MODELING AND SIMULATION OF CENTRIFUGAL PUMP

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

Master of Engineering

In

CAD/CAM & Robotics

By:
ANU BALA
(800981004)

Under the supervision of:

Mr. Satish Kumar
Assistant Professor
Mechanical Engineering Department

MECHANICAL ENGINEERING DEPARTMENT
THAPAR UNIVERSITY
PATIALA – 147004

DECLARATION

I hereby declare that the work which is being presented in the dissertation work entitled, **"MODELING AND SIMULATION OF CENTRIFUGAL SLURRY PUMP"**, in partial fulfilment 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 **Mr. Satish Kumar** 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 July, 2011
Place: - Patiala

(ANU BALA)

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

Supervisor:

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

Countersigned by:

(Dr. Ajay Batish)
Head, MED
Thapar University, Patiala

(Dr. S. K. Mohapatra)
Dean of Academics Affairs
Thapar University, Patiala

ACKNOWLEDGEMENT

I would like to express my most sincere appreciation and deep sense of gratitude and indebtedness to my guide **Mr. Satish Kumar,** Assistant professor Mechanical Engineering Department, 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: - July 15, 2011

AnuBala.

(ANU BALA)

Place: Patiala

ABSTRACT

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. Centrifugal pump radial-flow type is the most common in slurry service. A conventional centrifugal pump is designed to handle clear liquids. However when slurries are to be transported the conventional centrifugal pump has to be modified to handle solid liquid mixtures. 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. Conventional design method of centrifugal pump are largely based on the application of empirical and semi-empirical rules along with the use of available information in the form of different types of charts and graphs as proposed by successful designers. As the design of centrifugal pump involve a large number of interdependent variables, several other alternative design are possible for same duty. 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 pump, and provides the valuable hydraulic design information of the centrifugal pumps. Present work is aimed to analyze the pressure and velocity distribution inside the pump passage and evaluate the pump performance using the Fluent, a computational fluid dynamics simulation tool. A numerical model of an impeller and casing has been generated and the complex internal pressure and velocity distribution are investigated by using the fluent computational code. . Pressure and velocity distribution inside impeller of the centrifugal pump has direct influence due to change flow passage.

CONTENTS

List of figures and tables

NOMENCLATURE

SYMBOLS

D	Diameter, m
Q	Mass flow rate, m_3/sec
H	Head, m
BHP	Brake horse power, hp
N	Speed, RPM
P	Density of liquid, kg/m_3
$\vec{v}$	Velocity vector
r	Stress tensor
g	Acceleration due to gravity, m^2/sec
h_0	Enthalpy
μ_t	eddy viscosity
$\vec{F}$	Force vector, N
$\dot{m}$	Flow rate
E	Total energy, J
k	kinetic energy per unit mass, J/kg
U	Free stream velocity, m/s
ε	Turbulence dissipation rate, m^2/Sec
σ_k and σ_ε	Turbulent Prandtl numbers for k and ε, respectively

Suffix:

i	x coordinate
j	y coordinate
z	z coordinate
1	inner diameter
2	outer diameter

CHAPTER 1

INTRODUCTION

1.1 PUMP

A **pump** is a machine used to move liquid through a piping system and to raise the pressure of the liquid. A pump can be further defined as a machine that uses several energy transformations to increase the pressure of a liquid.

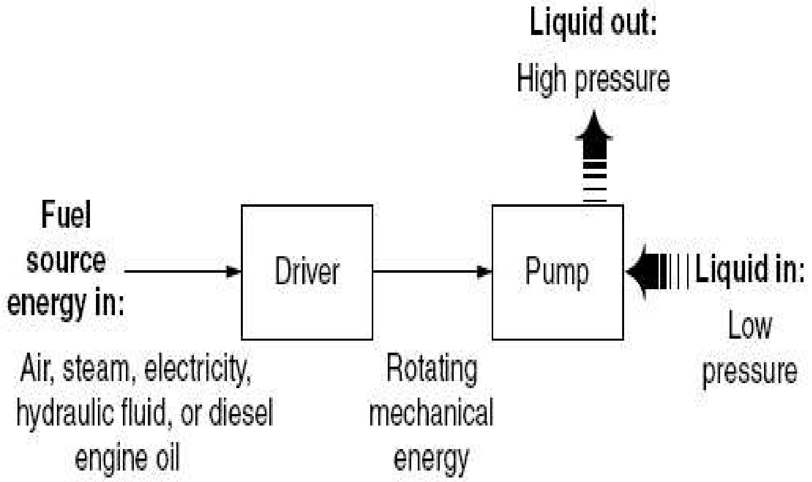


Figure 1.1 Working principle of Pump

1.2 TYPES OF PUMPS

1.2.1 RECIPROCATING PUMP

In reciprocating pumps the mechanical energy is converted into hydraulic energy by sucking the liquid into a cylinder in which a piston is reciprocating(moving backwards and forwards)which exerts the thrust on the liquid and increases its hydraulic energy(pressure energy),the pump is known as reciprocating pump. Reciprocating pumps are used where a precise amount of liquid is required to be delivered, also where the delivery pressure required is higher than that can be achieved with other types. Figure 1.2 shows line diagram of reciprocating pump.

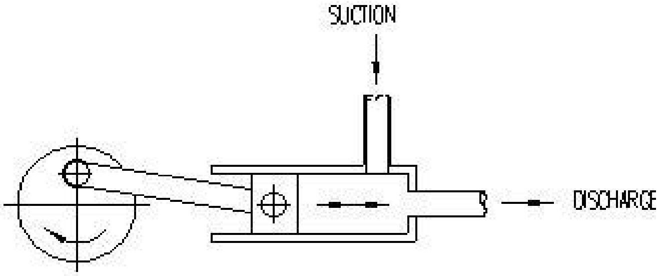


Figure 1.2 Reciprocating pump

1.2.2 ROTARY PUMP

Rotary pump is used to move heavy or very viscous fluids. These employ mechanical means such as gear, cam and screw to move the liquid.

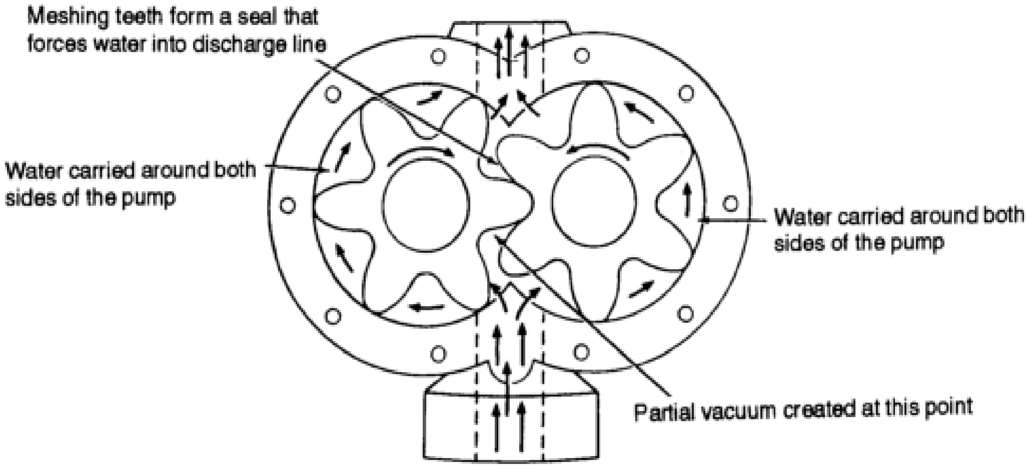


Figure 1.3 Sectional view of rotary pump

1.3 CENTRIFUGAL PUMPS

It is the rotodynamic machine. By rotating action develop the pressure able to lifting of liquid lower level to higher level. Centrifugal pump is explained with the following headings:

1.3.1 WORKING PRINCIPLE OF CENTRIFUGAL PUMP

Centrifugal pumps works on the basis of second law of Newton. Due to the rotation of the runner, called impeller the fluid at the inner radius moves to the outer radius & gain the Centrifugal head. Suction is created at the inlet to the pump which is called the eye. Continuous lifting of fluid thus takes place from sump to the pump while passing through the impeller the fluid take the energy from vane sin pressure & kinetic energy. A large amount of impeller outlet therefore made to convert the kinetic energy of fluid into pressure energy before the fluid enters the developing pipe.

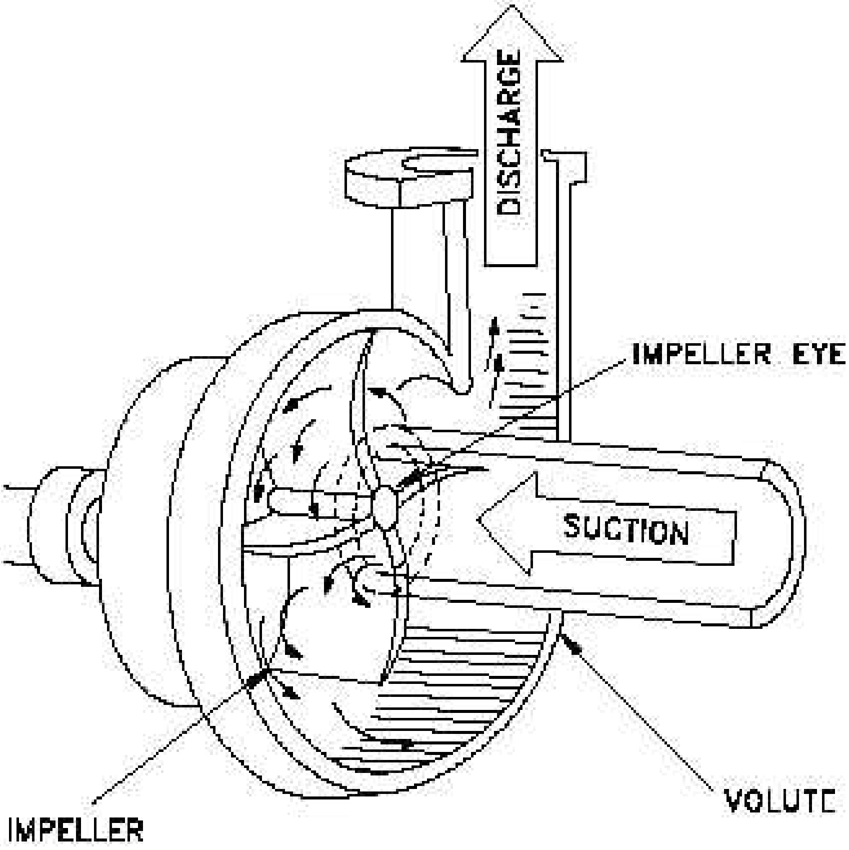


Figure 1.4 Working principle of Centrifugal pump

1.4 CLASSIFICATION OF CENTRIFUGAL PUMP

Based on their utility, design and Constructional features, centrifugal pumps can be classified with respect to the following characteristics.

1.4.1 Working head

a) **Low lift Centrifugal pumps**: Low lift centrifugal pumps are meant to work against heads up to 15m, Impeller is surrounded by a volute and there are no guide vanes.

b) **Medium lift:** Medium lift centrifugal pumps are used to build up heads as high as 40 m. They are generally provided with guide vanes.

c) **High lift:** High lift centrifugal pumps are employed to deliver liquids at heads above 40 m. High pumps are generally multistage pumps because single impeller cannot build up such a high pressure.

1.4.2 Types of casing

Pump casing should be so designed as to minimize the loss of kinetic head through eddy formation etc. Efficiency of the pumps largely depends on the type of casing.

a) Volute casing:

A casing operates on the principle of increasing the pressure energy in a free-vortex or spiral flow. In free-vortex, Angular momentum is constant.

So, $mvr = cons\tan t$ and $\therefore\ r \propto 1/v$.

Volute casing cross-section of the moving stream gradually increases from torque towards the discharging pipe. This increase in area results in a gradually decrease in velocity (kinetic energy) with corresponding increases in pressure. Most of the single stage pumps are built with volute casing.

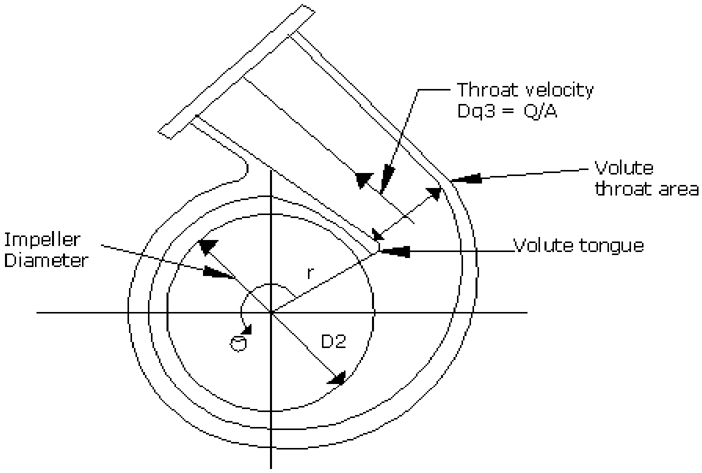


Figure1.5 Volute casing

b) **Volute with vortex chamber:** Vortex or whirl pool chamber: Annular space is provided between the volute and impeller. This arrangement arrests the formation of eddies and gives an improved performance.

c) **Diffusion pump:** Impeller surrounded by a guide wheel consisting of a number of stationary vanes or diffuser providing outlets with cross-section gradually enlarging towards the periphery. Water emerging from impeller flow past the guide vanes and as the section across flow increases, velocity falls and pressure is build up. Angle of guide vanes at the entrance should coincide with the direction of absolute velocity of water at impeller outlet. This arrangement is employed in all multistage pumps.

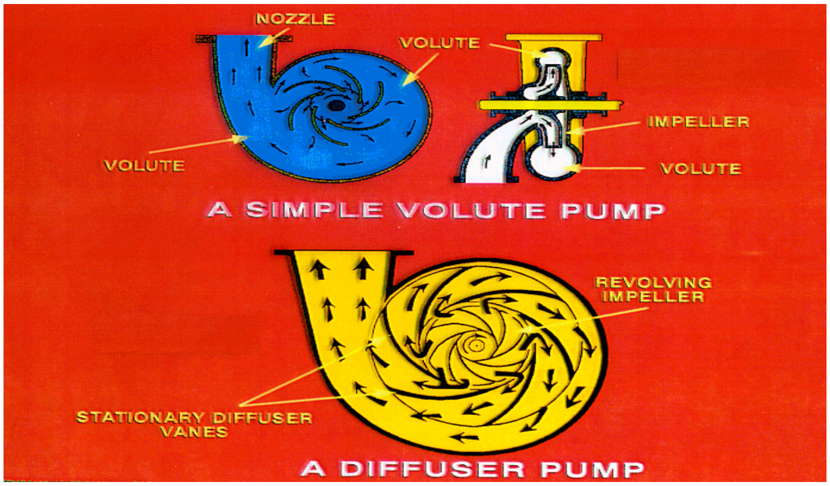


Figure 1.6 Diffuser pump

1.4.3 Number of stages

a) **Single stage pump:** It has one impeller keyed to the shaft. This is generally horizontal but can be vertical also. It is usually low lift pump.

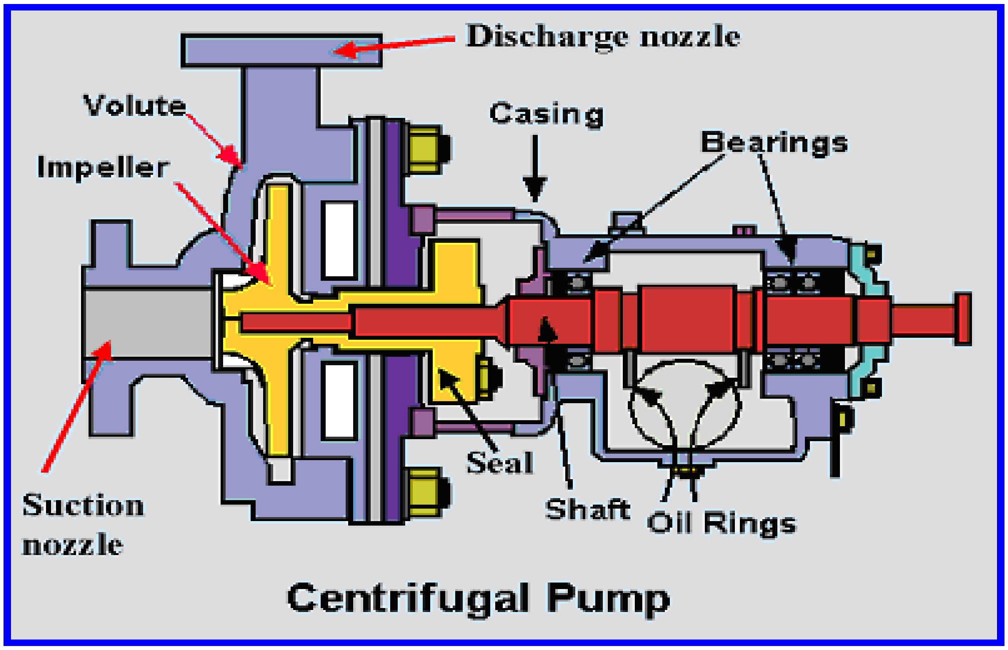


Figure 1.7: Single stage pump

b) Multistage pump: Multi Stage Centrifugal Pump:-It has two or more impellers keyed to a single shaft enclosed in the same casing. Pressure is built up in steps. The impeller is surrounded by guide vanes and the water is led through a by-pass channel from the outlet of one stage to the entrance of the next until it is finally discharged into a wide chamber from where it is pushed on to the delivery pipe . These pumps are used essentially for high working heads and the number of stages depends on the head required.

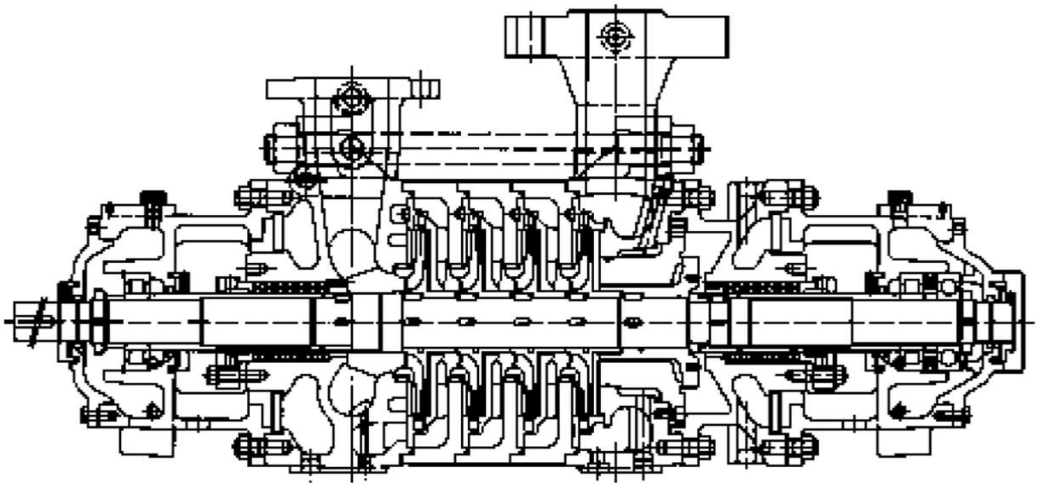


Figure 1.8 Multistage pump

1.4.4 Relative direction of flow through impeller

a) **Radial flow pump:** It is that pump in which the liquid flows through the impeller in the radial direction only. Ordinarily all the Centrifugal pumps manufacture with radial flow impeller.

b) **Mixed flow pump: :** In mixed flow pumps the liquids flows through the impeller axially as well as radially i.e. there is a combination of radial and axial flows. A mixed flow pump is just a modification of radial flow type in this respect that the former is capable of discharging a large quantity of liquid.

c) **Axial flow pumps:** In axial flow pumps the impeller is in the axial direction only. Axial flow pumps are usually designed to deliver very large quantities of liquid at relatively low heads. However, it is not justified to call axial flow pumps as centrifugal pumps because there is hardly any centrifugal action in their operation.

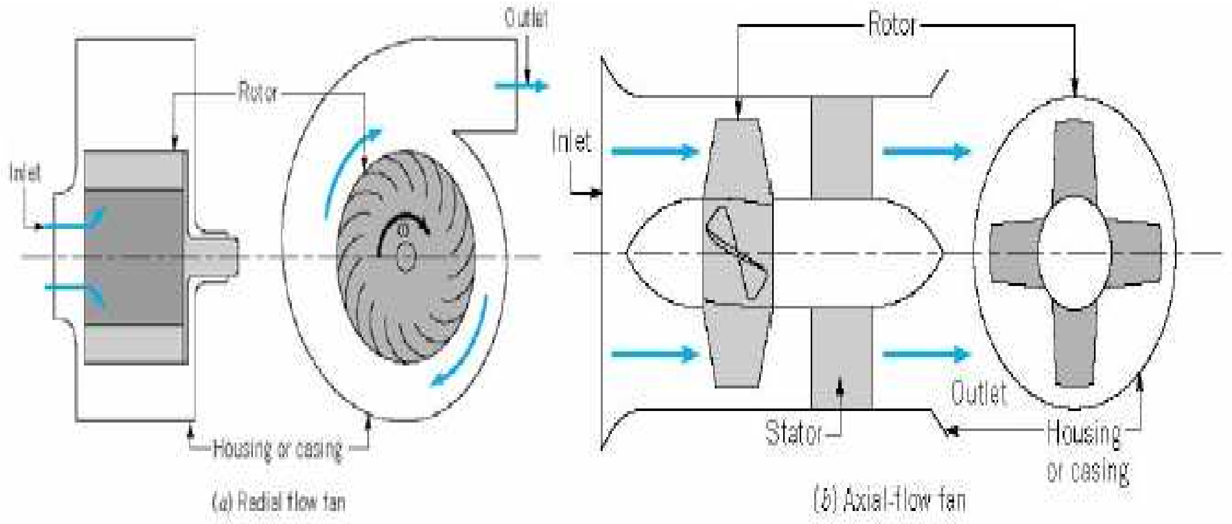


Figure 1.9 Radial and axial flow impellers

1.4.5 Number of entrance to impeller

a) **Single entry or single suction pump:** In a single suction pump liquid is admitted from a suction pipe on one side of impeller.

b) **Double suction pump:** In double suction pumps liquid enters from both sides of impeller. A double suction pump has an advantage that by this arrangement the axial thrust on the impeller is neutralized.

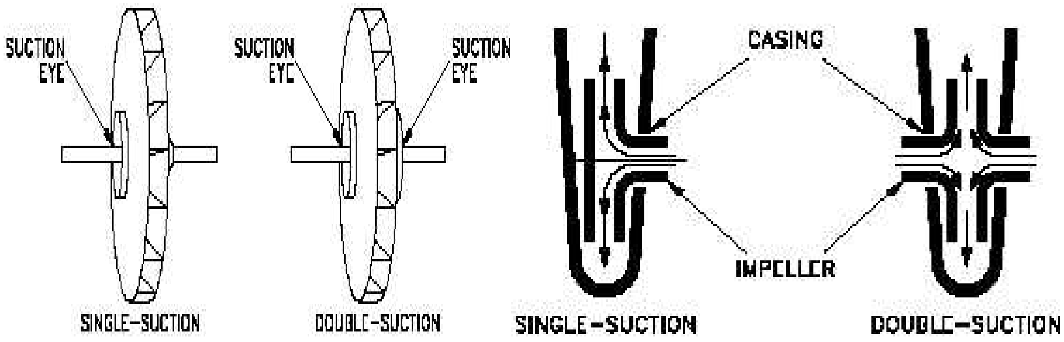


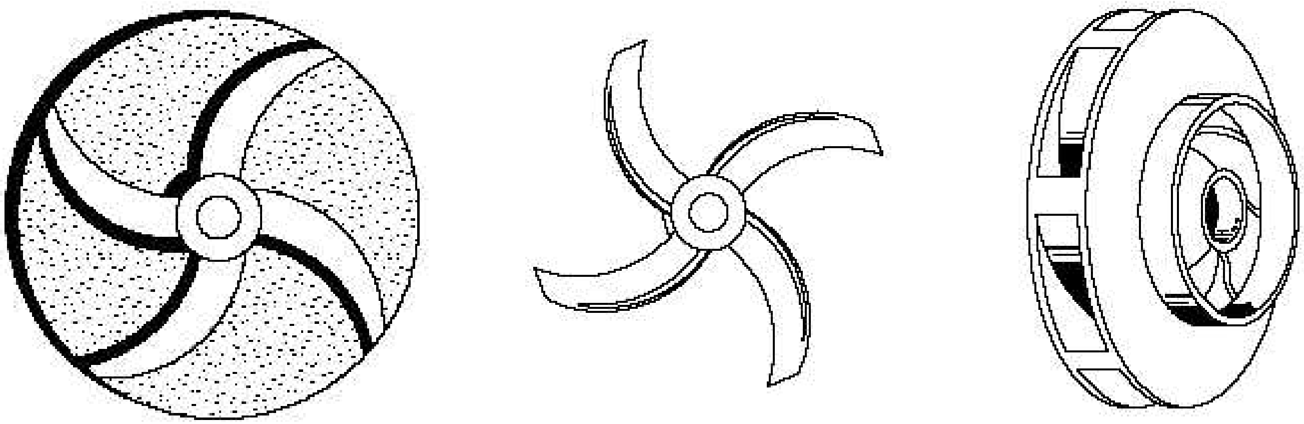
Figure1.10 Number of entrance to impeller

1.4.6 Liquid handled

Depending on the type and viscosity of liquid to be pumped, the pump may have a closed or open impeller.

a) Closed impeller:

An ordinary centrifugal pump is equipped with a closed impeller in which the vanes are covered with shrouds on both sides. This type is meant to handle non-viscous liquid such as ordinary water, hot water, hot oils and chemicals like acids etc, material of the impeller should be selected according to the chemical properties of liquid used. For hot water at temperature exceeding 150 degree celcius cast steel impeller is recommended.



(a) Semi-open impeller **(b) Open impeller** **(c) closed impeller**

Figure1.11 Types of impeller

b) **Semi open impeller:** The impeller is provided with shroud on one side only. This pump is used for viscous liquid such as sewage; paper pulp etc, choice of material for manufacturer of impeller is influenced by chemical nature of liquid to be handled.

c) **Open Impeller Pump:** The impeller is not provided with any shroud. Such pumps are used in dredgers, and elsewhere for handling mixture of water, sand, clay etc, it is generally made of forged steel.

1.4.7 Specific speed (N)

a) Slow speed radial flow runner- 10 to 30 rpm

b) Normal speed radial flow impeller- 30 to 50 rpm

c) High speed radial flow impeller- 50 to 80 rpm

d) Mixed flow runner- 80 to 160 rpm

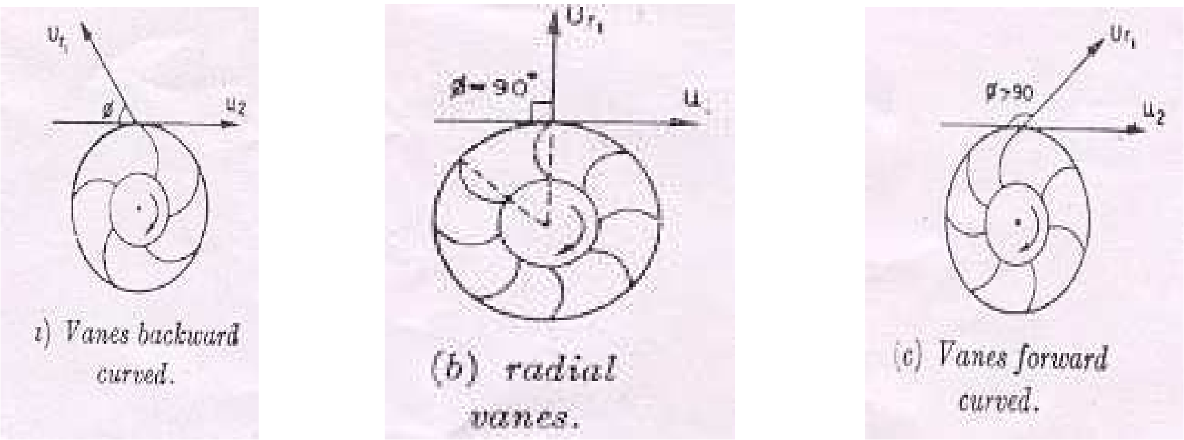
e) Axial flow runner- 110 to 150 rpm

1.4.8 Outlet blade angle

a) **Backward Blade:** Outlet blade curves in a direction opposite to that of motion , & the angle between the blade tip & the tangent to rotor at exit is below 90 ($\beta_2<90$)

b) **Radial Blade**: Liquid leaves the blade with relative velocity in a radial direction & angle $\beta_2=90$.

c) **Forward Blade:** Outlet tip of blade curves in the direction of motion & the angle between blade tip & the tangent to rotor at exit is obtuse ($\beta_2 > 90$)



i) Vanes backward curved.

(b) radial vanes.

(c) Vanes forward curved.

Figure 1.12 Outlet blade angle

1.4.9 Position of impeller

a) Horizontal impeller shaft pump

b) Vertical impeller shaft pump

The centrifugal pumps may be designed with either horizontal or vertical position of shaft, generally the pumps are provided with horizontal shaft. For deep wells and mines the pumps with vertical shaft are more suitable because the pumps with vertically disposed shaft occupy less space.

Figure 1.13 (a) Horizontal shaft pump **Figure1.13 (b) Vertical shaft pump**

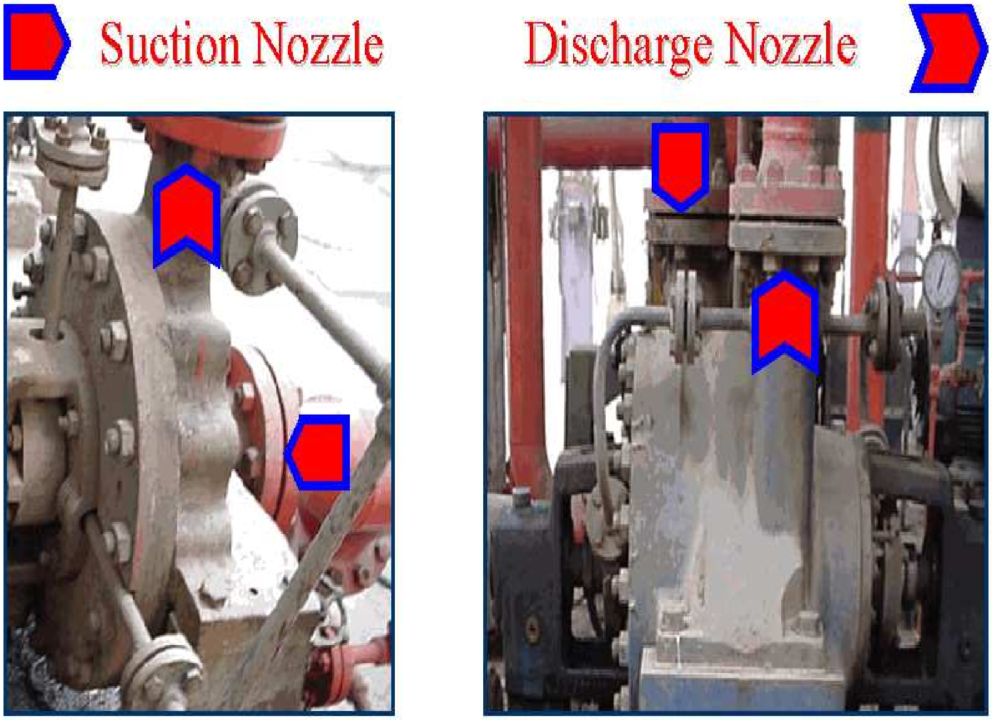
1.4.10Suction and Discharge Nozzle

a) End suction/Top discharge –

This pump is always of an overhung type and typically has lower NPSH because the liquid feeds directly into the impeller eye.

b) Top suction/Top discharge –

This pump can either be an overhung type or between-bearing type but is always a radially split case pump.



a) End suction/Top discharge b) Top suction/Top discharge

Figure 1.14 Position of suction & discharge

1.5 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

1.6 MERITS OF CENTRIFUGAL SLURRY PUMP

1) Simplified design

2) Easy installation

3) Easy maintenance

4) Lesser weight

5) Easy transportation of fluids.

1.7 CENTRIFUGAL PUMP FOR SLURRY

A conventional centrifugal pump is designed to handle clear liquids. However when slurries are to be transported the conventional centrifugal pump has to be modified to handle solid liquid mixtures. 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. Slurry pumps are available in variety of materials of construction to best handle the abrasion, corrosion and impact requirements of nearly any solids handling application, replaceable liners are used in critical areas of wear to reduce the costs of parts replacement. The materials used have different properties to prevent wear and erosion to ensure longer working life as compared to conventional pumps used for clear liquids. These modifications increase the hydraulic losses in the pump and deteriorate the pump performance. The efficiency of a centrifugal slurry pump is considerably low as compared to a conventional pump. The performance characteristics of slurry pumps are also poor as compared to the conventional pumps. The deterioration in the performance could be attributed to the modifications incorporated to allow free flow of solid particles along with the liquid to be transported. The designers of slurry pumps have to take the following parameters into account while designing the slurry pumps:

- Abrasive hardness
- Particle shape
- Particle size
- Particle velocity and direction
- Particle density
- Particle sharpness

During the design of such pumps, the requirements of longer life and reliability is to be balanced by the constrained of high initial costs and efficiency. Nearly all slurry pumps have larger diameter impellers than units for pumping clear liquids, to enable heads and capacities to be met at reduced rotational speed. Low speed operation is one of the most important wear reducing features of a slurry pump A centrifugal slurry pump is designed to handle solid-liquid mixture and is normally a single stage, end suction type having radial or mixed flow configurations to facilitate the motions of solid particles Some compromises are made in order to provide an acceptable pump life. Table 1.1 shows the design features, benefits, and compromises of the slurry pump.

Design feature	**Benefit**	**Compromise**
Thick Wear sections	Longer component life	Heavier, more expensive parts
Larger impellers	Slower pump speeds & longer component life	Heavier, more expensive parts
Speciality materials	Longer component life	expensive parts
Semi volute or concentric casing	Improved pump life	Loss of efficiency
Extra rigid power ends	Improved bearing lives	More expensive shafts & bearings

Table 1.1 SLURRY PUMP DESIGN FEATURES

1.8 TRANSPORTATION OF SLURRY

There has been a phenomenal growth in the demand of raw materials over the last few decades. This increase in demand has led to drastic changes in 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 1.15. 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.

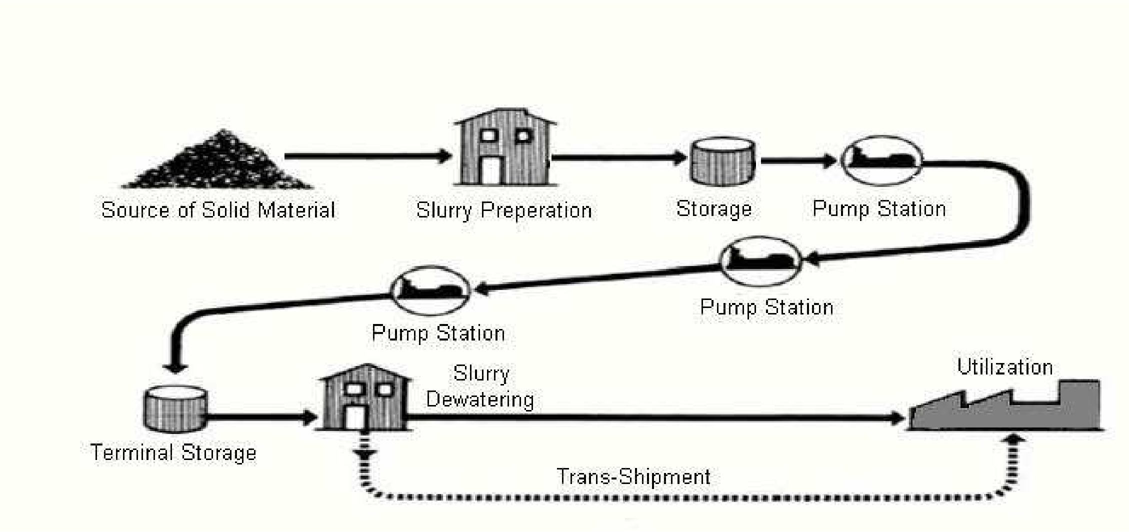


Figure 1.15 Transportation of slurry

1.8.1 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. 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.

CHAPTER-2

LITERATURE REVIEW

Parida et al. (1996) investigated the rheological and pipe flow behavior of ash samples from Talcher Thermal Power Station, Orissa. The viscosity of the fly ash slurry was found to be Newtonian in nature up to a solids concentration (C_W) of 50% and above this concentration the viscosity is non-Newtonian. The power law pseudo-plastic model correctly characterizes the non-Newtonian models the head loss of the slurry were predicted. It was indicated that the transportation cost of fly ash slurry reduces drastically if the same is transported at high concentrations instead of low concentrations.

Ward et al. (1999) investigated the hydraulic transportation of dense fly ash slurry using a stabilizing additive to prevent sedimentation of fly ash particles. But the addition of stabilizing additive increased the slurry viscosity for which a dispersing additive was to be used to solve the problem.

Ni et al [2000] have evaluated slurry pump performance of three types of narrow graded sands with deliver volumetric concentration up to 44% experimentally. It has been observed that high solid concentration causes strong influence on pump head, efficiency and power consumption and this influence behaves differently with different sand size. The pump efficiency in the coarse sand slurry service may drop almost 60%, compared to that to that of water service. The head ratio and efficiency ratio drop faster at solid volumetric concentration of about 35 %. For the fine and medium sand holds the concentration at least up to 35% for the coarse sand at efficiency ratio and Head Ratio are unity. However, the pump efficiency drop much faster than pump head when volumetric concentration less than 15%. A result, power requirement increases nonlinearly with slurry relative density. At exit two layers with different solid concentrations are observed in each passage of a pump impeller due to impeller rotation.

Eric dick et al. [2001] have used different methods of fluent code (Moving Reference frame, Mixing plane method & sliding mesh technique) to predict the performance of centrifugal pump. They took two pumps, one of which has a low specific speed in 2-d form & second one has medium specific speed & doubly curved vanes. They found that head is a function of flow rate & found quite closer to the experimental results above nominal flow rates. Effect of pressure variation on the impeller flow is analysed. Finally, sliding mesh technique has been found to be the most effective technique, which also considers the fluid inertia effect on flow.

Jose´ Gonza´lez et al. [2002] have studied the fluent code to predict the dynamic effects of unsteady fluid flow due to impeller-volute interaction in centrifugal pumps. Sliding mesh technique with k-ε turbulence model, time averaged Navier-Stokes equations on a commercial pump with backward curved blades are applied. They concluded that good agreement between the studied & experimental results generated for static pressure, unsteady distribution & dynamic forces at off-design operating conditions has been found.

Weidong Zhou et al. [2003] have predicted the off-design performance of two pumps with straight & twisted blades respectively using CFD code CFX, with advanced mesh technology for complex geometries and k-ε turbulence model. It was found that twisted bladed pump had better efficiency amongst the two. Similarly, comparing the two pumps both with twisted blades of different sizes; it was found that 25% of the design flow rate, the flow pattern is almost same. At 35- 40% of the design flow rate, the reverse flow arises and flow pattern changes.

Parida et al. (2003) using the flow and head loss characteristics of high concentration fly ash-bottom ash mixture slurry carried out the pipeline design for hydraulic back-filling of coal mines by considering the effects of solids concentration, bottom ash fraction, pipe diameter and flow velocity. A design chart was formulated to determine the pipe size and design transport velocity for a given backfilling rate and given H/L ratio i.e. length of vertical section of pipe (m) to length of horizontal section of the pipe (m).

K M Guleren et al.[2004] demonstrated a simulated work on flow in centrifugal vane pump having five curved blades with nine diffuser vanes. The pump model is generated in gambit & solid works & flow is analysed by fluent software. Analysis done at 15 different flow rates produces unstable results due to stalled region effects. Also these results have good agreement with the experimental ones. The performance curve and flow instabilities are also predicted very well by the fluent. Pressure fluctuations around the impeller and diffuser were found to be dependent on flow rate. The fluctuation is particularly more at off-design conditions but more stable at the design flow rate.

Migualasuaje et al. [2004]worked on inverse design methods to improve & optimize the design of centrifugal pump by using CFD tools and softwares like HELIOX, used for 1-D design analysis of pumps with volute or deswirl vanes, and REMIX, used for Meridio anal& blade to blade flow analysis. For this purpose, they used a centrifugal pump with vanes and analysed it with the help of CFD tools like CFD-BladeGEN+ and CFX TASC-FLOW. Very good results were obtained around the design point for partial, nominal and off-design flow rates, which confirm the reliability of the model.

Senapati et al. (2004) conducted the static studies, rheological behavior and design scale-up procedure for predicting head loss for fly ash samples collected from NALCO, Angul, India. Studies indicated that the maximum static settled concentration ($C_{W\ max}$) of around 69% can be achieved for the slurry and hence a slurry in the range of 60-67% by weight can be prepared for transporting through pipeline in laminar mode due to its non-settling nature. The preparation of such slurry was done using a helical ribbon mixture and a slurry concentration of 67% was easily achieved. The flow behavior of the ash slurry was affected by addition to bottom ash indicating lesser viscosity. The head loss of fly ash slurry was predicted using a non-Newtonian power law model.

A. Sellgren et al. [2004] modelled a selection of pump designs producing general relationships for the different pump casing, impeller and liner components for different duties. They then take these and show which offer the lowest cost of ownership for different services. The calculated values show total cost of ownership is affected significantly by changes in operating conditions (due to wear) and is a minimum, in most cases, at around a design specific speed.

J.M. Sankovic et al. [2004] demonstrated the results of the initial phase of obtaining experimental measurements throughout a rotary pump. Measurements in the volute exit/diffuser section were obtained at different pump speeds. Over the range of operating conditions tested, various flow phenomena were noted Reynolds shear stress measurements showed regions of significant stress levels that warrant further investigation.

G. R. Addie et al. [2005]worked on a 3 vane pump with 0.3m diameter impeller & open shroud with the help of unique model called "Viscosity correction method" to find the effect of properties of slurries on the performance & wear of pump. Different results obtained here point out the strong influence of the rheological behaviour on the choice of pump size and the power requirement. Also with the same method, the efficiency of non-Newtonian fluid was measured in a reasonable way but head reduction was under-estimated. For the underground fluid stated above, the method gives good results of viscosity at BEP flow rate but head and efficiency results were over-estimated.

John S. Anagnostopaupoloset al. [2006] demonstrated a 3-d turbulent flow simulation of centrifugal pumps solving RANS equation with control volume approach on Cartesian grid. A no. of controllable design variables is taken under consideration so as to optimize the geometry of impeller. Hydraulic efficiency, flow analysis & performances are calculated. Also the flow analysis vs. various design parameters is also drawn. The results are found to be in agreement with the experimental results. So, the use of design variables in impeller geometry along with grid generation algorithm constitute an effective tool for inverse designing of centrifugal pump, performance & design sensitivity analysis & optimization.

Carlos Santolaria et al. [2006] have worked on a low specific speed pump to predict the relation between the global variables & dynamic flow structure. For this purpose, a single stage centrifugal pump is used. The solver used is Fluent with k-ε turbulence model. Boundary conditions used are total pressure at inlet & pressure drop at outlet. Flow, pressure, torque fluctuations are analysed at different operating conditions. The flow structure changes inside the pump due to pressure & torque fluctuations have been correlated & the results are valuable explaining many flow characteristics. The relationship between global & local variables in turbo machinery plays a major role in defining the geometry of turbo machinery. It may help in future work to be done on design in this field.

M. H. ShojaeeFard et al. [2006] have simulated the 3-d fluid flow of a centrifugal pump, but with viscous oils as Newtonian fluids. Head correction factor is found to be related to operating conditions. Pressure distribution over the suction & pressure side of the blade, when flow rate is the nominal one, is the best & clearly appreciated.

Si Huang et al. [2006] have studied a multistage centrifugal pump with simulation & analysis in a commercial CFD software package using MRF technique, k-ε turbulence model. Performance is measured & connected with 3d flow field. Grid generation is done in GAMBIT and analysis in FLUENT with tetrahedron cell elements. Boundary conditions are mass flow inlet and at outlet, flow leakage takes place. A considerable interference of the velocity field at the impeller exit appears due to the interaction between impeller blades & diffuser vanes.

Vlasak et al.(2007) investigated the pipeline transportation of fly ash slurry at volume concentrations in the range of 22% - 31% and indicated that by adding bottom ash fractions the fly ash-bottom ash slurries reached slightly higher maximum concentration. He also revealed that a substantial reduction of the flow resistance can be achieved by addition of some drag reducing agents or by arrangements of particle size distributions or adopting some mechanical treatments. The flow behavior of the slurry was approximated by Yield-power law model. Since the addition of drag reducing agents causes a decrease of head loss therefore it is possible to use a lower operational velocity for the stabilized slurry, which brings a significant reduction in head loss. The effect of addition of Sodium hexa-meta-phosphate at 0.1% concentration (by weight) as an additive or rheological flow behavior of ash slurry was studied by Seshadri et al. (2008). By using the rheological data the head loss of fly ash slurry at high concentrations (C_W = 60%, 65% and 68%) were predicted using a Bingham plastic model. Since the additive modified the rheological behavior of the ash slurry a substantial reduction in head loss and energy consumption could be achieved.

A Benertam et al. [2007] showed the influence of solid particles on centrifugal pump having six impeller blades & found that size and density like parameters etc. of the solid particles influences the head reduction factor and efficiency. The present work is done using well defined conditions, which are not universal and may not be true for all operating conditions. The correction factors generated in this paper vary with parameters like concentration, particle size/shape and mixture density. The material used for the pump is also derivable from these parameters. The study is quite effective if the operating conditions used for this work could be made more general.

L. Pullum et al.[2007] made a model for check and analysis of performance of centrifugal pump which conveys pastes/coarse solid particles. They introduced characteristic shear rate factors and proposed a model based on pump geometry, fluid rheology & flow rate. The results show that moderate to high viscosity fluid generates a laminar flow. An overall equivalent viscosity is used with Hydraulic Institute Method to estimate viscous deration.

Low flow rates with increased viscous deration prove the sensitivity of model. Head reduction is a function of coarse solid concentration.

Adnan Ozturk et al. [2008] studied the effect of impeller diffuser radial gap ratio in a centrifugal pump. For this purpose, they took a non-traditional centrifugal pump &subjected its diffuser to different radial gaps with 15 different flow rates. Computational results are taken for static pressure, velocity vectors & turbulent kinetic energy distribution at design volume flow rate at different radial gaps. K.E. drops as the radial gap increases. Decreasing gap ratio decreases pump efficiency also & increases pressure fluctuations.

Khin Cho Thin et al. [2008] have demonstrated a work to find the best performance point by taking a design & model based on "Berman method". Many parameters like velocity fluctuations, theoretical, actual & net theoretical head, losses like friction, shock, recirculation etc. have been calculated keeping in mind only the two main parts of pump i.e. impeller & volute. This analysis shows that delivery head depends on the flow rate & this very connection is also called pump performance. All the above parameters are calculated by varying flow rates.

Mohammad F. Khalil et al. [2008] have studied that oil-in-water emulsion flow reduces the head and flow rate of the centrifugal pumps. As the holdup increase and temperature decrease, the reduction in head and flow rate increase. The change in the rheology of the emulsions with holdup, temperatures, and addition of surfactant is responsible for increasing the losses inside the pump and change in performance. Thus, the hydraulic efficiency decreases as the holdup increase and temperature decreases.

Kamal Elnahhas et al.[2009] found that Particle size distribution plays an important role to obtain higher concentrated slurries and higher carrying capacity. Therefore, economic advantage can be gained by adjusting particle size distribution. They investigated the flow behaviour and pressure frictional losses of settling slurries flowing in pipes.

Krishnan V. Pagalthivarthi et al. [2003] have considered the effect of various operational and geometry conditions on these quantities *viz.* wall shear stresses, solid concentration along the wall and tangential velocity along the wall. Solid concentration and solid wall shear stress increase monotonically from the upstream of the tongue region to the downstream of the belly region. These quantities are crucial in wear calculations along the casing wall.

H Chen et al. [2010] analysed the reasons of low efficiency under different operation condition based on the performance test and CFD numerical simulation approach. Analysis focuses on the relationship between pump efficiency and inner flow characteristics. The study of the pump took a step forward when a gap drainage blade was proposed. The author proposed a new flow controlling method which not only improves the low specific speed pump performance but enriches and develops the centrifugal pump design theory.

Oh and Kim[2001] developed a conceptual 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. Correlation provided was a function of number of vanes, vanes exit angle & the inlet-exit radius ratio. 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 and Gur [2001] have studied the effects of different solid-liquid mixture properties on the performance characteristics of a centrifugal enshrouded impeller pump, considering the variation of the tip clearance. The effect of the clearance between the impeller tip and the casing and of the solid concentration, density and mean diameter on the pump performance characteristics is investigated.

Stephan Bross et al[2002] predicted the influence of different design parameters on the wear behaviour of centrifugal slurry pump’s impeller suction sealing .For this purpose he developed a simple model and using this model he calculated the velocity field in the impeller suction side and also a comparison was done between analytical solution & numerical solution provided by a CFD package FLUENT.

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.

Goto Akira et al [2002] have proposed a computer aided design system for hydraulic parts of pumps including impellers, bowl diffusers, volutes and vaned return channels. Technologies include 3D-CAD modeling, automatic grid generations, CFD analysis and a 3D inverse design method.

Egin and Gur [2003] have evaluated some existing correlations to predict head degradation of centrifugal slurry pumps. A new correlation has been developed in order to predict head reductions of centrifugal pumps when handling slurries. The proposed correlation takes into account the individual effects of particle. The proposed correlation is therefore recommended for the prediction of performance factors of ‘‘small-sized’’ slurry pumps having impeller diameters lower than 850 mm size, particle size distribution, specific gravity and concentration of solids, and impeller exit diameter on the pump performance.

Kadambi et al [2004] have used Particle Image Velocimetry 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.

Graeme R. Addie et al [2005] have discussed numerical model of flow and particles. They have used the experiments which have been conducted to obtain the particle velocities inside an optically transparent acrylic pump using Particle Image Velocity (PIV). They have presented effect of different parameters on operating cost of pump. They concluded that wear parts cost of slurry pumps may be about 50% of the total operating cost of pumps.

Addie et al. [2007] have developed 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.

Yang et al [2007] have for evaluated the internal flow in the impeller of the centrifugal chemical pump by CFD FLUENT software. Standard k -ε (two-equation) turbulence model was used. Simultaneously the result of calculation is compared with PIV measurement. They found that the internal flow which is simulated in impeller is coincide with the general rule of flow in the impeller machinery, and validated with the result of PIV experiment.

CHAPTER-3

COMPUTATIONAL FLUID DYNAMICS

3.1 BASIC INTRODUCTION

Fluid mechanics is the study of fluids either in motion (fluid dynamics) or at rest (fluid *statics*) and the subsequent effects of the fluid upon the boundaries, which may be either solid surfaces or interfaces with other fluids. Both gases and liquids are classified as fluids, and the number of fluids engineering applications is enormous: breathing, blood flow, swimming, pumps, fans, turbines, airplanes, ships, rivers, windmills, pipes, missiles, icebergs, engines, filters, jets, and sprinklers, to name a few. When you think about it, almost everything on this planet either is a fluid or moves within or near a fluid. In the beginning, the use of these techniques was customary only in the areas of aerospace and nuclear technology. Subsequently, the use has spread to a variety of products, physical situations, and manufacturing processes. Some examples of interesting applications of computational modeling are cooling of electronics systems, rotating and reciprocating machinery, furnaces, and combustion chambers. Computational Fluid Dynamics (CFD) provides a qualitative or quantitative prediction of fluid flows by means of

- Mathematical/Analytical formulation (partial differential equations)
- Discrete and solution techniques
- Programming & software tools (solvers, pre- and post processors)

Fluid flows and related phenomena can be described by PDE's, which cannot be solved analytically except in few special cases. To obtain an approximate solution numerically, we use a discrete method which approximates the differential equations by a system of algebraic equations, which can then be solved on a computer (in the form of matrices also). The calculations are applied to small domains in space and/or time so the numerical solution provides results at discrete locations in space and time.

3.1.1 Principles of Fluid Motion

- Mass Conservation (Continuity Equation)
- Momentum conservation
- Energy Conservation

3.2 CFD APPLICATIONS

- Aerodynamics of aircrafts and vehicles
- Hydrodynamics of ships
- Power plants: combustion in IC engines and gas turbines
- Turbo machinery
- Electrical and electronic equipments cooling
- Chemical processes
- Buildings: wind loading, heating and ventilation
- Marine engineering
- Hydrology and oceanography

- Meteorology
- Biomedical engineering

3.3 CFD PROCEDURE

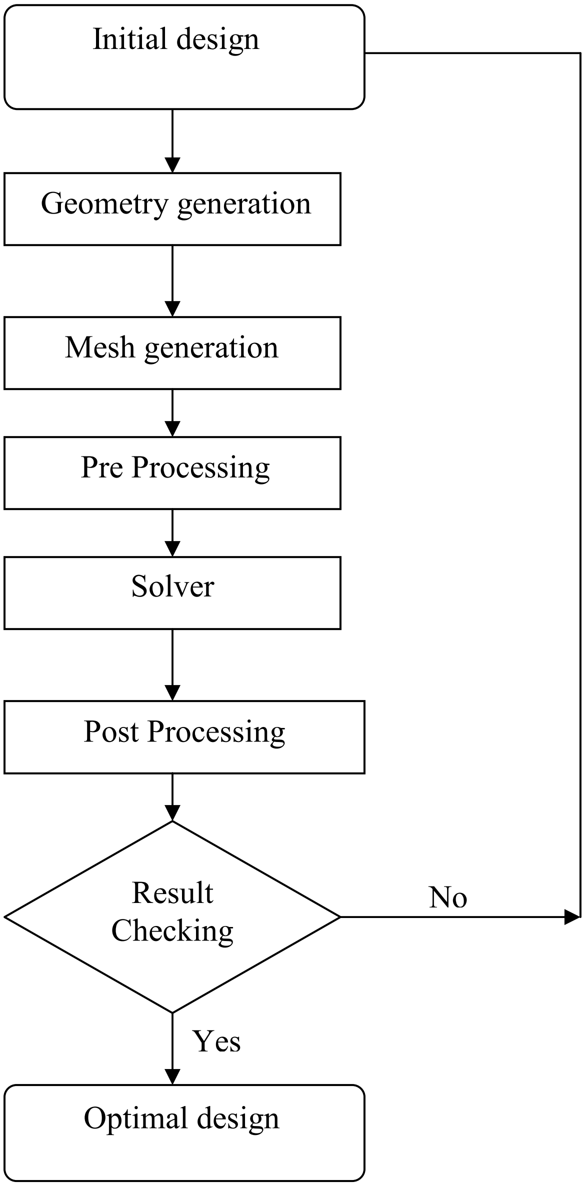


Figure 3.1 CFD procedure

All CFD components contain **main three elements.**

1. A **Pre-Processor**:

- Define the domain of interest
- Grid generation: sub-division of domain into smaller elements
- Selection of physical/chemical phenomena needed to be modeled
- Definition of fluid properties
- Specification of boundary conditions

2. A **flow solver**, which is used to solve the governing equations of the flow subjected to the conditions provided. There are three different methods used as a flow solver:

- **Finite element method** : In the finite element method, the fluid domain under consideration is divided into finite number of sub-domains, known as elements The summation of variation of the variable in each element is used to describe the whole flow field.

- **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.

- **Finite volume method** : The finite volume method is currently the most popular method in CFD. In this governing equations are in integral form and solution domain is sub divided into a finite number of control volume and conservation equation are applied to each control volume and computational nodes locate at the center of each control volume.

3. **A post-processor**:

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking
- Manipulation of output

3.4 BOUNDARY CONDITIONS

Boundary conditions are the set of conditions specified for the behavior 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.

3.4.1 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\

$$Vfluid = Vwall \tag{1}$$

- 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)

3.5 COMPUTATIONAL MODELLING

The modelling of the centrifugal pump is important step after understanding specification of the problem including geometry, flow conditions, and methodology of simulation. The dimensions of the pump were obtained by measuring the “50M WILFLEY” centrifugal slurry pump flow passage. The geometry of impeller, volute casing and inlet passage are modelled by using pre-processor Gambit 2.3.16. In this first we create the vertices of impeller blade profile, casing and inlet passage. These vertices are then joined to create the edges of all the three parts of the pump. These vertices can be joined by line, arc, splines curves, etc. Spline curves were used to create the edges of blades and the casing. These edges are further joined to create the faces of different parts of the pump. The faces are then joined together with stitch command to form volumes. These volumes are then moved to the places for assembly.

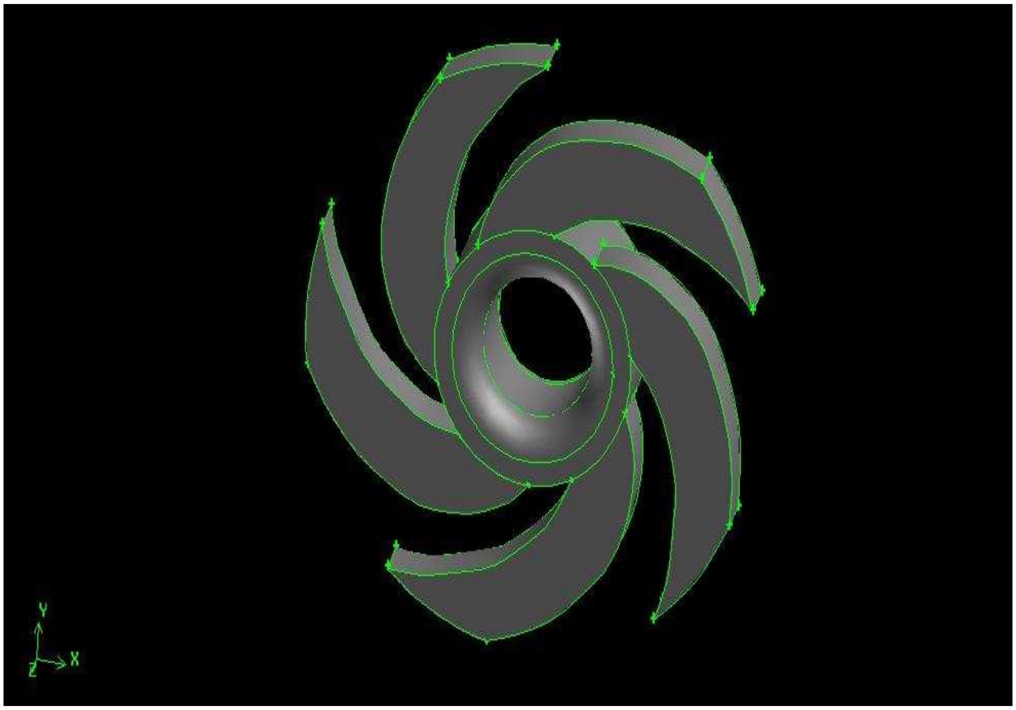


Figure 3.2 (a): Impeller

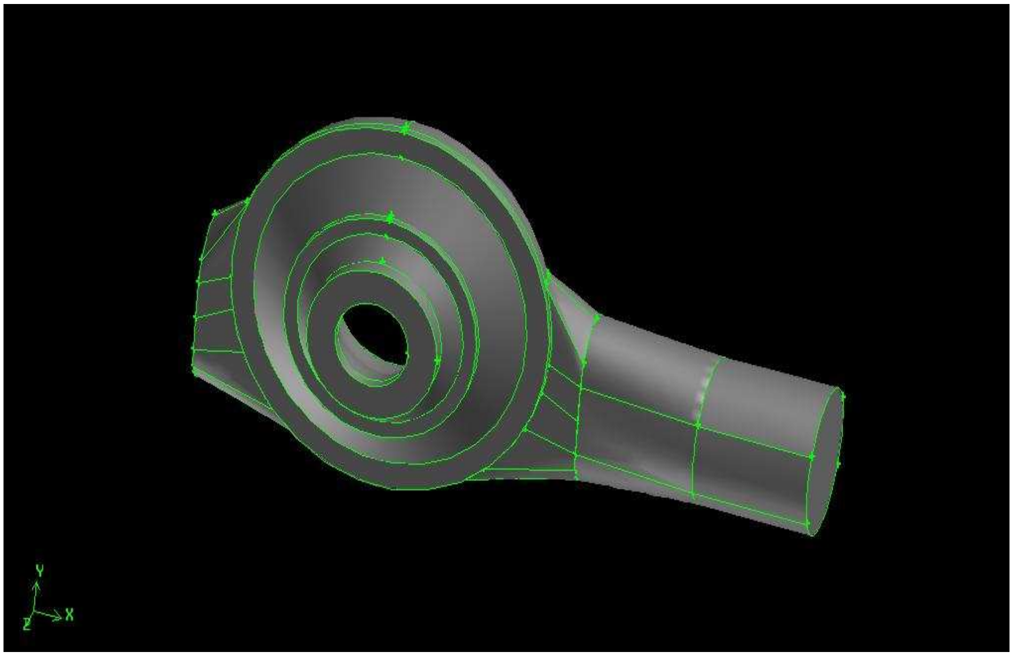


Figure 3.2 (b): Passage

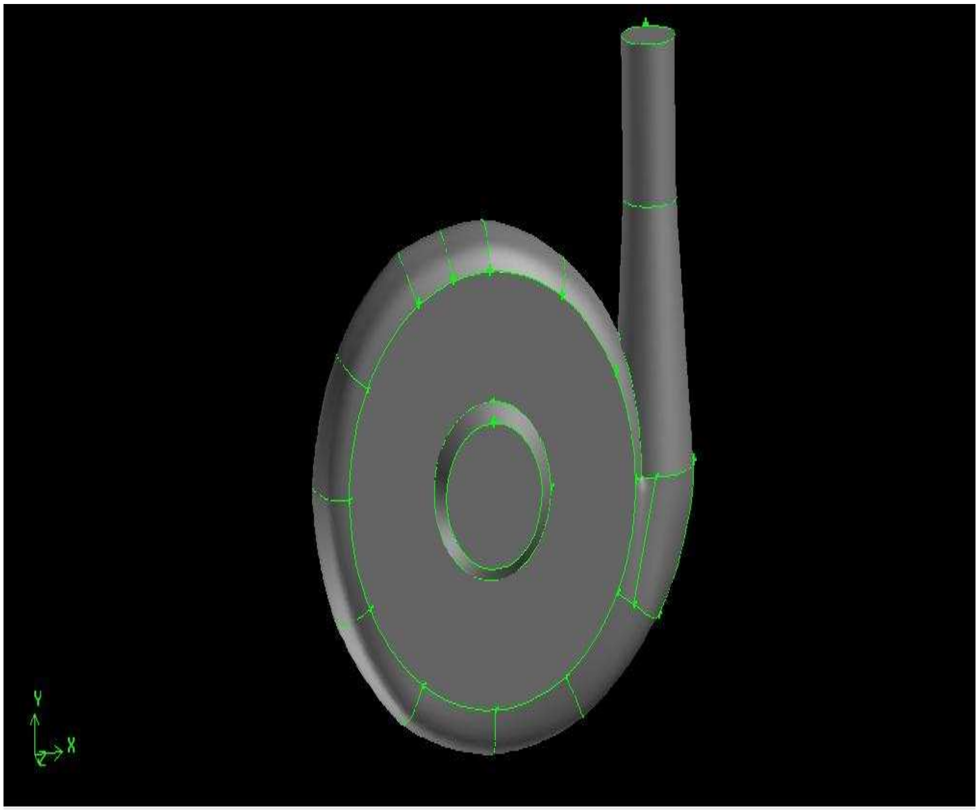


Figure 3.2 (c): Casing

3.6 MESH GENERATION

The next step after modeling the pump components is to descretize it into smaller mesh elements. This process of discretization is called meshing. Meshing can be done by using different types of elements like tetrahedral, hexahedral, wedge, etc. In the present work, coarse tetrahedral and hexahedral mesh with different interval is used in pump domain. The quality of mesh is checked by calculating the equisize skewness, aspect ratio & equiangle skewness.

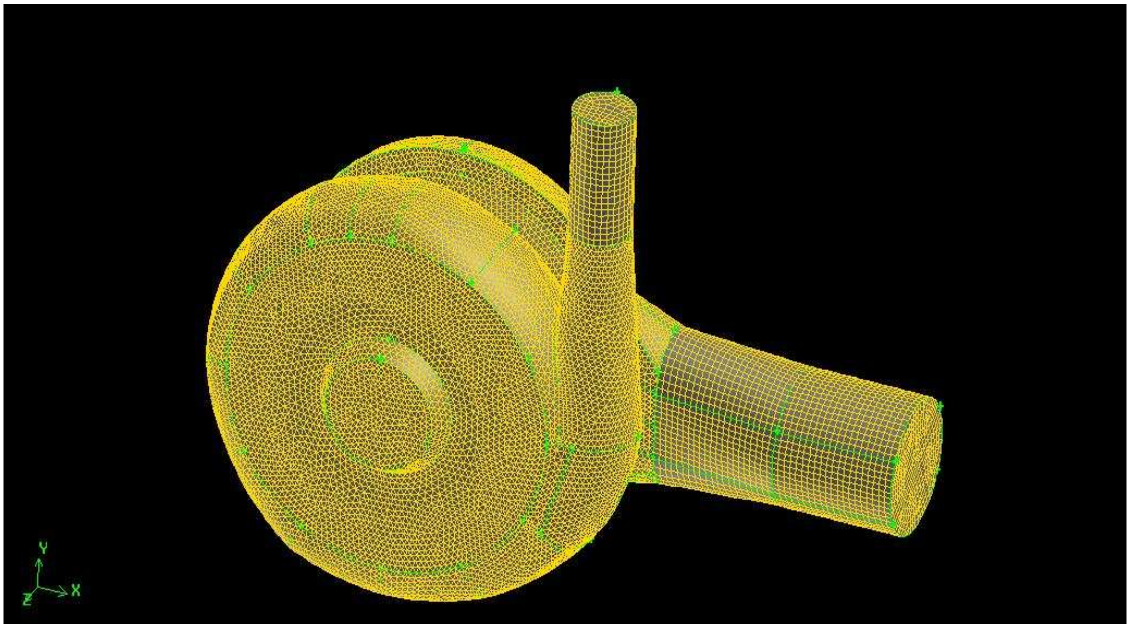


Figure 3.3 : A view of pump with mesh

3.6.1Examine of Mesh

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 'FLUENT' solver is not possible.

3.7 GRID INDEPENDENCE TEST

In the present work three types of mesh size are taken for checking the quality of the grid. The results are then compared at best efficiency point of the pump provided by the manufacturer. It is observed that a coarse mesh size in pump generates 874633 tetrahedral cells and a finer mesh generates 1154702 tetrahedral and hexahedral mixed cells in computational domain. Mesh quality of the centrifugal slurry pump assembly is shown in table 3.1. Table 3.2 show the grid independency test data for present pump simulation.

Sr no.	Mesh type	Parts	Mesh size	No. of element	Quality measuring aspect		
					skewness	Equi angle skewness	Aspect ratio
1	Tetrahedral Hexahedral	Impellor Casing Passage Inlet Outlet	2.5 3.5 4 4 4	446739 363488 267474 32292 3825	0.75	0.8	3.16
2	Tetrahedral mesh1	Impellor Casing Passage	2.5 3.5 4	1154072	0.75	0.79	3.12
3	Tetrahedral mesh2	Impellor Casing Passage	3 4 4	874633	0.75	0.79	3.16

Table 3.1: Grid quality check

Mesh type	Discharge in Kg/s	Head (mwc)
1	15.1	15.8
2	15.1	15.8
3	15.1	15.7

Table 3.2: Grid independence test

3.8 ASSUMPTIONS

The simulation of flow inside the centrifugal slurry pump is done on basis of following basic assumptions:

1. Steady state condition.
2. Incompressible fluid flow.
3. Constant fluid properties.
4. The impeller blades and casing walls are hydraulically smooth.

3.8.1 BOUNDARY CONDITIONS

The boundary conditions are specified as follows:

- Mass flow inlet is given at suction pipe entering section.
- Inlet passage faces, rotating faces of impeller and fixed faces of volute casing considered as wall
- At outlet face of delivery pipe section pressure outlet is applied.
- The suction pipe, impeller passages, volute casing, inlet passage and delivery pipe considered as Fluid zone.

3.8.2 SOLUTION PARAMETERS

1. 3-D double precision solver used to solve for simulation
2. Multiple reference frame technique(MRF) used to simulate the pump performance
3. Clear water used is taken as working fluid
4. Standard K-€ model is used for turbulence modeling.
5. Convergence criteria for continuity, velocity and turbulence parameters was set 10^{-4}
6. First order scheme is used for pressure correction as well as for solving momentum, turbulent kinetic energy and turbulence dissipation rate.
7. A simple scheme is used for pressure velocity coupling
8. To achieve convergence is less time under relaxation factor applied are 0.3 for pressure, 0.7 for momentum equation, 0.8 for turbulence kinetic energy and 0.8 for turbulence dissipation rate.

CHAPTER 4
RHEOLOGICAL PROPERTIES OF FLY ASH

The deposit of a large amount of fly ash and bottom ash discharged from coal-fired power stations is a serious problem. The amount of fly ash is larger than bottom ash. Considerable amount of recycle use is available, mainly byadding fly ash to cement. However, the addition of fly ash to cement is limited because the production rate of cement isleveled off, and also the concentration of fly ash in cementis limited. Thus, it is expected that large amount of fly ashmust be transported from power station to waste materialdeposit site in the near future. The deposit site is not necessarily located near the power station. Long distance transportation of fly ash with reduced cost must be considered.As a possible technique of fly ash transportation, hydraulictransportation of fly ash-water slurry is investigated in thisstudy. The design of a pipeline transportation system mustbe accomplished by means of the design procedure based onprecise knowledge of rheological behavior of dense fly ashslurries.

4.1 FLY ASH

Fly ash is one of the residues generated in combustion, and comprises the fine particles that rise with the flue gases. Depending upon the source and makeup of the coal being burned, the components of fly ash vary considerably, but all fly ash includes substantial amounts of silicon dioxide (SiO_2) (both amorphous andcrystalline) and calcium oxide (CaO) etc. In the past, fly ash was generally released into the atmosphere, but pollution control equipment mandated in recent decades now requires that it be captured prior to release.

In some cases, such as the burning of solid waste to create electricity, the fly ash may contain higher levels of contaminants than the bottom ash and mixing the fly and bottom ash together brings the proportional levels of contaminants within the range to qualify as nonhazardous waste in a given state, whereas, unmixed, the fly ash would be within the range to qualify as hazardous waste.

Fly ash is generally captured byelectrostatic precipitators or other particle filtration equipments before the flue gases reach the chimneys of coal-fired power plants, and together withbottom ash removed from the bottom of the furnace is in this case jointly known as **coal ash.**

Figure 4.1 Fly ash

4.2 PROPERTIES OF FLY ASH

4.2.1 PHYSICAL PROPERTIES

Fly ash has spherical particles with a very fine and smooth surface texture.Fly ash is composed predominantly of small, glassy, hollow particles with low to medium bulk density with an average diameter of <10 mm, high surface area and light texture which are aggregated into micron and sub-micron spherical particles of sizes ranging from 0.01 to 100 mm, with smaller particles entrapped within large spheres .Approximately 90-99% of fly ash consists of Si, Al, Fe, Ca, Mg, Na and K. Major matrix elements in fly ash are Si and Al. Fly ash contains all naturally-occurring elements.

4.2.2 CHEMICAL PROPERTIES

Fly ash has a high amount of silica and alumina in a reactive form. These reactive elements complement hydration chemistry of cement. When cement reacts with water, the hydration of cement begins. On hydration, cement produces C-S-H Gel. This C-S-H Gel binds the aggregates together and strengthens the concrete. However, one more compound is produced on hydration that is so different in behaviour. It is none other than the Calcium Hydroxide Ca(OH)2. In construction industry, it is generally referred to as Free Lime. Aggressive environmental agents like water, sulphates,CO_2 attack this free lime leading to deterioration of the concrete.

It is not only the chemistry provided by fly ash that compliments chemistry of cement, but also the physical properties of fly ash improve the rheology and microstructure of concrete by a great extent. Fly ash, on itself, cannot react with water, it needs free lime, produced on hydration of Portland cement, to trigger off its Pozzolanic effect. Once it is triggered, it can go on and on. In simple words, it means a much longer life for concrete structure.

4.3 BENCH 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 variation in the size of the particles 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 namely sieve analysis and hydrometer analysis, are employed to get this distribution. For coarser particle size distribution i.e. above 75 mm, sieve analysis is used whereas hydrometer analysis is used for finer particles i.e. below 75 mm. A known weight of representative sample of solid particles is taken and washed over a B.S. 200 mesh (75 mm). Both 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.

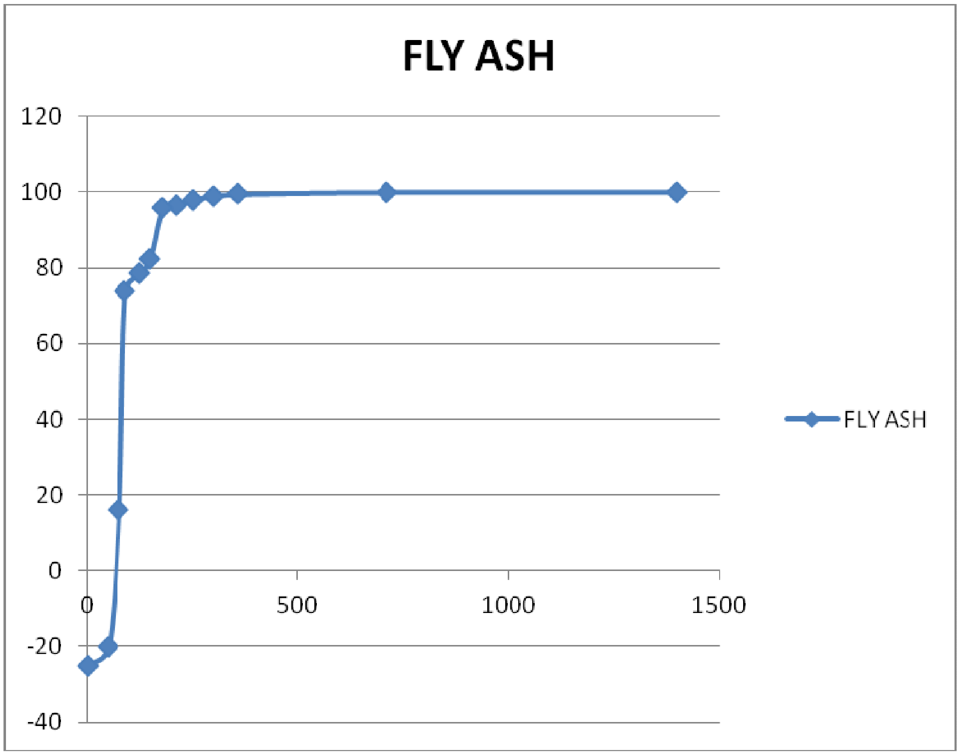


Figure 4.6: Size of particles v/s weight

4.3.2 STATIC SETTLED CONCENTRATION

The static settled concentration depicts the highest limit of solid concentration, which can be achieved by gravitational settling. The static settled concentration depends on a large number of parameters like specific gravity, shape and size distribution of particles, density and viscosity, of carrier fluid etc.

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 jar till the level of the solids become constant. This value of solid concentration in the settled portion of slurry is the static settled concentration. The slurry level at regular intervals of time was also recorded during the process of settling of the slurry to determine the settling rate of the slurry.

In this test take 480 gram of solid (fly ash) in jar and put 1120 gram water in jar to make the 30% concentration by weight. Then leave the jar for 2 hour. After 2 hours block the jar opening by one hand and vigorously mix the mixture and keep jar upright. Note the reading level of sand (fly ash) w.r.t time (sec) in each 10 second. Note the reading till you get steady static state. Calculate the static settled concentration, for the volume and specific gravity of same. Then plot concentration v/s time to get static concentration. The experimental results of Static Settled Concentration of fly ash is shown below.

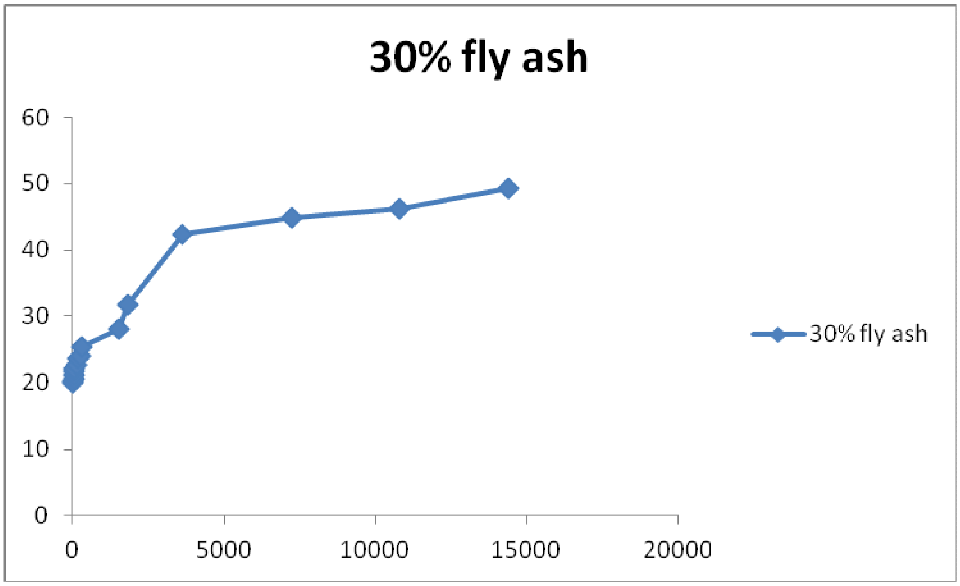


Figure 4.3 :- Static Settling Characteristics of fly Ash(30%)

4.3.3 pH 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.

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.

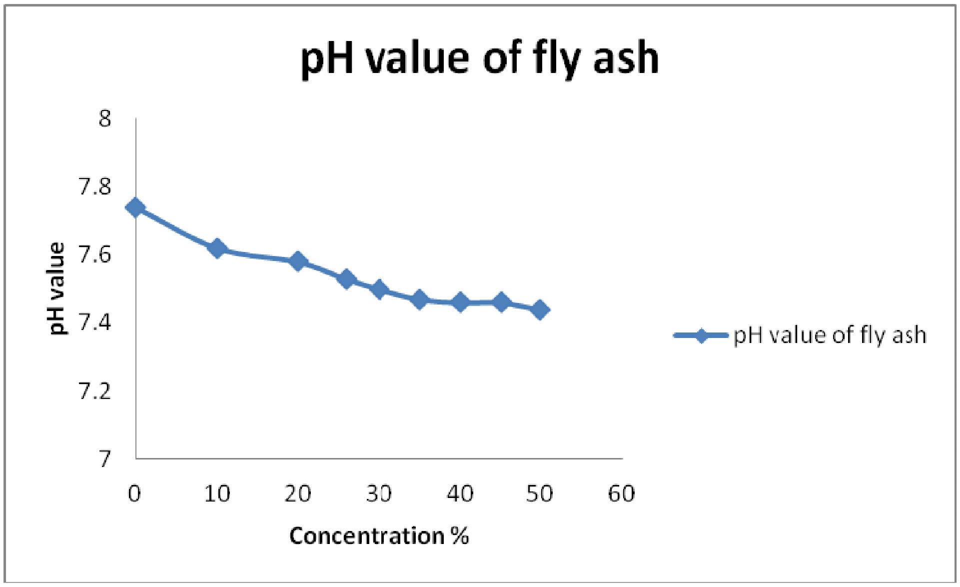


Figure 4.4 pH value of fly ash at different concentrations

4.3.4 RHEOLOGICAL BEHAVIOR OF SOLID-LIQUID MIXTURE

In the estimation of energy required for transporting solid particles using centrifugal pumps, viscosity of the slurry is a considerable parameter. Mixing of solid particles affects the viscosity of 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 fly ash to establish the rheological behavior of the mixtures.

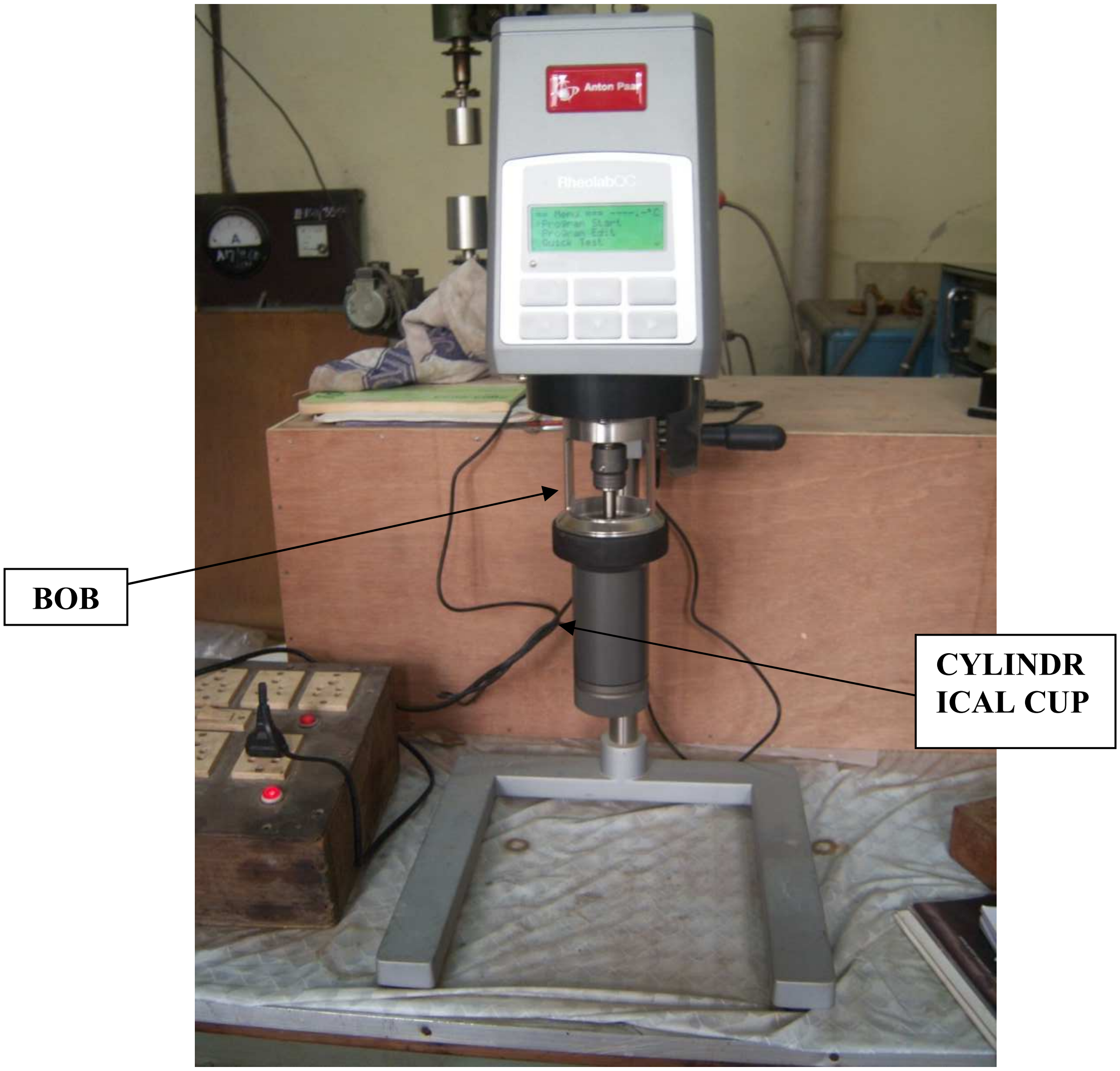


Figure 4.5: Setup of Rheometer

The Rheometer, (Make: Anton Paar, Gurgoan) shown in figure 4.5 has been used for determination of rheological characteristics of the slurries. A particular cylindrical geometry (called double gap) is used for particular type of fluid depending upon particle size of fluid. Slurry experiences a centrifugal force and thus a shearing action with the rotation of the bob. Before conducting tests on rheometer, the bob and cup assembly is done using a locking device and slurry is added into cup (cylinder) up to the particular mark. In this Rheometer, the experiment is done CSR (Controlled shear rate) and CSS (Controlled shear stress) basis. Most of the experiments are done on CSR (controlled shear rate) in which we input the particular shear rate [s^{-1}] and accordingly get the viscosity of particular concentration. The viscosity, shear stress and shear rate are recorded for each combination of shear rate and particular time intervals.

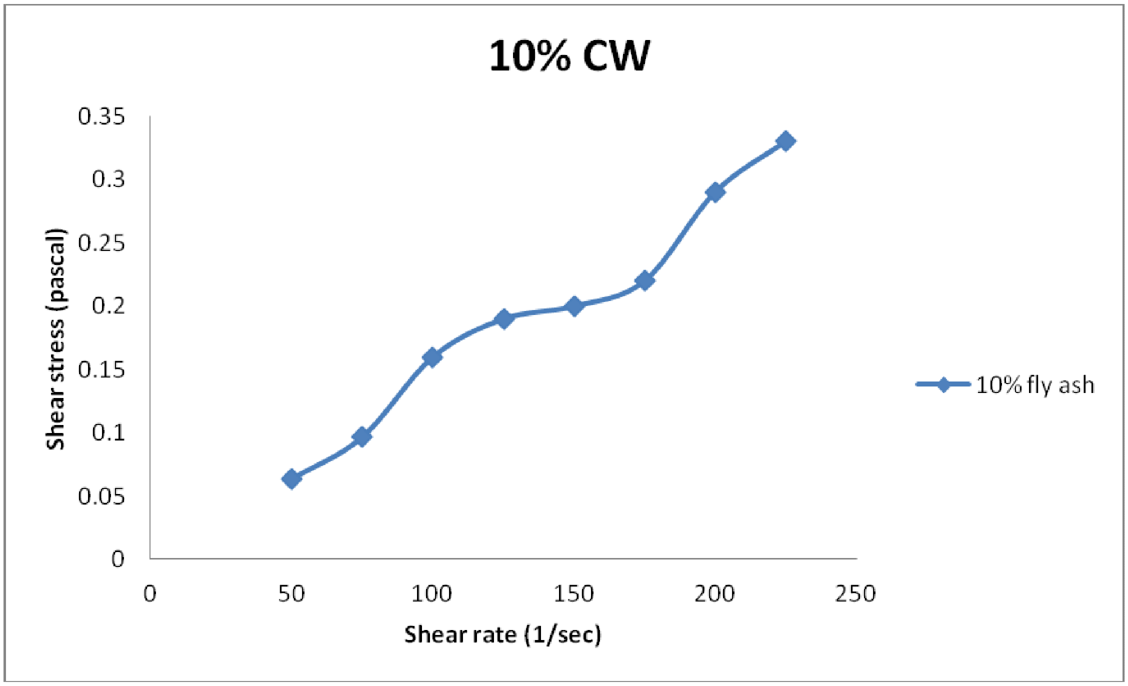


Figure 4.6 (a) Variation of shear stress of fly ash(10%)v/s shear rate

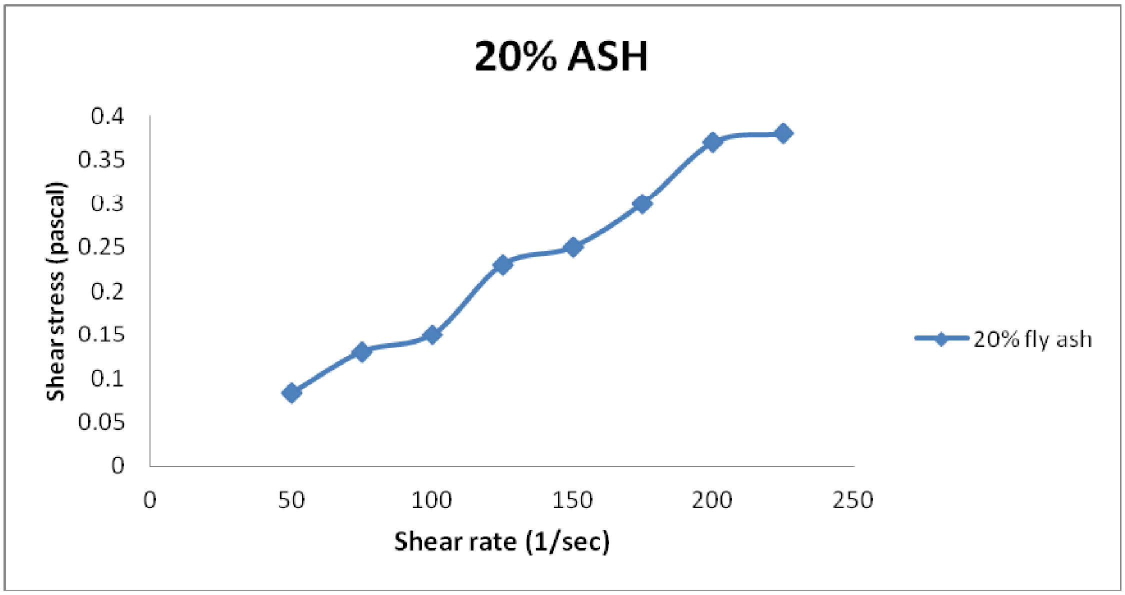


Figure 4.6 (b)Variation of shear stress of fly ash(20%)v/s shear rate

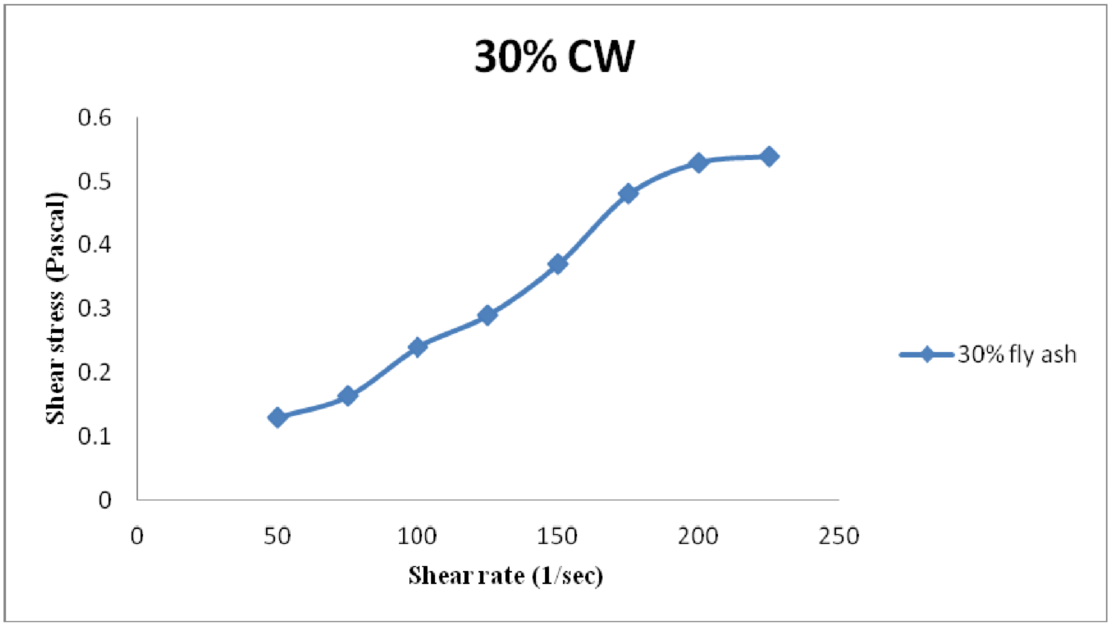


Figure 4.6 (c)Variation of shear stress of fly ash (30%)v/s shear rate

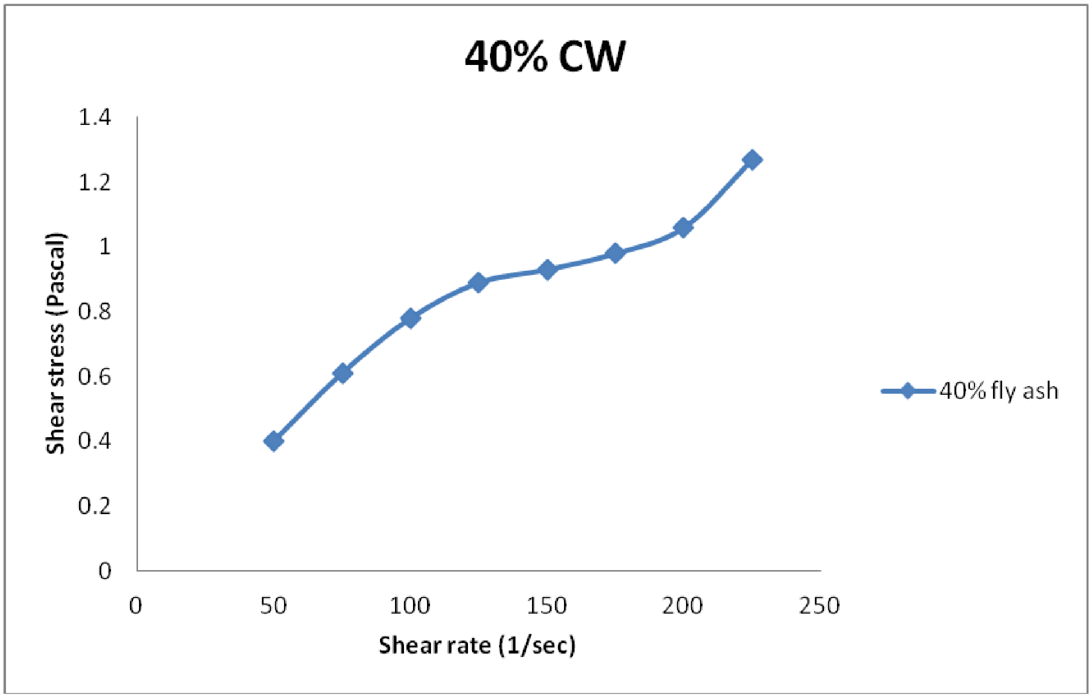


Figure 4.6 (d)Variation of shear stress of fly ash (40%)v/s shear rate

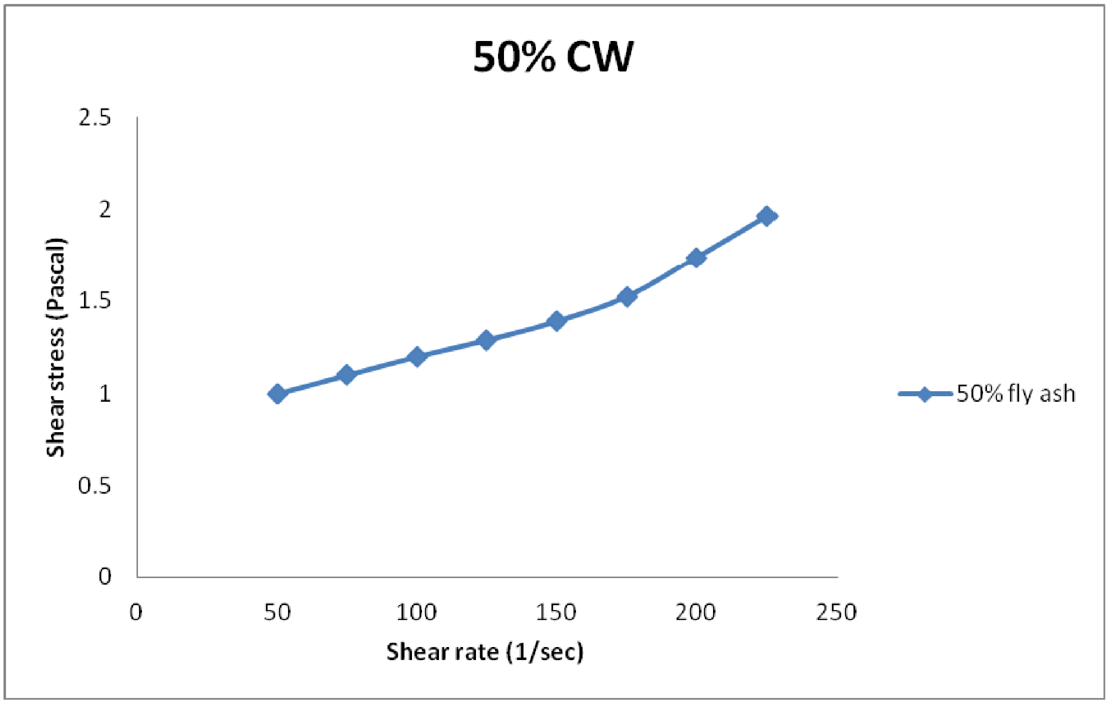


Figure 4.6 (e)Variation of shear stress of fly ash (50%)v/s shear rate (1/sec)

CHAPTER 5

NUMERICAL SIMULATION OF PUMP PERFORMANCE HANDLING FLY ASH

Performance of centrifugal slurry pump is mainly influence due to solid particle size; solid concentration and solid properties. There are various methods for prediction of performance such as loss analysis approach, artificial neutral network approach and various numerical methods. This chapter is dealing with the performance characteristics of centrifugal pump with water and slurry of fly ash and water at 10%, check suitability of affinity relation, testing procedure of centrifugal slurry pump, and artificial neutral network approach.

5.1 BOUNDARY CONDITIONS

The pump has various components like casing, passage, impeller and some interfaces and walls. The pump inlet was defined as mass flow rate boundary condition and pressure outlet was given at the pump outlet. The pressure outlet condition is appropriate when the flow exits at atmospheric pressure and back pressure is created in the flow and is best suited for pumps. The outer surfaces such as walls of casing and inlet passage were given wall boundary conditions. Following boundary types are considered for numerical simulation.

- 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.
- Solid zone: It is applied to impeller solid part.

5.2 SOLUTION TECHNIQUE

The multiple reference frame (MRF) model is used to simultaneously simulate flow through rotating impeller and stationary volute casing. The impeller is modeled in a rotating reference frame and suction pipe, inlet passage and casing are kept as stationary reference frame. The mixing multiphase model is used to simulate slurry transportation through pump and it uses a single fluid approach.

MRF is a steady-state approximation in which individual cell zones move at different rotational/translational speeds. This approach is appropriate when the flow at the boundary between these zones is nearly uniform. While the multiple reference frame approach is clearly an approximation, it can provide a reasonable model of the time-averaged flow for many applications. For example, the MRF model can be used for a turbo machinery application in which rotor-stator interaction is relatively weak. In mixing tanks, since the impeller-baffle interactions are relatively weak, large-scale transient effects are not present and the MRF model can be used. In general, any problems where transients due to rotor-stator interaction are small are candidates for the MRF model. Another potential use of the MRF model is to compute a flow field that can be used as an initial condition for a transient sliding mesh calculation. This eliminates the need for a startup calculation.

Following techniques are used for simulation of centrifugal slurry pump handling 10% concentrated (by weight) fly ash slurry.

- The most appropriate numerical scheme for the flow equations is segregated implicit solver.
- The standard k-ε model is used for turbulence modeling with standard wall function. The standard k- ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε).
- The mixture multiphase model is selected for slurry transportation and its phases are selected as phase-1 is water and phase 2 is fly ash slurry at any specific volume fraction.
- The mixture moment equation is solved using first-order-upwind-scheme.
- The first-order-scheme is used for pressure correction.
- The volume fraction equation is solved using first-order-upwind-scheme.
- SIMPLE type algorithm is used for pressure velocity coupling

Under relaxation factor are chosen as 0.3 for pressure, 0.7 for momentum equation, 0.8 for turbulent kinetic energy and 0.8 for turbulent dissipation rate. These values are selected for fast convergence of solution and better results. Residual tolerance 0.01 is used for convergence of velocity and turbulence parameters.

5.3 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 10% concentration at 1400 rpm with different mass flow rates. 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 behavior across different pump sections are velocity vectors and static pressure contours.

Velocity vectors: - Velocity vectors may be help to identify directional motion of fluid particles in impeller domain. It can be seen internal circulation and separation zones in impeller vane channels. The velocity vectors and static pressure contours at different flow rates are shown in Figure 5.1- 5.4. As the flow entering the impeller eye, it is diverted into the blade-to-blade passage. Due to the unsteady effect developed at upstream, the flow entering the passage is no longer tangential to the leading edge of impeller blade. Separation of flow can be observed at all passages leading edge.

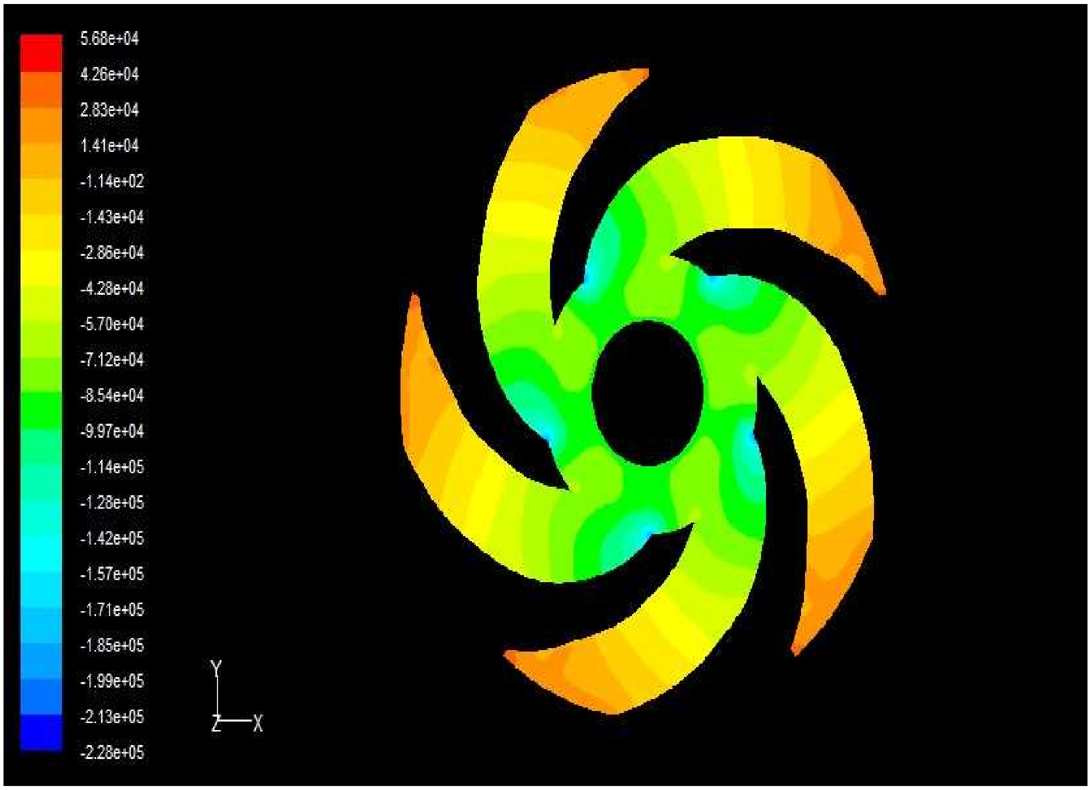


Figure 5.1: static pressure contours at impeller (N= 1400 rpm, Q=16.667 lps)

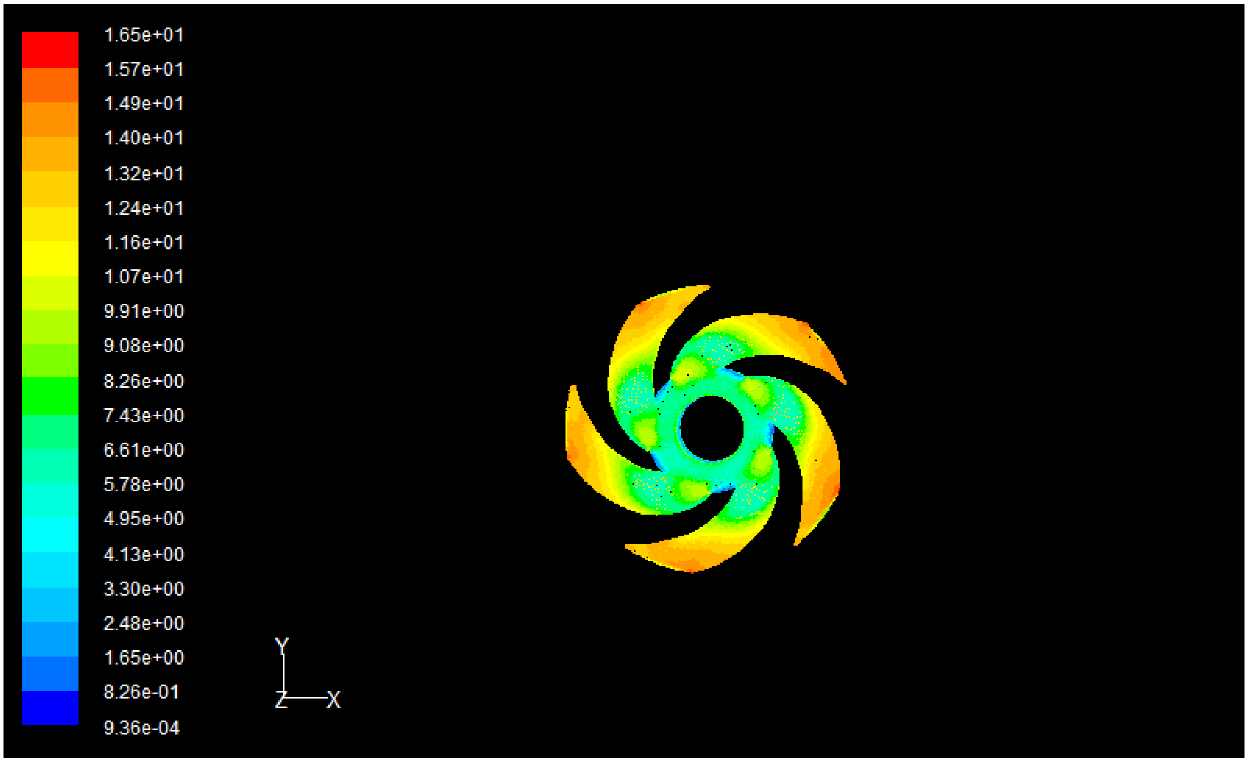


Figure 5.2: velocity vectors at impeller (N=1400 rpm, Q= 0.0 lps)

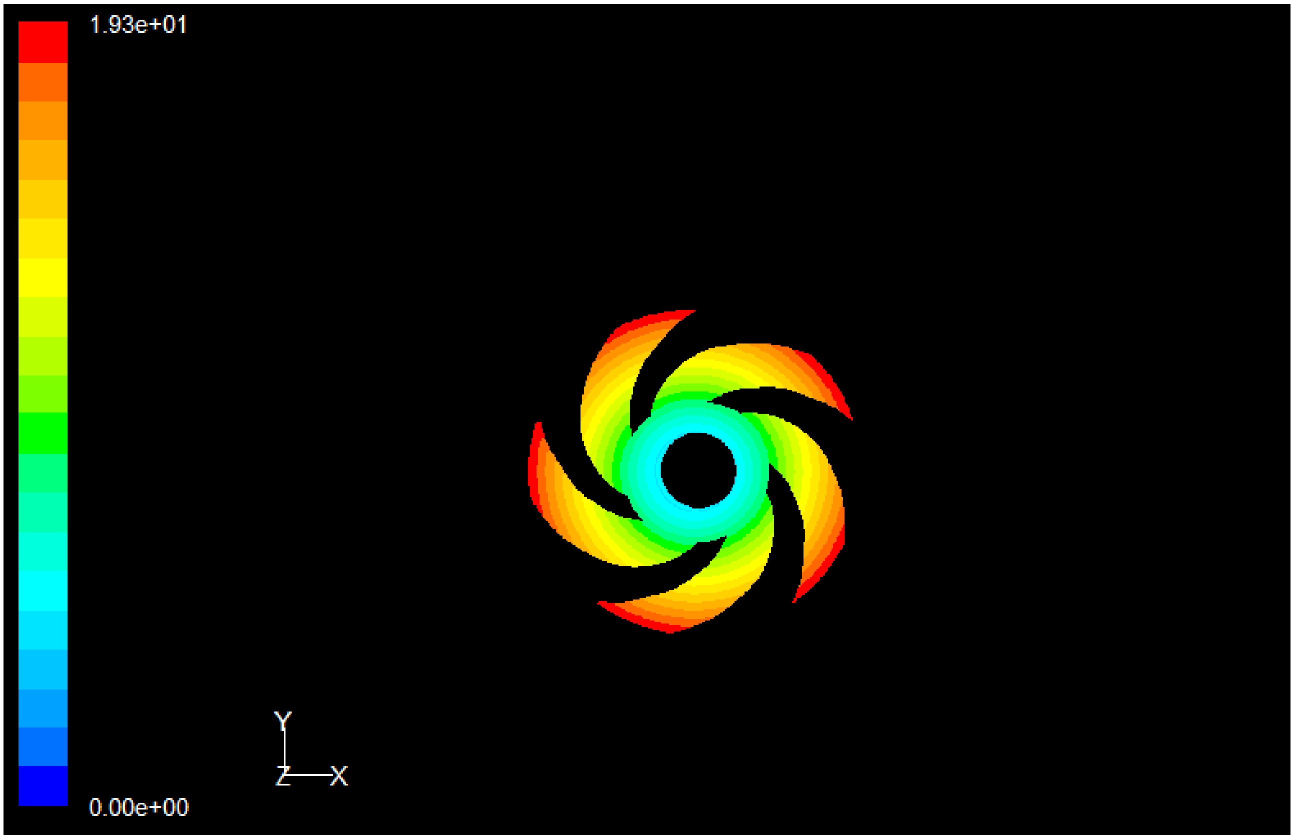


Figure 5.3: velocity contours at impeller (Q= 0.0 lps, N= 1400 rpm)

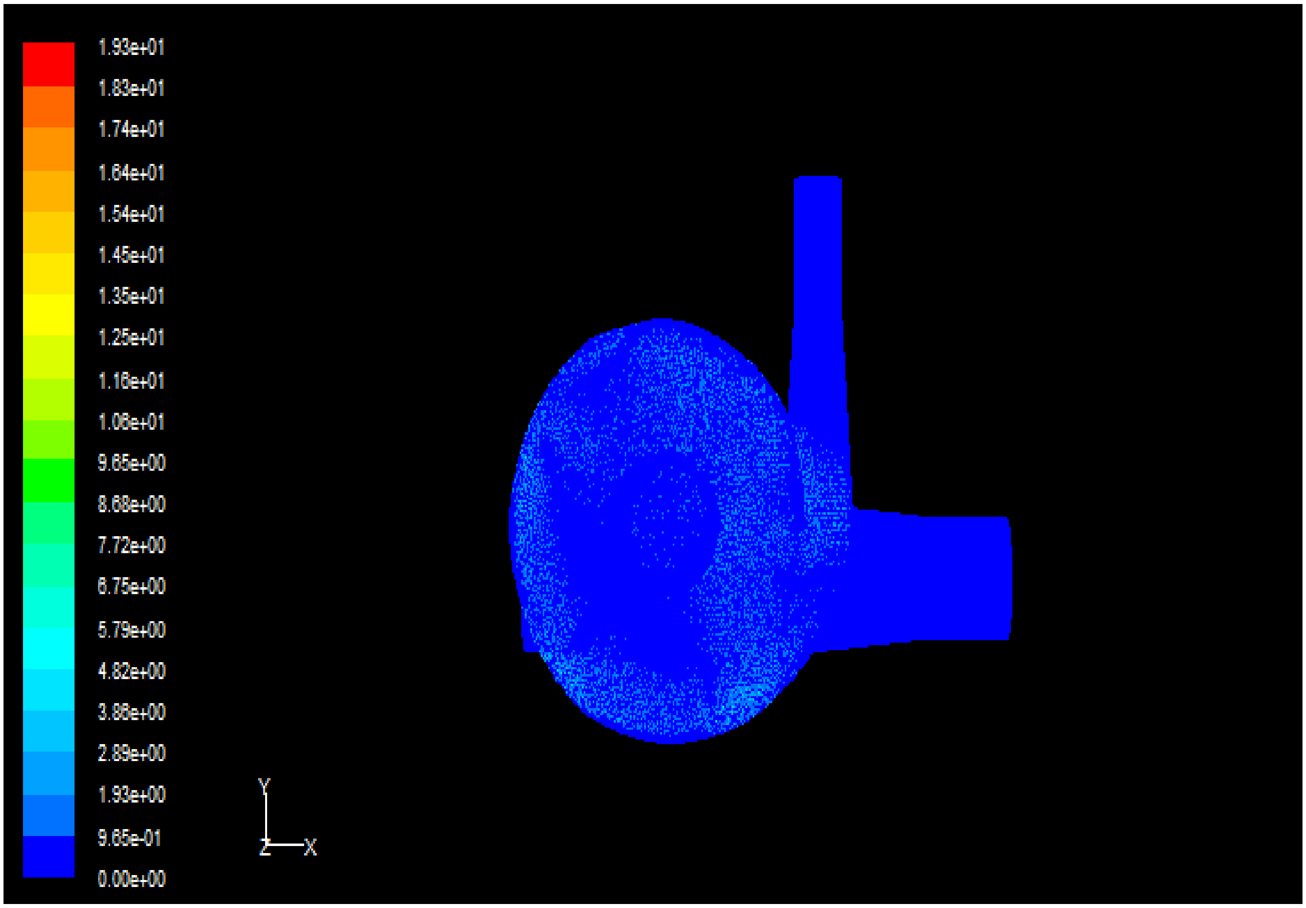


Figure 5.4: Velocity vectors contours at the pump (Q= 0.0 lps, N= 1400 rpm)

5.4 PERFORMANCE CHARACTERISTICS OF PUMP

1. The performance characteristic of the centrifugal slurry pump is predicted numerically using water and fly ash slurry. The solid concentration of fly ash in mixture is kept at 10% by weight. Head characteristics of the pump are predicted by CFD analysis at 1400 rpm, 1250 rpm, 1100 rpm and 900 rpm with water and the same results are taken for fly ash at 1400 rpm with 10%. The flow rate less than BEP, the flow field inside the pump is visualized at different flow rates. It is observed that the internal circulation at the vane outlet is developed due to some of the flow coming back into the impeller. The intensity of the internal circulation is found to be reducing with the increase in the flow rate as shown in the Figure 5.1 - 5.4.

2. At BEP flow rate, 15.1lps, the impeller passage flow at design point is very smooth and internal circulation at impeller vane outlet is completely disappeared. The flow follows the blade curvature profile from impeller passage entrance till the exit without any separation on blade pressure side.
3. The flow rate higher than BEP, 16.67 lps, the non-uniformity of the flow rates through the different vane channels increases and therefore shock losses in the volute casing increases.

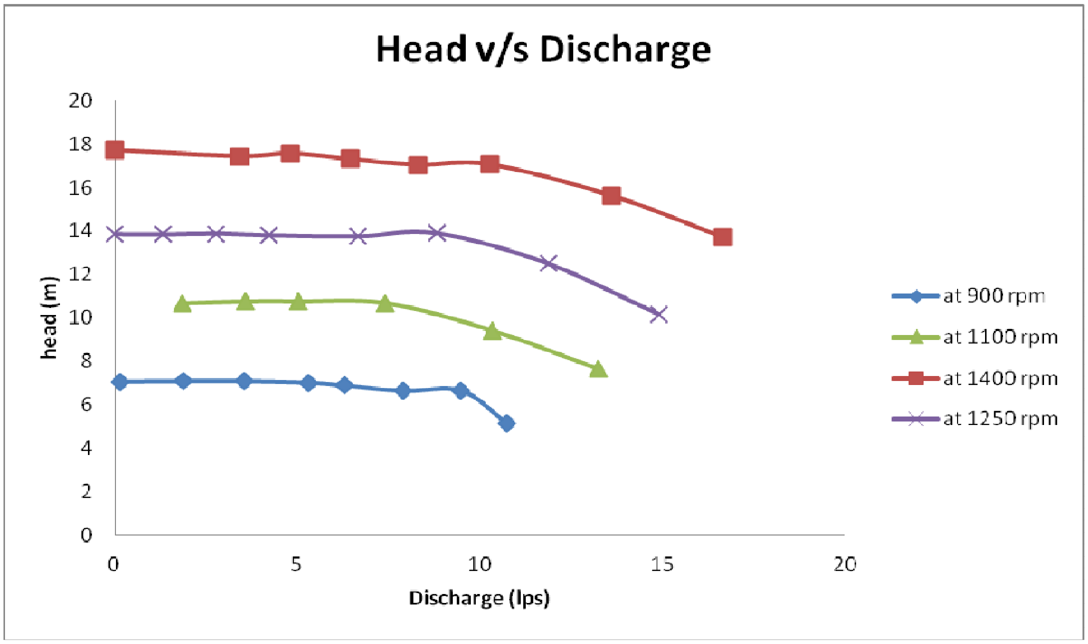


Figure 5.5: Discharge (lps) v/s head (m)

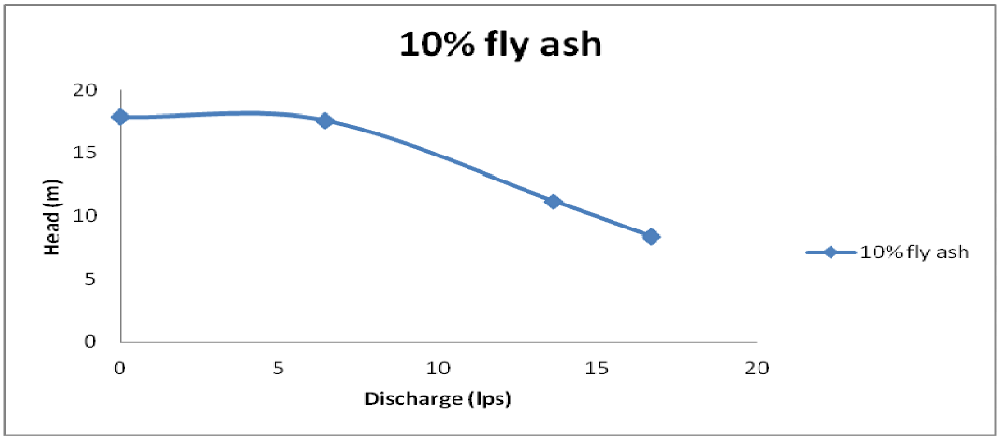


Figure 5.6: head v/s discharge (10% fly ash slurry)

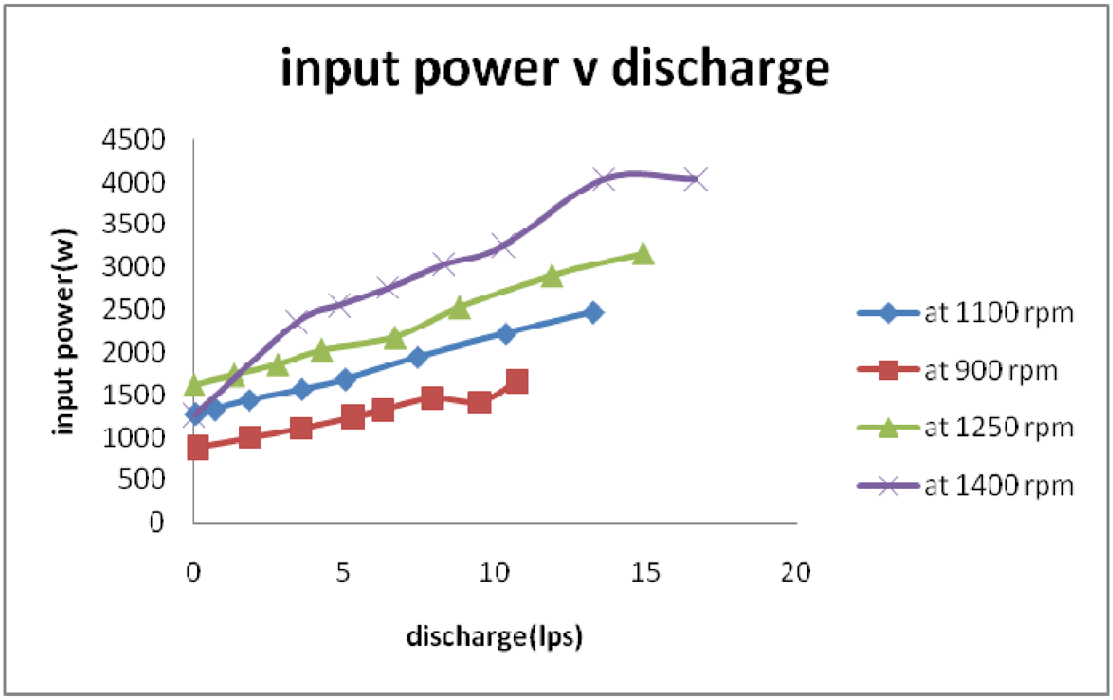


Figure 5.7: Discharge v Input power (for various mass flow rates)

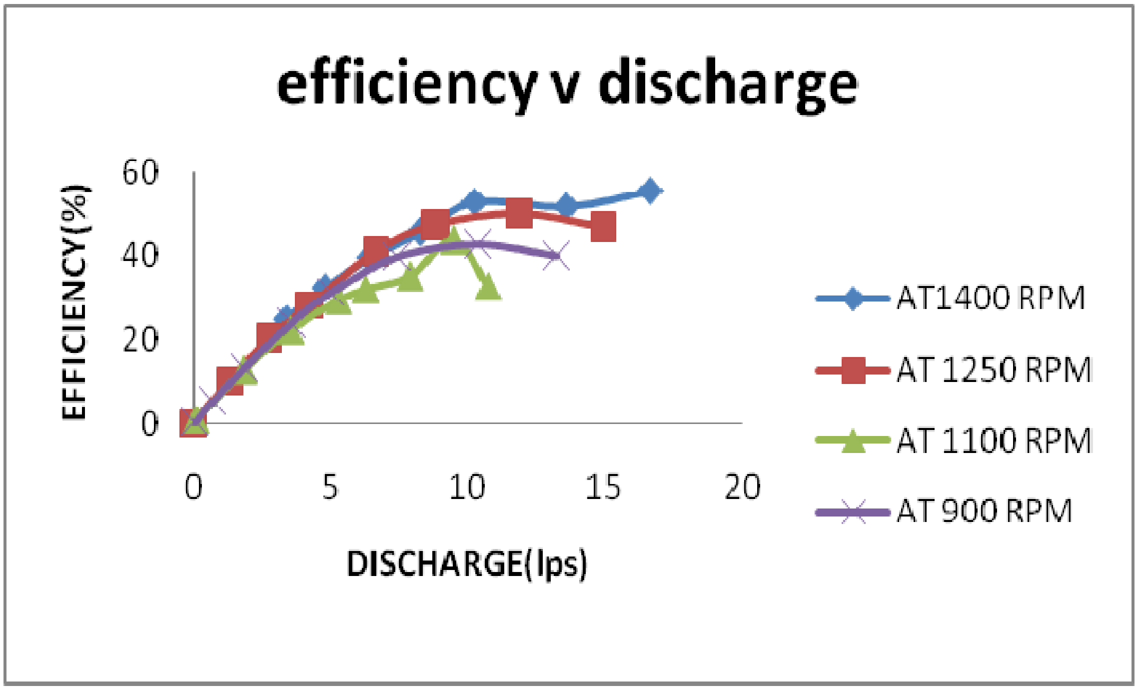


Figure 5.8: Efficiency v Discharge (for various mass flow rates)

CHAPTER-6

RESULTS, DISCUSSIONS & FUTURE SCPOE

A conventional centrifugal pump is designed to handle clear liquids. However when slurries are to be transported the conventional centrifugal pump has to be modified to handle solid liquid mixtures. 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. Conventional design method of centrifugal pump are largely based on the application of empirical and semi-empirical rules along with the use of available information in the form of different types of charts and graphs as proposed by successful designers. As the design of centrifugal pump involve a large number of interdependent variables, several other alternative design are possible for same duty. 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. The geometry of pump components impeller, casing, inlet passage and frame and follower plate is modeled using GAMBIT and PRO-E. The mesh is generated successfully using GAMBIT. Complex internal flow field, pressure and velocity distribution investigated using FLUENT commercial computational code. The simulation results are obtained at the operating speed 1400, 1250, 1100,900 rpm with different mass flow rates for transportation of clear liquid. The Simulation was performed by using k-Epsilon simulation modeling scheme. The performance results show that total static head is the function of the mass flow rate with constant operating speed. Numerical performance results of different turbulent modeling are compared with the experimental results at the same operating conditions.

Future Scope

1. Pressure and velocity distribution for pump impeller and casing can be calculated for bottom ash slurry.
2. Numerical simulation of centrifugal slurry pump can be evaluated by considering losses.
3. Similar computational simulation models can also be used for analyzing the pressure, velocity and stress distribution of the turbines, compressor, fan and blower.

REFERENCES

1. Eric Dick, Jan Vierendeels, Sven Serbruyns and John Vande Voorde, (2001) “Performance prediction of centrifugal pumps with CFD tools”. Task quarterly 5 No 4 (2001), 579–594, tq0405e7/580 26 I 2002 BOP s.c., http://www.bop.com.pl.

2. Jose´ Gonza´lez, Joaquı´n Ferna´ndez, Eduardo Blanco, Carlos Santolaria (2002) “Numerical Simulation of the Dynamic Effects Due to Impeller-Volute Interaction in a Centrifugal Pump”. Vol. 124, JUNE 2002 Copyright © 2002 by ASME Transactions of the ASME.

3. Weidong Zhou, Zhimei Zhao, T. S. Lee, and S. H.Winoto (2003) “Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamics”. International Journal of Rotating Machinery, 9(1): 49–61, 2003 Copyright °c 2003 Taylor & Francis 1023-621X/03 $12.00 + .00 DOI: 10.1080/10236210390147380.

4. K M Guleren and A Pinarbasi (2004) “Numerical simulation of the stalled flow within a vaned centrifugal pump”. Proc. Instn Mech. Engrs Vol. 218 Part C: J. Mechanical Engineering Science.

5. Miguel Asuajea, Farid Bakira, SmaÏne Kouidri‡a, Robert Reya (2004) “Inverse Design Method for Centrifugal Impellers and Comparison with Numerical Simulation Tools”. International Journal of Computational Fluid.
   Dynamics, 18: 2, 101 — 110.

6. G. R. Addie,A. Sellgren, (2005) “A First Slurry Pump Standard and some implications for paste systems.”GIW Industries, USA, Lulea University of Technology, Sweden.

7. John S. Anagnostopoulos (2006) “CFD Analysis and Design Effects in a Radial Pump Impeller”.Wseas Transactions on fluid mechanics. Issue 7, Vol. 1, July 2006 ISSN: 1790-5087

8. José González, Carlos Santolaria (2006) “Unsteady Flow Structure and Global Variables in a Centrifugal Pump”.Journal of Fluids Engineering Copyright © 2006 by ASME SEPTEMBER 2006, Vol. 128 / 937

9. M. H. ShojaeeFard, F. A. Boyaghchi and M. B. Ehghaghi (2006) “Experimental Study and Three-Dimensional Numerical Flow Simulation in a Centrifugal Pump when Handling Viscous Fluids”. IUST International , Journal of Engineering Science, Vol. 17, No.3-4, 2006, Page 53-60

10. Si Huanga, Mohammed F. Islamb, Pengfei Liu (2006) “Numerical simulation of 3D turbulent flow through an entire stage in a multistage centrifugal pump”. International Journal of Computational Fluid Dynamics, 20: 5,309 — 314.

11. A. Benretam, A. Haddouche, H. Cheghib & S. Saad (2007) “Influence of solid particles on centrifugal pump Characteristics”. Journal of Engineering & applied Sciences 2(1): 244-247,2007.

12. L. Pullum, L.J.W. Graham, and M. Rudman (2007) “Centrifugal pump performance calculation for homogeneous and complex heterogeneous suspensions”.The Journal of The Southern African Institute of Mining and Metallurgy, Volume 107, Referred paper, June 2007

13. Adnan Ozturk, Kadir Aydin, Besir Sahin and Ali Pinarbasi (2009) “Effect of impeller-diffuser radial gap ratio in a centrifugal pump”. Journal of Scientific & Industrial Research Vol. 68, March 2009, pp.203-213

14. Khin Cho Thin, Mya Mya Khaing, and Khin Maung Aye, (2008) “Design and Performance Analysis of Centrifugal Pump”. World Academy of Science, Engineering and Technology 46 2008.

15. Kamal El-Nahhas , Nageh Gad El-Hak, Magdy Abou Rayan and Imam El-Sawaf ,(2009) “Effect of particle size distribution on the hydraulic transport of settling slurries”. Thirteenth International Water Technology Conference, IWTC13 2009, Hurghada, Egypt.

16. Mohamed F. Khalil, Sadek Z. Kassab, Ashraf S. Ismail,and Ibrahim S. Elazab (2008),“Centrifugal pump performanceunder stable and unstable oil-water emulsions flow”. Twelfth International Water Technology Conference, IWTC12 2008, Alexandria, Egypt.

17. A. Sellgren, G. Addie, R. Visintainer and K. Pagalthivarthi (2004) , “Prediction of slurry pump component wear and cost”. V.P. Engineering, Research and Development, GIW Industries Inc., Grovetown, GA, USA

18. John M. Sankovic, Jaikrishnan R. Kadambi and Mehul Mehta, William A. Smith, Mark P. Wernet (2004), “PIV Investigations of the Flow Field in the Volute of a Rotary Pump”. NASA/TM—2004-212617 February 2004 FEDSM2003–45490.

19. Krishnan V. Pagalthivarthi, Pankaj K. Gupta, Vipin Tyagi, M. R. Ravi,” CFD Predictions of Dense Slurry Flow in Centrifugal Pump Casings”. International Journal of Aerospace and Mechanical Engineering 5:4 2011.

20. H Chen, W Liu, W Jian and P Wei, “Impellers of low specific speed centrifugal pump based on the draughting technology”. 25th IAHR Symposium on Hydraulic Machinery and Systems IOP Publishing IOP Conf. Series: Earth and Environmental Science 12 (2010) 012018 doi:10.1088/1755-1315/12/1/012018.

ANNEXURE I

Table1.1 Variation of head v/s discharge of water at different speeds

1400 RPM		1250 RPM		1100 RPM		900 RPM	
Discharge	Head	Discharge	Head	Discharge	Head	Discharge	Head
16.669	13.71	14.92	10.17	13.25	7.66	10.737	5.14
13.623	15.62	11.886	12.5	10.354	9.41	9.488	6.65
10.277	17.06	8.811	13.9	7.42	10.67	7.907	6.63
8.301	17.04	6.668	13.75	5.027	10.75	6.287	6.88
6.435	17.31	4.235	13.8	3.584	10.75	5.306	7.01
4.822	17.57	2.785	13.88	1.841	10.66	3.556	7.08
3.415	17.44	1.33	13.84	0.703	10.73	1.876	7.08
0	17.71	0	13.85	0.052	10.81	0.143	7.06

Table 1.2 Variation of head v/s discharge of fly ash slurry(10% by weight) at 1400 RPM

Discharge	0	6.435	13.62	16.667
Head	17.85	17.61	11.19	8.353

ANNEXURE II

Table 1.3 Variation of pH value v/s concentration of fly ash slurry

Concentration	0	10	20	25.9	30	35	40	45	50
pH value	7.74	7.62	7.58	7.53	7.5	7.47	7.46	7.46	7.44

Table 1.4 Variation of Shear stress v/s Shear Rate at different concentrations of fly ash slurry

	10%	20%	30%	40%	50%
Shear Rate (sec^{-1})	Shear Stress	Shear Stress	Shear Stress	Shear Stress	Shear Stress
0	0	0	0	0	0
50	0.064	0.083	0.13	0.4	1
75	0.097	0.13	0.163	0.69	1.1
100	0.18	0.15	0.24	0.78	1.2
125	0.19	0.23	0.26	0.89	1.288
150	0.18	0.25	0.37	0.93	1.39
175	0.22	0.3	0.51	0.98	1.523
200	0.29	0.37	0.45	1.06	1.738
225	0.33	0.38	0.54	1.27	1.96