ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 3, Issue 2, February 2018

Numerical Analysis of Fly Ash Slurry Transportation through Centrifugal Pump

[1] Patel Rajkumar, [2] Pravinkumar Hadgekar, [3] Dr. Sunil Chandel
[1] PG Student, [2] Assistant Professor, [3] Assistant Professor,
Department of Mechanical Engineering, Defence Institute of Advanced Technology (DU), Pune (411025), Maharashtra, India

Abstract: -- **In this work, the computational analysis of a centrifugal pump is done using ANSYS CFX. In this work, an effort is carried out on a numerical validation of a transportation of fly ash slurry through centrifugal pump, with results available in the literature. In this work, simulation is conducted at the different concentration of fly ash slurry as well as on water to analyse the flow behavior in a centrifugal pump. This all simulation can be divided into three parts 1st is creating centrifugal pump geometry using Vista CPD tool, in next part meshing is done using the TurboGrid tool and boundary conditions are given. In last part, simulations are carried out at different concentrations (i.e. 60.4% Cw, 65.2% Cw, and 70% Cw) of fly ash slurry. Interface model used for rotor and stator interaction is frozen rotor model (i.e. rotor is impeller and stator is volute). K-Epsilon Turbulence model is used in the simulation. Results of the simulation obtained are in a similar pattern of results available in the literature. The results show that at rated speed the head developer of the pump is reduced with an increase in solid concentration and slurry flow is strongly depends on the viscosity of the slurry.**

Keywords: Centrifugal Pump, Computational Fluid Dynamics, Fly ash slurr0079.

Nomenclature

C_w - concentration of solids by weight
atm - atmospheric pressure
rpm - rotation per minute
u_i - velocity component in particular direction
E_{ij} - rate of deformation
μ_t - eddy viscosity
ρ - density m^3/s
lps - liter per second

I. INTRODUCTION

Slurry flow has been one of the progressive technologies for conveying large quantity of materials over long distances .from researches it has been found that transportation of particulate material as slurry flow is much more economical and easy as well as environment friendly method [1]. Coal ash in thermal power plants and mineral ores in mining and process industries, are a few examples that can be mentioned. Literature has shown that the transportation of solid over long distances by pipelines is generally more economical compared to conventional mode of transportation [1]. Huge amount of coal ash are generated as large amount of coal is burnt in coal based thermal power plant. It is estimated that 12000 tons coal is used per day and it generated 4200 tons ash per day [1]. Ash generated in thermal power plan is mainly two types one is fly ash and the other is bed ash. Fly ash is a fine particulate powder whereas bed ash is course. In current trend use of this coal ash as raw materials for cement making, bricks making are also a developing sectors and use of coal ash as raw material is also increasing day by day. There for it becomes necessary to study slurry flow behavior for optimization of slurry transportation at optimal cost. In Indian thermal power plant mostly centrifugal pump is used as coal ash slurry transportation [1]. Centrifugal pump is a hydraulic machine which converts mechanical energy into pressure energy it is the second most used device in industry after electric motor, so that we can understand the importance of centrifugal pump in industry [2]. In centrifugal pump fluid is transported by means of centrifugal force acting on fluid through its impeller. Centrifugal pump works on a principle of forced vortex flow which means that when a certain mass of fluid is rotated by an external torque the pressure head of the fluid is increased it consist mainly two parts one is impeller and the second is volute casing. Impeller consists of a series of backward curved vanes and volute is a casing which covers the impeller and its shape is spiral type.

Analytical calculation of the slurry flow in pumping system is difficult and time consuming due to its complex shape of fluid flow passage [3]. Also many other factors has to be considered while analysing flow behavior of fluid in pumping system so analytical method of analysing flow behavior of fluid flow may give sometimes incorrect values.so to eliminate this researchers are turned to numerical methods as advancement in computing technology.

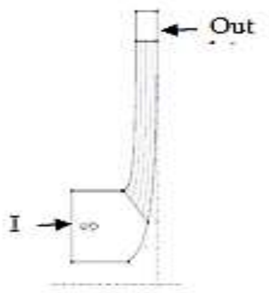

Fig. 1: 2D blade

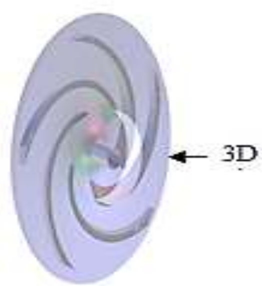

Fig. 2: Model of

In this work numerical simulation of a 3D flow of fly ash slurry and water at different concentration and mass flow rate is done in a centrifugal pump. For completing this work the geometry of a centrifugal pump is developed. it is a high performance CFD software which Works on a finite volume method. It is widely used for simulations of gas-hydraulic turbines, and other rotating machineries. Basically effort is carried out for analyzing flow behavior of fly ash slurry and water in centrifugal pump. In this simulation pressure field of flow in centrifugal pump is obtained. Objective of this work is to validate the results obtained in these simulations with the experimental results available in literature. The simulation is carried out at different concentration (i.e. 60.4% Cw, 65.2% Cw, and 70% Cw) of fly ash slurry. For evaluation of flow behavior of fly ash slurry, fly ash slurry is considered as one pure fluid with constant fluid properties. Main focus in this work is to get the relation between head obtained and discharge of the pump. Head is nothing but a point at which discharge of a pump straight up to the air, pump will lift the fluid at certain height this height is called head of the pump [4]. In centrifugal pump impeller is rotated by an electric motor, and shaft is used for power transmission between motor and pump. Head is dependent on the outside diameter of the impeller and its rotation [5]. By validating the results of this numerical simulation with experimental values, in future one can predict the flow behavior of different concentration fly ash slurry with this simple computational model which doesn't require big computing facility workstations.

Workstation used for this simulation
HP-Z420Workstation
Intel Xeon E5-1603 2.8 GHz 4-core Processor
8GB DDR3-1600 ECC RAM
WINDOWS 7 Professional 64-bit OS
NVIDIA Quadro 2GB Graphics
1TB 7200 RPM SATA HDD

II. GEOMETRY AND MODELLING

Many geometry and modelling tools are available in ANSYS for creating simulation model, for this simulation geometry is created using 'BladeGen' tool of the ANSYS WORKBENCH. It is a very handy tool for designing blades of any turbo machinery. Beauty of this tool is it automatically generates fluid path according to the data input.

Table I: Specification of pump used for this simulation.

Impeller	
1. Type	Closed
2. No. of vanes	5 (backward type)
Casing	
1. Type	Volute
2. Base volute radius	145 mm
Suction flange size	100 mm
Delivery flange size	50 mm
Rated speed	1450 (rpm)
Best efficiency point	
1. Total head	20 m
2. Discharge rate	15.1 (lps)

Table II: Viscosity of different concentration fly ash slurry.

% C_w	Slurry viscosity (Pa s)
0	0.891×10^{-3}
60.4	14.50×10^{-3}
65.2	53.40×10^{-3}
70	245.30×10^{-3}

A. Meshing

Structured grid is created on impeller using TurboGrid tool of workbench. Grid is generated assuming axisymmetric flow make flow passage domain simpler. Grid generation is one of the important stages in any simulation [6]. Grid generation of a whole impeller and single flow passage is shown in Fig. 3 and Fig. 4 Respectively assumptions taken are (1) fly ash slurry is a constant property fluid (2) steady

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 3, Issue 2, February 2018

state condition is assumed for fluid flow (3) Smooth walls are considered (4) Slurry is considered as incompressible

Table III: Mesh details.

	Rotor	Stator
No of elements	145368	150914
No of nodes	161400	54759

Boundary conditions

Input parameters and boundary conditions used for this simulations are given in Table IV for all different concentration flow the boundary conditions are at inlet p total is applied and at outlet boundary condition given is mass flow outlet.

Table IV: Input parameters for simulations.

Parameters	CFX Input
Simulation domain	Single impeller flow passage
Grid	structured
Fluid 1. Water 2. Fly ash slurry	Density(ρ) 1000 1440
Inlet	1 atm
Outlet	variable
Turbulence model	*k*-Epsilon model
Discretization	Second order
Turbulence intensity	1%
Maximum residual convergence 1. Type 2. Value	 RMS 0.0001

C. Turbulence model used

K-Epsilon turbulence model is used for the simulation. It is a commonly used model for simulating mean flow characteristics for turbulent flow condition. Two transport equation is solved which gives general description of turbulence of the model. In two equations one is T-kinetic energy (k) and second is rate of dissipation of turbulence energy (ε). In general standard and practical k-Epsilon model is used which has minimum unknown. And one can easily apply that to any turbulent simulation.

For T-kinetic energy (k)

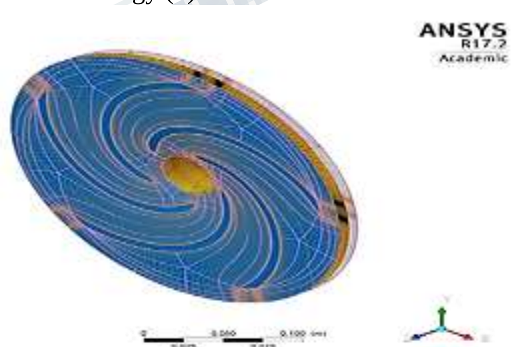

Fig. 3: Structured Mesh.

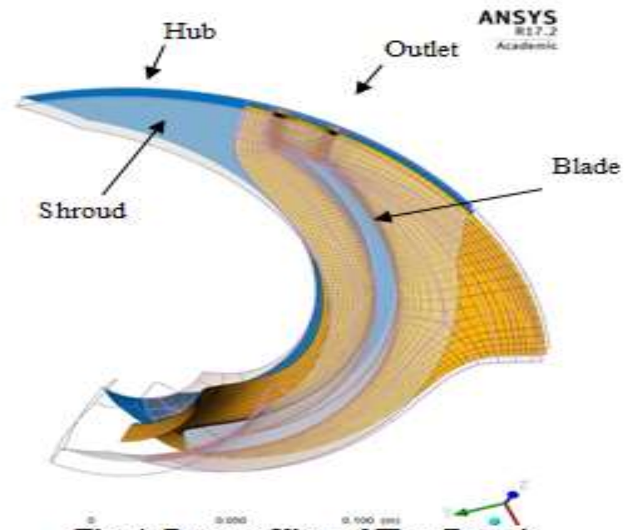

Fig. 4: Passage View of Flow Domain.

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right]+2\mu_t E_{ij}E_{ij}-\rho\varepsilon \qquad (1)$$

For dissipation of turbulence model (ε)

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_j}\right]+C_\varepsilon\frac{\varepsilon}{k}2\mu E_{ij}E_{ij} \qquad (2)$$

$$-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

D. Interface model used

Three types of mixing models are available in ANSYS CFX – (1) Frozen rotor, (2) Stage model and (3) Transient rotor stator. In impeller hexahedral mesh element is generated while in this work volute meshing is done with tetrahedral mesh element so it is very much important to select proper interface model between rotor and stator [7]. Also the one part is as stationary frame while other is as rotational frame. Frozen rotor model requires least amount of computational effort of the other mixing models, and suits this operating and computational condition so it is selected as the interface model.

E. Centrifugal pump 3D steady flow simulation

Simulation is done on ANSYS CFX. First simulations are carried out at different mass flow rate of water. Pressure contour of water at 22.6, 30, and 53.36 cubic meter mass flow rate are shown in Fig. 5, Fig. 6, and Fig. 7 respectively. Pressure contour of 65% Cw fly ash at 22.6, 30 and 38 cubic meter of mass flow rate is shown in Fig. 8, Fig. 9, and Fig. 10 respectively. And pressure contour of 70% Cw fly ash slurry at 20.6 and 30 cubic meter of mass flow rate is shown in Fig. 11 and Fig. 12.

IFERP

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 3, Issue 2, February 2018

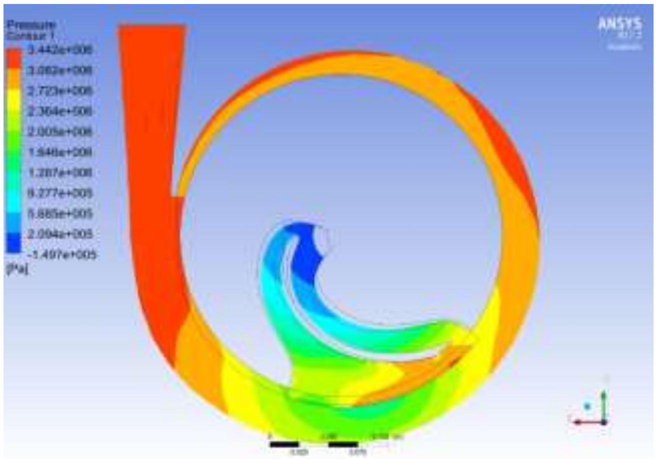

Fig. 5: Water 22.6 cubic meter per second.

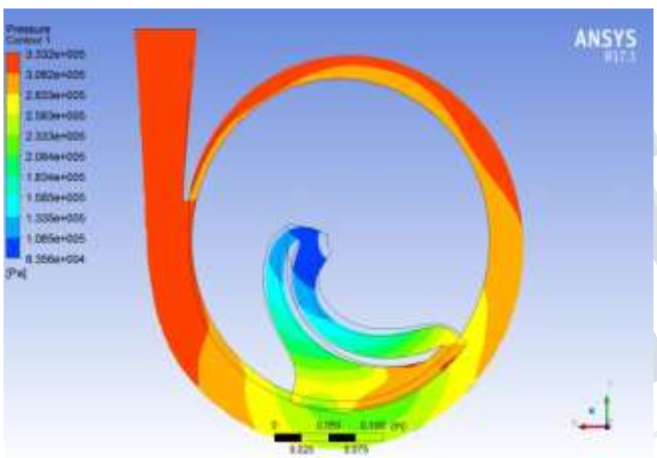

Fig. 6: 30 cubic meter per second

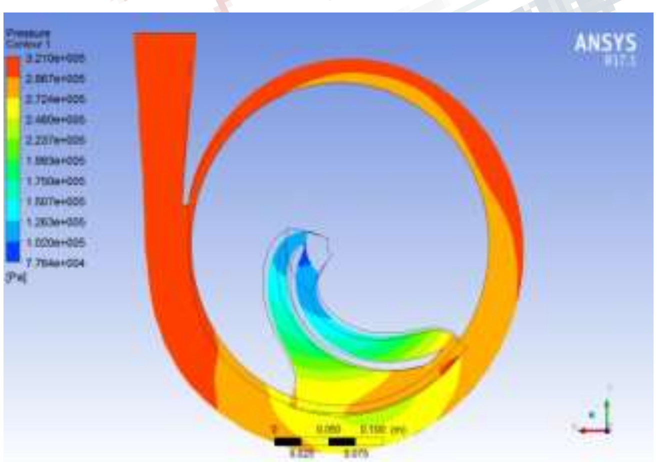

Fig. 7: 54.36 cubic meter per second.

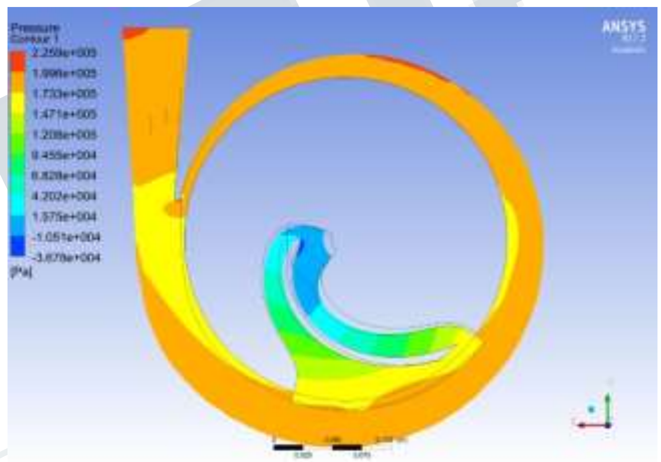

Fig. 9: 65% fly ash slurry 30 cubic meter per second.

Fig. 10: 65% fly ash slurry 38 cubic meter per second

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 3, Issue 2, February 2018

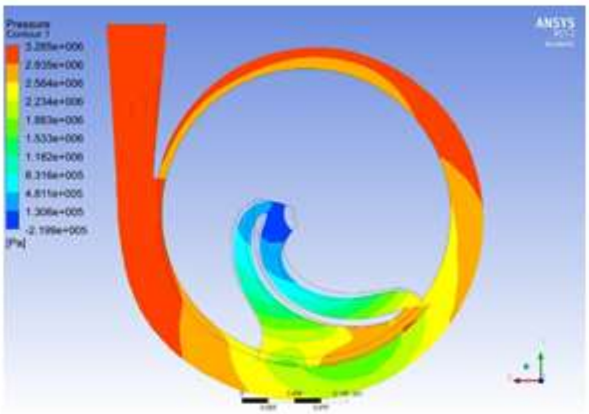

Fig. 11: 70% fly ash slurry 20.6 cubic meter flow rate

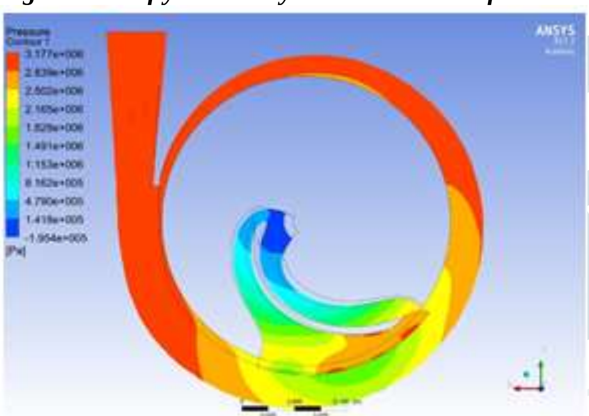

Fig. 12: 70% fly ash slurry 30 cubic meter flow rate.

III. PERFORMANCE CHARACTERISTICS

Performance head characteristics was numerically calculated by considering fly ash slurry as constant Property Fluid at different concentration, also the fly ash slurry is considered as one pure fluid throughout the simulation. And simulations are run at different mass flow rate of fly ash slurry. Fig. 13 shows the performance characteristic curve of centrifugal pump. By analyzing the curve we say that as the mass flow rate of the pump increase head obtained by the pump reduces.

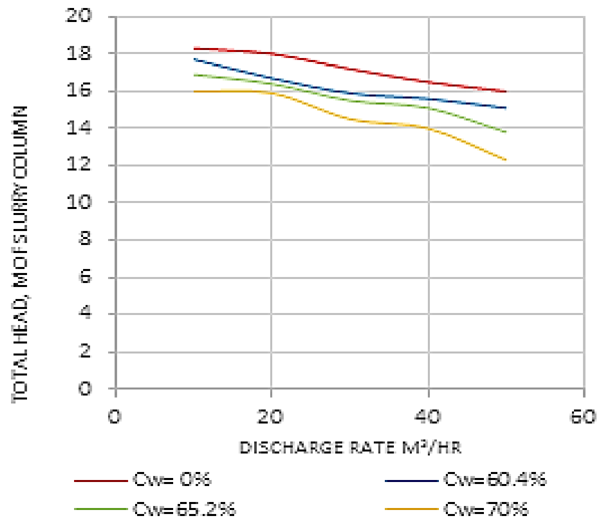

Fig. 13: Performance characteristic curve.

IV. CONCLUSION

In this work a centrifugal pump is created for fly ash slurry transportation. Simulation is done at different flow rate of water as well as fly ash slurry at 1450 rpm. Pressure contours for internal flow are obtained. Meshing is finely done with ANSYS CFX TurboGrid Obtained results are in similar pattern with the experimental results available. Results shows that at high concentration head developed is reduced. By analysing results we can say that when rotational speed is taken as constant total head depends on mass flow rate.

REFERENCES

[1] S. Chandel, "Studies on the Flow of High Concentration Coal Ash Slurry through Pipelines," Ph.D. thesis, IIT Delhi, 2010.

[2] R.S. Muttalli, S. Agrawal, H. Warudkar, "Simulation CFD of Centrifugal Pump Impeller Using ANSYS CFX," International Journal of Innovative Research in Science, Engineering and Technology, vol. 3, issue 8, Aug 2014.

[3] M. Asuaje, S. Kouidri, F. Bakir, F. Kenyery, R. Rey, "Numerical Modelization of the Flow in Centrifugal Pump: Volute Influence in Velocity and Pressure Fields," International Journal of Rotating Machinery, pp. 244-255, 2005.

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 3, Issue 2, February 2018

[4] Maitelli C. W. S de P, Bezerra v. M. de F, da Mata w, “Simulation of Flow in a Centrifugal Pump of ESP Systems Using Computational Fluid Dynamics,” Brazilian Journal of Petroleum and Gas, pp. 001-009, 2010.

[5] S. Rajendran and Dr. K. Purushuthaman, “Analysis of a centrifugal Pump Impeller using ANSYS-CFX,” International Journal of Engineering Research & Technology, vol. 1, issue 3, 2012.

[6] Erik Dick, Jan Vierendeels, Sven Serbruyns and John Vande Voorde, “Performance Prediction of Centrifugal Pumps with CFD-tools,” TaskQuarterly 5, no. 4, pp. 579-594, 2001.

[7] Abdulkadir Aman, Sileshi Kore and Edessa Dribssa, “Flow Simulation and Performance Prediction of centrifugal Pumps using CFD tool,” Journal of EEA, vol. 28, 2011.