Critical Velocities for Particle Transport from Experiments and CFD Simulations

Sajith Sajeev, Brenton McLaury, Siamack Shirazi

***Abstract*—**In the petroleum industry, solid particles are often present along with the produced fluids. It is imperative to keep particles from accumulating in flow lines. In this study, various experiments are conducted to study sand particle transport, where critical velocity is defined as the average fluid velocity to keep particles continuously moving. Many parameters related to the fluid, particles and pipe affect the transport process. Experimental results are presented varying the particle concentration. Additionally, CFD simulations using a discrete element modeling (DEM) approach are presented to compare with experimental result.

***Keywords*—**Particle transport, critical velocity, CFD, DEM.

I. Introduction

In the petroleum industry, oil, water, and gas are commonly produced and these fluids contain impurities. One of the impurities is sand. If the flow rate is above a certain velocity, the sand particles cause erosion in pipelines and equipment. If the flowrate is below a certain velocity, sand particles settle inside the pipeline. If the sand particles settle inside the pipeline, it leads to blockage and increased pressure drop. Hence, it is desirable to keep the particles continuously moving along the pipeline.

In this paper, computational fluid dynamics (CFD) is used to further analyze experimental results for sand particle transportation. In CFD, the fluid flow solution is obtained using a Eularian approach, and particle motion is obtained by applying a DEM.

II. Literature Review

A. Experiments

Many investigators have conducted experiments in liquid-sand flow. Durand and Condolios [1] conducted experiments in liquid-solid flows in horizontal pipelines and developed a correlation to predict the flow velocity needed to suspend particles. Oroskar and Turian [2] also developed a model to predict the flow velocity needed to keep particles suspended and studied the effect of particle diameter on critical velocity. Najimi [3] conducted experiments in liquid-sand and liquid-gas-sand flows at different concentration, particle diameter and viscosity. Najmi defined the critical velocity as the velocity at which the particles move continuously throughout the pipeline. Najimi reported that as concentration increases critical velocity increases. It is important to note that he examined concentrations at 1 percent by volume or less. Dabirian [4], [9]-[12] conducted experiments in gas-liquid-sand flows for horizontal and slightly inclined pipe.

B. CFD

The DEM approach was first developed by Cundall and Strack [8] to study the behavior of spheres. Many investigators have also performed CFD-DEM simulations. Parker [5] performed CFD-DEM validation for removal of gravel. Huang [6] conducted CFD-DEM coupling for the transient simulation of liquid-solid flow in a centrifugal pump. Akhshik [7] also conducted CFD-DEM stimulations for sand bed removal in the deviated well.

III. CFD Approach

In this work, simulations are performed using a combination of ANSYS Fluent 17.2.0 and EDEM 2.7. A simulation begins with a transient flow solution in Fluent. The Fluent-EDEM is coupled. The forces between fluid flow and particles are transferred using by a series of momentum source terms. The transient solid phase calculations then start in EDEM by considering the momentum source terms transferred from Fluent. The Fluent-EDEM coupling code then passes the drag forces on the particles to the Fluent mesh cells. The particle positions are then updated in EDEM. This particle position is then updated in Fluent, the iteration is then continued for the next time step.

DEM particle interaction is classified into two categories. The first category is for hard sphere model, where particles do not undergo any deformation. The second category is the soft sphere model where particles undergo deformation. Here the particle-particle and particle-wall interactions are assumed to cause no deformation, so the hard sphere model is used. The non-linear Hertz-Mindlin model is used. Since the heat transfer in the collison is assumed to be negligible, the Hertz-Mindlin heat transfer model is not used.

Lagrangian and Eulerian coupling models are available in Fluent-EDEM coupling. Eulerian coupling is used when the volume of the solids is high, more than 10%. The Lagrangian coupling is used when the volume percentage is less. Here since the volume percentage is small, Lagrangian coupling is used.

The experimental conditions and the corresponding experimental results used for performing and comparison of the CFD simulations are obtained from Najimi [3]. CFD simulations are performed for two different concentrations keeping the liquid velocity constant. Najimi [3] used water as

Sajith Sajeev is with the University of Tulsa, OK 74104 USA (phone: 918-625-2513; e-mail: sak795@utulsa.edu).
Brenton McLaury and Siamack Shirazi are with the Mechanical Engineering Department at The University of Tulsa, Tulsa, OK 74104 USA (e-mail: brenton-mclaury@utulsa.edu, siamack-shirazi@utulsa.edu).

the carrier fluid with a viscosity of 1 cP. 300 micron sand particles were used for the experiments. Najimi's experimental results showed that critical velocity increases with increase in volume concentration as presented in Table I.

TABLE I
VOLUME CONCENTRATION AND CRITICAL VELOCITY FOR 300 MICRON SAND

Volume Concentration (%)	Critical Velocity (m/s)
0.01	0.55
0.025	0.60

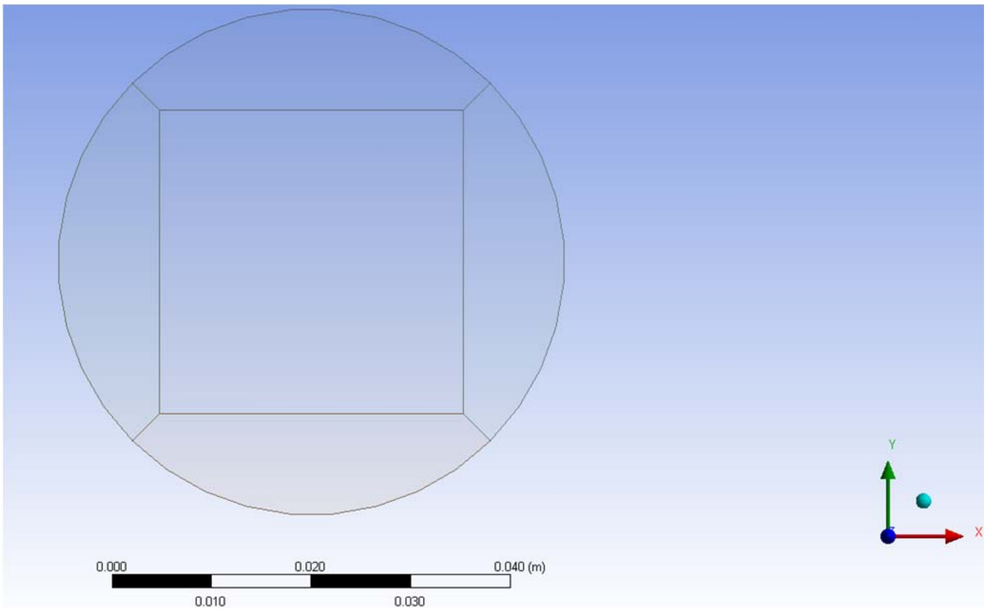

Fig. 1 Inlet face structured geometry

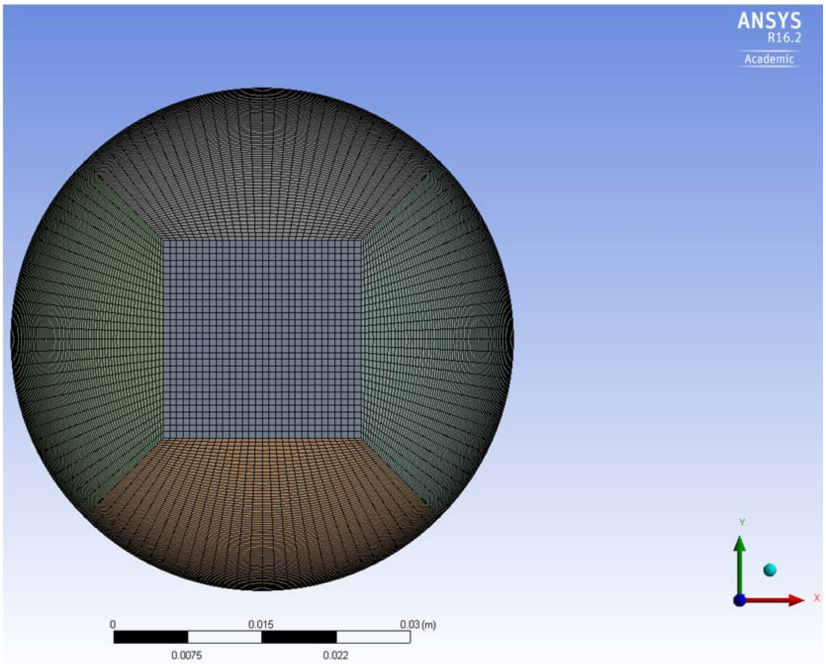

Fig. 2 Mesh for the fluent

CFD-DEM simulations were performed for volume concentrations of 0.01% and 0.025% with a fixed fluid velocity of 0.55 m/s with the goal of analyzing the difference in particle velocities with respect to concentration.

The geometry created is a pipe of 0.05 m diameter and length of 2 m. Fig. 1 shows the inlet face structured geometry

International Journal of Earth, Energy and Environmental Sciences
ISSN: 2517-942X
Vol:11, No:6, 2017

in fluent.

A structural mesh is created for the geometry in Fluent with an inflation layer created near the wall of the pipe to consider the boundary layer effect. The Fluent mesh has 3125000 elements and EDEM mesh has 279836150 elements. In EDEM, the mesh size recommended is less than the size of the smallest particle. Here a mesh size of 250 microns is used. Fig. 2 shows the mesh for Fluent, and Fig. 3 shows the mesh for EDEM.

A k-epsilon turbulence model is used in Fluent. A velocity inlet is used for inlet boundary condition. A pressure outlet is used for the outlet boundary condition. The sand particles are assumed as spheres. The acceleration due to gravity is taken into account and acts normal to the flow direction, since horizontal pipe is examined. Water is specified in Fluent as the fluid, and the properties of water are used in the cell zones. The diameter of the spheres is 300 microns, similar to the diameter of the sand particles used in the experiments. Fig. 4 shows the particle shape used in EDEM.

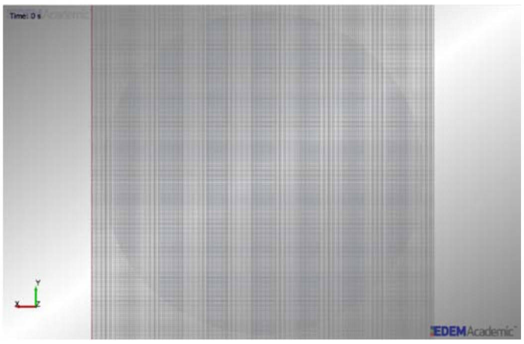

Fig. 3 Mesh for the EDEM

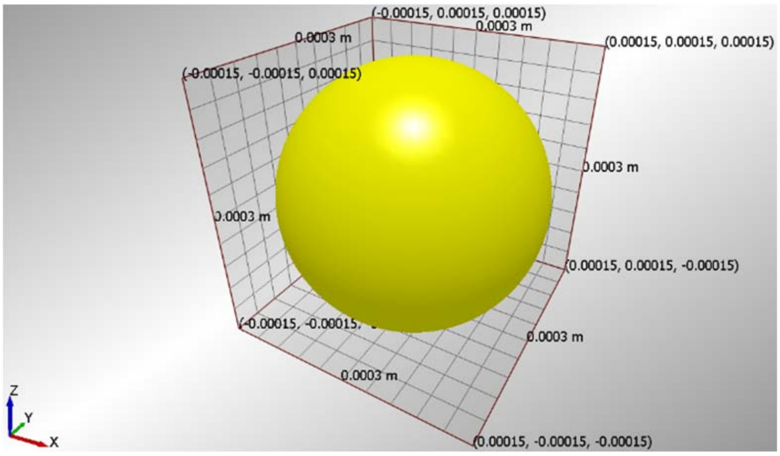

Fig. 4 Particle used in EDEM

The DEM properties of the sand particles and pipe wall are shown in Table II.

TABLE II
DEM PARAMETERS AND CONSTANTS

	Parameter	Constant
Sand	Poisson's Ratio	0.17
	Density	2650 Kg/m^3
	Shear Modulus	0.3 GPa
Pipe	Poisson's Ratio	0.3
	Density	1190 Kg/ m^3
	Shear Modulus	0.1 Gpa
Sand-Sand	Restitution	0.93
	Static Friction	0.63
Sand-Pipe	Restitution	0.7
	Static Friction	0.2

The time step in Fluent is 2e-03 s, and the time used in EDEM is 100 times smaller than that of Fluent, 2e-05 s. The flow is initially allowed to develop for 5 seconds in the Fluent simulation, and then the particles are injected. In the EDEM stimulation, the volume of the particles (spheres) in the fluid (water) is excluded, and the forces acting on the particles are transferred to the fluid providing two-way coupling (forces from particles are transmitted to the fluid and the forces from the fluid are also transferred to the particles).

IV. CFD RESULTS

The CFD results show the particle velocity at which the particle moves along the wall of the pipe for two different concentrations at the same fluid velocity.

The transient stimulation for solid particle concentration of 0.01% for 0 s, 6 s, 10 s and 16 s are Figs. 5 (a)-(d).

The transient stimulations for solid particle concentration of 0.025% for 0 s, 6 s, 10 s and 16 s are shown in Figs. 6 (a)-(d).

The results for particle velocity are shown in Table III.

TABLE III
CFD RESULT

Fluid Velocity (m/s)	Volume Concentration (%)	Particle Velocity (m/s)
0.55	0.01	0.20
0.55	0.025	0.16

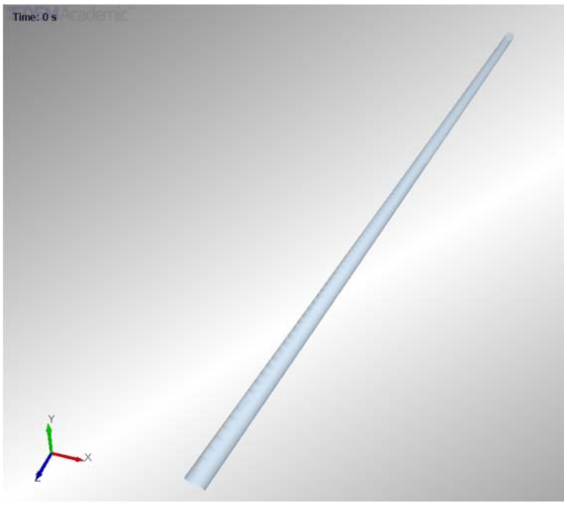

Fig. 5 (a) Stimulation at time 0 s

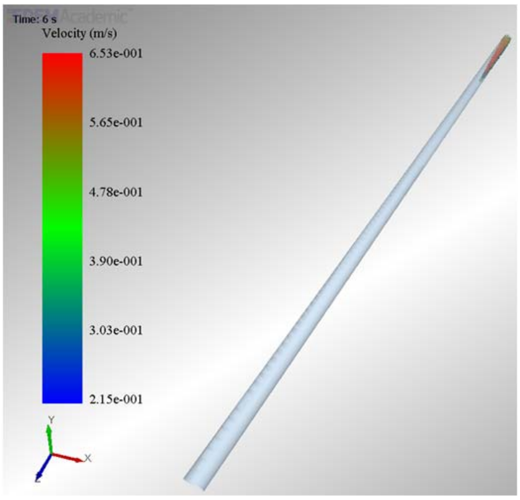

Fig. 5 (b) Stimulation at time 6 s

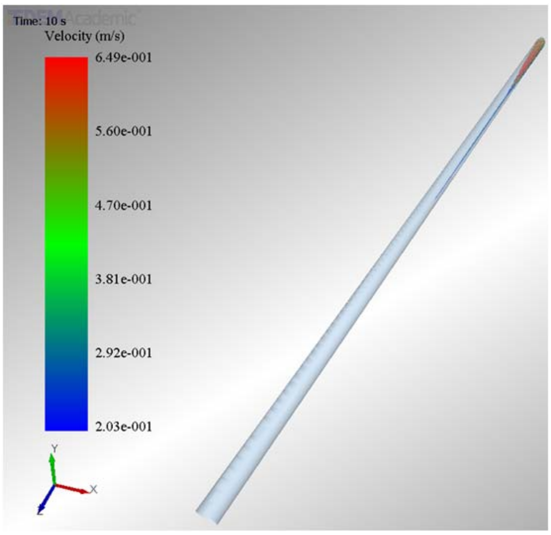

Fig. 5 (c) Stimulation at time 10 s

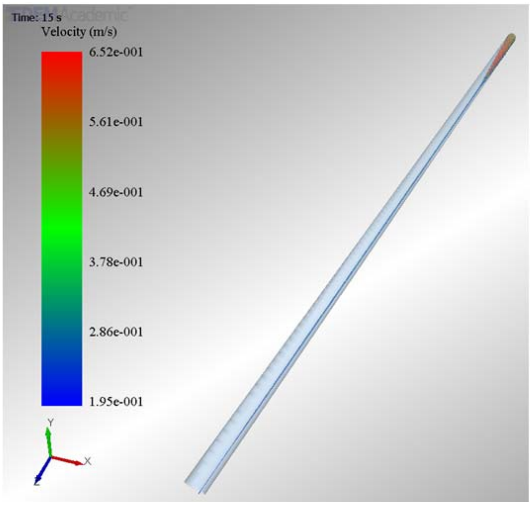

Fig. 5 (d) Stimulation at time 15 s

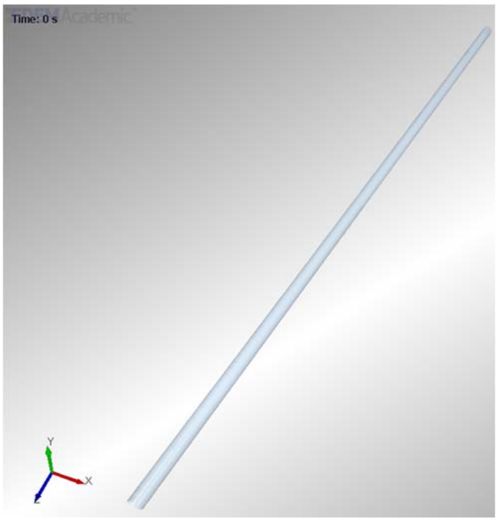

Fig. 6 (a) Stimulation at time 0 s

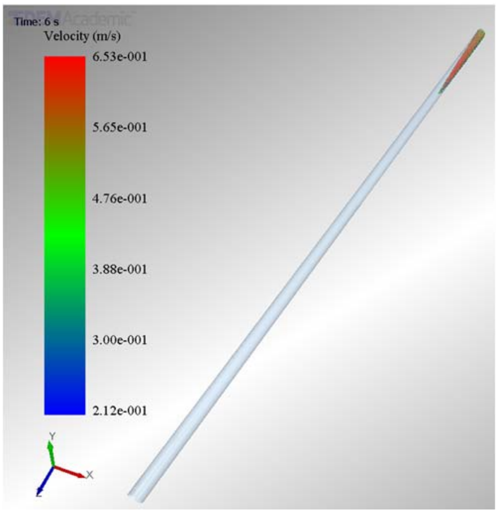

Fig. 6 (b) Stimulation at time 6 s

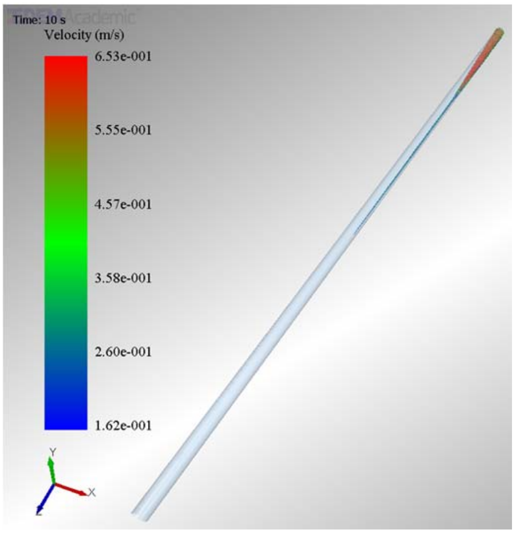

Fig. 6 (c) Stimulation at time 10 s

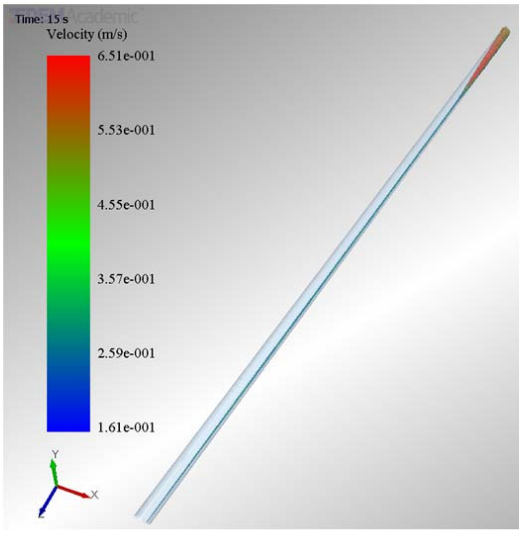

Fig. 6 (d) Stimulation at time 15 s

V. Conclusion

A CFD-DEM approach was used to study particle motion in water flowing in a horizontal pipe. The particle velocity decreases with increase in the particle volume concentration for a constant fluid flow rate. The CFD-DEM simulations demonstrate that there is a relation between particle velocity and particle concentration. The trend obtained is in agreement with experimental results.

Acknowledgment

The authors would like to thank the Tulsa University Sand Management Projects and The University of Tulsa for financial support.

References

[1] Durand, R., and Condolios, E, “Experimental Investigation of the Transport of Solids in Pipes,” Deuxieme Journee de Hydraulique, Societe Hydrotechnique de France, 1952, pp. 29–55.
[2] Oroskar, A.R., and Turian, R.M., “The Critical Velocity in Pipeline Flow of Slurries”, AIChe Journal, Vol.26, No4, pp.550-558,1980
[3] Najimi. Kamyar, “ Particle Transport in Single_phase and Multiphase Horizontal Pipes with Emphasis in the Effect of Viscosity”,2015
[4] Dabirian. Ramin, “Modeling and Experimental Investigation of Sand Transport in Gas-Liquid Stratified Flow”, Ph.D. Dissertation. The University of Tulsa, 2016.
[5] Parker I.,D. Ackerman, G. Langefeld, S.Cole “Simulation of Sub-sea Gravel Removal using the Coupled Computational Fluid Dynamics-Discrete Element Method,”
[6] Si Huang, Xianghui Su, Guangqi Qiu “Transient Numerical Stimulation for Solid Flow in a Centrifugal Pump by DEM-CFD Coupling ,”
[7] Siamak Akhshik, Mehdi Behzad, Majid Rajabi, “CFD-DEM Simulation of the Hole Cleaning Process in a Deviated Well Drilling: The Effects of Particle Shape,”, Particuology 2015
[8] P.A. Cundall, O.D.L. Strack,” A Discrete Numerical Model for Granular Assembiles,” Geotechnique 1979 Vol 1, pp 47-65.
[9] Dabirian, R., Mohan, R., Shoham, O., and Kouba, G., 2016. Critical Sand Deposition Velocity for Gas-Liquid Stratified Flow in Horizontal Pipes. Journal of Natural Gas Science and Engineering, Vol. 33, pp. 1-11.
[10] Dabirian, R., Mohan, R., Shoham, O., and Kouba, G., 2015. Sand Transport in Stratified Flow in a Horizontal Pipeline. SPE Annual Technical Conference and Exhibition, Houston, USA.
[11] Dabirian, R., Mohan, R., Shoham, O., and Kouba, G., 2016 Solid Particles Flow Regimes in Air-Water Stratified Flow in a Horizontal Pipeline. Peer review journal of SPE Oil and Gas Facilities, SPE-174960
[12] Dabirian, R., Mohan, R., Shoham, O., and Kouba, G., 2016 Sand Flow Regimes in Slightly Upward Inclined Gas-Liquid Stratified Flow. ASME Fluids Engineering Conference, Washington, DC, USA