Jurnal
Teknologi

Full Paper

NON-DIMENSIONAL DISTRIBUTION PATTERN ANALYSIS OF PARTICLE TRANSPORTATION IN SIMPLIFIED PIPELINE SYSTEM

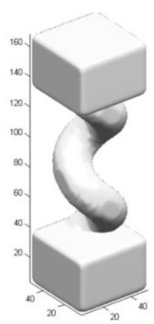
Article history
Received
15 April 2015
Received in revised form
29 September 2015
Accepted
12 November 2015

Mohd Zamani Ngali[a*], Kahar Osman[b], Nazri Huzaimi Zakaria[c]

[a]University Tun Hussein Onn Malaysia, P.0 Box 101, Parit Raja, Johor, Malaysia
[b]Faculty of Mechanical and Manufacturing, Universiti Teknologi Malaysia, UTM Johor Bahru, Johor, Malaysia
[c]Faculty of Mechanical Engineering, University Teknikal Malaysia Melaka, Hang Tuah Jaya, Durian Tunggal, Melaka, Malaysia

*Corresponding author
zamani@uthm.edu.my

Graphical abstract



Vertical curled pipeline

Abstract

Sustainable preservation of pipeline system that deal with particle transportation is more appealing these days. In petroleum industries for instance, sand transported through the pipelines pose serious problems ranging from blockage, corrosion, abrasion and reduction in pipe efficiency to loss of pipe integrity. Accurate four-dimensional simulation that caters the transient effect of the phenomena is used to promote sustainability in design, evaluation and maintenance procedures. This is employed to minimize conventional practices which are costly and inefficient. This work demonstrates the advantages of applying four-dimensional Splitting Fluid-Particle Solver to simulate particle transportation within a simplified pipeline system. Single-phase fluid with solid sphere particles are the assumptions while drift and gravitational forces are taken into account. Effect of fluid flow rate and particle weight alterations are observed within vertical curled and 2-1-2 segmental pipeline. Flow rate variation on multiple inputs shows that proper simulation is essential in order to predict fluid flow behavior prior to pipeline construction. Particle weight variation shows that simulation can lead to better prediction of potential areas of blockage, corrosion, abrasion and other piping system issues. This work proves that four-dimensional simulation can promote sustainability, cost effectiveness and efficiency of pipeline system management.

Keywords: Transportation, four-dimensional simulation, pipeline system

1.0 INTRODUCTION

When dealing with petroleum production operations, engineers have always found that crude oil and sand that eroded from the formation zones are often integrated as a flowing mixture throughout the pipeline system up to the well heads. This phenomenon continues upfront between well heads and flow stations.

The transported sand through the pipes leads to serious problems to the piping system. This includes blockage by the sand deposition, corrosion and

Nomenclature

F_E	External Force
F_B	Buoyant Force
F_D	Drag Force
C_D	Drag coefficient
A_p	Projected area of particle
$N_{Re,p}$	Particle Reynolds Number
ρ_f	Density of fluid
μ	Viscosity of fluid
ρ_p	Density of particle

abrasion by the sand hammering on the pipe surface. These problems will harshly reduce the pipe efficiency due to loss of pipe integrity. The loss in transport efficiency and maintenance lead to greater loss to the overall petroleum company revenue. Thus, more sustainable approach is required to reduce the loss of revenue on design slip-up and maintenance.

An in-house four-dimensional solver for fluid and particle distributions developed earlier by Ngali [1] is an excellent alternative for this purpose. This sort of integrated solver is capable of simulating sand deposition potential during the piping system design stage so that prevention and maintenance are optimized. Four-dimensional integrated fluid-particle solver is also suitable as pipeline system evaluation tool to assist Oil and Gas practitioners to evaluate critical parameters such as piping size and sand minimum transport condition (MTC) during the system design stage.

There are numbers of schemes available when simulation of fluid-particles interrelations are of interests. One of the most popular and suit the requirements of Splitting Methods by Karniadakis [2] which is used as the fluid solver in this work is Eulerian-Lagrangian scheme. Among others, Patankar *et al*. [3] are the pioneers to commence the application of Eulerian-Lagrangian technique in dealing with fluid-particle interactions.

Particle distribution solver is widely developed based on the Particle Equation of Motion formulation. The same concept is adopted in this work. The validation of fluid-particle solver coupling is focusing on the particle conformity characteristic. This attribute is vital in various phenomena such as sedimentation, deposition, ventilation and waste management.

Despite the fact that there are only limited academic resources discussing the characteristics and behaviours, there are few works have been done on particle conformity to fluid flow in the past few years. Kosinski *et al*. [4] among others, made a qualitative comparison of a single solid sphere particle path in a lid-driven cavity between his numerical solutions with experimental results by Tsorng *et al*. [5].

Another important parameter need to be considered for the simulation of fluid-particle distribution in pipeline system is the gravitational force. An important introduction of gravitational effect feature in Splitting Solver was made by Ngali *et al*. [1] where the aim was to improve the solver feasibility upon environmental air-particle related case studies. Even though the air and particle distribution analyses were done mainly for environmental studies such as dust ventilation, aerosol control or granular fertilizer blower distributions. The same solver is expected to be able to simulate this work's interest on pipeline system flow distribution.

The present study aims to simulate the air and particle distribution in the pipeline system. In reality, the pipeline system deal with varies shape and geometry such as horizontal, vertical, elbows, tees and etc fittings. Therefore, this simulationwill be developed in two types of case studies, namely vertical curled and 2-1-2 segmental pipelines. Based to this analysis, the different result will be obtained. The distribution of the particle for each case studies will be used as the primary result to solve the problem involving the real problem in the oil and gas industries.

2.0 RESEARCH METHODOLOGY

2.1 Particle Flow Theory

Particle Equation of Motion is used for the particle trajectory. This formulation is discretized separately from the fluid solver via Eulerian-Lagrangian technique. If particle of mass m moving through a fluid under the action of an external force F_E while the velocity of the particle relative to the fluid is u, the buoyant force on the particle is F_B and the drag is F_D, then we have,

$$m(du/dt) = F_E - F_B - F_D \tag{1}$$

Where,

$$F_D = C_D u^2 \rho_f A_p/2 \tag{2}$$

where C_D is the drag coefficient, A_p is the projected area of the particle in the plane perpendicular to the flow direction. Drag coefficient is a function of Reynolds number, denoted as,

$$N_{Re,p} = (u_s D_p \rho_f) / \mu_f \tag{3}$$

Where u_s is the velocity of approaching stream, Dp is the diameter of the particle, ρ_f is the density of fluid and μ_f is the viscosity of fluid. The drag coefficient is obtained by solving,

$$C_D = 24/(N_{Re,p}) \tag{4}$$

Equation (4) is the main formulation of drag coefficient since almost at any time, each particle Reymonds Number falls under unity value. In the developed algorithm however, conditional criteria for particle Reymonds Number equals to unity and greater than unity is thoroughly considered. Another important discussion for this work is on the introduction of F_B in equation (1), the buoyant force due to the gravitational correlation between particle and fluid dencity. The equation of buoyant force is,

$$F_B = mg\rho_f / \rho_p \tag{5}$$

Where m is the mass of the particle and g is the gravitational constant. From the equation (1), (2) and (5), we can combine it as,

$$d_u/d_t = [g(\rho_p - \rho_f)/\rho_p] - [C_D u^2 A_p \rho_f/2m] \qquad (6)$$

The change of particle velocity over a time increment can be found from the above equation.

2.2 Simulation Study

In crude oil transportation piping system, the momentum of sand particles carried across the pipe streamlines impinges the wall of the fittings. This undesirable fact results in erosion damage. Erosion of the fittings may result in failure of the piping system, which can be dangerous and expensive. Elbows and plugged tees are common geometries used in piping systems to transmit fluids. Both elbows and plugged tees are exposed to erosion when sand particles are present because particles deviate from the fluid streamlines and impact the wall when they pass through the geometries. Based on these cases, it is essential to have a method to determine the erosion rate for a given set of operating conditions to prevent any failures from occurring. This motivates the current work on the development of an in-house simulator with gravitational effect on particle distributions through pipelines. In order to investigate the effect of different types of pipeline arrangements, two distinctive pipelines with dissimilar characteristics are analysed, namely vertical curled and 2-1-2 segmental pipelines. The two geometrical structures are shown in Figure 1 below. These models represent simplified upstream and onshore petroleum pipelines respectively.

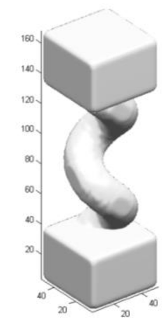
Four-dimensional simulations were done with dimensionless parameters. For the vertical curled pipeline,since the analysis was made non-dimensionally, the liquid flow rate was maintained to the value of one while the particle size was varied to investigate the effect of gravitational and drag forces. Particle diameter was set to be 0.0001 times the bottom compartment width with non-dimensional length of one. This setup representing relatively light particles.

Meanwhile, for relatively heavy particles, three times larger diameter was implemented. Since the flow analysis was made non-dimensional, the flow characteristics were set via the flow Reynolds Number while the flow velocities were kept as non-dimensional ratio. Vertical curled pipeline was set to have a suction flow of non-dimensional value of one flow rate at the top compartment while the four vertical wall boundaries of bottom compartment were kept 'open'. Particles were introduced at the floor of the bottom compartment to see the lifting phenomenon throughout the simulation.

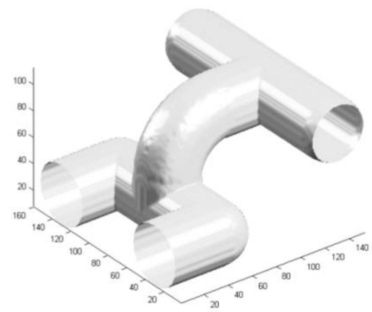
For the 2-1-2 segmental pipeline, the same dimensionless analyses were done. This time around, two sets of flow inlets were used for the same relatively light and heavy particle settings. The first set was done by keeping all openings 'open' except the top right aperture. Inward flow of dimensionless rate of one was used at this location while particles were introduced at the two bottom openings.

The second inlet flow set was arranged to investigate the effect of non-symmetric inlet flows with the same relatively light and heavy particles. The bottom left inlet was set with constant dimensionless flow rate value of one while the bottom right was set at flow rate value of 20 percent higher.

Analyses were done mainly to investigate the effect of particle size and mass on the distribution pattern through common pipeline arrangements. Qualitative observations and judgements were used widely in this account in order to convey the research findings as straightforward and as clear as possible.

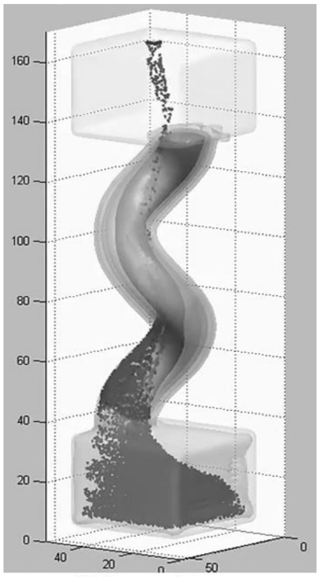


Vertical curled pipeline

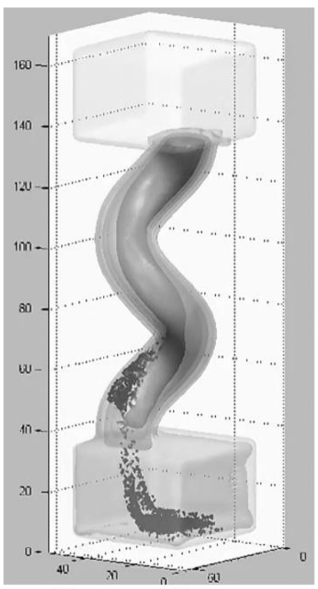


2-1-2 segmental pipeline

Figure1 Analyzed geometrical structures



Light particles



Heavy particles

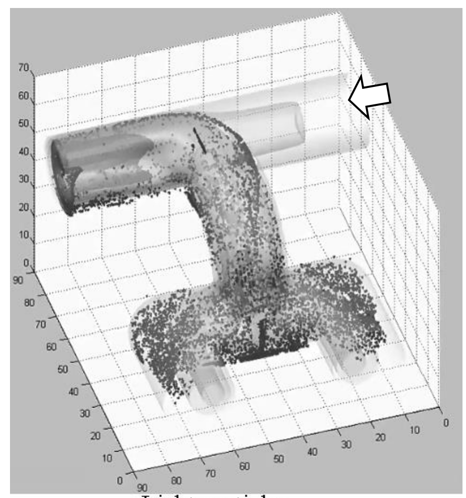
Figure 2 Light and heavy particle distributions in vertical curled pipeline

3.0 RESULTS AND DISCUSSION

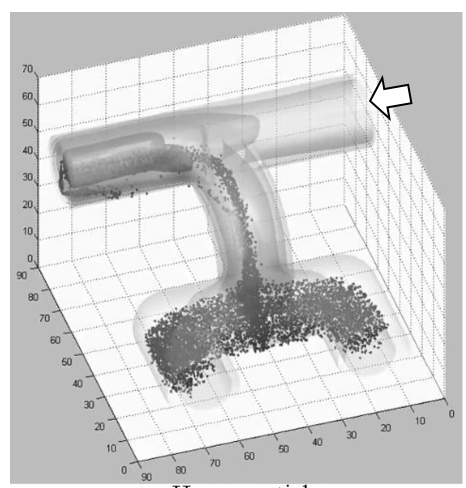
The first analysis was on the effect of mass and size of particles through vertical curled pipeline. Figure 2 shows the particle distribution at dimensionless time of 1.7. The layered surfaces within the pipeline illustrate the velocity isovalues where the darker layers represent higher velocity regions. Obviously, heavier particles lifted and moved upward relatively slower than the lighter particle. From the simulation of 5000 particles passing through the vertical curled pipeline, heavy particles acquired 2.5 times longer time as compared to light particles to fully exit the pipeline.

Another important observation is how the particles spread and hit the pipe surface. Lighter particles tend to spread to wider areas and collide with the wall in broader surface. Meanwhile for heavy particles, collision occured on more concentrated region. This condition is clearly shown in Figure 2 for the given instance of 1.7 dimensionless time where the collision surface area of heavy particles was less than 50% of the lighter particles collision surface area. Taking into account the momentum of the particles when collide with smaller pipe wall area, this simple simulation clearly showed that heavier particles promote higher degradation rate of the pipeline.

The severity of the collision impact by the particles to the pipeline inner surface is highly depends on the particles momentum. Since momentum is directly proportional to the mass and velocity of the particles, the severity of each impact can be compared by the product of particles mass and velocity. The defect probability at any given area throughout the pipeline can be determined by summing all the momentum of particles colliding on the specific region.



Light particles



Heavy particles

Figure 3 Light and heavy particle distributions in 2-1-2 segmental pipeline with single inflow

The first analysis on the 2-1-2 segmental pipeline was arranged in order that a single inlet flow involved as shown in Figure 3. The other three openings were left cavernous. With the same sets of light and heavy particles as it were earlier, the two bottom openings were introduced with the particle sets batch by batch. As the fluid flowed through the top straight pipe, the low pressure drove the fluid together with the particles in the lower pipelines toward the top straight pipe.

The simulation results once again showed that lighter particles flowed in wider distribution as compared to heavier particles. Due to the weight of the heavier particles, the path was concentrated at the bottom centre of the pipeline. This phenomenon could lead to particle deposition especially at pipe joints and junctions.

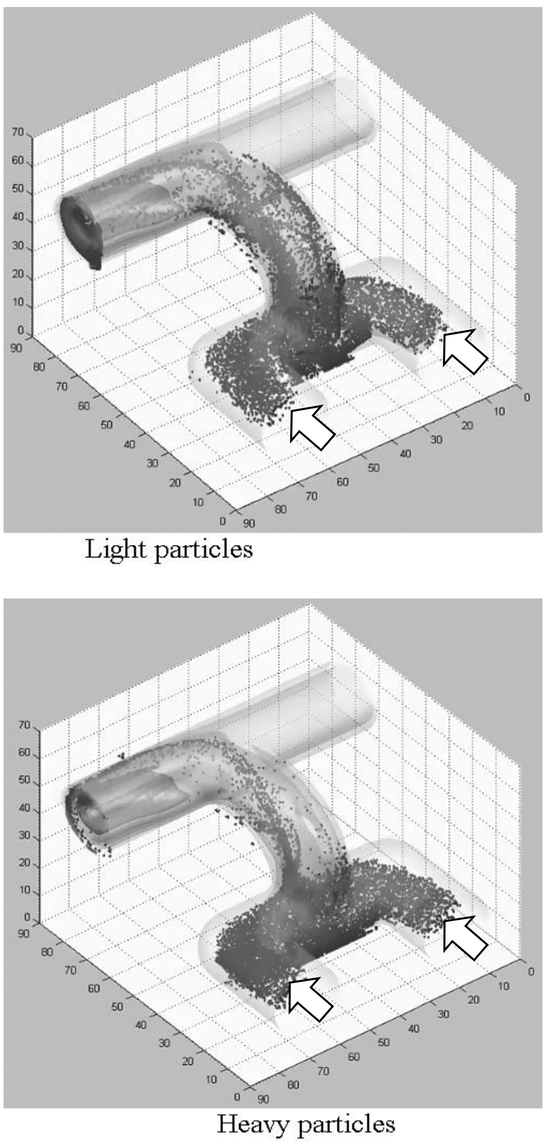


Figure 4 Light and heavy particle distributions in 2-1-2 segmental pipeline with two inflows

The final analysis was arranged with non-symmetric inlet flows as shown in Figure 4. This condition was achieved by increasing the flow rate of the bottom right inlet 20 percent higher than that through the bottom left inlet. An interesting phenomenon observed during the simulation where the top right opening was found to experience suction effect instead of expected lesser outflow. This occurrence led to none of the particles moved to the top right opening.

Particles distributions demonstrated almost identical pattern with the previous case study. The only difference was that the distributions were more chaotic than the previous ones. These were caused by the inlet arrangement. This last arrangement located two inlets at the bottom and these flows combined and mixed in higher velocity toward the single channel at the centre of the pipeline. This higher velocity flow that passed through the junctions created higher turbulence throughout the pipeline and caused the particles to distribute in more chaotic pattern.

The existence on turbulence flows in such pipeline structures contributes to more unpredictable particle existence consequences. As compared to the previous case study, these latest particle distributions were less predictable. Tendency for deposition, corrosion and other consequences were more arbitrarily occurred. The prediction complexity motivates us to apply simulation tools like the one used in this work since physical investigations are almost impractical to be carried out.

4.0 CONCLUSION

The main objective of this work which is to investigate the effect of particle weight on the particle distribution through simplified pipelines has been successfully accomplished. The application of in-house four-dimensional fluid-particle integrated solver was proven to be useful especially in comparing the particle distributions at any given instances. The importance on such simulations in design, evaluation and maintenance stages of pipeline system preservation become more significant when it deals with turbulent flows.

From the three analyses carried out in this work, we can conclude that the developed in-house integrated solver is capable of simulating fluid-particle distribution within pipeline system. The simulation results are capable of forecasting the potential hazards the pipeline may have even before the system is constructed. By simple calculation and analysis on the overall particle collision momentum and collision density, hazardous consequences such as blockage, corrosion, abrasion and reduction in pipe efficiency can be predicted accordingly.

Large industries such as petroleum upstream companies could benefit from this simulation tool with further advancement on the computational capabilities. Large scale equipments with huge demanding flow criteria acquire suitable upgrading on the current solver especially on computational memory management, boundary layer formulation and crunching efficiency

Acknowledgement

This research is supported by the Fundamental Research Grant Scheme (FRGS), Grant No: FRGS/2/2013 Vot 1420 by Ministry of Education, Malaysia.

References

[1] Ngali, M. Z., N. H. Zakaria, O. Kahar, A. Khalid, B. Manshoor, and I. Zaman. 2014. Gravitational Effect Formulation on In-House Air-Particle Flow Solver. *Applied Mechanics and Materials*, Trans Tech Publications. Switzerland. 660: 699-703.

[2] Karniadakis, G., M. Israeli, and S. Orszag. 1991. High-order Splitting Methods for the Incompressible Navier-Stokes Equations. *Journal of Computational Physics*. 97: 414-443.

[3] Patankar, N. A., and D. D. Joseph. 2001. Modeling and Numerical Simulation of Particulate Flows By The Eulerian-Lagrangian Approach. *International Journal of Multiphase Flow*. 27(10): 1659-1684.

[4] Kosinski, P., A. Kosinska, and A. C. Hoffmann. 2009. Simulation of Solid Particles Behaviour in a Driven Cavity Flow. *Powder Technology Journal*. 191: 327-339.

[5] Tsorng, S. J., H. Capart, J. S. Lai, and D. L. Young. 2006. Three-dimensional Tracking of the Long Time Trajectories of Suspended Particles in a Lid-Driven Cavity Flow. *Experiments in Fluids*. 40: 314-328.