JJMIE

Volume 8 Number 2, April. 2014
ISSN 1995-6665
Pages 56 - 65

Jordan Journal of Mechanical and Industrial Engineering

Effect of Asymmetric Branches on Solid Particles Distribution in Central Gas Stations (CGSs)

M. Bakhshesh[1], E. Goshtasbi Rad[1*], M. Mehrvar[2]

[1] *Department of Mechanical Engineering, Shiraz University, Shiraz, Iran*
[2] *Department of Chemical Engineering , Ryerson University, 350 Victoria Street, Toronto, Ontario*

Abstract

Natural gas flows always contain solid particles in various points of transport and distribution lines. These particles can accumulate where the passages become narrow, whereas in regulators and gas burner devices, this might cause some problems. Therefore, natural gas should be filtered and then decompressed in central gas stations (CGSs) before entering into the pipelines of city or residential areas. Therefore, the kind and the number of filters should be determined, depending on particle accumulation.

In the present work, Lagrangian method is used for studying the accumulation and deposition of solid particles in a pipeline with asymmetric branches. Moreover, the effect of some parameters, such as Stokes number, Reynolds number, and symmetric branches on the amount of outlet particle accumulation, is investigated.

The effect of curvature ratio on the penetration of particles in a 90^o bend as well as the effect of Reynolds number upon particle deposition is evaluated and the results have been compared with the available data from the open literature. Finally, the dependence of volume portion of particles is shown with flow rate. The results show that the dependency is higher at high flow velocities. In all current cases, particles do not affect the flow field due to their low volume portion.

Keywords: *Two-Phase Gas-Solid Flows, Asymmetry Branch, Deposition, and Penetration Particles.*

1. Introduction

Two phase gas-solid flows can be detected in various engineering applications and industrial process. Solid particles can cause a lot of damage to transport pipeline equipment and central gas stations due to corrosion within gas pipelines. One major approach for removing solid particles from gas is using dry gas filters. Distribution of solid particles in central gas station is non-uniform because of asymmetric branches.

Many studies have been carried out to characterize the penetration of aerosol through bends. Cheng and Wang developed a model based on an analytical laminar flow solution to investigate the behavior of aerosol particles in 90^o bends. They proposed a correlation for particles penetration as a function of the Stokes number and curvature ratio [1].

Carne and Evanes utilized a numerical method to predict the behavior of aerosol particles in a laminar air flow of 90^o bend and curvature ratios from 4 to 20 and their results showed reasonable agreement with those of Chang and Wang [2].

Cheng and Wang re-examined the deposition of particles in pipe bends. They concluded that in the laminar flow regime the aerosol particle deposition was mainly a function of Stokes number and Reynolds number for different curvature ratios [3]. However, since the flow regime in transport systems with bends is usually turbulent, their model is not generally applicable.

Pui experimentally evaluated the deposition of aerosol particles in 90^o bends for different Reynolds numbers and curvature ratios. For turbulent flow, it was assumed that the penetration does not depend on neither curvature ratio nor Reynolds number and presented a correlation for predicting the aerosol particle in 90^o bend [4].

Tsai and Pui used a three-dimensional numerical model to examine the aerosol particle deposition for laminar flow in a pipe with a 90^o bend. They observed significant variation in the deposition efficiency by changing the amount of curvature ratio. They also witnessed the effect of the inflow velocity profile on the deposition efficiency. For parabolic inlet velocity profile, their results could be confirmed with the results of Cheng and Wang [5].

One of the first numerical studies of the particle motion in a horizontal pipe was performed by Ottjes who considered the effects of Magnus lift force and inelastic particle-wall collisions [6]. Moreover, more complex geometries were considered by Tsuji and Morikaw who utilized a numerical simulation of gas-solid flow in a

* Corresponding author. e-mail: goshtasb@shirazu.ac.ir.

horizontal channel [7]. In all the above investigations, the effect of particles on the gas flow has been neglected.

A remarkable amount of numerical studies on particulate flows in pipes have been done in 1991. Three-dimensional solutions of horizontal pneumatic conveying were presented for particle loading ratios between 1 and 10 including two-way coupling and wall impact effect of non-spherical particles. Collisions of particles, recognized as an effective and important parameter at high loading flow, has not been considered in their work. Flow turbulence effects on the particle motion could be neglected due to the large size of particles [8].

An effort was made to predict the pressure drop in horizontal pneumatic conveying of coarse particles by Tashiro [9]. Tashiro use an Euler-Lagrange approach and considered the effect of the lift force, inter particle collisions, and particle-wall collisions without roughness. However, a two-way coupling for modeling of particle behavior has not been considered. Further, the results have shown considerable differences with experimental correlations.

Various experimental studies have been done by Hurber and Sommerfeld [10] and Yilmaz and Levey [11] on dust conveying systems with solids mass loading of larger than 0.3. It has been observed that the particle behavior in such flow systems is partly complex and strongly affected by the particle size, the wall roughness, the conveying velocity, and the radius of the bend. However, the low limit of gas-solid flow dilution in these models has not been elucidated.

Lumley estimated that particle-particle interaction is negligible for spherical particles with volumetric concentration below 0.3% [12].

At low Reynolds numbers, the fluid and particle velocities are equal for the neutrally buoyant particles [13, 14].

Hajji and Pascal investigated the dispersion of heavy particles in homogeneous isotropic turbulent gas-solid flows. The Lagrangian approach was used for the simulation of particles' dispersion and the effects of particles' inertia and drift velocity on fluid turbulence were investigated [15].

In order to study the dispersed gas-solid flows in pipe systems, Hurber and Sommerfeld made an Eulerian/Lagrangian approach. They considered the effects of important parameters such as turbulence, two-way coupling, particle transverse lift force, particle-wall collisions including wall roughness and inter particle collisions [16].

Sommerfeled, Lain and Kussin used Reynolds stress model for studying the particle motion in a horizontal channel flow and observed that for high mass loading of particles, wall roughness and inter–particle collisions have a significant impact on the particle behavior [17].

The motion of solid particles in horizontal and vertical pipes was investigated by Kuan and Schwarz. Their study confirmed the role of drag coefficient and inlet conditions in the calculations of particle track in a vertical duct. Furthermore, in the analyzing a two-phase flow in a horizontal duct, it was shown that the gravitational force has the dominant influence on the distribution of particles [18].

Kuan investigated single and two phase flows in a 90^o bend. In a simulation, gas turbulence was solved for differential Reynolds stress models, whereas a Lagrangian approach was used to predict particle tracks. The effects of some important parameters such as one-way coupling, turbulence dispersion, pressure gradient effects, transverse lift force, and surface roughness were considered. Two major conclusion of his study were that fine particles mainly tend to follow the gas motion more firmly and gas flow near the outer wall of bend decelerates in the presence of adverse pressure gradient as well as accelerates under the influence of favorable pressure gradient [19].

Dehbi studied particle-turbulence interactions close to walls of 90^o bend, where anisotropic effects are considerable. Flow field was simulated with the FLUENT and particle dispersion in turbulent boundary layer flows was analyzed. The results confirmed that for smaller particles, inertia effects are reduced. Therefore, the model offers more accurate prediction for turbulent field in the boundary layer [20].

Hadinoto and Curits examined the effect of Reynolds number on the gas-phase turbulence in a vertical downward pipe. Moreover, they investigated the effect of particles on the turbulence modulation. 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 Reynolds number [21].

Mando and Lightstone studied the effect of particles on the turbulence equations. They utilized Eulerian/Lagrangian approach for the source terms. Their results were compared with the standard and consistent models obtained data as well as with experimental results [22].

Ono and Kimura used two types of elbows with different curvature ratios to study the interaction between flow separation and secondary flow due to the elbow curvature for high Reynolds numbers. They concluded that the flow separation always formed in the short-elbow while secondary flows intermittently occurred in the long-elbow case [23].

Lain and Sommerfeld investigated pneumatic conveying of spherical particles both in horizontal ducts and circular pipes. The Eulerian-Lagrangian approach was used for three dimensional numerical solution. They employed k–ε and Reynolds Stress models to solve flow field in a pipe with high particle volumetric percent and demonstrated that inter-particle collisions have a significant effect on velocity profile in a pipe. Furthermore, many research papers showed that the wall collisions frequency is considerably higher in the pipe in comparison with the channel [24].

The purpose of the present study is to evaluate the effects of curvature ratio, Reynolds and Stokes numbers on deposition of particles in 90^o bends to validate the computational assumption and to investigate asymmetric branches upon distribution of particles in a pipe line. To do this, the geometry of an asymmetric pipeline system is drawn. Afterwards, the structural grid is generated in the computational domain and flow field is simulated. Finally, particles are injected into the flow field.

To study the grid independency, meshes with different resolutions are generated. Since Brownian motion and diffusion phenomenon do not affect the behavior of particles and their deposition remarkably, these parameters are neglected. Since in dilute two phase flows drag and gravity forces are effective parameters [25], these parameters are considered in this study. Since volumetric concentration of particles in the flow field is below of 0.1%, particle-particle interaction is neglected and the assumption of one way coupling between particles and fluid is used [20]. This means that the flow field is not affected by particles. Therefore, after solving the flow field, the behavior of particles is investigated by using the Lagrangian method. It is worth mentioning that, in Eulerian-Lagrangian approach, the fluid phase is considered as a continuum media by solving the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles through the computational domain. This approach is the most accurate way to predict the dispersion of particles in the flow field [26].

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

The most important parameters in the problem of flow through bends with circular tubes are Reynolds number and curvature of the bend. These two parameters characterize dimensionless Dean number that is defined as:

$$De = \frac{Re}{\delta^{1/2}} \tag{1}$$

Where, Re is the Reynolds number and δ is the curvature ratio of the bend [27].

One of the major recognition parameters of gas-solid flow is dimensionless Stocks number, that is a measure for deposition of aerosol particles in a bend and is defined as:

$$Stk = \frac{C \rho_p d_p^{\,2} U}{9 \mu d_t} \tag{2}$$

Where C is the Cunningham's correction slip factor very close to unity for particles above 1 µm diameter; ρ_p, d_p and μ are the particle density, particle diameter and dynamic viscosity of fluid, respectively [20].

2. Governing Equations

2.1. Gas phase

The gas flow properties and turbulence quantities are calculated by solving a set of Reynolds averaged Navier-Stokes equations:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \bar{V} \right) = 0$$

$$\frac{\partial}{\partial t}\left(\rho \bar{\bar{V}} \right) + \nabla . \left(\rho \overline{\bar{V}\bar{V}} \right) =$$

$$-\nabla \bar{P} + \nabla . \left[\mu (\nabla \bar{\bar{V}} + \nabla \bar{\bar{V}}^T) \right] + \frac{\partial}{\partial x_j} (-\rho \overline{V_i' V_j'}) \tag{3}$$

2.2. Particle Motion in Dispersed Phase

It is assumed that only drag and gravity forces are significant. Therefore, particle velocities and trajectories in bends are calculated by solving the equation of motion for particles in Lagrange frame simultaneously [10]:

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

$$\frac{dx_{p,i}}{dt} = u_{p,i} \tag{4}$$

Where U and U_P are local gas and particle instantaneous velocities, respectively.

u_i and $u_{P,i}$ are gas and particle velocity components as well as $x_{P,i}$ is the particle position coordinate.

The drag coefficient can be obtained by:

$$C_D = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \tag{5}$$

In the above equation, a_1, a_2, and a_3 are constants which can be applied to spherical particles for a wide range of Re number. The relative Reynolds number can be defined by the following equation [20]:

$$\mathrm{Re} = \frac{\rho d_p \left| u - u_p \right|}{\mu} \tag{6}$$

In order to solve the equations of motion, i.e. Eq. (4), for each tracking particle in the flow domain, instantaneous fluid velocity components at all particle locations need to be determined. The inclusion of instantaneous fluid velocity components shows that the effects of turbulence are taken into account in calculating the particles motion. The present study adopts a classical stochastic approach for estimating the fluid fluctuating velocities.

3. Simulation of Flow Field

For calculating the three-dimensional flow field and particle tracking, the flow field is solved and subsequently a large number of particles are injected into it.

Turbulent flow is modeled by K-ε and Reynolds stress model (RSM) methods. The results of the K-ε solution are used as primary solution for RSM.

Since the Reynolds stress model includes the effects of streamline curvature and rotation in different curvatures, it can be used to illustrate the rotation of flow in a pipeline with asymmetric branches. In addition, this model can predict turbulent anisotropic stress in various points of knees [9, 11].

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

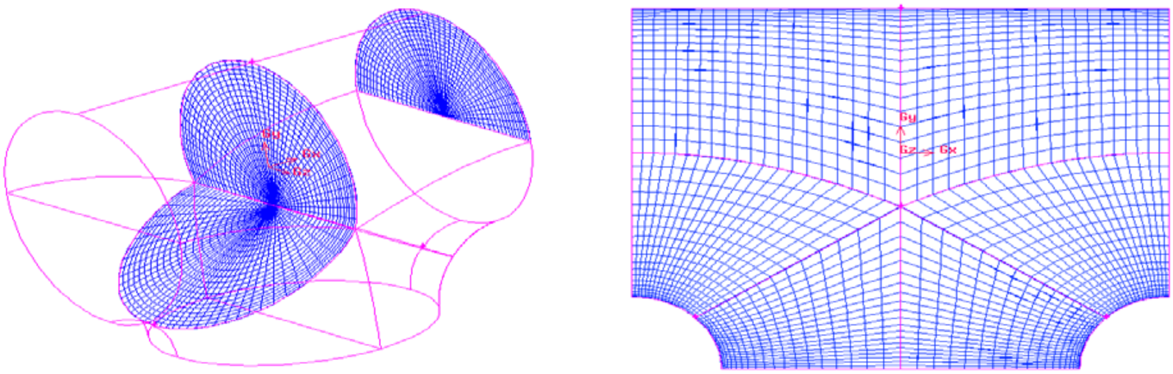
4. Particles Injection into the Flow Field

The injected particles velocity in the inlet is adjusted to the flow velocity [18]. The penetration of aerosol through bend and asymmetric branches of a pipeline is calculated by tracking a large number of particles released simultaneously. A plane is put in the inlet and particles are injected to flow by this plane. Particles are uniformly distributed over the face. It is worth mentioning that the rate of exit particles is independent of the number of particles injected by inlet plane. So, particles injection is done in any step with different particle number and this procedure is continued until the particles distribution in outlet becomes independent of the inlet particle number. The number of particles is 1500 for asymmetry pipeline. It can be shown that the rate of penetration in the pipeline does not change for a different number of injected particles. Since turbulent dispersion of particles is modeled by stochastic methods, particles injection is done several times for any inlet flow rate with a different particles size. Final results are obtained by averaging the data of each injection time.

For a 90° pipe- knee, it is assumed that the wall is stick. Therefore, some of the particles can deposit in the knee according to the flow velocity. Thus, a trap boundary condition on the walls is used. But, for asymmetry pipeline, walls of the knee are dry and all exited particles are filtered. For this reason, the reflect boundary condition is used on pipe-knee walls. Outlet particle accumulation has been studied in order to determine the number and size of required filters in any outlet region.

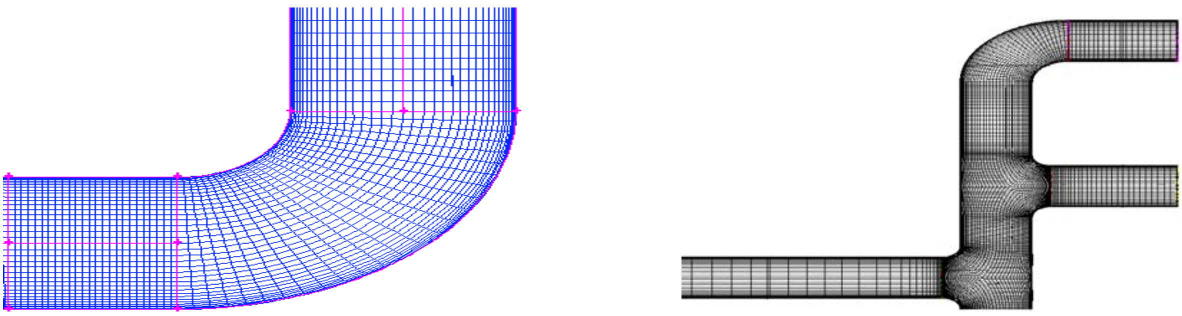
5. Grid Generation

O topology is used for grid generating in the pipe and for T junction; grids are generated as shown in Figure 1. In order to have an accurate solution and modeling of the boundary layer, the grid must be fine in the vicinity of the walls. Figure 2 shows the representation of computational grid structures at pipe-knee junction and asymmetric branches in central gas station. Figures 2 and 3 show the representation of computational grid structures at pipe-knee junction and asymmetric branches in central gas station. In this study, the flow solution and particles injection are performed for 190,651 cells. About 76,000 of total cells are triangular and have been used in the pipe centerline whereas other ones are hexahedral elements.



a) Three dimensional view T junction b) Two dimensional view T junction

Figure 1. The manner of grid generation at three ways



a) Pipe- knee junction b) Three ways and asymmetric branches

Figure 2. The manner of grid generation

5.1. Grid Study for Main Geometry

Three grids with different cells have been generated for the main geometry in Figure 3. Table 1 shows no remarkable differences between velocity and pressure values in the first three ways for 135,667 and 190,6851 cells.

5.2. Study of Grid Quality with Focusing on Aspect Ratio

Table 2 shows the distribution percent of cells in four ranges of different aspect ratios. In all the knees, the three ways and the inlets, the amount of this parameter is in the range of 1-5. In the pipe which the flow is horizontal, less than twenty-five percent of cells, have aspect ratios higher than 15. In regions where the flow is in direction of axis coordinates, cells with partly high value of aspect ratio could not cause any significant problems in numerical solution and convergence trend.

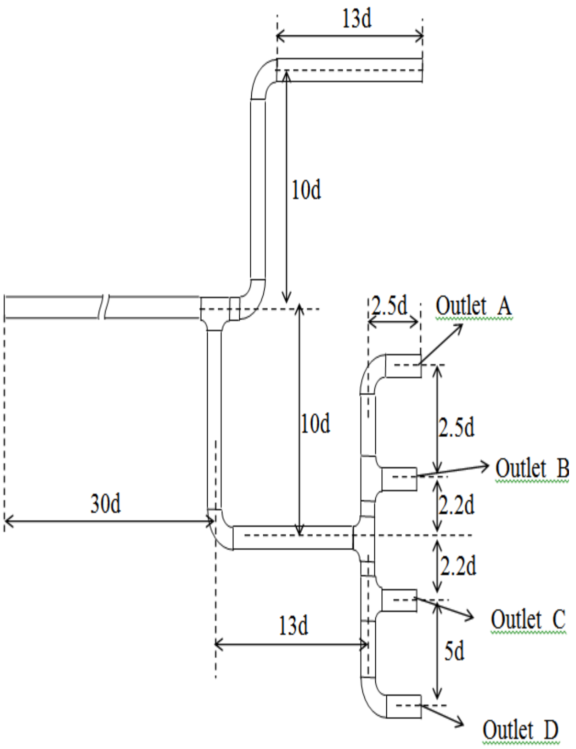


Figure 3. A Part of asymmetric branches in pipe line

Table1. Grid study for asymmetric branches shown in figure 3

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

Table2. Grid Study of grid quality from Aspect ratio standpoint for Figure 3

Aspect ratio	Percentage of cells
$1 < Q_{AR} < 5$	50%
$5 < Q_{AR} < 10$	17%
$10 < Q_{AR} < 15$	11%
$Q_{AR} > 15$	22%

6. Results and Discussion

After grid study, three pipe-knees with different curvature ratio $R/a = 2,4,10$ have been analyzed (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 the bend exit plane and its symmetry are satisfied as shown in Figure 4. These results confirm the previous experimental study done by Mac Farland in 1997 [27].

The use of the (K-ε) model for the flow solution leads to a high difference in the percentage of particles penetration in comparison with experimental results which has been shown in Figure 5. Consequently, unlike RMS, this model cannot predict the particle behavior properly.

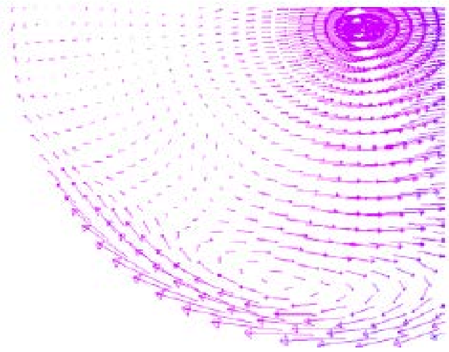
Figure 5 shows that, for three pipe-knees, particles penetration decreases with the increase of Stocks number. The Stocks number is varied by changing the flow rate through bend. With the increase of flow velocity, particles impact walls of pipe-knee with higher velocity when they want to change their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

6.1. Effect of Curvature Ratio on Particles Behavior

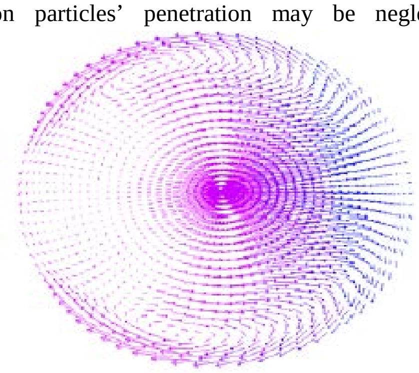
The effect of the curvature ratio on the behavior of particles in a 90^0 bend has been investigated. It is assumed that the stocks number is varied by changing the flow rate. As it is presented in Table 3, the particle penetration in bend is increased when the curvature ratio rises from 2 to 4, whereas when this increment continues from 4 to 10, the penetration of particles does not change considerably. This confirmed that increasing the bend curvature ratio can be recognized as an effective and reasonable method for augmenting transport particles. An increase in the curvature ratio from a certain bound, could lead to the waste of costs.

6.2. Effect of Reynolds Number on Particles Penetration in Bend

For considering the effect of Reynolds number on particles penetration, it is assumed that the Reynolds number varies while the Stokes number remains constant. The results are compared with experimental data and showed a reasonable agreement with what had been done by Mac Farland in 1997 [27]. Figure 6 demonstrates that by changing the Stokes number, particles' penetration does not change remarkably. Therefore, the effect of Reynolds number on particles' penetration may be neglected.



a) Velocity vectors in downstream of bend exit plane



b) Symmetry of velocity vectors in downstream of bend exit plane

Figure 4. The display of velocity vector

Table 3. Percentage of particles penetration versus Stocks number

Stocks number	**Percentage of particles penetration for** $R/a=2$	**Percentage of particles penetration for** $R/a=4$	**Percentage of particles penetration for** $R/a=10$
0.1	78%	88%	88.5%
0.15	70%	75%	77%
0.2	58%	68%	70%
0.25	49%	57%	57%
0.3	47.5%	55%	56%
0.35	38%	45%	46%
0.4	32%	39%	40%

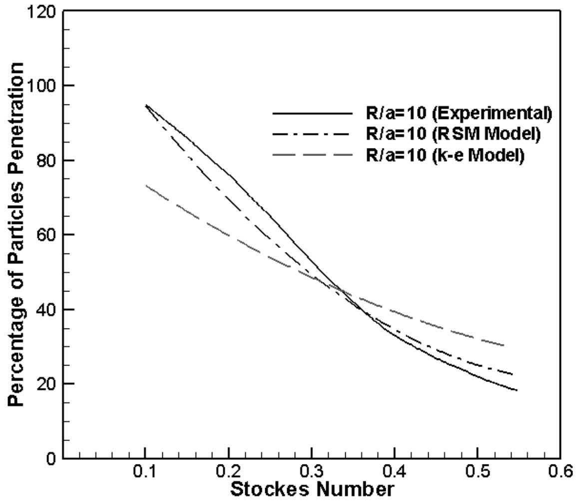


Figure 5. K-ε Model in comparison with RSM and experimental samples.

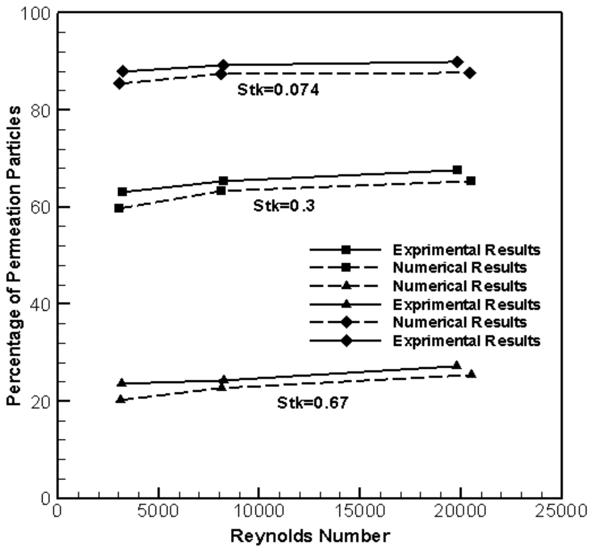


Figure 6. Percentage of particles penetration versus Reynolds number

7. Particles Distribution in Pipeline System with Asymmetric Branches

A part of asymmetric branches used in pressure reduction station is shown in Figure 3. For various inlet velocities, continuity and conservation of flow rate are conserved according to Table 4. Fluid in branches flows to the paths which cause fewer resistances against the flow. Therefore, it is expected that the most flow rate passes through the upper pipe. Velocity contour is shown in Figure 7.

The most flow rate passes through the upper part of the pipe. This is illustrated in Figure 7. Since particle does not affect the flow field and flow carries particles to the outlet, the most of accumulation particle in the outlet is in the upper part. This is shown in Table 5. Therefore, this region requires filters with higher quality and size to collect particles. Furthermore, the least number and smallest size are in *A* and D as shown in Figure 7.

For more investigation about the effect of asymmetry in branches, it is assumed that exit B in Figure 7 is blocked. In fact, flow geometry contains one inlet and three outlets A, C and D as shown in Figure 8. Results of particles injection into flow field have been shown in table 6. It could be concluded from Table 6 that the percentage of exit particles at outlet A increases to a nearly twice value, as compared with the cases in which there are four output regions at the lower part of the branch. This could be due to closing the B output region. Neither flow nor particles can exit through this outlet. Therefore, filters of higher quality and size are required at outlet A for collecting particles. Since flow tends to pass from shorter paths, the passing flow rate amount through the C region is higher than that of the D region. This fact is illustrated in Figure 7 which shows a velocity contour. C and D outputs are shown by $G-G$ and $F-F'$ cross sectional segments in Figure 7, respectively. Since flow propels particles to the outlet, it is expected that particles accumulation in C output region be more than D. Therefore, filters of higher quality and size are required for collecting particles at C output region in comparison with D, in Figure 7.

Table 4. Percentage of exit flow rate at outlet five-regions is shown in figure 3

Velocity inlet $(\frac{m}{s})$	Flow rate inlet $(\frac{kg}{s})$	Percentage of flow rate at upper pipe	Percentage of flow at A region	Percentage of flow at B region	Percentage of flow at C region	Percentage of flow at D region
4	0.63493	57.74	8.29	11.94	13.19	8.83
8	1.26987	57.48	8.9	12.1	12.91	8.6
12.3	1.95242	57.35	8.81	11.85	12.93	9.06

Table 5. Percentage of exit particles in five-outlet regions shown in figure 3

Velocity inlet $(\frac{m}{s})$	Diameter particle (μm)	Percentage of particles accumulation at upper pipe	Percentage of particles accumulation at A region	Percentage of particles accumulation at B region	Percentage of particles accumulation at C region	Percentage of particles accumulation at D region
4	10	56.8	9.5	11.7	11.1	10.9
8	10	56.1	10.1	12	10.7	10.1
12.3	10	56.1	10.8	11.7	11.6	10.8

Table 6. Percentage of exit particles with closing one of the outlets (B)

Percentage of exit flow at upper branch	Percentage of exit particles at upper branch	Percentage of exit flow at A region	Percentage of exit particles at A region	Percentage of exit flow at C region	Percentage of exit particles at C region	Percentage of exit flow at D region	Percentage of exit particles at D region
57.52	56.8	16.9	17.2	14.56	13.8	11.01	11.7

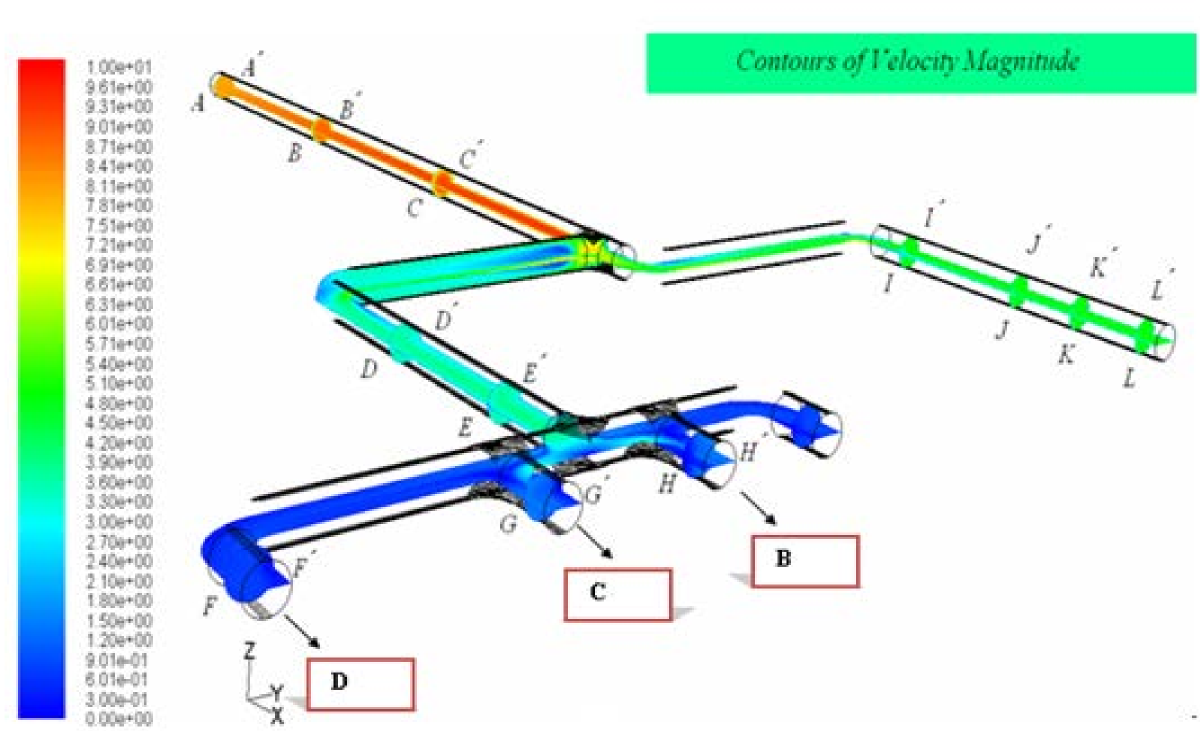


Figure 7. Velocity contour in pipe line with asymmetric branches

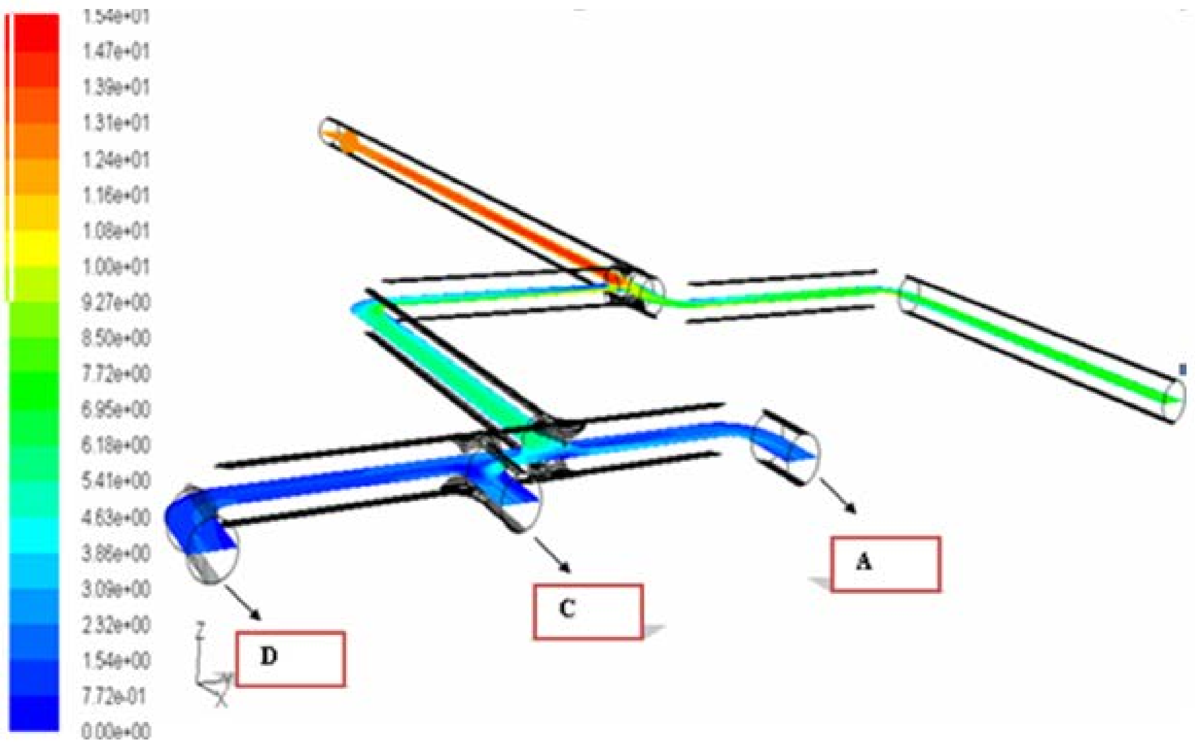


Figure 8. Velocity contour with closing one of the outlets (B)

Table 7. Percentage of exit particles with different boundary conditions at outlet

Percentage of exit particles at upper branch	**Percentage of exit particles at A region**	**Percentage of exit particles at B region**	**Percentage of exit particles at C region**	**Percentage of exit particles at D region**
45.2	7.9	19.5	19.8	7.2

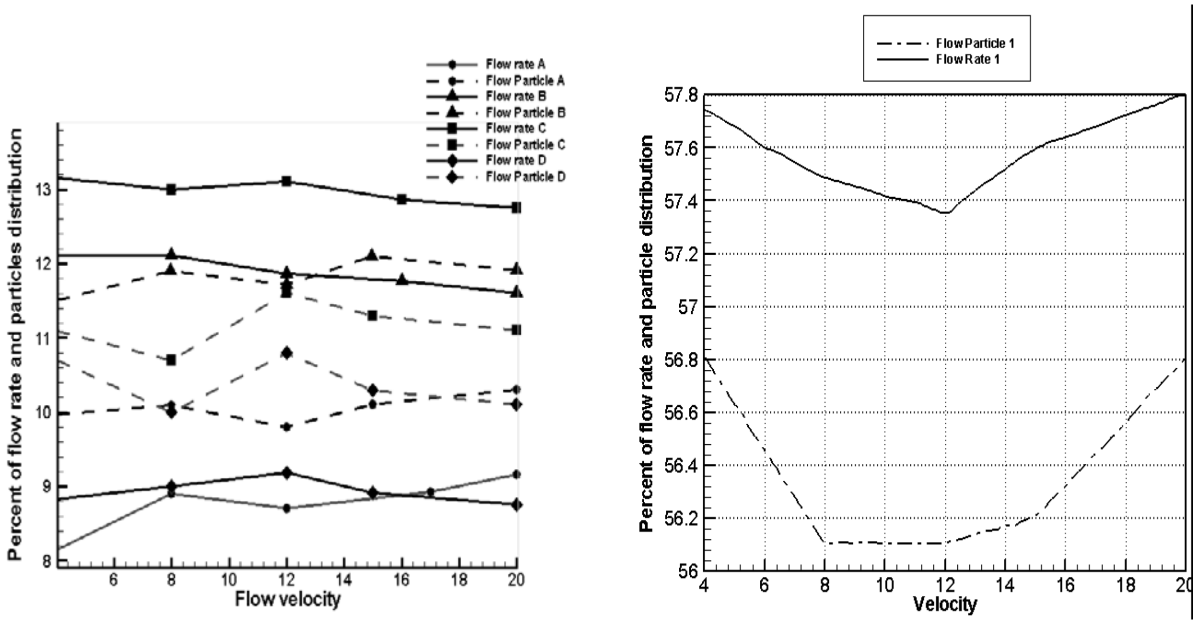
8. Effect of Pressure Variation on the Particles Distribution in Outlet

Different boundary conditions are considered at the outlets. In B and C, the value of pressure is set to *5.04 MPa* and in A and D *5.6 MPa* working pressure is exerted as boundary condition. Since the working pressure is supposed to be equal to atmospheric pressure, methane density should be calculated in *5.6 MPa* working pressure. Percentage of exit particles in various outlet regions for 12.3 m/s velocity inlet and 10-micrometer particle has been shown in Table 7.

Table 7 shows particles accumulation at the upper outlet as remarkable. In addition, due to the pressure reduction in B and C relative to A and D, the flow rate increases at B and C. Therefore, flow propels more particles to these regions. As a result, at B and C, under mentioned circumstances, filters of higher quality and size are needed.

9. Dependence of Flow Rate and Particles Distribution on the Fluid Velocity

The dependence of volume portion of particles and flow rate on fluid velocity is indicated in Figure 9. This also proved that this dependence increases at high flow velocities. This is mainly due to the drag force increment at higher velocities, which make it significantly larger than gravity force, whereas particle behavior is more unaffected by the flow in the velocities lower than 7 m/s. In figure 9, dependence of volume portion of particles at A, B, C and D regions has been shown with flow rate as well as dependency has been shown for upper part of pipe. In both of them, dependency is higher at high flow velocities because of increase drag force at higher flow velocities.



a) At A, B, C and D outlet regions b) At upper exit region

Figure 9. Graphical relation particles distribution and flow rate

10. Effect of Particles Size upon Particles Distribution in Outlet

The form of particles distribution in the outlets is directly related to flow distribution at these regions. Therefore, for all injected particles with any diameter and size, it is expected that the amount of particles' accumulation at the upper exit region is more than that in other regions, while at A and D regions it is less than in other regions. Besides, flow tends to direct larger particles through simpler passage to exit regions. By increasing particles diameter, amount of particles' accumulation changes. Admittedly, the amount of particles accumulation reduces in the lower parts of branch and obviously increases in the upper exit region with the same amount. Since all the particles are in the size of micro and there is no remarkable difference in their size, particles distribution especially in the lower parts of branch is not considerable. Figure 10 shows the effect of particle size on the distribution of particle accumulation. More particles exit from the upper part and at the lower, a little difference is present between four branches.

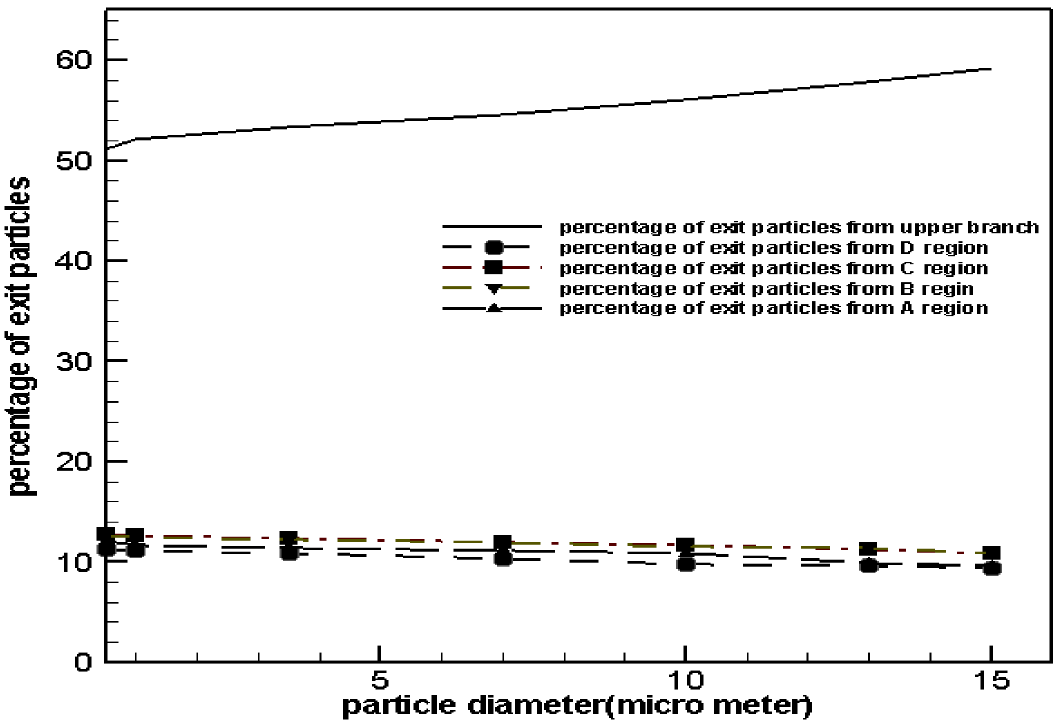


Figure 10. Particle distribution from exit regions versus particle diameter

11. Conclusions

Deposition of particles increases in the pipe-knee as the inlet flow velocity rises as opposed to the particles' behavior in the pipe without knee and branch. The increase in bend curvature ratio to a certain limit is relatively effective and plausible method for augmentation of transport particles. If curvature ratio exceeds from a certain ceiling, it leads to a waste of costs. It is worth noting that exit particles percentage is not found as a function of Reynolds number and it depends on the Stokes number only. Asymmetry in branches of piping systems has a very powerful effect on the distribution of particles. So, the required filters for trapping particles are dependent on the position of branches in these piping systems. In addition, at high flow velocities, particles behavior depends on flow pattern. This may be attributable to the significant increase in the drag force in comparison with gravity force.

References

[1] Cheng, Y.S., Wang, C.S.," Inertial deposition of particles in a bend". Journal of Aerosol Science, Volume 6, Issue 2, 1975, Pages139-145.

[2] Crane, RI., Evans, RL., "Inertial deposition of particles in a bent pipe". Journal of Aerosol Science, Volume 8, Issue 3, 1977, Pages161-170.

[3] Cheng Y.S., Wang, C.S., "Motion of particles in bends of circular pipes". Atmospheric Environment, Volume 15, Issue 3, 1981, Pages 301-306.

[4] Pui DYH, Romay-Novas, F., Liu BYH., "Experimental study of particle deposition in bends of circular cross section". Aerosol Sci Technol; Volume7, Issue 3, 1987, Pages 301–315.

[5] Tsai C.J., Pui DYH, "Numerical study of particle deposition in bends of a circular cross-section-laminar flow regime". Aerosol Sci Technol; Volume12, 1990, Pages 813–831.

[6] Ottjes, J. A., "Digital simulation of pneumatic particle transport". Chemical Engineering Journal, Volume 33, Issue 6, 1978, Pages 783-786.

[7] Tsuji, Y., Morikaw, Y., "Numerical simulation of gas-solid two-phase flow in a two-dimensional horizontal channel". International Journal of Multiphase Flow, Volume 13, Issue 5, 1987, Pages 671-684.

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

[9] Tashiro, H., Peng, X., Tomita,Y., "Numerical prediction of saltation velocity for gas-solid two-phase flow in a horizontal pipe". Powder Technology, Volume 91, Issue 2, 1997, Pages141-146.

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

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

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

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

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

[15] Hajji,L., Pascal,Ph., Oesterel,B., " A simple description of some inertia effects in the behaviour of heavy particles in a turbulent gas flow".International Journal of Non-Linear Mechanics, Volume31, Issue 3, 1996, Pages387-403.

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

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

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

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

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

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

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

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

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

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

[26] Taylor, G. "diffusion by continuous movements" proc. of the London Mathematical Society, Volume 20, 1921, Pages 193-212.

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