International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:11, No:2, 2017

Numerical Investigation of Pressure Drop and Erosion Wear by Computational Fluid Dynamics Simulation

Praveen Kumar, Nitin Kumar, Hemant Kumar

***Abstract*—**The modernization of computer technology and commercial computational fluid dynamic (CFD) simulation has given better detailed results as compared to experimental investigation techniques. CFD techniques are widely used in different field due to its flexibility and performance. Evaluation of pipeline erosion is complex phenomenon to solve by numerical arithmetic technique, whereas CFD simulation is an easy tool to resolve that type of problem. Erosion wear behaviour due to solid–liquid mixture in the slurry pipeline has been investigated using commercial CFD code in FLUENT. Multi-phase Euler-Lagrange model was adopted to predict the solid particle erosion wear in 22.5° pipe bend for the flow of bottom ash-water suspension. The present study addresses erosion prediction in three dimensional 22.5° pipe bend for two-phase (solid and liquid) flow using finite volume method with standard *k-ε* turbulence, discrete phase model and evaluation of erosion wear rate with varying velocity 2-4 m/s. The result shows that velocity of solid-liquid mixture found to be highly dominating parameter as compared to solid concentration, density, and particle size. At low velocity, settling takes place in the pipe bend due to low inertia and gravitational effect on solid particulate which leads to high erosion at bottom side of pipeline.

***Keywords*—**Computational fluid dynamics, erosion, slurry transportation, *k-ε Model.*

I. Introduction

THIS paper presents erosion prediction by using commercial CFD codes. Slurry is a mixture of solid particles and fluid held in suspension solution [1]. Water is used as most common fluid for preparation of the slurry due to its bulk availability. Transportation of the slurry through pipeline has adverse effect on material transportation system. The physical characteristics of slurry are dependent on many factors such as particle size distribution, solid concentration in the liquid phase, turbulence level, temperature, conduit size, and viscosity of the carrier [2]. This slurry transportation method has various advantages such as less noise and pollution. Slurry pipelines are used to transport solid materials using water for short or long distances. The rheological characteristics of slurry depend on particle size, concentration. CFD simulation is performed on the slurry flow in pipeline bend for analysis of pressure drop in pipeline [3]. CFD is the analysis of slurry system involving fluid flow by means of computational simulation [4]. CFD is one of the branches of fluid mechanics that use numerical methods and algorithms to solve and analyse problems that involve fluid flows [5]. It has become an essential tool for design, analysis, and refinement of industrial equipment and research work. Nowadays erosive wear is a serious problem in thermal power plants. Slurry pipelines are commonly used for transportation of bottom ash from plant to ash pond in form of slurry. These systems are involved the excessive wear and pressure drop in slurry pipeline. CFD simulation is a modern technique for the analysis of erosion wear and pressure drop in pipeline bend. Commercial CFD tool FLUENT is used to evaluate the pressure drop and erosive wear at different velocities. Pipeline is modelled with help of SolidWorks 2012. The CFD simulation results show good agreement with Pereira et al. [6].

Praveen Kumar, Nitin Kumar and Hemant Kumar are with the Mechanical Engineering Department, Thapar University, Patiala, India (phone: +919815306452, e-mail: Praveen.dimenssion@gmail.com).

II. Computational Simulation of Pipeline

CFD is one of the branch of fluid mechanics that uses numerical methods and algorithms to solve problems that involve fluid flows [7]. CFD is the analysis of system involving fluid flow by means of high performance computer based simulation. The use of CFD reduces the development cost of new products and cuts the time to market of these products [8].

A. Methodology

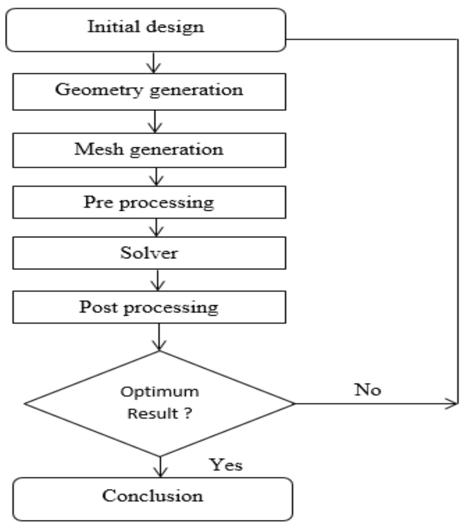

Fig. 1 Methodology adopted in CFD [9]

B. Rheological Properties

Physical properties of bottom ash are given in Table I. The specific gravity of bottom ash was found as 2.25.

TABLE I
RHEOLOGICAL PROPERTIES OF BOTTOM ASH

S.No.	Concentration (Cw)%	Yield stress (Pa)	Slurry viscosity (cP)	Water viscosity (cP)	Relative viscosity (cP)	Flow behaviour
1	0	0	-	0.995	1	Newtonian
2	15	0	1.305	0.995	1.315	Newtonian
3	25	0	1.9	0.995	1.91	Newtonian
4	35	0	3.25	0.995	3.26	Newtonian
5	45	0	4.95	0.995	4.965	Newtonian

TABLE II
MESH QUALITY PARAMETER

S.No.	Mesh Type	Mesh Size	Number of Elements	Quality Parameter		
				Equi Size Skewness	Equi Angle Skewness	Aspect Ratio
1	Tetrahedral Elements	5	103727	0.805	0.85	1.85
2	Tetrahedral Elements	4	201482	0.84	0.85	1.84
3	Tetrahedral Elements	3	477222	0.83	0.85	1.84

III. COMPUTATIONAL DOMAIN AND MESH

CFD code contains three main elements which are preprocessor, flow solver, and postprocessor.

A. Modeling of Pipeline

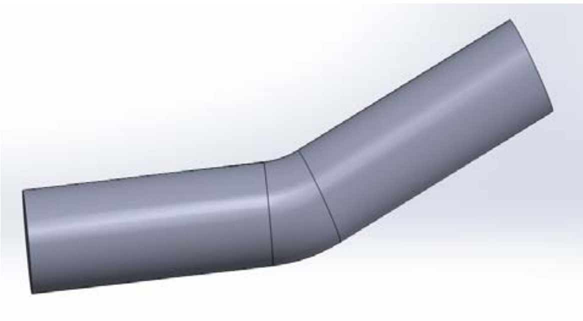

Fig. 2 Model of the pipe flow configuration with the elbow being considered for analysis

To investigate numerical analysis on the pipeline of 22.5° bend and 68 mm diameter, the model of pipeline bend has developed in modelling software SolidWorks 2012 which is shown in Fig. 2.

B. Grid Generation

The model is meshed using ANSYS R15.0 and gets three sets of meshed Quad symmetric cells. The number of meshed cells is listed in the Table II which contains mesh type, mesh size, and number of element. Dimension of the pipeline bends are taken from industry data. The pipeline bend model is modelled by using SolidWorks 2012 (Dassault System) and imported the IGES file of SolidWorks in the ANSYS 15. The 3D meshing of 22.5° bend pipeline is shown in Fig. 3.

T-grid tools are used for generating meshing of the 22.5° pipeline bend and it is very effective meshing tool. It consists of triangular, tetrahedral and prismatic, hexahedral elements. Pipeline bend is generated by using tetrahedral meshing in FLUENT. Mesh quality parameters which are skewness and aspect ratio are calculated as 0.85 and 1.85, respectively. 477222 elements are generated.

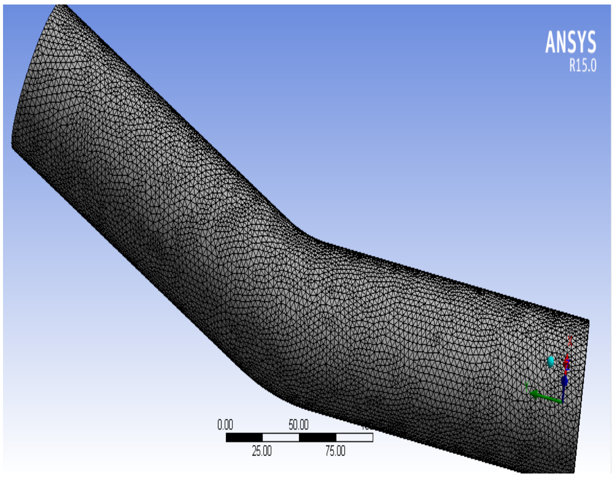

Fig. 3 Geometry and computational mesh distribution for the pipe bend model

C. Grid Independency Analysis

In the present work, three types of mesh size using for evaluate the grid independency test. It is observed that coarse 5 mm mesh size in pipeline bend generates 103727 tetrahedral elements. Fineness of 4 mm grid size in pipeline bends 201482 tetrahedral elements generates, and finer mesh of 3 mm grid size in pipeline bend produces 477222 tetrahedral elements in CFD domain. Mesh quality of pipeline bend is shown in Table II. The mesh size of 3 mm shows the more accurate results. So, 3 mm mesh size is used for simulation.

D. Quality of Mesh

The quality of mesh is checked due to existence of skewness parameters that affect the accuracy of CFD simulation.

Each element has a value of skewness between 0 and 1. The skewness is divided in two parts, i.e. 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.

E. Boundary Conditions and Solution Parameters

The simulation of flow in pipeline bend is done on the basis of steady state conditions. The boundary conditions are specified as constant velocity in X-direction at velocity inlet. Turbulence viscosity ratio and intensity are taken as 10 and 2%, respectively. Pressure outlet at discharge end is given as absolute pressure, and target mass flow rate is 4 kg/s, i.e. constant. Backflow intensity and turbulence viscosity ratio are selected as 2 and 10% respectively for outlet. 3D double precision solver is used to solve for simulation. Clear water is used as working fluid. Standard *k-ε* model is most extensively used for turbulence. Two transport equations *k-ε* model independently solve for turbulent velocity and length scale. Second order scheme is used for turbulence dissipation rate as well as pressure correction, turbulent kinetic energy. To achieve convergence within less time, relaxation factor applied is 0.3 for pressure, 0.8 for turbulent kinetic energy, 0.8 for turbulence dissipation rate, and 0.7 for momentum.

IV. Computational Results and Discussion

A. Pressure Drop with Water in Pipe Line

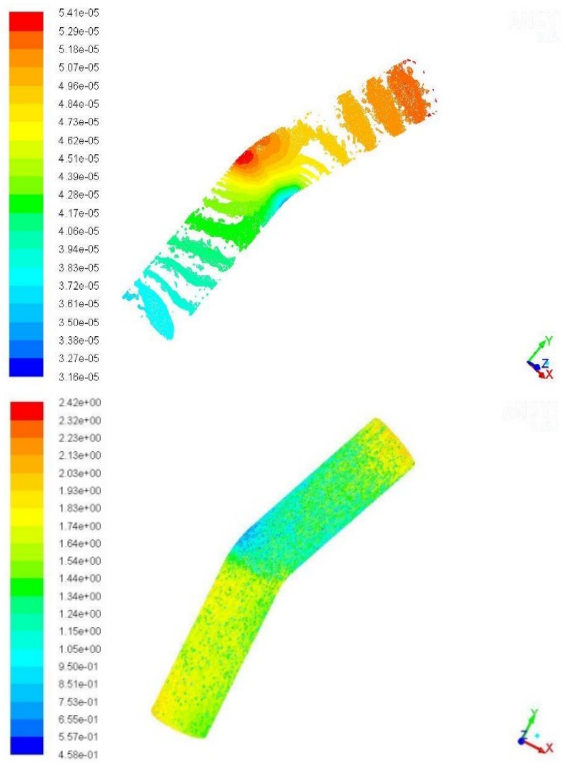

Fig. 4 Pressure and Velocity contour of the water flow for velocity 2 m/s

The pressure drop across the bend is a function of flow velocity and solid concentration. In the present study, the first pressure drop is measured with clear water then ash solid of the bottom ash with different velocity at same flow discharge. Figs. 4 and 5 show pressure and velocity vector contour at same discharge with velocity 2 and 4 m/s. Pressure contours predict that the pressure is maximum at initial point of pipe bend and decreases across the bend and becomes minimum at the bend. It is observed that increase in flow velocity of slurry increases the pressure drop in pipe. Kinetic energy of particle changes into the pressure energy due to velocity that causes maximum pressure at the outer periphery of pipe. Turbulence becomes more intense at boundary of pipe and drops at the center of pipe. The maximum heat dissipation rate is observed at boundary of pipe due to kinetic energy of incoming fluid which is immediately transferred to the wall of pipe after collision to the boundary.

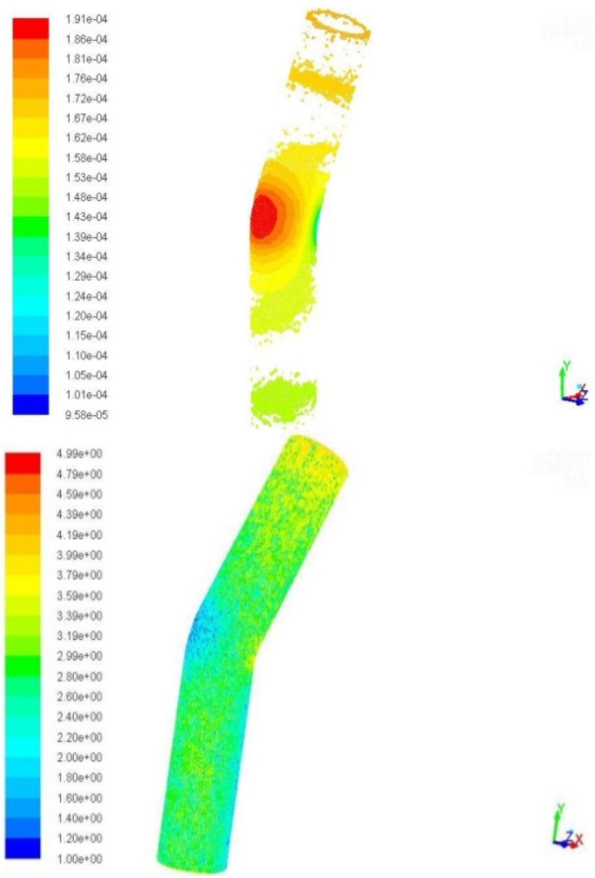

Fig. 5 Pressure and Velocity contour of the water flow for velocity 4 m/s

B. Erosion Wear Studied by Computational Method

Fig. 6 illustrates the erosion contour inside elbow for 2 and 4 m/s velocity with 45% solid concentration of ash particles. It was found that maximum erosion occurs after the midpoint along the symmetry of pipe bend where the profile of velocity starts, and pressure is maximum. It is also observed that martial loss increases with increase in mass flow rate due to increase of particles kinetic energy. This could be due to fact that particle rebound back at bend will have more rebounding velocity.

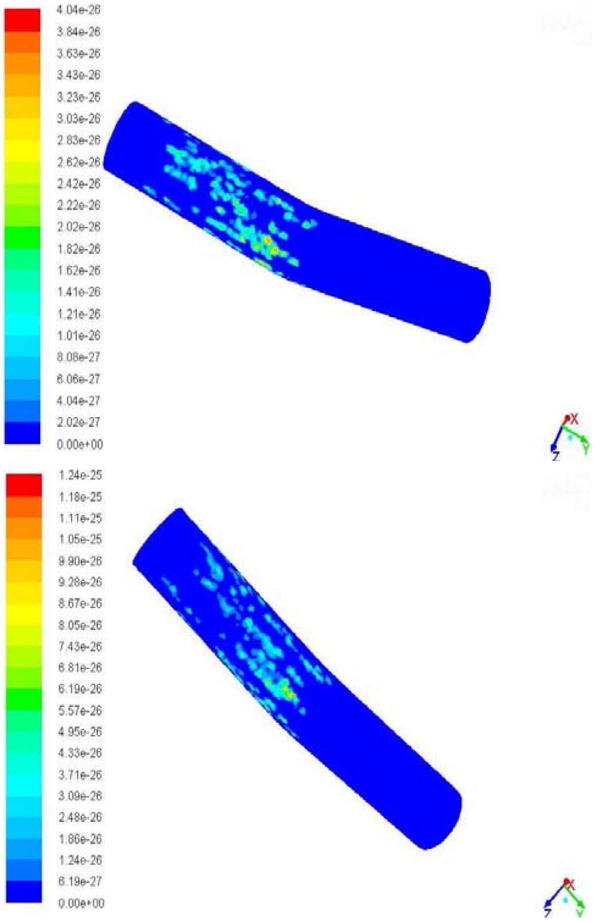

Fig. 6 Location of maximum erosion in 22.5° bend for velocity of 2 m/s and 4 m/s at 45% concentration of ash

V. Conclusion

On the basis of present study, the following conclusions can be drawn:

1. At constant flow rate, the pressure drop increases with increase of the slurry concentration.
2. The pressure at outer periphery is greater as compared to the inner section of pipeline.
3. CFD FLUENT results give the pressure and velocity contour which is necessary for pressure drop analysis.
4. Erosion wear depends upon velocity and impact angle and it is observed that wear rate is lesser in 22.5° bend as compared to 90° bend.

References

[1] Singh, K. and Lal, K., 2012. Effect of cetylpyridinium chloride, Triton x-100 and sodium dodecyl sulfate on rheology of fly ash slurry. *International Journal of Scientific and Research Publications*, *2*(8), pp.1-5.

[2] Aktaş, Z. and Woodburn, E.T., 2000. Effect of addition of surface active agent on the viscosity of high concentration slurry of a low-rank British coal in water. *Fuel Processing Technology*, *62*(1), pp.1-15.

[3] Ahmed, W.H., Ching, C.Y. and Shoukri, M., 2007. Pressure recovery of two-phase flow across sudden expansions. *International journal of multiphase flow*, *33*(6), pp.575-594.

[4] Ekambara, K., Sanders, R.S., Nandakumar, K. and Masliyah, J.H., 2008. CFD simulation of bubbly two-phase flow in horizontal pipes. *Chemical Engineering Journal*, *144*(2), pp.277-288.

[5] Naik, H.K., Mishra, M.K. and Rao Karanam, U.M., 2009. Rheological characteristics of fly ash slurry at varying temperature environment with and without an additive. In *2009 World of Coal Ash (WOCA) Conference-May 4*(Vol. 7).

[6] Pereira, G.C., Souza F.J., Martins, D.A.M. 2014. Numerical prediction of the erosion due to particles in elbows. *Powder Technology*, *261*, pp.105–117.

[7] Senapati, P.K., Mishra, B.K. and Parida, A., 2010. Modeling of viscosity for power plant ash slurry at higher concentrations: Effect of solids volume fraction, particle size and hydrodynamic interactions. *Powder Technology*, *197*(1), pp.1-8.

[8] Naik, H.K., Mishra, M.K. and Rao, K.U., 2011. Evaluation of flow characteristics of fly ash slurry at 40% solid concentration with and without an additive.

[9] Malik, S., Aggarwal, L., Dua, A., 2014. Pressure Drop Analysis of Water and Fly Ash Mixture Flow through Straight Pipeline. IJERA.