivative is replaced with an approximate difference formula (that can generally be derived from a Taylor series expansion). The computational domain is usually divided into grids, and the solution will be obtained at each nodal point. The FDM is easiest to understand when the physical grid is Cartesian, but through the use of curvilinear transforms the method can be extended to domains that are not easily represented by brick-shaped elements. The discretization results in a system of equation of the variable at nodal points, and once a solution is found, then we have a discrete representation of the solution.

4.4.4.4 Flow equations

The steady, incompressible, Reynolds Averaged Navier-Stokes (RANS) equations are employed for the flow calculations in polar coordinates and a rotating with the impeller system of reference. These equations establish that changes in momentum in infinitesimal volumes of fluid are simply the sum of dissipative viscous forces (similar to friction), changes in pressure, gravity, and other forces acting inside the fluid: an application of Newton's second law.

The Navier-Stokes equations are differential equations which, unlike algebraic equations, do not explicitly establish a relation among the variables of interest (e.g. velocity and pressure). Rather, they establish relations among the rates of change. For example, the Navier-Stokes equations for simple case of an ideal fluid (inviscid) can state that acceleration (the rate of change of velocity) is proportional to the gradient (a type of multivariate derivative) of pressure.

Contrary to what is normally seen in solid mechanics, the Navier-Stokes equations dictate not position but rather velocity. A solution of the Navier-Stokes equations is called a velocity field or flow field, which is a description of the velocity of the fluid at a given point in space and time. Once the velocity field is solved for, other quantities of interest (such as flow rate, drag force, or the path a "particle" of fluid will take) may be found.

A two dimensional, laminar incompressible flow with constant viscosity is described by:

$$\text{Momentum equation: } \rho \frac{\partial u}{\partial \tau} + \rho u \left(\frac{\partial u}{\partial x} \right) + \rho v \left(\frac{\partial u}{\partial y} \right) = \mu \left[\left(\frac{\partial^2 u}{\partial x^2} \right) + \left(\frac{\partial^2 u}{\partial y^2} \right) \right]$$

$$\text{And continuity equation: } \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

There are several methods of discretising a given differential equation, but finite volume method is used in ANSYS-CFX.

4.5 MULTIPHASE FLOW

Multiphase flow is a flow in which more than one fluid is present. In general, the fluids consist of different chemical species, such as air-water. In some applications, they may represent different thermodynamic phases of the same species, such as steam-water.

It is important to distinguish between multicomponent and multiphase flow. A multicomponent fluid is assumed to consist of a mixture of chemical species which are mixed at the molecular level. In this case, single mean velocity and temperature fields, etc., are solved for the fluid. Examples are gaseous mixtures, and solutes in liquids.

The fluids in a multiphase flow are assumed to be mixed at macroscopic length scales, much larger than molecular. Examples are gas bubbles in a liquid, liquid droplets in a gas or in another immiscible liquid, etc. In this case, it is necessary to solve for different velocity and temperature fields, etc., for each fluid. These may interact with each other by means of interfacial forces and heat and mass transfer across the phase interfaces.

4.5.1 Eulerian-Eulerian Model

The Eulerian-Eulerian model is one of the two main multiphase models that has been implemented in ANSYS CFX; the other is the Lagrangian Particle Tracking Model. Within the Eulerian-Eulerian model, certain interphase transfer terms used in momentum, heat and other interphase transfer models, can be modeled using either the Particle Mode, the Mixture Mode or the Free Surface Model. In particular, the calculation of the interfacial area density, used for all inhomogeneous transfer models for a given fluid pair, is calculated according to one of these models. The model used for a particular pair of phases is set on the Fluid Pairs tab. The available option depend on the morphology of each phase of the pair (for example, continuous, dispersed, etc.), and on settings found on the Fluid Models tab (homogeneous options, free surface model

option).

4.5.2 Inhomogeneous Multiphase Flow

Inhomogeneous multiphase flow refers to the case where separate velocity fields and other relevant fields exist for each fluid. The pressure field is shared by all fluids. The fluids interact via interphase transfer terms. The particle and Mixture Models are both inhomogeneous multiphase models.

4.5.3 Homogeneous Multiphase Flow

Homogeneous multiphase flow is a limiting case of Eulerian-Eulerian multiphase flow where all fluids share the same velocity fields, as well as other relevant fields such as temperature, turbulence, etc. The pressure field is also shared by all fluids.

4.5.4 Multi-component Multiphase Flow

It is possible to combine the notions of multi-component and multiphase flows. In this case, more than one fluid is present, and each such fluid may be a mixture of chemical species mixed at molecular length scales. An example is air bubbles in water in which ozone gas is dissolved in both the gaseous and liquid phases. In this case, mass transfer of common species may occur by diffusion across the phase interface.

4.5.5 Volume Fraction

Multiphase modeling employs the notion of interpenetrating continua. Although phases are mixed at length scales much larger than molecular, they are also assumed to be mixed at length scales smaller than you scales much larger than molecular, they are also assumed to be mixed at length scales smaller than you Thus, each phase is assumed to be present in principle in each control volume, and assigned a volume fraction equal to the fraction of the control volume occupied by that phase. This variable appears as <fluid>.Volume Fraction in the results file. The variable <fluid>.Conservative Volume Fraction is also available but should not usually be used for post-processing. The conservative volume fraction is used by the CFX-Solver; it may exceed 1 and may not sum to unity over all fluids during the solution. However, these criteria are enforced in a converged solution.

4.5.6 Free Surface Flow

Free Surface flow refers to a multiphase situation where the fluids (commonly water and air) are separated by a distinct resolvable interface.

4.5.7 Surface Tension

Surface tension is a force that exists at a free surface interface which acts to minimize the surface area of the interface. It gives rise to effects such as a pressure discontinuity at the interface and capillary effects at adhesive walls.

4.5.8 Maximum Packing

For dispersed phases, you can specify a maximum packing parameter. This is the volume fraction of the phase at its state of maximum packing. It is most commonly used for compact solid dispersed phases, for example, as you would find in fluidized beds.

The maximum packing parameter is unity by default for a dispersed fluid. For a dispersed solid phase, it may range from 0.5 to 0.74, the latter being the maximum possible packing for solid spheres. For most applications, the default value 0.62 suffices.

The maximum packing parameter is used in correlations for certain drag laws, and models for particle collision forces. Unfortunately, it is not possible to numerically guarantee that volume fractions are bounded above by the maximum packing parameter. Consequently, you may observe volume fractions higher than the maximum packing.

4.6 POST-PROCESSING

Post-processing provides the CFD professional with easy-to-use powerful result visualization features for structured, unstructured, and hybrid grids. It provides an in depth view of data with visualization tools such as cutting planes, contouring, streamlines, line plots, data probes and animation. Animations can be created by saving CFD solutions with or without skipping certain number of time steps and playing the saved frames in a continuous sequence. Animations are important tools to study time dependent developments of vertical /turbulent structures and their interactions.

4.7 MODELING OF THE PUMP COMPONENTS

For the purpose of numerical analysis on the pump, the dimensions of the pump are required to generate a model in the software. So the dimension of all the components of the centrifugal slurry pump was obtained by reverse engineering techniques. The experimentation setup was dismantled and all the valves were closed to prevent any leakage of water or slurry. The pipes connected to the pump were disconnected and the water in the setup was drained. The pump assembly was disassembled and all the parts were separated. The assembly consisted of the casing, impeller and suction discs and follower plate. Since the pump was firmly placed on the foundation the suction passage extending from the flange to the frame was not removed.

The first step after dismantling is:

1. The dimensions that could be taken directly like impeller outer diameter, casing outlet diameter, etc were taken using vernier calipers and steel rule.
2. For dimensions that could not be taken directly with instruments moulds of plaster of paris and clay were made.

The dimensioning of different parts of the centrifugal slurry pump is disused below:

4.7.1 Impeller

The impeller of centrifugal slurry pump is the enclosed type and has 5 vanes. The tail of the impeller extends from the centre of the head through the discs till the suction passage where it rests on a disc connected to the shaft and embedded in the inlet passage. This tail section is circular in shape; this section consists constant diameter over a certain length and increases gradually over the end forming a bell mouth at the end. This bell mouth lies in the suction passage from where the flow enters the pump. The main purpose of the giving bell shape to the tail of the impeller is the smoothening of the flow to ensure uniform feed to the impeller. A fillet is given at the inlet in the impeller head where the flow enters the vanes. Due to this fillet the width of the vane increases at the inlet. Since the impeller is closed type and the vane at inlet is wider than the outlet, it was not possible to measure the dimensions at the impeller inlet. The profile of the vane at inlet was reconstructed by making a mould out of m-seal. The other parts of the impeller were modelled using plaster of paris moulds. The dimensions were then taken from

these moulds by plotting them on the graph. Points were marked on the graph sheet with respect to a reference.

4.7.2 Casing

The casing of centrifugal slurry pump is semi-volute type having 275 mm base circle diameter and 11 degrees tongue angle. The casing was also modeled by using same technique as of impellor i.e. making plaster of paris moulds. The casing was divided into 30 degrees span total making 12 sectors. Once divided moulds of each of these sectors were casted. The moulds were not casted for the entire sector but only for the area beyond the base circle, this was done to optimize the requirement of plaster of paris and also because only the area which is increasing was required. These 12 sectors were then put together to reconstruct the casing and check the conformation with the casing. These sectors were then grinded on the edges to get a clear cross-section on the graph. Each sector was then drawn over a base circle in the software.

4.7.3 Follower plate, Frame and Flange

The dimensions of the follower plate and frame were taken directly as the geometry was simple both to measure and construct in CAD software. The cross-section of the disc was modeled according to the dimensions and then rotated about an axis to model the discs. The flange was made by joining the cross-section at pump inlet and the flange connected to the pipe. The cross-section at pipe end was circular while it had a different cross-section at the inlet. The two sections made and then joined.

4.7.4 Inlet passage

The inlet passage of the centrifugal slurry pump had a complicated structure with several fillets and sections at different angles. For this reason several moulds of different areas were made; from these sections geometry of passage at different locations was obtained. These sections were extended and combined together to model the fluid portion in the inlet passage.

4.8 PREPARATION OF CAD MODEL AND MESH GENEATION

The next step after taking the dimensions are preparing the cad model. The cad model was modeled by using the ICEM module of the ansys 14. For modeling of impellor the points

obtained on graph were then plotted in the software following which a curve with a suitable radius was fitted through the points to get the smooth surface. In case of casing each sector was then drawn over a base circle in the software. The cross-section was drawn by connecting the points with Nerbs. Once the cross-section of each sector was made, the casing was constructed with the bottom up approach by creating faces and volumes. Similarly modeling was done of other parts of the pump and they were later moved to their respective places for assembly purpose. The cad model of casing, impellor and inlet passage is shown on figure 4.1, 4.2 and 4.3:

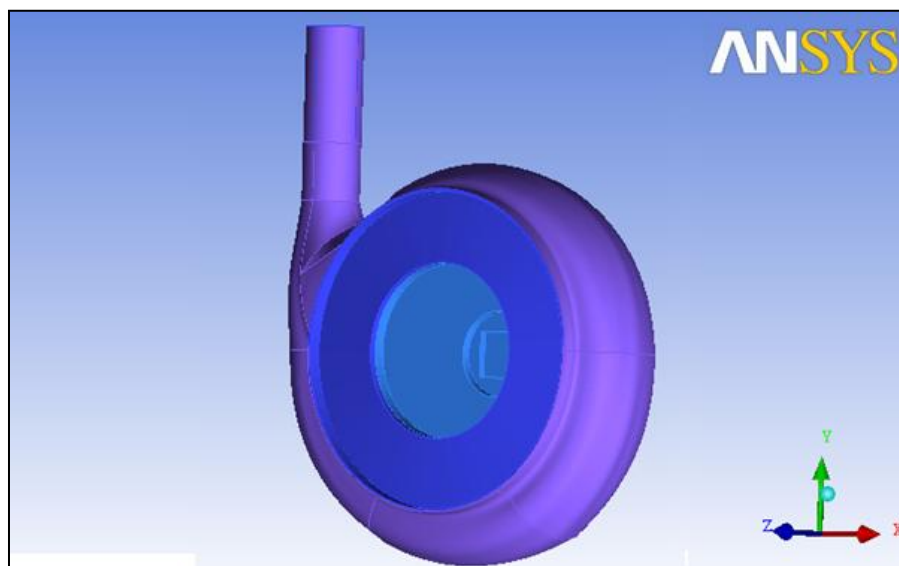


Figure: 4.1 isometric view of geometry of outlet casing

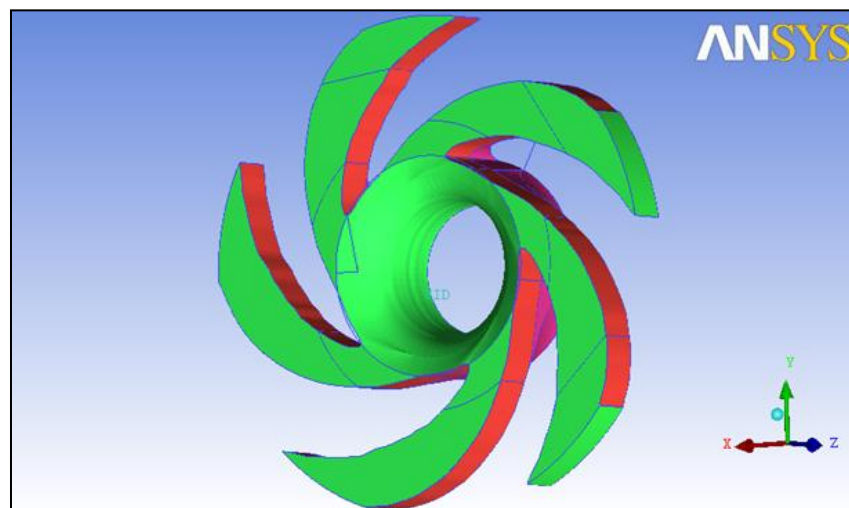


Figure 4.2 isometric view of Geometry of impeller

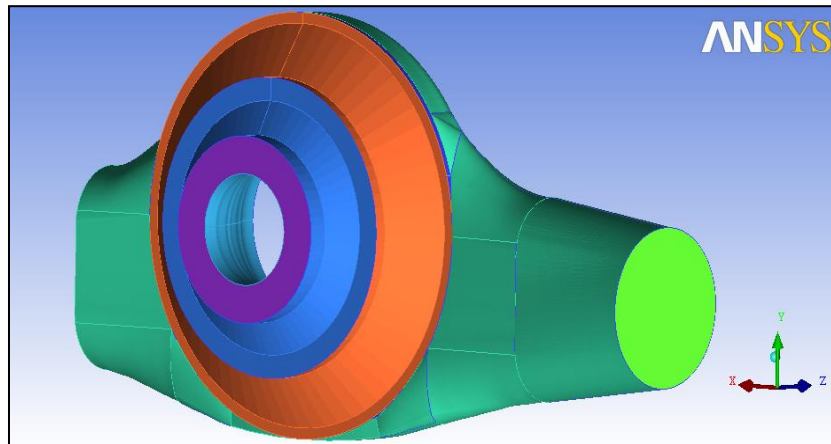


Figure 4.3 CAD model of inlet passage

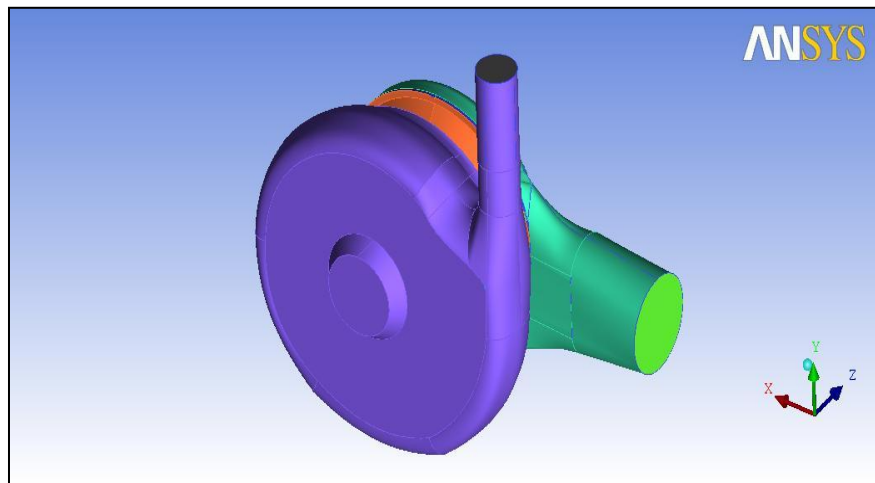


Figure 4.4 Assembled view of centrifugal pump

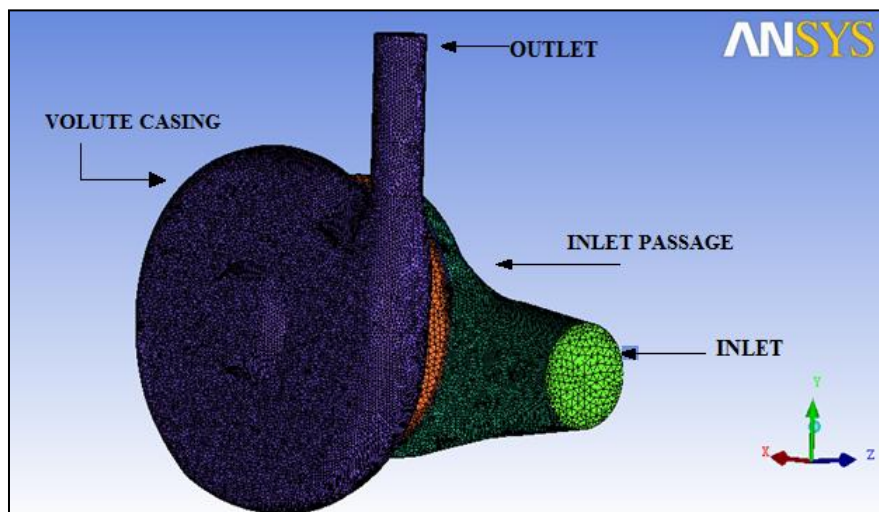


Figure 4.5 Meshed model of centrifugal pump

The first step after modeling the pump components is discretize it into smaller elements. The model is discretized so that the affect of external or internal forces acting on a body can be captured at any point on the body. This process of discretization is called meshing. The larger the number of these elements, finer will be the mesh which will give accurate result. But a large number of elements or a fine mesh needs greater computational capabilities and time. To optimize the meshing, a finer mesh is used at points where the forces have maximum impact and a coarser mesh is used at other locations. For meshing, the model was taken to a commercially available Ansys 14 ICEM was used. It provides a large variety of tools for meshing. The tetrahedral type of elements was used for the discretization of all the three components of the centrifugal pump. The meshed photographic view of pump is shown in figure 4.5.

The information about no. of elements is given in table 4.1

Table 4.1 Number of Elements

Part name	No. of elements
Inlet passage	180988
Casing	185360
Impellor	92538

Table 4.2 Properties of Fly Ash

Concentration	10%	20%	30%	40%	50%
Viscosity	0.966	1.771	2.524	3.246	3.945

Elements	c	Mg	Al	Si	P	K	Ca	Ti	Fe	Cu	Zn	O
Molar Mass	12.01	24.30	26.98	28.08	30.97	39.09	40.07	47.86	55.84	63.54	65.38	15.99

4.9 BOUNDARY CONDITIONS

Boundary conditions are the set of conditions specified for the behaviour of the solution to a set of differential equations at the boundary of its domain. Boundary conditions are important in determining the mathematical solutions to many physical problems. These conditions specify the flow and thermal variables on the boundaries of a physical model. They are, therefore, a critical

component of simulation and it is important that they are specified appropriately. The boundary conditions are defined on cell faces and they do not have a finite thickness and they provide a means of introducing a step change in flow properties.

It is important to check the quality of mesh, because parameter such as skewness affects the accuracy of the CFD simulation. Each element has of value of skewness between 0 and 1. The skewness is classified in two ways, EquiAngle skew and EquiSize skew. The smaller value of equiAngle skew and equisize skew are more acceptable. It is also important to verify that all of the elements in mesh have positive area/volume otherwise the simulation in solver is not possible.

EquiAngle Skew

The EquiAngle Skew (Q_{EAS}) is the measure of skewness that is defined as follows:

$$Q_{EAS} = \max \left\{ \frac{\theta_{\max} - \theta_{eq}}{180 - \theta_{eq}}, \frac{\theta_{eq} - \theta_{\min}}{\theta_{eq}} \right\}$$

Where,

θ_{\max} & θ_{\min} = maximum and minimum angles between the edges of the element, degrees

θ_{eq} = angle corresponding to an equilateral cell of similar form. For triangular and tetrahedral elements, $\theta_{eq} = 60^\circ$ For quadrilateral and $\theta_{eq} = 90^\circ$ for hexahedral elements.

By definition, $0 \leq Q_{EAS} \leq 1$

Where, $Q_{EAS} = 0$ describes an equilateral element, and $Q_{EAS} = 1$ describes a completely degenerate (poorly shaped) element. In general, high-quality meshes contain elements that possess average Q_{EAS} values of 0.1 in two-dimensional case and 0.4 in three-dimensional case.

EquiSize Skew

The EquiSize Skew (Q_{EVS}) is a measure of skewness that is defined as follows:

$$Q_{EVS} = \frac{(S_{eq} - S)}{S_{eq}}$$

Where, S = area in 2-dimensional or volume in 3-dimensional case of the mesh element,

S_{eq} = maximum area in 2-dimensional or volume in 3-dimensional case of an equilateral cell the circumscribing radius of which is identical to that of the mesh element.

$$0 \leq Q_{EVS} \leq 1$$

4.9.1 Types of boundary conditions

The following boundary conditions at the walls are used with the equations of motion

- No slip conditions

At fluid wall interface, there must be no slip\

$$V_{fluid} = V_{wall}$$

The conditions that are applied for numerical simulation of the centrifugal slurry pump simulations are

- Mass flow inlet: It is the inlet face at suction pipe entry.
- Wall: It is applied to Inlet passage faces, rotating faces of impeller and fixed faces of volute casing.
- Pressure outlet: It is applied on outlet face at delivery pipe entry.
- Fluid zone: It is applied suction pipe, impeller passages, volute casing, inlet passage and delivery pipe.

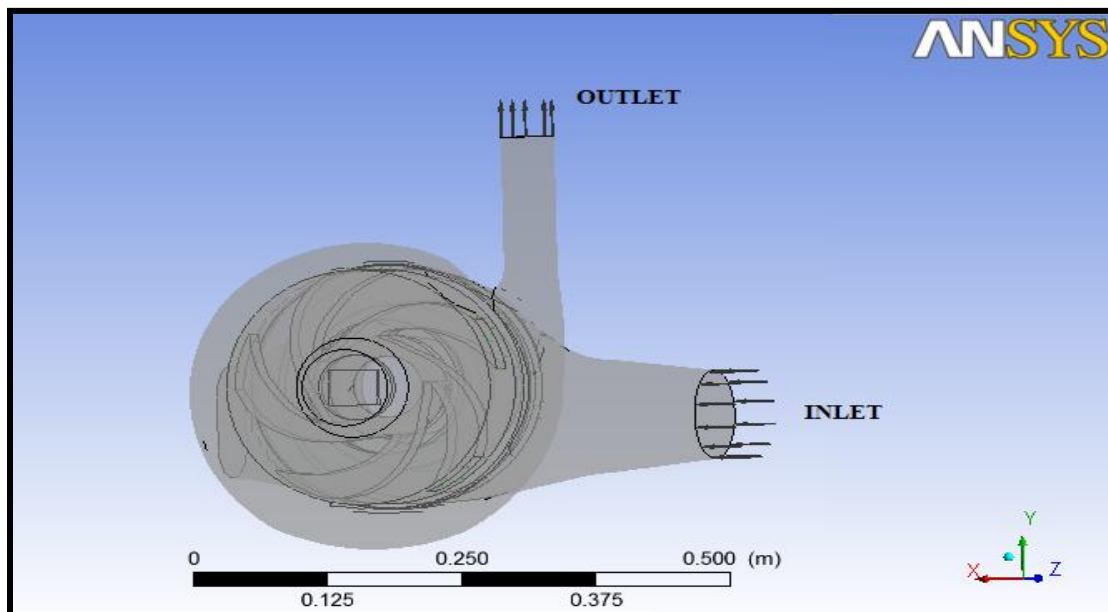


Figure 4.6 Centrifugal Pump with boundary conditions applied

4.10 PERFORMANCE CURVES OF FLY ASH AT VARIOUS SPEEDS:

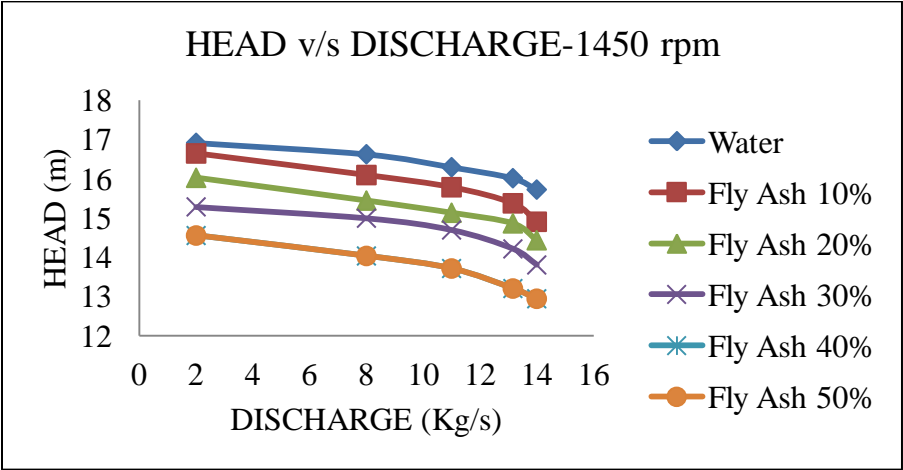


Figure 4.7 Performance curve between Head v/s Discharge of Fly Ash at 1450 rpm

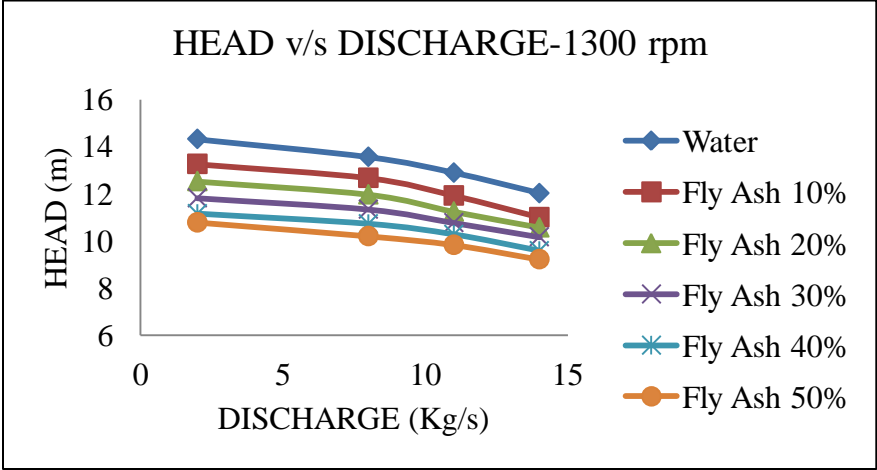


Figure 4.8 Performance curve between Head v/s Discharge of Fly Ash at 1300 rpm

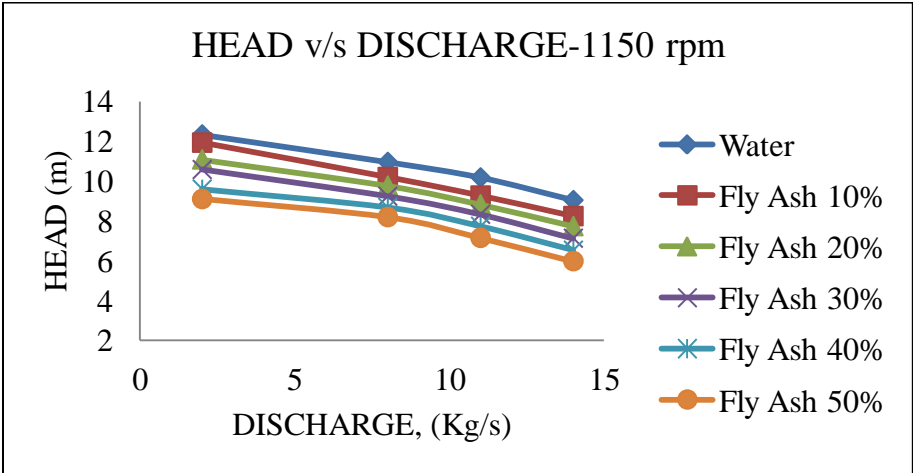


Figure 4.9 Performance curve between Head v/s Discharge of Fly Ash at 1150 rpm

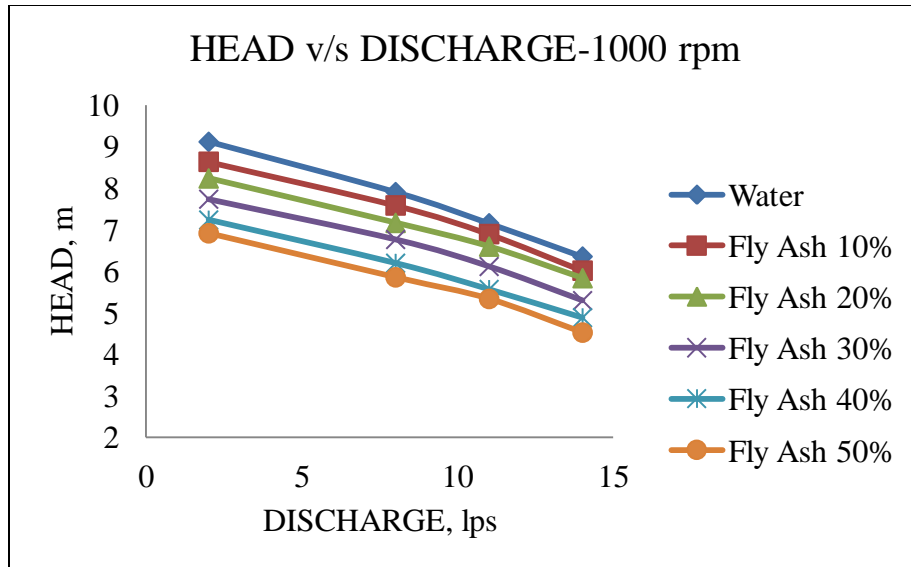


Figure 4.10 Performance curve between Head v/s Discharge of Fly Ash at 1000 rpm

Numerical Simulations has been performed at different speeds namely; 1000rpm, 1150rpm, 1300rpm, 1450rpm for evaluation of Centrifugal Slurry Pump performance handling Fly Ash slurry with concentrations 10%, 20%, 30%, 40% and 50%. From figures 4.7 to 4.10 for fly ash slurry it is observed that with the increase of mass flow rate, there is a decrease in the head developed by the pump and the input power increases linearly.

It is also seen that with the increase of speed of the pump, both the head and input power increases at a given flow rate condition

With the increase of the concentration of the slurry, head decreases and input power increases for handling fly ash.

4.11 PRESSURE DISTRIBUTION OF IMPELLER AND VELOCITY DISTRIBUTION OF PUMP AT 1450 RPM OF FLY ASH AT DIFFERENT CONCENTRATIONS:

4.11.1 at 10% concentration:

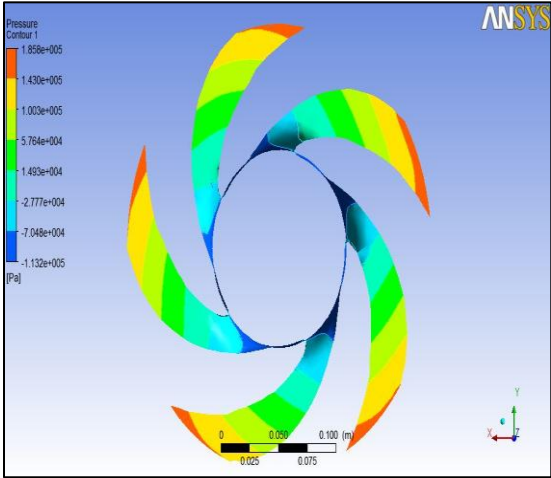


Figure 4.11 a)

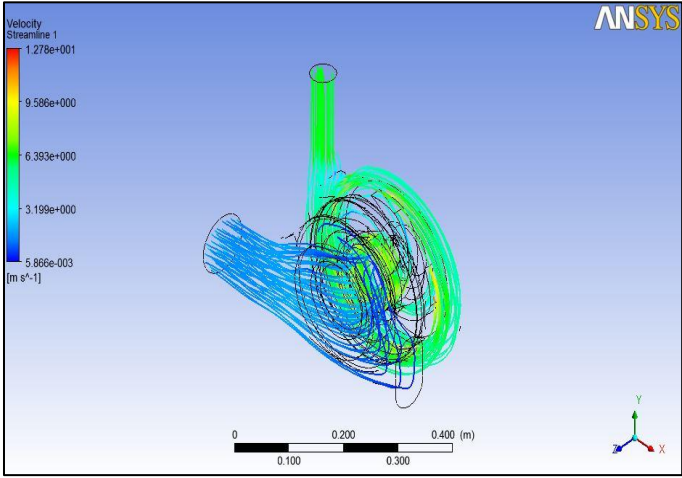


Figure 4.11 b)

4.11.2 at 20% concentration

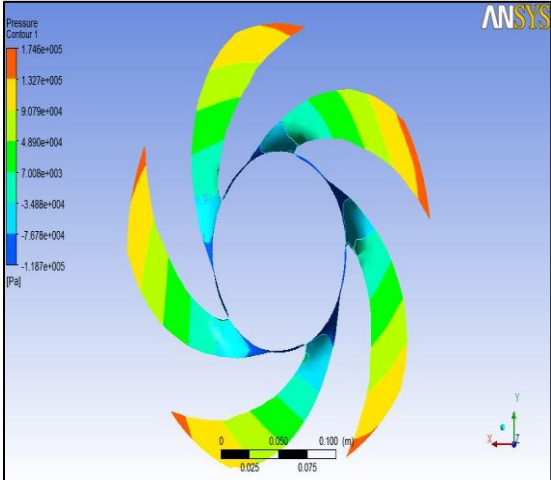


Figure 4.12 a)

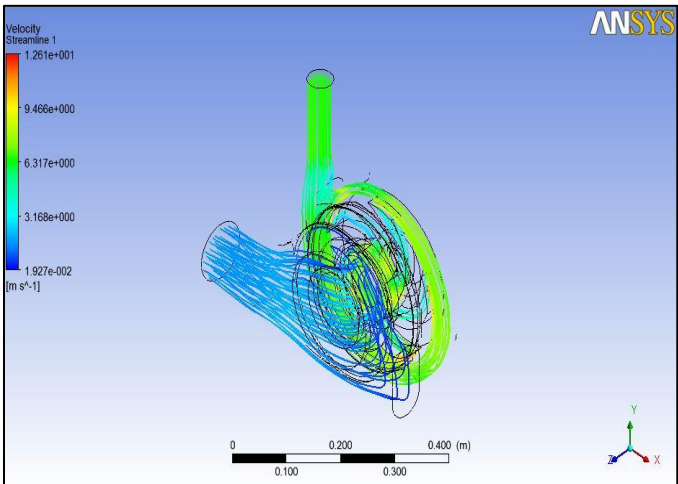


Figure 4.12 b)

4.11.3 at 30% concentration

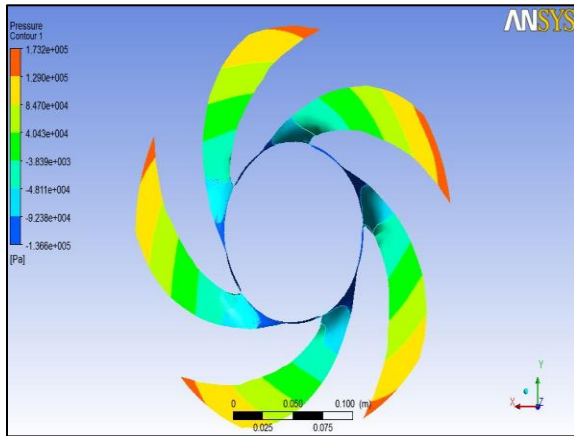


Figure 4.13 a)

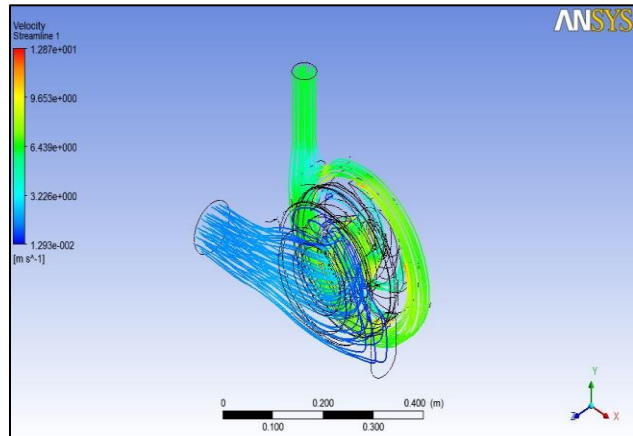


Figure 4.13 b)

4.11.4 at 40% concentration

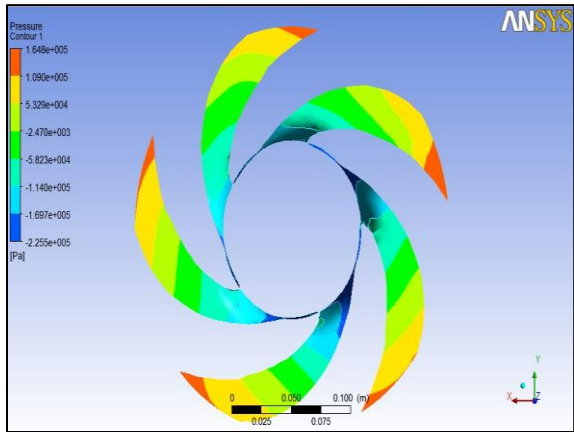


Figure 4.14 a)

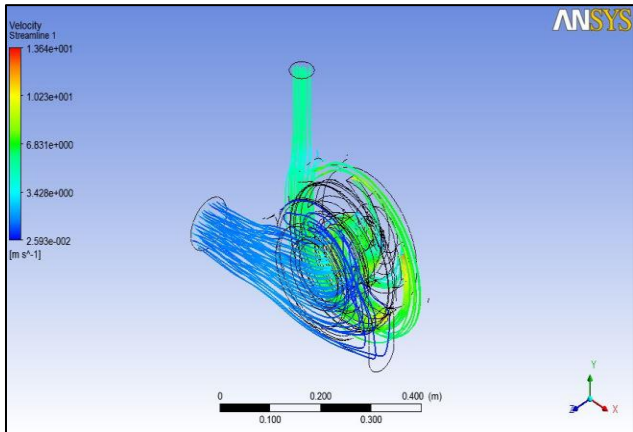


Figure 4.14 b)

4.11.5 at 50% concentration

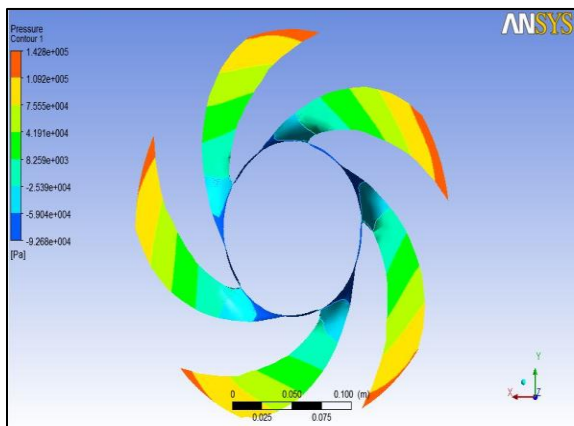


Figure 4.15 a)

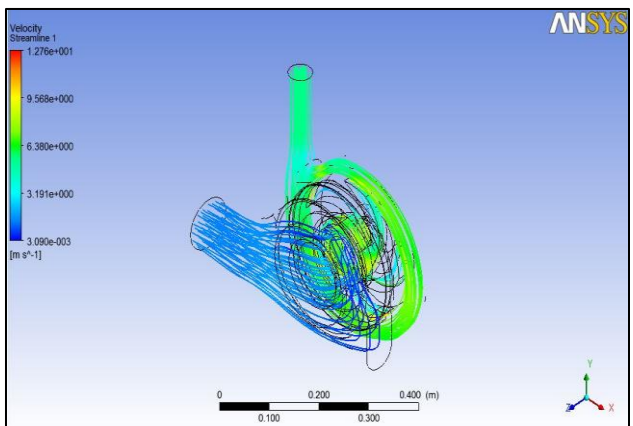


Figure 4.15 b)

The performance characteristic of the centrifugal slurry pump has been predicted numerically handling fly ash slurry. Head, power and efficiency characteristics of the pump are predicted by CFD analysis at 1000rpm, 1150 rpm, 1300rpm and 1450 rpm, and at different concentrations namely 10%, 20%, 30%, 40% 50% graphically in Figures 4.11 to 4.15.

For different flow rates at different speeds and concentration, the relative velocity and pressure distribution in the impeller and the centrifugal slurry pump is obviously different. At small flow rates and low speeds, relative velocity and pressure is lesser. As the pump flow rate increases and the speed increases, the relative velocity increases gradually with the increase in the pressure distribution over the impeller.

4.12 Validation of results

The numerical result was found compatible with the experimental result of water and fly ash. The numerical and experimental results of water at 1450rpm is validated which is being shown in the below figure.

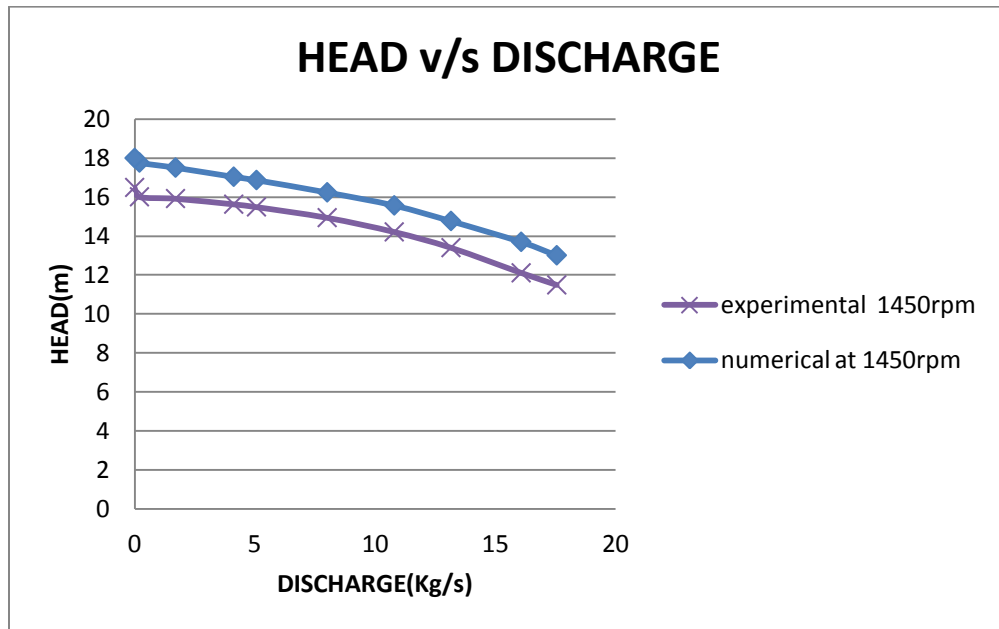


Figure 4.16 numerical-experimental result of head v/s discharge

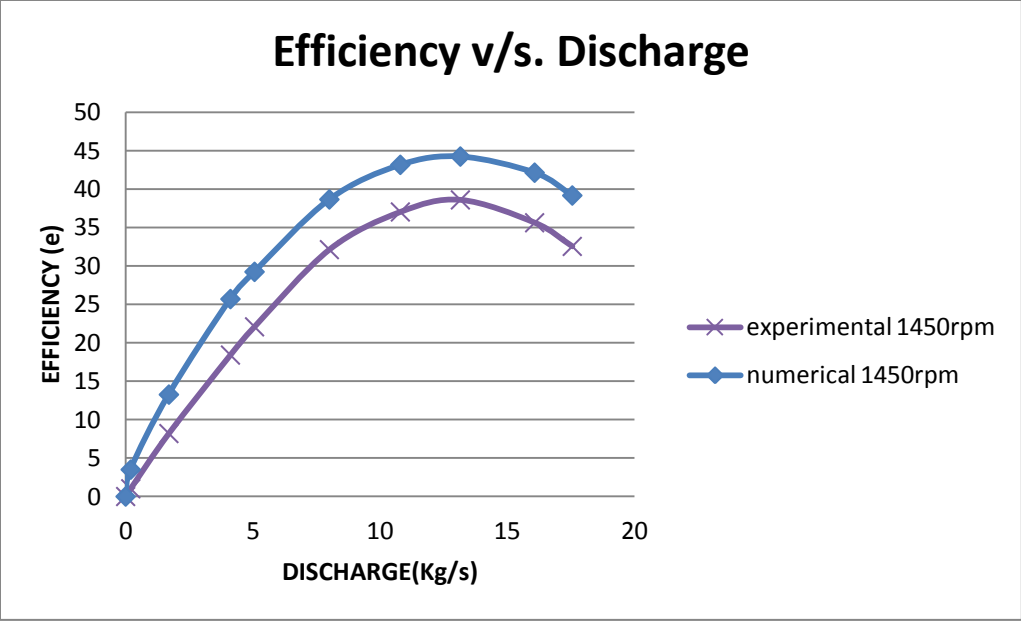


Figure 4.17 numerical-experimental result of head v/s discharge

CONCLUSION AND FUTURE SCOPE

Conclusion:

The performance characteristics of centrifugal slurry pump are evaluated experimentally handling clear water, bottom ash and fly ash slurry. It also includes the numerical simulation of flow through pump handling 10%, 20%, 30%, 40%, and 50% (by weight) concentrated bottom ash and fly ash slurry. The simulation results are obtained at the operating speeds namely 1000rpm, 1150rpm, 1300rpm and 1450rpm with different mass flow rates. The complex pump internal flow field was investigated by using numerical methods. The complex internal flow of the centrifugal pump simulation has permitted to study the internal flow pattern and pressure distribution of the pump operating at 1000rpm, 1150rpm, 1300rpm and 1450 rpm.

Based on the present investigation the following conclusions can be obtained

- It is observed that, head and efficiency of centrifugal slurry pump decreases with increase in solid concentration (by weight).
- In Numerical simulation, Pressure contours shows that the pressure increases from inlet to outlet. It has a minimum value at inlet and increases at the tip of the vane.
- Velocity Vectors shows the velocity is high at the tailing edge of the impeller vane as compare to the leading edge of vane.

At design point, the internal flow or velocity vector is very smooth along the curvature along the blades. However, flow separation developed at the leading edge due to non-tangential inflow conditions. Also due to high speed of the impeller the relative pressure developed is very high. Therefore to compensate the high pressure in the impeller either the thickness can be increased or the material with desirable properties such as hardness can be used.

Future Scope:

- To analyze the hydraulic losses and erosion wear of pump and pipeline in pilot plant.
- Some additive can be added in bottom ash and fly ash slurry to improve performance characteristics of pump.

References

1. Miner, S. M., "3-D Viscous Flow Analysis of an Axial Flow Pump Impeller" *International Journal of Rotating Machinery*, 1997, Volume 3, No. 3, pp 153-161
2. Oh and Kim, Conceptual Design, Optimization of mixed-flow pump impellers using mean streamline analysis. *Proceedings of Institution of Mechanical Engineers*, 2001, Vol. 215.
3. Benra Friedrich-Karl, 2001, Economical Development of Efficient Centrifugal Pump Impellers by Numerical Methods. *World Pumps*, Vol. 416, pp. 48-53.
4. Das, S., Das, P. K., and Maiti, B., 1998, CAD of Centrifugal Pump Impeller with Blades of Single Curvature, 25th National and 1st International Conference on Fluid Mechanics & Fluid Power, pp. 214-223.
5. Goto, A., 1992, Study of Internal Flows in a Mixed-Flow Pump Impeller at various Tip Clearances using Three-Dimensional Viscous Flow Computations. *ASME Journal of Turbomachinery*, vol.114, pp.373-382.
6. Jude Lucian and Covshi Dorel Homent, 1998, Numerical analysis of the Inviscid Incompressible Flow in Two-Dimensional Radial-Flow Pump Impellers, *Engineering Analysis with Boundary Elements*, Vol. 22, Issue 4, pp. 271-279.
7. Kaupert, K.A., Holbein, P. and Staubli, T., 1996, A First analysis of Flow Field Hysteresis in a Pump Impeller. *ASME Journal of Fluids Engineering* 118, pp.685-691.
8. Miner, S. M., 2000, Evaluation of Blade Passage Analysis Using Coarse Grids, *ASME Journal of Fluids Engineering*, Vol. 122, pp. 345-348.
9. Pagalthivarthi, K.V., Kapoor Rajat and Ramanathan, V., 1998, Finite Element Prediction of Turbulent 2D Flow in Centrifugal Slurry Pump Casings, 25th National and 1st Intl. Conf. on Fluid Power, December 15-17, pp. 70-79.
10. Roco, M.C., Hamelin, P. and Davidson, G., 1989, Experimental Study on Centrifugal Slurry Pumps, Freight Pipelines, Proc. 6th Intl. Symp. On Freight Pipelines, Hemisphere Publishing Corp., New York, pp. 386-389.
11. Roco, M.C., Marsh, M., Addie, G.R. and Maffett, J.R., 1986, Dredge Pump Performance Prediction, *J. of pipelines*, Vol. 5, pp. 171-190.

12. Rudramoorthy, R., Prabu, T. and Mohanakrishnan, A.S., 2003, CFD analysis of High Speed Single Stage Submersible Pump Impeller, Proceedings of 30th National Conference on Fluid Mechanics and Fluid Power, December 11-13, pp.558-561.
13. Sun, J. and Tsukamoto, H., 2001, Off-Design Performance Prediction for Diffuser Pumps. ASME Journal of Power and Energy, Proceedings of I. Mech. E, Part A, pp.215.
14. Tamm, A., Ludwig, G. and Stoffel, B., 2001, Numerical, Experimental and Theoretical analysis of the Individual Efficiencies of a Centrifugal Pump. Proceedings of ASME FEDSM' 01 New Orleans, Louisiana, May 29- June 1 pp. 23-34
15. Walker, C.I., Wells, P.J. and Pomat, C., 1993, The Effect of Impeller Geometry on the Performance of Centrifugal Slurry Pumps, Freight Pipelines (ed. G.F. Round), Elsevier Science Publishers Limited, New York, pp. 247-258.
16. Zhou, W. D. and Ng, E. Y. K., 1998, 3-D Viscous Flow Simulation of Mixed Flow Water Pump Impeller with Tip-Clearance Effects, proceedings of the 4th International conference and Exhibition on Pumps and Systems., Singapore: HQ Link Pte Ltd., pp. 189-198.
17. Majidi, K. and Siekmann H. E., (2000), "Numerical calculation of secondary flow in pump volute and circular casings using 3D viscous flow techniques" International Journal of Rotating Machinery, Volume 6, No. 4, pp 245-252
18. Robert Ray., frank Kenyrey., Numerical modelization of the flow in centrifugal pump, Journal of rotator machinery, vol 3, 2005, 244-255.
19. J.H Horlock. J.D. Denton., A review of some early design practice using computational fluid dynamics and current prospective, Journal of ASME, vol 127, 2005
20. Cheah, K.W., Lee, T. S., Winoto, S. H. and Zhao, Z.M., 2007, "Numerical Flow Simulation in a Centrifugal Pump at Design and Off-Design Conditions", International Journal of Rotating Machinery, Volume 200.
21. John S. Anagnostopoulos, A fast numerical method for flow analysis and blade design in centrifugal pump impeller, Journal of computer and fluid, 2008, pp 284-289
22. Behrouz, J., Hajari, A. and Mehdi M., 2010, "The Flow Simulation of a Low- Specific-Speed, High- Speed Centrifugal Pump" Applied Mathematics and Modeling , Elsevier publication

23. Lamloumi Hedi, 2 Kanfoudi Hatem, 3 Zgolli Ridha, 2010 “Numerical Flow Simulation in a Centrifugal Pump “vol 5, pp 300-304
24. Krishnan V. Pagalthivarthi, Pankaj K. Gupta, 2011, “CFD Predictions of Dense Slurry Flow in Centrifugal Pump Casings” vol 5, 2011, pp 254-266