Numerical 3D simulation of developing turbulent stratified gas-liquid flow in curved pipes consisting of entrained particles through this type of flow

M. A. Rasoulian (ali.rasoulian@aut.ac.ir), A. Rasteh (aminrasteh@aut.ac.ir), A. Farokhipour (farokhipour.ali@aut.ac.ir), Z. Mansoori (z.mansoori@aut.ac.ir), M. Saffar-Avval1*, (Mavval@aut.ac.ir), Ali Haj (alihaj@aut.ac.ir), Goodarz Ahmadi (gahmadi@clarkson.edu)

a. Department of Mechanical Engineering, Amirkabir University of Technology, Tehran, Iran
b. Energy Research Center, Amirkabir University of Technology, Tehran, Iran
c. Department of Mechanical and Aeronautical Engineering, Clarkson University, Potsdam, NY, USA

Abstract:
Since curved pipes are widely used in industrial equipment, predicting multiphase flows in these geometries is of great importance. In the present study, a computational model for predicting the velocity profile is developed and used to study the developing turbulent gas-liquid flows in curved pipes. In order to discretize and solve the three-dimensional steady-state momentum equations, the finite volume scheme on staggered grids besides central difference and QUICK scheme have been used. Moreover, the k-ε model is employed to reflect the nature of turbulence in the flow. In order to address the needs for sooner convergence and convenient mapping of the physical domain, the computations have been performed in a new toroidal coordinate system. Particle tracking has been done using Lagrangian approach in which two-way coupling regime is considered. In terms of validation, the numerical simulation results for the straight duct (infinite curvature), have been compared with the analytical solution and previous experimental results. Moreover, injection of particles through the flow indicates that, in each section of the bend, trade-off between centrifugal and pressure gradient forces plays a key role on particles motion. In last section, the effects of particle diameter and bend curvature on particle motion have been examined.

Keywords: Gas-liquid multiphase flow, stratified flow, Particle tracking, Curved pipe, Numerical analysis

1. Introduction

Behavior of multiphase flows in curved pipes is of interest to many industrial applications. Since bends are widely exploited, particularly, in pieces of equipment of gas and oil industry, the computational modeling of such flows in these geometries can help to predict the flow behavior passing through them to a large extent. The stratified flow velocity profiles of both gas and liquid phases, pressure losses along the passage and secondary flows occurring along the pipe, are significantly affected by curved section. Furthermore, in the production procedures, since the entrainment of sand particles to transmission lines frequently carrying gas-liquid flows is an inevitable issue during this process, their behavior should be carefully considered to prevent from the probable consequences such as pressure loss, erosion and equipment failure. Among the geometries, bends are more prone to erosion phenomenon due to the impingement of particles with the wall downstream of the curved section at which particles, because of their inertia, are not able to follow the streamlines and, consequently, detach from them and impact the inner surface of the pipe. There are many pieces of research available in the literature mostly conducted experimentally and mainly devoted to the pressure drop across the bend, the bulk flow measurements and mean void fraction of gas and liquid phases [1, 2]. In many applications, however, more detailed information of turbulent stratified flow such as gas and liquid velocity profiles besides the location and strength of secondary flows is required to be able to predict the particle motion and erosion wear caused by entrained particles in multiphase flows through the bend.

The multiphase flows in curved geometries are generally more complex than those in straight ones. This may be why there is rather limited amount of research in this area. The available literature also includes semi-empirical studies; for instance, Gardner and Neller [3] implemented visual and experimental studies for bubble/slug flows using a pipe of 76 mm diameter in a vertical 90° elbow with radii of curvature of 305 and 610 mm. In their work, they used a traversing probe to measure the local time averaged air concentration. They reported that gas can flow either on the outside or the inside of the bend depending on the balance between the centrifugal force and the gravity. The trade-off between these two forces is expressed in terms of Froude number, \(Fr_q = V^2/\alpha g \sin \theta\), where \(V\) is the mixture mean velocity, \(\alpha\) is the radius of curvature of the bend, and \(\theta\) is the stream-wise angle across the bend. That is noteworthy that the phases are flowing in radial equilibrium when the Froude number is equal to unity. For \(Fr > 1\), the gas is displaced towards the inside of the bend while for \(Fr < 1\), the gas moves to the outside of the bend. An experimental study was performed on two-phase mixture flowing in an inverted U-bend by Usui et al. [4, 5]. They measured some figures such as the local void fraction distribution, averaged void fraction over the bend with the aid of quick closing valves and slip ratio both through and in the vicinity of the bend. As a result of their work, they found that the secondary flow in bend would have governing effects upon the distribution of void fraction across the whole cross section of the pipe. Das et al. [6] and Bandyopadhyay et al. [7] performed an experimental study of gas/non-Newtonian liquid flow through a 12.7 mm curved pipe. They managed to develop an empirical correlation to estimate the pressure drop over the passage as a function of dynamic variables and physical parameters of the system. In another investigation by Ribeiro et al. [8], drop sizes upstream and downstream of a 90°, 32 mm internal diameter bend in a horizontal plane were measured. Their results demonstrated that the drop sizes downstream were 50–75% of those upstream. From their point of view, this trend was attributed to thicker films caused by the initial deposition which is

* Corresponding author.
E-mail address: mavval@aut.ac.ir (M. Saffar-Avval)
induced by the secondary flow occurring in the gas phase. Moreover, they found that the ratio of downstream to upstream entrained fraction would decrease with a rise in the upstream entrainment fraction. Azzi et al. [9] considered different correlations estimating the pressure loss for the variety of flow conditions, the bend orientation and geometries and, eventually, decided that, in order to calculate adequately the two-phase flow pressure loss in both vertical and horizontal ducts in wider parameter ranges, the effect of gravitational force in experiments should be taken into account.

Comprehensive studies focusing on the effect of the bend on the flow fields of a two phase air/water mixture were performed by Wang et al. in horizontal [10], [11] and vertical return bends [12], [13]. In these studies, for the horizontal experiments, return bends with 3~6.9 mm diameter and curvature ratios (curvature radius / pipe diameter) of 1.5, 2.5 and 3.5 were used while a single bend geometry (6.9 mm ID and curvature ratio of 1.5) was exploited for the vertical cases and the distribution of the phases was observed via still photography. In their investigation, the effect of parameters such as tube size, mass flux and curvature ratio were examined. Finally, with the aid of visual observation in the horizontal bends, the transition of flow regime from stratified to annular flow were detected. On the other hand, for the vertical return bend having \( D = 6.9 \text{ mm} \), some unique phenomena such as flow reversal and frozen slug flow were observed which were absent in previously reported results for horizontal configurations. However, such these phenomena (the flow reversal and freezing slug) were hardly seen at a smaller tube (\( D = 3 \text{ mm} \)) owing to the effect of the surface tension and flow inertia.

Spedding and Benard [14] measured the pressure drop for two-phase air/water pipe flow and three-phase air/oil/water through a 26 mm ID pipe and elbow (curvature radius / pipe diameter = 0.654) for vertical to horizontal flow. Two-phase results were highly dependent on the flow regimes present in the system. Three-phase data also showed noticeable but not such dramatic differences. Recently, several investigators implemented a few local measurements across an elbow by a wire-mesh sensor. Abdulkadir et al. [15] considered the flow fields of a 90° bend (curvature radius / pipe diameter = 2.3) with the aid of advanced instrumentation including wire-mesh sensor (WMS) and Electrical Capacitance Tomography (ECT) to give cross-sectionally resolved data on void fraction and high speed video. Although, for the vertical 90° bend, they observed that bubble, stratified, slug and semi-annular flows were present downstream, conversely, the flow regime showed the same configurations upstream of the bend for the horizontal 90° bend. As another result of their work, they found that ECT and WMS predicted same flow pattern signatures. Furthermore, in both the vertical and horizontal 90° bends, the gravitational force tends to move the liquid and gas to the inside and outside of the bend, respectively. Different slug characteristics such as the liquid slug body length distribution were identified by Kesana [16]. They also obtained the frequency of the slugs from void fraction data using dual wire-mesh sensors in the horizontal flow orientation. In their investigation, they examined the discrepancies between the properties of the slug flow and pseudo-slug flow regimes upstream and downstream of an elbow. In another study by dual wire-mesh sensor, Vieira et al. [17] focused on the two-phase stratified-slug transition, stratified wavy and annular flow regimes before and after a horizontal stainless steel elbow with an internal diameter of 76.2 mm and 144 mm radius of curvature. Analysis of time series void fraction data from WMS resulted in the determination of mean void fraction, local time average void fraction distribution, liquid phase distribution around the peripheral surface, interfacial structure velocities, as well as characteristic signatures within the cross-section of pipe before and after the elbow. In their research, the effect of some factors such as the superficial liquid and gas velocities and liquid viscosity on the measured parameters were also examined. The results indicated that the distribution of gas and liquid phases and interfacial velocities are significantly altered even 20D downstream of the elbow. A comprehensive experimental study in gas-solid heat transfer in a pipe with different configurations (horizontal, inclined and vertical positions) was implemented by Mohktarifar et al. [18]. The effects of parameters such as pipe slope and solid particles feed rate on the Nusselt number were examined. Results showed that the Nusselt number, in a dilute regime of the mixture, went down at lower solids feed rate; conversely, at higher solids feed rates, it went up. Importantly, near pipe position of 45°, Nusselt number took higher values.

In terms of CFD simulations, Nojabaei et al. [19] investigated heat transfer in turbulent liquid-solid suspension upward flow in a vertical pipe at which four-way coupling regime was considered. During the simulation, the effects of loading ratio, particle density and particle diameter on heat transfer coefficient were discussed. The results illustrated that particle to liquid density had a prominent effect on heat transfer and turbulence characteristics. The turbulent heat transfer in gas-solid flows through an inclined pipe with various inclination angles and under constant wall heat flux condition was investigated by Pishvar et al. [20]. As result of their study they reported that increases up to a certain angle led to a significant surge in Nusselt number. Dabirian et al. [21] undertook research using a commercial CFD code for 4 – in horizontal pipe carrying stratified turbulent air/water and presented the results such as liquid holdup, liquid height, wall shear stress and velocity profiles. Haj et al. [22] used a new form of toroidal coordinate system to address the need for convenient mapping of the physical domain where fluid flow pattern was considered laminar and stratified. They also employed bipolar coordinate in curved pipe cross section. They gained the results with the aid of assumptions by which the effect of liquid phase was considered as boundary condition at interfacial surface and the angular displacement of liquid phase was negligible. They reported that several vortices were formed in the gas phase which their size and strength varied along the curved section.

Currently, simultaneous Particle Image Velocimetry (PIV) measurements of stratified turbulent air/water flow in a horizontal pipe were performed using water droplets as tracers in the gas-phase by Ayati et al. [23]. The two-phase flow measurements indicated expected behaviour for low flow rates, and very interesting features related to the turbulence structure at higher flow rates that led to wavy interfaces. Inspired by ongoing approach, Vestøl et al. [24] produced detailed velocity profile measurements over a range of operating conditions of two phase gas/liquid flow with low liquid fractions in horizontal and inclined pipes. The experiments were implemented in a 15 m long stainless steel pipe with internal diameter of 56 mm at room temperature and atmospheric outlet pressure. In general, higher axial velocities were observed in the gas phase. Higher velocity gradients were observed close to the pipe wall and interface. Based on the horizontal gas-liquid flow measurements, the maximum velocity was located close to halfway between the interface and the upper wall.
Since the entrainment of solid particles into pipes carrying crude oil is an inevitable and inseparable part of production procedure in oil and gas industry, prediction of particles behaviour, particularly particle deposition, can prevent from operational problems such as decrease in production, excessive pressure drop, erosion, corrosion and, consequently, equipment failure. The operational condition in which continuous movement of particles through the pipe is guaranteed, will be the best approach to confront the deposition issue. This condition can be managed by operating above the critical particle deposition velocity, which is the velocity that satisfies the desire operational condition. In order to address this requirement, a few investigation regarding the particles in two-phase flows have been performed. Stevenson et al. [25] experimentally measured the average velocity of particles in smooth stratified flow through curved pipes. They found that particle velocity followed a linear trend with respect to the average velocity of the liquid phase of the flow. Moreover, it was seen that larger particles move more quickly than smaller ones along the pipe. As another result of their experiments, they understood that particle velocity decreased with increasing the liquid viscosity. This could be justified by consideration of the particle size relative to the depth of the viscous sublayer. Another experimental study was conducted by Dabirian et al. [26] in a 4-inch horizontal pipe carrying a stratified flow to examine the behaviour of solid particles (glass beads) with concentration ranges of 100 to 10,000 ppm. In their research, flow regime sequences for three ranges of particle size were investigated. They introduced the gas velocity, particle concentration and liquid holdup as key parameters in the creation of different particle regime flows. Furthermore, they reported the high dependence of critical particle deposition velocity to particle size as their sizes increased, the critical particle deposition velocity rose. Also, when the concentration of particles went up, for the smaller particles, the critical velocity remained fairly unchanged while gradually increased for larger ones (≥ 125 μm). Moreover, some of the most important characteristics of the stratified air-water flow in mini-channel were studied with the aid of a rule-based fuzzy inference system by Zehabiyan-Rezaie [27]. Their analysis demonstrated that the system is a viable approach to avoid the significant computational cost of the numerical simulations. In case of CFD prediction of erosion phenomenon, Banakermani et al. [28] performed simulations for a range of elbow angles for two different flow orientations; horizontal inlet and outlet flow directions (H-H flow) and the cases in which vertical inlet existed and outlet flows were in the horizontal plane or with some angles (V-H flow). As an important result, they found that total annual eroded volume in the H-H case configuration is larger than V-H case figure. Moreover, they results showed that in both V-H and H-H cases, over the range of mass loadings under investigation, the maximum erosion rate increased gradually when the elbow angle went up from 15° to 90°; however, the rate of total eroded volume remained relatively constant over the sand rate range.

The thrust of this work is studying the developing turbulent stratified gas-liquid and in horizontal curved pipes consisting of entrained particles. A computational model for predicting velocity profiles and secondary flows has been developed. The three-dimensional steady-state momentum equations, with the aid of the k-ε turbulence model, are discretized and solved using the finite volume scheme. The computations have been performed in the extended form of toroidal and bipolar coordinate system. Axial velocity contours and secondary flows are predicted in the gas phase in the curved pipe at four axial sections over the bend. Particle tracking is implemented by Lagrangian approach considering two-way coupling regime. The behavior of particles has been discussed in three cross-sectional position of the pipe along the curved section and the effects of particle diameter and bend curvature on particles motion have been examined. Experimental results and analytical data for the case of developed stratified gas-liquid flow in curved pipe are used in order to validate the present results.

2. Mathematical formulation

2.1. Coordinate system

In this study, in order to match the coordinate system with the flow configuration in curved geometries, the orthogonal toroidal coordinate system is used for simulation of the turbulent stratified flow in curved pipes. The toroidal coordinate system is depicted in Fig. 1. The liquid and gas domains in stratified pipe flow, as shown in Fig. 1, are modelled by the bipolar coordinate system at the pipe cross section described as

\[ x = c \frac{\sinh \eta}{\cosh \eta - \cos \xi} \quad (1) \]

\[ z = c \frac{\sinh \eta}{\cosh \eta - \cos \xi} \quad (2) \]

\[
\begin{cases}
\gamma \leq \xi < \pi, & -\infty < \eta < \infty, \quad \text{for upper fluid (air)} \\
\pi \leq \xi < \pi + \gamma, & -\infty < \eta < \infty, \quad \text{for lower fluid (water)}
\end{cases} \quad (3)
\]

where \( c \) and \( \gamma \), as illustrated in Fig. 1, are focal length in bipolar coordinate system and the half view angle of the interface from centre of the pipe, respectively. In other words \( \gamma \) is equal to half the angle subtended by the centre of the pipe and the gas–liquid interface and is defined as
\[ \gamma = \cos^{-1} \left[ 1 - \frac{2hL}{D} \right] \]  

(4)

Here \( h \) and \( D \) represent the liquid height and pipe internal diameter, respectively. In this study, in order to address the needs for sooner convergence and convenient mapping of the physical domain, the new form of toroidal coordinate system is created and used. Therefore, the Navier-Stokes governing equations need to be stated in the frame of aforementioned coordinate system. For this, bipolar coordinate system should be expressed in the reference Cartesian coordinate system \( XYZ \). The position of any point through the curved geometry can be defined by the vector \( \vec{x} \). Thus, \( \vec{x} \) can be calculated by:

\[ \vec{x} = \vec{R}(s) + \vec{r} \]  

(5)

where \( \vec{R}(s) \) and \( \vec{r} \) (in the reference Cartesian coordinate system \( XYZ \)) are described as

\[ R(s) = a \cos \left( \frac{s}{a} \right) i + a \sin \left( \frac{s}{a} \right) j \]  

(6)

\[ \vec{r} = c \frac{\sinh \eta}{\cosh \eta - \cos \xi} \vec{N}(s) + c \frac{\sinh \eta}{\cosh \eta - \cos \xi} \hat{k} \]  

(7)

where \( a \) is the radius of curvature of the curve, \( s \) is axial direction (stream-wise direction) and \( \vec{N}(s) \) represents the principal normal vector of \( s \), and is described by:

\[ \vec{N}(s) = -\cos \left( \frac{s}{a} \right) i - \sin \left( \frac{s}{a} \right) j \]  

(8)

Thus, the position of each point inside the curve in the reference Cartesian coordinate system \( XYZ \) can be defined by the vector \( \vec{x} \) as

\[ \vec{x} = \left[ a \cos \left( \frac{s}{a} \right) + c \frac{\sinh \eta}{\cosh \eta - \cos \xi} \cos \left( \frac{s}{a} \right) \right] \hat{i} + \left[ a \sin \left( \frac{s}{a} \right) + c \frac{\sinh \eta}{\cosh \eta - \cos \xi} \sin \left( \frac{s}{a} \right) \right] \hat{j} \\
+ c \frac{\sin \eta}{\cosh \eta - \cos \xi} \hat{k} \]  

(9)

The corresponding scale factors (metric coefficients) are provided by

\[ h_\xi = \sqrt{\left( \frac{\partial \chi_X}{\partial \xi} \right)^2 + \left( \frac{\partial \chi_Y}{\partial \xi} \right)^2 + \left( \frac{\partial \chi_Z}{\partial \xi} \right)^2} = \frac{1}{\cosh \eta - \cos \xi} \]  

(10)

\[ h_\eta = \sqrt{\left( \frac{\partial \chi_X}{\partial \eta} \right)^2 + \left( \frac{\partial \chi_Y}{\partial \eta} \right)^2 + \left( \frac{\partial \chi_Z}{\partial \eta} \right)^2} = \frac{1}{\cosh \eta - \cos \xi} \]  

(11)

\[ h_s = \sqrt{\left( \frac{\partial \chi_X}{\partial s} \right)^2 + \left( \frac{\partial \chi_Y}{\partial s} \right)^2 + \left( \frac{\partial \chi_Z}{\partial s} \right)^2} = 1 + \frac{c \sinh \eta}{a \left( \cosh \eta - \cos \xi \right)} \]  

(12)

2.2. Governing equations

The Reynolds-Averaged Navier–Stokes (RANS) continuity and momentum equations, for steady-state incompressible and viscous flow in the curvilinear new coordinates system \( (\xi, \eta, s) \), are given as [29]

- Continuity

\[ \frac{1}{h_{\xi} h_{\eta} h_s} \left[ \frac{\partial \left( h_{\xi} h_{\eta} u \right)}{\partial \xi} + \frac{\partial \left( h_{\xi} h_{\eta} v \right)}{\partial \eta} + \frac{\partial \left( h_{\xi} h_{\eta} w \right)}{\partial s} \right] = 0 \]  

(13)
\[ \frac{1}{h_{\xi}h_{\eta}} \left[ \frac{\partial}{\partial \xi} \left( \rho h_{\xi}U \right) + \frac{\partial}{\partial \eta} \left( \rho h_{\eta}V \right) + \frac{\partial}{\partial s} \left( \rho h_{s}W \right) \right] \]
\[ + \frac{\rho V}{h_{\xi}} \left( U \frac{\partial h_{\xi}}{\partial \eta} - \frac{\partial h_{\eta}}{\partial \eta} \right) + \frac{\rho W}{h_{\eta}} \left( W \frac{\partial h_{\eta}}{\partial \eta} \right) \]
\[ \frac{1}{h_{\xi}h_{\eta}} \left[ \frac{\partial p}{\partial \xi} + \mu \left( \frac{1}{h_{\xi}h_{\eta}} \left[ \frac{\partial}{\partial \xi} \left( \rho h_{\xi}U \right) + \frac{\partial}{\partial \eta} \left( \rho h_{\eta}V \right) + \frac{\partial}{\partial s} \left( \rho h_{s}W \right) \right] \right) \right] \]
\[ - \frac{1}{h_{\xi}h_{\eta}} \left[ \frac{\partial}{\partial \eta} \left( h_{\eta} \frac{\partial h_{\xi}}{\partial \xi} \right) + \frac{\partial}{\partial s} \left( h_{s} \left( \frac{\partial h_{\xi}}{\partial s} - \frac{\partial h_{\xi}}{\partial \xi} \right) \right) \right] \]
\[ + \rho g_{\zeta} + S_{\xi}' + \nabla \left( -\rho u'v' \right) \]

\[ \eta \text{ momentum} \]
\[ \frac{1}{h_{\xi}h_{\eta}} \left[ \frac{\partial}{\partial \xi} \left( \rho h_{\xi}V \right) + \frac{\partial}{\partial \eta} \left( \rho h_{\eta}V \right) + \frac{\partial}{\partial s} \left( \rho h_{s}W \right) \right] \]
\[ + \frac{\rho U}{h_{\xi}} \left( V \frac{\partial h_{\xi}}{\partial \eta} - U \frac{\partial h_{\eta}}{\partial \eta} \right) + \frac{\rho W}{h_{\eta}} \left( W \frac{\partial h_{\eta}}{\partial \eta} \right) \]
\[ \frac{1}{h_{\xi}h_{\eta}} \left[ \frac{\partial p}{\partial \eta} + \mu \left( \frac{1}{h_{\xi}h_{\eta}} \left[ \frac{\partial}{\partial \xi} \left( \rho h_{\xi}U \right) + \frac{\partial}{\partial \eta} \left( \rho h_{\eta}V \right) + \frac{\partial}{\partial s} \left( \rho h_{s}W \right) \right] \right) \right] \]
\[ - \frac{1}{h_{\xi}h_{\eta}} \left[ \frac{\partial}{\partial s} \left( h_{s} \frac{\partial h_{\eta}}{\partial s} - \frac{\partial h_{\eta}}{\partial \eta} \right) \right] \]
\[ + \rho g_{\zeta} + S_{\eta}' + \nabla \left( -\rho u'v' \right) \]

\[ s \text{ momentum} \]
\[ \frac{1}{h_{\xi}h_{\eta}} \left[ \frac{\partial}{\partial \xi} \left( \rho h_{\xi}W \right) + \frac{\partial}{\partial \eta} \left( \rho h_{\eta}W \right) + \frac{\partial}{\partial s} \left( \rho h_{s}W \right) \right] \]
\[ + \frac{\rho U}{h_{\xi}} \left( W \frac{\partial h_{\xi}}{\partial \eta} \right) + \frac{\rho V}{h_{\eta}} \left( W \frac{\partial h_{\eta}}{\partial \eta} \right) \]
\[ \frac{1}{h_{\xi}h_{\eta}} \left[ \frac{\partial p}{\partial s} + \mu \left( \frac{1}{h_{\xi}h_{\eta}} \left[ \frac{\partial}{\partial \xi} \left( \rho h_{\xi}U \right) + \frac{\partial}{\partial \eta} \left( \rho h_{\eta}V \right) + \frac{\partial}{\partial s} \left( \rho h_{s}W \right) \right] \right) \right] \]
\[ - \frac{1}{h_{\xi}h_{\eta}} \left[ \frac{\partial}{\partial \xi} \left( h_{s} \frac{\partial h_{\xi}}{\partial s} - \frac{\partial h_{\xi}}{\partial \xi} \right) \right] \]
\[ + \rho g_{\zeta} + S_{s}' + \nabla \left( -\rho w'v' \right) \]

Here \( u, v, w \) are the velocity components, \( U, V, W \) are the mean velocity components, \( u', v', w' \) are the fluctuation velocity components and \( S_{\xi}', S_{\eta}', S_{s}' \) are the source or sink terms when the particles exist in the flow field and the career fluid is influenced by them (two-way coupling regime) and, in \( \xi, \eta \) and \( s \) directions, are respectively defined by...
\[
S_p = \varphi_p \left( -\rho_p \left[ \frac{du}{dt} - g_z \right] \right) 
\]
(17)

\[
S_p = \varphi_p \left( -\rho_p \left[ \frac{dv}{dt} - g_y \right] \right) 
\]
(18)

\[
S_p = \varphi_p \left( -\rho_p \left[ \frac{dw}{dt} - g_i \right] \right) 
\]
(19)

here \( \varphi_p \) and \( \rho_p \) are particle mass loading and density, respectively. Apart from the situation in which the flow regime has been considered two-way coupling, these terms are neglected. The last term on the right hand sides of momentum equations reflects the effect of turbulence through the flow field in terms of turbulent stress. In order to solve the RANS equations, extra modelling for Reynolds stress term \( (\rho u'v') \) is required. This term is defined by [30]

\[
\tau_{ij} = -\rho u_i' v_j' = \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} 
\]
(20)

where \( k \) is the turbulence kinetic energy per unit mass quantified by the mean of the turbulence normal stresses. Thus, it is computed by

\[
k = \frac{1}{2} \left( u'^2 + v'^2 + w'^2 \right)
\]
(21)

For the case of this work, among several turbulence models, which are developed to calculate the Reynolds stress term, the two-equation standard \( k-\varepsilon \) turbulence model is selected. This turbulence model, which can provide excellent performance for flows in which boundary layers under adverse pressure gradient, separation and recirculation exist, has been successfully applied in this CFD simulation. The two equations of the standard \( k-\varepsilon \) turbulence model for calculation of the turbulence kinetic energy, \( k \), and dissipation rate, \( \varepsilon \), are, respectively, written as follows

\[
\nabla \left( \rho k \vec{V} \right) = \nabla \left[ \frac{\mu}{\sigma_k} \nabla k \right] + 2 \mu S_i S_i - \rho \vec{c} 
\]
(22)

\[
\nabla \left( \rho \varepsilon \vec{V} \right) = \nabla \left[ \frac{\mu}{\sigma_\varepsilon} \nabla \varepsilon \right] + C_{1_\varepsilon} \frac{\varepsilon}{k} 2 \mu S_i S_i - C_{2_\varepsilon} \rho \frac{\varepsilon^2}{k} 
\]
(23)

where \( \mu_t \) represents the eddy viscosity and is computed as follows

\[
\mu_t = \rho C_f \frac{k^2}{\varepsilon} 
\]
(24)

In these equations, \( S_{ij} \) represents component of rate of deformation and the model constants are \( \sigma_k = 1.00, \sigma_\varepsilon = 1.30, C_{1_\varepsilon} = 1.44, C_{2_\varepsilon} = 1.92 \) and \( C_f = 0.09 \).

2.3. Particle tracking

In this step of the study, the sand particles are injected from the pipe inlet into the flow and tracked using the Lagrangian approach. Inspired by Newton’s second law, the particle motion is a direct consequence of trade-off between forces acting on them, which is defined by

\[
m_p \frac{d\vec{V}_p}{dt} = \vec{F}_D + \vec{F}_A + \vec{F}_P + \vec{F}_G
\]
(25)

where, \( m_p \), particle mass, \( \vec{V}_p \), velocity of the particle, \( \vec{F}_D \), Drag force, \( \vec{F}_A \), added mass force, \( \vec{F}_P \), pressure-gradient force, and \( \vec{F}_G \) is the resultant of gravity and buoyancy forces. These forces are calculated according to the Eqs. (26) to (29) [31]
\[ \bar{F}_d = \frac{3 \rho_f m C_d}{4 \rho_p d_p} \left| \vec{V} - \vec{V}' \right| \left( \vec{V} - \vec{V}' \right) \]  
(26)

\[ \bar{F}_s = -\frac{1}{2} m \frac{\rho_f}{\rho_p} \frac{dV_s}{dt} \]  
(27)

\[ \bar{F}_p = -\frac{1}{4} \pi d_p^2 g \]  
(28)

\[ \bar{F}_e = m_p \left( 1 - \frac{\rho_f}{\rho_p} \right) g \]  
(29)

where \( \rho_f, V_f, \) and \( d_p \) are, respectively, fluid density, fluid velocity, and particle diameter. \( C_d \) is the drag coefficient based on Schiller and Naumann model [32] and \( \text{Re}_p = \frac{\rho_f d_p \left| \vec{V} - \vec{V}' \right|}{\mu_f} \), where \( \mu_f \) is the dynamic viscosity of fluid, represents the relative Reynolds number. Fourth-order Range-Kutta method is used to solve the Eq. (25).

The Crowe model [33] has been used to model the collision of the particle with the wall and the gas-liquid interface. The return velocity is usually less than the collision velocity, and the reason for this drop is the energy transfer from the particle to the wall. The return velocity depends on several important factors such as wall and particle material, incident angle and incident velocity. According to this model, the particle velocity components after the collision can be calculated as follows

\[ \begin{align*}
    v_x &= v_x^{(o)} + \varepsilon_x f (e + 1) v_y^{(o)} \\
    v_y &= -\varepsilon_y f (e + 1) v_y^{(o)} \\
    v_z &= v_z^{(o)} + \varepsilon_z f (e + 1) v_z^{(o)}
\end{align*} \]  
(30)

where \( v_x^{(o)}, v_y^{(o)}, v_z^{(o)} \) are components of incident velocity and \( v_x, v_y, v_z \) are components of return velocity. \( f \) represents skin friction coefficient of wall, \( e \) is coefficient of restitution and \( \varepsilon_x \) and \( \varepsilon_z \) represent direction cosines in \( x-z \) plane. The coefficients of restitution and friction on the wall and interface are \( e_w = 0.95, e_{int} = 0.6, f_w = 0.3 \) and \( f_{int} = 0.5 \).

3. Definition of problem and model

The current research concentrates on a horizontal curved pipe with an internal diameter of 100 mm and the radius of curvature of \( a = 300 \) mm which carries the two-phase turbulent stratified flow with low liquid holdup. The length of the 90° curved pipe is 471 mm. Liquid holdup at the bottom of the pipe is assumed to have a depth of 10 mm. That is, water occupies 5.76% of the cross section of pipe and, consequently, the bulk of the pipe volume is filled by gas. There is a reasonable assumption during the simulation by which liquid height has been considered unchanged. In this situation, the gas phase can be separately modelled and the effect of liquid phase appears as boundary conditions at the gas-liquid interface. The continuous phase is air with density of 1.2 kg/m³ and dynamic viscosity of \( 1.51 \times 10^{-5} \text{ N.s/m}^2 \).

Influenced by centrifugal force, the gas-liquid interface in the bend section moves from the horizontal position. Based on assumption of the smooth interface between gas and liquid, Castillo [2] acquired an analytical expression for angular deviation of gas-liquid interface against the pipe axis depicted in Fig. 2 and given as,

\[ \varphi = \frac{w^2}{ag} \left( 1 - \cos \left( \sqrt{\frac{8y}{K_0 - l}} \right) \right) \]  
(31)

where \( \varphi \) is the liquid free surface angle with the horizontal surface as shown in Fig. 2, \( w \) is the mean stream-wise (axial) velocity of liquid phase, \( y \) is the distance between cross-sectional center of gravity of liquid phase and pipe axis, and \( K_0 \) represents cross-sectional gyration radius of liquid part about the pipe axis. According to Eq. (31), the maximum angular deviation would be equal to \( 2w^2/aga \). That is worth mentioning that Eq. (31) is only reasonable for very small angular deviations.
In this investigation, the mean velocities of the liquid and gas phases are considered 0.02 m/s and 4 m/s, respectively. According to these values and Eq. (31), the maximum displacement would be 0.016°. Since the angular deviation is negligible, in this study, the alteration of gas-liquid interface from completely horizontal position has been ignored. In order to ensure that the flow pattern is not varying as the flow passes through the curved section, qualitative analogy with the experimental results obtained by Vieira et al. [17] has been drawn. Experiments for the gas-liquid stratified wavy flow with gas void fraction of 0.93, superficial liquid velocity of 0.03 m/s and superficial gas velocity of 9 m/s passing through the 90° elbow with curvature ratio, $\delta$ (pipe radius, $R$/curvature radius, $a$) of 0.3342, the flow pattern has not been displaced from horizontal position. In this work, since the curvature ratio of the considered bend is 0.17 (50/300), the induced centrifugal force is much less and, consequently, the corresponding maximum angular deviation is smaller; therefore, the aforementioned assumption is not far away from the fact.

4. Numerical method

The utilization of extended toroidal coordinate system boosts it to develop an effective computational technique for study of the stratified flow in curved passages. The computational domain of the gas/liquid flow has been illustrated in Fig. 3. For ease of computation, the domain is separated for a given value of liquid holdup and then, the flow fields will be solved independently for each phase by specifying the appropriate boundary conditions at the interfacial surface.

The Navier-Stokes equations, which are expressed in the orthogonal coordinate system ($\xi, \eta, s$), are used to describe the gas phase flow field through the bend. A finite volume method on staggered grids is used to discretize the governing equations. Central difference and QUICK scheme are used to discretize the diffusion and convection terms, respectively. Moreover, the k-ε model is employed to reflect the nature of turbulence in the flow field.

In terms of boundary conditions, at the walls the impenetrability and no-slip boundary condition are enforced. At the inlet, a Poiseuille velocity distribution has been considered while a constant pressure outflow at the outlet of the curved pipe is applied. The main purpose of this work is to predict the gas phase velocity field; therefore, a simplifying assumption at the gas/liquid interface has been made by which the axial velocity of interface is constant during the simulation and equal to the mean axial velocity of the liquid phase and the other components of the velocity are considered zero.

In order to validate the present CFD code, the simulation results have been compared with the experimental data by Bovendeerd et al. [34] for the case of developing single phase flow in a 90° bend pipe. Here, pipe internal radius is 4.0 mm, radius of curvature is 24.0 mm and Reynolds number is 700. In Figs. 4 and 5, the predicted dimensionless axial velocity profile in the symmetry plane at two sections of the curve at $z =$ 1 and 1.7 (where $z = R\theta/(aR)^{0.5}$, $R$ and $\theta$ are pipe radius and axial direction along the curve, respectively, as shown in Fig. 6), are compared with the experimental measurements by Bovendeerd et al. [34]. The vertical axis in these figures represent the dimensionless axial velocity, which is made non-dimensional with the mean axial velocity. On the other hand, the horizontal axis is dimensionless distance from the outer wall of the bend. ($x_1$ is the distance from the outer wall). It can be seen that the predicted dimensionless axial velocity profiles are in good agreement with the experimental measurements.

To examine the validation of the present model for the stratified gas-liquid flows, a condition in which the bend radius of curvature tends to infinity has been considered and the results are compared with those for the straight pipes. The predicted dimensionless axial velocity profile of the gas-liquid phase for fully developed stratified flow in a straight pipe is compared with the analytical solution provided by Brauner [35] in Fig. 7. The axial velocities have been made non-dimensional by

$$V_x = \frac{R^2}{4\mu_L} \frac{dp}{ds}$$  \hspace{1cm} (32)

For this case, to obtain the exact value of the interfacial velocity, its value has been initialized and then, the gas phase equations are solved and axial gas velocity profile is evaluated. In the next step, the corresponding averaged shear stress at the interface is calculated and, consequently, is applied to the liquid phase. Eventually, the liquid phase is solved and the interface velocity is calculated. The new interface velocity is then applied to the gas phase and the solution process will be repeated until the gas-liquid interfacial velocity becomes unchanged and the solution converges. Figure 7 demonstrates that the present model prediction is in excellent agreement with the analytical solution.

Lastly, for validation of turbulence in fully developed pipe flow, the results are compared with experimental ones provided by Laufer [36]. In experiments, pipe internal radius was 127 mm, radius of curvature tends to infinity (straight pipe) carrying flow with Reynolds number of 50000. This comprehensive comparison consists of three items including axial velocity profile, turbulence kinetic energy and dissipation rate displayed in Figs. 8, 9 and 10, respectively. These parameters have been made dimensionless by
Although comparing the CFD simulation results with axial velocity and turbulence parameters measured experimentally revealed a satisfactory agreement (discrepancy < ±15%), there is a slight deviation near the wall mainly caused by the difference between initial y* and actual one leading to the right selection of appropriate turbulence model.

5. Results and discussion

When the two-phase flow passes through the bend, the maximum axial velocity of the gas phase tends towards the outer wall and the secondary flow forms and evolves along the pipe. Figure 11 shows axial velocity contours and Fig. 12 displays the corresponding secondary flows occurring in the gas phase through the curved pipe at four axial sections: s = 0.03 m, 0.1 m, 0.14 m and 0.18 m. The gas Reynolds number is about 27000 in all cases.

Figure 11 depicts that the central maximum stream-wise velocity region gradually disappears and is shifted to the region farther away from the inner wall (left) and closer to the gas-liquid interface, top and outer pipe wall (right). Near the inner wall, the axial velocity magnitude is very low while near the outer wall is quite high. Therefore, the radial velocity gradient is very high at the outer wall although this gradient is much lower between the inner wall and maximum axial velocity region. The contour lines of the axial velocity are asymmetric with respect to horizontal centerline and this asymmetric behavior becomes more evident along the pipe from s = 0.03 m to s = 0.18 m. At s = 0.18 m the contours seem to be highly skewed.

Figure 12 shows that at s = 0.03 m, the secondary flows in the gas phase generate four vortex zones. Two vortices are in the vicinity of gas-liquid interface and pipe wall while the other two larger vortices are created in the upper part of the pipe cross section. By drawing an analogy among the secondary flows at all axial sections, as provided in Fig. 12, it can be understood that the two lower vortices located in the corners of gas-liquid interface and that one near the inner wall become stronger along the pipe while the other one does not vary noticeably.

In this step, the behaviour of entrained particles through the pipe carrying considered flow will be examined. Table 1 contains the parameters describing the geometry, flow, and entrained particles.

Particle injection region from the inlet surface is ranged as follows

\[
\begin{align*}
0.8511 \leq & \xi \leq 1.8998 \\
-0.8306 \leq & \eta \leq 0.8306
\end{align*}
\]

According to Fig. 13, the particles are enforced to move towards the outer wall during the passage through the bend influenced by the centrifugal force. Moreover, they are continuously under the influence of gravitational force. On the other hand, pressure gradient in the radial direction occurs in the flow field; therefore, in each section of the bend, the trade-off between aforesaid forces is different and plays a key role on particles motion. Near the internal walls, the dominant force is the gravity and the particles tend to have lower concentration than the outer wall of the bend.

As shown in Fig. 14 and Fig. 15, by decreasing the radius of curvature, the particles are more oriented towards the inner wall. The particles are mostly inclined to the vortex located on the right side of the pipe, near the interface, and are still more likely to collide with the interface.

Furthermore, in order to investigate the effect of particle diameter on their path, three particle diameters of 5, 50 and 200 microns are injected from the center of inlet surface. The direction of movement of these particles is depicted in Fig. 16. By decreasing the diameter of the particles, the pressure-gradient force, the added mass force and the gravity force decrease
with the third power of the diameter, but the drag force decreases with the first power of the particle diameter and, consequently, centrifugal force acting on the particles overcomes other forces and pushes the particles towards outer wall.

6. Conclusions

For simulating the gas-liquid stratified flow in a horizontal curved pipe, a new toroidal coordinate system has been developed and utilized for convenient mapping of the physical domain and the gas-liquid interface. The 3-D steady-state momentum equations are discretized and solved using the finite volume scheme. Comparing the numerical data with the previous experimental results, the validity of the numerical model is examined and it is shown that the new model can precisely handle developed stratified gas-liquid flows in curved pipe, particularly at low radius of curvature.

In general, higher axial velocities are observed in the gas phase. Based on the horizontal gas-liquid flow simulation, the maximum velocity is detected close to halfway between the interface and the upper wall. This could be attributed to secondary flows similar to the results acquired by Meknassi et al. [37]. The current results confirm the presence of secondary flows in the gas phase indicating that the flow is not fully developed.

The present results illustrates that the contour lines of the axial velocity in gas-liquid stratified flow are asymmetric with respect to horizontal centerline. At the axial section from \( s = 0.03 \) m to 0.18 m, the secondary flow forms four major vortex (recirculation) zones. Furthermore, the two secondary vortices near the corners of the gas-liquid interface are found to be stronger than the other vortices and exist at all sections in the curved pipe.

Entrained particle tracking is performed with the aid of Lagrangian approach considering two-way coupling regime between the flow and particles. To solve the equation of particle motion, the fourth-order Range-Kutta method is exploited. Finally, the results obtained by the particle tracking indicates that, in each section of the bend, the trade-off between centrifugal and pressure gradient force plays a key role on particles motion as near the inner surface, the dominant force is the gravity and the particles tend to have lower concentration than the outer wall of the bend. In terms of the effect of particle size, decreasing the particle diameter results in enhancing the centrifugal force which pushes the particles towards outer wall. Furthermore, the comparison of particles motion through bends with various curvature demonstrates that by decreasing the radius of curvature, the particles are more oriented towards the inner wall. The particles are mostly inclined to the vortex located on the right side of the pipe, near the interface, and are still more likely to collide with the interface.

**Notation**

- \( a \) curvature radius, m
- \( D \) pipe diameter, m
- \( c \) focal length in bipolar coordinate system, m
- \( h_L \) liquid height, m
- \( h_\eta, h_\xi, h_z \) scale factors
- \( R \) pipe radius, m
- \( p \) pressure, Pa
- \( s \) axial direction, m
- \( u \) velocity component in \( \eta \) direction, \( \text{m} \text{s}^{-1} \)
- \( v \) velocity component in \( \xi \) direction, \( \text{m} \text{s}^{-1} \)
- \( w \) axial velocity, \( \text{m} \text{s}^{-1} \)
- \( w_{\text{mean}} \) mean axial velocity, \( \text{m} \text{s}^{-1} \)
- \( X, Y, Z \) Cartesian coordinates, m
- \( k \) turbulence kinetic energy
- \( g \) gravity

**Greek letters**

- \( \gamma \) half-angle subtended by liquid at center of pipe
- \( \delta \) curvature ratio
- \( \eta, \xi \) bipolar coordinates
- \( \theta \) Stream-wise angle
\( \mu \) viscosity, Pa.s
\( \rho \) density, kg.m\(^{-3}\)
\( \varphi \) Angular displacement of gas-liquid interface about the pipe axis, radian
\( \varepsilon \) turbulence dissipation rate

Subscripts

\( G \) gas
\( L \) liquid
\( \eta, \xi, \zeta \) new toroidal coordinates
\( p \) particle

References

Mohammad Ali Rasoulian received his BSc degree in Mechanical Engineering from Ferdowsi University of Mashhad, Mashhad, Iran, in 2011, and his MSc degree in Mechanical Engineering from Amirkabir University of Technology (Tehran Polytechnic), Tehran, Iran, in 2014. Currently, he is a research assistant at Energy Research Center of Amirkabir University of Technology. His research interests include Computational Fluid Dynamics (CFD), Particle Transport, Windbreaks and Wind Flows, Solid Particle Erosion and Multiphase Flows.

Amin Rasteh received his BSc degree in Mechanical Engineering from Yasuj University, Iran, in 2011, and his MSc degree in Mechanical Engineering from Amirkabir University of Technology (Tehran Polytechnic), Iran, in 2014. He is currently a research assistant at Energy Research Center in Amirkabir University of Technology. His research interests include Computational Fluid Dynamics, Convection Heat Transfer, Solid Particle Erosion, Multiphase Flows and Turbomachines.

Ali Farokhipour received his BS degree in Aerospace Engineering from K. N. Toosi University of Technology, Iran, in 2011, and an MS degree in Mechanical Engineering from Amirkabir University of Technology (Tehran Polytechnic), Iran, in 2013. where he is currently a PhD degree student. His research interests include computational fluid dynamics, gas turbine, non-premixed combustion, solid particle erosion and multiphase flows.

Zohreh Mansoori completed her PhD in Mechanical Engineering from Amirkabir University of Technology (Polytechnic), Tehran, Iran. She is Associate Professor and the Head of Energy Research Center in Amirkabir University of Technology. She has published more than 40 papers in reputed journals and has been serving as an Editorial Board Member of repute.

Majid Saffar-Avval is a Professor of Mechanical Engineering at Amirkabir University of Technology (AUT), Tehran, Iran. He received his BSc and MSc degrees from Sharif University of Technology and his PhD degree from the Ecole Nationale des Arts et Metiers (ENSAM), Paris, France, in 1985. He has been teaching at AUT since then. He was head of the Mechanical Engineering Department from June 2000 to June 2002 and head of the Energy and Control Center of Excellence from May 2007 to March 2012 at AUT. His research contributions are in the fields of two-phase heat transfer, advanced thermal systems, energy management, and bio-heat transfer.

Ali Haj received his MSc degree in Mechanical Engineering from Amirkabir University of Technology (Tehran Polytechnic), Iran, in 2016. His research interests include heat transfer enhancement, two-phase flow modelling and Computational Fluid Dynamics (CFD).

Goodarz Ahmadi received his Ph.D. degree in Mechanical Engineering from the Purdue University, West Lafayette, IN, USA, and is now Professor of Mechanical Engineering at Clarkson University, USA. His research work concerns: Particle Transport, Deposition and Removal, Aerosol Mechanics, Stochastic Processes in Engineering, Advance Theory of Turbulence, Multiphase Flow Modelling.
Table 1. Simulated cases in particle tracking section.
Fig. 1 (a) Schematic diagram of the new toroidal coordinates system $(\xi, \eta, s)$ (b) The bipolar coordinate at the pipe cross section.

Fig. 2 Angular displacement of gas-liquid interface about the pipe axis.

Fig. 3 Computational domain of the gas and liquid flows.

Fig. 4 The dimensionless axial velocity profile at the AA plane; $z = 1, \delta = 1/6, Re = 700$ [34].

Fig. 5 The dimensionless axial velocity profile at the AA plane; $z = 1.7, \delta = 1/6, Re = 700$ [34].

Fig. 6 Computational domain of the bend.

Fig. 7 Comparison of the predicted dimensionless axial velocity profile along the vertical centerline with the analytical solution of Brauner [35], $h_L = D/10, \mu_G/\mu_L = 0.01$.

Fig. 8 Comparison of the predicted dimensionless axial velocity profile along the vertical centerline with the experimental results by Laufer [36].

Fig. 9 Comparison of the predicted dimensionless turbulence kinetic energy profile along the vertical centerline with the experimental results by Laufer [36].

Fig. 10 Comparison of the predicted dimensionless turbulence dissipation rate profile along the vertical centerline with the experimental results by Laufer [36].

Fig. 11 Axial velocity contours in the gas phase at $s = 0.03, 0.1, 0.14$ and 0.18.

Fig. 12 Secondary flow in the gas phase at $s = 0.03, 0.1, 0.14$ and 0.18.

Fig. 13 Concentration of particles in the bend (curvature radius= 0.3 m)

Fig. 14 Concentration of particles in the bend (curvature radius= 0.25 m)

Fig. 15 Concentration of particles in the bend (curvature radius= 0.1274 m)

Fig. 16 Comparison of motion of particles with different sizes
Table 1. Simulated cases in particle tracking section.

<table>
<thead>
<tr>
<th>Pipe Orientation</th>
<th>$H - H$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bend length</td>
<td>0.2 m</td>
</tr>
<tr>
<td>Curvature radius</td>
<td>0.1274, 0.25, 0.3 m</td>
</tr>
<tr>
<td>Particle diameter</td>
<td>200 µm</td>
</tr>
<tr>
<td>Particle density</td>
<td>2500 kg/m$^3$</td>
</tr>
<tr>
<td>Particle initial velocity</td>
<td>70% of gas velocity</td>
</tr>
<tr>
<td>Gas velocity</td>
<td>2 m/s</td>
</tr>
<tr>
<td>Interface velocity</td>
<td>0.01 m/s</td>
</tr>
<tr>
<td>Liquid film thickness</td>
<td>0.01 m</td>
</tr>
</tbody>
</table>
Fig. 1 (a) Schematic diagram of the new toroidal coordinates system ($\xi, \eta, s$) (b) The bipolar coordinate at the pipe cross section.

Fig. 2 Angular displacement of gas-liquid interface about the pipe axis.

Fig. 3 Computational domain of the gas and liquid flows.
Fig. 4 The dimensionless axial velocity profile at the AA plane; $z = 1$, $\delta = 1/6$, $Re = 700$ [34].

Fig. 5 The dimensionless axial velocity profile at the AA plane; $z = 1.7$, $\delta = 1/6$, $Re = 700$ [34].

Fig. 6 Computational domain of the bend.
Fig. 7 Comparison of the predicted dimensionless axial velocity profile along the vertical centerline with the analytical solution of Brauner [35], $h_e = D/10$, $\mu_e/\mu_l = 0.01$.

Fig. 8 Comparison of the predicted dimensionless axial velocity profile along the vertical centerline with the experimental results by Laufer [36].
Fig. 9 Comparison of the predicted dimensionless turbulence kinetic energy profile along the vertical centerline with the experimental results by Laufer [36].

Fig. 10 Comparison of the predicted dimensionless turbulence dissipation rate profile along the vertical centerline with the experimental results by Laufer [36].
Fig. 11 Axial velocity contours in the gas phase at $s = 0.03, 0.1, 0.14$ and $0.18$. 
Fig. 12 Secondary flow in the gas phase at $s = 0.03, 0.1, 0.14$ and 0.18.

Fig. 13 Concentration of particles in the bend (curvature radius= 0.3 m)
Fig. 14 Concentration of particles in the bend (curvature radius = 0.25 m)

Fig. 15 Concentration of particles in the bend (curvature radius = 0.1274 m)

Fig. 16 Comparison of motion of particles with different sizes