Enhancement of a cylindrical separator efficiency by using double vortex generators

Ali M. Jawarneh *, Ahmad Al-Shyyab, Hitham Tlilan, Amer Ababneh

Department of Mechanical Engineering, The Hashemite University, Zarqa 13115, Jordan

1. Introduction

Cylindrical separators for solid–liquid separation are a new increasing technology for the oil and sand industry. The cylindrical separator is an attractive alternative to the conventional vessel-type separator. As a result, it is essential to develop predictive tools for design and to be able to improve the technology of the cylindrical sand–liquid separators.

Separation technology has a dominant role in many process industries. A good example is that of crude and shale oil. There are many reasons for the importance of separation, like the ever-increasing demands on product purity, the gradual reduction in the quality of raw materials and the growing demands for environmental acceptability of waste materials. Separation of oil–sand is vital in oil sands production and processing. Solids may be separated in settling tanks/basins or by mechanical devices. A wide variety of commercial separators are used. Some of the more commonly used types are static inclined screen, vibrating screen, rotary screen, belt press, perforated roll press and screw press. Each of these separator types has one or more disadvantages, such as high initial cost, high operating cost, high maintenance cost or inadequate degree of separation.

Different forms of swirl and vortex technologies have been developed during the last 30 years, their major functions have been found in the vortex combustor, liquid atomizer, vortex flow meter, Hilsch vortex tube and many others. A variety of opinions have developed regarding the application of these technologies, which vary from overwhelming support to reservations of their effectiveness. Design and performance of swirl/vortex devices depend on the understanding of their characteristics, such as the velocity and the pressure distribution, the strength of the centrifugal force, the vortex tube geometry and the size of the solid particles.

The separator is a multivariable device and has been the subject of intensive research particularly in the last 60 years. Published literature in this area is overwhelming and has to some extent been reviewed by Bradley [1] and Svarovsky [2]. Various aspects of the separators have been thoroughly studied by many researchers in an effort to improve or more efficiently control its performance. Some of the work has been related to the influence of design and operating parameters of the separator, some to mathematical modeling and simulation, and others to design modifications. Studies on the turbulent flow field in cyclones are needed for improving the cyclone performance. For experimental studies, Deotte [3] studied the velocity field at low Reynolds number in a small-size...
cylindrical cyclone; Hoekstra et al. [4] measured the flow field in
gas cyclone for turbulent swirling flow; Lu et al. [5] measured
the Reynolds stresses using Laser Doppler Velocimetry (LDV) diag-
nosis in a liquid–liquid hydrocyclone with double cones and cylin-
ders and reported that the turbulence is anisotropic. A 3-d particle
dynamic-analyzer was employed by Su and Mao [6] to measure the
gas–solid two-phase flow in square cyclone separator. The flow
fields had the feature of Rankine eddy in the central part and pseudo-
dee eddy region of weak swirling intensity near the cyclone
wall. The quasi-laminar motion of particles enhanced the turbulent
motion at the corners due to particle–particle/wall collision. Hu et al.
[7] studied the 3-d strongly swirling turbulent flows in a cyclone
separator and measured the velocities using LDV. Their results
showed that the tangential velocity profiles in the separation space
have a typical Rankine-vortex structure and its distribution is
asymmetric in the annular space while the distribution of axial
velocity in the exit tube is entirely different from that in the sepa-
ration space. Jawarneh et al. [8] have generated a swirling flow by
using a single vortex generator and utilizing an expression that
was developed for the pressure drop while Yilmaz et al. [9] pro-
duced a swirling motion by a radial guide vane swirl generator.

The majority of these studies were done only for the separation
space for a single vortex, whereas the flow field in the annular
space and at the exit is negligible. The flow behavior in separator
is quite complex. This complexity of flow processes has led design-
ers to rely on empirical equations for predicting the separator per-
fomance. These empirical relationships are derived from an
alysis of experimental data and include the effect of operational
and geometric variables. Different sets of experimental data lead to
different equations for the same basic parameters. However, these
models suffer from the inherent deficiency as any other empirical
models; these models can only be used within the extremes of the
experimental data from which the model parameters were determined.

Many studies have been done focusing on theoretical studies
the Navier–Stokes equations to compare the flow field in hydrocy-
clones and gave an analytical solution, with overly simplifying
assumptions. In the region near the central axis, the vortex conser-
avation was applied, with inviscid and rotational flow assumptions,
which yields axial and radial velocity distributions. The lack of an
adequate turbulence description led others to be cautious to use
this analytical solution. Hwang et al. [11] reported analytical solu-
tion but due to the approximations introduced, their results are
only valid in delineating the main features of the vortex flow. In
view of this shortcoming, mathematical models based on fluid
mechanics are highly desirable.

The major obstacle in numerical modeling of complex turbulent
swirling flows is the selection of appropriate turbulence closure
model. In simple flow cases the standard $k – \varepsilon$ model performs
well. However, for strongly swirling flows that involves rigorous
streamline bending it fails. The last conclusion is clearly evident
in a variety of studies; see for example the work of Nallasamy
[12] and Weber [13]. There are several previous CFD investigations
of the flow behavior in the cylindrical separator. Erdal et al. [14]
presented CFD simulation utilizing a commercial code called CFX.
They used axis-symmetric assumptions for two-phase flow. The
tangential inclined inlets were simulated by specifying rotational
velocity components, in addition to the axial and radial velocity
component. An expression was developed for an equivalent inlet
tangential velocity for the axisymmetric model and the effects of
the inlet swirl velocity to the axial velocity on the flow behavior
was also carried out. Motta et al. [15] presented a CFD model for
an axis-symmetric flow for rotational two-phase flow in a gas–li-
quid cylindrical separator. Zhou and Soo [16] predicted time-aver-
ged axial and tangential velocities and the pressure distribution
for gas–solid flow in a cyclone separator using the $k – \varepsilon$ model.
Lu et al. [17] simulated turbulent flows in liquid–liquid hydrocy-
clones using the $k – \varepsilon$, renormalization group of $k – \varepsilon$, and the Rey-
nolds stress model. It has been shown by Boysan et al. [18] that
the standard $k – \varepsilon$ model is inadequate to simulate flows with swirl
because it leads to excessive turbulence viscosities and unrealistic
tangential velocities. For strongly swirling flow, renormalization
group (RNG) $k – \varepsilon$ model is adopted in the present numerical
calculation because it provides an option to account for the effect
of swirl on turbulence by modifying the turbulent viscosity.

An absolute majority of the studies have been made on standard
conically shaped hydrocyclones and confined to a single vortex.
Furthermore, the connection between a specific measured flow
field and the separation efficiency of the separators are seldom
made in the literature. An increased knowledge of how a certain
change of the flow field influences the overall separation process
would be of great benefit for the continued development of
separators.

To develop mechanistic models to estimate the efficiency of the
cylindrical separator by using double vortex generators, informa-
tion about details of flow such as velocity and volume fractions dis-
tributions is required. To date, there are no experimental data
available on local measurements of axial and tangential velocities and volume fractions in the double vortex separator. Thus, computational fluid dynamics (CFD) codes allow the simulation of the separator without the expense of experimental setup and measurements.

The majority of previous works have aimed to study the flow of solids in separators by using a single vortex generator, and up to date there is no attempt to study the separator with double vortex generators in order to separate the solids. Therefore, in this paper we discuss the ability of the double vortex cylinder to separate the oil from sand with different diameters and to create a localized residence zone for solid particles at some distance from the end walls of the vortex cylinder using numerical techniques based on $k-\varepsilon$ turbulent and multiphase flows models. The mixture-granular multiphase and RNG-based $k-\varepsilon$ turbulence models are implemented in CFD code developed by Fluent 6.2 [19], which is based on finite volume method.

2. Numerical method

Advances in computational technology in recent years have lead to models based on fundamental fluid flow taking central stage as tools that will enhance the understanding of flow fields within the separator and bring about improvements in the design of these process units. CFD package Fluent 6.2 [19] is one of the efficient means to study the fluid dynamics of many physical systems and it has been used widely in engineering applications. The use of CFD will go some way to alleviate the problems of using empirical engineering models based on correlation formulae established for a limited range of parameters dictated by the experimental data. Since the simulation will involve the combined effect of turbulent and multiphase flows, the mixture-granular multiphase model and RNG-based $k-\varepsilon$ turbulence model are implemented in this study.

2.1. Geometry and materials

The cylindrical separator configuration for the present simulation is shown schematically in Fig. 1, which was described in a previous study by Jawarneh and Vatistas [20]. It has a cylindrical configuration with constant cross-sectional area (the radius $R_{e} = 7$ cm) and a central axis outlet and circumferential inlets. Swirl is imparted to the fluid via two vortex generators shown in Fig. 1. Each one has four perpendicular flow inlets where the compressed flow is induced. A number of openings of a circular cross-section ($d_{in}$) are drilled at a specified angle ($\varphi = 30^\circ$). When the sand and liquid mixture flows through an inclined inlet sections through the swirlers, it is guided to enter the cylindrical separator in the tangential direction so that swirl is formed inside the vortex chamber. The two generators are mounted at the two ends of the vortex chamber. Each generator has eight holes with diameter $d_{m}$ of 1.267 cm and inlet area $A_{1,2}$ of 10 cm$^2$. For the geometry reported here, the chamber length $L$ is equal to 42 cm, the radius of the exit hole $R_{e}$ is 1.75 cm. The mixture flow rate $Q_{in}$ through each generator is 0.014 m$^3$/s. The density $\rho_{f}$ of engine-oil, defined as the primary or continuous phase, is 889 kg/m$^3$ and its viscosity $\mu_{f}$ is 1.06 kg/m.s. The density $\rho_{p}$ of sand particles, defined as the secondary phase, or dispersed phase, is specified 2500 kg/m$^3$. Several particle sizes ($d_{p}$) and their corresponding feed volume fractions ($q_{f}$) are given in Table 1.

The problem is considered to be incompressible, steady, axi-symmetric, turbulent swirling flow. In this case, due to the simplicity of the cylindrical separator geometry and the explicit treatment of the vortex generator, the separator was set up as 2-d axisymmetric model with a transport equation for the swirl, see Fig. 2.

2.2. Governing equations

2.2.1. Mixture model

The mixture model uses a single-fluid approach and allows the phases to be interpenetrating. It solves the continuity equation for the mixture, the momentum equation for the mixture, and the volume fraction equation for the secondary phases, as well as algebraic expressions for the relative velocities.

The continuity equation for the mixture is

$$\nabla \circ (\rho_{m} v_{m}) = 0 $$

where $v_{m}$ is the mass-averaged velocity:

$$v_{m} = \frac{\sum_{k=1}^{n} q_{k} \rho_{k} v_{k}}{\rho_{m}}$$

and $\rho_{m}$ is the mixture density:

$$\rho_{m} = \sum_{k=1}^{n} q_{k} \rho_{k}$$

$q_{k}$ is the volume fraction of phase $k$ and $n$ is the number of phases, in the present case $n$ was set to be 5.

The momentum equation for the mixture can be obtained by summing the individual momentum equations for all phases. It can be expressed as

$$\nabla \circ (\rho_{m} v_{m}) = -\nabla p + \nabla \circ [\mu_{m} (\nabla v_{m} + \nabla v_{m}^{T})] + \rho_{m} \mathbf{g} + \nabla \circ \left( \sum_{k=1}^{n} q_{k} \rho_{k} \mathbf{v}_{k} \right)$$

Fig. 1. Schematic of the vortex chamber, Jawarneh and Vatistas [20].

A.M. Jawarneh et al. / Energy Conversion and Management 50 (2009) 1625–1633
where \( \mu_m \) is the viscosity of the mixture as defined by Batchelor [21],
\[
\mu_m = \mu_f \left( 1 + \frac{5}{2} \chi_p \right) \tag{5}
\]
\( v_{dr,k} \) is the drift velocity for secondary phase \( k \):
\[
v_{dr,k} = v_k - v_m \tag{6}
\]
The relative velocity or slip velocity is defined as the velocity of a solid phase \( p \) relative to the velocity of the fluid phase \( q \):
\[
v_{pq} = v_p - v_q \tag{7}
\]
The mass fraction for any phase \( k \) is defined as
\[
c_k = \frac{\alpha_p \rho_p}{\rho_m} \tag{8}
\]
The drift velocity and the slip velocity are connected by:
\[
v_{dr,p} = v_{pq} - \sum_{k=1}^{n} c_k v_{pk} \tag{9}
\]
Manninen et al. [22] suggested the form of the relative velocity as
\[
v_{pq} = \frac{\tau_p (\rho_p - \rho_m) d_p^2}{\rho_p} a \tag{10}
\]
where \( d_p \) is the diameter of the particles of secondary phase, \( a \) is the secondary-phase particle's acceleration and \( \tau_p \) is the particle relaxation time. The drag function \( f \) is taken from Schiller and Naumann [23]:
\[
f = 0.0183 R_e \tag{11}
\]
The Reynolds number \( R_e \) is defined based on the average axial velocity as
\[
R_e = \frac{4Q_m}{\pi D_s} \tag{12}
\]
and the acceleration \( a \) is of the form
\[
a = g - (v_m \circ \nabla) v_m \tag{13}
\]
From the continuity equation for secondary phase \( p \), the volume fraction equation for secondary phase can be obtained:
\[
\nabla \circ (\alpha_p \rho_p v_m) = - \nabla (\alpha_p \rho_p v_{dr,p}) + \sum_{q=1}^{n} (\alpha_m v_p - \alpha_p v_{pq}) \tag{14}
\]
Granular properties in the mixture model as the collisional viscosity and kinetic viscosity is modeled as given by Gidaspow et al. [24].

2.2.2 RNG \( k - \varepsilon \) model

The RNG-based \( k - \varepsilon \) turbulence model is derived from the instantaneous Navier–Stokes equations, using a mathematical technique called renormalization group (RNG) methods. The analytical derivation results in a model with constants different from those in the standard \( k - \varepsilon \) model, and additional terms and functions appears in the transport equations for \( k \) and \( \varepsilon \), see Choudhury [25]. The RNG \( k - \varepsilon \) model is similar in form to the standard \( k - \varepsilon \) model but it has shown substantial improvements over the standard \( k - \varepsilon \) model where the flow features of strong streamline curvature, vortices, and rotation are included. So, the effect of swirl on turbulence is included in the RNG model, enhancing the accuracy for swirling flows.

Transport equations for the RNG \( k - \varepsilon \) model:
\[
\frac{\partial (\rho_k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \frac{\alpha_{meff} k}{\rho} \right) \frac{\partial k}{\partial x_j} - \rho \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} - \rho \frac{\partial}{\partial x_i} \left( \frac{\alpha_{meff} \varepsilon}{\rho} \right) \tag{15}
\]
\[
\frac{\partial (\rho \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \frac{\alpha_{meff} k}{\rho} \right) \frac{\partial \varepsilon}{\partial x_j} + C_1 \frac{\varepsilon}{k} \left( \rho \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} - C_2 \rho \frac{\partial^2 u}{\partial x_i^2} + R_c \right) \tag{16}
\]
The quantities \( \alpha_k \) and \( \alpha_\varepsilon \) are the inverse effective Prandtl numbers for \( k \) and \( \varepsilon \) and for high Reynolds number their values are \( \alpha_k = \alpha_\varepsilon = 1.393 \). The effective Viscosity \( \mu_{eff} \) is the sum of the laminar \( \mu \) and turbulent viscosities \( \mu_t \) of the mixture:
\[
\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon} \tag{17}
\]
The model constants \( C_1 \), \( C_2 \) and \( C_{\mu} \) are 1.42, 1.68 and 0.0845, respectively. The main difference between the RNG and standard \( k - \varepsilon \) models lies in the additional term in the \( \varepsilon \) equation given by
\[
R_c = \frac{C_{\mu} \rho n \eta (1 - \eta / \eta_0) \beta^2}{1 + \beta^4} \tag{18}
\]
where \( \eta = S k / \beta \), \( \eta_0 = 4.38 \), \( \beta = 0.012 \). The swirl number \( S \) is defined as the ratio of the axial flux of angular momentum to the axial flux of axial momentum:
\[
S = \frac{\int_0^{R_s} \rho V \cdot \omega \cdot r^2 dr}{\int_0^{R_s} \rho V \cdot r^2 dr} \tag{19}
\]
where \( R_s \) is the separator radius, \( V_\theta \) and \( V_z \) are the tangential and axial velocities components.

Turbulence is affected by swirl in the mean flow. The RNG model provides an option to account for the effects of swirl by modifying the turbulent viscosity appropriately. The modification takes the following functional form:
\[
\mu_t = \mu_{inf} \left( \Psi, S, \frac{k^2}{\varepsilon} \right) \tag{20}
\]
where $\mu_0$ is the value of turbulent viscosity calculated without the swirl modification, and $\Psi$ is a swirl constant that assumes different values depending on whether the flow is swirl-dominated or only mildly swirling. For strongly swirling flows as in the present work a value of 0.08 was used.

2.3. Boundary conditions and numerical schemes

At the two inlets a uniform velocity components are used. The same velocity is specified for both the fluid and particulate phases. The total inlet velocity vector $\mathbf{V}_{in}$ at both two inlets has two components $V_{inx}$ and $V_{iny}$, and they are related to each other by

$$V_{in} = q_{in} \cos \phi, V_{in} = q_{in} \sin \phi, q_{in} = \frac{Q_{in}}{A_{in}}$$ (21)

We should refer that the vortex at the right swirler (generator) acts clockwise while the left one acts counter clockwise. At the outlet boundary there is no information about the variables and some assumptions have to be made. The diffusion fluxes in the direction normal to the exit plane are assumed to be zero. The pressure at the exit is neglected since it does not have the space to develop. A grid independent solution study was made by performing the simulations for three different grids consisting of 33,000, 37,000 and 48,000 nodes.

$$\frac{\partial p}{\partial r} = -\rho \mathbf{V}_{avg}^2$$ (22)

At the solids walls, the no-slip condition was applied where the velocities at the walls were specified to be zero. The center-line boundary was considered axis of symmetry. Regarding the inlet boundaries for $k$ and $\varepsilon$, the turbulence intensity, $l$, can be estimated from the following formula derived from an empirical correlation for pipe flows:

$$l = \frac{V'}{V_{avg}} = 0.16(R_e)^{-1/8}$$ (23)

where $V_{avg}$ is the average axial velocity. An approximate relationship between the turbulence length scale, $L_o$, and the physical size of the separator diameter ($D_o = 2R_o$) is

$$L_o = 0.07D_o$$ (24)

The relationship between the turbulent kinetic energy, $k$, and turbulence intensity, $l$, is

$$k = \frac{3}{2} (V_{avg})^2$$ (25)

The turbulence dissipation rate $\varepsilon$ can be determined as

$$\varepsilon = C^\mu \frac{k^{3/2}}{L_o}$$ (26)

In this study, the standard wall functions (Launder and Spalding [26]) are implemented for wall boundaries for $k$ and $\varepsilon$. The boundary condition for $k$ imposed at the wall is

$$\frac{\partial k}{\partial n} = 0$$ (27)

where $n$ is the local coordinate normal to the wall. The turbulence dissipation rate $\varepsilon$ is computed from

$$\varepsilon = \frac{C^{\mu} k^{3/4}}{y^p}$$ (28)

where $\kappa$ is von Kármán constant (= 0.4187), and $y_p$ is distance from point $p$ to the wall.

A phase-coupled SIMPLE algorithm (Vasquez and Ivanov [27]) is adopted. The second order upwind schemes were used for momentum, swirl velocity, turbulence kinetic energy, turbulence dissipation rate, and volume fraction. Convergence was assumed when the residual of the equations dropped more than three orders of magnitude. Triangular mesh elements and an unstructured grid were used for the separator. The mesh is sufficiently refined in order to resolve the expected large flow parameter gradients. The under-relaxation parameters on the velocities were selected 0.3–0.5 for the radial and axial, and 0.9 for the swirl velocity components. Segregated, implicit solver, which is well-suited for the sharp pressure and velocity gradients, has been applied to separator. When using the present turbulence model it is necessary to run the simulation for a significant number of iterations, beyond normal convergence criteria. Experience has shown that typically 6000 iterations needed before the peak tangential velocity in the simulation stabilizes. A grid independent solution study was made by performing the simulations for three different grids consisting of 33,000, 37,000 and 48,000 nodes.

![Fig. 3. Comparison of the flow field at a plane of $z = 300$ mm obtained from the measurements and CFD simulation: (a) mean swirl velocity; (b) mean radial pressure.](image_url)
Fig. 4. Contour of volume fractions: (a) phase-1, liquid-phase; (b) phase-2, $d_p = 250 \mu m$; (c) phase-3, $d_p = 200 \mu m$; (d) phase-4, $d_p = 105 \mu m$; (e) phase-5, $d_p = 100 \mu m$. 
3. Results and discussion

In order to gain confidence and understand the modeling methodology that is required to adequately simulate the separator. The experimental work of Georgantas et al. [28] is used as a case study to validate the modeling approach presented in this paper. We should refer here due to lack of experimental data from literature that using a double vortex separator technology, the work of the last reference which is based on a single vortex separator has been adopted as a benchmarking for the present simulation because simply nothing better is available. The present predicted results have been compared against the published study by Georgantas et al. [28] for the air model. Fig. 3 compares tangential velocity and radial pressure profiles. The result presented in Fig. 3(a) where obtained using the renormalization group of $k - \varepsilon$ model. The tangential velocity increases sharply with radius in the central core region then it decreases. This is typical radial transition between free and forced-vortex regions. The two vortex modes are clearly captured and compares well with the measured values. The mean pressure distribution profile $C_p$ is defined according to the following equation:

$$
C_p = \frac{2(p(r) - p(r = 1))}{\rho V_{in}^2}, \quad \text{where} \quad r = \frac{r}{R_0}
$$

(29)

It is obvious from Fig. 3(b) that the renormalization group of $k - \varepsilon$ model can capture the experimental points. It is evident that the maximum pressure occurs at the side wall of the separator due to the action of the centrifugal force. The pressure decreases at a progressively higher rate as the center of the separator approached.

The most important feature of the cylindrical separator with double vortex generators is its ability to create a localized residence zone for solid particles, as shown in Fig. 4, at some distance from the end walls of the vortex chamber. At that level the suspension process can take place. The balance of forces in the horizontal plane at the level of the particles residence zone should be analyzed in order to predict the separation efficiency. Upon the introduction of the mixture through the two swirlers, the particles are suspended by the action of the axial drag force produced by the right swirl generator while the particles are preventing from leaving the exit port by the action of left swirl generator. So, the two axial drag forces acting in opposite directions at the periphery of the chamber are high enough to confine the particles in a narrow zone where the two vortices merge. The tangential drag force accelerates the particles towards the periphery while the radial drag force pushes the particles toward the center of the chamber. For particle equilibrium in the horizontal plane, the centrifugal force must be balanced by the radial drag force. The separation action of the cylindrical separator is based on the effect of centrifugal force where the necessary vortex action is produced by pumping the fluid tangentially into the vortex generators. The two vortices distribute the solid particles over the radius of the separator. Sand particles of different size were injected through the two inlets of the separator and their volume fractions within the separator were tracked. Results from the present simulation of volume fraction contours for all phases are shown in Fig. 4. The distribution of particle volume fractions are evaluated over the cross-sectional $(r-z)$ plane of the cylindrical separator using the contour maps. In the case of $250 \mu m$ particle, for instance, the portion shown in warm colors such as red and orange seem to indicate considerably higher concentrations compared with those shown in cold colors such as blue and sky blue.\footnote{For interpretation of color in Fig. 4, the reader is referred to the web version of this article.}

Fig. 4a depicts the liquid volume fraction distribution, it was seen that the flow field in the separator consisted of the strong swirling flow of high velocity around the center (forced-vortex region) and weak swirling flow of low velocity near the wall (free-vortex region), and local eddies existed at the corners. The strong swirling flow in the center gave rise to centrifugal force and threw the particles to the outer free-vortex region, where particles were likely to be collected due to the weak swirling intensity, especially the larger particles. Larger particles as shown in Fig. 4b were more possible to be collected at the mid-separator because they exchanged more momentum and energy during the collision and they could not trace the liquid flow as well as smaller particles did due to its larger inertia. So, the solids are to be trapped at the wall. As the particle diameter decreases as seen in Fig. 4c and d, the drag and slip velocity decrease allowing the solids phase to be transported more easily by the continuous phase increasing the
dispersion of solids dimensionless axial location $\frac{Z}{L}$ of 0.56.

To examine the effect of particle diameter on the separation efficiency, strategically two axial locations are considered where significant separation occurs. The first location is chosen at the exit plane and the second location is selected close to the mid-separator at axial location $Z$ of 0.56. It can be seen from Figs. 5 and 6 that the concentration for all particle sizes at the exit plane are zero up to the dimensionless exit radius $r_e$ of 0.2, thus pure liquid-phase will flow through this area, then the concentration linearly increases up to 0.35, 0.4, 0.5, 0.6, and 0.75 for 100 $\mu$m, 150 $\mu$m, 200 $\mu$m, and 250 $\mu$m particle, respectively. Fig. 6 shows that all the solid phases have the same concentration of 0.042 at non-dimensional exit radius of 0.75. Fig. 7 shows volume fractions distributions around the mid-separator at dimensionless axial location $\frac{Z}{L}$ of 0.56. It can be seen from previous figures that the solid volume fraction distributions are low and relatively flat in the central region and rapidly increase near the wall.

4. Conclusion

The numerical methodology described in this paper provides confidence that the double vortex separator technology will be of significant value to people designing and using separators. The mixture-granular multiphase model and RNG-based $k-\varepsilon$ turbulence model are able to predict the flow features inside the separator such as the tangential velocity and the radial pressure profiles. The prediction shows the behavior of the mean tangential velocity distribution where a forced-vortex inside the core and a free-vortex outside the core are existed and agree with the experimental data. In addition, the maximum pressure occurs at the side wall of the separator due to the action of strong centrifugal force and it decreases sharply as the center of the separator approached. The collected sand particles at the mid-separator are predicted using the present models and the best performance is obtained when the bottom to top mixture flow ratio is equal to unity where the strength of both vortices are the same. The sand and liquid mixture flows through an inclined inlet sections. As a result of the tangential inlets, two vortices are formed causing the sand and liquid to separate due to the centrifugal force. The sand moves toward the wall, while the liquid flows to the center and exits from the exit orifice. The larger particles are forced to remain near the periphery of the separator, while the smaller ones are held closer to the central axis. The separation process does not end by the arrival of the particles onto a collecting surface. If the process is to be continuous, the collected particles have to be transported and discharged from the separator.

References
