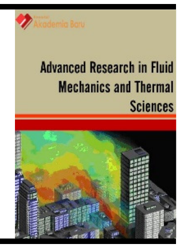
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Flow Pattern and Particle Contamination in Fluid Flow through Mitre Bend

Open Access

Ainaa Maya Munira Ismail[1,*], Fazila Mohd Nawawi[2], Jamaludin Md. Sheriff[2]

[1] Department of Mechanical Engineering, Faculty of Mechanical Engineering, UiTM Cawangan Johor, Kampus Pasir Gudang, Malaysia
[2] Department of Mechanical Engineering, Faculty of Mechanical Engineering, UTM Skudai, Johor Malaysia

ARTICLE INFO

Article history:
Received 29 October 2016
Received in revised form 1 December 2017
Accepted 9 December 2017
Available online 10 December 2017

ABSTRACT

Flow pattern exists on miniscule to astrophysical scales, it is difficult to understand the fluid flow behavior and predict how they will react especially when involving particle contamination in those fluid flows. Recently, many research have investigated the relationship between flow pattern and behavior with a particle deposition through numerous types of bend but not too concerned about the flow pattern with different types of particle distributions. Therefore, this paper presents the investigation on the flow pattern of particle contamination in fluid flow through square mitre bend with different types of particles. The aim of this article is to establish the flow behaviour through square mitre bend with the effect of the particles and to study the relationship between flow pattern with related parameters such as flow velocity for different types of particles. This research was performed on flow through 90° square mitre bend in horizontal to vertical orientation and water was used as flow medium. Flow pattern is caused by three types of particles, namely Sand, Nylon and Polyethylene. Besides, particle size was in the range of 50µm to 200µm and was categorized as a Particle 1 (Sand A), Particle 2 (Sand B), Particle 3 (Nylon), Particle 4 (Polyethylene Solid) and Particle 5 (Polyethylene Powder). All experimental results were compared with those form simulation technique. Particle Image Velocimetry (PIV) was used to determine velocity and pattern field of the flow. Then, High Speed Camera (HSC) was also used to observe the particle concentration through the pipe bend. Meanwhile, simulation results were illustrated by using commercial Computational Fluid Dynamics (CFD) Software, i.e ANSYS FLUENT. The results indicate that the flow pattern was influenced by particles and bending conditions, while different particle properties affects particle concentration and distributions. Henceforth, fine particles tend to be fully suspended owing to their small volume and lightweight condition, but coarse particles are easily separated from the liquid.

Keywords:
Flow pattern, flow behavior, particle contamination

1. Introduction

In general, particles tend to settle down to the bottom of pipes due to the action of gravity force forming different flow patterns which can be indicated by particle concentration profile. As predicted, fine particles tend to be fully suspended due to their small volume and light weight condition, but coarse

* *Corresponding author.*
E-mail address: ainaaismail@ymail.com (Ainaa Maya Munira Binti Ismail)

particles are easily separated from the liquid. Previous studies were carried out to investigate the flow behavior and pattern affected from particle through bend in laminar or turbulent flow. The studies included the understanding of the effect of a bend on particle cross section concentration and segregation and the impact of particle flow to bend and flow phenomenon. Those studies have been conducted in different ways and some were solved in limited conditions and solutions. Most of the investigation of flow behavior involved only one particular type of particle. Pulverized coal is used as particle with volume mean diameter of 53.3 mm to study solid flow behavior in bend of pneumatic conveying system [5]. Particles of Polydisperse glass spheres with the range of size between 5-150mm were used to measure the deposition in industrial bends [10]. For the purpose of analyzing the slurry flow regimes downstream of a pipe bend, particle used were 100μm diameter glass beads [8]. With reference to [9], particle used were assumed to be solid and spherical. Five size groups (1, 3, 5, 9, and 16 mm) of particle distribution and deposition were taken into consideration for this investigation. Significant loss of particles occurs in a bend as a result of inertial deposition. The particle cross sectional concentrations is not in uniform for a few meters after a bend and the particle tend to concentrate around pipe wall [3]. The motion of particles in a bend using the developed laminar flow field from empirical formula stated that the flow of Reynolds number has significant effects on particle deposition in bends [1].

Research of the turbulent flow in a pipe with sudden expansion of geometry identified that flow over sudden expansion of geometry generates vortices due to the separation flow which resulted from the sudden pressure change in the fluid. The research of flow separation from the surface of a solid body, and the study of global changes in the flow field which was developed as a result of the separation are among the most difficult problems and fundamental of fluid dynamics [11]. Then, to obtain accurate data, it is important to correct for the losses of particles in bends as well as other parts of a sampling system [2]. In order to prevent flow blockage phenomenon and to reduce the impact of particles on the wall of the bend, the new swirling flow technique were proposed in [4]. The new swirling flow technique is called as Swirling flow pneumatic conveying (SFPC). The results indicate that the overall pressure drop of the swirling flow pneumatic conveying shows a lower tendency than that of axial flow pneumatic conveying, when it was in the lower velocity range. On other hand, the minimum velocities can be decreased by the swirling flow pneumatic conveying. From the particles flow pattern visualization, it shows that the impact of particles on the wall of the bend can be reduced using swirling flow.

Moreover, based on the reviews on other literatures, the solid flow behavior in bends was successfully investigated. A solid flow-metering device based on pressure drop measurements in dilute phase conveying systems was developed beforehand. This task explores a similar condition flow blockage that has been observed with the usage of coal. In conveying pulverized coal pneumatically to a blast furnace using compressed nitrogen, from the observation done shows that the conveying system would plug with passage of time due to coal deposition [5]. Next, by comparing the particle size distributions upstream and downstream of bends that had geometries and flow conditions similar to those used in industrial ventilation, it can be concluded that a penetration was not a multiplicative function of bend angle as theory predicts, but it is due to the developing nature of turbulent flow in bends [6]. By referring to the research about the flows, secondary flow has been investigated because of its effect to the stability of downstream bend. It is important to study on flow behavior especially secondary flow behavior. The research about secondary flow at various angles in a square and circular cross section pipe bends was implemented and it shows that secondary flow occurred in a sharp bend of square or circular cross section. Secondary flow pattern appears more clearly when the velocity was increased [7]. Then, characterization of the pipe bend influence to downward and horizontal flows on transition of velocities between slurry flow regimes in a horizontal pipe was presented [8]. It can be noted that an important challenge brought by slurry flows concerns on the availability of reliable non-intrusive measurement methods, like tomography, to obtain concentration and velocity profiles required to design and scale-up slurry flow processes. Since this particle affected the flow pattern, it is necessary to investigate the means of particle deposition concentration. So, in this study, three different types of particles were used, namely Sand, Nylon and Polyethylene. The particle size was in range the range between 50μm to 200μm. The Reynolds Number used varied from 5.00×10^2 to 1.00×10^3 (Laminar flow).

2. Methodology

To execute this investigation, an experiment was set up by using Particle Image Velocimetry (PIV) and High-Speed Camera (HSC). Then, the results obtained were validated by using CFD software, FLUENT in particular.

2.1 Experimental Setup for PIV

PIV was mainly used in fluid mechanics for its ability to obtain relevant instantaneous velocity fields. For this experiment, a simple square pipe bend model made from 3mm thickness perspex was assembled. Perspex was used as a flow channel material because of its transparent feature which allows light to pass through. Geometry or Modeling of the experimental test section is one of the important factors to fulfill the investigation. Regarding that, the construction of the model was strictly referred from the general standard for modeling a bend and some related information from previous study. For this investigation, by referring to the British Standard [12], the acceptable range of diameter or the bend design, d (cm) is 8 <d<10. Thus, the bend of the pipe constructed is 90° mitre bend with cross section of 8 cm. The mechanic properties involved in this investigation is the Reynolds Number (*Re*) to determine the condition of flow whether it is laminar, transition or turbulent flow. *Re* is denoted by

$$Re = \frac{\rho v d}{\mu} \quad (1)$$

where;
ρ= Density of fluid= 998 kg/m^3
v= Velocity of the flow (m/s)
d = Cross Section of Diameter Pipe (m)
μ= Dynamic viscosity of fluid =1.002x10^{-3} kg/ms

Before starting the experiment, the water was to reach its fully developed stage. A fully developed flow exists when the velocity profile is no longer changing with increasing distances downstream. So, it was important to obtain the entrance length, *Le.* But, to obtain the *Le*, hydraulic diameter (*D*) and flow velocity (*V*) must be measured first. The measured flow velocities are shown in Table 1.

Table 1
Flow Velocities

Points	1	2	3	4
Velocity, V (m/s)	0.052	0.0411	0.03994	0.0385

Hydraulic diameter (*D*) is denoted as

$$D = \frac{4(ab)}{2\,(a+b)} \quad (2)$$

Since this study considered square cross-section, the width *a*, and the height, *b*, of the duct are the same. From the equation above, the hydraulic diameter obtained is 0.08 m. Moreover, known that for laminar flow *Re*<1000, transition flow *2000<Re<4000* and turbulent flow *Re>4000* [13], then, *Le* can be defined by

$$\frac{Le}{d} = 0.06Re_d \; for \; Laminar \; flow \tag{3}$$

$$\frac{Le}{d} = 4.4Re_d \; for \; Turbulent \; flow \tag{4}$$

In this investigation, four flow velocities were used to observe the flow pattern of particle contamination in fluid flow through 90° mitre bend. The test section with appropriate *Le* and 90° bend is illustrated in Figure 1. Water was used as a medium of a flow and the particles categorization can be reviewed in Table 2, while the illustration of experiment shown in Figure 2.

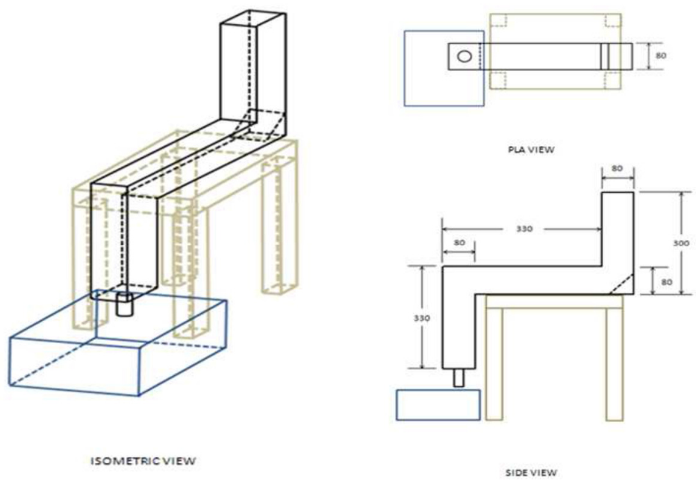


Fig. 1. The experimental test section

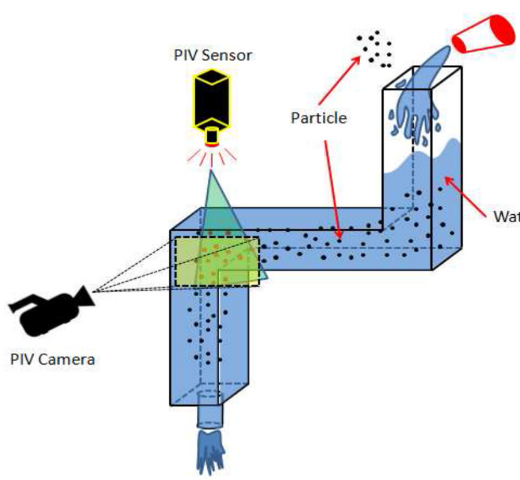


Fig. 2. The illustration of experimental set up

Table 2
Particles Categorization

Particles Specifications	Particle 1	Particle 2	Particle 3	Particle 4	Particle 5
Type	Sand	Sand	Nylon	Polyethylene	Polyethylene
Size (μm)	5 x 10^{-5}	1 x 10^{-4}	1 x 10^{-4}	1 x 10^{-4}	1.75 x 10^{-4}
Particle Volume, v (m^3)	6.54 x 10^{-14}	5.24 x 10^{-13}	6.28 x 10^{-6}	1.27 x 10^{-5}	2.81 x 10^{-12}
Flow rate, Q (kg/s)	1.31 x 10^{-11}	1.05 x 10^{-10}	1.26 x 10^{-3}	2.51 x 10^{-3}	5.61 x 10^{-10}

2.2 Experimental Setup for HSC

HSC was used to obtain the particle concentration through a pipe bend. The particle flow image was illustrated in the HSC figure. Hence, it is very suitable to capture the particle images through a pipe bend because of its capability to record thousands of high resolution images per second. It gives better understanding about the particle movement and concentration in a pipe bend and indirectly obtains the accurate answer for a real particle flow behavior. The setting of HSC was similar as PIV. The camera captures a flow movement and particle distribution when it passes through bend test section.

2.3 CFD Simulation

Numerical analysis in this paper was aimed to investigate the velocity and flow pattern of the particle contamination in fluid flow through mitre bend. The CFD software, FLUENT with simple algorithm was applied to simulate fluid flow problems. Finite-volume method was utilized to solve the governing equations for fluid. Moreover, it provides the capability to use different physical models such as incompressible or compressible, inviscid or viscous, laminar or turbulent, and others. In FLUENT, parameter data defined are:
i. Second order implicit, steady solver
ii. Material: Water
iii. 2-Dimensional

Then, K-Epsilon model is chosen as viscous model because of its simple complete model, more related when applying Laminar flow with 2 equations needed, faster convergence and high accuracy.

3. Results and Discussion

3.1 PIV and FLUENT Results

The main objective of this investigation is to establish the flow behavior through the square mitre bend with the effect of the particle and to study the relationship between flow pattern and related parameters. Thus, the relationship between flow pattern for different velocities and relationship between flow pattern for different particles can be determined. The study was conducted by using three types of particle, namely Particle 1 (Sand A), Particle 2 (Sand B), Particle 3 (Nylon), Particle 4 (Polyethylene Solid) and Particle 5 (Polyethylene Powder). Each particle has their own flow rate, Q that has been calculated and there were $Q_{Sand\ A} = 1.3089 \times 10^{-11} kg/s, Q_{Sand\ B} = 1.30472 \times 10^{-10} kg/s, Q_{Nylon} = 1.2566 \times 10^{-3} kg/s, Q_{PE\ Solid} = 2.5133 \times 10^{-3} kg/s, Q_{PE\ Powder} = 5.6123 \times 10^{-10} kg/s$. Regarding that, the result data were defined in 4 flow velocities that have been measured for each particle. The particle flow velocity measured were V_1=0.052 m/s, V_2=0.0411 m/s,

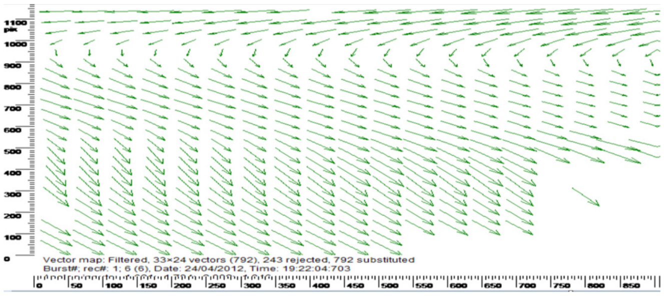
V_3=0.03996m/s and V_4=0.0385 m/s. In order to obtain the flow pattern and velocity flow of particle contamination in fluid flow through mitre bend in the experiment, PIV method was used as an experimental solution while FLUENT software as a CFD simulation solution.

3.2 Formation of Vortex (Flow Pattern)

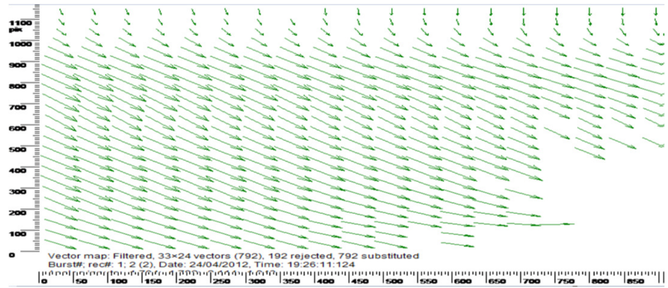
The main vortex flow features can be described in terms of general dynamical, and in some cases, in terms of topological concepts about the local and overall properties of flows, such as the strain/vorticity ratio, the helicity, the genus of vortical surfaces, and an important new parameter related [15]. Figure 2 and Figure 3 illustrate the vortex formation of 5 particles in bend configuration at V_1=0.052 m/s by using PIV and FLUENT respectively. Based on Figure 3.1 (a) and (b), it is shown that the vortices have been developed and evenly scattered all over the pipe bend surrounding. The flow patterns are clearly seen, and it was in a smooth behavior. Both configurations were in the same streamlines. As for Figure 2 (c) and (d), the vortices developed in unstable condition. The streamlines are not uniform, and as we can see on Figure 2 (c), the flow velocity was high before the particle passed through the bending part while in Figure 2 (d), there was almost no particle velocity before it crossed the bending condition. This is because, the particle movement was not aligned with the fluid flow. The flow pattern formed was not in smooth condition. Figure 2 (d), shows that the streamlines were fully developed. There is a center of vortex which occurred near the lower wall of bending part. Figure 3 illustrates the vortex developed at pipe bend by using FLUENT at 5 different particles in the same velocity, V_1=0.052 m/s. The streamlines were fully developed for 5 particles. In Figure 3 (a) and (b), the flow patterns have high concentration at the lower wall of the bend part. This is because the particle used (Sand A and Sand B) was heavy and the streamlines were evenly scattered. In Figure 3 (c), (d) and (e), the particle flow was already having a high concentration before it passed through the bending part. The streamlines were also scattered all over the pipe.

3.3 Strength and Size of Vortex

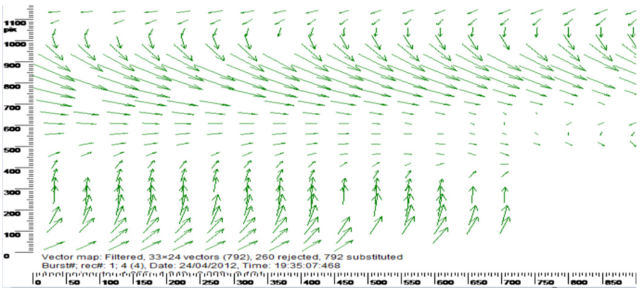
The strength of the vortex is mostly determined by the starting conditions. Further development of the vortex is determined by the conservation of its angular momentum. In a rotating system, the analogy is that the vortex flows are principally generated by the meridional flow field while the centrifugal and Carioles forces only act to change the vortex vector direction [7]. Figure 3 (a) and (b), indicates the vortices developed by Sand A and Sand B. It shows that the streamlines were fully developed. The strength and size of vortex are uniform and constant. Based on the streamlines in Figure 3 (c) and (d), it was indicated that the vortices were developed by Nylon and PE Solid respectively. With reference to Figure 3 (c), the vortices have higher strength and size at the upper wall of pipe. The vortices circumstance occurred before it passed through the bending part. The illustration shows that the streamlines were poorly developed because of inconstant formed size of vortex. In Figure 3 (d), it is shown that the strength and size of vortices increase when it crosses the bending part. As indicated in Figure 4 (e), it shows the result of PE Powder by using PIV. The result indicates that the center of vortex occurred at lower wall of bending part. The streamlines were clearly developed, and those strength and size of vortices were almost uniform. Meanwhile, Figure 4 illustrates the vortex developed at pipe bend by using FLUENT at 5 different particles in the same velocity, V=0.052 m/s. The streamlines were clearly developed for 5 particles. In Figure 3 (a) and (b), the strength and size of vortices were high at the lower wall of the bend part. The streamlines were evenly scattered. As for Figure 4 (c), (d) and (e), the particle flow was already having a high concentration before it passed through the bending part. The streamlines were also scattered all over the pipe. Those 3 results show that the strength and size of vortices were almost even.



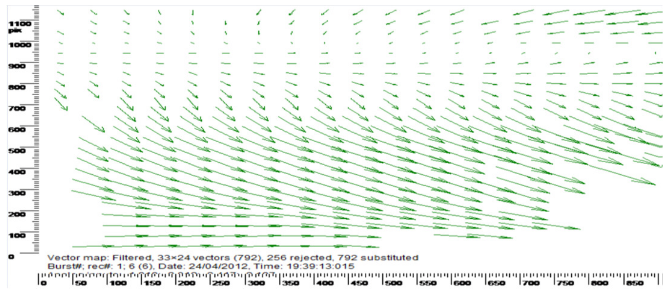
(a) Sand A



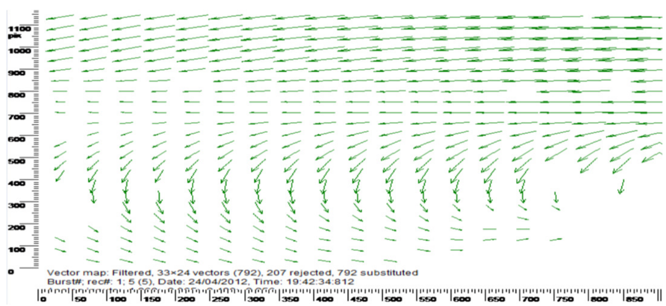
(b) Sand B



(c)Nylon



(d) PE Solid

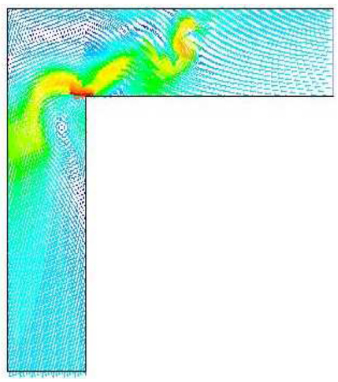


(e) PE Powder

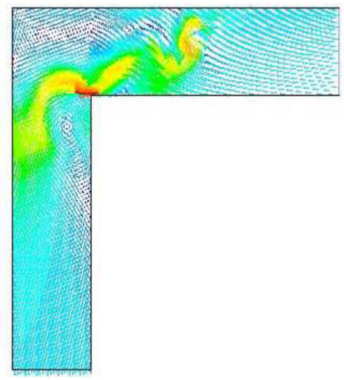
Fig. 3. Streamlines of pipe bend in 5 particles at V_1= 0.052 m/s by using PIV

3.4 Velocity at Bending Condition

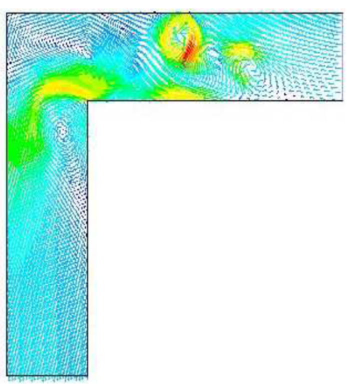
Figure 5 illustrates the vortices configuration for Sand A flow at 4 velocities by using PIV. Figure 5 (a) shows the result when flow velocity is 0.052m/s. The vortices were evenly scattered and uniform. It was fully developed, and the particle flow was constant at any point in pipe. As indicated in Figure 5 (b), the flow velocity applied is 0.0411 m/s. It shows that the flow started to construct inconstantly and disordered. The particle velocity was high at some point in a pipe before it passed through the bending part. Meanwhile, Figure 5 (c) presents the result when velocity is decreased to 0.03995 m/s. The velocity was not evenly scattered and there was no particle vortices along the right left wall of pipe. The streamlines were not uniform and constant. Furthermore, Figure 5 (d) illustrates the vortices configurations when velocity is 0.0385 m/s. As we can see, there is no particle velocity appears in the left wall of pipe. Since the velocity was very slow, so the effect of force was not clearly seen. Regarding that, in this case most particles were influenced by centrifugal force and momentum of flow velocity. Hence, the particle velocity was almost similar, and it moves in consonance with gravity. Whenever the direction of the flow is changed at a bend or elbow, the velocity distribution across the pipe is distributed. A centrifugal effect causes the maximum velocity to occur towards the outside of the bend or elbow whilst at the inside of the bend or elbow the flow is slowed or even reversed in direction if the flow is separated from the wall and a vena-contracted formed [14].



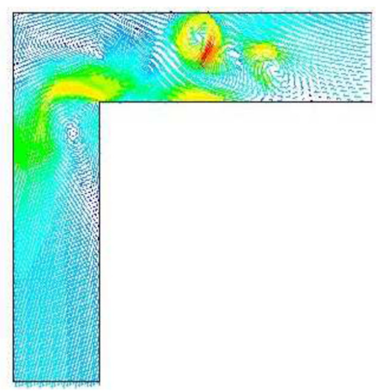
(a)Sand A



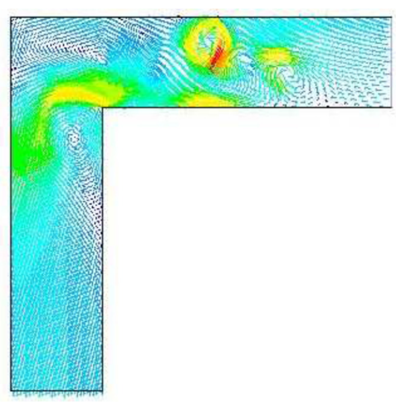
(b)Sand B



(c)Nylon

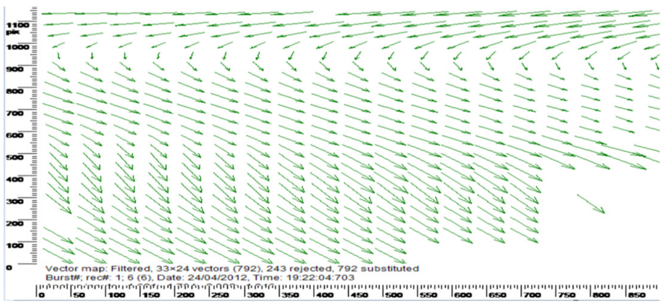


(d)PE Solid

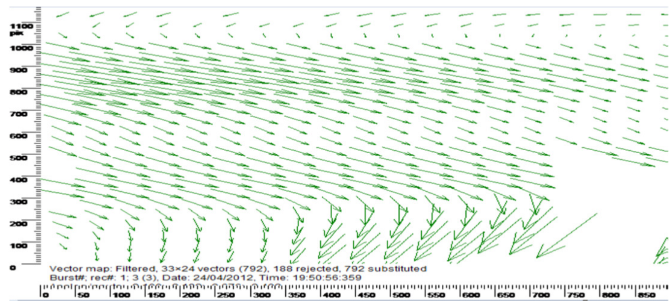


(e)PE Powder

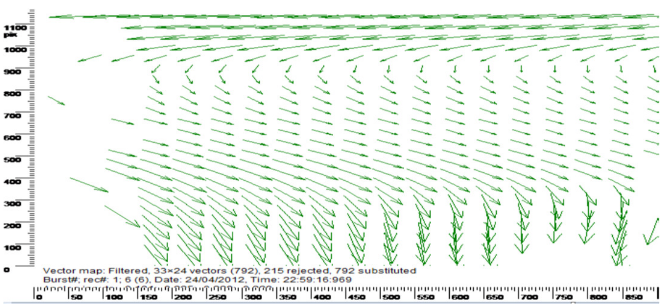
Fig. 4. Streamlines of pipe bend in 5 particles at V_1= 0.052 m/s by using FLUENT



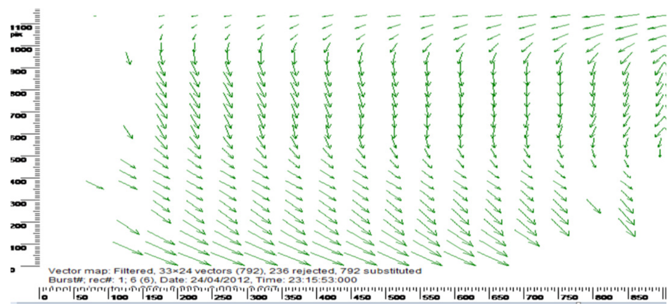
(a) V=0.052 m/s



(b) V=0.0411 m/s



(c) V=0.03996 m/s



(d) V=0.0385 m/s

Fig. 5. Streamlines at a pipe bend for a Sand A at 4 velocities by using PIV

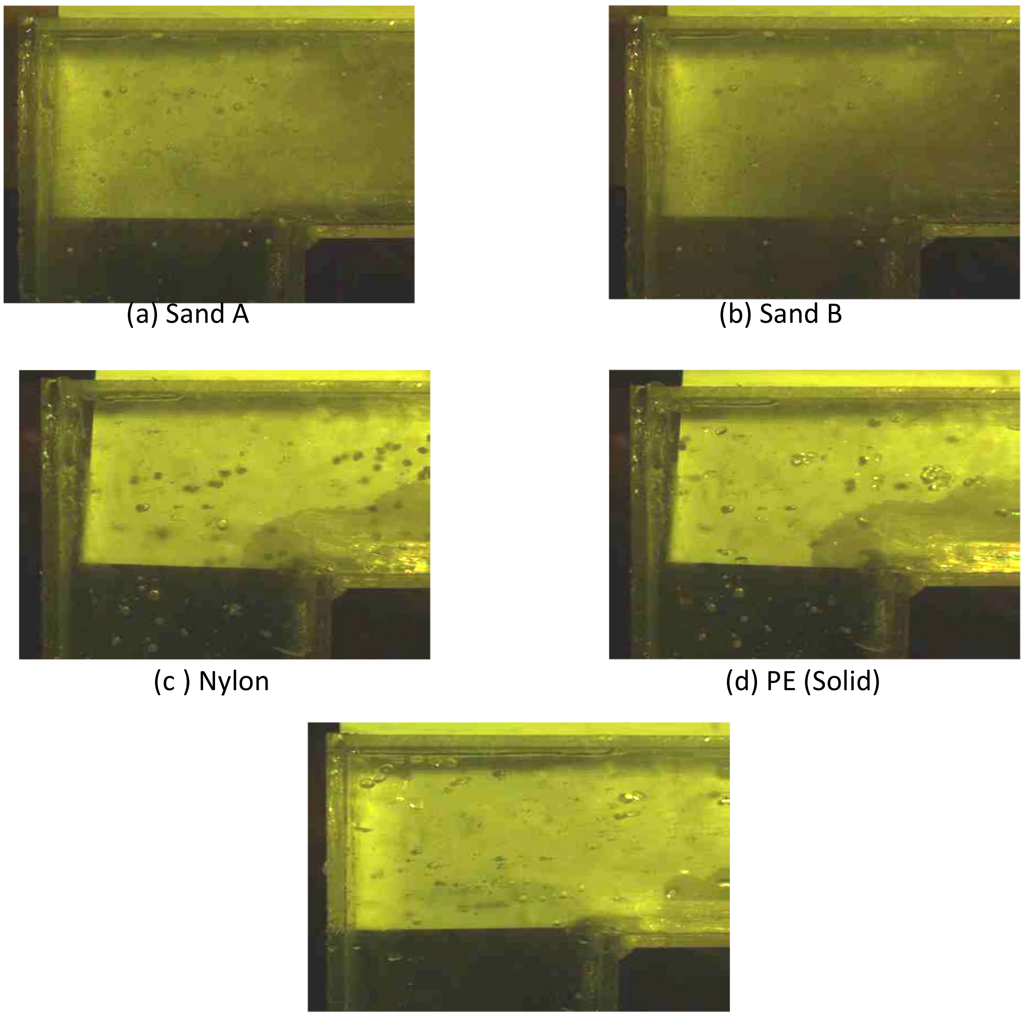
3.5 Particle Concentration

Figure 6 shows the 5 different particles concentration in a pipe bend at V=0.0411m/s. The particles were visualized as bubble-like form. Based on Figure 6 (a) and (b), it is clearly seen that the particles (Sand A and Sand B) were distributed at all the pipe surfaces but have higher concentration at lower wall of pipe. The particles schematically move along the pipe bend. Moreover, Figure 6 (c) shows that the particles flow randomly at the pipe surrounding. This is because Nylon is lightweight and has a small flow rate. The same goes to particle concentration of PE solid and PE Powder. The results of particle concentrations can be revised on Figure 6 (d) and (e). The results obtained were in good argument when compared to literature study. As represented in [3], pipe diameter, bend radius and different particle properties influence particle concentrations. In the other hand, it was mentioned that the bend significantly contributes to the suspension of particles [8].

3.6 Particle Distribution

Figure 6, 7, 8 and 9 indicate the graph of velocities against the particle distribution for 5 particles at V_1 = 0.052 m/s, V_2 = 0.0411 m/s, V_3 = 0.0399 m/s and V_4 = 0.0385 m/s. As stated, the particles were classified as Particle 1(Sand A), Particle 2 (Sand B), Particle 3(Nylon), Particle 4(PE Solid) and Particle 5(PE Powder). Figure 3.5 shows the particle distribution at V_1=0.052 m/s. It shows that Sand A and Sand B were distributed at position 2 to 10. The particles were distributed in a smooth condition and followed the flow of velocity. Meanwhile, Nylon and PE Solid distribution was random and not uniform respectively. The higher position of both was only at point 8. For PE Powder, it moved randomly at initial point and become linear when it reaches at point 4. The result was similar for particle distribution at

V_2=0.0411 m/s as shown in Figure 7. Figure 8 illustrates the particle distribution at V_3= 0.03996 m/s. Sand A and Sand B were distributed smoothly and constantly as compared to Nylon and PE Solid. While for PE Powder, it moved randomly at initial point and become linear when it reaches at point 4. This was similar as shown in Figure 9, where V_4 = 0.0385 m/s. The same phenomenon occurred because of the segregation of particles. The real particulate phase (powder, granular material) is composed of particles with a range of sizes, so this lead to segregation of particles in the pipe section following a bend [2].



(a) Sand A

(b) Sand B

(c) Nylon

(d) PE (Solid)

(e) PE Powder

Fig. 6. Particle Concentration and Distribution at V_2= 0.0411 m/s by using HSC

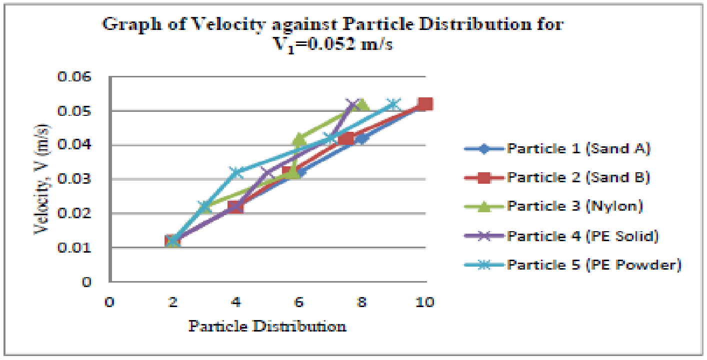


Fig. 7. Graph of Velocity against Particle Distribution for V_1=0.052 m/s

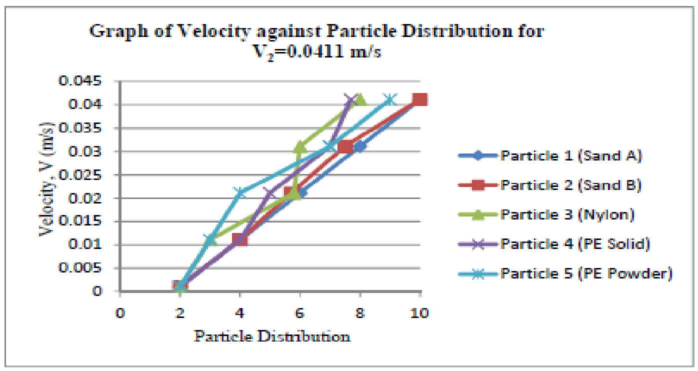


Fig. 8. Graph of Velocity against Particle Distribution for V_2=0.0411 m/s

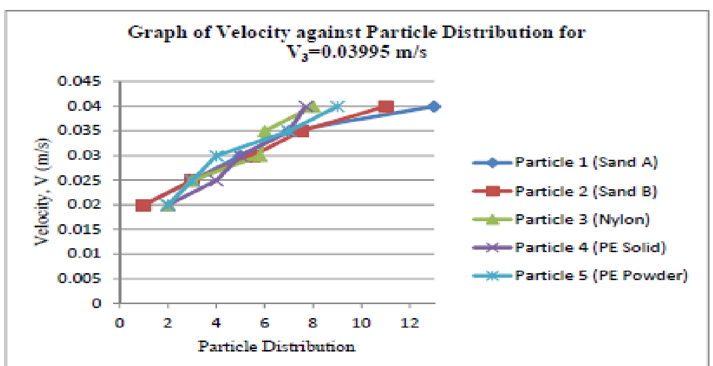


Fig. 9. Graph of Velocity against Particle Distribution for V3=0.03996 m/s

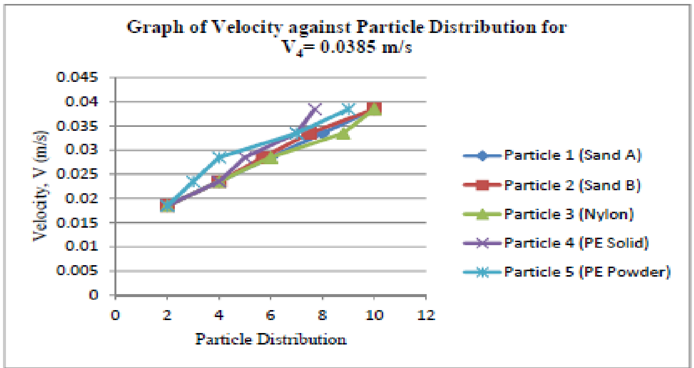


Fig. 10. Graph of Velocity against Particle Distribution for V_4=0.0385 m/s

4. Conclusions

The flow of Reynolds number played important role in flow pattern. It was proven that laminar and turbulence gave different results of flow pattern. For this study, the flow patterns appear more evidently when the velocity was increased from lower to higher value. Therefore, it can be concluded that the flow pattern was influenced by particles and bending conditions. Meanwhile, for particles concentration, most of the particle flow was governed by 3 main factors; Centrifugal Force, Momentum and Gravity. These 3 main factors gave big impacts to the flow pattern. It is known that laminar was very slow so that water flow velocity is higher than particle flow. Moreover, different particle properties influence the particle concentrations and distributions. Thus, fine particles tend to be fully suspended owing to their small volume and lightweight condition, but the coarse particles are easy to be separated from the liquid.

Acknowledgement
The authors would like to thank the Ministry of Higher Education (MOHE), Universiti Teknologi MARA (UiTM) and Universiti Teknologi Malaysia (UTM) for the support given to this research.

References
[1] Mori, Yasuo. "Study on forced convective heat transfer in curved pipes-1. Laminar region." *Int. J. Heat Mass Transfer* 8 (1965): 67-82.

[2] Cheng, YUNG-SUNG, and C. S. Wang. "Motion of particles in bends of circular pipes." *Atmospheric Environment (1967)* 15, no. 3 (1981): 301-306.

[3] Levy, Avi, and David J. Mason. "The effect of a bend on the particle cross-section concentration and segregation in pneumatic conveying systems." *Powder Technology* 98, no. 2 (1998): 95-103.

[4] Li, Hui, and Yuji Tomita. "An experimental study of swirling flow pneumatic conveying system in a horizontal pipeline." *Journal of fluids engineering* 118, no. 3 (1996): 526-530.

[5] Venkatasubramanian, Srikanth, Hiroyuki Tashiro, George E. Klinzing, and Kenneth Mykelbust. "Solids flow behavior in bends: assessing fine solids buildup." *Powder technology* 113, no. 1-2 (2000): 124-131.

[6] Peters, Thomas M., and David Leith. "Measurement of particle deposition in industrial ducts." *Journal of aerosol science* 35, no. 4 (2004): 529-540.

[7] S.M.Doma (2006).Master Thesis:Study on Secondary Flow Phenomenon in a Square and Circular Pipe Bend.UTM Skudai.

[8] Giguère, R., L. Fradette, D. Mignon, and P. A. Tanguy. "Analysis of slurry flow regimes downstream of a pipe bend." *Chemical Engineering Research and Design* 87, no. 7 (2009): 943-950.

[9] Sun, Ke, Lin Lu, and Hai Jiang. "A computational investigation of particle distribution and deposition in a 90° bend incorporating a particle–wall model." *Building and Environment* 46, no. 6 (2011): 1251-1262.

[10] Peters, Thomas M., and David Leith. "Particle deposition in industrial duct bends." *Annals of Occupational Hygiene* 48, no. 5 (2004): 483-490.

[11] Wong, M. K., L. C. Sheng, CS Nor Azwadi, and G. A. Hashim. "Numerical study of turbulent flow in pipe with sudden expansion." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 6 (2015): 34-48.

[12] STANDARD, BRITISH, and BSEN ISO. "Plastics piping systems for hot and cold water installations—Polypropylene (PP)—." (2004).

[13] R.W Johnson. (1998). The Handbook of Fluid Dynamic. Chapter 10, pg 10.8.

[14] Lehat, Md Sufian. "Study on Secondary Flow Through a Pipe Bend Using PIV and CFD." PhD diss., Universiti Teknologi Malaysia, 2006.

[15] Hunt, J. C. R., René Delfos, Ian Eames, and Richard J. Perkins. "Vortices, complex flows and inertial particles." *Flow, Turbulence and Combustion* 79, no. 3 (2007): 207-234.