Numerical simulation of the motion of cellulose fibers in a centrifugal cleaner

Zhijun Hu\textsuperscript{a}, Jiang Lin\textsuperscript{a}, Jessica Wu\textsuperscript{b}

\textit{a)} Key Laboratory of Recycling and Eco-treatment of Waste Biomass of Zhejiang Province, Zhejiang University of Science and Technology; Hangzhou 310023, China
\textit{b)} Department of Chemical Engineering, University of New Brunswick, Fredericton, E3B 5A3, Canada
*Corresponding authors: foxhuz@163.com

ABSTRACT

Centrifugal cleaners are commonly used in the pulp and paper industry to separate contaminates from pulp fibers. To reveal the separation mechanism of cellulose fibers and impurities in a centrifugal cleaner, three-dimensional computational fluid dynamics (CFD) models were established based on experimental analyses with the inlet flow rate and outlet diameter as the variables. The incompressible three-dimensional Navier-Stokes equations were applied to describe the fluid motion. Numerical simulation results showed that an increased inlet flow rate could improve the efficiency by enhancing the centrifugal force on particles. Secondary swirling patterns were predominant for centrifugal cleaners with smaller lower outlets. The crowding effect played an important role in the separation of heavy contaminants, and the separating efficiency was proportional to the inlet flow rate and inverse proportional to the lower outlet diameter.

Keywords: cellulose fibers; centrifugal cleaner; numerical simulation; separation

1. INTRODUCTION

The behavior of long, flexible fibers in suspension plays an important role in many applications, including pulp and paper manufacture, polymer melts, food processing and fiber-reinforced composites.\textsuperscript{1-3} The characteristics of fiber suspension and final products highly depend on the velocity profile, turbulent property of the suspension, the shape and flexibility of the individual fiber as well as the interactions between each fibers. The principal factors, including dynamic properties of these particles, their size and aspect ratio, and the flow velocity, affect the transportation, separation and deposition pattern of fibers. Even though there are a large amount of studies about laminar fiber suspension, the behaviors of turbulent fiber suspension, commonly used in engineering, are still not well-understood because of the complex phenomena of turbulence and the fiber motion.

Fiber suspension flowing through a centrifugal cleaner is widely used in pulping process especially in pulp producing from waste paper. High specific gravity contaminants such as sands and dirt solids can be removed from fiber suspension in the centrifugal cleaner due to the density difference and particle shape. Instead of rotating the equipment as a centrifuge, the centrifugal action in the centrifugal cleaner introduces the feed stream at a relatively high velocity, tangentially into a cylindrical body. This creates a vortex that tends to cause high-density components to move to the wall.\textsuperscript{4}

The lower portion of the cyclone is a convergent cone. Material collected on the wall is discharged from the bottom of the cone. The bulk of the flow forms an inner vortex that is raised to the top of the unit and discharged through a central pipe. It is important to know the fiber orientation distribution and rheological properties in order to understand the characteristics of motion and separation between fibers and contaminants. Since the geometric design and the settings of a cleaner can affect its efficiency directly, it is necessary to explore the characteristics of flow fields in the cleaner. However, due to the inherently opaque property of the pulp suspension, it is difficult to measure the flow field through direct instruments.

Computational fluid dynamics (CFD) is a very useful tool to obtain details from the flow inside a cleaner. Since centrifugal cleaner has been extensively used in various industries, a large amount of experimental and theoretical studies have been reported in the literature. CFD for the simulation of transport, separation and deposition of particles has been developed, assuming particles as spheres can reflect the flow field accurately and give scientific predictions.\textsuperscript{5} Bryant et al.\textsuperscript{6} observed that the vortex could touch the cone wall, and particle re-entrainment could occur in some centrifugal cleaners. Leith and Dirgo,\textsuperscript{7} Bhatia and Cheremisinoff\textsuperscript{8}, as well as Rongbiao et al.\textsuperscript{9} found that the cone opening size had an effect on the separating efficiency. Results by Bohnet,\textsuperscript{10} Michael and Martin\textsuperscript{11} indicated that the grade efficiency was associated with the pressure drop. Other researchers had also developed numerical methods to study both individual fiber dynamics and fiber aggregate suspension rheology.\textsuperscript{12, 13} These simulations could accomplish physical experiments and provide data that was hard to obtain through measurement. The complicated swirling turbulent flow in a cyclone places great demand both on the numerical techniques and the turbulence models applied in the CFD codes. It has been shown that\textsuperscript{14} various flow
uncontrolled imperfections, such as short-circuiting and circulatory eddies could limit the separation performance. Wang et al.\textsuperscript{15} used a more complex turbulence model which allowed asymmetrical flow patterns. This realistic model indicated that the flow in the vortex was not symmetrical with the physical axis of the unit. Raoufi et al.\textsuperscript{16} used CFD to simulate and optimize vortex in conventional cyclones. Zhao et al.\textsuperscript{17} used CFD to characterize two types of cyclones with the conventional single inlet (SI) and spiral double inlets. The standard $k$-$\varepsilon$, RNG $k$-$\varepsilon$ and realizable $k$-$\varepsilon$ models were not optimized for strongly swirling flows found in cyclones. The Reynolds Stress turbulence model yielded an accurate prediction on swirl flow pattern, axial velocity, tangential velocity and pressure drop on cyclone simulation.\textsuperscript{18} A model for particle-level simulation of fiber suspensions was used to simulate the separation of fiber and contaminants in centrifugal cleaner.

There are two main methods describing the motion of fiber in turbulent flow. One is called the Lagrange method, using various forces exerted on fiber to calculate the fiber orientation throughout the entire field by solving dynamic equations of each fiber in a known fluid velocity field.\textsuperscript{19,20} However, the existence of fibers can affect the turbulent flow. The other one is called Direct Numerical Simulation (DNS), which can be used to describe the interaction between fibers and fluid, for example, the Lattice Boltzmann method.\textsuperscript{21} The fiber suspension flowing in the turbulent field has been widely discussed in the literature. Nan et al.\textsuperscript{22} presented a linear viscoelastic sphere-chain model based on the discrete element method. Regarding to the damping behavior of deformed flexible fibers, the results suggested that the damping coefficient of an individual fiber mainly depended on the aspect ratio of the fiber and the viscous bond damping coefficient. Tavakol et al.\textsuperscript{23} analyzed both the dispersion and deposition of fibers in various sizes in a fully developed turbulent pipe, and duct flows for a range of flow Reynolds numbers. Fiber translational and rotational equations of motion were solved using the Lagrangian approach assuming one-way interaction. It was shown that using appropriate stochastic models led to a satisfactory evaluation of dispersion and deposition of fiber in turbulent flows. Marchioli et al.\textsuperscript{24} studied rigid fiber in a channel with turbulent flow, investigating near-wall phenomena, fiber alignment and aggregation. Lindström et al.\textsuperscript{25} developed the model by taking into account the particle inertia and the intermediate to long-range hydrodynamic interactions between the fibers. They derived an approximation of the non-creeping interaction between fiber segments and the surrounding fluid, and took into account the two-way coupling between the particles and the carrying fluid. The simulations successfully reproduced the different regimes of motion for thread-like particles and were subsequently used to study paper forming.\textsuperscript{26}

However, very little information is available on how the centrifugal cleaner size and shape affect the flow characteristics and separating efficiency when other dimensions are fixed. The objective of this research was to establish a three-dimensional model for numerical simulation of the motion of cellulose fibers in a centrifugal cleaner. A steady vortex flow and Lagrangian particle tracking method was used to trace the dispersion of particle and the separation between fibers and heavy contaminants. It was assumed that the particles had no direct impact on the generation or dissipation of turbulence in the continuous phase. A Lagrangian particle tracking method was used to trace the dispersion of particle and its trajectory. The effects of various inlet velocity and outlet diameter on the velocity field, streamline and the separation was analyzed.

2. EXPERIMENTAL

2.1 Mathematic Models

A commercial computational fluid dynamics (CFD) software, Fluent 6.3 (Ansys, Lebanon, USA), was used to solve the flow governing equations. The incompressible three-dimensional Navier-Stokes equations were used to describe the fluid motion. A steady vortex flow was designed in this case to emphasize the effects of dispersion between fiber and heavy contaminants. It is assumed that no direct impact assigned to the particles on the generation or dissipation of turbulence in the continuous phase. A Lagrangian particle tracking method was used to trace the particles on their trajectories. The trajectories of individual particles can be tracked by integrating the force balance equations on the particles. The continuity and steady-state Navier–Stokes equations for viscous incompressible Newtonian fluid are as follows.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

(1)

$$\frac{d\mathbf{u}_p}{dt} = \mathbf{F}_D(u - u_p) + g_x + \mathbf{F}_i$$

(2)

$F_c$ in the special circumstances consists of several other forces, such as quality force, Brown power, Stanford lift, etc. Only Stanford lift is considered in the circumstances, other forces were neglected.

g_x denotes the x component of gravity vector and $F_D(u-u_p)$ represented the drag force per unit particle mass given by

$$F_D = \frac{18 \mu}{\rho_p d_F^2} \frac{C_D \text{Re}_p}{24}$$

(3)

In the above relations $\rho_p$ stood for the fiber density, $\mu$ is the dynamic viscosity, $d_F$ indicates fiber equivalent diameter, and $u_p$ is the fiber velocity. With the Reynolds number and constants $a_1$, $a_2$ and $a_3$, the drag coefficient for spherical particles can be found using the equation below.

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

(4)
2.2 Computational mesh-grid generation

The geometrical dimensions of the centrifugal cleaner were designed for simulation depicted in Fig.1 (a). Structured mesh was pictured in Fig.1 (b). The effect of grid refinement was evaluated in the simulation process. The meshes adopted for the simulation cases were 515,624 tetrahedral elements. No-slip conditions were chosen as wall boundary conditions. The grid nodes near the wall were estimated based on the wall function. Particles were assumed to be elastically reflected by wall.

![Fig. 1 Three-dimensional model and grid representation of the cyclone of a centrifugal cleaner (diameter of 75mm, upper outlet of 16 mm and lower outlet of 4 mm)]

2.3 Initial conditions

RSM turbulence model was designed to describe the liquid flow, and the fibers were traced in the Lagrangian frame. The standard wall function was set for the turbulence calculations. In all these formulations, the pressure-velocity coupling was solved based on the SIMPLEC scheme. The liquid phase was taken as water, with the density known as 1000 kg/m³, viscosity of 0.001 mPa·s, and the inlet velocity varied from 1 to 12 m/s, while the outlet pressure was the atmospheric pressure. Second-phase initial conditions were defined as cylinder-shaped pulp fiber. \( l_{pw} \) and \( d_{pw} \), std for fiber length and diameter respectively, and the fiber shape was substituted by equivalent surface sphere. \( d \) was the radius of spherical, which was shown in the following equation:

\[
d = 0.708 \sqrt{d_{pw}^3 + 2d_{pw}^{5/3}l_{pw}}
\]  

(5)

The distribution of Rosin-Rammler formula was adopted for discrete phase particle size distribution

\[
Y_d = \exp \left[-\left(\frac{d}{d_{50}}\right)^n\right]
\]

(6)

In the formula (6), \( Y_d \) presented the mass fraction of particles, and \( n \) was the distribution index. In the equation, discrete-phase wet density was used as 1150 kg/m³, pulp concentration of 1.0%, the total flow rate of velocity of 0.013 kg/s, the average diameter of 0.346 mm, \( d_{\text{max}} \) of 0.694 mm, \( d_{\text{min}} \) of 0.060 mm, and \( n \) of 3.50.

3. RESULTS AND DISCUSSION

3.1 The effect of lower outlet diameter

All the simulation runs under two conditions with two different lower outlet pipe diameters, 4 mm and 8 mm, at different inlet velocities, varying from 1 to 12 m/s. The lower outlet pipe diameters differed mainly from 2 mm to 10 mm. The particles that reached the outlet were trapped and reported, so that the separation characteristics of the cleaner could be determined. During each run, particles were injected simultaneously. When the inlet velocity was 10.2 m/s, the overflow and underflow velocity distributions of pulp fluid in the axial section near the exit position were shown in Fig. 2 and Fig. 3, with different lower outlet pipe diameter. The color indicated velocity, which meant that the darker the color was the greater the value Z coordinate represented the direction of axial of the cleaner was.

Fig. 2 showed a distinct overflow leading the fiber flow into the upper outlet. Comparing Fig.2 (a) and Fig.2 (b), the velocity was higher for 4 mm lower outlet than 8 mm lower outlet. Fig. 3 showed an umbrella-typed fluid discharge from the inlet of the centrifugal cleaner. The secondary swirling patterns were predominant in the centrifugal cleaner with lower outlet of 4 mm. The similar phenomenon was observed during the actual production operations. The predicted velocity distribution was set to compute the mass fluxes at the outlets. The predicted mass separating rates of fibers and contaminants were presented in Fig. 4 for two different lower outlet diameters. The lower outlet diameter could be varied to control the underflow rate.
3.2 Influence of inlet flow rate on the separating efficiency

Figs. 4 showed the CFD prediction of separating efficiency on the cleaners with different outlet diameters. The CFD predicted the influence of various vortex cleaners on the separating efficiency with an acceptable deviation from Lim et al\textsuperscript{27} experiments.

As shown in Fig. 4, as the inlet flow rate increased, the separating rate increased. A growth in feed flow rate could improve the separating efficiency by enhancing the centrifugal force on heavier particles. Due to the rise of the feed rate, the centrifugal force effect was exaggerated, and the heavy particles were carried to the outer layer and went underflow. However, the separating efficiency for the lower outlet diameter of 4 mm started to decrease when the velocity exceeded 11 m/s, while it continued to increase slightly for the lower outlet of 8 mm.

From the simulation, it was observed that when the lower outlet diameter increased, the separating efficiency of both heavy contaminants and fiber particles dropped. The decrease in efficiency for fiber particles was due to the large ratio flow into underflow. In addition to the fluid split effect, the crowding effect played a role in the separation of heavy contaminants.

Crowding effect referred to the crowding of particles between the negative pressure core and conical wall especially near the lower outlet region. For centrifugal cleaner with lower outlet of 8 mm, because of the rise in the tangential velocities, the flow of heavy contaminants along the conical wall increased. The bulk volume of heavy contaminants in the lower outlet region became greater than the capacity of the lower outlet pipe, then a portion of heavy contaminants were carried upward to the overflow. It could be concluded that with increase of flow rate and decrease in lower outlet diameter the separation rate of centrifugal cleaner was improved, but more heavy contaminants would flowed upward to the overflow.
3.3 Velocity field along the center-plane

The velocity field in a cleaner had three components: tangential, axial and radial. Since the flow was strongly swirling, the tangential velocity component was more important than the axial and radial components. Fig. 5 presented a different velocity field, especially in the upper part of a cleaner. The velocity distributions were displayed along the center-plane which bisected the centrifugal cleaner vertically across its inlet.

As shown in Fig. 5 (a) the tangential velocity profile for the centrifugal cleaner could be seen as two regions with an outer region of quasi-free vortex flow surrounding an inner region of quasi-forced vortex flow. The velocity vector plot displayed some minor flow patterns in the flow field of centrifugal cleaner. The fluid was flowing downward along the wall of centrifugal cleaner, and the flow direction reversed along the lower part of the conical section. Thus a refuence zone existed beneath the inlet region was obviously in the velocity vector plot. The upward fluid flow was moving quickly near the central negative pressure core as seen in Fig. 5 (b). The contour graph for the tangential velocity component showed high swirl grades both at the outer and inner rims of the centrifugal cleaner.

3.4 Analysis of separation mechanism of centrifugal cleaner

The flow pattern in centrifugal cleaner could roughly be visualized as sketched in Fig. 6. Thus, the flow was often viewed as consisting of an outer downward-spiraling vortex and an inner upward-spiraling. The heavy particles were separated from the fluid by centrifugal force to the wall.

Along the wall they were transported to the lower outlet by gravity and the downwardly directed flow. The separating mechanism of fiber could be interpreted by a two-step escape. In the first escape rate, as the experiment results of fluid flow field in centrifugal cleaner showed, in the range of 0.25 D below the upper outlet, the short-cut flow was monitored. The ratio between the particles short-cut flow mass flow rate and the inlet particles mass flow rate was defined as the first escape rate. It could be used to evaluate the influence of the upper outlet depth on particles collection efficiency.

In the second escape rate, on the circular surface of the inner swirl region upon the vortex upper outlet, the upward axial velocity and the concentration of the particles were the escape particles mass flow rate caused by the inner swirl and refuence. The ratio of the mass flow rate to the inlet particles mass flow rate was defined as the second escape rate. For the first escape rate, some pulp particles entered into short-circuit from the rotating outer region to the inner core directly and exited through the cleaner outlet. Very little or none separation occurred with this component. For the second escape rate, large eddies swept fluid from close to the wall back into the bulk region, so that the good fibers could be separated with contaminants effectively. The above separation principles could be used to estimate the changing of separating efficiency of centrifugal cleaner caused by the inner swirl entrainment. The difference between the second escape rate and the refuence rate could be applied to measure the secondary separation ability of the inner swirl in centrifugal cleaner.

4. CONCLUSIONS

A steady vortex flow was applied in this case to allow the simulation focusing on the separation effects between fibers and heavy contaminants. Fiber flexibility, fiber–fiber interactions and the two-way coupling between phases were neglected. The particles were assumed with no direct
impact on the turbulence in the continuous phase. RSM
turbulence model, sphere particles model and Lagrangian
frame were used to describe the liquid flow and the fibers.
The underflow rate was adjusted by the lower outlet
diameter. The predicted flow characteristics of the outlet
section for various inlet velocities and outlet diameters were
in good agreement with experimental data. It was shown
that most of the heavy particles were gathering to the
underflow, and light particles flowed to the overflow.

The results showed that an increase in inlet flow rate
could improve the efficiency by enhancing the centrifugal
force on particles. When lower outlet diameter was enlarged
to a certain portion, the inflow fluid would carry more
fibers. The figures showed that the secondary swirling
patterns were predominant in the case of centrifugal cleaner
with smaller lower outlet. The crowding effect played a big
role in the separation of heavy contaminants, and the
separating performance of a centrifugal cleaner.

CFD technology had a great potential in understanding
the fluid flow behavior in a centrifugal cleaner. The
proposed models predicted well the relationships between
the structure design and the operating parameters and the
separating performance of a centrifugal cleaner.

ACKNOWLEDGMENTS

The authors wish to express their thanks to the China
Natural Science Foundation under Grant No.51376162, Key
Laboratory of Recycling and Eco-treatment of Waste
Biomass of Zhejiang Province under Grant No.2016REWB30.

REFERENCES

1. Xi Z. P., Zhu J. L., Tang H. P., et al. Progress of application
researches of porous fiber metals. Materials, 2011, 4(4):
816-824.
2. Londono-Hurtado A., Osswald T. A., Hernandez-Ortiz J. P.,
Modeling the behavior of fiber suspensions in the molding
of polymer composites. Journal of Reinforced Plastics &
Composites, 2011, 30: 781-790.
3. Jarabo R., Fuente E., Moral A., et al. Effect of sepiolite on
the flocculation of suspensions of fibre-reinforced cement.
Cement and Concrete Research, 2010, 40: 1524-1530.
4. Geoff Covey. Separation performance of centrifugal cleaners.
Appta Journal, 2009, 62(1): 31-36.
5. Griffiths W. D., Boysan F. Computational fluid
dynamics(CFD) and empirical modelling of the performance
of a number of cyclone samplers. Journal of Aerosol Science,
1996, 27: 281-304.
6. H.S. Bryant, R.W. Silverman, F. A. Zenz, How dust in gas
affects cyclone pressure drop, Hyrocarb. Process. 1983, 62 :
87-90.
7. J. Dirgo, D. Leith, Performance of theoretically optimized
cyclones, Filtr. Sep. 1985, 22: 199-225.
8. M.U. Bhatta, P.N. Cheremisinoff, Cyclones, in: P.N.
Cheremisinoff (Ed.), Air Pollution Control and Design for
Industry, Marcel Dekker, New York, 1993.
9. Rongbiao S.H., Xiang K.W., Park K.W., et al. Effects of cone
dimension on cyclone performance, Aerosol Sci. 2001, 32 :
549-561.
10. Bohnet M. Cyclone separation for fine particles and difficult
operating conditions, KONA Powder Part. 1994, 12: 69-76.
11. Michael M., Martin W. Pressure loss and separation
characteristics calculation of a uniflow cyclone with a CFD
method, Chem. Eng. Technol. 2000, 23: 753-758.
12. Joungh. C. G., Phan-Thien N., Fan X. J., Direct simulation of
flexible fibers. J. Non-Newton. Journal of Fluid Mechanics,
2001, 99 (1): 1-36.
13. Petrie C. J. S. The rheology of fibre suspensions. J.
Non-Newton. Journal of Fluid Mechanics, 1999, 87 (2):
369-402.
14. Ohtake, T., Usuda, M., Kadoya, T. A fundamental study of
hydrocyclones Part 1. Flow pattern in the hydrocyclone,
Japan Tappi J. 1987,41(2): 60.
15. Wang, B; Xu, D.L.;Xiao, G.X. and Yu, A.B. Numerical study
of gas-solid flow in a cyclone separator Proc. 3rd Intl. Conf. on CFD in Minerals and
Process Industries, Melbourne, 2003.
16. Raoufi A., Shams M., Farzaneh M., et al. Numerical
simulation and optimization of fluid flow in cyclone vortex
finder. Chemical Engineering & Processing, 2008,
47(1):128-137.
17. Zhao B., Su Y., Zhang J.. Simulation of gas flow pattern and
separation. Chemical Engineering Research & Design, 2006,
84(12):1158-1165.
18. Sommerfeld M., Ho C. H. Numerical calculation of particle
transport in turbulent wall bounded flows. Powder Technology, 2003, 131:1-6.
19. Lin, J. Z., Li, J., Zhang, W .F. The forces exerted on a
cylindrical particle in the elongational shear flows. Int. J.
Nonlinear Science and Numerical Simulation, 2004, 5(1):
9-16.
20. You Z. J., Lin J. Z., Yu, Z. S. Hydrodynamic instability of
fiber suspensions in channel flows. Fluid Dynamics Research, 2004, 34(4): 251-271
21. Qi, D. W. A new method for direct simulations of flexible
filament suspensions in non-zero Reynolds number flows, Int.
J. for Numerical Methods in Fluids, 2007, 54 (1):103-118
22. Wenguang Nan, YueShe Wang, Huiping Tang. A viscoelastic
model for flexible fibers with material damping. Powder
Technology, 2015, 276: 175-182
23. Tavakol M. M., Abouali O., Yaghoubi M., et al. Stochastic
dispersion of ellipsoidal fibers in various turbulent fields.
Journal of Aerosol Science, 2015, 80: 27-44.
24. Marchioli, C., Fantoni, M., Soldati, A. Orientation,
distribution, and deposition of elongated, inertial fibers in
turbulent channel flow. Physics of Fluids. 2010, 22(3):89-106
25. Lindström, S. B., Uesaka, T. Particle-level simulation of the
motion of flexible fibers in viscous fluid flow. Physics of Fluids. 2007, 19(11): 733-742
26. Lindström, S. B., Uesaka, T. Particle-level simulation of
forming of the fiber network in papermaking. International
Journal of Engineering Science., 2007, 46(9): 858-876
27. Lim K.S., Kim H .S, Lee K.W. Characteristics of the
collection efficiency for a cyclone with different vortex
finder shapes, Aerosol Sci. 2004, 35: 743–754.
28. Hu L.Y., Shi M. X. Three-dimensional time-averaged flow

www.Bioresources-Bioproducts.com

Peer-Reviewed

125
structure in cyclone separator with volute inlet. Journal of Chemical Industry and Engineering, 2003, 54(4): 549-556.