Gas-Solid Flow Field and Particle Distribution in Bends with Circular Cross Section

Mehdi Bakhshesh and Majid Bakhshesh

Abstract– **Natural gas always contains quantitative solid particles in various points of transport and distribution lines. These particles can accumulate in small passages such as regulators, gas burner devices and make difficulties. So, before natural gas enters into the city pipelines and residential areas must be filtered in central gas station (CGS) in addition to pressure reduction. In this paper, effect of flow Reynolds number and Stokes number upon particle deposition in bends have been investigated and compare the results with what has been found in literature. In addition, the effect of curvature ratio on penetration of particles in a 90^o bend has been studied.**

Keywords– Two Phase Flows, Gas-Solid, Deposition, Penetration Particles and Bend

I. INTRODUCTION

Two phase gas-solid flow is observed in various engineering application and industrial process. Solid particles which are generated by corrosion within the gas pipeline can blemish to equipment of transport pipelines and central gas stations. So it is necessary to use suitable technique to remove solid particles out of the gas. This task is done by dry gas filters. Therefore, accumulation and deposition processes of solid particles are very important to investigate. Hence, it is useful to study the effect of solid particles size and quantities in bends and three ways.

Many studies are carried out to characterize the penetration of aerosol through bend.

A number of numerical studies on particulate flows in pipes have been done in 1991, three –dimensional calculations of horizontal pneumatic conveying were performed for particle loading ratios between 1 and 10 including two- way coupling and the effect of wall impact of non-spherical particles. However, collisions between particles, that are very important at such high loading, were not considered. Since large particles (i.e., particle diameter) were considered, flow turbulence effects on the particle motion could be neglected [1]. An attempt to predict pressure drop and saltation velocities in horizontal pneumatic conveying of coarse particles was recently presented by Tashiro et al. using an Euler- Lagrange approach and including lift force, inter particle collisions, particle- wall collisions without roughness. But, they didn't use two-way coupling for modeling of particle behavior. The comparison with experimental correlations however still showed considerable difference [2].

Various experimental studies have done in dust conveying systems with solids mass loading >0.3 by Hurber and Sommerfeld [3], Yilmaz and Levey [4]. It is generally accepted that the particle behavior in such flow systems is very complex and is strongly affected by particle size, wall roughness, conveying velocity, and the radius of the bend. However it is not clear from these studies whether the same also applies to a more dilute gas- solid flow regime (that is, solid mass loading << 0.3). In the investigation of Lumley, it calculated that particle-particle interaction is expected to be negligible for spherical particles with volumetric concentration below 0.3% [5].

At low Reynolds numbers, the fluid and particle velocities are identical, even for large concentration of solids, if the particles are neutrally buoyant.(Cox and Moson 1971,Yianneskss and Whitelaw1983) [6], [7].

Hajji and Pascal investigated dispersion of heavy particles in homogeneous isotropic turbulent gas-solid flows. They used Lagrange approach for simulation of dispersion of particles and investigated how fluid turbulence affected by inertia and drift velocity particles [8].

Hurber and Sommerfeld used an Euler/Lagrange approach for the calculation of dispersed gas-solid flows in pipe systems. The calculations included all important effects, such as turbulence, two-way coupling, particle transverse lift force and particle-wall collisions including wall roughness, and inter particle collisions [9].

Sommerfeled, Lain and Kussin used Reynolds stress model for studying the particle motion in a horizontal channel flow and calculated that for high mass loading of particles, wall roughness and inter –particle collisions have a dramatic influence on the particle behavior in a horizontal channel [10].

In the investigation of Kuan and Schwarz upon horizontal and vertical pipe, the motion of solid particles was analyzed. The importance of drag coefficient and inlet conditions to particle track calculations in a vertical duct was showed. In the analysis for two – phase in horizontal duct, it was shown that gravitational force has stronger influence over the distribution of particles [11].

Kuan .B.T. predicted single and two – phase flows in a 90o bend. In the simulation, gas turbulence was solved with a

Majid Bakhshesh is with the Department of Mechanical Engineering, Behbahan branch, Islamic Azad University, Behbahan, Iran

Mehdi Bakhshesh is with the Department of Mechanical Engineering, Behbahan branch, Islamic Azad University, Behbahan, Iran, (Corresponding author phone: +98-916-6717254; Fax: +98-612-4231020, Email: mehdi.bakhshesh @yahoo.com)

differential Reynolds stress models, whereas particle tracks were predicted through a Lagrange approach taking into account one-way coupling, turbulence dispersion, and pressure gradient effects. The calculations also considered effects such as transverse lift force which encourages particle setting in the gas flow and surface roughness which way critically affect near-wall development of fine particle concentration. Flow measurements, including mean and fluctuating gas and solid velocities were performed on the duct's plane of symmetry and results were compared with the laboratory data. It calculated that fine particles tend to better follow the gas motion. Also it was found that within the bend, the gas flow near the outer wall decelerates in presence of an adverse pressure gradient while the near-wall, flow accelerates under the influence of a favorable pressure gradient [12].

Dehbi studied particle-turbulence interactions close to walls of 90 bend where anisotropic effects are significant. Flow field was simulated in fluent software and then particle dispersion in turbulent boundary layer flows was analyzed. It calculated that for the smaller particle size where inertia affects reduced, model offers better prediction of turbulent field in the boundary layer [13].

Hadinoto and Curits studied the Reynolds number influence on the gas-phase turbulence in a vertical downward pipe. Also, they investigated the effect of particle in the turbulence modulation as a function of Reynolds. They concluded that in the presence of high inertia particles at a low particle loading, the intensity of gas-phase turbulence in the pipe core is increased with increasing Re relative to the unladen flow [14].

In the investigations of Mando and co-worker, the effects of particles on the turbulence equations using Euler/Lagrange approach for source terms was investigated. Results compared with what obtained with the so-called standard and consistent approaches and with experimental data and concluded that performance of model surpass that of both the standard and the consistent model [15].

In the investigations Ono and Kimura, two types of elbows with different curvature ratios were studied to determine the interaction between flow separation and the secondary flow due to the elbow curvature under high Reynolds number. They concluded that the flow separation always occurred in the short-elbow while the flow separation occurred intermittently in the long-elbow case [16].

Sun and Jiang studied deposition and distribution of particles due to the presence of duct bends using the Eulerian method and a Lagrangian trajectory method. They used Reynolds stress model and concluded that particle deposition has a peak value near bend outlet and enhanced deposition behind bends is result of bend-induced turbulence [17].

Lain and Somerfield investigated pneumatic conveying of spherical particles in horizontal ducts and a circular pipe. The Euler-Lagrange approach was used in three dimensional numerical calculations. They used k–ε and a Reynolds Stress turbulence model for two- way coupling and concluded that inter-particle collisions have an important influence velocity profile in a pipe for high particle volumetric percent and the wall collisions frequency is considerably higher in the pipe in comparison with the channel [18].

The main goal of this study is to evaluate the effects of curvature ratio, Reynolds number and stokes number upon deposition of particles in 90^o bends to validate the computational assumption.

In this research, computational grids are generated by Structure mesh. Afterwards, flow field is simulated by computational fluid dynamic. At last, particles are injected into the flow field. Meshes with different resolution are produced to insure grid independencies. Since Brownian motion and diffusion phenomenon are not effective in the behavior of particles and their deposition, in this research those parameters are neglected.

In dilute two phase flows drag and gravity forces are effective parameters whereas, in dense flows inter-particle collisions influence the particle motion [19].

Since volumetric concentration of particles in the flow field is below 0.1%, particle- particle interaction is neglected and assumption of one way coupling between particles and fluid is used [6]. This means that particles will not affect on flow fluid and after solution of flow field, behavior of particles are investigated by using Lagrange method.

In Euler- Lagrange approach, the fluid phase is treated as a continuum by solving the time-averaged Navier-Stokes equation, while the dispersed phase is solved by tracking a large number of particles through the calculated flow field. This approach is the most suitable way to predict the dispersion of particles in the flow field [21].

The dispersion of particles due to turbulence in the fluid phase is predicted using stochastic tracking model. The stochastic tracking model includes the effect of instantaneous turbulent velocity fluctuations on the particle trajectories through the use of stochastic method.

Important parameters of flow through bends of circular tubing are Reynolds number and curvature of the bend. These two parameters characterize non-dimensional Dean number which is defined as:

$$De = \frac{Re}{\delta^{1/2}} \quad (1)$$

Where, Re is the Reynolds number and the parameter δ is the curvature ratio of the bend [20].

$$\delta = \frac{R}{a} \quad (2)$$

Where R and a are curvature radius of the bend and tube internal radius respectively.

One of the important recognition parameters of gas-solid flow is non-dimensional Stocks number that is a feature for deposition of aerosol particles in a bend and is defined as:

$$Stk = \frac{C\, \rho_p\, {d_p}^2\, U}{9\, \mu\, d_t} \quad (3)$$

Where C is the Cunningham's correction slip factor and is very close to unity for particles further than 1µm, ρ_p is the particle density, d_p is the particle diameter and µ is the dynamic viscosity of fluid [13].

II. GOVERNING EQUATIONS

A. Gas phase

The gas flow properties and turbulence quantities are calculated by solving a set of Reynolds averaged Navier-stokes partial differential equations [2].

$$\frac{\partial\rho}{\partial t} + \nabla.\left(\rho\,\bar{\vec{V}}\right) = 0 \tag{4}$$

$$\frac{\partial}{\partial t}\left(\rho\bar{\vec{V}}\right) + \nabla.\left(\rho\overline{\vec{V}\vec{V}}\right) = -\nabla\bar{P} + \nabla.\left[\mu\left(\nabla\bar{\vec{V}} + \nabla\bar{\vec{V}}^{T}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{V_i'V_j'}\right) \tag{5}$$

B. Particle motion in dispersed phase

It is assumed that only drag and gravity is significant. Therefore, Particle velocities and trajectories in bends are calculated by solving the equation of motion for particles in Lagrange frame simultaneously [2].

$$\frac{du_{p,i}}{dt} = \frac{3\rho}{4\,\rho_p\,D_p}C_D\left(u_i - u_{p,i}\right)\left|\vec{U} - \vec{U}_p\right| + g$$
$$\frac{dx_{p,i}}{dt} = u_{p,i} \tag{6}$$

$\vec{U}$ and $\vec{U}_p$ are local gas and particle instantaneous velocities, respectively, u_i and $u_{p,i}$ are gas and particle velocity components, $x_{p,i}$ are the particle position coordinates;

The drag coefficient is obtained from the expression: [20]

$$C_D = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \tag{7}$$

Where, a_1, a_2, a_3 are constants which apply to spherical particles for wide range of Re number.

The relative Reynolds number defined as [20] :

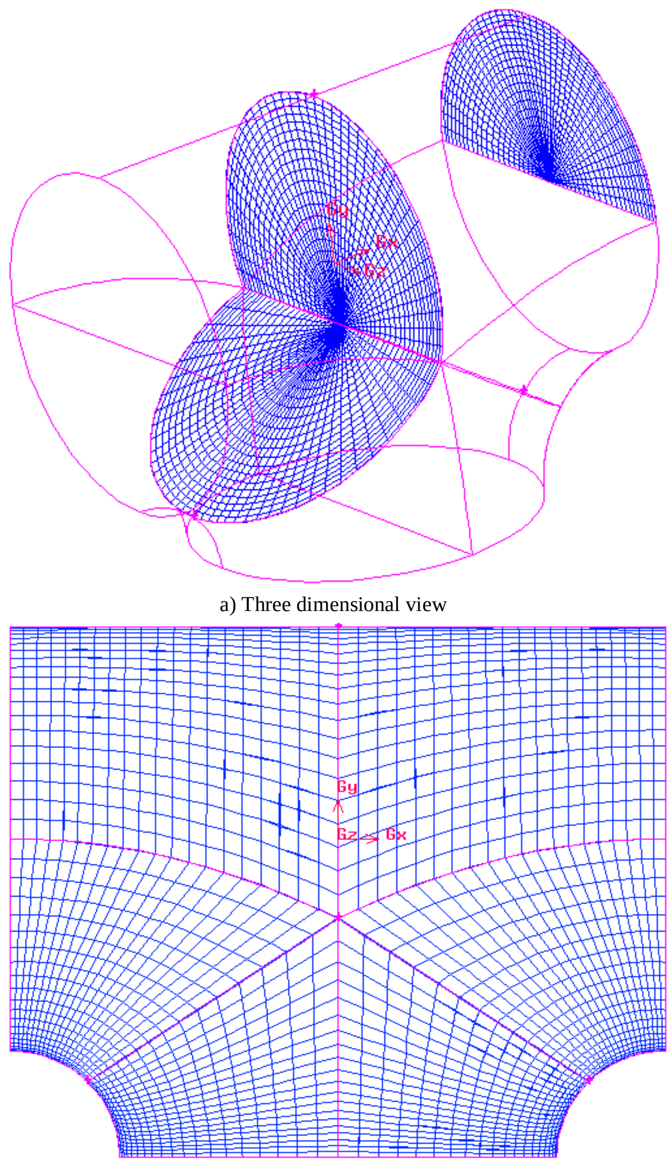
$$Re = \frac{\rho\,d_p\left|u - u_p\right|}{\mu} \tag{8}$$

In order to solve the equations of motion (6) for every tracking particle in the flow domain, instantaneous fluid velocity components at all particle locations need to be determined in advance. The inclusion of instantaneous fluid velocity components shows that the effects of turbulence are taken into account in the calculation of particle motion. The present work adopts a classical stochastic approach for the estimation of fluid fluctuating velocities.

III. GRID GENERATION

A. Strategy

O topology is used for grid generation in the pipe, and for T_junction in three ways, grids are generated as shown in Fig. 1. For correct solution of flow and exact modeling of boundary layer, grid must be fine in vicinity of walls. Other point that must be having in mind is compression or expansion in grid at various components of junctions such as pipe and knee or pipe and three ways. In Fig. 2 grid generation manner at pipe-knee has been shown. In this research, flow solution and particles injection are performed for 190651 cells in this geometry. About 76000 cells of total elements are triangle elements that have been used in pipe centerline and other cells are hexahedron element.



a) Three dimensional view

b) Two dimensional view T junction

Fig. 1. The manner of grid generation at three way

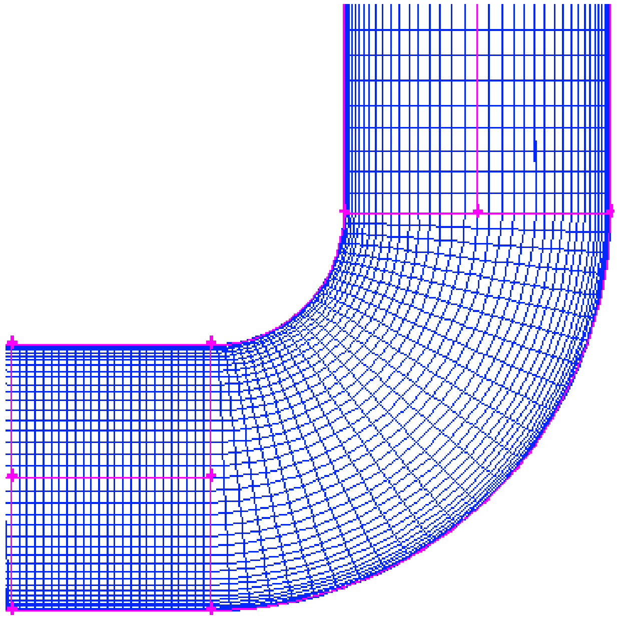


Fig. 2. The manner of grid generation at pipe- knee junction

B. Grid study for geometry

Three grids with different cells, have been generated. Table 1 shows no high difference between velocity and pressure values in three ways for 135667 and 1906851 cells.

Table. 1. Grid study for bend

Number of cells	Velocity at first three way	pressure at first three way
828288	10.1147	6.8517
1356670	10.9124	7.2619
1906851	10.9213	7.2859

IV. SIMULATION OF FLOW FIELD

For the numerical prediction of three- dimensional flow field and particle tracking, at first the flow field is computed and then a large number of particles are injected into the field.

Turbulent flow is modeled by k-ϵ and RSM. First k-ϵ model is used. Next, results that obtained from this model are used as primary solution for RSM.

Although k-ϵ turbulence model is used in a wide range of applications, for Eulerian two phase models [1] and Euler-Lagrange approach [2], it does not account for anisotropic flow [3]. Since the Reynolds stress model includes the effects of streamline curvature and rotation in the geometries with different curvatures, it is somewhat proper for showing of rotation flow in a pipeline with asymmetric branches. In addition, this model can predict turbulent anisotropic stress in various points of knee. [4], [8].

In all cases, velocity inlet and pressure outlet are used as boundary conditions. Velocity inlet changes for different cases, while zero gauge pressure is imposed at the exit.

V. PARTICLES INJECTION INTO THE FLOW FIELD

The injected particles velocity in inlet is set to the flow velocity [2]. The penetration of aerosol through bend is calculated by tracking a large number of particles released simultaneously. A plane is defined in inlet and particles are injected to flow by this plane. Particles are uniformly distributed over the face. It is worth that the rate of exit particles should be independent of particle number injected by inlet plane. So particles injection is done in any step with different particle number and this procedure is repeated until particles distribution in outlet is independent of inlet particle number. Injection with larger number of particles showed no change in the rate of penetration in bend. Since turbulent dispersion of particles is modeled by stochastic methods, for any inlet flow rate and any dimension particle, particles injection is done for several times. Final result is obtained by averaging of injection results.

For a 90° pipe - knee it is assumed that wall is stick. Therefore, some of particles can deposit in the knee according to flow velocity. So in this case, trap boundary condition on walls is used.

VI. RESULTS AND DISCUSSION

After grid study, three pipe-knees with different curvature ratio $\frac{R}{a} = 2, 4, 10$ have been analyzed. Where R and a are curvature radius of the bend and tube internal radius respectively. For all bends the tube diameter is 16 mm.

Velocity vectors in downstream of bend exit plane and its symmetry are presented in figures 3 and 4 respectively.

These results confirm the experimental previous researches that had been done by Mac Farland in 1997 [20].

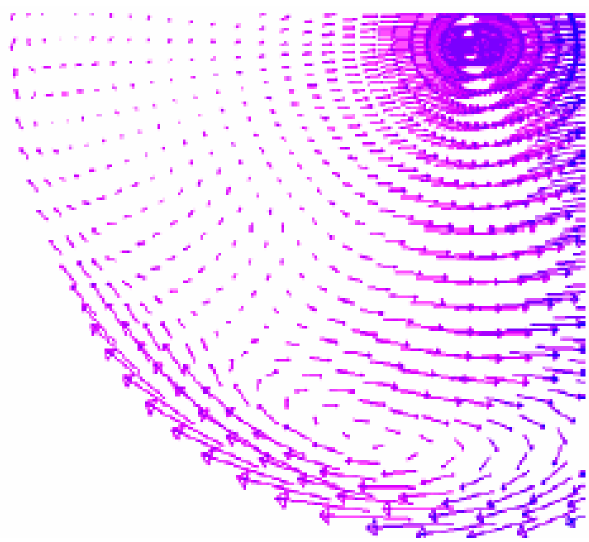


Fig. 3. Velocity vectors in downstream of bend exit plane

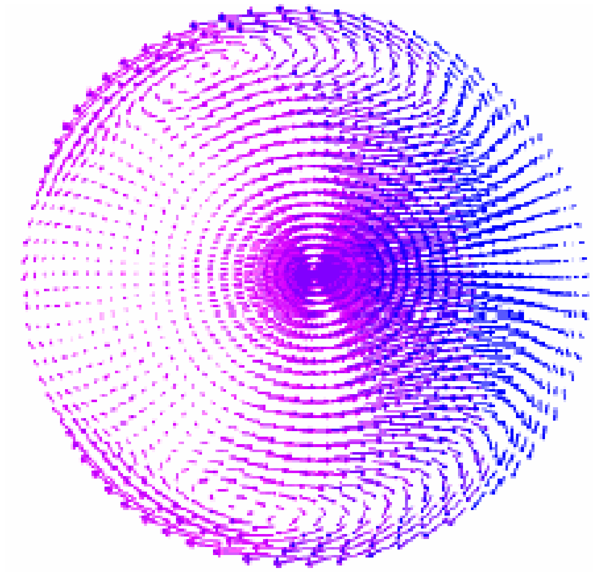


Fig. 4. Symmetry of velocity vectors in downstream of bend exit plane

The use of k-ϵ model for flow solution leads to high difference in percentage of particles penetration in comparison with experimental results. This has been shown in figure 5. Thus, this model cannot predict particle behavior properly against RSM.

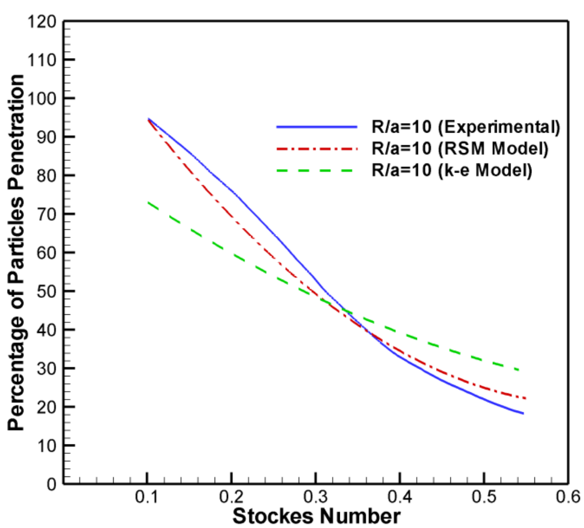


Fig. 5. K-ϵ Model in comparison with RSM and experimental samples

Fig. 5 shows that for three pipe-knees, particles penetration decreases with increase of stocks number. The stocks number is varied by changing the flow rate through bend. With increase of flow velocity, particles impact walls of pipe-knee with higher velocity when change their passage and to travel with flow. Therefore, more particles are trapped in these regions.

Effect of curvature ratio on particles behavior:

The effect of curvature ratio on the behavior of particles in a 90 bend has been investigated. It is assumed that the stocks number is varied by changing flow rate. Figure 9 shows that the particle penetration in bend increases when the curvature ratio rises from 2 to 4, but when curvature ratio increases from 4 to 10 the penetration of particles does not differ considerably. This shows the increase of bend curvature ratio for augmentation of transport particles is relatively effective and acceptable. If curvature ratio exceeds from a certain bound, only cause to increasing economic costs.

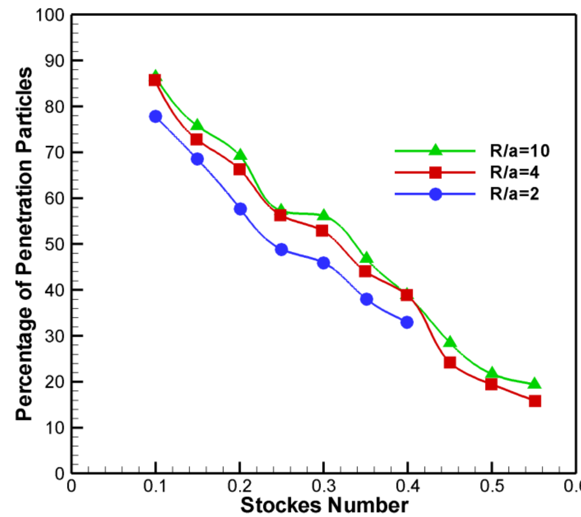


Fig. 6. Percentage of particles penetration versus stocks number

Effect of Reynolds number on particles penetration in bend

For considering the effect of Reynolds number on particles penetration, it is assumed that stokes number is constant and the Reynolds number varies. The results were compared with experimental data and showed reasonable agreement [20]. Figure 7 shows that with changing of stokes number, particles penetration does not change considerably. Therefore, the effect of Reynolds number on particles penetration can be neglected.

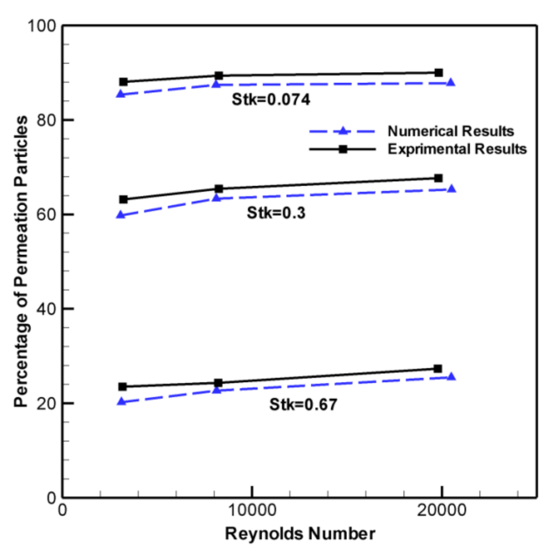


Fig. 7. Percentage of particles penetration versus Reynolds number

VII. CONCLUSIONS

Effect of flow Reynolds number and Stokes number upon particle deposition in bends have been investigated and compare the results with what has been found in literature. In addition, the effect of curvature ratio on penetration of particles in a 90^o bend has been studied. Deposition of particles increases in the pipe- knee with increase of inlet flow velocity and this is against of particles behavior in the pipe without knee and branch. Increase of bend curvature ratio to certain bound is relatively effective and acceptable for augmentation of transport particles. If curvature ratio exceeds from a certain bound, only cause to increasing economic costs. Also, Percentage exit particles does not vary by Reynolds number and it is only dependent on stokes number and results show reasonable agreement with those of McFarland research.

REFERENCES

[1] Y. Tsuji, N. Shen, "Lagrangian simulation of dilute gas – solid flows in a horizontal pipe", Advanced Powder Technology, Vol 2, Issue1, pp. 63-81, 1991.

[2] N. Hurber, M. Sommerfield, "Charactereization of the cross-sectional particle concentration distribution in pneumatic conveying systems", Powder Technology, Vol 79, Issue 3, pp. 191-210, 1994.

[3] H. Tashiro, X. Peng, "Numerical prediction of saltation velocity for gas-solid two-phase flow in a horizontal pipe", Powder Technology, Vol 91, Issue 2, pp. 141-146, 1997.

[4] A. Yilmaz, E. k. Levey, "Roping phenomena in pulverized coal conveying lines", Powder Technology, Vol 95, pp. 43-48, 1998.

[5] J. Lumley, "Two phase and non- Newtonian flows", Turbulence, Topics in Applied Physics", Vol 12, Pages 289-324, 1978.

[6] R. Cox, S. Mason, "Suspended particles in fluid flow through tubes", Annual Review of Fluid Mechanics, Vol 3, Pages 291-316, 1971.

[7] M. Yianneskis, J. Whitelaw, "*Velocity characteristics of pipe and jet flows with high particle concentrations*", Fluids Sec., Mechanical Engineering Department, Imperical College of Science and Thechnology, 1983.

[8] L., Hajji, B. Oesterel, "A simple description of some inertia effects in the behaviour of heavy particles in a turbulent gas flow", International Journal of Non-Linear Mechanics, Vol 31, Issue 3, pp. 387-403, 1996.

[9] N. Hurber, M. Sommerfield, "Modeling and calculation of dilute – phase pneumatic conveying in pipe systems". Powder Technology, Vol 99, Issue 1, pp. 90-101, 1998.

[10] M. Sommerfeled, S. Lain, J. Kussin, , "Analysis of transport effects of turbulent gas - particle flow in a horizontal channel", in *Fourth International Conference on Multiphase Flow, ICMF, 2001*.

[11] B. Kuan, M. Schwarz, "Numerical prediction of dilute particulate flows in horizontal and vertical ducts", Proceedings in *Fourth International Conference on CFD in the Minerals and Process Industries CSIRO, Australia, pp. 135-140, 2003.*

[12] B. T. Kuan, "CFD simulation of dilute gas – solid flows with different solid size distribution in a curved 90˚ duct bend", Journal of ANZIAM., Vol 46, pp. 44-763, 2005.

[13] A. Dehbi, "A CFD model for particle dispersion in turbulent boundary layer flows", Journal of Nuclear Engineering and Design, Vol 238, Issue 3, pp. 707-715, 2008.

[14] Hadinoto, Kunn., Curits, Jennifer. "Reynolds number dependence of gas-phase turbulence in particle-laden flows", Powder Technology, Vol 195, pp. 119-127, 2009.

[15] M. Mando, M. F. Lightstone, C. Rosendahl, "Turbulence modulation in dilute particle-laden flow". International Journal of Heat and Flluid Flow, Vol 30, Issue 2, pp. 331-338, 2009.

[16] A. Ono, , N. Kimura, H. Kamide, A. Tobita, "Influence of elbow curvature on flow structure at elbow outlet under high Reynolds number condition", Nuclear Engineering and Design, Vol 241, Issue 11, pp. 4409-4419, 2011.

[17] Ke. Sun, Lu, Lin., Jaing, Hai. "A numerical study of bend-induced particle deposition in and behind duct bends",. Bulding and Environment, Vol 52, pp. 77-87, 2012.

[18] Lain, Santiago., Sommerfeld ,Martin. "Numerical calculation of pneumatic conveying in horizontal channels and pipes", International Journal of Multiphase Flow, Vol 39, pp. 105-120, 2012.

[19] M. Founti, A. Klipfel, "Experimental and computational investigation of nearly dense two-phase sudden expansion flows", Experimental Thermal and Fluiid Science, Vol 17, Issue 1-2, pp 27-36, 1998.

[20] A. R. McFarland, H. Gong, "Aerosol deposition in bends with turbulent flow", Environ. Sci. Technol., Vol 31, pp. 3371-3377, 1997.

[21] Z. Zhang, Q. Chen, "Comparison of the Eulerian and Lagrangian methods for predicting particle transport in enclosed spaces", Atmospheric Environment, Vol41, Issue 25, pp. 5236-5248, 2007.