An Analysis on the Velocity Field of Decanter Centrifuge on the Basis of Fluent

Qiyun Cheng, Hongbin Liu, Yuxin Tian

Abstract—**Because of the complex internal structure and the high-speed operation inside, the real-time monitoring of decanter centrifuge's internal flow field is impossible. The natures of internal flow field directly effects the production capability, separation and operating life of centrifuge. The paper is finished upon the basis of high-speed decanter centrifuge LW355-1235. We use Fluent, the large-scale commercial fluid computation software, to obtain the distribution of velocity field by simulating the internal flow field with the use of RNG turbulence model under different circumstances. The whole process is carried out with existence of revolving speed discrepancy. The result shows that every velocity component of fluid is getting bigger when we increase the revolving speed of drum. Besides, the gap between the simulation value and theoretical value is getting wider, and the axial velocity turns to be slower. For fluid phases with different concentrations, the declining rates of axial velocity are tend to be the same. Besides, the separation is getting worse when the speed discrepancy gets smaller. It also provides basis for the further analysis of high-speed decanter centrifuge.**

Index Terms—**decanter centrifuge, analysis of velocity field.**

I. INTRODUCTION

Horizontal screw discharging centrifuge(called decanter centrifuge for short) is a fast-revolving centrifuge which is able to absorb materials continuously, separate materials and classify them, as well as unload finished products with spiral pusher[1]. It is generally used in after-treatment stage of material separation, especially in solid-liquid separation. Emphasis has been laid upon the analysis of centrifuge all the time, including the study of centrifuge's internal flow field, the optimization of the operating condition and the improvement of structure. Shengfei Zheng [2] used the model RSM and DPM of Fluent to analyze the distribution of both pressure and velocity of decanter centrifuge's three-dimensional flow field. Zhu Guorui[3] simulated the centrifuge flow field M-2301 with Fluent and obtained its solid phase distribution. Yu Ping and Liu Jingguang[4] analyzed the relation between the speed of centrifuge separation field and structural property parameters with Model DPM of Fluent. Fu Shuangcheng and Dong Lianzhu[5] analyzed the velocity field and internal pressure field of decanter centrifuge with model RNG K-ε of Fluent. The multiphase flow model Euler is used in this study. The author used the turbulence model RNG K-ε in the multi-reference frame(MRF) to analyze how solid particles precipitate with the influence of internal velocity field in decanter centrifuge, thus providing important reference for improving the structural design of decanter centrifuge and increasing the separation efficiency.

II. THE SELECTION AND INTRODUCTION OF MODEL RNG K-ε

The internal flow field of decanter centrifuge, which belongs to turbulence model, is a complex three-dimensional irregular flow of unsteady state and rotation. So, the simulation is set on the basis of turbulence flow. The physics parameters of the fluid in turbulence model, like speed, pressure, temperature, and time are altered randomly over time and space. With all factors to be considered, the model RNG k-ε is chosen for the simulation.

Model RNG k-ε is based on the strict statistics technology. It is similar to the standard model k-ε, but the following improvements has been made: A condition is added in formula ε of model RNG, which takes the turbulence eddy into consideration, thus effectively improving its accuracy. Theory RNG provides a analytic formula for turbulence Prandtl number, but the standard model k-ε uses the constant provided by users. Standard model k-ε is a model of high reynolds number, while theory RNG provides a analytic formula with the low reynolds number flow viscidity taken into account. The effectiveness of those formulas depends on whether the near wall area is correctly treated. Those properties make the model RNG k-ε more convincing and accurate than model k-ε in wider range of flow.

III. THE GOVERNING EQUATION OF FLUID MECHANICS[6]

The cornerstone of computational fluid mechanics is composed of governing equation, continuity equation, momentum equation and energy equation. Those equations are the mathematical description of the 3 basic physics disciplines that all the fluid mechanics must follow.

(1) Conservation of mass;
(2) F=ma(the second discipline of Newton);
(3) Conservation of energy.

Those disciplines can be described with Navier-Stokes[7] equation within the Newtonian fluid(In CFD, the continuity equation, momentum equation and energy equation are generally called Navier-Stokes equation).Conservation of energy is not involved in calculation this time, so it will not be discussed.

Mass conservation equation：

$$\frac{\partial \rho_f}{\partial t} + \nabla \cdot (\rho_f v) = 0$$

momentum conservation equation:

$$\frac{\partial \rho_f v}{\partial t} + \nabla \cdot (\rho_f vv - \tau_f) = f_f$$

In the equation, t stands for time, f_f for volume force vector, ρ_f for fluid density, v for fluid velocity vector, τ_f for shear tensor. It can be presented as:

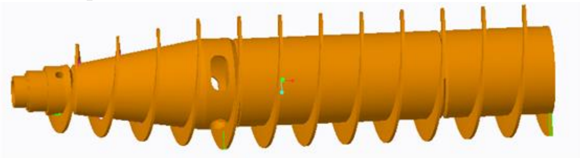
$$\tau_f = (-p + \mu \nabla \cdot v)I + 2\mu e$$

In the equation：p stands for fluid pressure, μ for dynamic viscosity, e for velocity tensor of stress.

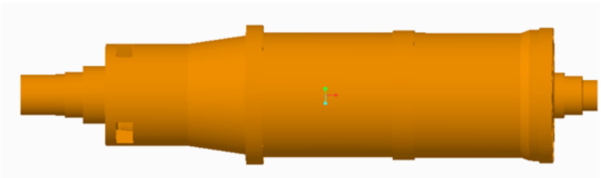
IV. Introduction of Model and Boundary Conditions

Introduction of model

With software UG 8.0, a 3D modeling is made for high-speed decanter centrifuge in its real size. The spiral auger conveyor and drum assembly of decanter centrifuge are showed in picture 1 and 2.

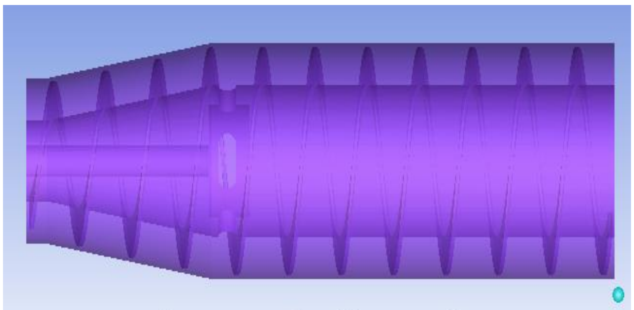


Picture 1 Spiral auger conveyor

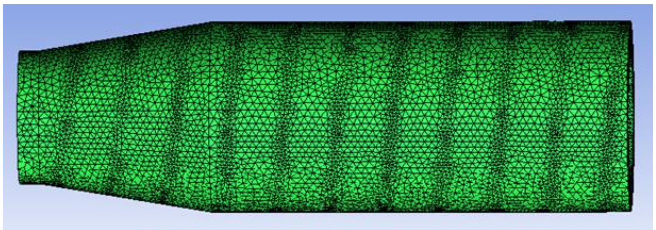


Picture 2 Drum assembly

The picture 3 shows the cutaway view of decanter centrifuge's runner. We carry out a non-structural mesh generation of decanter centrifuge's flow field with the use of software ICEM-CFD. The result is showed in picture 4. The types of volume mesh are mainly tetra and mixed, and volume mesh is generated automatically. The minimum length of wire mesh is 8mm. Altogether 903807 grids and 161145 nodes are generated in the system. Second order upwind discretization is adopted in momentum equation, and Simple algorithm is adopted in pressure-velocity coupling.



Picture 3 cutaway view of decanter centrifuge's runner



Picture 4 Non-structural mesh of decanter centrifuge

Flow field model is relatively simple. Materials can be put in the machine directly from inlet, and the exits are set on the 2 end faces. The main structure parameters of decanter centrifuge are showed in diagram 1.

Table 1 the main structure parameters of centrifuge

Items	Values
Outer diameter of drum's column D/mm	355
Length of drum's column l_1/mm	855
Length of drum's cone l_2/mm	380
Depth of drum's liquid pool h/mm	40
Taper angle of drum α/ (°)	9
Screw diameter $d_1$mm	350
Lead S/mm	110

The velocity inlet is adopted in the boundary of entrance. Processing capability Q=40 m^3/h, velocity at entrance $v = 6.14m/s$, density of liquid phase $\rho_1 = 998.2kg/m^3$, density of solid phase $\rho_2 = 2600kg/m^3$, viscosity $\mu_1 = 0.03pa \cdot s$, the average diameter of solid phase particles $d_s = 50\mu m$, relative pressure P=0.3MP, volume fraction of suspension liquid is 30%, and the turbulence intensity at entrance is 5%. Exit boundary is set as the pressure-outlet, as well as spiral and drum are set as the rotating wall. Revolving speed of drum is 3200r/min, and that of auger conveyor is 3150r/min. So, a discrepancy of 50r/min exists. Fluid must meet the demand of non-slipping on all the walls.

V. The Result and Analysis on Simulation

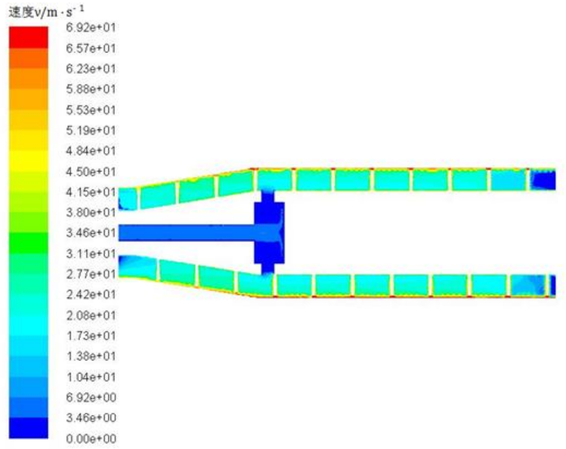
A. The existence of lag coefficient

The drum and spiral push the surrounding liquid layer to rotate together when decanter centrifuge is working. However, the revolving speed of liquid is not consistent with that of drum when liquid enter into centrifuge. There is a certain degree of speed lag. By dimensional analysis and retrogression treatment of experimental data, Sun Qicai[8] solved the N-S equations and deduced the computational formula of angular velocity field in swirling flow field. Besides, by working together with Dong Lianzhu to compare the simulating value and theoretical value, he confirmed that the relative error between simulating lagged value of tangential velocity and the calculated value in record is under 5%.

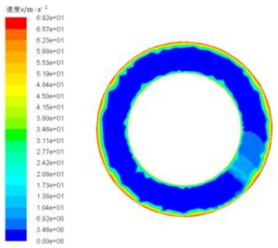
B. The result and analysis on the velocity field simulation

To show the properties of flow field clearer and more realistic, the simulation is carried out with the existence of discrepancy in revolving speed of spiral and drum.

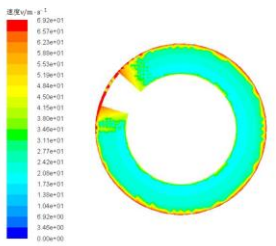
The distribution of velocity field is observed at the cutaway X=0. See picture 5.



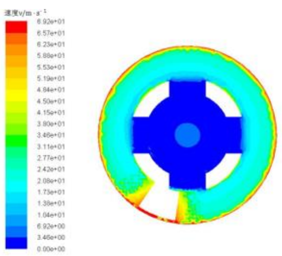
Picture 5 (a) X=0 the distribution of velocity field of decanter centrifuge



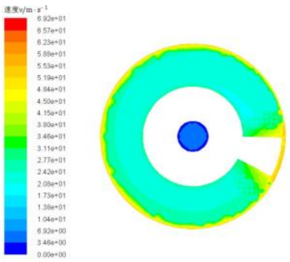
(b)z=100mm



(c)z=100mm



(d)z=800mm



(e)z=1100mm

(1) The velocity of suspension liquid first entering into centrifuge alters dramatically in the direction of radius. Take a look at picture (b), the circumferential velocity grows from 0m/s to 69.2m/s, and the varying gradient is relatively big. The varying gradient tends to be narrow and stabilized while fluid enters into the middle section of drum.

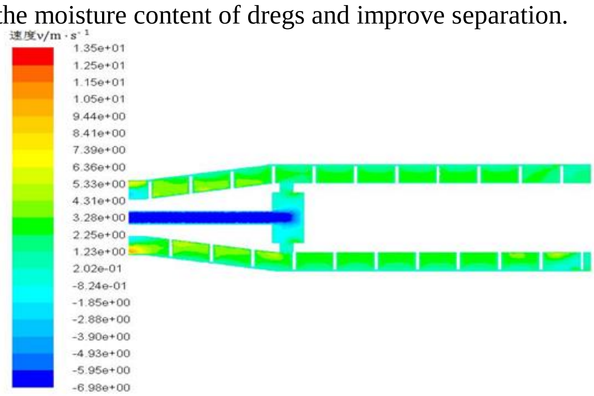
(2) In separation room, the circumferential velocity of fluid is almost 0, as showed in picture (d). The reason can be explained in several steps. First fluid is obstructed when it enters from feeding pipe to separation room, then the axial velocity turns to 0 immediately while fluid gains speed along wise direction, next, the suspension liquid comes back to column under the influence of backflow, and accelerates gradually from column to make separation more sufficient.

(3) The maximum of fluid's circumferential velocity is more or less decreased around the dregs discharge exit, and the velocity varying gradient is narrowed as well, as showed in picture (e). That's because the radius of cone is decreased gradually. From equation (3), we know the circumferential velocity is in direct proportion to radius, and the decrease of cone's circumferential velocity is conducive to the backflow of liquid phase. The solid phase can be discharged from dregs discharge exit with the use of spiral conveyor, which will promote the further solid-liquid separation, thus improving the separation effect of centrifuge.

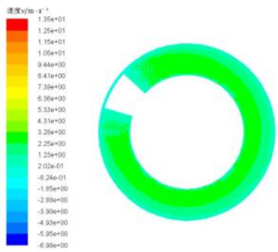
C. Axial velocity

Another physical quantity to measure the production capability of centrifuge is axial velocity. The higher the flow velocity is, the more capable the centrifuge can be.
However, the too-high axial velocity will disturb the already-precipitated dregs, which will do harm to the separation effect and increase the moisture content of dregs. From picture 6 we can find that the axial velocity in this simulation varies between -0.288m/s—-3.28m/s, and the varying gradient is relatively small, which accords with the actual situation.

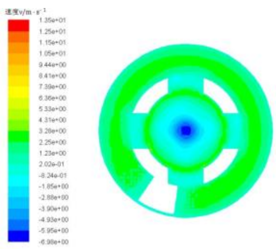
The axial velocity of fluid is complex and unsteady, because it is influenced by the internal geometric structure in centrifuge during fluid flow, especially when there is screw blade. When the axial velocity around the cone's dregs discharge exit appears to be negative, as showed in picture 6 (c), it means fluid is flowing backward, which will decrease the moisture content of dregs and improve separation.



Picture 6(a) The distribution of axial velocity in decanter centrifuge



(b)z=500mm

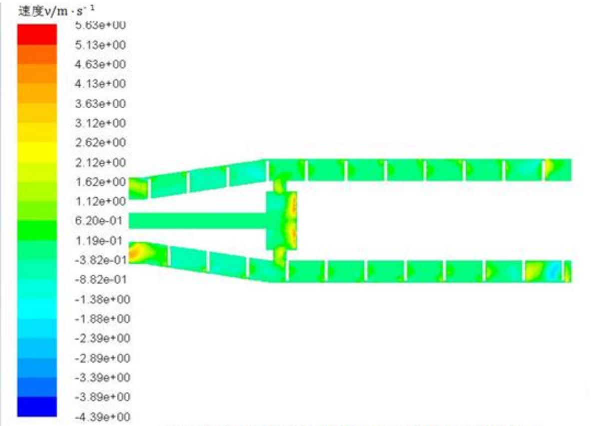


(c)z=800mm

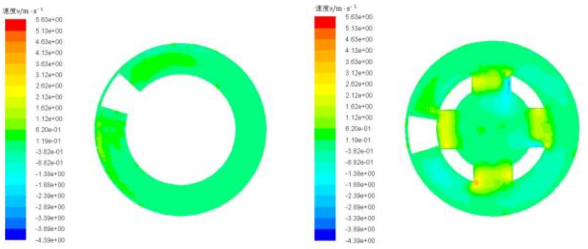
D. Radial velocity

According to laminar flow theory, the varying gradient of radial velocity in the radial direction can be neglected. However, this is far from real situation, where the internal flow field of decanter centrifuge belongs to turbulence flow. The internal flow field is different from laminar flow in velocity distribution. According to Prandtl's boundary layer theory[9], there exists the velocity gradient area around the drum wall, and the velocity gradient is relatively big at boundary layer. Picture 7 shows the cloud image of distribution of radial velocity.

From contour maps in picture 7, we can find that the radial velocity in separation room varies dramatically from -0.119m/s to 4.13m/s when fluid enters into separation field. So the change in radial velocity caused by turbulence can not be ignored. According to turbulence theory, diffusion occurs in this position will break the balance of suspension liquid, and hinder suspension fluid from entering into separation field steadily and smoothly, both of which will severely effect separation. So, in later optimum structural design, the internal structure of separation room should be taken into consideration to avoid the diffusion of suspension fluid in separation room and improve the effect of solid-liquid phase stratification.



Picture 7 The distribution of radial velocity in decanter centrifuge



（b）z=500mm (c)z=800mm

VI. CONCLUSION

Turbulence model RNG k-ε in Fluent is used in this simulation, with the influence of screw as well as the discrepancy between screw and drum in revolving speed to be considered, thus making the simulation of internal 3-dimentsional flow field in decanter centrifuge more realistic. The regularities of distribution of each velocity component in flow field are confirmed. The conclusions and suggestions are as follows:

(1) Circumferential velocity is the main cause of centrifugal sedimentation in centrifuge. From simulation, we know circumferential velocity is in direct proportion to revolving speed of drum. In the later design, to improve the revolving speed of drum as good as possible on the basis of meeting the structural requirement is the trend in future.

(2) According to simulation result, the axial velocity is negative in separation room, which means backflow occurs in separation room and will adverse to the separation of suspension liquid. In later design, we should take into consideration the separation room structure's influence on solid-liquid separation, so as to design the proper runner structure to improve separation efficiency. From the cloud map of simulation result, we find that the axial velocity tends to decrease as the revolving speed of drum increases, and the declining gradient tends to be the same in different liquid phases. The axial velocity in cone of drum appears to be negative, and liquid phase flow backwards, both of which are conducive to solid-liquid separation. In later design, to increase the cone to the utmost on the basis of a smooth dregs discharge will improve separation.

(4) According to the analysis on radial velocity, the radial velocity varies dramatically from -0.119m/s to 4.13m/s in separation room, which will severely influence the stability of suspension liquid, and hinder the circumferential velocity of suspension liquid. To solve this, we suggest to add a secondary accelerator in separation room, which will counterbalance the severe change of velocity, and get a preacceleration of one circumstance for suspension liquid, thus making suspension liquid to get close to the revolving speed of drum faster to improve separation efficiency.

(5) There is difference between simulation and real situation. So, in later simulation, the complexity of real situation should be taken into account, the appropriate simplification and hypothesis should be made as well, to obtain more accurate data to guide design.

REFERENCES

[1] Q.C. Sun, D.W. Jin. Centrifuge structural principle and design calculations, Machinery Industry Press in Beijin.1987.
[2] S.F. Zheng, X. Ren, L.J. Xie, Numerical simulation of flow field in decanter centrifuge, Light Ind. Mach. 27 (6) (2009) 26-29.
[3] G.R. Zhu, W. Tan, Y. Yu, L.Y. Liu, Experimental and numerical study of the solid concentration distribution in a horizontal screw decanter centrifuge, Ind. Eng. Res. 52 (48) (2013) 17249-17256.
[4] P. Yu, W. Lin, X.L. Wang, et al., Velocity simulation analysis on centrifugal separation field of horizontal spiral centrifuge, J. Mech. Eng. 47 (24) (2011)151-157.
[5] L.D. Dong, S.C. Fu, H.X. Yuan, Numerical simulation on pressure filed in a decanter centrifuge, Chem. Ind. Eng. Pro. 33 (2) (2014) 309-313, 336.
[6] X.G.Song,L.Cai,H.Zhang, ANSYS Flow structure interaction analysis and engineering design, China Water Power Press in Beijin.2012.
[7] W.P.Li, Computational Fluid Dynamics, Huazhong University of Science and Technology Press in Wuhan.2004.
[8] Q.C.Sun,F.J.Zeng, Settlement centrifuge drum the fluid velocity field,J. Fluid Engineering. 1990(11)1-5.
[9] J.Z.Lin,X.D.Ruan,B.G.Chen, Hydromechanics, Tsinghua University Press in Beijin.2005.

First Author Qiyun Cheng, A postgraduate study in School of Mechatronic Engineering, Southwest Petroleum University, China.

Second Author Hongbin Liu, A associate professor. in School of Mechatronic Engineering, Southwest Petroleum University, China.
Third Author Yuxin Tian, A postgraduate study in School of Mechatronic Engineering, Southwest Petroleum University, China.