CFD Analysis on Impeller of Slurry Mixture for Performance Improvement

Swapnil Jain[1] Somendra Kumar Dewangan[2] Dr. Deepak Sharma[3]
[1]Associate Professor [2]Student [3]Principal
[1,2,3]Department of Mechanical Engineering
[1,2]Shri Shankaracharya Engineering College, Junwani, Bhilai, India [3]Shri Shankaracharya Institute of Engineering & Technology, Junwani, Bhilai, India

Abstract— A slurry mixture impellers are the most important part of thermal power plant. In this paper, Computational Fluid Dynamics (CFD) is carried out on the performance of slurry mixture blade. The performance is based on the pressure variation on the mixture blade. Computational investigation was conducted on blades with a linear velocity and density is 4 m/s and 1440 kg/m3 respectively. In the present work, analysis has done with two different material namely grey cast iron and nickel. The pressure was compared between these two materials. After calculating the result, we will suggest which material is good for impeller. The slurry impeller is modelled using CREO software and CFD analysis is carried out using ANSYS software. A k- ε turbulence model is used in CFD analysis. The model has been developed with actual geometric parameters.

Key words: Computational Fluid Dynamics, Slurry Mixture Impeller, Pressure Variation

I. INTRODUCTION

Slurry mixture impeller is a rotating device that mixes ash and water in the ash hopper. A steam generating boiler simply cannot operate without an ash hopper. Ash is the residue remaining after the coal is incinerated. Ash produced in large quantity from boilers that to utilize it completely converting into useful products is not always possible. In this condition ash is required to be dumped in ash pond which are usually far away from the plant area. The effective method of disposing ash is by pumping it in form of slurry through ash hopper. Slurry can be classified in low /high density depending on slurry concentration. The carrying medium is always water. Ash generated in power plant is about 30-40% of total coal consumption and hence the system is required to handle ash for its proper utilization or disposal. Ash generally classified in two categories namely fly ash and bottom ash. Fly ash is generated in ESP which got carried out with the flue gases. It also consists of economizer ash and air pre heater ash. Bottom ash is generated below boiler furnace of the steam generator. Bottom ash is heavier than fly ash.

Fig. 1: Slurry mixture impeller

A. Slurry

Slurry is a mixture of solids and liquid, generally water. Slurry is dependent on physical characteristics such as distribution of particles, size and concentration of solid in liquid phase. The example of slurry flows as ash-water mixture which is mixes in the ash hopper. The flow of slurry is generally classified into two categories, one is non-settling slurry and other is settling slurry.

B. Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is the process of obtaining numerical approximations to the solution of the governing equations. CFD provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of

1) Mathematical modeling (partial differential equation)
2) Numerical method (discretization and solution techniques)
3) Software tools (solver, pre-processor and post-processor)

II. PROBLEM STATEMENT

A steam generating system cannot give up ash directly into atmosphere. Because of lighter weight of ash there is a chance of spread of ash in environment which may be dangerous for our health. So that water is mixed with ash to form slurry before leaving the plant through ash hopper. In ash hopper, mixture impeller is placed for proper mixing of ash and water. While mixing slurry becomes dense so that impeller blade is deformed. This is the serious problem that affect the working of ash hopper.

Fig. 2: Blade before working

Fig. 3: Deformed blade after working

III. METHODOLOGY

Based on problem identification, firstly make a model of impeller blade having actual geometry from plant site. The thickness of blade is 20 mm. The other parameters as shown in fig. below:

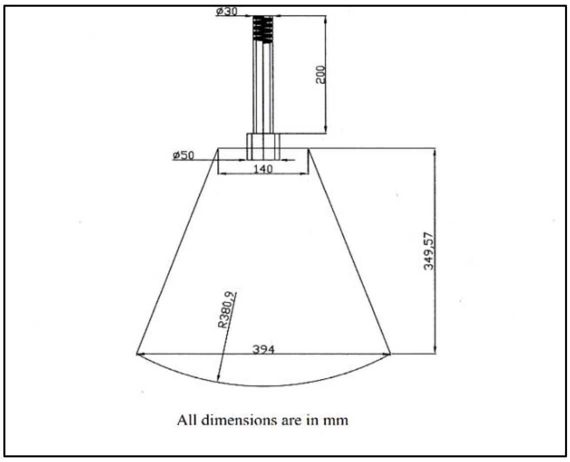

Fig. 4: Geometry of impeller blade

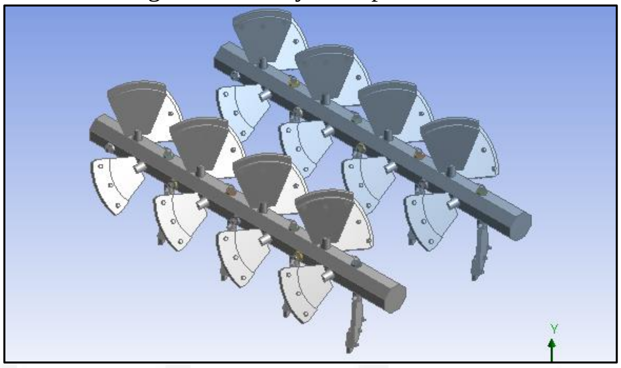

Fig. 5: Assembled model of mixture impeller

A. Mesh Generation

Mesh generation is the process of subdividing a region to be modeled into a set of small control volumes. In each control volume there will be one or more value of the dependent variable (e.g. Velocity, temperature, pressure etc.). Meshing is a method to define and break up the model into small elements.

The quality of mesh is dependent on orthogonal quality and aspect ratio. Orthogonal quality ranges from 0 to1.

Minimum Orthogonal Quality = 7.56318e-02

Maximum Aspect Ratio = 8.04631e+01

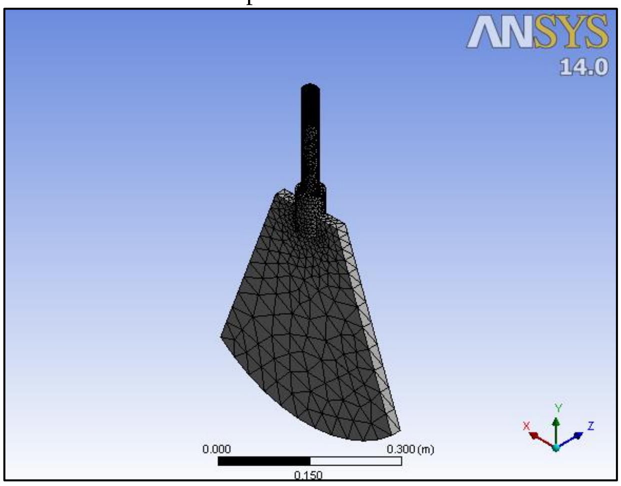

Fig. 6: Meshing of blade

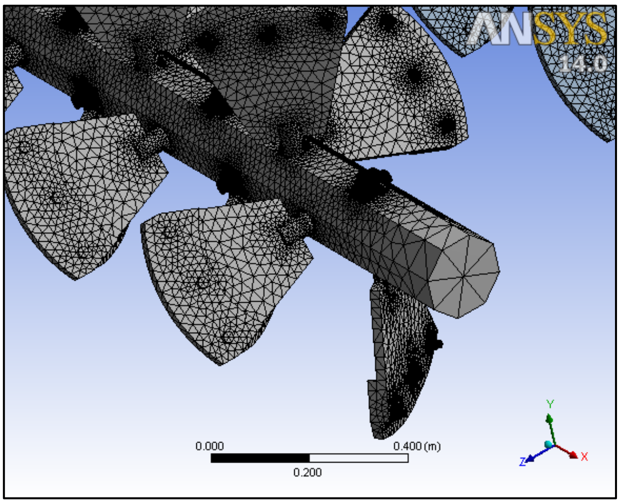

Fig. 7: Meshing of impeller

No. of Nodes	588865
No. of Elements	3192439

Table 1: Mesh Statistics

B. Pre-processing

In pre-processing, material properties, cell zone conditions and boundary conditions are applied on the model. The particle size is 20 µm. Applied cell zone conditions are:

At Inlet – Velocity inlet,
At outlet – Pressure outlet.

C. K-Epsilon Turbulence model:

The k-epsilon (k- ε) model proposed by Lounder and Spalding (1974) makes use of two model equation, one for the turbulent kinetic energy k and one for the rate of dissipation of turbulent kinetic energy per unit mass ε. These equations are:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial t}(\rho k v_i) = \frac{\partial}{\partial t}\left[\frac{\mu_j}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + 2\mu_T S_{ij} S_{ij} - \rho\varepsilon \quad (1)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial t}(\rho\varepsilon\overline{v}_j) = \frac{\partial}{\partial t}\left[\frac{\mu_T}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_T S_{ij} S_{ij} - 2C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (2)$$

Transport equation (1) and (2) are solved iteratively along with Reynolds averaged Navier-Stokes equation. Using k and ε, velocity and length scales are defined as follows:

$$q = \sqrt{k}, \qquad L = \frac{k^{\frac{3}{2}}}{\varepsilon}$$

The standard k- ε model is well established and the most widely validated and used eddy viscosity turbulence model. Various modifications of the standard model have been suggested to account for the near-wall region and low Reynolds number turbulent flows.

IV. Results & Discussions

The results displayed in the form of contour plot and vector plot from the CFD (Computational Fluid Dynamics) simulation. The contour plot of pressure for both materials has shown in figure below. The pressure varies around the whole assembly. The figure shows that pressure is maximum on the blade and the value of pressure for grey cast iron and nickel is 2.372 bar and 1.29718 bar respectively. The variation of vector plot of static pressure are also displayed.

A. For Grey Cast Iron

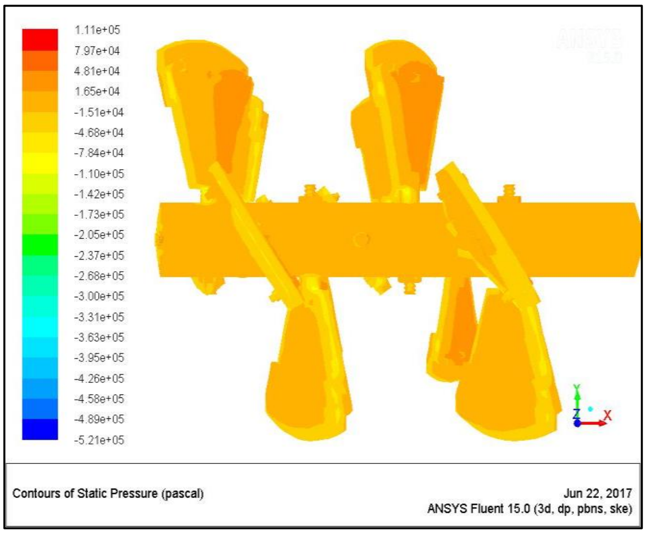

Fig. 8: Contour of Pressure (1)

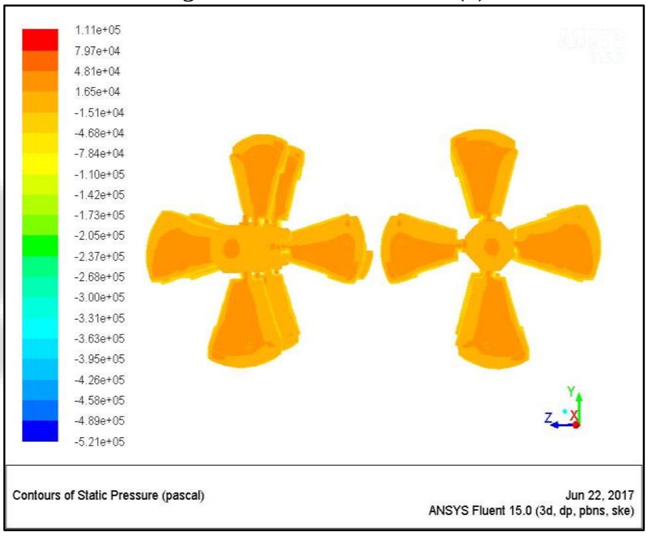

Fig. 9: Contour of Pressure (2)

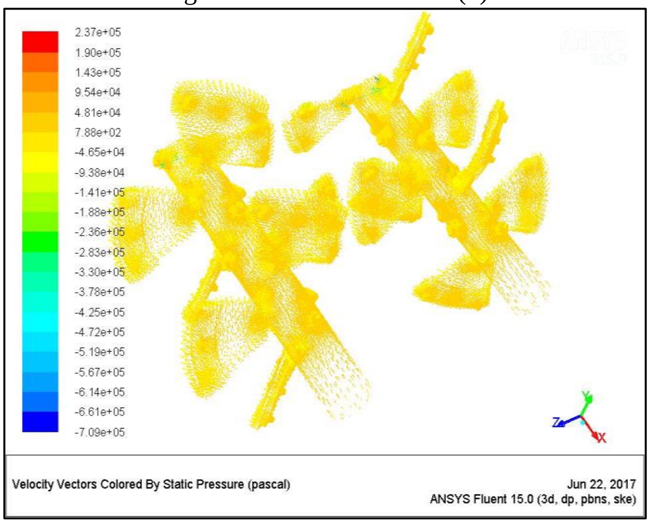

Fig. 10: Velocity vector of static pressure

B. For Nickel

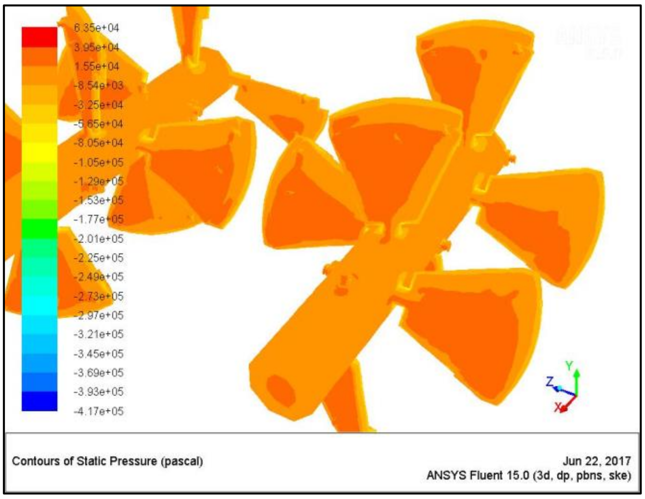

Fig. 11: Contour of Pressure (1)

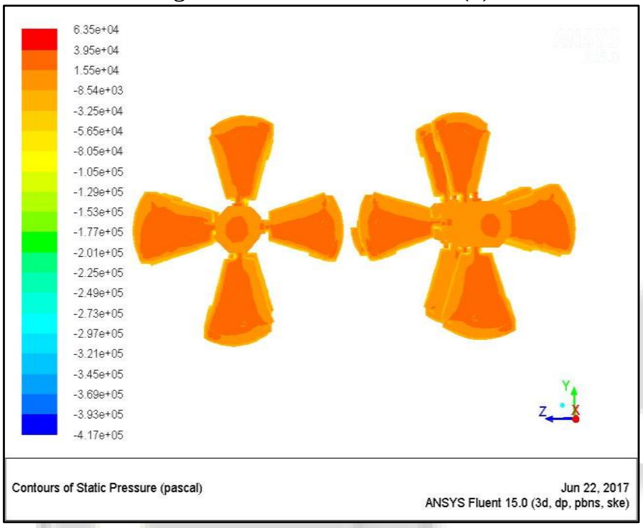

Fig. 12: Contour of pressure (2)

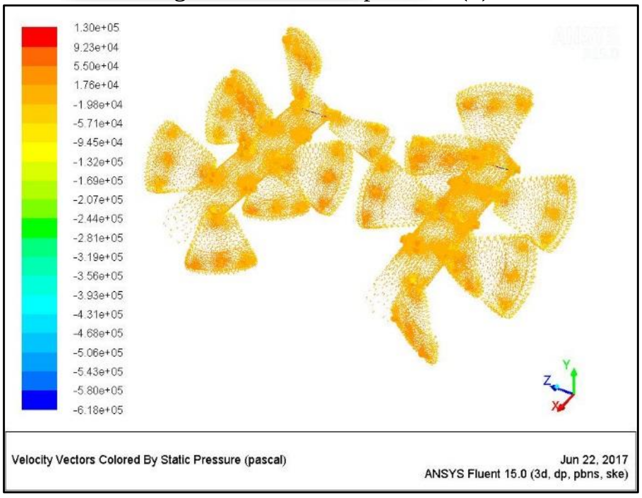

Fig. 13: Velocity vector of static pressure

V. CONCLUSION

A slurry mixture impeller is modeled by using CREO software and this model is simulated in ANSYS FLUENT software in terms of pressure variation. The CFD analysis has done by using k-epsilon (k- ε) turbulence model. The pressure around the blade for both the materials were calculated and found that pressure on grey cast iron is larger than nickel. As we further increases the velocity and slurry concentration (density more than 1440kg/m^3), the pressure also increases in same manner, and at a stage when pressure will be maximum there is a chance of deformation.

So that from this paper result we can conclude that we need a material for impeller which resist the maximum pressure. After simulation of these two materials on the basis of our result we can suggest "nickel" for impeller design. To improve the performance, reduce deformation and get longer life of impeller.

VI. FUTURE SCOPE

The scope of future work in this topic is very vast. A lot of work can be done in this field. The following suggestions for further work as:

1) The material selected for mixture blade in this work is grey cast iron and nickel so that different material can be selected for the analysis in future work.
2) The CFD approach can be used to simulate the similar work with different density and slurry concentration.
3) Flat mixture blade can be extended to curved blade.
4) Different blade angle can be selected in future work.

REFERENCES

1) Walker, C.I. Well, P.J and Pomat C., "The Effect of Impeller Geometry on the Performance of Centrifugal Slurry Pumps", Elsevier Science Publishers Limited, New York, 247-258, 1993.
2) Gandhi, B. K., Singh, S. N. and Seshadri, V.," Performance characteristics of centrifugal slurry pumps", Transactions of ASME, Journal of Fluid Engineering, (123), 271-280, 2001.
3) P.Gurupranesh, R.C.Radha, N.Karthikeyan, CFD Analysis of centrifugal pump impeller for performance enhancement, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE).
4) Verma, A.K., Singh, S.N., Seshadri, V., "Pressure drop for the flow of high concentration solid-liquid mixture across 90^0 horizontal conventional circular pipe bend", Indian Journal of Engineering & Material Sciences, 13, 477-483, 2006.
5) Rakesh Joshi, Computational Investigation of Flow Field in a Centrifugal Pump, Thapar University, 2010
6) Harsimran Singh, Investigation of Flow of Centrifugal Slurry Pump Handling Bottom Ash, Thapar University, 2011
7) Ugwu, K.E., Ofomatah, A.C., Eze, S.I., "Influence of solid concentration on the flow characteristics and settling rate of coal water slurries", Journal of Energy and Natural Resources, 2(3), 21-24, 2013
8) Vlasak, P., Chara, Z., Krupicka, J., Konfrst, J., "Experimental investigation of coarse- particles water mixture flow in horizontal and inclined pipes", Journal of Hydrology Hydromech., 62 (3), 241-247, 2014.
9) Eswara Rao, G.A., Naidu, M.K., "Numerical simulation on the slurry flow through straight pipe for the evaluation of pressure drop:, 1(7) , 81-87, 2014
10) Raghavendra S Muttalli, Shweta Agrawal & Harshla Warudkar, "CFD Simulation of Centrifugal Pump

Impeller Using ANSYS-CFX", International Journal of Innovative Research in Science, Engineering and Technology, Vol.3, Issue 8, August 2014.

