Elsevier Editorial System(tm) for Journal of Petroleum Science and Engineering
Manuscript Draft

Manuscript Number:

Title: Influence of Inlet Angle on Flow Pattern and Performance of Gas-Liquid Cylindrical Cyclone Separator

Article Type: Full Length Article

Keywords: Separator, cyclone separator, GLCC, turbulence model, multi-phase flow, Computational Fluid Dynamics.

Corresponding Author: Dr. Le Van Sy, Ph.D.

Corresponding Author's Institution: PetroVietnam University

First Author: Le Van Sy, Ph.D.

Order of Authors: Le Van Sy, Ph.D.

Abstract: Gas-Liquid Cylindrical Cyclone Separator (GLCC) is widely used in the petroleum industry with potential field applications. Its performance is strongly influenced by the inlet configuration. The 27-degrees optimal inclined inlet angle has experimentally observed for GLCC with the same diameter of body and inlet. For other GLCCs, the effect of inlet angle on flow pattern and their performance has not investigated. The main target of the current study is to understand deeply the changes of flow pattern with respect to different inclined angles and flow conditions. Twelve GLCCs with different inclined angles were numerically investigated with using the Reynold Stress turbulence model to predict the flow pattern with GLCC. The distribution of radial, axial and tangential velocity profiles and their maximum magnitudes with respect to the change of inlet angle were carefully considered in this study.

Cover

Gas-Liquid Cylindrical Cyclone Separator (GLCC) and compact separation equipment are becoming industry standard with more than 4200 field applications. Performance of separator equipment is strongly influenced by the inlet configuration. The 27-degrees inclination angle of the inlet has been optimized for conditions of equal GLCC and inlet diameters. For GLCCs where the inlet diameter is smaller than the GLCC diameter, the optimum inlet inclination angle might be not equal to 27 degrees. Currently, there are no mechanistic models to predict the effect of different inlet configurations on performance of compact separators. Furthermore, there has been little or no fundamental work published on effect of inlet angle on the flow pattern and performance of GLCC separators. In this paper, a numerical simulation (CFD) in combination with practical experiments have been used as a potential tool which can help to better understand the effect of inclined inlet angle on a complex flow patterns of the GLCC separators in which have small diameter ratio of the body and inlet.

Influence of Inlet Angle on Flow Pattern and Performance of Gas-Liquid Cylindrical Cyclone Separator

Abstract:

Gas-Liquid Cylindrical Cyclone Separator (GLCC) is widely used in the petroleum industry with potential field applications. Its performance is strongly influenced by the inlet configuration. The 27-degrees optimal inclined inlet angle has experimentally observed for GLCC with the same diameter of body and inlet. For other GLCCs, the effect of inlet angle on flow pattern and their performance has not investigated. The main target of the current study is to understand deeply the changes of flow pattern with respect to different inclined angles and flow conditions. Twelve GLCCs with different inclined angles were numerically investigated with using the Reynold Stress turbulence model to predict the flow pattern with GLCC. The distribution of radial, axial and tangential velocity profiles and their maximum magnitudes with respect to the change of inlet angle were carefully considered in this study.

INFLUENCE OF INLET ANGLE ON FLOW PATTERN AND PERFORMANCE OF GAS-LIQUID CYLINDRICAL CYCLONE SEPARATOR

Abstract: Gas-Liquid Cylindrical Cyclone Separator (GLCC) is widely used in the petroleum industry with potential field applications. Its performance is strongly influenced by the inlet configuration. The 27-degrees optimal inclined inlet angle has experimentally observed for GLCC with the same diameter of body and inlet. For other GLCCs, the effect of inlet angle on flow pattern and their performance has not investigated. The main target of the current study is to understand deeply the changes of flow pattern with respect to different inclined angles and flow conditions. Twelve GLCCs with different inclined angles were numerically investigated with using the Reynold Stress turbulence model to predict the flow pattern with GLCC. The distribution of radial, axial and tangential velocity profiles and their maximum magnitudes with respect to the change of inlet angle were carefully considered in this study.

Keywords: Separator, cyclone separator, GLCC, turbulence model, multiphase flow.

1. Introduction

In petroleum production industry, separating the single-phases of gas, oil, and water from multiphase product is an important stage of production process. The vessel-type separators have popularly used for this task which have large size, bulkiness and high cost of purchase and operation. GLCC separator being a potential alternatives for conventional one was patented Chevron and Tulsa University [1]. GLCC is a compact separator with no rotating part which consisted of the vertical cylindrical body welded to a downward inclined tangential inlet and two outlets, one for gas collection at upper part and other for liquid collection at lower part (Fig. 1-a). The multiphase mixture is fed from inclined inlet in tangential direction with GLCC body in which produced a vortex flow. Due to gravitational and centrifugal forces, heavy phases are pushed radially toward the wall of GLCC body, downward and collect at lower outlet while lighter phases are pulled radially forward the center part, upward and collect at upper outlet. The main advantages of GLCC are low operating and manufacturing cost, minor maintenance, and easy installation and operation. Therefore, GLCC separators are becoming industry standard with more than 4800 field applications [3]. Although they have potential applications, complex phenomenon effected on the separating efficiency have not studied completely in the past [1-2,14,15].

For traditional hydrocyclone separators, researches have been investigated the influence of inlet dimensions [3-5] and the inlet angle on their hydrocyclone performance which derived the unconsenting results. Misiulia et al. [6] investigated the effects of inlet angle on the flow pattern and pressure drop by using CFD simulation. The results showed that increasing in the inlet angle decreased the average static pressure and the tangential velocity component of the flow, so the collection efficiency of the separator will be reduced. In addition, the increase of inlet angle also decreases the pressure drop in the cyclone. Qian et al. [7,8] performed the numerical analysis for a hydrocyclone separators with different inlet angles of 0°, 30°, and 45° at the same inlet flow velocity. The results presented that increase of the inlet angle decreased the pressure drop and total separating efficiency of the cyclone increased. The author found a 45°-inlet angle is an optimal angle value for the cyclone. Bernardo et al. [9] also numerically investigated the effect of inlet angles (30°, 45° and 60°) on pressure drop and the separating efficiency of an industrial-sized cyclone. The results showed that increasing in inlet angle decreased the total pressure drop and the separating efficiency for the cyclone with inlet angle of 60°, but separating efficiency increased for the cyclone with smaller inlet angles. Funk [10] performed the experiments on the cyclones which have the square and rectangular inlets inclined the angles of −10°, 0°, and 10°. The author concluded that the performance of cyclone is decreased in the case of square and inclined inlet.

For GLCC separators, Kouba [1] and a researching group at TUSTP performed the experiments on three GLCCs with downward inclined inlet that have equal diameter of body and inlet. Kouba et al. [1] experimented on two GLCCs with the small diameters of 2.54cm and 5.08cm while TUSTP tested on GLCC separator with larger diameter of 7.62cm. Kouba [1] observed experimentally that the optimal inlet angle is 27° which allowed to retards significantly the onset of liquid carry-over (LCO) in comparison with horizontal inlet in a low pressure condition. Currently, this optimal value has been used for the most GLCC designs in defiance of the different diameters between the inlet and GLCC [1-2,11,12,15]. Therefore, the most researches only focused on the effect of inlet geometry [11-13], inlet nozzle design [12-14] and inlet position [11, 12] on operating envelop of GLCC.

In summary, most of researches have been investigated to understand better the effect of inlet area, inlet dimensions, inlet section shape, number of inlets, and inlet position on flow pattern and performance of conventional hydrocyclone separators. A few studies in effect of inlet angle on the performance were performed but the results were still contradictory. For cylindrical cyclone separators, there are no mechanistic models and fundamental work published on the effect of different inlet configurations on a performance of compact separators [2-15]. In this study, the numerical simulation combined to practical experiments was used to investigate the effect of inclined inlet angle to a complex flow behavior of GLCC with small diameter ratio of the body and inlet. To do this task, the single-phase CFD simulations with different turbulence models are firstly performed to compare to the axial and tangential profiles between simulated results and experimental data. Secondly, the best suit turbulence model is chosen for all numerical simulations. The simulated results are compared to the experimental data from TUSTP [12-13]. Finally, the effects of the inclined inlet angles (from 5° to 55°) on the flow pattern at different mass flow rates will be analyzed in details to understand deeply the important aspects of the flow behavior that it is difficult to observe by the practical experiments.

2. Former experimental settings

Experimental case	Fluid type	Mass flow rate (kg/s)	U_{av} (m/s)	Re number
1	Water/glycerin	3.39	0.545	7570
2	Water	0.63	0.102	9285
3	Water	4.54	0.731	66,855

Table 1. Erdal's the experimental parameters [13]

Edral [12] performed the practical experiments at TUSTP to measure locally the single-phase swirling flow inside the GLCC. GLCCs with different diameter of body and inlet is made of clear acrylic with dimensions showed on Fig. 1-b. The inlet configuration is inclined an angle of 27° with respect to the horizontal plane. Local measurement of axial and tangential velocities were measured at 24 different axial locations in the range from a distance of 31.7cm to 89.9cm below the inlet by using a Laser Doppler Velocimeter (LDV). Then, the values of turbulence kinetic energy were approximated by using the formula of the radial velocity fluctuations [5,12]. Two fluids used for the experiments and experimental conditions are shown in Table 1. Before the experiments, air within GLCC is removed by opening a blee valve and thus water is covered whole GLCC body.

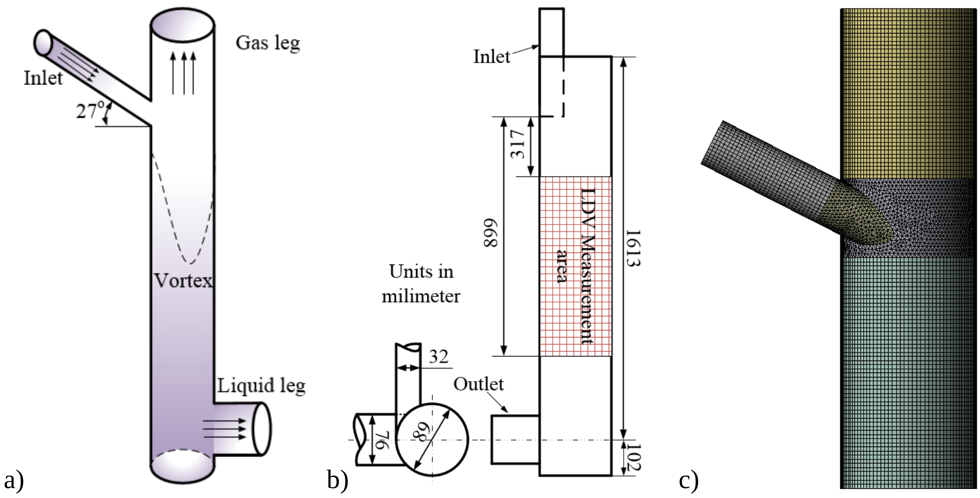


Fig. 1. Geometric dimensions of GLCC, measuring position, and mesh

3. Numerical Simulation of Swirling Flow within cyclone

3.1 Selecting of turbulence model

Selection of a proper turbulence model for CFD simulation of swirl flow inside cyclone separator is a very important task which effects directly on the obtained results. The swirl flow within cyclone separator is almost turbulence, complex and anisotropic behavior [3-10]. Normally, the existing empirical turbulent models were established based on some simplified assumptions which can applied for one or more specific physic behaviors of the flow in industrial applications. Thus, observation of practical experiments and analysis of measured velocity component help to identify better a turbulent model which can predict well the flow pattern for a wide range of operating conditions of cyclone separator.

In the past, there are many efforts to perform CFD simulation for GLCC separator which were reported with contract conclusions for different turbulent models. Erdal et al. [12-13] performed 3D steady-state simulation of GLCC with a high Reynold swirling water flow using both the standard k-ε model and a Reynold Stress Model (RSM). The author reported that both two models captured correctly the swirl flow inside GLCC but predicting the flow behavior of the standard k-ε model is better than one of RSM. Gupta [18] investigated analysis of the tangential velocity component by using PTV and numerical simulation. The author concluded that the RNG k-ε model showed a good agreement between simulated results and experimental data. Rainier et al. [14] used many turbulent models (in two groups of Reynolds Averaged Navier Stoke (RANS) and Large Eddy Simulation (LES)) to simulate the flow in GLCC with different inlet configurations [12]. The result showed that the high-Re realizable k-ε model gave the best prediction of flow behavior in GLCC. LES model also gave a good prediction the velocity profiles of swirl flow. However, LES model required fine mesh, small time step, and very long time to obtain a solution convergence which could not use for optimizing the operating conditions of GLCC.

For simulating hydrocyclone, many efforts have been investigated to predict both axial and tangential velocity profiles using the standard k-ε models [17-21]; modified k-ε models [22], a Differential Reynolds stress (DRS) [23-25]. The results showed that the standard k-ε models could not present a fluctuating motion due to the presence of swirl intensity and insufficient for computing strong swirling flows [24-25]. Some researchers suggested to use the Renormalization Group (RNG) k-ε model which can predict correctly the swirl flow and fairly rotational flow in hydrocyclone separator [31]. However, other researchers noticed that standard and RNG k-ε model were failed to give correctly the effects on turbulence of extra strain and body forces because turbulence in hydrocyclones is anisotropic [22-29]. Slack [25] showed that Reynolds stress model (RSM) presented better the anisotropic turbulence of the fluid flows in hydrocyclone than the conventional k-ε model.

In summary, there are no turbulent model which can predict correctly all hydrodynamic properties of swirl flow in cyclone separators. It is difficult to find a turbulence model which can utilize for all behaviors of swirl flow. Each turbulent model can use well for one or more aspects of swirl flow but it is weak prediction for other applications of the swirl flow. Most of the previous studies on CFD simulation of GLCC separator are restricted to single phase flow with a few simplified assumptions because the complex behavior of swirl flow within GLCC separator. The CFD simulation to predict correctly the swirl flow behavior helping better understanding of its nature are still a challenge for the further studies.

3.2 Model, mesh independence and boundary conditions

The current study is focused on the effects of single inclined inlet angle on the flow pattern of GLCC separators which were modeled with twelve inlet angles from 5º to 55º. The general dimensions of GLCC were presented on the Fig. 1. The mesh for numerical analysis was generated by dividing GLCC body into many dependent geometric blocks. Most of these blocks were identified well to generate the mesh with hexahedral elements by using intelligent tools of ANSYS Meshing 15.0 [30]. The blocks with complex geometries at intersections between inlet/outlet and GLCC body were meshed with tetrahedral elements (Fig 1-c). The boundary layers near the GLCC wall were generated into a structured mesh with the first layer thickness which is calculated correctly to be sure high boundary resolution requirements and the near-wall flow (the value of $y^+ = 1$ for low Reynold solver and y+ value is in the range from 30 to 300 for other cases). Three refinement meshes were generated to test for the grid dependent (number of cells is 950.000, 1.500.000, and 1.650.000, respectively) which all important measures (mesh orthogonality, skewness, aspect ratio… etc) suggested for mesh quality were kept in the best range of high mesh quality [30]. Comparing the results of model with 1.500.000, and 1.650.000 cells presented a small discrepancy.

The velocity inlet condition of the mass flow rates was used for the inlet in a perpendicular direction with inlet section. Two working fluids used for experiments with the properties shown in Table 1. The condition of fully developed flow was applied at both outlets for exiting the gas and liquid. No slip condition was used for the remaining boundaries. Three kinds of turbulence model were evaluated with different options from Ansys Fluent 15.0 [30]. These turbulence models used to simulate the single-phase flow of GLCC separator: i) the k-ε models: the standard, RNG with option accounting swirl effects, and realizable k-ε variants; ii) the RSM model with three options for the pressure-strain term: a linear pressure-strain, a quadratic pressure-strain.

3.3 Numerical scheme

The finite volume method has been carried out to discretize the partial differential equations described the CFD model. For all simulations of this study, pressure-velocity coupling scheme was used with the SIMPLEC algorithm (Semi-Implicit Method Pressure-Linked Equations Consistent). The schemes of pressure interpolation such as standard, linear, body force weighted, second order and PRESTO (Pressure Staggered Option) have been evaluated. The results showed that the second-order interpolation scheme is the best suit for pressure interpolation. Concerning the discretization of turbulent kinetic energy and turbulence dissipation rate, two schemes of QUICK (Quadratic Upstream Interpolation for Convective Kinetics) and first order upwind have also compared. The QUICK scheme derived high accuracy in predicting the swirl flow of GLCC separator.

3.4 Validation of the numerical model

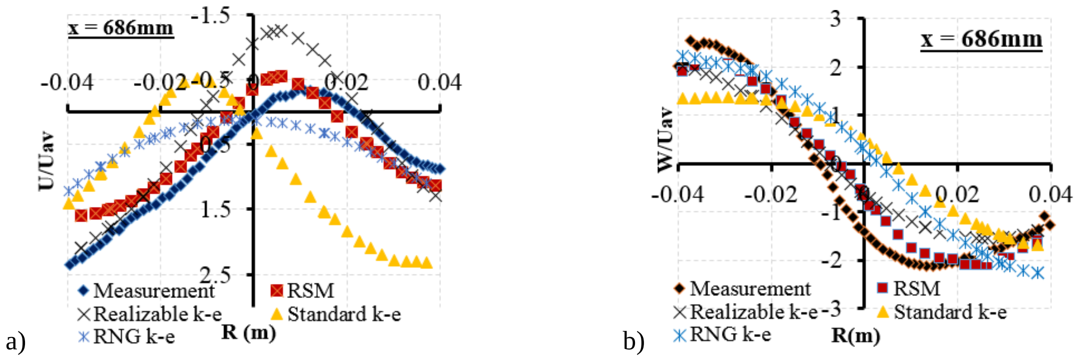


Fig. 2. Axial (a) and tangential (b) velocity profiles

Comparing between the measured results and CFD simulated results is an important step to verify the reliability of CFD simulation. The CFD simulated results with the turbulence models from common standard k-ε model to more complex RSM model used with different options were compared with the results measured directly from LDV by Erdal [12]. The comparison between the predicted and measured tangential and axial velocity profiles at distance of x =869mm were shown in Fig. 2 for high-Reynolds version. They showed that tangential velocity profile derived from RSM model is the best agreement with experimental data with the maximum discrepancy of 4%. The Realizable k– ε model used with the two-layer wall-treatment also agreed very well to experiment. Other turbulence models showed overestimated prediction of the velocity components of swirl flow inside GLCC body. The standard k-ε model, Realizable k-ε model, and RNG k-ε model used without enhanced options have not optimized to predict the strong swirl flow of GLCC separator [5-9, 12-14]. The RSM model and the realizable k–ε model used with the two-layer wall-treatment presented an agreeable prediction of swirl flow pattern. The convergence of the simulations with RSM model become difficult for GLCC models with high inclined inlet angles which need a more investigation of meshing optimization. Finally, the comparison between numerical schemes of pressure interpolation, discretization of turbulent kinetic energy and turbulence dissipation were performed in each turbulence model. The results showed that the QUICK scheme is the best for the discretization of turbulent kinetic energy and turbulence dissipation and a second-order scheme for the pressure interpolation.

4. Results and Discussion

4.1 Flow pattern

The velocity patterns of the flow field within GLCC are characterized by the distribution of velocity field in it. The velocity field is induced from a straight flow positioned tangentially with vertical cylindrical pipe which creates a complex swirl flow inside GLCC body. The flow velocity is resolved into three components following the axes of coordination (*u*, *v*, and *w*) [12]. The velocity components extracted from CFD simulations are compared with the experimental data at the same sections in Erdal's experiments [12]. For each inlet velocities, the velocity components at different sections are showed in Fig. 3-4 which showed a good agreement between experimental and simulation results with using RSM model. This means that RSM turbulence model is the best capacity to predict complex flow behavior which is not only for conventional separators reported by many researchers [14,23-25] but also for cylindrical cyclone separator.

4.2 Axial Velocity

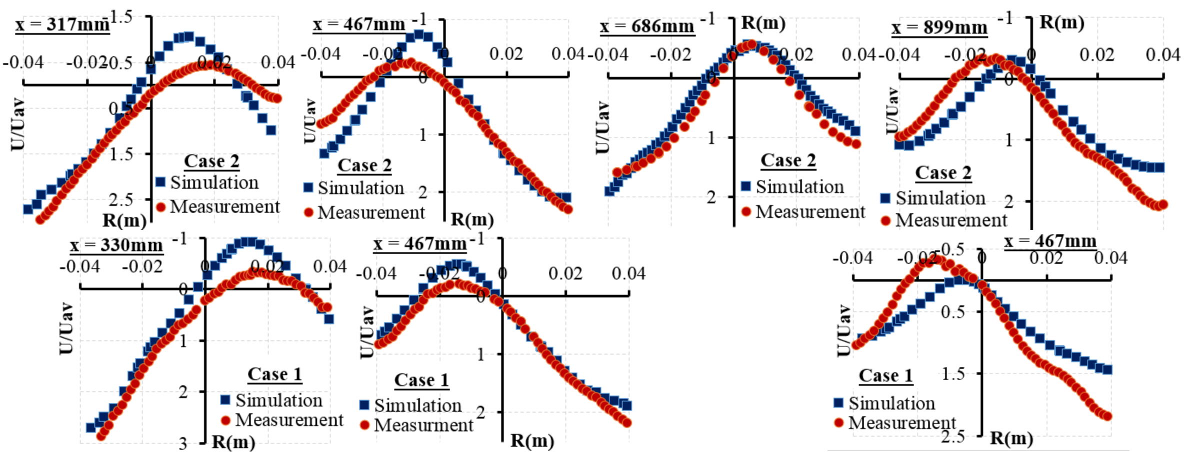


Fig. 3. Axial velocity pattern in Case 1

The profile of axial velocity component is identified in many previous studies which is depended on the flow regimes [3-15]. The axial velocity profile in Case 2 (low Reynold number of 9285) is quite accuracy to the experimental data in comparison with other flow rates (see Fig. 3). A typical profile of the axial velocity is divided into two regions with respect to outer downward flow close to the cyclone wall and inner upward flow towards the cyclone center. The axial velocity profiles at the sections near inlet and outlet section where have a strong swirl flow are slight over-predicted while ones at middle sections are very accuracy. Particularly, the axial velocities at the center region (forced vortex region) of GLCC is higher than other regions (near wall region). The distribution of axial velocity clearly presents the separated regions of axial velocities included low and high positive axial velocity region corresponding downward flow and negative axial velocity region corresponding upward flow. Due to helical flow inside GLCC, the upward flow can located at left side or right side of y-axis depending on the measured plane. The axial velocities near the wall is high and positive magnitude at one side of wall and are still smaller positive magnitude at other side of the wall. The magnitude of both upward and downward flow is decreased when the flow is far from inlet. Thus, the helical pitch (wavelength) of the vortex increases with the increase of axial distance from inlet. Similar observations can be found for high flow rates of Case 1 and 3.

4.3 Tangential Velocity

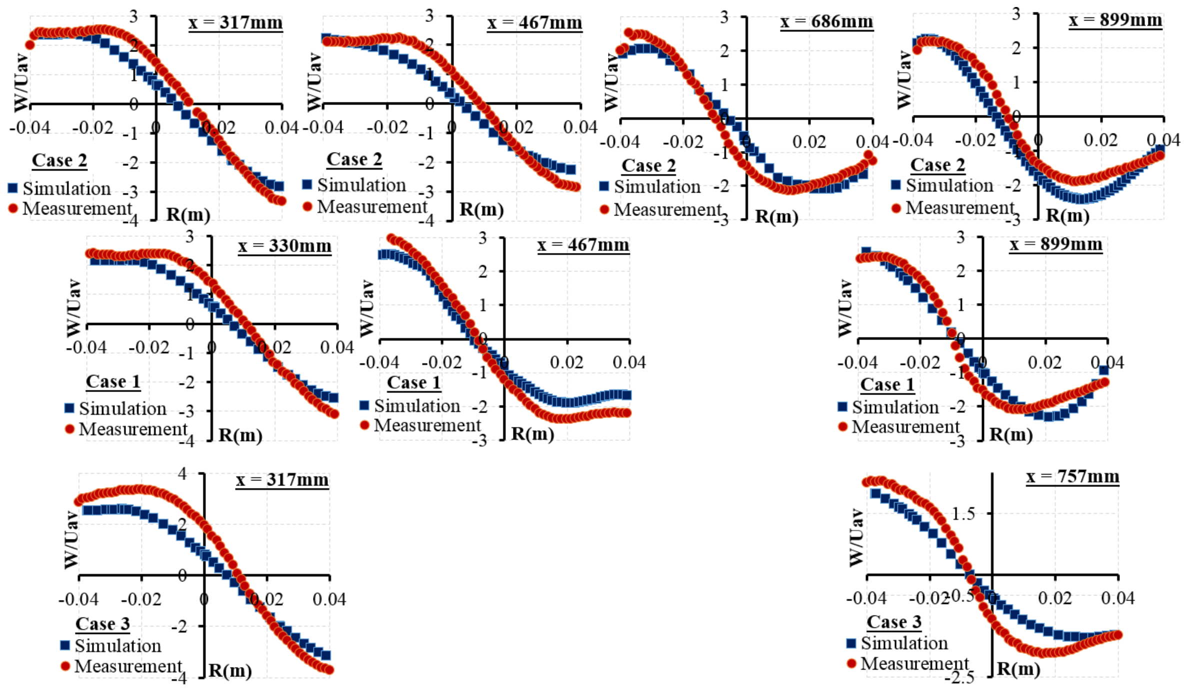


Fig. 4. Tangential velocity pattern

Tangential velocity of cyclone flow is key component which is interested in most of studies about its flow hydrodynamics [17-25]. It affects the flow field that forms a swirling motion and interacts with existing flow in the radial direction producing a centrifugal force [8]. Fig. 4 showed tangential velocity profiles with respect to the different flow rates corresponding to the Reynold numbers from 9285 to 68855. The tangential velocity profiles showed the contrary results with axial velocities. Most predictions is under-predicted at all sections and tangential velocity profiles at middle sections are also the best agreement to the experiments. The tangential velocity profile is divided into two clear regions of positive and negative velocities at each side of x-axis. This behavior is due to viscous flow to be rotated about GLCC axis that showed the in/out direction of the flow in the measured plan. The magnitude of tangential velocities are very high near the GLCC wall but they were dramatically decreased towards the GLCC center. The maximum magnitude of tangential velocities at each measured plans is decayed in the axial direction from inlet.

The tangential velocity profiles in Case 1 and 3 (high Reynold number of 66855) showed that a good agreement between predicted and measured results. A slight discrepancy among the velocity profiles at the sections near inlet and outlet section is due to the effect of Reynold number. The maximum discrepancy is at the section close to the inlet corresponding to high inlet flow rate (Case 3). It can be explained that this region has very strong flow which contributed into the discrepancy. However, these simulated results obtained from using RSM turbulence model are still better than other models of previous studies [5,14].

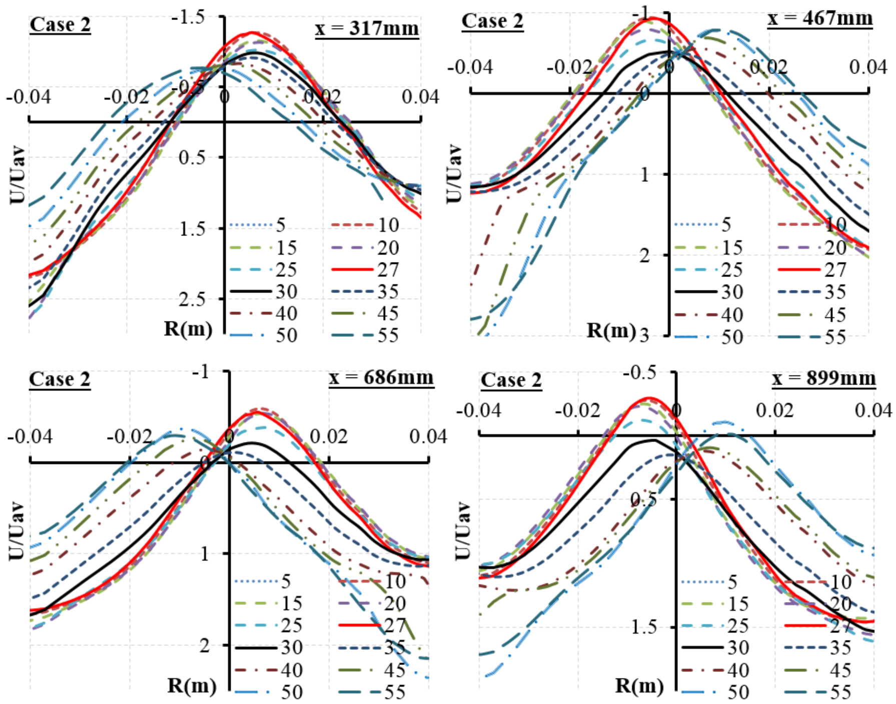
5. Effects of inclined inlet angles on flow pattern

The experimental investigation to understand clearly the complex flow behavior within the GLCC body is an expensive method and much time while CFD simulation with a proper turbulence model and boundary condition showed the potential advantages such as no dedicated measurement, short time and low cost. The CFD simulation is the best way to optimize the operating performance of GLCC before they are fabricated. The above comparisons between simulated result and experimental data are very important which exhibited the appropriate CFD model to be able to predict well the complex flow behavior of GLCC separator. This model can be utilized to study the effect of operating parameters on GLCC performance that have not been investigated by the practical experiments.

In this study, the twelve GLCC models with different inlet angles were simulated with the same meshing properties and boundary conditions as shown in Section 3 to be sure the accuracy of the obtained results. The axial and tangential profiles are extracted from the simulated models at four sections located at x-distances below inlet section (Fig. 5,12). The magnitude of velocity components were scaled with the mean velocities in the GLCC. The model with 27^{o} inclined inlet angle was used to validate the experimental data from Erdal's experiments.

5.1 Axial velocity

The axial velocity is an important component of the flow in GLCC body which presents the movement of fluid flow in axial direction toward the outlet. The distribution of axial velocity profiles with respect to different inlet angles are showed Fig. 5. The axial velocity profile presents two flow streams (upward and downward flow) existing in GLCC body. The upward flow is near the cylindrical centerline directed to inlet while the downward flow near the wall directed to the bottom of GLCC at narrow radial distance. The shape of the axial velocity profiles near the wall are significantly affected by the inlet angle. Larger inlet angle derives higher magnitude of axial velocity, however, this intend only happens at a wall side. This may be explained by movement of swirl flow inside GLCC. The shape of axial velocity profiles also depends on the vortex helical pitch (vortex wavelength) (Fig. 6). The top (negative region) of axial velocity profiles locating at left or right side of GLCC centerline is due to the measured locations and viscous flow conditions which effects to the wavelength of vortex inside GLCC. Fig. 6-a-b showed a good agreement of between axial measured contour from Erdal's experiments (Figure 6-a) and simulated contour (Fig. 6-b) for Case 1 (left side of each Fig.) and Case 3 (right side of each Fig.).



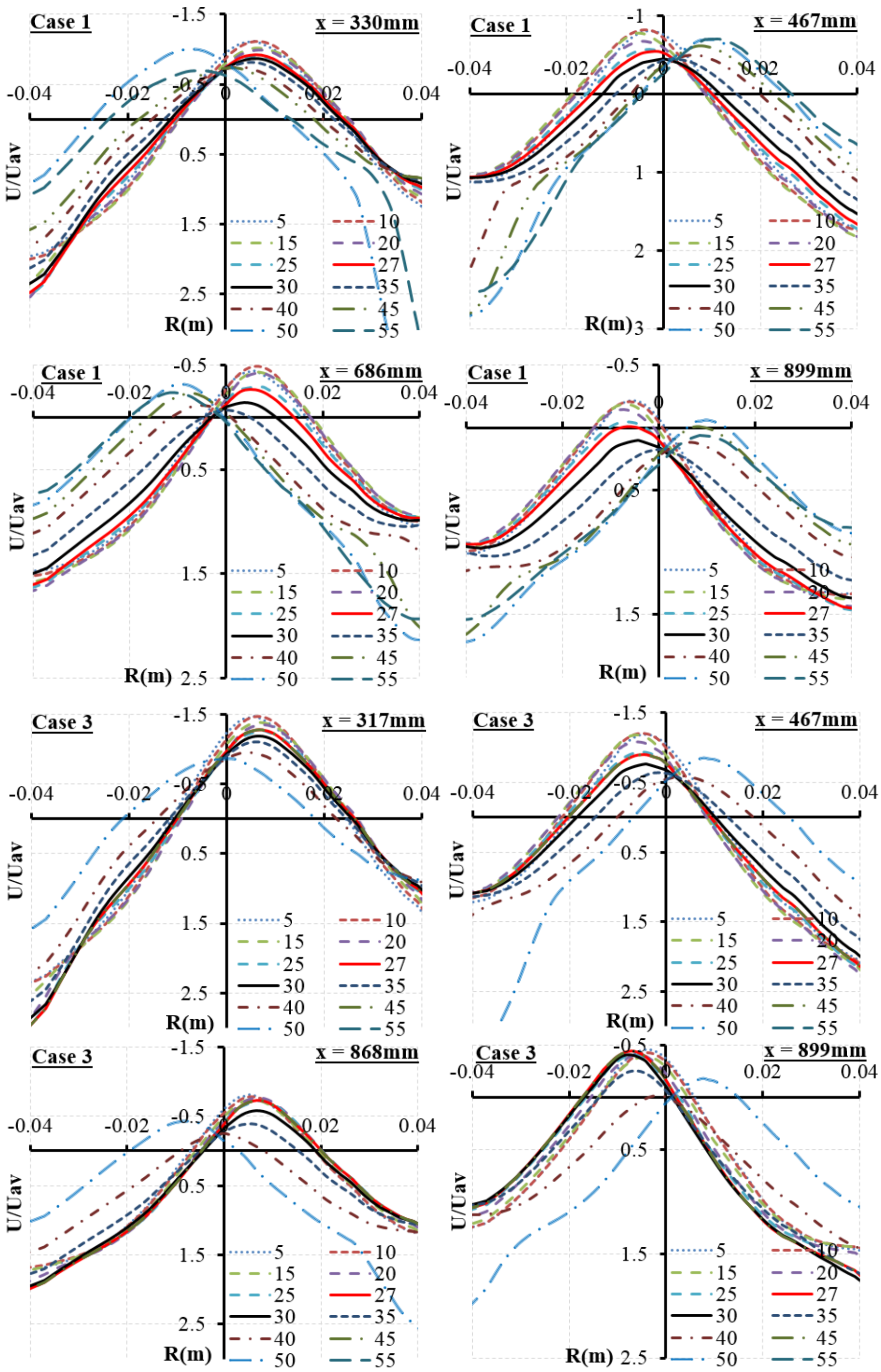


Fig. 5. Axial velocity pattern

Variation of axial velocities is slightly small in the range of inlet angle from 5º to 35º. However, they varied significantly for inlet angles that is larger than 35º. In range of inlet angle from 5º to 35º, the magnitude of axial velocities near wall region is decreased at one side and is increased in the other side while they are only increased in the region near the central axis of GLCC body (Fig. 5). This means that upward flow near GLCC centerline is always increased when inlet inclined angle increases. This trend of axial velocity is kept in the section near the inlet while they are reversed about GLCC centerline on the measured plans toward the bottom of GLCC. The velocity of downward flow near the wall has high positive magnitude and is decayed as the fluid flow moves far from inlet toward the outlet. This decay trigger off an increase of the vortex wavelength in axial direction toward outlet (see Fig. 6). Thus, the vortex of the flow is stretched in this direction which can contribute into the effect of gas-carried under (GCU) phenomenon (gas bubbles move down toward the liquid leg) on the GLCC performance [1,11,15].

The change of maximum axial velocities of upward (lower graph of each Fig.) and downward (upper graph of each Fig.) flow with respect to different inlet angles at four measured plans are showed on the Fig. 7 a-b-c. This graph showed clearly that the maximum axial velocities are very small change in the range of 5-35°. In this range, the slight decreasing trend is found in the maximum axial velocity of both downward and upward flow while opposite trend of the flows is exhibited for larger inlet angles. However, the maximum axial velocities of the downward flow at right below inlet section increases significantly with inlet angle from 5-20° but then they decrease considerably in the range of inlet angle of 45-50°. However, it increases sharply at the inlet angles which are larger than 50°. This may be explained by effect of the flow entering GLCC body interacted to the swirl flow in the vortex region which results in a negative pressure pushed the fluid flow toward the bottom of GLCC. Hence, the axial velocity of downward flow increased in this region near the inlet section. Particularly, the maximum axial velocity of downward flow fluctuated remarkably in the case of large inlet angle and high flow rate (Fig. 7 b-c).

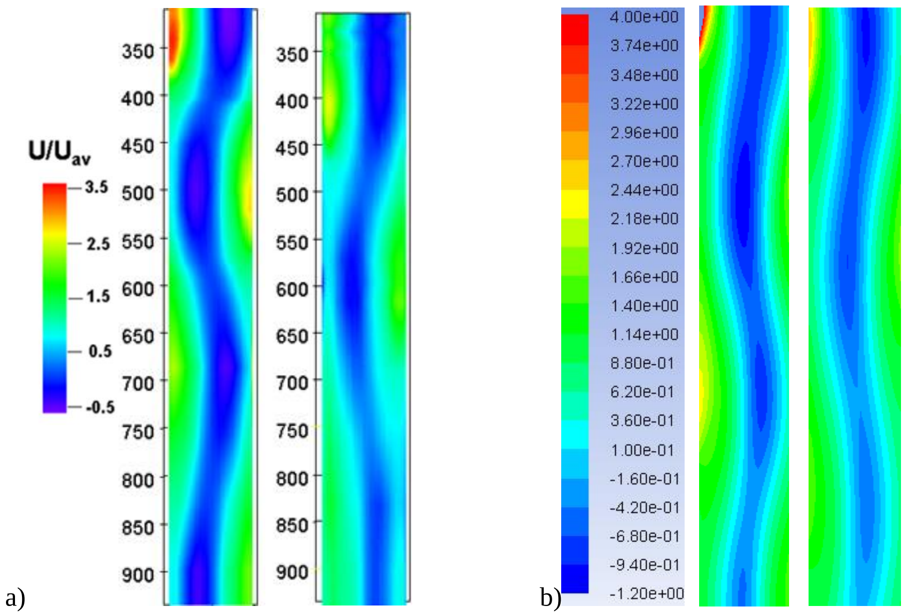


Fig. 6. Comparison of axial contours between the simulated and measured data

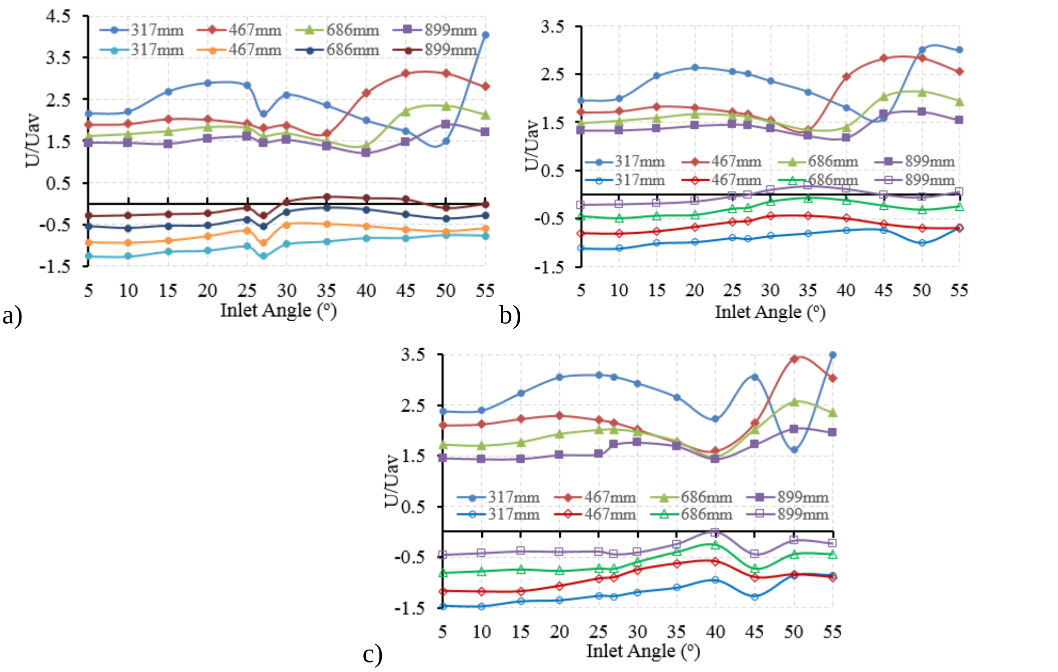
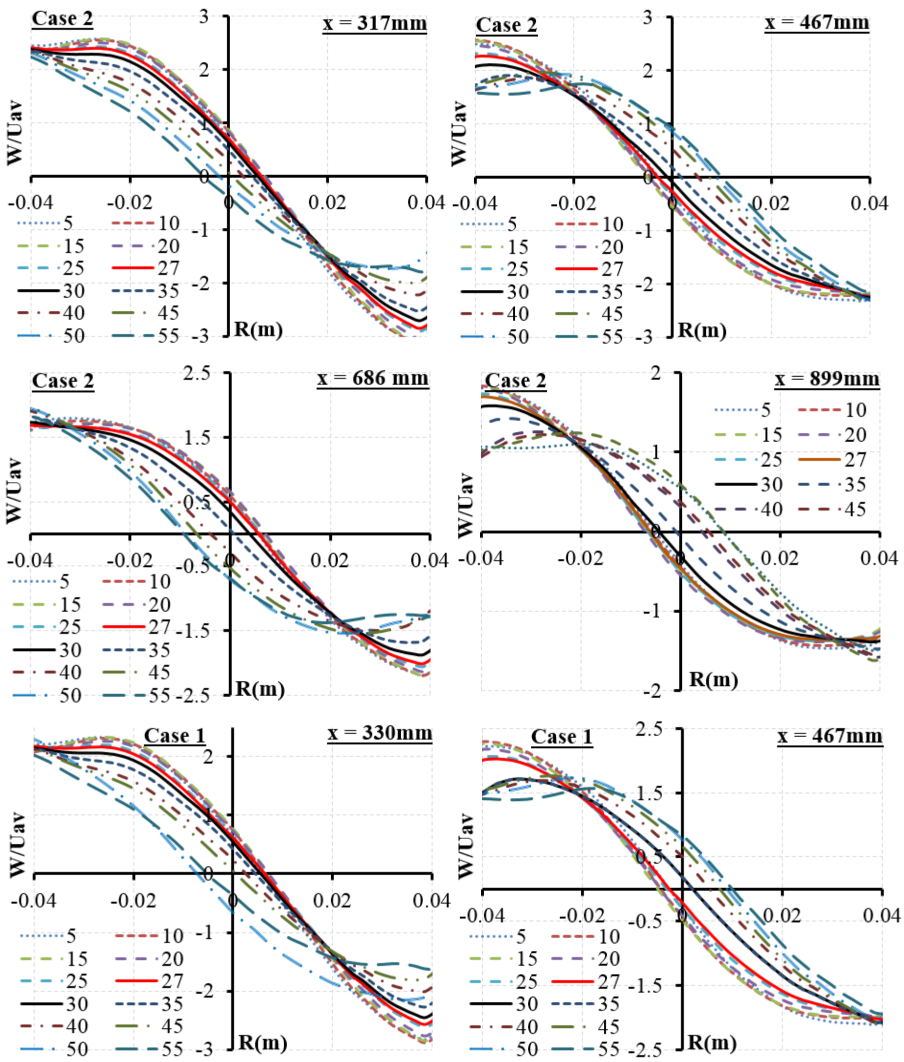


Fig. 7. Distribution of maximum axial velocity in Case 1 (b), Case 2 (a), Case 3 (c)

The maximum axial velocity of upward flow is almost independent on the change of inlet angles. There is a very small change of the maximum axial velocity at the inlet angle which are larger than 35°. This can be explained that same mass flow rate (same mean axial velocity) is provided for the simulations with different inlet angles.

5.2 Tangential velocity

The inclined inlet of GLCC separator is used for enhancing the stratification of multiphase flow before entering the GLCC body in tangential direction. Thus, tangential velocity component has dominant role to create the centrifugal force for separating the phases within GLCC. The tangential inlet of GLCC produces the larger intense of centrifugal force pushed radially the fluid flow to the cylindrical wall which increases the separating performance. The variation of the tangential velocity profiles with various inlet angles are presented in Fig. 8 for flow rate of Case 2-1-3, respectively. Due to the viscous flow is rotated with GLCC body, the distribution of tangential velocity profiles is divided into the positive region on the left side and negative region on the right side. Similar to the case of axial velocities, the magnitude of tangential velocities is significantly decayed when the GLCC flow moves toward the outlet (Fig. 8). The distribution of flow in GLCC also have helical shape under the actions of centrifugal force, thus, the tangential velocities are low in radial region toward the GLCC center but are very high in the region near the wall. The effect of the inlet angle on the tangential velocity is not significant at the small angles (less than 35°) while the significant effect occurred in the inlet angles are larger than 35°. Surprisingly, the variation of tangential velocity with respect to different inlet angles is only very high at a side and does not vary at other side. For small inlet angles, the location of zero tangential velocities will decide the location where has high variation of tangential velocity near the wall via the variation of inlet angle. If the location of zero-tangential velocity being in left side of y-axis, high variation region of tangential angle near the wall falls in the fourth quadrant, and conversely (Fig. 8).



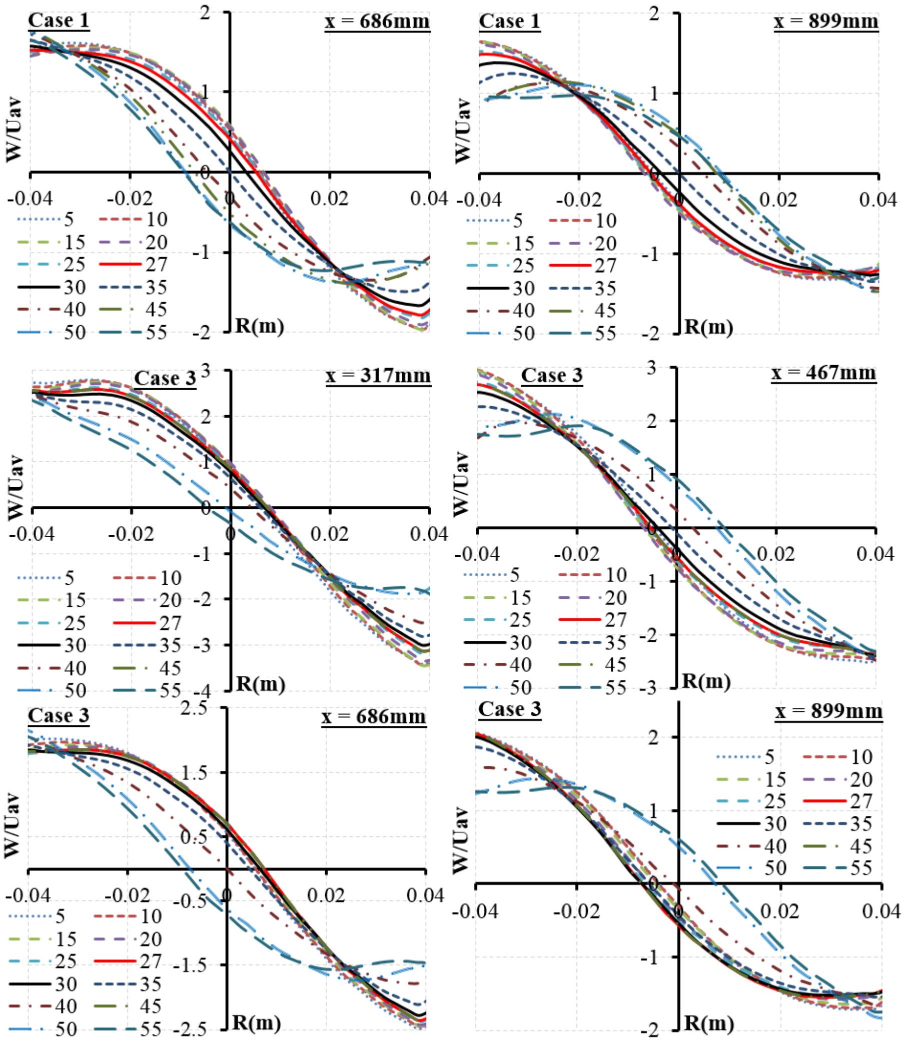
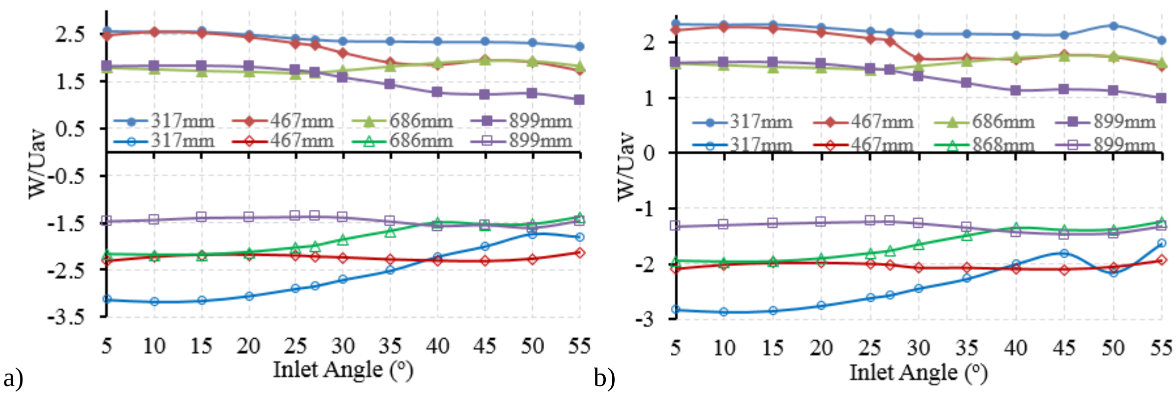
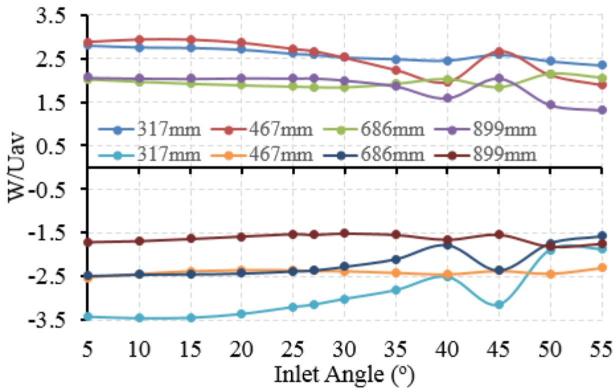


Fig. 8. Tangential velocity pattern in Case 3

The maximum tangential velocity depends on the inlet angle which happens mainly at the wall region. This is one of the most important factors effecting the GLCC performance. In the GLCC with high inlet angle, the tangential velocity near the wall region has the large change. The Fig. 9 showed the effect of inlet angle on the maximum tangential velocity at four measured plan. The (lower) upper graph presented the change of maximum (negative) positive tangential velocity with respect to different inlet angles. The maximum tangential velocity is almost independent on the change of inlet angle at the nearest and the farthest from inlet plan where has less effect of swirl flow. The maximum tangential velocity is increased slightly at the inlet angles which is larger than 35° because the fluid flow is pushed in axial direction and interacts to the upward flow which is decrease the intensity of tangential velocity. The flow is high turbulence at very large inlet angle which is reduce the separating performance of GLCC.





c)

Fig. 9. Distribution of maximum tangential velocity in Case 1 (b), Case 2 (a), Case 3 (c)

5.3 Radial Velocity

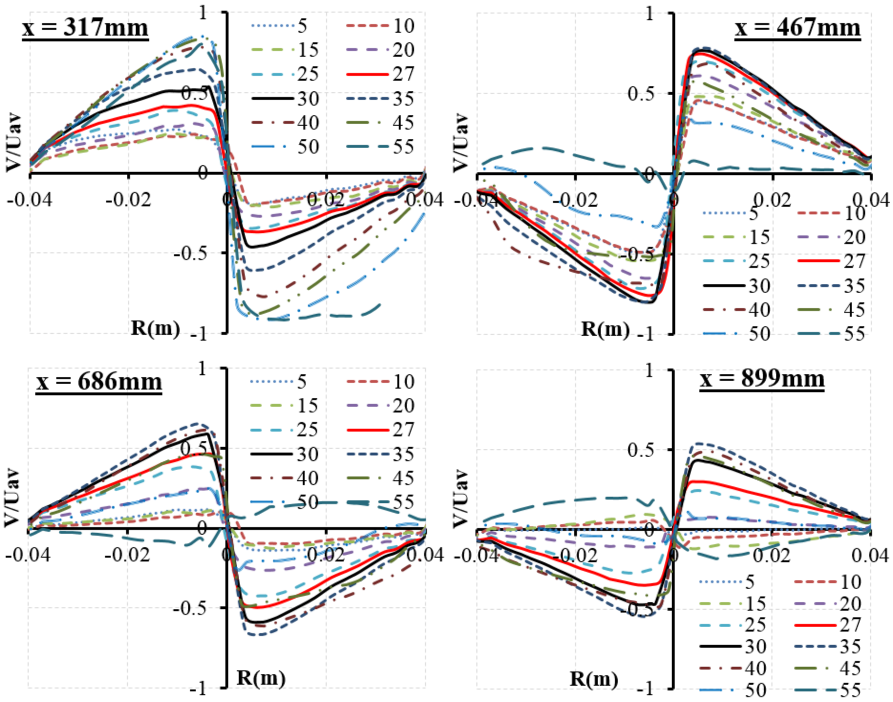


Fig. 10. Radial velocity pattern in Case 2

The radial velocity component has slight effect on the fluid flow bypass which is negligible in the hydrodynamic analysis and development of the mechanistic model [1,11,15]. In the practical experiment, the radial velocity has not directly measured which has usually calculated through the mathematical continuous equation with the assumption of an axisymmetric swirl flow. In this study, an excellent agreement between the simulated and measured results of axial and tangential velocity profiles, thus radial velocity component extracted from the simulation may be a reliable result.

The radial velocity profiles have an axial symmetry and their magnitudes is much smaller than ones of axial and tangential velocity. The radial velocity is close to zero at the wall region, increases quickly toward the GLCC centerline and obtain the peak at the region near center of GLCC (Fig. 10). There is a deeply decrease of radial velocity in narrow distance before zero radial velocity is at GLCC centerline. The radial velocity profile is divided into two regions with opposite sign but the absolute value of radial velocity is slightly equal. It can be explained that the vortex core of swirl flow is eccentrically moving in helical path which created two different pressure regions. The fluid flow intends to move radially toward the centerline due to the vortex core appearing near the wall toward the center where created a low pressure in this region. Similar to axial and tangential velocity, there is a decay of radial velocity toward the bottom of GLCC.

The effect of inlet angle on the radial velocity in the GLCC separator is presented in Fig. 11. The maximum radial velocity increases when the inlet angle increases. In vortex region, there is a slight increase in radial velocity with inlet angle range of 5-30° while significant increase of radial velocity is found for the larger inlet angles. It may be showed that increasing the inlet angle in range of 5-30° can accelerate the bubbles entrained the vortex core that has positive effect on the GLCC performance. However, increasing excessively the inlet angle results the GCU phenomenon which has negative effect on the GLCC performance.

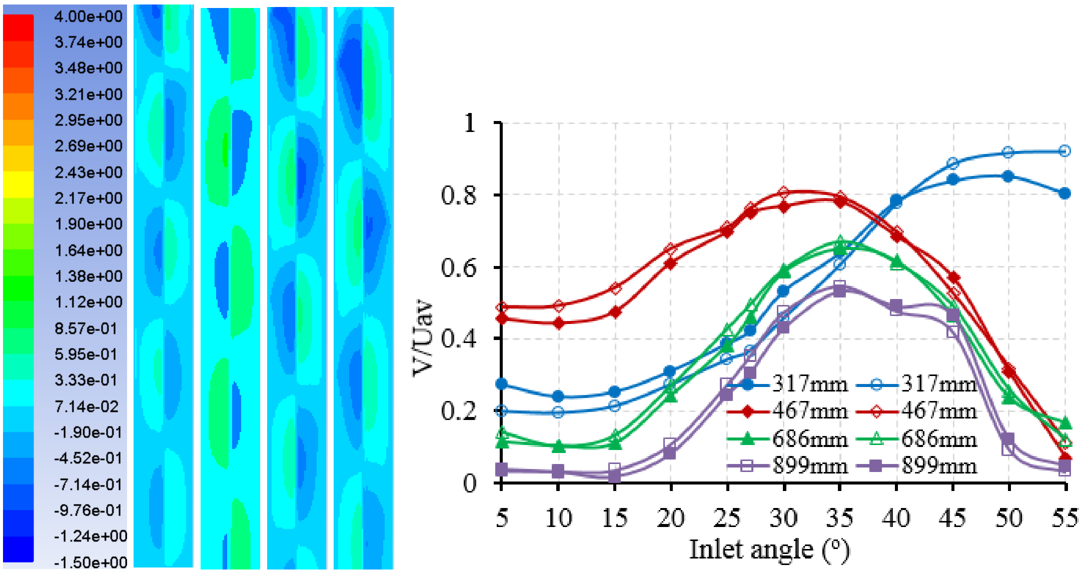


Fig. 11. Radial velocity contours and its maximum magnitude in Case 2

6. Conclusions

In this paper, using a numerical simulation (CFD) combined to practical experiments is a potential tool which can help to better understand the effect of inclined inlet angle on a complex flow patterns of the GLCC separators in which have small diameter ratio of the body and inlet. The following conclusions can be extracted from this study:

- The flow patterns of tangential and axial velocities and their maximum magnitude changed insignificantly when the inclined inlet angles are from 5° to 35° but they increased significantly for the inlet angle being larger than 35°.
- In the range of 5° - 35°, the axial velocities of downward flow are decreased at one side and is increased in the other side while one of upward flow are always when inlet inclined angle increases. The velocities of downward flow were decayed as the flow moves far from inlet toward the outlet. Thus, the vortex of the flow is stretched in this direction which can contribute into the effect of GCU phenomenon on the GLCC performance.
- The maximum axial velocities of both downward and upward flow were slightly decreased in 5° - 35° while they increased for larger inlet angles. The maximum axial velocity of upward flow is almost independent on the change of inlet angle. The maximum tangential velocity depends significantly on the inlet angle which happens mainly in the wall region. The maximum tangential velocity is almost independent on the change of inlet angle at the nearest and the farthest from inlet section.
- The radial velocity profiles have an axial symmetry and their magnitudes is much smaller than ones of axial and tangential velocity. The maximum radial velocity is increased when the inlet angle is increased. However, increasing excessively the inlet angle results the GCU phenomenon which has negative effect on the GLCC performance

7. Acknowledgements

The authors gratefully acknowledge the financial assistance from PetroVietnam University (PVU) in the project No. GV1508.

References

1. Kouba G. E. A, O. Shoham, Review of Gas-Liquid Cylindrical Cyclone Technology. International Conference of Production Separation Systems, Aberdeen, UK, April, 1996.
2. Mohan Ram, Internal report 2013, TUSTP.
3. Hsiao. T.-C, S. Huang, C. Hsu, C. Chen, P. Chang, Effects of the geometric configuration on cyclone performance. Journal of Aerosol Science, 86 (2015), 1–12.
4. Iozia D.L., D. Leith, Effect of cyclone dimensions on gas flow pattern and collection efficiency, Aerosol Science and Technology 10 (3) (1989) 491–500.

5. Elsayed K., C. Lacor, The effect of cyclone inlet dimensions on the flow pattern and performance, Applied Mathematical Modelling, 35 (2011), 1952–1968.
6. Misiulia D., Anders G. G., T. S Lundstrom, Effects of the inlet angle on the flow pattern and pressure drop of a cyclone with helical-roof inlet. Chemical Engineering Research and Design, 102 (2015), 307-321.
7. Qian F., Y. Wu, Effects of the inlet section angle on the separation performance of a cyclone, Chemical Engineering Research and Design, 87-12 (2009), 1567–1572.
8. Qian F., M. Zhang, Effects of the inlet section angle on the flow field of a cyclone, Chemical Engineering & Technology, 30-11 (2007), 1521–4125.
9. Bernardo S., M. Mori, A. Peres, R. Dionisio, 3-D computational fluid dynamics for gas and gas-particle flows in a cyclone with different inlet section angles, Powder Technology, 162-3 (2006), 190–200.
10. Funk, P.A., Hughs, S.E., Holt, G.A.,. Dust cyclone design. Applied Engineering in Agriculture. 17-4 (2001), 441–444.
11. S. Movafaghian, J.A. Jaua-Marturet, R. Mohan, O. Shoham, G. Kouba, The effects of geometry, fluid properties and pressure on the hydrodynamics of gas–liquid cylindrical cyclone separators, International Journal of Multiphase Flow. 26 (6) (2000), 999–1018.
12. Erdal, F., Shirazi, S. Local velocity measurements and computational fluid dynamics (CFD) simulations of swirling flow in a gas–liquid cylindrical cyclone separator, Engineering Technology Conference on Energy, vol. 145 , Houston, Texas, 2011, 23–30.
13. Erdal, F., Shirazi, S., Shoham, O., Kouba, G. CFD simulation of single-phase and two-phase flow in gas–liquid cylindrical cyclone separators. SPE Journal 2, 1997, 436–446.
14. Rainier H, Caroline G, Noël M., Numerical investigation of swirling flow in cylindrical cyclones. Chemical engineering research and design, 89 (2011), 2521–2539.
15. Gomez L. E., Ram S. Mohan, O. Shoham, J. D. Marrelli, G. E. Kouba. State-of-the-Art Simulator for Field Applications of Gas-Liquid Cylindrical Cyclone Separators. SPE Annual Technical Conference and Exhibition, Houston, Texas, Oct-1999.
16. Gupta, A., Kumar, R.,. Three-dimensional turbulent swirling flow in a cylinder: experiments and computations. International Journal of Heat and Fluid Flow 28 (2007), 249–261.
17. Concha, F. (2007) Flow Pattern in Hydrocyclones, Kona-Powder and Particle, 25, pp. 97-132.
18. He, P., Salcudean, M. and Gartshore, I. S. A numerical simulation of hydrocyclones, Chemical Engineering Research & Design, 77, (1999), pp. 429-441.
19. Malhotra, A., Branion, R. M. R. and Hauptman, E. G. Modelling the flow in a hydrocyclone, The Canadian Journal of Chemical Engineering, 72 (1994), pp. 953-960.
20. Dyakowski, T. and Williams, R. A. Modelling turbulent flow within a small diameter hydrocyclone, Chemical Engineering Science, 48, (1993), pp. 1143-1152.
21. Hargreves, J. H. and Silvesters, R. S. (1990) 'Computational fluid dynamics applied to the analysis of deoiling hydrocyclone performance.', Trans. Inst. Chem. Eng, 68, pp.365–383.
22. Dai, G. Q., Li, J. M. and Chen, W. R. Numerical prediction of the liquid flow within a hydrocyclone, Chemical Engineering Journal, 74 (1999), pp. 217-223.
23. Small, D.M., Fitt, A.D. and Thew, M.T. The influence of swirl and turbulence anisotropy on CFD modelling for hydrocyclones. In, Claxton, D., Svarosky, L. and Thew, M. (eds.) Hydrocyclones '96. London, UK, Professional Engineering, (1996), 49-61.
24. Cullivan, J. C., Williams, R. A., Dyakowski, T. and Cross, C. R. 'New understanding of a hydrocyclone flow field and separation mechanism from computational fluid dynamics', Minerals Engineering, 17, (5), (2004) pp. 651-660.
25. Slack, M., Cokljat, D. and Vasquez, S. A. Reynolds-Stress Model for Eulerian Multiphase. Proc 4th Int. Symp on Turbulence Heat and Mass Transfer. Begell House Inc, (2003), pp. 1047–1054.
26. Delgadillo, J. A. and Rajamani, R. K.. Computational fluid dynamics prediction of the air-core in hydrocyclones, International Journal of Computational Fluid Dynamics, 23, (2), (2009), pp. 189-197.
27. Mousavian, S. M. and Najafi, A. F. Numerical simulations of gas-liquid-solid flows in a hydrocyclone separator, Archive of Applied Mechanics,79(5), (2008b), pp.395-409.
28. Narasimha, M., Brennan, M. and Holtham, P. N. Large eddy simulation of hydrocyclone-prediction of air-core diameter and shape, International Journal of Mineral Processing, 80, (1), (2006), pp. 1-14.
29. Pericleous, K. A. and Rhodes, N. The hydrocyclone classifier - A numerical approach, International Journal of Mineral Processing, 17, (1-2), (1986), pp. 23-43.
30. Ansys documentation 15.0, www.ansys.com

***Highlights**

Highlights

Gas-Liquid Cylindrical Cyclone Separator (GLCC) and compact separation equipment are becoming industry standard with more than 4200 field applications. Performance of separator equipment is strongly influenced by the inlet configuration. The 27-degrees inclination angle of the inlet has been optimized for conditions of equal GLCC and inlet diameters. For GLCCs where the inlet diameter is smaller than the GLCC diameter, the optimum inlet inclination angle might be not equal to 27 degrees. Currently, there are no mechanistic models to predict the effect of different inlet configurations on performance of compact separators. Furthermore, there has been little or no fundamental work published on effect of inlet angle on the flow pattern and performance of GLCC separators. In this paper, a numerical simulation (CFD) in combination with practical experiments have been used as a potential tool which can help to better understand the effect of inclined inlet angle on a complex flow patterns of the GLCC separators in which have small diameter ratio of the body and inlet.

Results

- The flow patterns of tangential and axial velocities and their maximum magnitude changed insignificantly when the inclined inlet angles are from 5^o to 35^o but they increased significantly for the inlet angle being larger than 35^o.
- In the range of 5^o - 35^o, the axial velocities of downward flow are decreased on one side and increased on the other side while one of upward flow is always increased when inlet inclined angle increases. The velocities of downward flow were decayed as the flow moves far from inlet towards the outlet. Thus, the vortex of the flow is stretched in this direction which can contribute into the effect of GCU phenomenon on the GLCC performance.
- The maximum axial velocities of both downward and upward flow were slightly decreased in 5^o - 35^o while they increased for larger inlet angles. The maximum axial velocity of upward flow is almost independent on the change of inlet angle. The maximum tangential velocity depends significantly on the inlet angle which happens mainly in the wall region. The maximum tangential velocity is almost independent on the change of inlet angle at the nearest and the farthest from inlet section.
- The radial velocity profiles have an axial symmetry and their magnitudes is much smaller than ones of axial and tangential velocity. The maximum radial velocity is increased when the inlet angle is increased. However, increasing excessively the inlet angle results the GCU phenomenon which has negative effect on the GLCC performance.