iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 **Issue:** V **Month of publication:** May 2018

DOI: http://doi.org/10.22214/ijraset.2018.5255

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue V, May 2018- Available at www.ijraset.com

Numerical Analysis of Sand-Water Slurry Flow through Horizontal Pipeline for Various Particle Size at High Velocity

Shofique Uddin Ahmed[1], Shuvam Mohanty[2], Gaurav Chandra[3]
[1,2]Student, M. Tech, Department Of Mechanical Engineering, Amity University Gurgaon, Haryana
[3]Assistant Engineer,Public Work Department, Uttrakhand, GOVT. of Uttrakhand,Udham Singh Nagar, India

Abstract: This present study represents a numerical analysis of sand water slurry flow considering different sand particle sizes through horizontal pipeline of 5.5m length and 103 mm diameter at high velocity. Computational fluid dynamics was used for the numerical simulation; Eulerial two phase model was selected for modeling the multiphase flow and RNG K-epsilon model was adopted for modeling the turbulent flow. 90μm, 150μm and 270μm sand particle (Sp. Gravity 2.65) sizes were considered for this present study for a mixture flow velocity of 5.4m/s at various solid volumetric concentration levels (20%, 30% and 40%). The behaviour of various flow parameters viz. concentration distributions, velocity distributions and pressure drop were analyzed from the numerically simulated results. The influence of particle size and solid volumetric concentrations on various flow parameters were analyzed in this study. Finally the simulated results of pressure drop were validated with the experimental data available in previous literature.
Keywords: 3-D CFD modeling, Slurry flow, Eulerian model, RNG K-epsilon model, Concentration distribution, Velocity distribution, Pressure drop.

I. INTRODUCTION

Demands for conveyance of solid materials through pipelines over a long distance have increased dramatically over the last few decades. This is because this mode of transportation of solid materials is more economical, more environment friendly, causes less air pollution and less road traffics than the conventional mode of solid material transportations. Many industries like power generation industries, pharmaceutical, construction industries, city municipality, oil and gas industries food processing industries etc handle the solid material transportation through pipelines. Generally solid materials are mixed with fluid and slurry is formed, then the slurry is made to flow through the pipeline for transportation. The flow patterns of these slurry (Solid-fluid mixture) shows a remarkable difference with the flow patterns of pure fluid flow through the pipeline and hence to obtain a detailed information about the slurry flow process behaviour of different flow parameters need to be observed carefully. Slurry flow is a multiphase complex flow problem. Computational fluid dynamics (CFD) is a numerical analysis platform where a wide range of multiphase flow problems can be analyzed with greater ease and low cost which would have been almost impossible with experimental work. CFD allows the researchers to adopt different complex multiphase models for proper modeling of the slurry flow.

II. LITERATURE REVIEW

The primary objective of many researchers over the years has been the development of a generalized solution for the accurate prediction of flow patterns and flow parameters behaviour of a slurry flow. Researches in this area began back in third decade of 20th century when O'Brien [1] and Rouse [2] performed an experimental investigation of a slurry flow containing very low volume fraction in gravity based open channel flow using a diffusion model. Since then many research have been conducted for the analysis of various slurry flow parameters like concentration distribution, velocity distribution and pressure drop. Some of the preeminent work in this field includes the work of Kaushal et.al [3], Kaushal and Tomita [4], Kaushal and Tomita [5], Gillies and Shook [6], Gillies et. Al [7], Gillies and Shook [8], Gillies et. al [9], Roco and Shook [10], Roco and Shook [11], Seshadri et. al [12], Karabelas [13], Shook et. al [14] and Shook and Daniel [15] who focused their study on the concentration distribution of solid particles in slurry flow. Further studies have been taken place aiming for the velocity distributions and pressure drop in a slurry flow. Some of the noticeable works in this area includes the work of Kaushal et. al [3], Kaushal and Tomita [4], Kaushal and Tomita [5], Wilson et. al [16], Ghanta and Purohit [18], Mishra et. al [19], Sundqvist et. al [20], Gillies et. Al [7], Doron et. al [21], Wasp et. al [22]. Some of the recent studies in this area include the work of Gopaliya and Kaushal [23]; in their study Eulerian model along with

RNG K-epsilon model were applied for CFD simulation of sand-water slurry flow through horizontal pipeline. The pressure drop characteristics along with other flow parameters were predicted in their study. Arvind kumar [24] conducted a numerical CFD simulation for prediction of pressure drop for 448 µm silica sand-water slurry flow through pipe bends and straight pipelines, adopting Euleian model and K-epsilon model. He concluded that pressure gradient increases with flow velocity across the pipe bend and pressure values are maximum at outer surface and minimum at inner surface across the pipe bend. Umesh Kumar et. al [25] performed an experimental investigation to predict the pressure drop characteristics for a silica sand-water based bi-modal slurry flow through horizontal pipe. They concluded that for bi-modal slurry the pressure drop increases rapidly when fine particles are added and reduction of pressure drop is larger at low velocity as compared to high velocity. Tamer Nabil et. al [26] used CFD technique for the numerical analysis of sand-water slurry flow through horizontal pipe. In their study they predicted the characteristics of various flow parameters such as concentration distribution and velocity distribution and their effect on pressure drop. Predicted pressure gradient data was compared with their experimental pressure gradient data and was found to be in a good agreement.

In this present study sand-water slurry flow is numerically analyzed considering sand particle size of 90µm, 150µm and 270µm for a high flow velocity of 5.4m/s at different solid volumetric concentration level (20%, 30% and 50%) through 103mm diameter and 5.5 m long horizontal pipeline selecting Eulerian two phase model along with RNG K-epsilon approach. The 3D simulated results of pressure drop are then compared with the experimental data present in the work of Gillies et. al [27] for the validation of the present work.

III. MATHEMATICAL MODELS

Selection of proper mathematical model is essential for every numerical analysis. It provides us precise information about the flow patterns and flow variables within the flow domain. Slurry flow being a multiphase flow (Solid-fluid); preferred model is Eulerian two phase model because of its ability of solving the momentum and mass equations for both the phase (solid and fluid phase) individually. Further granular version of Eulerian model was selected because it takes into account of particle collision and lift force which is essential for slurry flow.

A. Eulerian Model

Eulerian model assumes that the flow consists of separate fluid and solid phases which results in formation of a continuum so that the volumetric concentration of fluid (α_f) and solid (α_s) is equal to 1. i.e. $\alpha_f + \alpha_s = 1$

B. Governing Equations

1) Continuity equation

$\nabla.(\alpha_t \rho_t \vec{v}_t) = 0$, *t* being either *solid* or *fluid.* (1)

2) Momentum equation for fluid phase

$$\nabla.\left(\alpha_f \rho_f \vec{v}_f \vec{v}_f\right) = -\alpha_f \nabla P + \nabla.\bar{\bar{\tau}}_f + \alpha_f \rho_f \vec{g} + K_{sf}\left(\vec{v_s} - \vec{v_f}\right) + C_{vm}\alpha_f \rho_f (\vec{v_s}.\nabla\vec{v_s} - \vec{v_f}.\nabla\vec{v_f}) + C_L \alpha_s \rho_f \left(\vec{v}_f - \vec{v}_s\right) \times (\nabla \times \vec{v_f}) \tag{2}$$

3) Momentum equation for solid phase

$$\nabla.(\alpha_s \rho_s \vec{v}_s \vec{v}_s) = -\alpha_s \nabla P - \nabla P_s + \nabla.\bar{\bar{\tau}}_s + \alpha_s \rho_f \vec{g} + K_{sf}\left(\vec{v_f} - \vec{v_s}\right) + C_{vm}\alpha_s \rho_f \left(\vec{v_f}.\nabla\vec{v_f} - \vec{v_s}.\nabla\vec{v_s}\right) + C_L \alpha_s \rho_f \left(\vec{v}_s - \vec{v}_f\right) \times (\nabla \times \vec{v_f}) \tag{3}$$

C. Turbulence Model for Fluid Phase

RNG K-epsilon model was selected for modeling the turbulence of the flow. The Reynolds stress tensor for fluid phase is given by

$$\overline{\overline{\tau_{t,f}}} = -\frac{2}{3}\left(\rho_f k_f + \mu_{t,f}\nabla\vec{v}_f\right)\bar{\bar{I}} + \mu_{t,f}(\nabla\vec{v}_f + \nabla\vec{v}_f^{\,tr}) \tag{4}$$

D. Turbulence Model for Solid Phase

$$\tau_{F,sf} = \alpha_s \rho_f K_{sf}^{-1}\left(\frac{\rho_s}{\rho_f} + C_{vm}\right) \tag{5}$$

E. Wall Functions

Collection of empirical and semi-empirical formulae which help in calculating the simulated results with higher precision at the wall boundary of the flow domain are called wall functions. Because of the presence of higher gradient terms in the flow problem, the near wall boundary needs special treatment during the analysis. This can be achieved by selecting proper wall functions available in turbulence models. Standard wall function of RNG K-epsilon model has been selected for this present study.

IV. NUMERICAL SOLUTION

A. *Generation of Geometry and Mesh*

103 mm diameter and 5.5 m long pipe was modeled in ANSYS workbench. Hexa core structured elements with non uniform meshing was introduced for mesh generation. Near wall boundary was refined by selecting seven smooth transition inflation layers of size 0.272 with a growth ratio of 1.2. Number of elements of the generated mesh was found to be 213594.

B. *Boundary Conditions*

Three boundary conditions namely inlet, outlet and wall boundary was imposed on the flow domain. At inlet boundary mixture velocity and volume fraction for solid and fluid phase was introduced. At wall boundary no slip boundary conditions was selected and the outlet boundary was treated as pressure outlet.

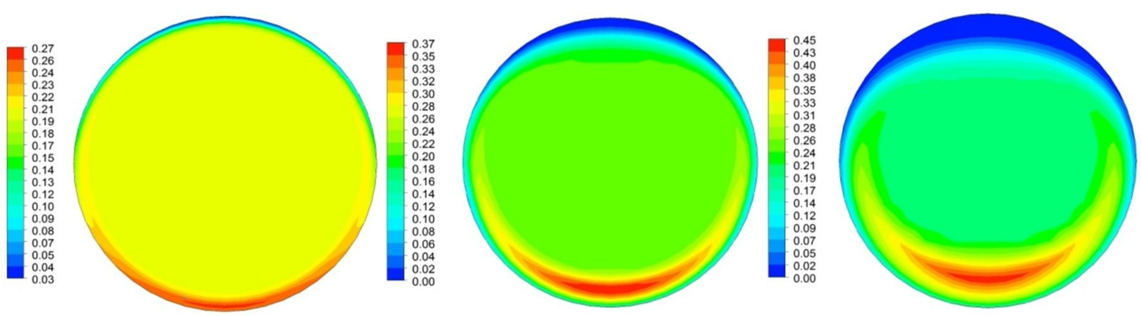
C. *Solution Strategy*

Mass and momentum equations were solved using finite difference approach. Turbulent kinetic energy,momentum equation and turbulent dissipation rate were solved using 2nd order discretization scheme while solid volumetric concentration was solved selecting 1st order disretization scheme.

V. RESULTS AND DISCUSSION

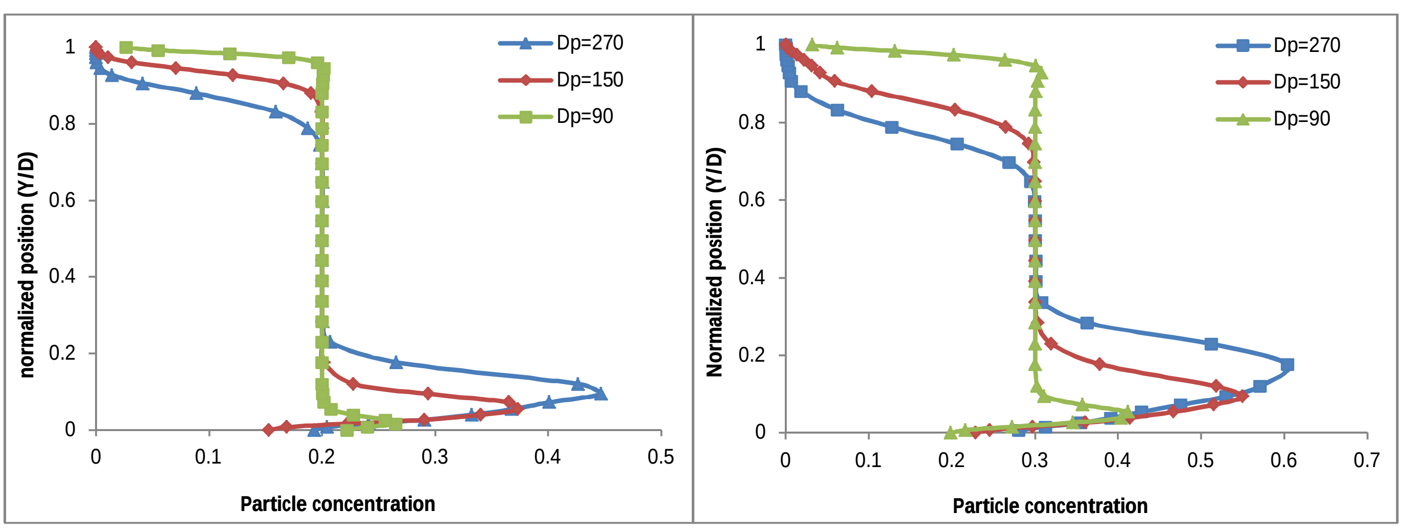
A. *Effect of volume fraction and size of the solid particles on particle concentration distribution and particle flow velocity distribution*

Effects of particle size and solid volume fraction on particle concentration distribution and velocity distribution are discussed in this section.



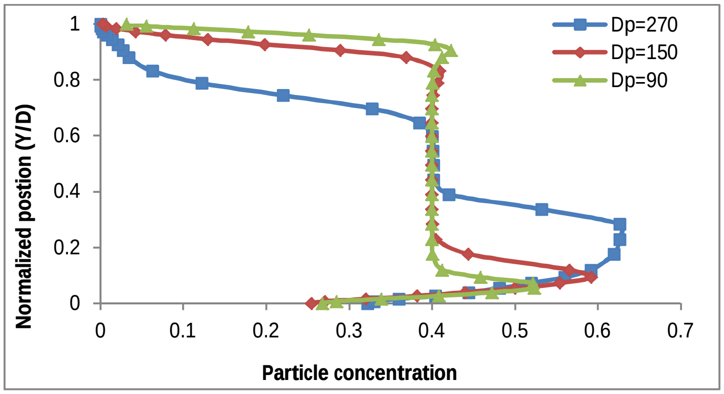
(a) Particle size= 90µm (b) Particle size=150µm (c) Particle size=270µm

Fig. 1: Contours of particle concentration distribution for different particle size at 20% volume fraction.



(a) Volume fraction=20% (b) Volume fraction=30%

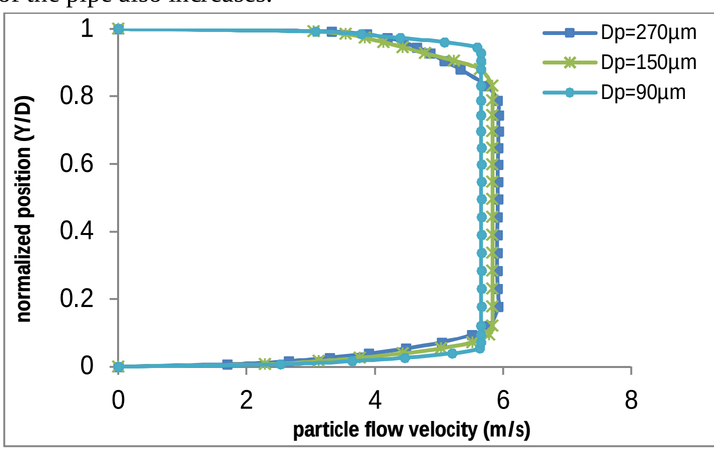
International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue V, May 2018- Available at www.ijraset.com



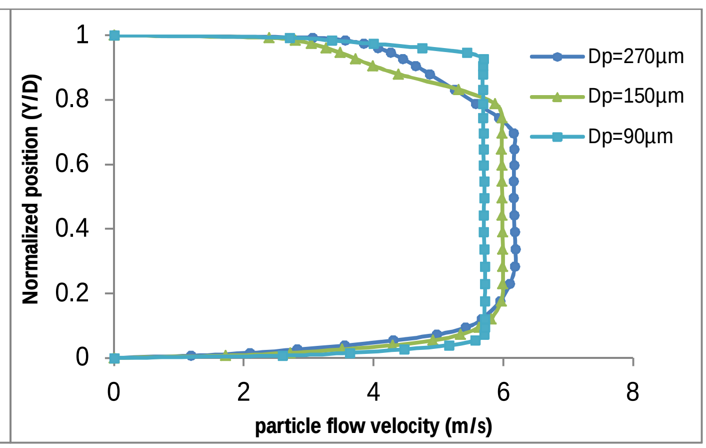
(c) Volume fraction=40%

Fig. 2: Plots of particle concentration distribution for various particle sizes at volume fraction of (a) 20%, (b) 30%, (c) 40% respectively.

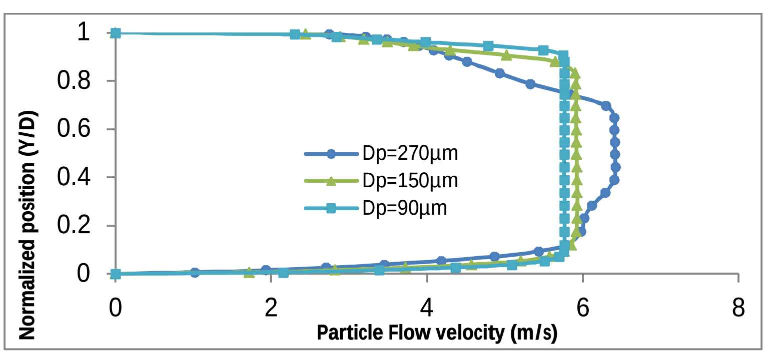
Fig. 1 shows the contours of concentration distribution of soliAd particles for different particle size at 20% volume fraction across the cross section of the pipe outlet. Fig. 2 shows the plots of solid particle concentration distribution along vertical centerline of the pipe cross section at pipe outlet for different particle size at 20%, 30% and 40% volume fraction respectively. From these figures it can be observed that when the particle size is smaller (say 90µm) the distribution of particle concentration is symmetrical and the particles show buoyant nature thereby suspending with the fluid. As particle size increases (say 270µm) the concentration distribution becomes asymmetrical and particles tend to settle down at the bottom o the pipe due to gravitational effect. Furthermore it can be noticed that when volume fraction increases from 10% to 40% for each particle size the concentration of particles become more suspended. Moreover with the increase in volume fraction the thickness of the bed formed by the solid particles at the bottom of the pipe also increases.



(a) Volume fraction=20%



(b) Volume fraction=30%



(c) Volume fraction=40%

Fig. 3: Plots of particle velocity distribution for different particle size at (a) 20%, (b) 30% and (c) 40% volume fraction respectively.

Fig. 3 represents the plots of solid particle flow velocity distribution for different particle sizes at 20%, 30% and 40% volume fraction respectively. From this figure it can be observed that for each constant volume fraction the particle flow velocity increases as the particle size increases from 90µm to 270µm. large particle sizes are related to the rise in turbulence in the flow because of strong particle-particle bond formation, rise in turbulence causes a surge in the flow velocity. Further it can be noticed that for each constant particle size the particle flow velocity becomes more distorted and asymmetrical in nature as the volume fraction changes from 20% to 40%. This is due to the fact that high volume fraction causes a hike in flow turbulence as well as accumulating more space at the pipe bottom which causes a reduction in effective flow area and increase in flow velocity.

B. Validation of the Study

The simulated results of pressure drop against particle flow velocity for a particle size of 270µ at 20% volume fraction is compared with the experimental data collected from the work of Gillies et. al [27]. The comparison showed that the simulated results are in good agreement with the experimental data.

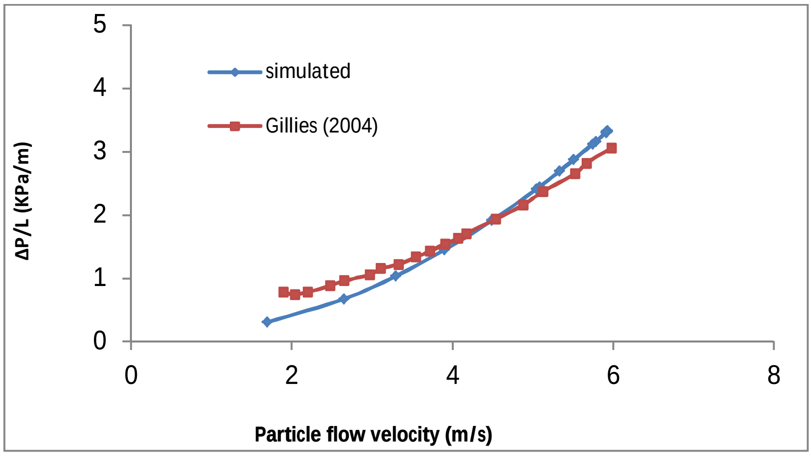


Fig. 4: Comparison of simulated pressure drop and experimental pressure drop over particle flow velocity for 270µm sand particle at 20% volume fraction.

VI. CONCLUSION

In this present study various flow parameters of a sand-water slurry flow through horizontal pipeline of 103mm diameter and 5.5m length is numerically analyzed using CFD technique. 90µm, 150µm and 270µm sand particle size with 20%, 30% and 40% volume fraction were considered in this analysis and their effect on particle concentration distribution and particle flow velocity were discussed. It was concluded that the particle size affects concentration distribution smaller particle being distributed symmetrically with the fluid and remains suspended in the fluid. Whereas the larger particles tend to accumulate at the bottom of the pipe forming a bed and reducing the effective flow area. Further conclusions can be drawn that the thickness of the bed becomes more at high volume fraction for all particle size. Particle size and volume fraction also affects the particle flow velocity; larger particle size at high volume fraction distorts the particle flow velocity distribution making it more asymmetrical.

REFERENCES

[1] O'Brien, M.P., 1933. Review of the theory of turbulent flow and its relations to sediment transport. Transaction of the American Geophysical Union 14, 487–491.

[2] . Rouse, H., 1937. Modern conceptions of the mechanics of fluid turbulence. Transactions of ASCE, 102, 463–505.

[3] Kaushal et. al (2005). Effect of particle size distribution on pressure drop and concentration profile in pipeline flow of highly concentrated slurry. International Journal of Multiphase Flow, 31, 809–823.

[4] . Kaushal, D.R., Tomita, Y., 2002. Solid concentration profiles and pressure drop in pipeline flow of multisized particulate slurries. International Journal of Multiphase Flow, 28, 1697–1717.

[5] Kaushal, D.R., Tomita, Y., 2003. Comparative study of pressure drop in multisized particulate slurry flow through pipe and rectangular duct. International Journal of Multiphase Flow, 29, 1473–1487.

[6] Gillies, R.G., Shook, C.A., 2000. Modeling high concentration settling slurry flows. The Canadian Journal of Chemical Engineering, 78, 709–716.

[7] Gillies, R.G., Hill, K.B., Mckibben, M.J., Shook, C.A., 1999. Solids transport by laminar Newtonian flows. Powder Technology, 104, 269–277.

[8] Gillies, R.G., Shook, C.A., Wilson, K.C., 1991. An improved two layer model for horizontal slurry pipeline flow. The Canadian Journal of Chemical Engineering, 69, 173–178

[9] Roco, M.C., Shook, C.A., 1984. Computational methods for coal slurry pipeline with heterogeneous size distribution. Powder Technology, 39, 159–176.

[10] Roco, M.C., Shook, C.A., 1983. Modeling of slurry flow: The effect of particle size. The Canadian Journal of Chemical Engineering, 61, 494–503.

[11] . Seshadri, V., Malhotra, R.C., Sundar, K.S., 1982. Concentration and size distribution of solids in a slurry pipeline. In: Proc. 11th Nat. Conference on Fluid mechanics and fluid power, B.H.E.L., Hyderabad.

[12] Seshadri, V., 1982. Basic process design for a slurry pipeline. In: Proc. The Short Term Course on Design of pipelines for transporting liquid and solid materials, IIT, Delhi.

[13] Karabelas, A.J., 1977. Vertical distribution of dilute suspensions in turbulent pipe flow. AIChE Journal, 23, 426–434.

[14] Shook, C.A., Daniel, S.M., Scott, J.A., Holgate, J.P., 1968. Flow of suspensions in pipelines. The Canadian Journal of Chemical Engineering, 46, 238–244.

[15] Shook, C.A., Daniel, S.M., 1965. Flow of suspensions of solids in pipeline: I. Flow with a stable stationary deposit. The Canadian Journal of Chemical Engineering, 43, 56–72.

[16] Wilson, K.C., Clift, R., Sellgren, A., 2002. Operating points for pipelines carrying concentrated heterogeneous slurries. Powder Technology, 123, 19–24.

[17] . Kaushal, D.R., & Tomita, Y. (2007). Experimental investigation for near-wall lift of coarser particles in slurry pipelines using γ-ray densitometer. Powder technology, 172(3), 177-187

[18] Ghanta, K.C., Purohit, N.K., 1999. Pressure drop prediction in hydraulic transport of bi-dispersed particles of coal and copper ore in pipeline. The Canadian Journal of Chemical Engineering, 77, 127–131.

[19] . Mishra, R., Singh, S.N., Seshadri, V., 1998. Improved model for prediction of pressure drop and velocity field in multisized particulate slurry flow through horizontal pipes. Powder Handling and Processing Journal, 10, 279–289.

[20] Sundqvist, A., Sellgren, A., Addie, G., 1996. Slurry pipeline friction losses for coarse and high density products. Powder Technology, 89, 19–28.

[21] Doron, P., Granica, D., Barnea, D., 1987. Slurry flow in horizontal pipes-experimental and modeling. International Journal of Multiphase Flow, 13, 535–547.

[22] Wasp, E.J., Aude, T.C., Kenny, J.P., Seiter, R.H., Jacques, R.B., 1970. Deposition velocities, transition velocities and spatial distribution of solids in slurry pipelines. In: Proc. Hydro transport 1, BHRA Fluid Engineering, Coventry, UK, paper H4.2, pp. 53–76.

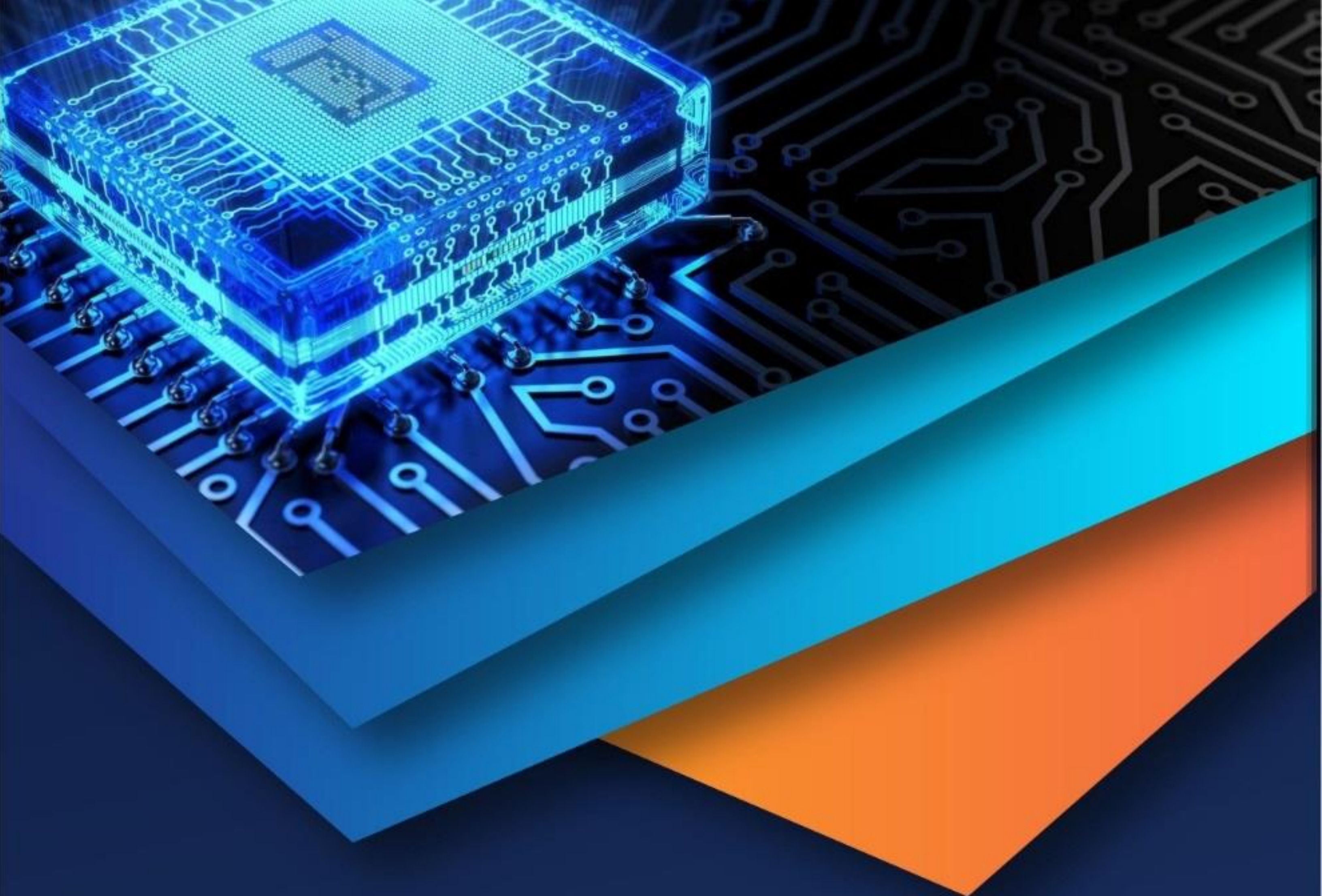
[23] Gopaliya, M. K., & Kaushal, D. R. (2016). Modeling of sand-water slurry flow through horizontal pipe using CFD. Journal of Hydrology and Hydromechanics, 64(3), 261-272.

[24] Kumar, A. (2016). CFD Modeling for Slurry Flow Through Bends and Straight Pipe Line. In CAD/CAM, Robotics and Factories of the Future (pp. 615-623). Springer, New Delhi.

[25] Kumar, U., Singh, S. N., & Seshadri, V. Experimental Investigation on pressure drop characteristics of Bi-modal slurry flow in a straight horizontal pipe.

[26] Nabil, T., El-Sawaf, I., & El-Nahhas, K. (2013, November). Computational fluid dynamics simulation of the solid-liquid slurry flow in a pipeline. In Proc. 17th International Water Technologies Conference IWTC17 (Vol. 57).

[27] Gillies, R. G., Shook, C. A., & Xu, J. (2004). Modelling heterogeneous slurry flows at high velocities. The Canadian Journal of Chemical Engineering, 82(5), 1060-1065



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY