CFD Modelling and Simulation of slurry through Centrifugal Pump in ASH Handling Plant

Authors

Vishwatesh Raj¹, P.N. Ulhe²

Department of Mechanical Engineering, SSBT'S College of Engineering and Technology,
Bambhori, Jalgaon (M.S)

ABSTRACT:-

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. 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. Pressure and velocity distribution inside impeller of the centrifugal pump has direct influence due to change flow passage.

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.

1.2 Types of Pumps

- Rotary Pump
- Reciprocating Pumps
- Centrifugal Pump

1.3 Centrifugal Slurry Pump Merits

- Simplified design
- Easy installation
- Easy maintenance
- Lesser weight
- Easy transportation of fluids

1.4 Centrifugal Pump for Slurry

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 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

2. LITERATURE REVIEW:-

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.

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.

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.

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.

3. COMPUTATIONAL FLUID DYNAMICS

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)

3.1 Principles of Fluid Motion

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

3.2 CFD Procedure

Figure 3.1 CFD procedure

All CFD components contain main three elements.

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

B. 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
- Finite difference method
- Finite volume method

C. 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

3.3 Computational Modeling

The geometry of impeller, volute casing and inlet passage are modelled by using pre-processor Gambit 2.3.16

Fig:-a) Impeller

Fig: b) Passage

Fig:- c) Casing

3.4 Mesh Generation

The next step after modelling the pump components is to discretize 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:- A view of Pump with mesh

3.5 Mesh Examine

It is important to check the quality of mesh, because parameter such as skewness affects the accuracy of the CFD simulation. Each element has a value of skewness between 0 and 1. The skewness is classified in two ways, Equi-Angle skew and EquiSize skew. The smaller value of equi-Angle 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.

3.6 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 1, Table 2 shows the grid independency test data for present pump simulation.

Table 1:- Grid Quality check

S.No	Mesh Type	Parts	Mesh Size	No. Of Element	Quality Measuring Aspects		
					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	Impeller Casing Passage	2.5 3.5 4	115407 2	0.75	0.79	3.12
3	Tetrahedral mesh2	Impeller Casing Passage	3 4 4	874633	0.75	0.79	3.16

Table 2:- Grid Independence Test

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

4. RHEOLOGICAL PROPERTIES OF FLY ASH

Fly Ash:- Fly ash is one of the residues generated in combustion, and comprises the fine particles that rise with the flue gases. fly ash includes substantial amounts of silicon dioxide (SiO_2) (both amorphous and crystalline) and calcium oxide (CaO) etc.

Fig:- Fly Ash

4.1 Fly Ash Properties:-

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 the Calcium Hydroxide $\text{Ca}(\text{OH})_2$. 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.

Physical Properties:- Fly ash has spherical particles with very fine and smooth surface texture. Fly ash is composed predominately 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 size ranging from 0.01 to 100 mm, with smaller particles entrapped within large spheres.

Bench Scale Tests:-

A. pH value:- The pH suspension was read on the digital display unit when equilibrium value was reached.

Fig:- pH value of fly ash at different concentration

B. Rheological Behavior of Solid-Liquid Mixture:- 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 established the rheological behavior of the mixtures.

Fig- Variation of Shear stress of fly (10%) v/s shear rate

Fig- Variation of Shear stress of fly (20%) v/s shear rate

Fig- Variation of Shear stress of fly (30%) v/s shear rate

Fig- Variation of Shear stress of fly (40%) v/s shear rate

Fig- Variation of Shear stress of fly (50%) v/s shear rate

C. Particle Size Distribution (PSD)

The particle size distribution of the fine particles collected (<75 μ m) is then determined using the standard hydrometer analysis.

Fig- Size of Particles v/s weight

Concentration of Static settled of Particle

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.

Fig- Static Settling Characteristics of fly Ash (30%)

3.2 CFD Solution Technique:-

The multiple reference frame(MRF) model is used to simultaneously simulate flow through rotating impeller and stationary volute casing. MRF is a steady-state approximation in which individual cell zones move at different rotational/translation speeds. This approach is appropriate when the flow at the boundary between these zones is nearly uniform.

3.3 Post Processing of CFD Result:-

The numerically simulated flow field of centrifugal slurry pump was analysed to investigate the performance characteristics of centrifugal slurry pump at 10% concentration at 1400 rpm with different mass flow rates.

Velocity vectors:- Velocity vectors may be help to indentify directional motion of fluid particles in impeller domain. It can be seen internal circulation and separation zones in impeller vane channels.

Fig- Static Pressure contours

At impeller(N=1400rpm, Q=16.667 lps)

Fig- Velocity vector at impeller (N=1400rpm, Q=0.0lps)

Fig-Velocity Contours at impeller (N=1400rpm, Q=0.0lps)

Fig- Velocity vectors contours at the pump (Q=0.0lps, N=1400rpm)

3.4 Performance Characteristics of Pump:-

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%.

Fig- Discharge (lps) v/s head (m)

Fig- Head v/s discharge (10% fly ash slurry)

Fig- Discharge v/s input power (for various mass flow rates)

Fig- Efficiency v/s Discharge (for various mass flow rates)

RESULTS:-

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 modelling scheme. The performance results show that total static head is the function of the mass flow rate with constant operating speed.

Future Scope

Pressure and velocity distribution for pump impeller and casing can be calculated for bottom ash slurry.

Numerical simulation of centrifugal slurry pump can be evaluated by considering losses.

Similar computational simulation models can also be used for analyzing the pressure, velocity and stress distribution of the turbines, compressor, fan and blower.

REFERENCE:-

1. Miguel Asuajea, Farid Bakira, Smaïne Koudriġa, 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.
2. 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.
3. 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.
4. Si Huang, 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.

5. 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 .
6. 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.
7. Eric Dick, Jan Vierendeels, Seven Sebruyns and John Vande Voorde, (2001) "Performance prediction of centrifugal pumps with CFD tools". Task quarterly 5 No 4 (2001),579-594, tq0405e7/58026 I 2002 Copyright© 2002 by ASME Transactions of the ASME.
8. 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.
9. 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.

Annexture-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

Annexture-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