INVESTIGATION OF FLOW OF CENTRIFUGAL SLURRY PUMP HANDLING BOTTOM ASH

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

Master of Engineering

In

CAD/CAM &ROBOTICS

By:
Harsimran Singh
(800981012)

Under the supervision of:

Dr. S.K. Mohapatra
Senior Professor (Mechanical Engineering Department)&
Dean of Academic affairs.

(July 2011)

MECHANICAL ENGINEERING DEPARTMENT
THAPAR UNIVERSITY
PATIALA – 147004

DECLARATION

I hereby declare that the work which is being presented in the dissertation work entitled, **"INVESTIGATION OF FLOW OF CENTRIFUGAL SLURRY PUMP HANDLING BOTTOM ASH"**, in partial fulfillment of the requirements 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 **Dr. S.K Mohapatra**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.

Date: - 15-07-2011
Place: - Patiala

Harsimran Singh
(HARSIMRAN SINGH)

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

Supervisor:
Dr .S.K Mohapatra
Senior Professor &Dean of Academic Affairs
Mechanical Engineering Department
Thapar University, Patiala

Countersigned by:

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

S.K. Mohapatra
Dean Of Academic Affairs
Thapar University,Patiala

ACKNOWLEDGEMENT

I would like to express my most sincere appreciation and deep sense of gratitude and indebtedness to my guides **Dr. S.K. Mohapatra** Senior Professor Mechanical Engineering Department& Dean of Academic Affairs, Thapar University, Patiala for their continuous indefatigable guidance, which paved me on to the path to carry this project. I am highly indebted to them for their painstaking efforts and invaluable suggestions during the period of work.

Date: - 15-07-2011

Place: Patiala

(Harsimran Singh)

ABSTRACT

Centrifugal pumps are used for transportation of solids and liquid 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. 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 of bottom ash slurry at different speeds.A numerical model of an impeller and casing has been generated using *ANSYS 12*.The pump characteristics of centrifugal slurry pumps are influenced by various particle parameters such as concentration and flow velocity of slurries.

CONTENTS

List of figures and tables

CHAPTER 1

INTRODUCTION

1.1 PUMP:

Pump is a mechanical device which is used to increase the pressure of a liquid. It is also used for rising fluid from a lower level to a higher level. Pumps are used wherever any quantity of liquid must be moved from one place to another. Pumps are found in such services as steam power plants, water supply plants, sewage, drainage or irrigation, oil refineries, chemical plants & steel mills, food processing factories & mines, dredging or jetting operations, hydraulic power services & almost every ship whether driven by diesel or steam engine.

1.1.1 Classification of pump:

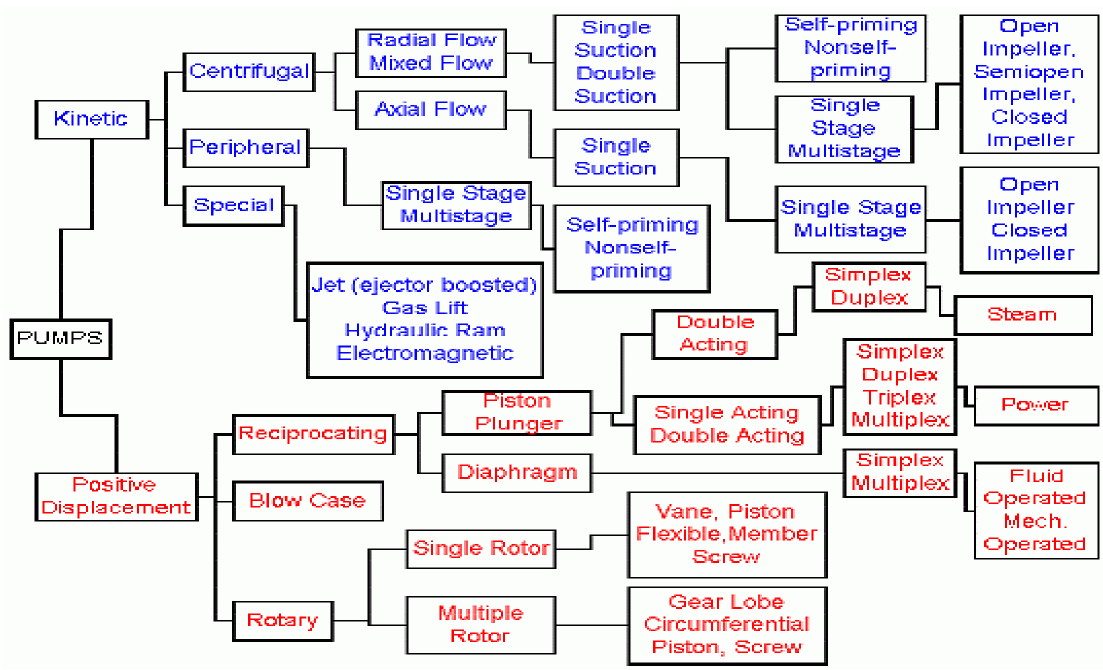


Figure 1.1 Classification of pump

1.1.2 Reciprocating Pump:

In a reciprocating pump, a volume of liquid is drawn into the cylinder through the suction valve on the intake stroke and is discharged under positive pressure through the outlet valves on the discharge stroke. The discharge from a reciprocating pump is pulsating and changes only when the speed of the pump is changed. This is because the intake is always a constant volume. Often an air chamber is connected on the discharge side of the pump to provide a more even flow by evening out the pressure surges. Reciprocating pumps are often used for sludge and slurry.

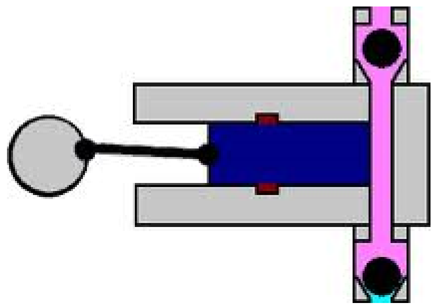


Figure 1.2: Reciprocating Pump

1.1.3 Centrifugal Pumps:

A centrifugal pump is a roto-dynamic pump that uses a rotating impeller to increase the pressure and flow rate of a fluid. Centrifugal pumps are the most common type of pump used to move liquids through a piping system. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward or axially into a diffuser or volute chamber, from where it exits into the downstream piping system. Centrifugal pumps are typically used for large discharge through smaller heads.

Centrifugal pumps are most often associated with the radial flow type. However, the term "centrifugal pump" can be used to describe all impeller type rotodynamic pumps including the radial, axial and mixed flow variations.

The mechanical Energy is converted into pressure energy by centrifugal force, hence these pumps are known as centrifugal pumps. These pumps are used for Low precision and low pressure requirements.

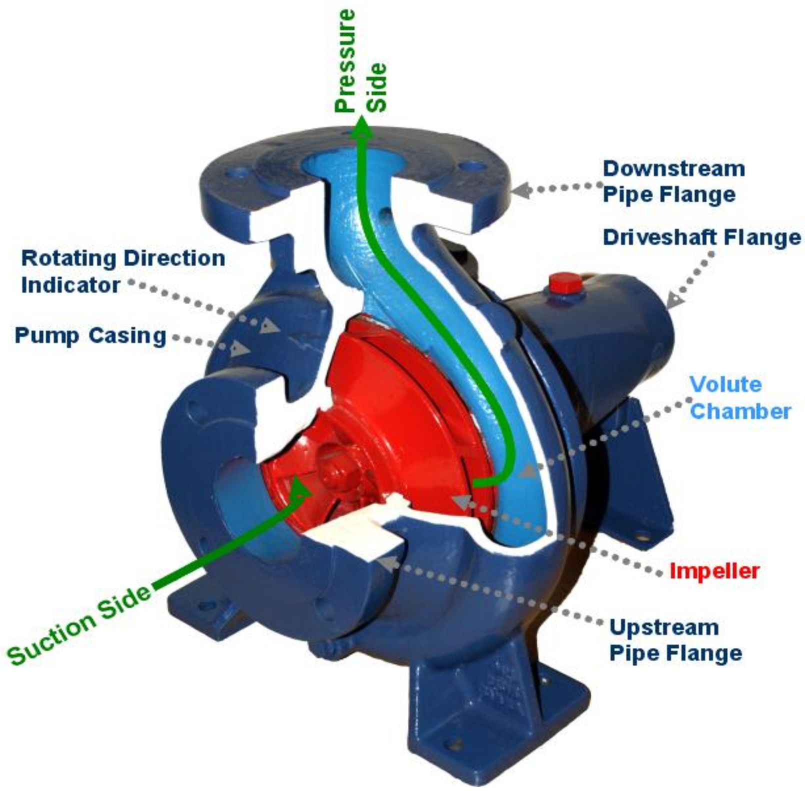


Figure 1.3: Centrifugal pump

1.2 WORKING PRONCIPLE OF CENTRIFUGAL PUMP:

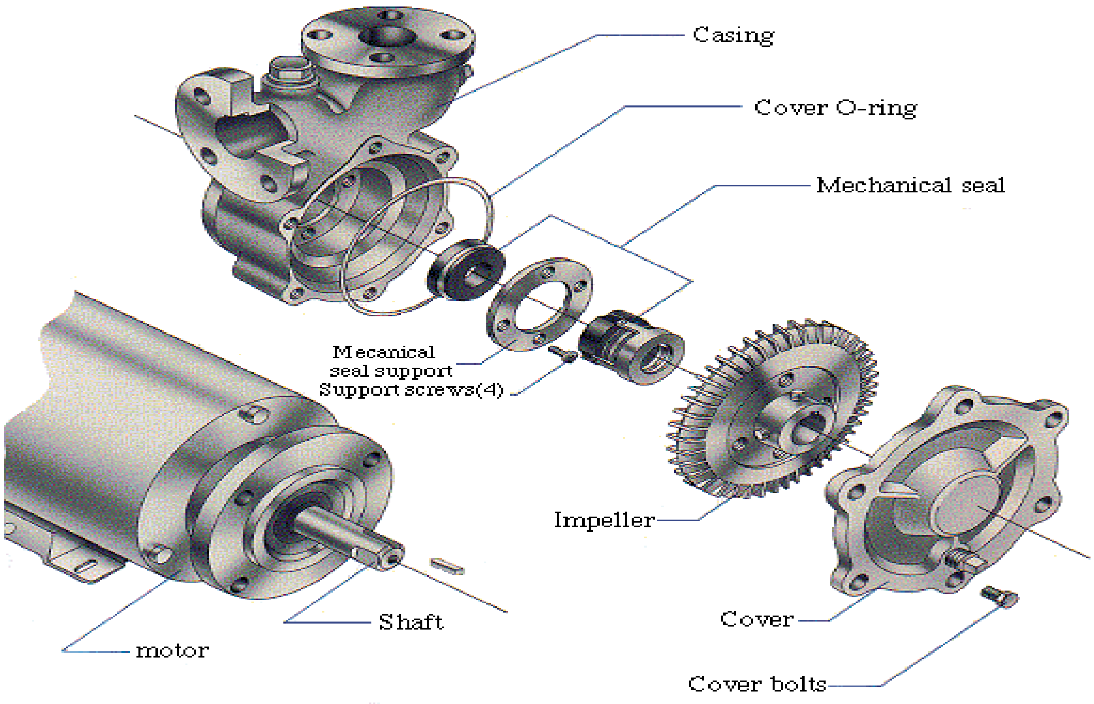
Pump is adding energy to a fluid. The energy addition is achieved by means of centrifugal force in centrifugal pump. When fluid enters into the impeller eye (Center of the impeller where vacuum exists because of pumping), the fluid will be thrown out of impeller eye by a centrifugal force which adding the velocity to the fluid. The volute surrounding the impeller guides the fluid to the discharge and it converts the velocity energy to pressure energy(head).when u keep the discharge valve closed, it will attain the shut off head. Once the discharge valve is opened the fluid will start to flow and head will start decreases since the energy addition is shared by pressure energy and velocity energy. Impeller is rotated by means of a motor or engine, a low pressure region is created at the impeller eye causing the liquid to be sucked into the eye. As the impeller rotates the sucked fluid is thrown out with force through the periphery of the impeller now the diffuser comes into action the diffuser due to its shape of varying cross-sectional area causes the liquid to slow down and from bernoullis principle, a reduction in kinetic head will be compensated by an increase in pressure head.

Pumps are used throughout society for a variety of purposes. Early applications include the use of the windmill or watermill to pump water. Today, the pump is used for irrigation, water supply, gasoline supply, air conditioning systems, refrigeration (usually called a compressor), chemical movement, sewage movement, flood control, marine services, etc.

1.3COMPONENTS OF A CENTRIFUGAL SLURRY PUMP:

Common components of a Centrifugal Pump are:

- Moving Components
    - Impeller
    - Shaft
- Non Moving Components
    - Motor
    - Bearings
    - Glands
    - Mechanical seals
    - Casing



:

Figure 1.4 Components are briefly shown in the figure

1.3.1 Impeller:

Impeller is the part which provides the desired pressure energy to the fluid. It changes the direction of flow of fluid in pumps. Impellers can further be classied depending upon various factors as follows:

- Based on suction criterion
    - Single Suction

      These Pumps have Fluid inlet from one side only.
    - Double suction

      These impellers have fluid inlets from both the sides. These impellers are used for very high discharge requirements.
- Based on Design criterion
    - Closed impeller

This type of impeller has shrouds on both its sides. This type of impeller is used for fluids with high viscosity. These impellers are generally usedclean liquids because tolerances airtight at the eye and the housing, and there is no room for suspended solids, crystals or sediments

Figure 1.5 Closed Impellor

- Semi closed impeller

  This type of impeller is closed on one side only. They are used for comparatively low viscous fluids. The efficiency of these impellers is governed by the limited free space or tolerancebetween the front leading edge of the blades and the internal pump housing wall. This type of impellers is generally used at the bottom of a tank to empty it.

Figure 1.6 Semi closed Impellor

- Open type impeller

  This type of impeller is used for high discharge and low viscosity fluids. With its open tolerances for moving and grinding solids they are not high efficiency devices. This type of impeller doesn’t provide high head discharge.

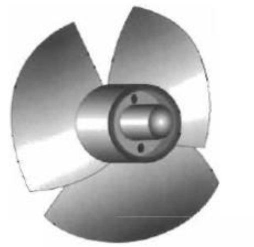


Figure 1.7 Open type Impellor

- Based on Direction of flow:
    - Axial flow
    - Radial flow
    - Mixed flow

As their name suggests these types of impellers are classified according to the direction of flow of fluid at the outlet.

1.3.2Volute Casing:

A volute is a curved funnel that increases in area as it approaches the discharge port. The volute converts kinetic energy into pressure by reducing speed while increasing pressure, helping to balance the hydraulic pressure on the shaft of the pump. The name "volute" is inspired by the resemblance of this kind of casing to the scroll-like part near the top of an ionic order column in classical architecture, called a volute. In free-vortex, angular momentum is constant . Most of the single stage pumps are built with volute casing.

1.3.3 Seal Chamber and stuffing Box:

Seal chamber and Stuffing box both refer to a chamber, either integral with or separatefromthe pump case housing that forms the region between the shaft and casing where sealingmedia are installed. When the sealing is achieved by means of a mechanical seal, thechamber is commonly referred to as a seal chamber. When the sealing is achieved by meansof packing, the chamber is referred to as a Stuffing Box. Both the seal chamber and thestuffing box have the primary function of protecting the pump against leakage at the point. where the shaft passes out through the pump pressure casing. When the pressure at thebottom of the chamber is below atmospheric, it prevents air leakage into the pump. When thepressure is above atmospheric, the chambers prevent liquid leakage out of the pump. The sealchambers and stuffing boxes are also provided with cooling or heating arrangement forproper temperature control.

1.3.4 Gland:

Gland is the part of pump which separates the electric motor and the pump. It provides the sealing between the motor and pump impeller. It consists of an Mechanical seal with sound ground stones, flush, quench.

Figure 1.8Mechanical Seal

1.3.5 Throat bushing:

It forms a restricted clearance between the impeller and the pump, it is used for the proper alignment and adjustment of the shaft.

Throttle Bushing:

It forms a close clearance around the shaft at the outer end of the mechanical seal, preventing the slippage of mechanical seal and further strengthening the sealing.

Bearing housing:

Bearing housing of the pump is the part which holds the bearings in the pump which supports the pump drive shaft and the impeller. The shaft rotates inside the bearings. Different types of bearings can be used depending upon the weight f the shaft and the power to be transmitted to the impeller. Bearing housing also holds lubricating oil but largely, self Lubricating bearings or the bearings with lubricant filled with seals are generally used.

1.4 HEADS OF PUMP

Suction head: Distance between center line of pump and water level of sump is called suction head

Delivery head: Distance between center line of pump and level of discharge is called delivery head

Static head: It is the sum of suction and delivery head. If H_s and H_d be the suction and delivery heads resp. the static head

$$H_{staticH} = H_s + H_d$$

Manometric head: It is the head measured across the pump inlet and outlet flanges, it is expressed as the increase in pressure energy per unit weight of liquid handled by the impeller

$$H_{mano} = \frac{(P_d - P_s)}{\rho g} + H_g$$

H_{mano}=H_{th}(theoretical head developed by pump) +loss of head in pump

Manometric head is the different between the delivery head (+) and suction (-) pressure head (difference between reading shown by gauge plus vertical distance between the pressure tapping for suction and delivery gauge)

Total Gross or effective head: This is the actual head against which the pump has to work. It is equal to the static head plus all the head losses occurring to flow before through, and after the impeller.

$$H = \frac{(P_d - P_s)}{\rho g} + H_g + \frac{(V_{d_2} - V_{s_2})}{2g}$$

Where

H =total or effective head in meters of liquid column

V_d = velocity of water in delivery pipe

V_s=velocity of water in suction pipe

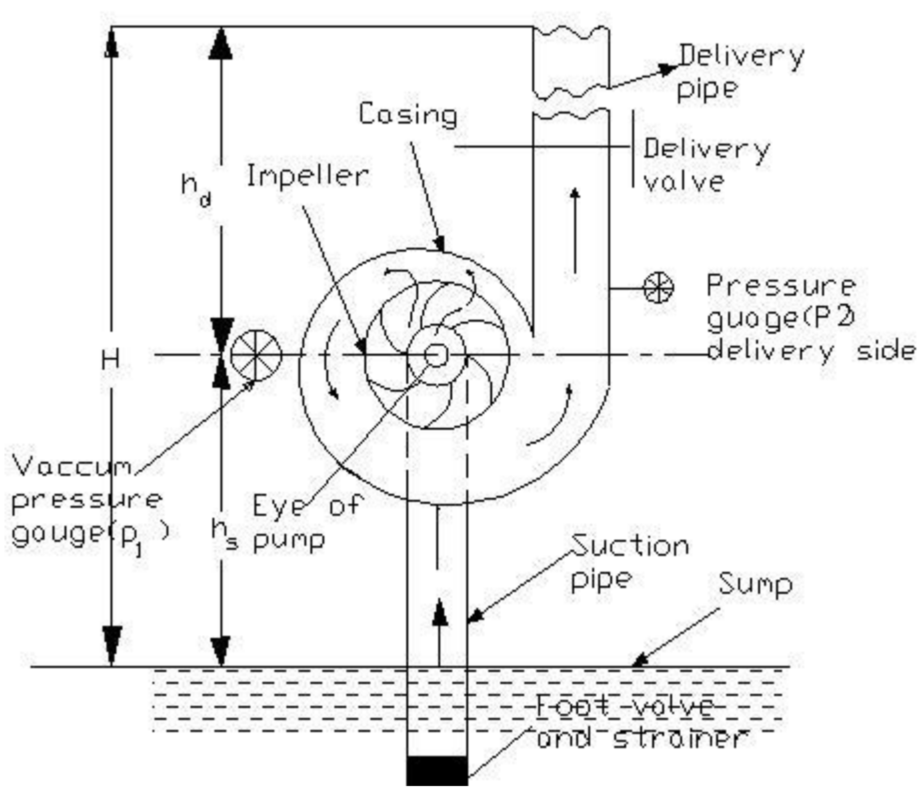


Figure 1.9 Head of centri fugal Pump

If H_L be the total loss of head, then the manometric head,

$$H_{mano} = H_{static} + H_L$$

The above heads are independent of the density of the liquid being raised. A centrifugal pump rotating at a particular speed will raise water, oil or mercury to the same height. But pressure generated in pump will be different in each case also power required to be different.

1.5 PERFORMANCE OF PUMPS-CHARACTERISTIC CURVE

A pump is usually designed for a particular speed, flow rate and head, but in actual practice the operation may be at some other condition of head, and for the changed condition the behavior of the pump is less efficient than the quantity the value of velocity of flow of liquid through impeller will be changed. As a result, the value of v, r and vr will be changed, and at the same time the loss will be increased so the efficiency of pump will lowered, therefore, in order to predict the behavior and performance of a pump under varying condition, tests are performed, and the result of test are plotted, the curve thus obtained are known as characteristic curve of a pump. Characteristic curve are usually prepared for the centrifugal pumps are main and operating characteristic curve, constant efficiency curves, constant head & constant- discharge curve.

1.5.1 Main characteristics

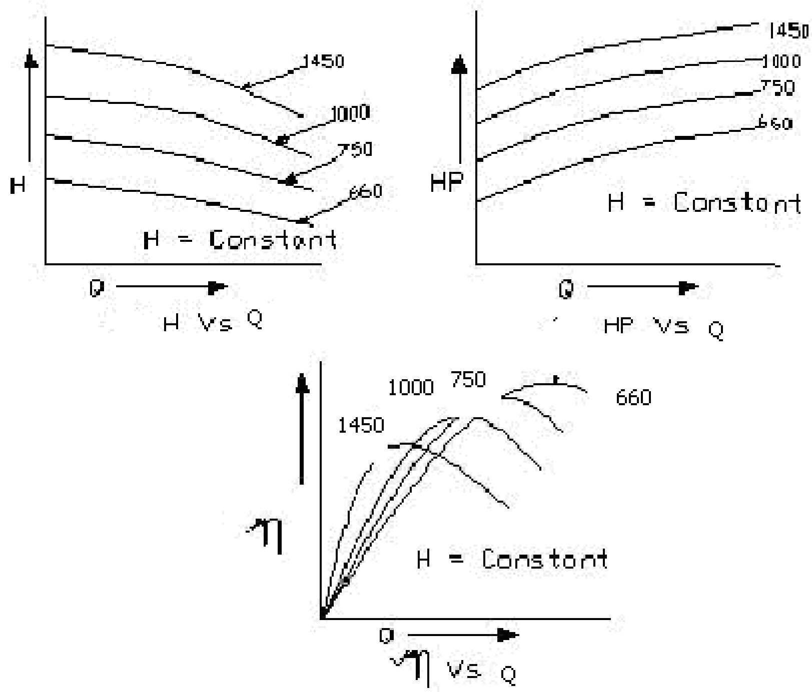


Figure 1.10Main characteristic curves of centrifugal pump.

These are obtained by fixing the speed at some arbitrary value of and plotting separately H, HP, & η against Q. The rate of flow Q is varied by means of the quantity H, HP, η are calculated. A number of different values of N are chosen and one such set of curves is drawn for each speed.

1.5.2 Operating characteristics

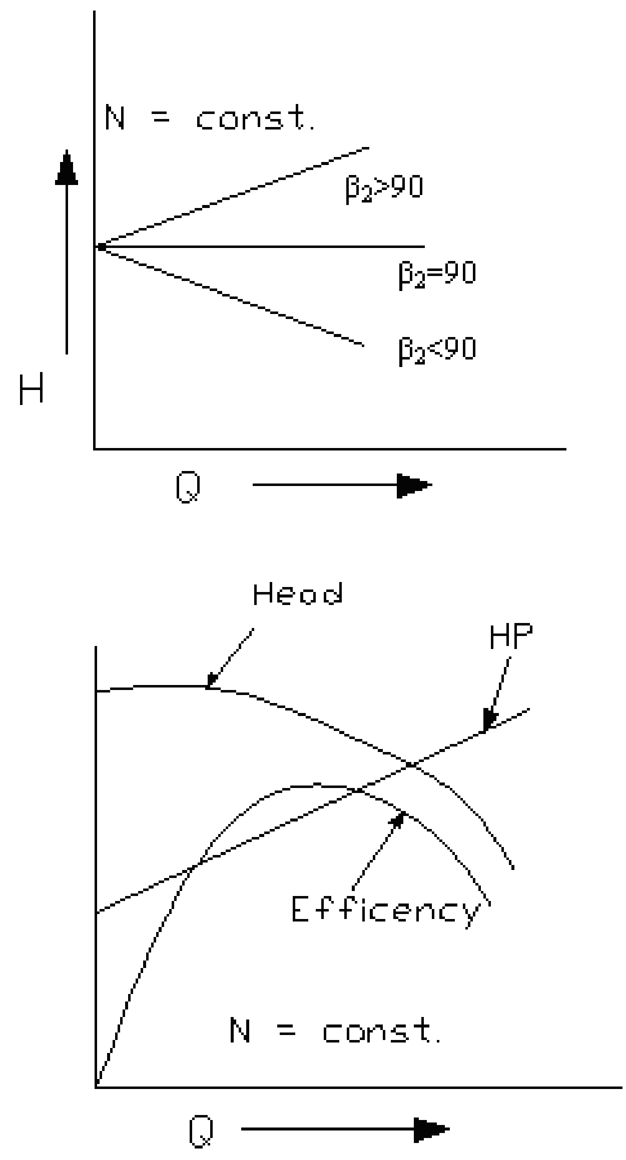


Figure 1.11Operating characteristic curves of centrifugal pump

During operation the pump must run at a constant speed. Normally, this is the designed speed. The particular set of main characteristics which correspond to the designed speed is mostly used in operation and is therefore known as operating characteristics.

1.5.3Muschal Curve or Constant Efficiency Curve

With the help of data obtained from the above curve, a series of constt. Efficiency curve can be obtained. They facilitate the job of the salesman and enable the prospective customer to see directly the range of operation with a particular efficiency,

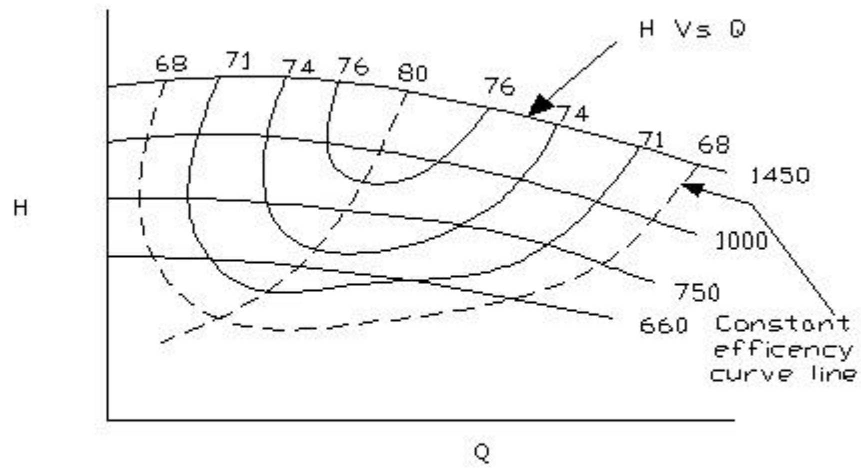


Figure 1.12Muschal Curve

1.5.4 Constant Head and Constant Discharge Curve

It is quite possible that a pump may be required to deliver water at a certain height, in which case it is fixed. If for some reason the speed varies, discharge will also be affected. The performance of the pump under such condition, it is necessary to drawn a constant H curve by plotting Q Vs N. Similarly to determine the speed required to discharge a certain quantity at diff. pressure it is convenient to draw constant Q curves showing H against N.

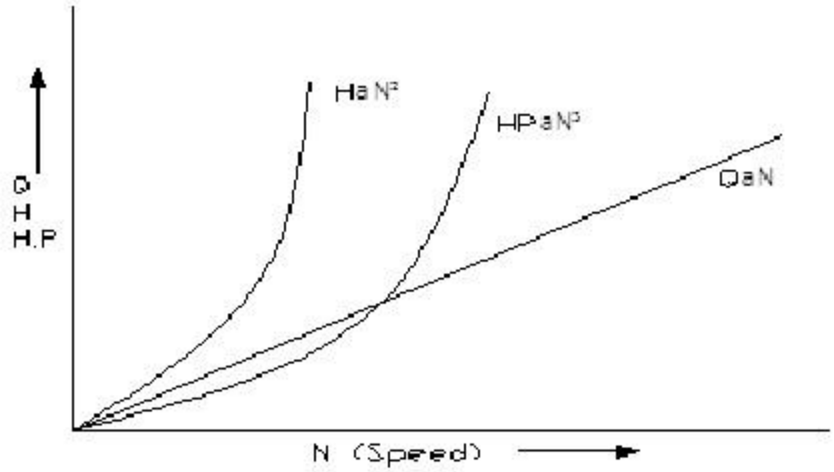


Figure 1.13 Constant Head and Constant Discharge Curve

1.6 CENTRIFUGAL PUMP APPLICATIONS

Pumps are used wherever any quantity of liquid must be moved from one place to another. Pumps are found in such services as steam power plants; water supply plants; sewage; drainage or irrigation; oil refineries, chemical plants and steel mills; food processing factories and mines; dredging or jetting operations; hydraulic power services and almost every ship whether driven by diesel or steam engine. While these pumps have much in common, they are varied to meet special requirements and particular needs of each service.

- Petroleum Industry
- Chemical Industry
- Textile Industries
- Paper Industry
- Sewage and Sump Services
- Irrigation, Drainage and Flood Control
- Mining and Construction

1.7 FACTORS AFFECTING PUMP PERFORMANCE

The presence of solids in the liquid affects the performance of pump. Some of the important factors which influence the performance of the pump are:

- Concentration of solids
- Density of solids
- Size and size distribution of particles
- Rheological parameters of the slurry
- Geometry and size of pump
- Flow field inside the pump.

1.8 SLURRY

Slurry is a mixture of solids and liquids. Its physical characteristics are dependent on many factors such as size and distribution of particles, concentration of solids in the liquid phase, size of the conduit, level of turbulence, temperature, and absolute (or dynamic) viscosity of the carrier. Nature offers examples of slurry flows such as seasonal floods that carry silt and gravel. A slurry mixture is a mixture of a carrying fluid and solid particles held in suspension. The most commonly used fluid is water, however in some cases air is also used such as in pneumatic conveying. Theoretically, a single-phase liquid of low absolute (or dynamic) viscosity can be allowed to flow at slow speeds from a laminar flow to a turbulent flow. However, a two-phase mixture, such as slurry, must overcome a deposition critical velocity or a viscous transition critical velocity. If the slurry's speed of flow is not sufficiently high, the particles will not be maintained in suspension. On the other hand, in the case of highly viscous mixtures, if the shear rate in the pipeline is excessively low, the mixture will be too viscous and will resist flow.

1.8.1 TYPES OF SLURRY FLOWS

There are generally two types of flows:

1.8.1.1 Homogeneous flows

In homogeneous flows solids are uniformly distributed throughout the liquid carrier. For example copper concentrate slurry after undergoing a process of grinding and thickening. Drilling mud, sewage sludge, and fine limestone behave as homogeneous flows.

1.8.1.2 Heterogeneous flows

In heterogeneous flows, solids are not uniformly mixed in the horizontal plane. Heavier particles tend to settle down and lighter particles tend to float. Sliding bed may form in the pipe, with the heavier particles at the bottom and the lighter ones in suspension. Heterogeneous slurries are encountered in many places mining, phosphate rock mining, and dredging applications. Heterogeneous flows require a minimum carrier velocity.Although the transportation of solids in the form of slurries is basically older than history. The blood circulating system in mammals involves the use of a positive displacement pump forcing slurry of solid corpuscles in liquid serum through a complex pipeline. The slurry transportation of solids through long pipelines and with the help of centrifugal pumps has been undertaken only in almost last 25 years.

1.9TRANSPORTATION OF SLURRY

Over the last few decades there has been a phenomenal growth in the demand of raw materials. This rise in demand has led to drastic changes the existing techniques of mining, food processing, power generation and other sectors where transportation of suspended solids play a major role. Due to this change, there has been an increase in requirements in slurry transportation. A typical slurry transport system is shown in the figure 1.10. Pump plays a vital role in transportation of slurry; different types of pumps are available for slurry transportation, but centrifugal pumps are most widely used in slurry transportation applications due to the following reasons:-

• Higher flow rates can be obtained.

• Pulse free flow can be obtained.

• Higher initial and maintenance cost of positive displacement pumps.

• Solid particle of any size can be transported.

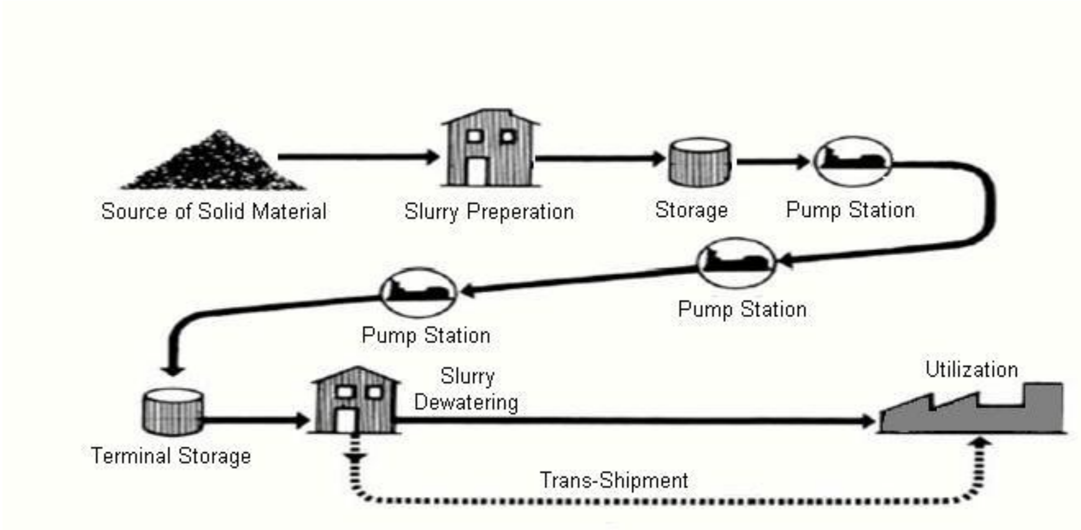


Figure 1.14 Transportation of slurry

1.9.1 Pumps used for transportation of slurries

The choice of pumps or pumping systems for slurry transport will depend not only on the flow, head required, suction conditions, type of installation and location, as for any other pump application, but also on the slurry flow regime and properties.

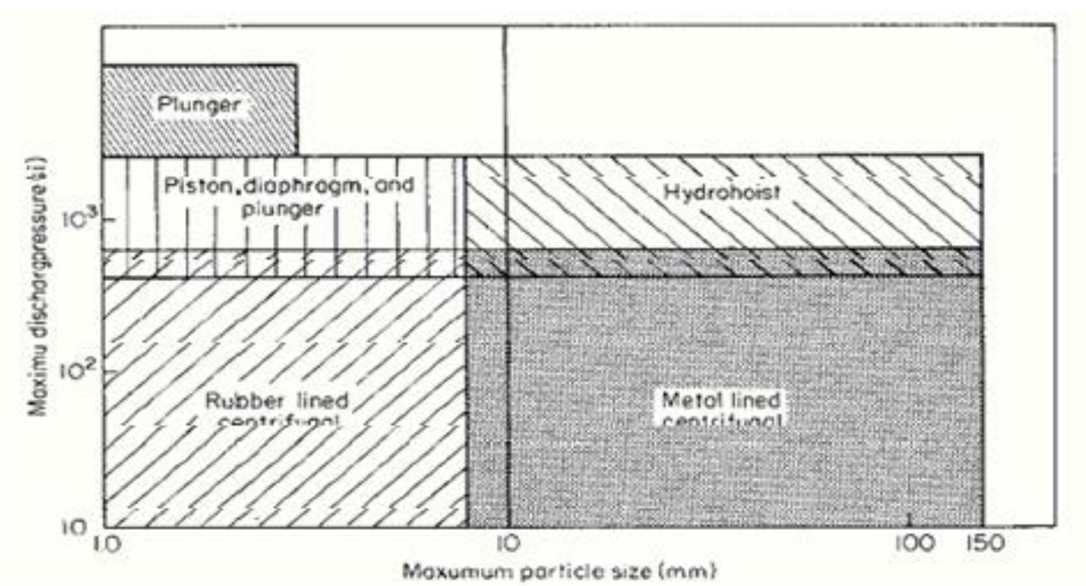


Figure 1.15 Pumps used for transportation of slurries

Rot dynamic pumps, of which the centrifugal or radial-flow type is the most common in slurry service, are usually considered for the higher flow, lower head duties, whereas conversely, positive-displacement reciprocating types tend to be used for the lower flow, high pressure applications, e.g. long-distance pipelines. However, relatively high pressures may also be achieved with centrifugal pumps, depending on casing pressure limitations, by arranging them in series. 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. Figure 1.14 gives a pictorial idea of pump application depending upon discharge and particle size required.

CHAPTER 2

LITERATURE REVIEW

Sturge et al(1975) reported a method for calculating two-dimensional incompressible and in viscid flow within a centrifugal impeller when the flow separation from the suction side has been developed. They assumed that mixing of the through flow with the separated region is suppressed. The flow in an impeller passage was divided into two regions. The flow in the main region was regarded as in viscid and irrigational, while in the wake region there was no flow. The axial inlet and radial outlet impeller was used for calculation. In this report, they had shown the streamline pattern and the overall parameters for the flow in a centrifugal impeller which were observed to change by altering the flow rate and the impeller shape.

Wiedenrothet al.(1978) from his experiments on two-bladed and three-bladed impeller pumps with slurries of different solid concentrations has shown the existence of a relationship between the head loss, solid concentration and particle Reynolds number. Based on the experiments on a rubber-lined slurry pump.

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 (Cv=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.

Gotoet al.(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.

Liu et al. (1994) have measured velocity in a centrifugal pump impeller passages, the volute, the inlet and exit ducts, by use of Laser-Doppler Velocimeter. The pump has a radial flow impeller with four backswept blades and a single volute. In this study, they found that the flow in the impeller passages depart at the curvature of the blade surfaces at off-design conditions particularly at the suction surface and front shroud.

Kaupertet 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.

Ni et al. (1996) have experimentally evaluated the effect of high delivered volumetric concentration (c_{vd}) on characteristics of a slurry pump. They performed extensive experiments by using three sorts of narrowly graded sands for the observation of pump and pipeline characteristics. They conclude that high solid concentration has a strong influence on the pump head, efficiency and power consumption and this influence behaves differently with different sand size. The pump efficiency in coarse sand slurry service may drop almost 60% compared to that of water service, when c_{vd} = 42%. Within the measured range of concentrations in each passages may experience similar stratification process occurred in pipelines. The mechanical friction regime in the impeller passages could be similar to that in pipelines. Therefore the delivered volumetric concentration and the size affect the head loss in the same way both in pumps and pipelines.

Miner, S. M.et al.[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

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.

Zhou et al.(1998) have developed a three-dimensional time-marching, incompressible Navier-Stokes equation solver using the pseudo-compressibility technique to study the flow field through a mixed-flow water pump impeller. The applicability of the original code was validated by comparing it with the experimental and computational results available in the literature.

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.

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.

Chung et al.[1999] has developed optimum design code of the pump. They determined the geometric and fluid dynamic variables under the appropriate design constraints. Optimization problem has formulated with a non-linear objective function to minimize losses, net positive suction head required and product price of a pump stage depending on the weighting factor selected as the design compromise. Optimal solution obtained, efficiency $NPSH_R$ depends design variable of centrifugal pump. Selected in the range of weighting factor 0 to 1.designer can easily find the optimum value of design variable to meet their particular requirement of pump design.

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 et al.(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.

Sinha and Katz (2000) have used Particle Image Velocimetry (PIV) system for measurement of the unsteady flow structures and turbulence in a transparent centrifugal pump with a vanned diffuser. A high speed digital CCD camera was used to record the images. After the experimental study, they found that the entire flow field was dominated by a series of wakes, generated by the impeller blades, and diffuser vanes into the volute. Boundary layer on the diffuser vanes was reported to get affected strongly by unsteady flow field associated with the impeller.

Ogut,A.et al.[2000]haveprovidedaninsightintotheeffectivenessoffluidinjectionas a boundary layer control method in suppressing or eliminating flow separation in the vanneddiffuserat off-design flow conditions.The reverseflow wasobservedalongthe hubandshroudwalls. Thephenomenonofflowseparationalongthe walls was also reduced byinjectingthefluid.Thepressurerecoverywillbemaximumifinjection ratewas3%tothe60%ofdesignflowrate.

Majidi, K.et.al.[2000]haveobservedthesecondaryflowinvoluteandcircularcasings ofcentrifugalpumps.Thestatic pressurewasnotdistributeduniformlyattheoutletofthe impellerwhichresultsintheradialthrust. Also themaximum value of relativevelocityoccursatthe peripheryofthe impeller.Theanalysisshowsthatthecurvatureofthe casingcreates pressuregradientsthat causevorticesatcross-sectionalplanesofthe casings.

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.

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.

Benraet al.(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.

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.

Gandhi et al. [2001] have studied erosion wear at various locations inside the volute casing of a centrifugal slurry pump for the flow of solid-liquid mixtures. They reported that the wear increases all along the volute periphery with increase in the amount of solid suspended in the mixture and wear smaller when the pump operates near the (BHP).

Gandhi et al. (2001) have studied the performance of two centrifugal slurry pumps for three solids materials having different particle size distribution (PSD) in terms of head, capacity and power characteristics. The results have shown that values of head and efficiency ratios are not only depended on solid concentration but are also affected by PSD of the solids and properties of slurry. They conclude that the head and efficiency of the pump decrease with increase in solid concentration, particle size and slurry viscosity, the decrease in the head being 2-10% higher than that of the efficiency. The presence of finer particles (<18 μm) in coarse slurries substantially attenuate the loss of the performance of the pump in terms of head and efficiency

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.

Chung M K et al. [2001] developed a simple and accurate correlation for the slip factor of centrifugal impeller. He investigated the radius of relative eddy inscribed by two adjacent vanes and the exit circle of a flow channel in the impeller to obtain the correlation.

Engin, T. et al. [2001] have conducted the experimental study on the effects of solids in an unshrouded centrifugal pump impeller by varying the tip clearance when both handling water and solid-water mixtures. The tip clearance loss mechanisms seem to be similar to those in single phase pumping and a steady deterioration in pump performance was observed by increasing the tip clearances. The tip leakage losses increase as the tip clearance increases. The head reduction factor appears to be almost independent of the variation in the tip clearance, and which allows correlating the solids effects on the pump performance regardless of changes in the tip clearance. A slight decreasing trend was also observed in efficiency ratio when the tip clearance was increased.

Gandhi et al. [2002] have evaluated performance characteristics of a centrifugal slurry pump at different rotational speeds with water as well as solid-liquid mixture. They found that the affinity relations applicable to conventional pumps for head and capacity can be applied to slurry pumps handling water and slurries at low concentrations (<20% by weight). For higher solids concentrations, these relationships needed to be corrected by taking into account the effect of solids

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 -ε (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.

Kato, C. et al. [2003] have observed boundary interface between impeller and volute casing by using overset grids from dual frames of reference. The overall grid was composed of several grid sets, and appropriate transactions take place at the interface regions. Large-eddy simulation was applied to the prediction of internal flows in a high–specific-speed, mixed-flow pump stage that possesses weak instability in its head-flow characteristics at low flow-rate ratios. The head-flow characteristics were also developed, although the large eddy simulation predicted the stall point at a lower flow-rate ratio than the measurements flow rate. The phase-averaged distributions of the meridional- and tangential-velocity components were also compared with those measured by an LDV.

Nursen, E. C. et al. [2003] have developed an incompressible flow solver for the pump volute. The developed flow solver provides detailed pressure and velocity distribution information inside the volute, and the calculated results were verified by means of the experimental results.

Kadambi et al. [2004]) have used Particle Image Velocitimetry to investigate the velocities of the slurry in the impeller of a centrifugal slurry pump for sodium-iodide solution (NaI) and 500micron glass beads slurry. The experiments conducted at 725 rpm, 1000rpm speed, and1%, 2%, 3% volumetric concentration. They observed that the in clear fluid flow conditions for both the pump rpm, flow separation takes place on the suction side of the blade in the region below the blade tip. For the same flow conditions, the flow moves smoothly along the suction side of the blade depicting a recirculation zone. The intensity of this recirculation zone decreases at the higher concentration of 3% due to particle inertia effects. On the pressure side of the blade the particles are pushed along the blade surface and can result in the frictional wear.

Hergt, P. et al. [2004] have observed the unsteady velocity, pressure and flow angle at the impeller outlet of a centrifugal pump with and without volute casing at five operating points using the hotwire technology and a fast response single hole cylindrical probe. The test fluid was air. While the velocities and pressures depend only on the axial coordinate and were rotationally symmetrical. If there was no casing around the impeller, the influence of the volute on the circumferential distribution of these quantities increases with the deviation of the operating point from the design point.

Xu, C. et al. [2005] have calculated the detailed flow structures in the volute and the compressor performance was calculated for different tongues. It was observed that the flow in the volute sections has a single vortex structure, as opposed to two counter-rotating vortices. The round tongue creates significant blockage near the tongue. This blockage forces secondary flow centre away from the tongue area. It was shown that the round tongue produces better performance than the sharp tongue. The flow simulation was observed to better understand the volute flow mechanisms and provide design guidance in volute design to meet performance goals.

Addie et al. [2007] havedeveloped ANSI/HI standard of centrifugal slurry pump. They studied the effect of slurry on pump performance; net positive suction head required and wear by using the ANSI/HI standard.

Pullum et al [2007] have calculated the performance reduction of the centrifugal slurry pump by using Hydraulic Institute method for handling non-Newtonian coarse particle suspensions. Suspensions up to 38% v/v of coarse particles with mean diameters in the range of 1.1 < d50 < 3.4 mm suspended in carrier fluids with dynamic yield stresses of $0 < \tau_\gamma < 17.2$ Pa and shear thinning indices in the range $0.35 < n < 0.79$ were examined. They found that the reduction in the head is the function of coarse solid concentration.

Min-Guan, Y. et al.[2007] have observed the phenomena of two-phase flow with salt crystallizing in the chemical pump, the 3-D turbulent flow in the impeller of chemical pump was simulated at the condition of rinsing. The internal flow between the impellers of chemical pump was investigated. Based on the Reynolds-averaging N-S equations and the standard *k*-ε two equations turbulent model, the simulations of turbulent flow between the impellers were performed using the flow computing software Fluent under different operating conditions. Based on the analysis of the calculated results of velocity and pressure profiles in the chemical pump and experimentally observed phenomenon of flow impact, secondary flow and recirculation, some design improvements were proposed, which give suggestions on the optimal design and internal two-phase flow study of the chemical pump.

Sellgrenet al.(2007) suggested that the head loss depends on the basic mineral structure and chemical composition of the solid particles in addition to their size, shape and distribution.

CHAPTER 3

EXPERIMENTAL PERFORMANCE OF CENTRIFUGAL SLURRY PUMP

The line diagram of experimental set-up of centrifugal slurry pump used for performance evaluation is shown in Figure 3.1.The setupdetails of “50M WILFLY” centrifugal slurry pump shown in Figure 3.3. Slurry is prepared for experiments in hopper shaped mixing tank having a stirrer arrangement which keepsthe slurry mixture homogeneous during experimentation. The height of the mixing tank is1.55 m and has square cross shape at top (1m x1m).

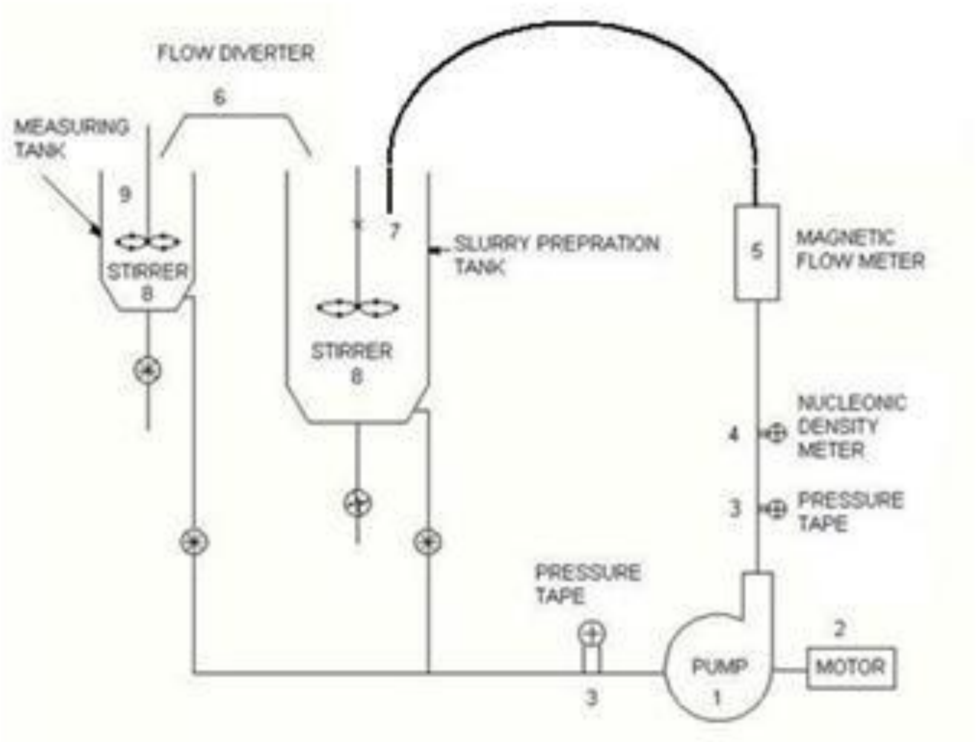


Figure 3.1 Line Diagram of Experimentation setup

In the view of importance of slurry transportation through centrifugal slurry pump, aim of the present work is to evaluate performance of pump handling clear water and bottom ash slurry experimentally in the pilot plant test rig at IIT Roorkee. The photographic view of the experimental setup is shown in fig in Figure 3.2.

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

The return pipe which brings flow back to the mixing tank was earlier consisting two bends and length of the pipe was much more almost touching the roof of the room. Due to these bends many losses were occurring. This results in decrease in head of the pump and the flow rate. The solution to this problem was the removal of bends and decrease in length of the pipe. In place of the bend the pipe was given curved trajectory. The losses were effectively decreased and more accurate readings of the pump was obtained.

3.1 COMPONENTS IN CENTRIFUGAL SLURRY PUMP SETUP

3.1.1 Centrifugal slurry pump: 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 loop has been chosen as 4 m/s well above the settling velocity.

The pump used in the experimental setup for the performance evaluation handling ash and water mixture is “50M WILFLY” centrifugal slurry pump. The pump consists of impellor having 5 blades and volute casing. The manufacturers data of the “50M WILFLY” centrifugal slurry pump is shown in table 3.1:

Figure 3.3Centrfugal slurry pump

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

3.1.2 Motor: 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.

3.1.3 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 m^3
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

3.1.4 VFD (Variable Frequency Drive): 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.

3.1.5 Electro-magnetic Flow Meter: For continuous monitoring of flow rate pre-calibrated electro-magnetic flow meter is installed as shown in the figure 3.4.

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

1. Pipe diameter : 50 mm NB
2. Range of discharge rate : minimum 0-15 lps
3. Accuracy : ± 0.5%
4. Application : abrasive slurry flow measurement
5. Maximum operating pressure : 5 bar
6. Maximum liquid temperature : 60· C
7. Max. Specific gravity of fluid : 2.2
8. Display : 3 1/2 digit digital display for discharge and velocity
9. Repeatability : Teflon or Ni- Chrome coated to work in abrasive slurry environments.

Figure 3.4 Electro-magnetic Flow Meter

3.1.6 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;

Detector	:	Scintillation counter, stainless steel
Nominal pipe diameter	:	50 mm
Power supply	:	230/115 V AC
		18 to 32 V DC
Operation temperature	:	0 to + 50°C
Display	:	LCD-display

3.1.7 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;

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

Figure 3.5 Pressure transducers

3.2EXPERIMENTAL PERFORANCE OF PUMP

Before the experimentation, all the plug valves were first closed, and the mixing tank, separation chambers, suction and delivery measurement tubes were properly filled up with water and all the air bubbles were purged off. The evolution of the pump performance, first suction plug valve connected to the mixing tank was opened fully and the pump was run by starting motor through VFD. At desired flow condition the delivery plug valve was opened and adjusted. The measurements data noted by different instruments namely pressure transducer, tachometer, magnetic flow meter, wattmeter, voltmeter, and ammeter. This procedure was repeated for various openings of delivery valve to cover the entire operating range of the pump at the different speeds. The speed of pump was varied by variable frequency drive (VFD). Also the input power to electric motor was measured by a power analyzer.

Calculation of Head

Total head developed by the pump is calculated from the suction and delivery head values obtained after applying due corrections for velocity head and potential head. The pump input power is measured to calculate the motor efficiency. The calculation procedure is given below.

1. Calculation of head (H_{total})

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

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)

$(\frac{P_d}{\rho g})$= Delivery pressure gauge reading delivery side (m)

$(\frac{P_s}{\rho g})$= 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:

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

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

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

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 Cw =solid concentration, % by weight

ρ_s = mass density of solid material, kg/m^3

ρ_l = mass density of liquid, kg/m^3

ρ_m= mass density of solid- liquid mixture , kg/m$^{3\backslash}$

4. Power output

$$P_{out} = \rho g Q H$$

Where H =Total head (m)

ρ = mass density of water, kg/m^3

g= gravitational acceleration m/sec^2

Q = flow rate, m^3/s

P_{out}= Output power,kW

5. Efficiency of pump

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

P_{in}- Input power of pump

P_{in}=T×ώ

Where T=Torque of pump shaft,N-m

ώ = 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.

3.3EXPERIMENTAL PERFORMANCE OF CENTRIFUGAL SLURRY PUMP WITH WATER

The performance of the pumps was determined with clear water at the speeds of 1250 rpm and 1400rpm which is tabuled in table 3.2 and 3.3.

Table 3.2Experimental results of Performance of centrifugal pump with water at 1250 rpm

S.No.	Q in lps	Speed in rpm	Actual head at Observed speed in m of H2o	Q at 1000 rpm in lps	Rated head at rated speed in 1000 rpm	Specific discharge	Specific head at 1000 rpm
1	14.92	1250	10.19354	11.936	6.523866	39.01967	3.311243
2	11.886	1250	11.52973	9.5088	7.379025	31.08504	3.745301
3	8.811	1250	12.42256	7.0488	7.950438	23.0431	4.035328
4	6.668	1250	12.70502	5.3344	8.131211	17.43858	4.127081
5	4.235	1250	12.88873	3.388	8.248788	11.07565	4.186758
6	2.785	1250	12.92963	2.228	8.274963	7.283512	4.200044
7	1.33	1250	13.6357	1.064	8.726845	3.478302	4.429401
8	0	1250	13.60475	0	8.70704	0	4.419349

Table 3.3 Experimental results of Performance of centrifugal pump with water at 1400 rpm

S.No.	Q in lps	Speed in rpm	Actual head at Observed speed in m of H2o	Q at 1000 rpm in lps	Rated head at rated speed in 1000 rpm	Specific discharge	Specific head at 1000 rpm
1	16.669	1400	12.80453	11.90643	6.532924	38.923	3.31584
2	13.623	1400	14.53861	9.730714	7.417657	31.81049	3.76491
3	10.277	1400	15.6695	7.340714	7.994645	23.99739	4.057765
4	8.301	1400	15.90306	5.929286	8.113807	19.38331	4.118248
5	6.435	1400	16.17015	4.596429	8.250078	15.0261	4.187414
6	4.822	1400	16.2875	3.444286	8.309947	11.25965	4.2178
7	3.415	1400	16.40707	2.439286	8.370952	7.974222	4.248764
8	0	1400	17.22345	0	8.787476	0	4.460175

Figure 3.6 shows that maximum value of actual head developed by pump is 16.2875 (m) 1400 rpm and at 4.822 litre/sec of discharge. It shows that with increase in discharge, head is decreased. The pump efficiency characteristic with bottom ash is shown in figure 3.9. It shows that all the data points lie on a smooth curve and the maximum efficiency of 34.372 % is measured. Figure 3.8 shows that the pump input power increases steadily with increase in discharge rate and its maximum value is 6.092 kW at 16.669 litre/sec discharge.

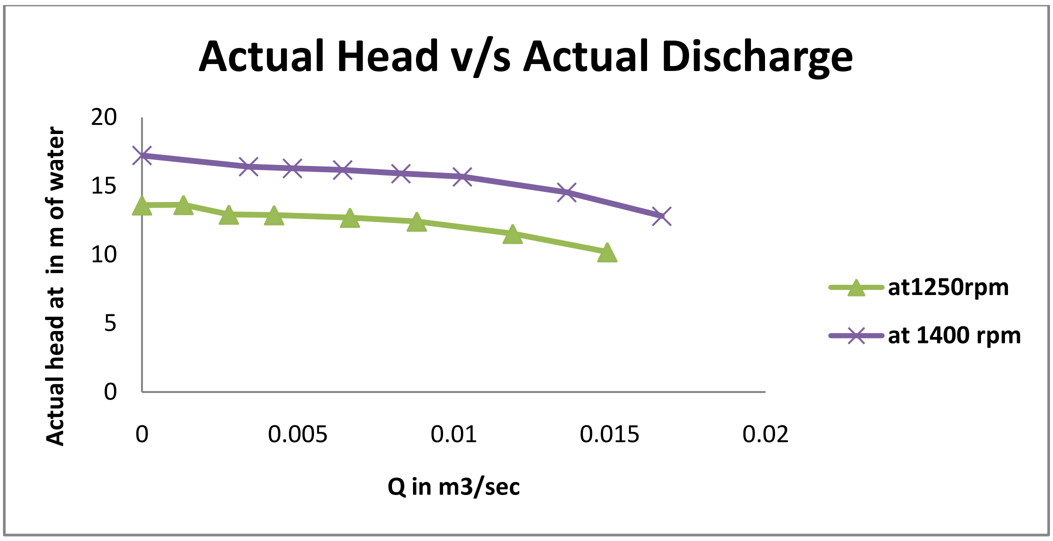


Figure 3.6 Comparison of actual head and actual discharge

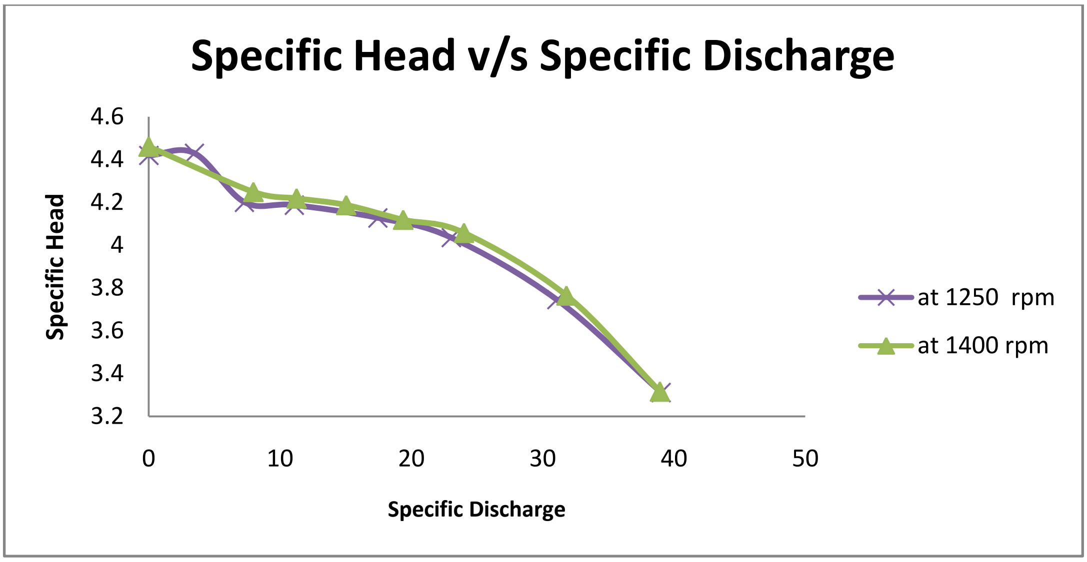


Figure 3.7 Comparison of specific head with specific discharge

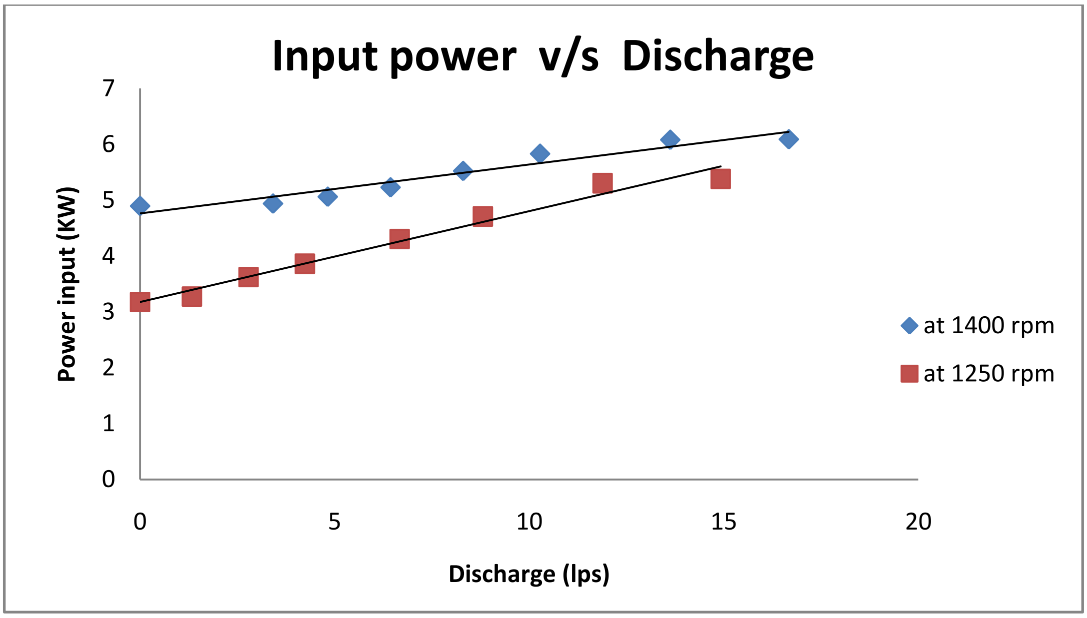


Figure 3.8 Input power-Discharge characteristics of centrifugal slurry pump

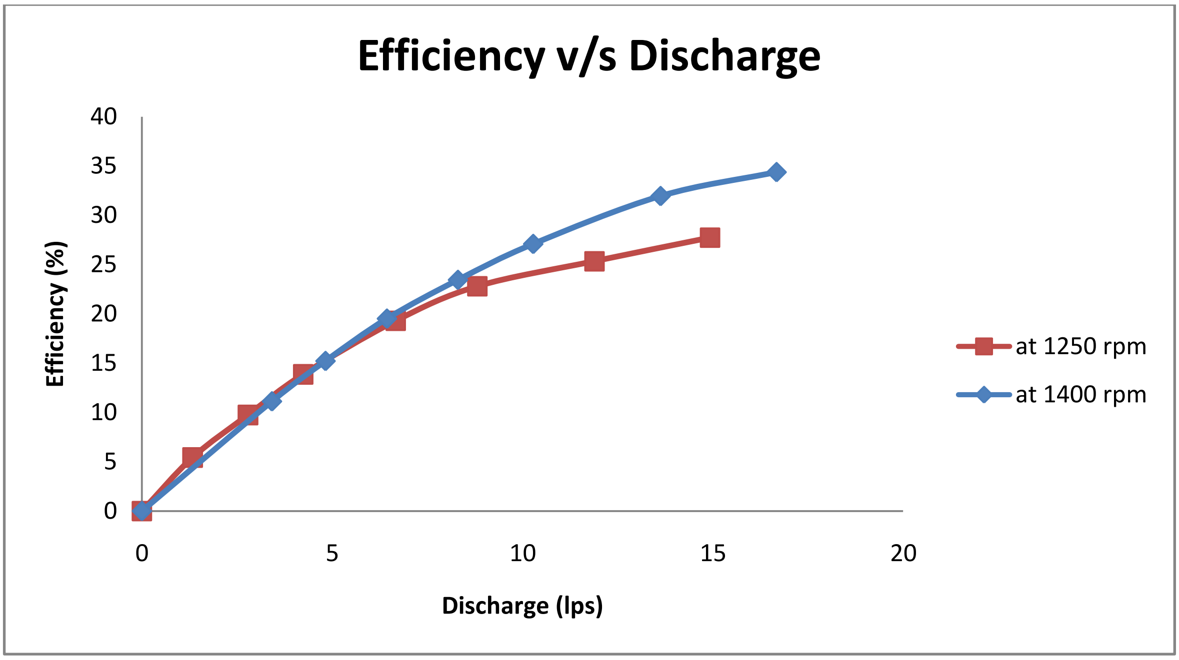


Figure 3.9 Efficiency-Discharge characteristics of centrifugal slurry pump

CHAPTER 4

RHEOLOGICAL BEHAVIOUR OF BOTTOM ASH

4.1 BOTTOM ASH

Bottom ash is agglomerated ash particles, formed in pulverized coal furnaces that are too large to be carried in the flue gases and impinge on the furnace walls or fall through open grates to an ash hopper at the bottom of the furnace. Physically, bottom ash is typically grey to black in color, is quite angular, and has a porous surface structure.

Bottom ash applications include its use as a:

- Filler material for structural applications and embankments
- Aggregate in road bases, sub-bases, and pavement
- Feed stock in the production of cement
- Aggregate in lightweight concrete products
- Snow and ice traction control material

4.2 PROPERTIES OF BOTTOM ASH

4.2.1 Physical properties

Bottom ash contains angular particles. They have very porous surface texture. Bottom ash particles range in size from a fine gravel to a fine sand with very low percentages of silt-clay sized particles. Bottom ash is predominantly sand-sized, usually with 90 to 100 percent passing below 2000 micron sieve size, 90 to 80 percent passing a below 1400 micron sieve size, 30-20 percent passing below 75 micron sieve size.

4.2.2 Chemical properties

Bottom ash from boiler slag is composed principally of silica, alumina, and iron, with smaller percentages of calcium, magnesium, sulfates, and other compounds. The composition of the bottom ash or boiler slag particles is controlled primarily by the source of the coal and not by the type of furnace.

Figure 4.1 Bottom ash

4.3BENCH SCALE TESTS

Standard methods are used in the laboratory to determine the physical properties of solids, carrier fluid and slurry. A brief description of these tests is presented here:

4.3.1 Particle Size Distribution (PSD)

The changes in the particle size in the solid sample and the percentage of particles present in different pre-selected size ranges are determined to establish the particle size distribution (PSD). Two methods are employed to get this distribution namely. One is Sieve analysis (for coarse particles i.e. > 75mm) and other is Hydrometer analysis (for fine particles i.e. < 75mm).Asamplewith known weight of solid particles is taken and washed over. The material retained over the sieve as well as the finer particulate material are dried in an oven. The dried coarser material is sieved through a set of standard sieves. Special care is taken to ensure that the sample is properly dried. The sample retained on each sieve is collected and the percentage retained on each sieve is calculated using the standard procedure. The particle size distribution of the fine particles collected (<75 mm) is then determined using the standard hydrometer analysis. The experimental result of Bottom Ash is shown in table 4.1.

Table 4.1 particle size distribution of bottom ash

S.No	Size in microns	Wieght	% Finer Bottom
1	1005	--	100%
2	710	1.7	99.15%
3	500	21.7	88.30%
4	355	20.85	77.86%
5	250	31.9	61.93%
6	150	48.57	37.64%
7	103	42.1	16.59%
8	50	31.27	0.96%
9	Below 50	1.65	0.13%

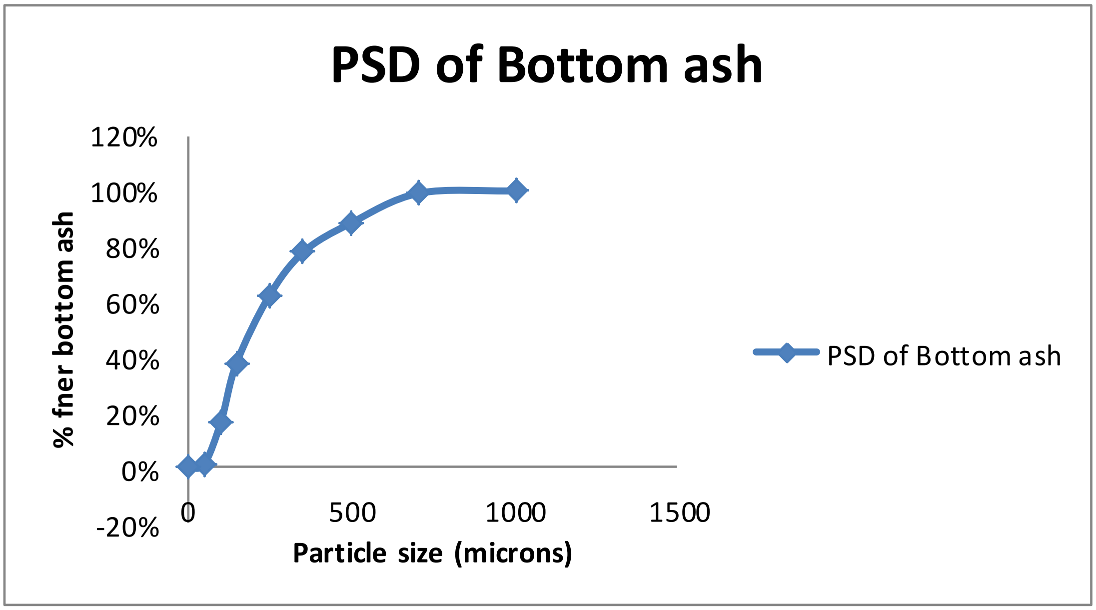


Figure 4.2 Particle Size distribution of bottom ash

4.3.2 P^{H} VALUE

A pH meter was used for measurement of the pH value of the slurry of any given solid concentration. The electrode of the meter was first moistened with tap water and then calibrated with a buffer solution of a known pH value. It is cleaned by rinsing vigorously with distilled water and then immersed in the slurry sample whose pH value was to be determined. The pH suspension was read on the digital display unit when equilibrium value was reached. The experimental results of ph value of bottom ash are shown in table 4.2.

In this test first prepare 7 PH or 4 PH solution dissolve 1 buffer tablet in 100 ml of distilled water. Insert the probe PH meter in the solution and wait for 5 minutes till readings get stabilized. If reading is not equivalent to the PH of solution, then change the control and make the readings equivalent to the PH value of solution. Remove the probe and dip into the distilled water beaker and then dip into the solution or slurry of which PH value is to be determined. Wait for 5 minutes till the reading is stabilized.

Table 4.2 pH value of bottom ash at different concentration

Cw, %	0	20	25	30	35	40	45	50
pH	7.72	7.64	7.62	7.62	7.61	7.6	7.58	7.58

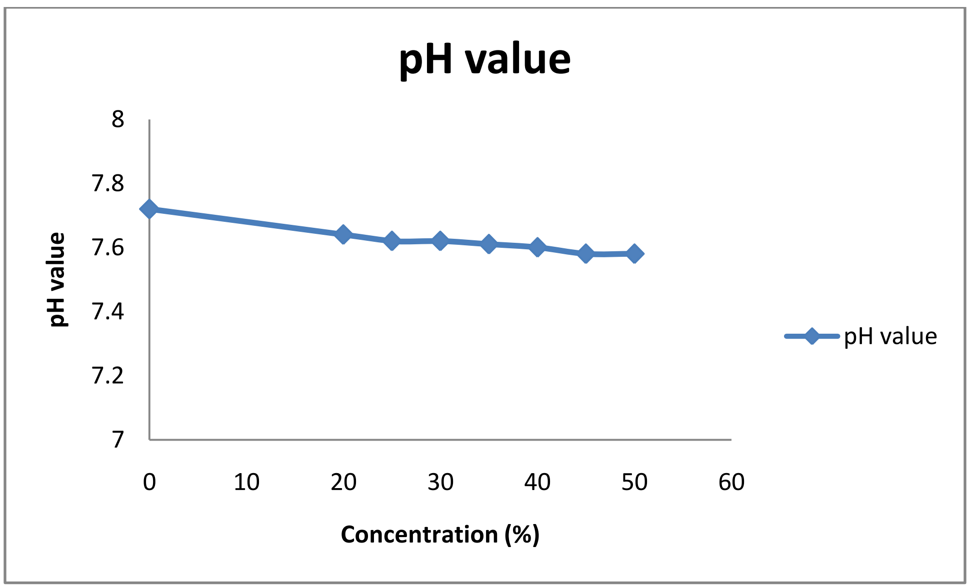


Figure 4.3 pH value of bottom ash

4.3.3 Static Settled Concentration

The static settled concentration is an important parameter which decides the performance of the pump at different levels of slurry concentration. Static settling predicts the behavior of the ash with water or transporting fluid. It is the procedure to test the time and speed at which the ash particles settle themselves in the containing fluid.The static settled concentration depends on parameters like specific gravity, shape and particle size distribution, density and viscosity, of carrier fluid etc. It is well accepted that the optimum concentration for solids transportation is around 5 to 10% lower than the static settled value.

In the present study, the static settled concentration has been determined by preparing a slurry sample of intermediate concentration i.e. 30% (by weight) and allowing it to settle in a graduated measuring beaker till the level of the particles of ash become constant. This value of solid concentration in the settled portion of slurry is the static settled concentration. The readings of slurry levels at taken at regular intervals of time to determine the settling rate of the slurry.

In this test take 150 gm of Bottom ash in jar and put 350 gm water in jar to make the 30% concentration by weight. Then keep the mixture unshaken for 2 hours. After 2 hours, shake the jar well enough to mix the ash properly with the carrier fluid. Reading levels are taken every 10 seconds in the initial stage. Note the reading till you get steady state. The computations for settled concentration of ash with specific gravity and the volume of the concentration are done. Then concentration v/s time graphs are generated to get static concentration.The experimental results of Static Settled Concentration of bottom ash is shown below.

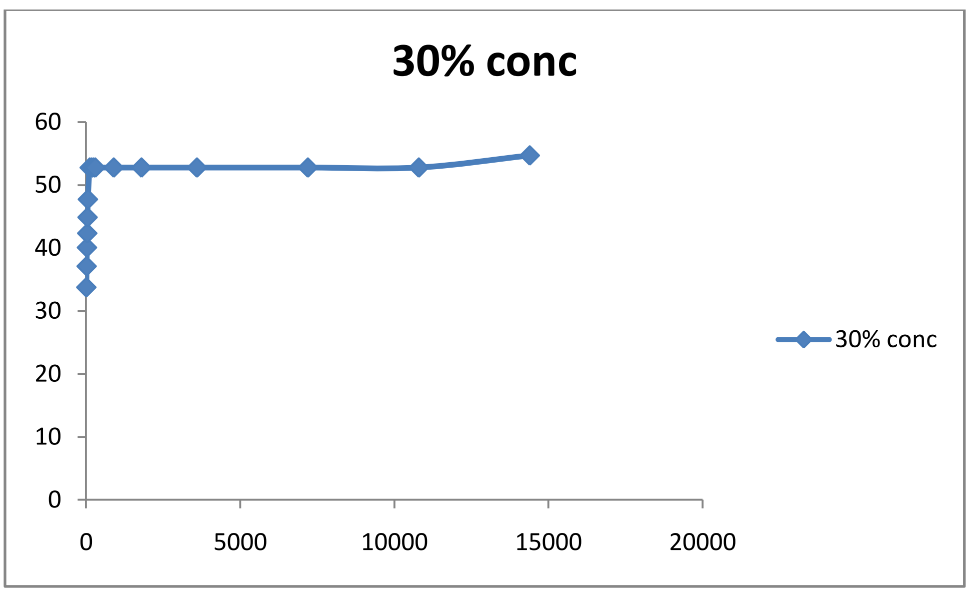


Figure 4.4 Static Settling Characteristics of Bottom Ash

4.3.4 Rheological Behavior of Solid-Liquid Mixture

Rheology is the study of the flow of matter: primarily in the liquid state, but also as 'soft solids' or solids under conditions in which they respond with plastic flow rather than deforming elastically in response to an applied force. It applies to substances which have a complex molecular structure, such as muds, sludges, suspensions, polymers and other glass formers (e.g. silicates), as well as many foods and additives, bodily fluids (e.g. blood) and other biological materials.

To predict the level of input energy in transporting slurry through pumps, viscosity measurement is an important aspect.Presence of solid particles alters the viscosity of carrier fluid. Addition of solids in the carrier fluid beyond a certain proportion results in the mixture depicting Non-Newtonian behavior. Therefore the variation of shear stress with shear-rate has been measured for a wide range of concentrations varying from 10 to 50% (by weight) for bottom ash and water slurries to establish the rheology of the mixtures.

Figure 4.5Setup of Anton PaarRheometer

The Rheometer, (Make: Anton Paar, Gurgoan) shown in figure 4.9, was used to measure the viscosity for the water slurry mixture. The mixture was poured in double gap geometry and the shear stress was applied by rotating double gap stirrer which is immersed in mixture. The varieable shear rate was given and accordingly the value of shear stress and viscosity was calculated.The experimental results of viscosity of bottom ash are shown in figure 4.7.

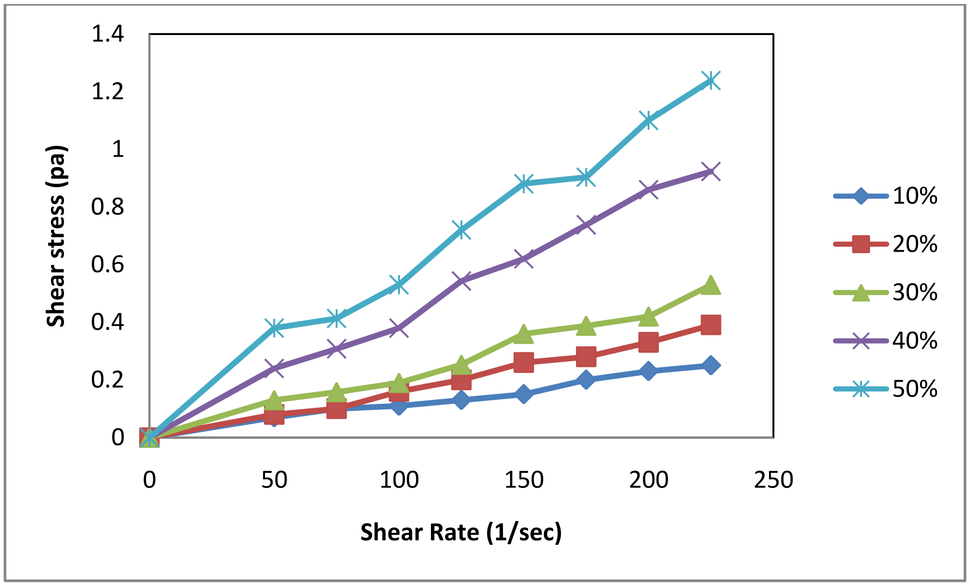


Figure 4.6 Variation of shear stress of bottom ash with shear rete

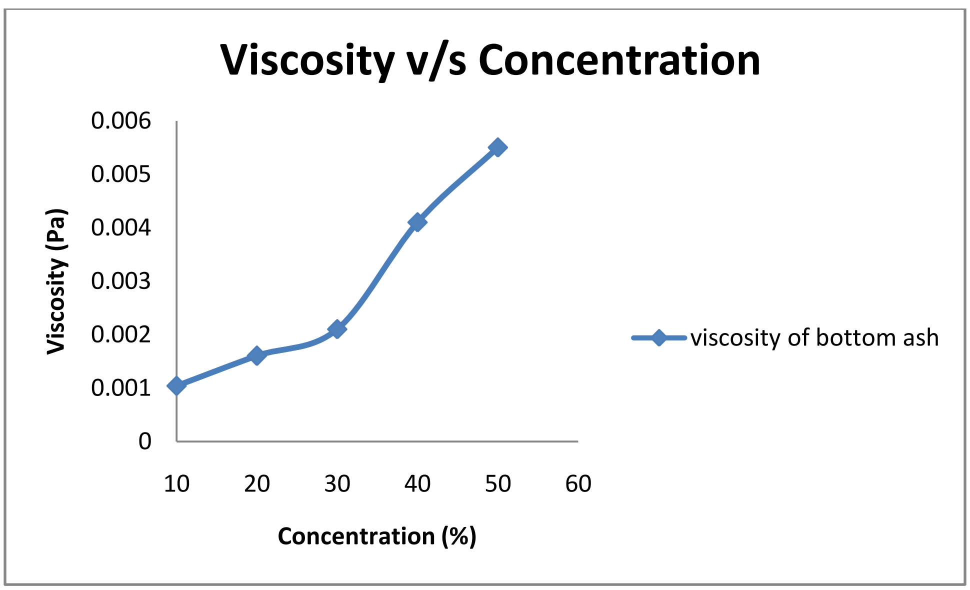


Figure 4.7 Variation of viscosity of bottom ash with concentrations

From the figure 4.6 and 4.7 observed that shear stress is the function of shear rate. Shear stress increase with the increase in shear rate and also observed that the bottom ash upto 30 %shows Newtonian behavior. Beyond this non-newtonian behavior was observed.

CHAPTER 5

MODELLING OF CENTRIFUGAL PUMP

In the recent years, Computational Fluid Dynamics (CFD) has been increasingly used for a wide range of engineering applications. Some examples of interesting applications of computational modeling are cooling of electronics systems, rotating and reciprocating machinery, furnaces and combustion chambers. CFD is finding its way into process, chemical, civil, and environmental engineering. Optimization in these areas can produce large savings in equipment and energy costs and in reduction of environmental pollution.
Computational fluid dynamics (CFD) is one of the branch of Engineering, using numerical methods and algorithms to solve and analyse 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 dynamicsis the art of replacing such PDE systems by a set of algebraic equations which can be solved using digital computers.

The basis of computational fluid dynamics is the reduction of continuum deferential equation describing the dynamics of the fluid (Navier stokes+ Mass & energy conservation equation) into a system of algebraic equation of finite number of “grid” points, and the solving of the equation at these limited no. of points only.

Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of

- Mathematical modeling (partial differential equations)
- Numerical methods (discretization and solution techniques)
- Software tools (solvers, pre and post processing utilities)

Principle of Fluid Motion

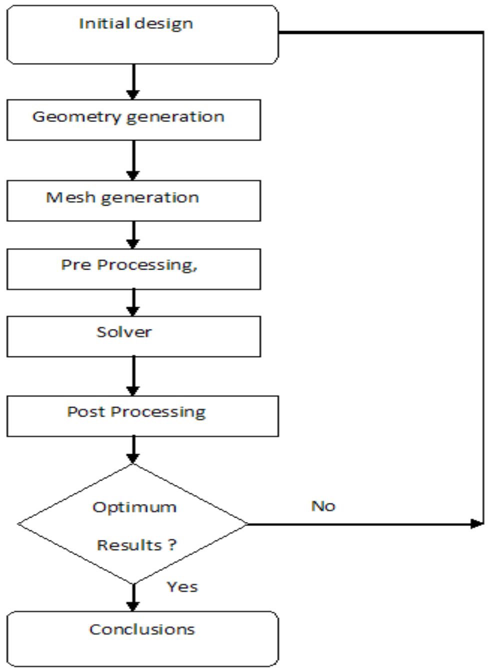
1. Mass Conservation
2. Newton’s Second of Law
3. Energy Conservation
4. Equation of State for Idealized Gas

Experiments vs. Simulations

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

Experiments	Simulations
Quantitative description of flow phenomena using measurements • 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	Quantitative prediction of flow phenomena using CFD software • For all desired quantities • With high resolution in space and time • For all actual domain • For virtually any problem and realistic operating condition
Error sources: measurement errors, flow disturbance by the probes	Error sources: modelling, discretization, iteration, implementation

CFD Methodology



Solver

There are three methods using for discretizing a given differential equation,

- FDM (Finite Difference Method):- Finite difference method utilizes the Taylor series expansion to write the derivatives of a variable as the differences between values of the variable at various points in space or time.
- FEM (Finite Element Method):- In the finite element method, the fluid domain under consideration is divided into finite number of sub-domains, known as elements. A simple function is assumed for the variation of each variable inside each element. The summation of variation of the variable in each element is used to describe the whole flow field.
- FVM (Finite Volume Method):- The finite volume method is currently the most popular method in CFD. The main reason is that it can resolve some of the difficulties that the other two methods have. Generally, the finite volume method is a special case of finite element.

All CFD codes contain three main elements:

1. A **pre-processor**, which is used to input the problem geometry, generates the grid; define the flow parameter and the boundary conditions to the code.
2. A **flow solver**, which is used to solve the governing equations of the flow subject to the conditions provided. There are three different methods used as a flow solver:
   - Finite difference method
   - Finite element method
   - Finite volume method
3. A **post-processor**, which is used to massage the data and show the results in graphical and easy to read format.

5.1 GOVERNING EQUATIONS OF CFD

The physical aspects of any fluid flow are governed by the following three fundamental Principles:

a) Conservation of mass

b) Conservation of momentum (Newton’s second law)

c) Conservation of energy (first law of thermodynamics)

Conservation of Mass equation

The equation for conservation of mass also called the continuity equation can be written as follows:

$$\frac{\partial \rho}{\partial t} . \nabla(\rho \vec{v}) = S_m \qquad (5.1)$$

The equation is the general form of the mass conservation equation and is valid for incompressible as well as compressible flows. The source S_m is the mass added the continuous phase from the dispersed second phase (e.g., due to vaporization of quid droplets) and any user-defined sources.

For steady state compressible fluid flow the continuity equation is given by:

$$\rho(\nabla \vec{v}) = 0$$

Where, $\nabla = \frac{\partial}{\partial x_i}\hat{i} + \frac{\partial}{\partial x_i}\hat{j} + \frac{\partial}{\partial x_i}\hat{k}$ and $V = u_i\,\hat{i} + u_j\,\hat{j} + u_k\,\hat{k}$

Momentum conservation equation

Conservation of momentum in an inertial (non-accelerating) reference frame is describe as:

$$\frac{\partial \rho}{\partial t}(\rho \vec{v}) + \nabla.(\rho \vec{v}\vec{v}) = -\nabla p + \nabla(\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \qquad (5.2)$$

Where,

$\rho \vec{g}$ = gravitational body force,

F = external body forces (e.g., that arise from interaction with the dispersed phase), respectively. F also contains other model-dependent source terms such as porous media and user-defined sources.

The stress tensor $\bar{\bar{\tau}}$ is given by;

$$\overline{\overline{\tau}} = \mu[(\nabla \vec{v} + \nabla \vec{v}^{T})] = -\frac{2}{3}\nabla.\vec{v} I$$

Where, the second term on the right hand side is taken for consider the effect of volume dilation.

For steady state incompressible fluid flow, the momentum conservation equation is given by

$$\nabla.(\rho \vec{v}\vec{v}) = -\nabla p + \nabla.(\overline{\overline{\tau}}) + \rho \vec{g} + \vec{F}$$

These fundamental principles can be expressed in terms of mathematical equations which in their most general form are usually partial differential equations. CFD is the art of replacing the governing partial differential equations of fluid flow with numbers and advancing these numbers in space and / or time domain to obtain a final description of complete flow field of interest. With the development of high-speed digital computers, CFD has become a powerful tool to predict flow characteristics in varied problem, in an economical way.

5.2MODELING 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:

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.

Figure 5.1 Impellor

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.

Figure 5.2 Casing

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 a 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.

Figure 5.3 follower plate **Figure 5.4 frame**

Figure 5.5 Flange

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.

Figure 5.6 Dismantled pump for dimensioning of inlet passage

5.3 PREPRATION 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 12. 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 5.7, 5.8 and 5.9:

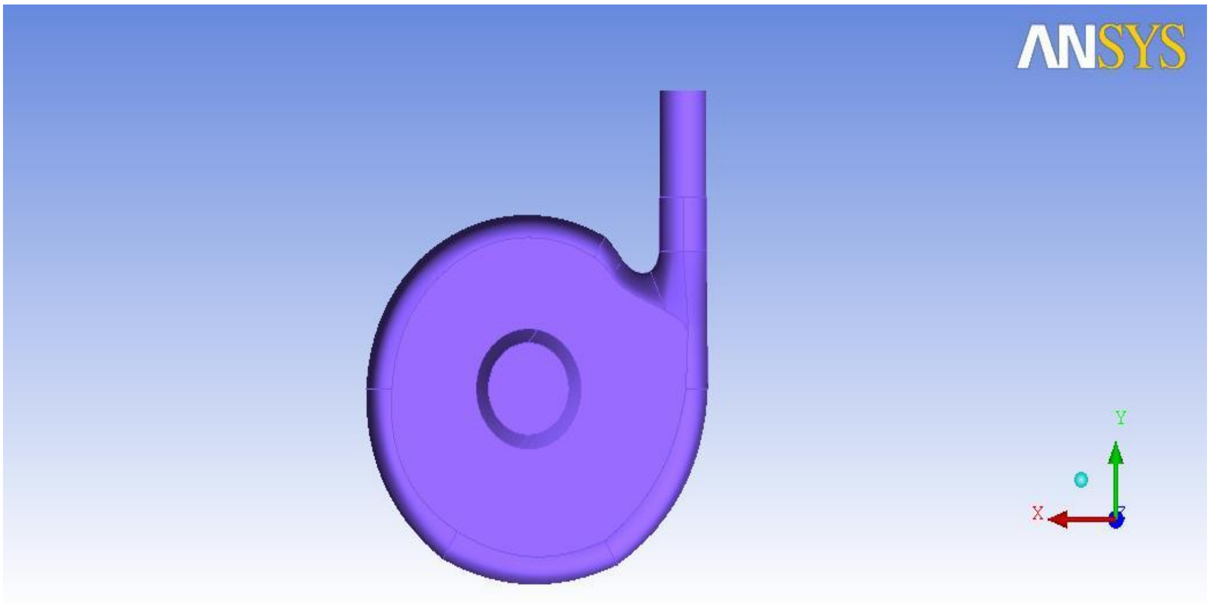


Figure 5.7 CAD model of casing

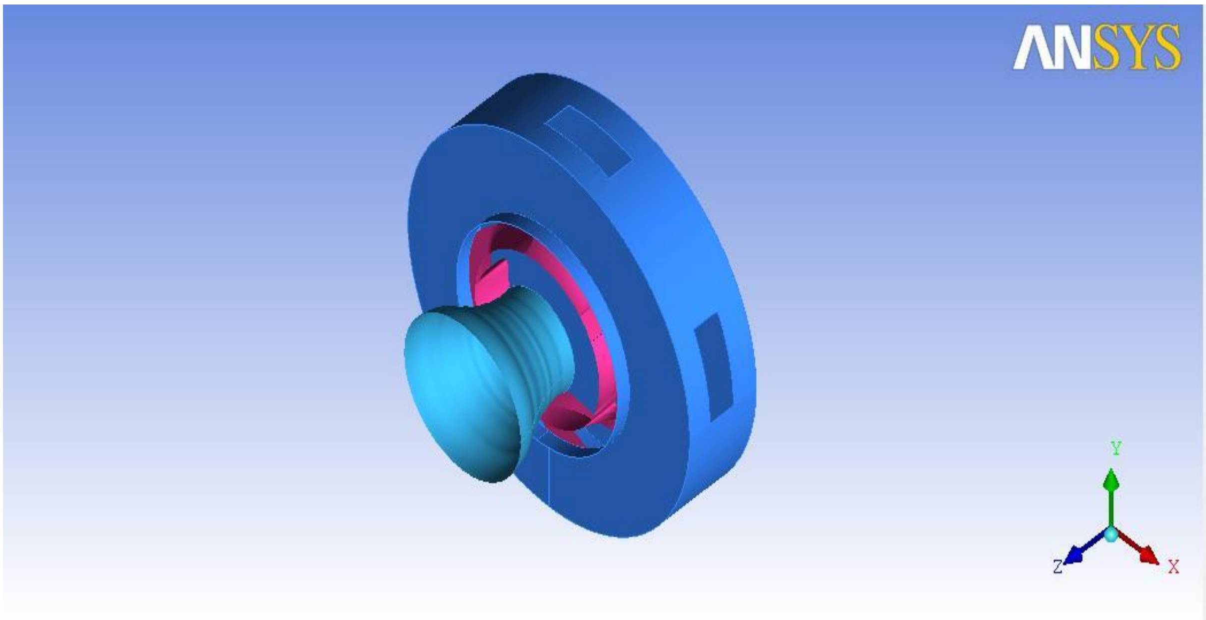


Figure 5.8 CAD model of impellor

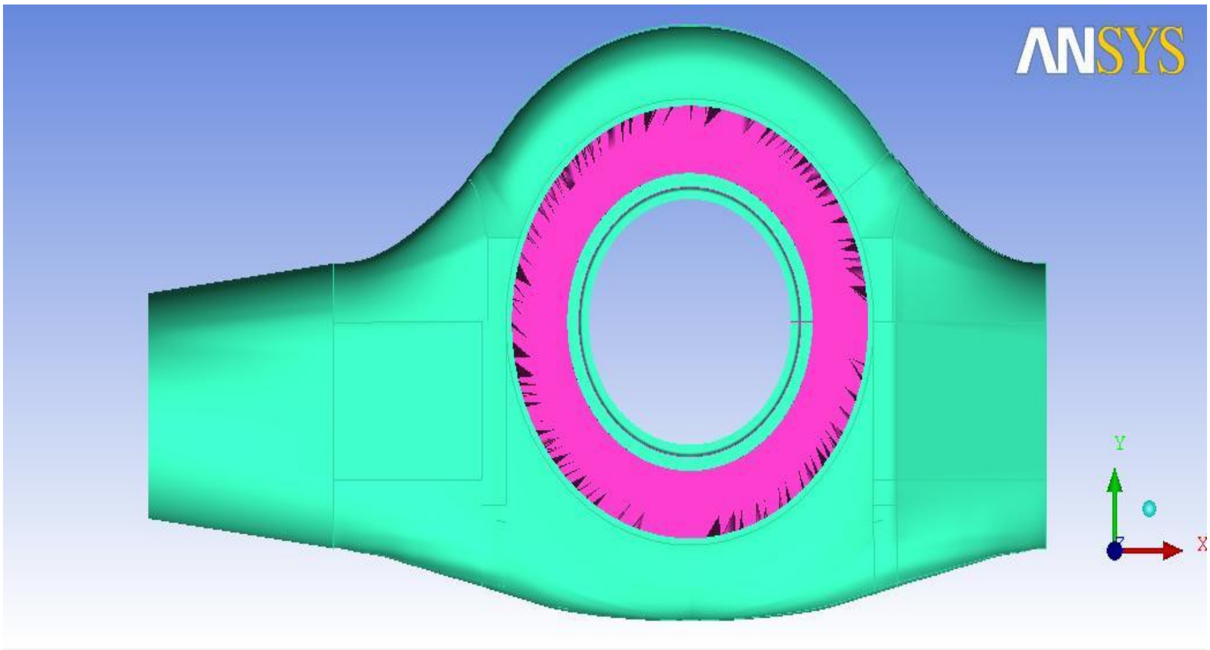


Figure 5.9 CAD model of inlet passage

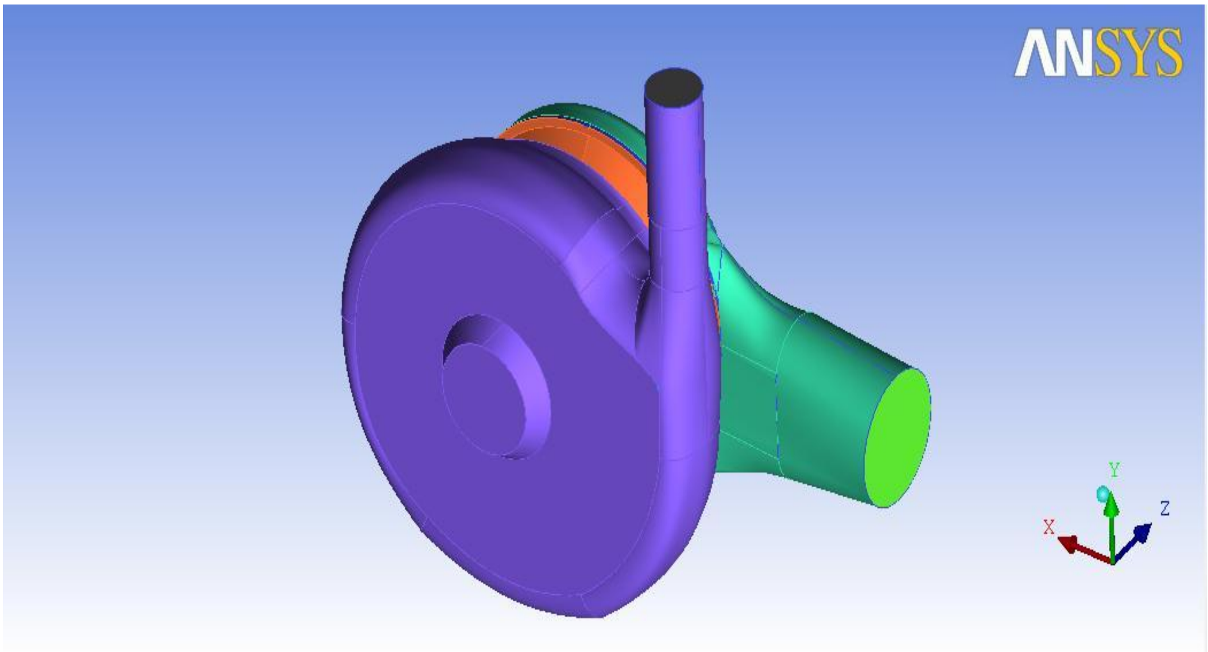


Figure 5.9 The assembled view of centrifugal pump

The first step after modeling the pump components is descretize it into smaller elements. The model is descretized 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 12 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 5.11.

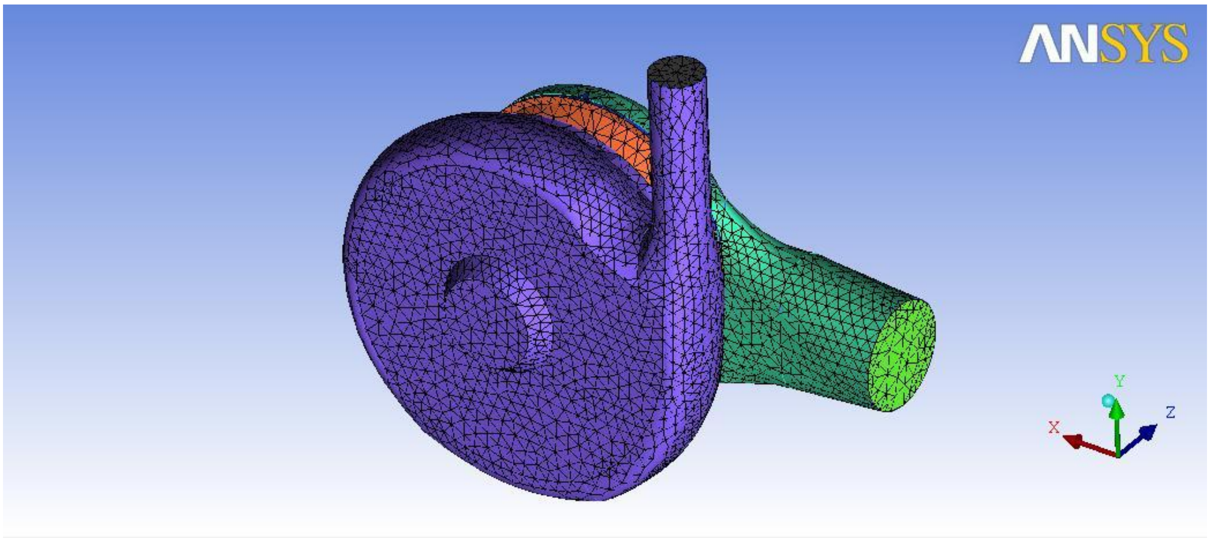


Figure 5.11 Meshed model of centrifugal pump

The information about no. of elements is given in table 5.1

Table 5.1 Number of elements

Part name	No. of elements
Inlet passage	112285
Casing	143056
Impellor	28130

5.4 BOUNDRY 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:

$$Q_{EAS} = \max\left\{\frac{\theta_{\max} - \theta_{eq}}{180 - \theta_{eq}}, \frac{\theta_{eq} - \theta_{\min}}{\theta_{eq}}\right\} \quad (5.3)$$

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, $\theta_{eq} = 60^0$For quadrilateral and $\theta_{eq} = 90^0$ 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_{EAS}) is a measure of skewness that is defined as follows:

$$Q_{EVS} = \frac{(S_{eq} - S)}{S_{eq}} \quad (5.4)$$

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.

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

5.4.1 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\

$$Vfluid = Vwall \qquad (1)$$

• Temperature field

The principle different types of boundary conditions are as follows:

• Dirichlet boundary condition (first type)

• Neumann boundary condition (second type)

• Robbins boundary condition (third type)

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.

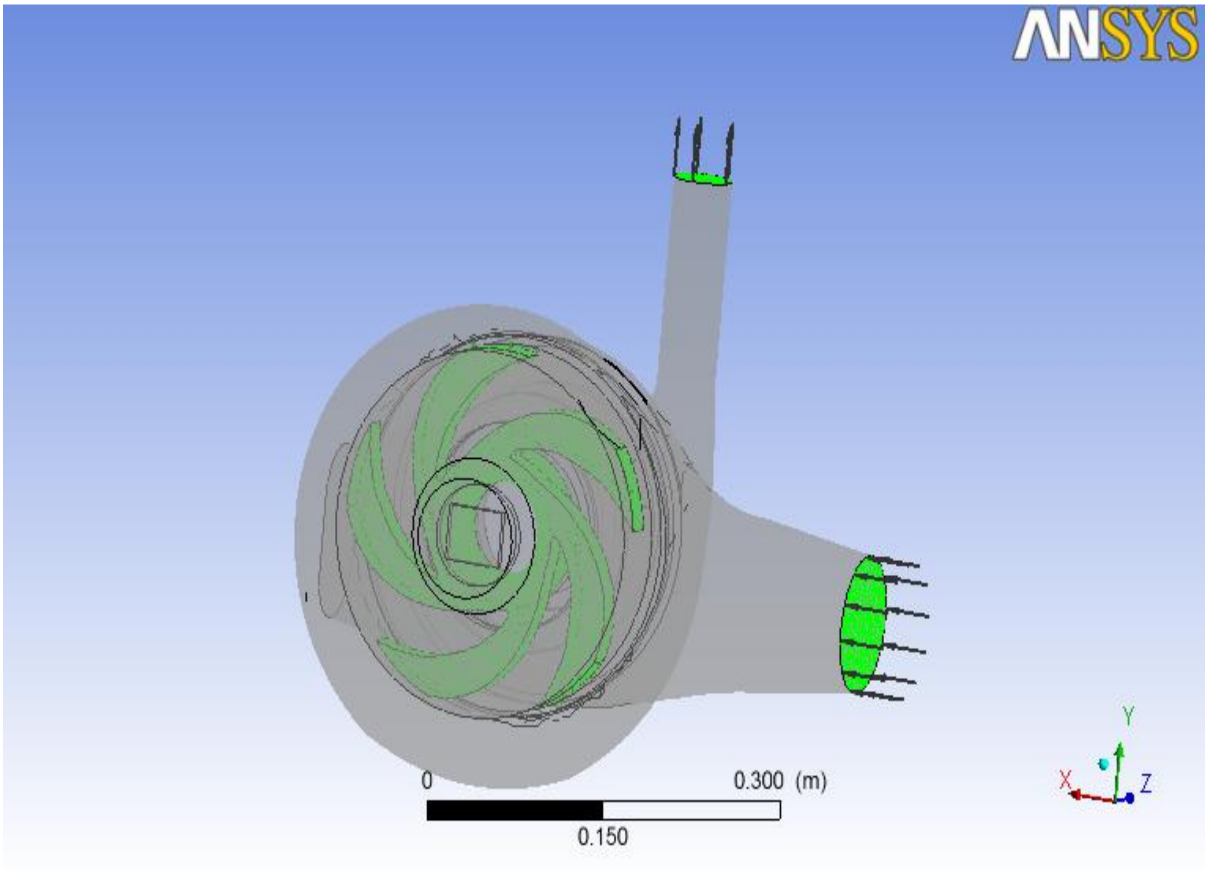


Figure 5.12 Centrifugal Pump with boundary conditions applied

5.5 POST PROCESSING OF CFD RESULT

The numerically simulated flow field of centrifugal slurry pump was analyzed to investigate the performance characteristics of centrifugal slurry pump at 1400rpm, 1250rpm, 1100 rpm and 900 rpm. In this the velocity and pressure contours are developed and gives the information about the maximum velocity and pressure region developed on the impeller. The study of flow field is divided into following two parameters behaviour across different pump sections are velocity vectors and static pressure contours. The velocity streamlines is also shown in fig. below.

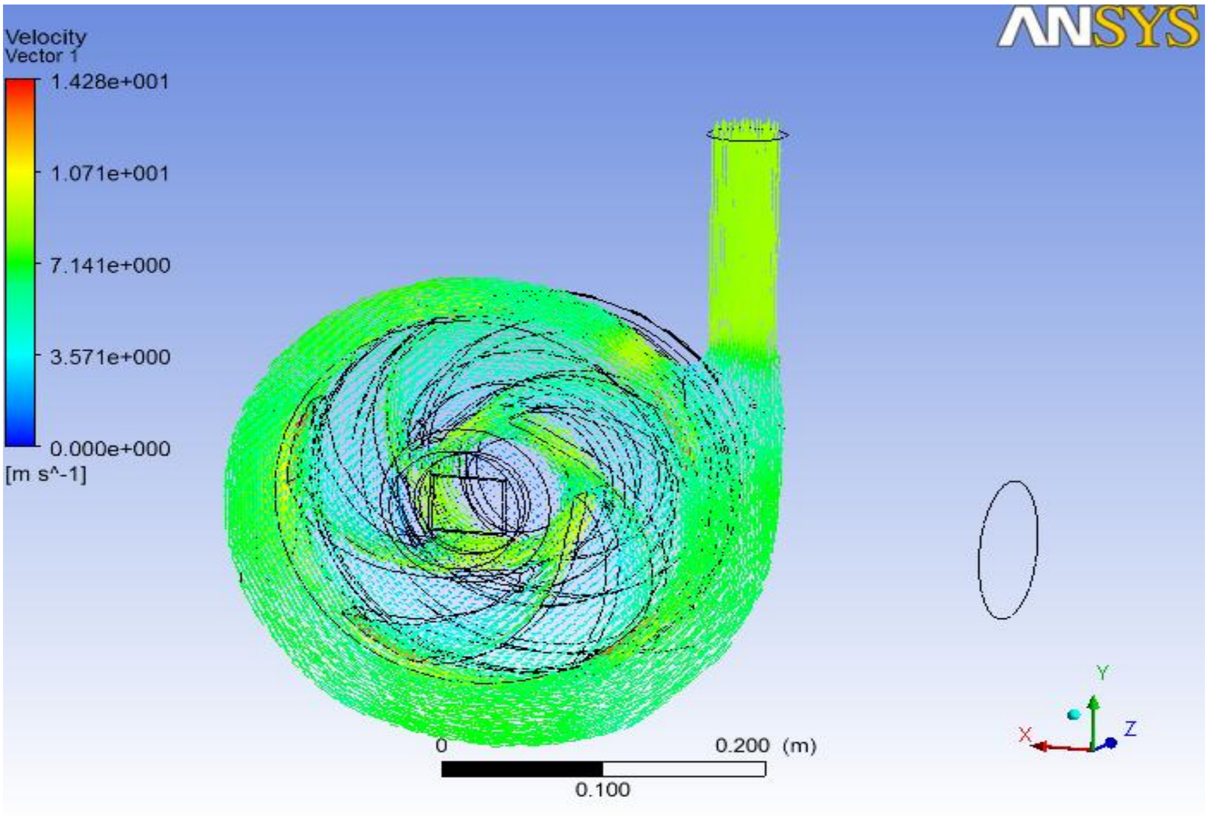


Figure 5.13 Velocity Vectors in casing and impellor of centrifugal pump

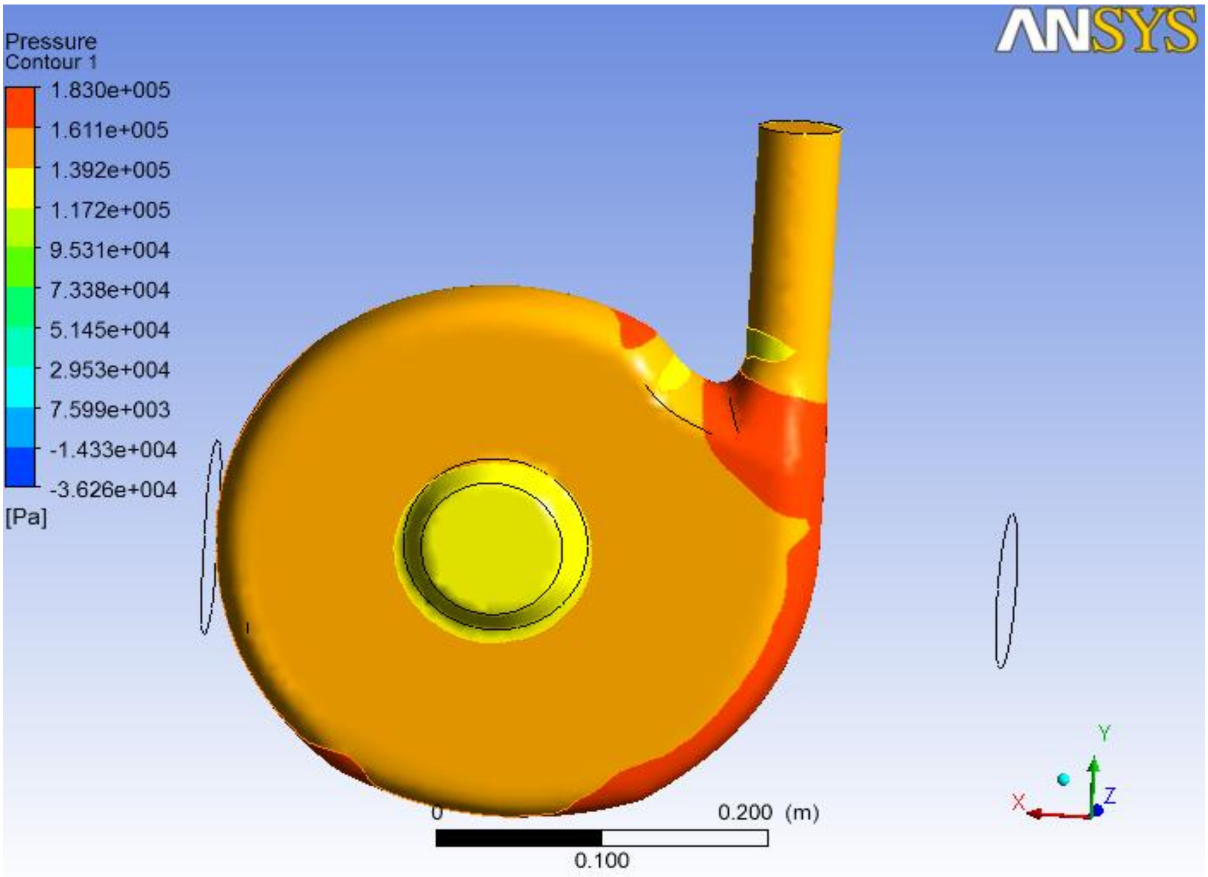


Figure 5.14 Pressure contours in casing of centrifugal pump

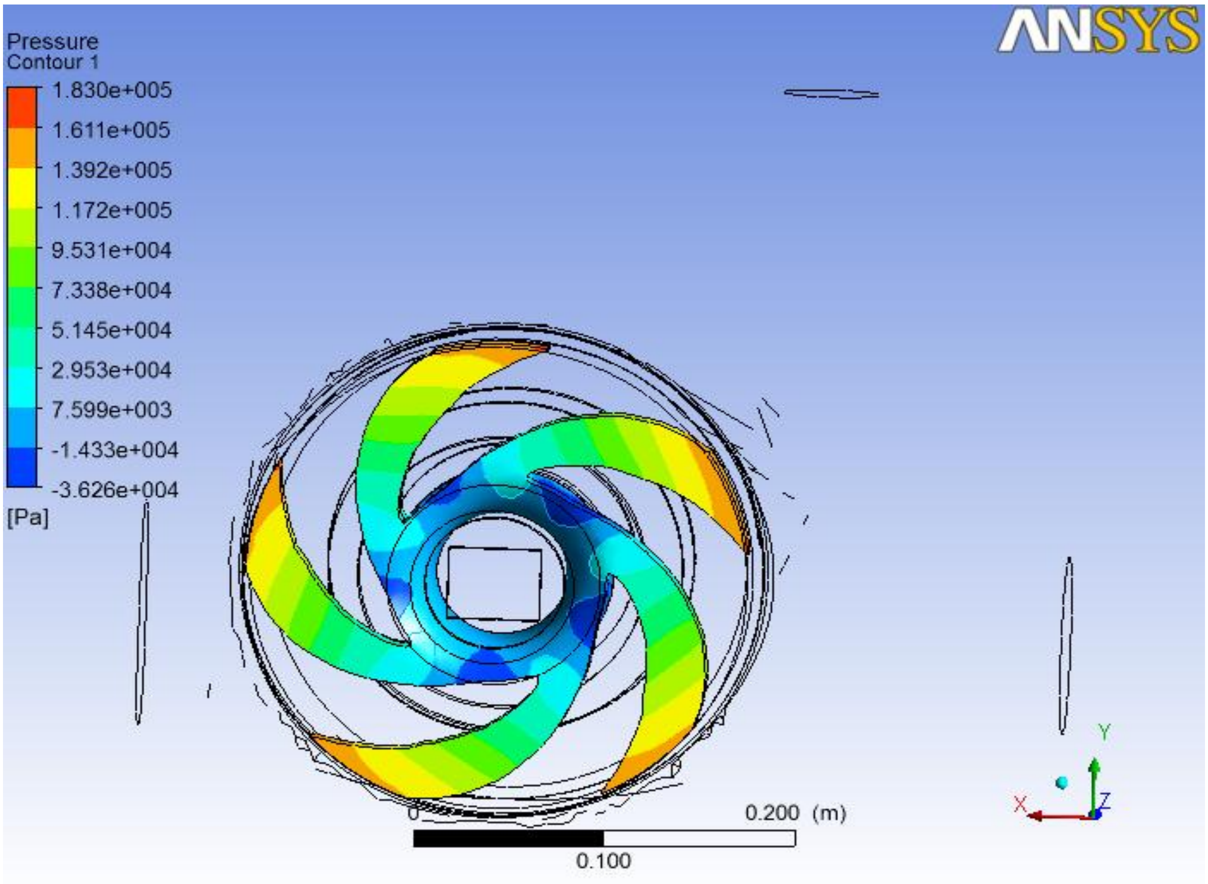


Figure 5.15 Pressure contours and impellor of centrifugal pump

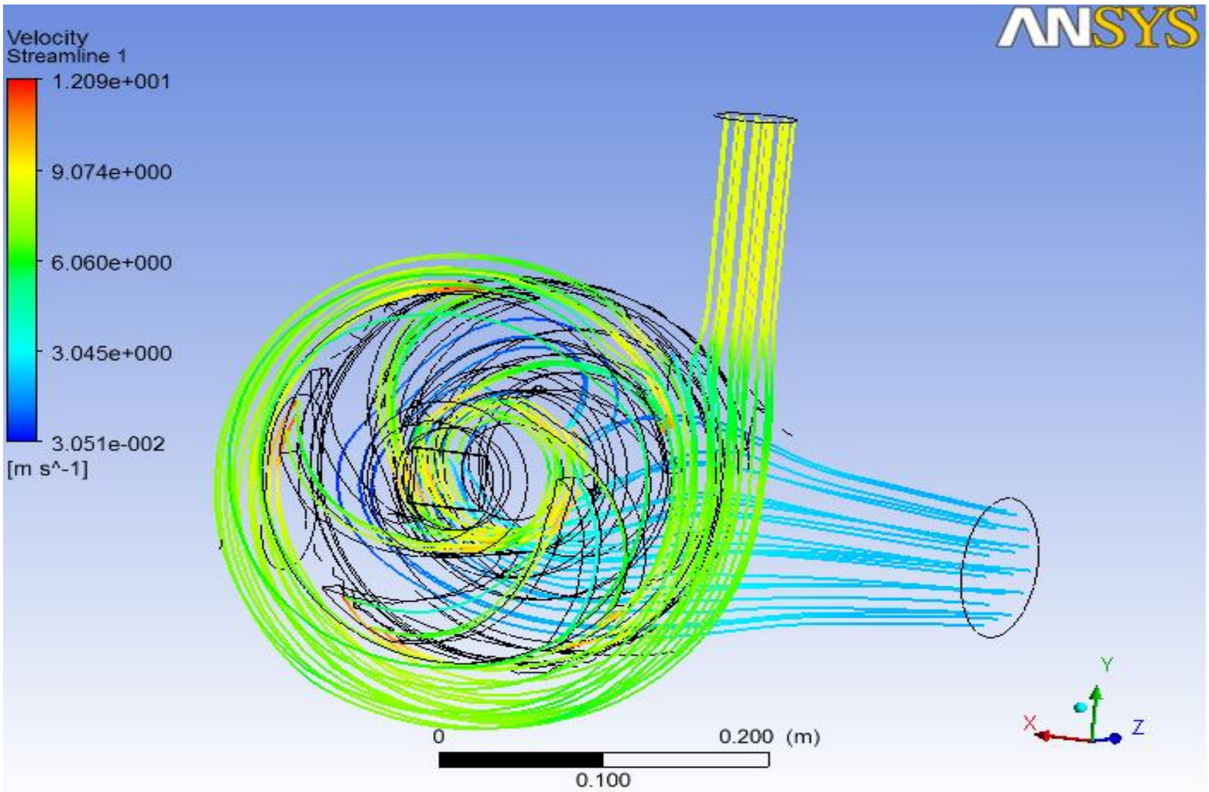


Figure 5.16 Velocity streamlines of centrifugal pump

5.6 PERFORMANCE CHARACTERISTICS OF PUMP

The performance characteristic of the centrifugal slurry pump has been predicted numerically handling water. Head, power and efficiency characteristics of the pump are predicted by CFD analysis at 1400rpm, 1250rpm, 1100 rpm and 900 rpm and explained graphically in Figures 5.17-5.19. Figure 5.18 shows that the pump input power increases steadily with increase in discharge rate. The pump efficiency characteristic with water is shown in Figure 5.19, depicts that all the data points lie on a smooth curve and the maximum efficiency of 75 % is measured. Figure 5.17 shows that maximum value of actual head developed by pump is 17.86 (m) at 1400 rpm and at 0 lps of discharge. It shows that with increase in discharge, head is decreased. Also the results with slurry with 10% concentration was calculed and shown below.

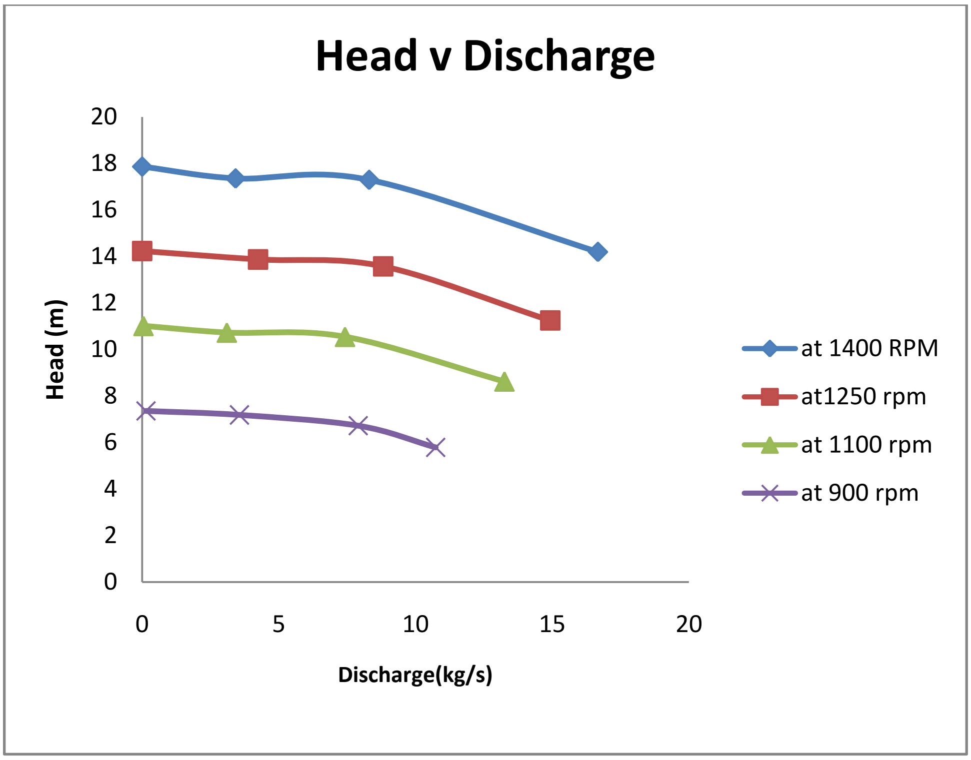


Figure 5.17 Numerical Head-Discharge characteristics of pump handling slurry at different rpm

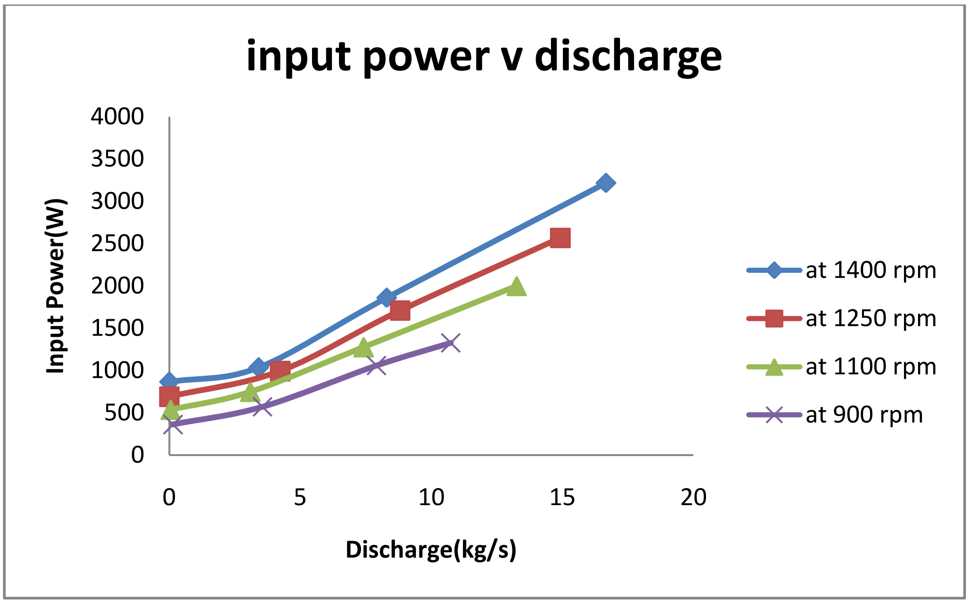


Figure 5.18 Variation of Input Power v/s Discharge characteristics of pump handling slurry at different rpm.

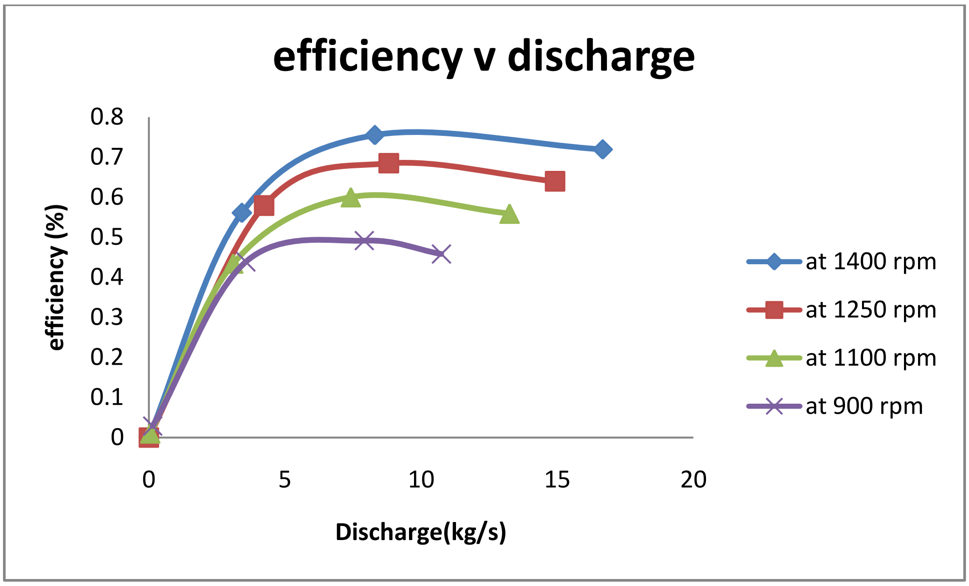


Figure 5.19 Variation of Efficiency v/s Discharge characteristics of pump handling slurry at different rpm.

Table 5.2 Comparison of results

Discharge	Speed	Experimental Results	Numerical Results
16.67	1400	12.8	14.19
8.301	1400	15.9	17.29
3.415	1400	16.41	17.35
0	1400	17.22	17.86
14.92	1250	10.19	11.25
8.811	1250	12.42	13.37
4.235	1250	12.89	13.87
0	1250	13.6	14.23

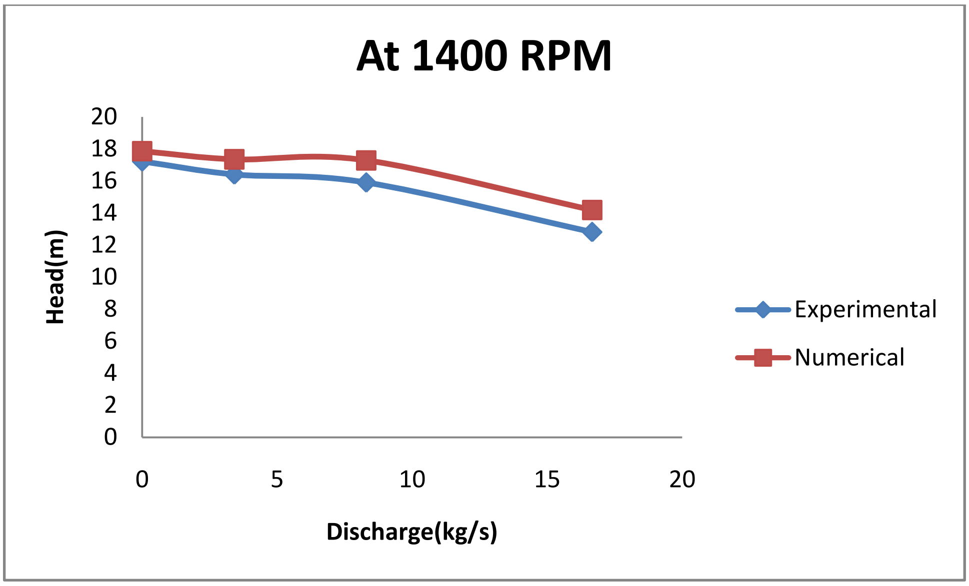


Figure 5.20 Comparison of results at 1400 RPM

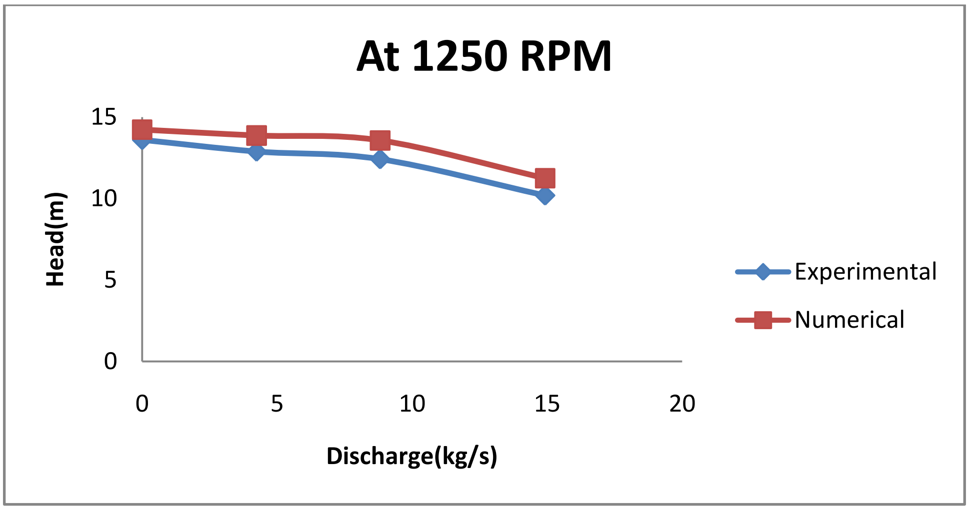


Figure 5.21 Comparison of results at 1250 RPM

Table 5.3 Results with water, 10% and 20% concentration

Discharge	Speed	Numerical Results water	Head calculated with slurry (10% cw)	Head calculated with slurry (20% cw)
16.67	1400	14.19	13.06	12.01
8.301	1400	17.29	16.32	15.22
3.415	1400	17.35	16.45	15.38
0	1400	17.86	16.78	15.67
14.92	1250	11.25	10.14	9.04
8.811	1250	13.37	12.49	11.36
4.235	1250	13.87	12.76	12.65
0	1250	14.23	13.37	13.24

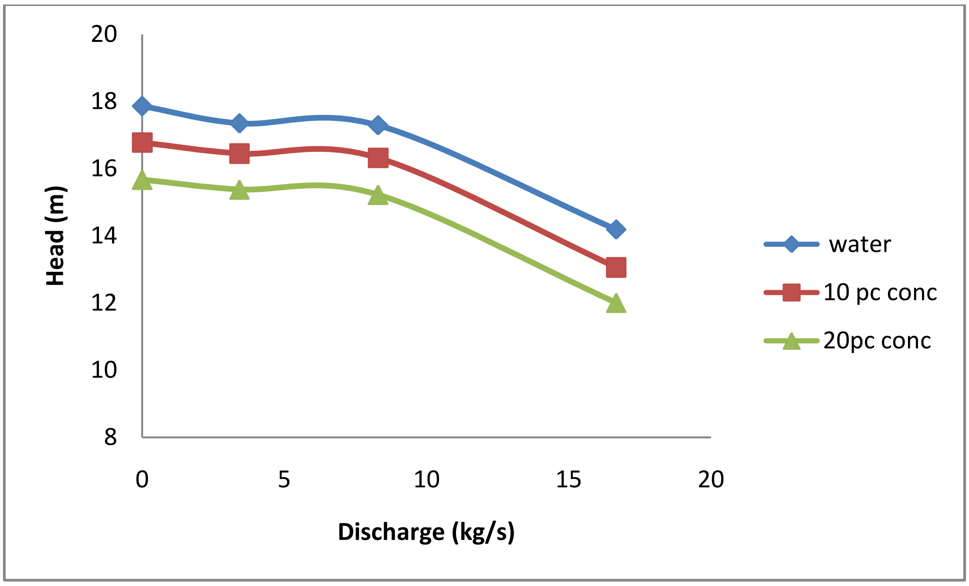


Figure 5.22 Results with water, 10%, 20% concentration

CHAPTER 6

CONCLUSION AND FUTURE SCOPE

Conclusion

Centrifugal pumps are used for transportation of solids and liquid 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. 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.The performance characteristics of centrifugal slurry pump are evaluated experimentally handling clear water. Also present work includes, study the rheological properties of bottom ash like particle size distribution, PH value, static settled concentration, specific gravity and viscosity of bottom ash. It also modelingof pump handling using ANSYS 12.The experimental results are obtained at the operating speed 1400rpm, 1250 rpm, with different mass flow rates. The pump characteristics of centrifugal slurry pumps are influenced by various particle parameters such as concentration and flow velocity of slurries.

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 slurry to improve performance characteristics of pump.

Reference

1. Sturge, D. P. and Cumpsty, N. A., 1975, Two-Dimensional Method for Calculating Separated Flow in a Centrifugal Impeller, ASME Journal of Fluids Engineering, pp. 581-597.

2. Wiedenroth, W., 1978, Experimental Work on the Transportation of Solid-Liquid Mixtures through Pipeline and Centrifugal Pumps, Hydrotransport 5, BHRA Fluid Engineering, Hannover (Germany).

3. 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.

4. 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.

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. 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), Esevier Science Publishers Limited, New York, pp. 247-258.

7. 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.

8. Maida, 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

9. Oh and Kim,(2001)Conceptual Design, Optimization of mixed-flow pump impellers using mean streamline analysis. Proceedings of Institution of Mechanical Engineers, Vol. 215.

10. Benrafriedrich-Karl, 2001, Economical Development of Efficient Centrifugal Pump Impellers by Numerical Methods. World Pumps, Vol. 416, pp. 48-53.

11. 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.

12. 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.

13. Miner, S. M., 2000, Evaluation of Blade Passage Analysis Using Coarse Grids, ASME Journal of Fluids Engineering, Vol. 122, pp. 345-348.

14. Pagalthivarthi, K.V., KapoorRajat 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.

15. 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.

16. 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.

17. 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

18. 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.

19. Zhou, W., Zhao, Z., Lee, T. S. and Winoto, S. H., 2003. Investigation of Flow through Centrifugal Pump Impeller Using Computational Fluid Dynamics. International Journal of Rotating Machinery, 9(1), pp. 49-61.

20. Ni, F, Vlasblom, W. J., and Zwartbol, A., ‘‘Effect of High Solid Concentration on Characteristics of a Slurry Pump,’’ Hydrotransport 14, BHRA Fluid Engg. Maastricht, TheNeatherland, 1996, pp. 141–149.
21. 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.
22. Oh and M.K. Chung (1999), “Optimum values of design variables versus specific speed for centrifugal pumps”. Proceedings Institutionof Mechanical Engineers,1999, Vol. 213 ,
23. Ogut, A. and Pastor, D. G., “Simulation of flow in turbopumpvaneless and vaned diffusers with fluid section” International Journal of Rotating Machinery, 2000, Volume 6, No. 1, pp 57-65.
24. Majidi, K. and Siekmann H. E., “Numerical calculation of secondary flowin pump volute and circular casings using 3D viscous flow techniques” International Journal of Rotating Machinery,2000, Volume 6, No. 4, pp 245-252
25. B.K Gandhi, S.N. Singh, V. Seshadri, “Variation of wear along the volute casing of a centrifugal slurry pump”. JSME International Journal, 2001, Vol.44, 2001.
26. Gandhi, B. K., Singh, S. N., and Seshadri, V., ‘‘Performance Characteristics of Centrifugal Slurry Pumps,’’ 2001, ASME J. Fluids Eng., **123**, pp. 271–28.
27. Oh and Kim, “Conceptual Design, Optimization of mixed-flow pump impellers using mean streamline analysis”. Proceedings of Institution of Mechanical Engineers, 2001, Vol. 215.
28. K S Paeng, M K Chung,”A New Slip Factor For Centrifugal Impellers”, Proceedings of Institution of Mechanical Engineers, 2001,Vol 215 Part A.

29. Engin, T., and Gur, M., ''Performance Characteristics of a Centrifugal Pump Impeller with Running Tip Clearance Pumping Liquid-Solid Mixtures,''ASME J. Fluids Eng.,2001, 123, pg. 532–538.
30. B.K Gandhi, S.N. Singh, V. Seshadri, "Effect of Speed on the Performance Characteristics of a Centrifugal Slurry Pump", Journal of fluid Engineering, February, 2002.
31. Kato, C., Mukai, H., and Manabe, A., "Large-eddy simulation of unsteadyflow in a mixed-flow pump" International Journal of Rotating Machinery, 2003, volume9, pp 345–351.
32. Nursen, E. C., and Ayder, E., "Numerical calculation of the three- dimensional swirling flow inside the centrifugal pump volutes" International JournalofRotatingMachinery,2003, volume9,247–253.
33. J R.Kadambi,Charoenngam P,Subramanian A, Mark P.Wernet,John M.Sankovic, Addie G, CourtwrightR,"Investigation of Particle Velocities in a Slurry pump using PIV:Part 1,The Tongue and Adjacent Channel Flow" Journal Of Energy Resources Technology,ASME. December 2004, Vol-126/271.
34. Hergt, P.,Meschkat, S. and Stoffel, B., "The flow and head distribution within the volute of a centrifugal pump in comparison with the characteristics of the impeller without casing"Journal of Computational and Applied Mechanics, 2004, Volume 5, pp 275-285.
35. Xu, C. and Muller, M., "Development and design of a centrifugalcompressor volute" International Journal of Rotating Machinery, 2005, volume 3, pp190–196.
36. A.Sellgren, "The new ANSI/HI centrifugal slurry pump standard", 17th International conference on the hydraulic transport of solids,The southern African Institute of Mining and Metallurgy and the BHR group, 2007.
37. L.Pullum,L.J.W Graham and M.Rudman, "Centrifugal pump performance calculations for homogeneous and complex heterogeneous suspensions",17th International conference on the hydraulic transport of solids,The southern African Institute of Mining and Metallurgy and the BHR group,2007.

38. Min-Guan Yang,Dong Liu,Xiang Dong, ”Analysis of turbulent Flow in the Impeller of a Chemical Pump”,Journal of engineering science and technology,Vol-2,N0.- 3,2007,Pg-218-225.