RESEARCH ON CFD SIMULATION OF A NEW TYPE OF HIDROCLON USED IN THE WINEMAKING INDUSTRY

CERCETĂRI PRIVIND SIMULAREA CFD A UNUI NOU TIP DE HIDROCLON FOLOSIT ÎN INDUSTRIA VINIFICAȚIEI

***BĂETU M.M.*[1], *CÂRLESCU P.*[1], *ȚENU I.*[1]**
e-mail: marius.baetu@yahoo.com

Abstract. *Currently, in winemaking industry, is seeking solutions for a separation of solid particles, whether organic or anorganic origin, both in must and in the final stage of the wines clarifycation. The advantages of using separation through hydrocyclones is given by the short time of separation, separation quality and not least, by the continued operation of the facility. New constructive forms for hydrocyclones have been possible as a result of the possibilities offered by CFD simulations (Computational Fluid Dynamics). In this work were made more CFD simulations for the optimization of a new hydrocyclone model, in terms of geometric and functional, with a view to increasing the efficiency of separation. The optimized hydrocyclone, proposed in the paper, has undergone experimental study and the CFD simulation coupling the Reynolds turbulence model for fluid flow with the discrete phase model for tracking the particles trajectory that will be separated inside.*
Key words: *winemaking, hydrocyclone, CFD simulations*

Rezumat. *În prezent, în industria vinificației, se caută soluții pentru o separare a particulelor solide, fie ele de origine organică sau anorganică, atât din must cât și în etapa finală de clarificare a vinurilor. Avantajele utilizării separării prin hidrocicloane este dată de timpul scurt al separării, calitatea separării și nu în ultimul rând de funcționarea continuă a instalației. Noi forme constructive pentru hidrocicloane au fost posibile ca urmare a posibilităților oferite de simulările CFD (Computational Fluid Dynamic). În această lucrare au fost realizate mai multe simulări CFD pentru optimizarea a unui nou model de hidrociclon, din punct de vedere geometric și funcțional, în vederea creșterii eficienței separării. Hidrociclonul optimizat, propus în lucrare, a fost supus studiului experimental, iar simularea prin CFD cuplează modelul de turbulență a lui Reynolds pentru curgerea lichidului cu modelul fazelor discrete pentru urmărirea traiectoriei particulelor ce urmează a fi separate în interiorul acestuia.*
Cuvinte cheie: *vinificație, hidrociclon, simulare CFD*

INTRODUCTION

The need to introduce a hydrocyclone in the technological flow of production of wines, is to separate a large amount of anorganic and organic particles, up to the dimensions of 8–10 µm (the size of a cell yeast).

New constructive forms for hydrocyclones were possible as a result of the possibilities offered by CFD simulations (**C**omputational **F**luid **D**ynamic).

---

[1]University of Agricultural Sciences and Veterinary Medicine Iasi, Romania

Ultimately, many experiments were carried out to determine the flow within a hydrocyclone. Knowles et. al.(1973) carried footage at high speed to track anisol drops moving in a hydrocyclone and to determine rates of the liquid flow. More recently, a number of researchers have carried out a series of measurements with laser (Laser Doppler Velocimetry - LDV) and by electrical impedance tomography (Dai, 1999; Hsieh and Rajamani, 1991).

Experimental methods proposed are expensive and difficult to implement from a technical perspective, being limited to a dispersed liquid phase. Along with experimental efforts, various analytical models have been established, in order to control and determine the flow characteristics of a hydrocyclone (Chen et. al., 2000; Nageswararao et. al., 2004; Plitt. et. al., 1976).

Most of the works have addressed the CFD simulation for determining flow and particle trajectory in conventional geometry hidrocicloane in various sizes.

As a result of experiments conducted by Băetu et. al. (2012), using a hydrocyclone whose geometry is shown in figure 1a, the authors demonstrated that the best speed, to obtain a must with a lower turbidity, is 1800 rpm.

Under the same experimental conditions and the particle trajectory simulation analysis of the must, it was observed that the separation efficiency for particles of 100 µm diameter is insignificant (4.28%).

The paper presents a new three-dimensional geometric model of hydrocyclone, which was constructive optimized by numerical modeling of the must flow and particle trajectories which initially are foundin the must. The CFD simulation of the hydrocyclone shows the particles separation degree

MATERIAL AND METHOD

The hydrocyclone, whose geometry is shown in figure 1a, used in experimental research, has two exits on the vertical axis: an outlet at the bottom, which typically is collected solid fraction (4), and an outlet at the top where it is discharged the clarified liquid or with a small amount of solid fraction (3). The entrance of the solid - liquid mixture passes through the top of the hydrocyclone and is positioned tangentially (2). As a result of the tangential entry, the suspension is partially agglomerated on the walls of the hydrocyclone, due to the effect of the centrifugal force, and the fluid flow and the form construction part pull the solid particles to the bottom of the hydrocyclone.

In order to increase the probability, that the most of the particles with a diameter of 100 µm to be separated, it recourse to lengthen the separation chamber of the hydrocyclone, by doubling the capacity, as shown in figure 1b.

The ultimate goal is to identify the optimal flow for a efficiently separation of the particles with a diameter of 100 µm and the evolution of these particles in conjunction with the must turbulence when flicking the hydrocyclone from entrance to the exit.

Physical parameters of the must and particles, introduced into the hydrocyclone, have the following values: must viscosity η = 0.0018 Pa/s, must density ρ_l = 1085 kg/m^3, particles density ρ_P = 1130 kg/m^3. The particle flow introduced in hydrocyclone trough the inlet connection shall be 10% of the must flow-rate. This flow of particulates must not exceed 10 – 12% and it fits in the recommendation of the FLUENT textbook (Ansys-Fluent – User Guide, 2010).

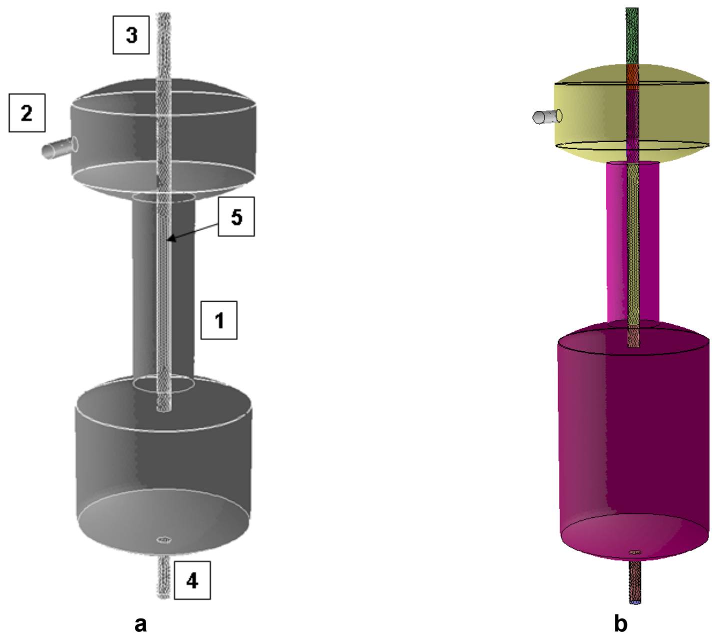


Fig. 1 – The geometry of the originally hydrocyclone (a) and the geometry of the modified hydrocyclone (b):
1 – cylindrical body; 2 – input connection for must; 3 – output connection for the partially cleared must; 4 – connection for sediment purging; 5 - inner pipe collection for partially cleared must.

Experimental tests were performed more by varying the pump speed *n* of the hydrocyclone resulting different flow rates and different inlet speeds (table 1).

Table 1

Outline experimental conditions

Pump revolution n (rot/min)	**Speed**		**Flow rate**	
	u (m/s)	$\mathbf{u_P}$ (m/s)	$\mathbf{Q_l}$ (kg/s)	$\mathbf{Q_P}$ kg/s)
1200	2.62	2.62	0.358	0.0372
1500	3.10	3.10	0.423	0.0441
1800	3.63	3.63	0.488	0.0508
2100	3.93	3.93	0,531	0.0553

(u- must speed, u_p- particles speed, Q_l – inlet flow rate of the must, Q_p – inlet flow rate of the particles)

The separation of particles from the must, assume a knowledge of the nature of existing particles from the physical point of view, the particle diameter, geometric shape and the particles concentration in the must volume. The particles introduced in hydrocyclone separation simulation are considered to have a spherical shape and the size was 100 µm.

In order to characterize the turbulent flow occurring as a result of must rotation within the hydrocyclone, it is necessary to apply an appropriate turbulence model. RSM turbulence model (**R**eynolds **S**tress **M**odel) describes with a good precision the anisotropic turbulence (Wang et al., 2008; Xu et al. 2009).

The processing of the system of partial differential equations composed of continuity equation, moment equation, Reynolds tensions equation (**RSM**) and the equation of turbulent energy dissipation rate ε was performed using a segregated solver (equations are calculated so that the continuity equation is satisfied and local) with the program ANSYS - FLUENT V 3.6.26.

Simulated solids hydrocyclone movement is carried out by selecting the Lagrange model probabilistic (stochastic) tracking particle trajectory, model called DPM (**D**iscrete **P**article **M**odel) and was also performed with the program ANSYS - FLUENT V 6.3.26.

The simulation of the must flow and particle movement in hydrocyclone, in a reasonable time, was conducted with TYAN workstation (2XCPU-Intel Xeon 3.33 GHz RAM - 16 GB DDR3).

The results of processing are given in the speed fields form, the Reynolds (Re) turbulence form, the current lines form and particle trajectory form, for several sizes.

RESULTS AND DISCUSSIONS

Following the CFD simulation of the experimental data, it was found that the best results are obtained at a inlet rate flow of 0.531 kg/s. This was achieved by changing the flow speed to 2100 rpm.

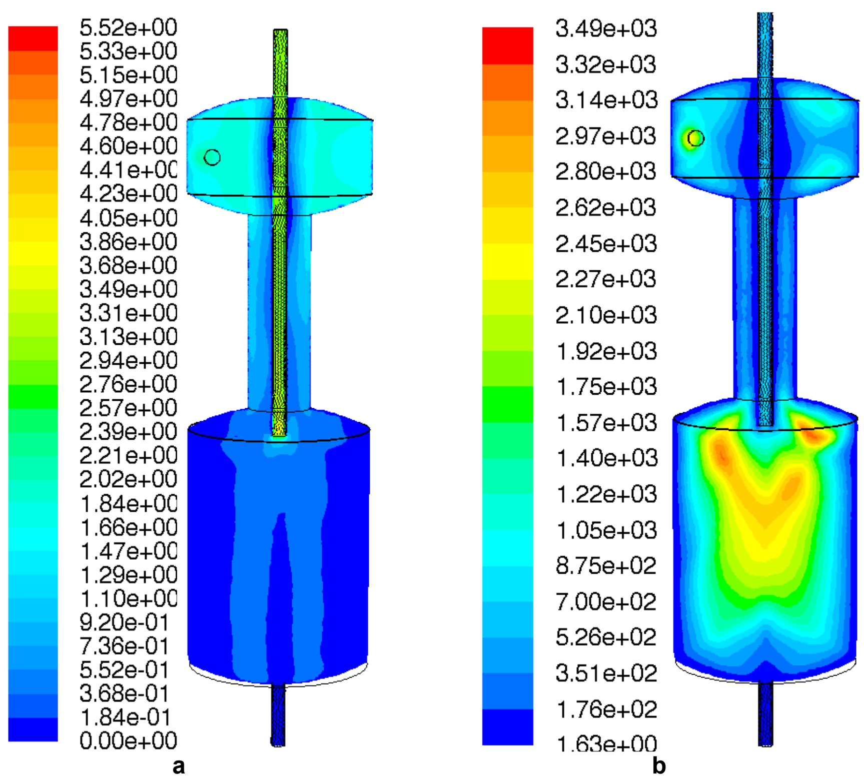
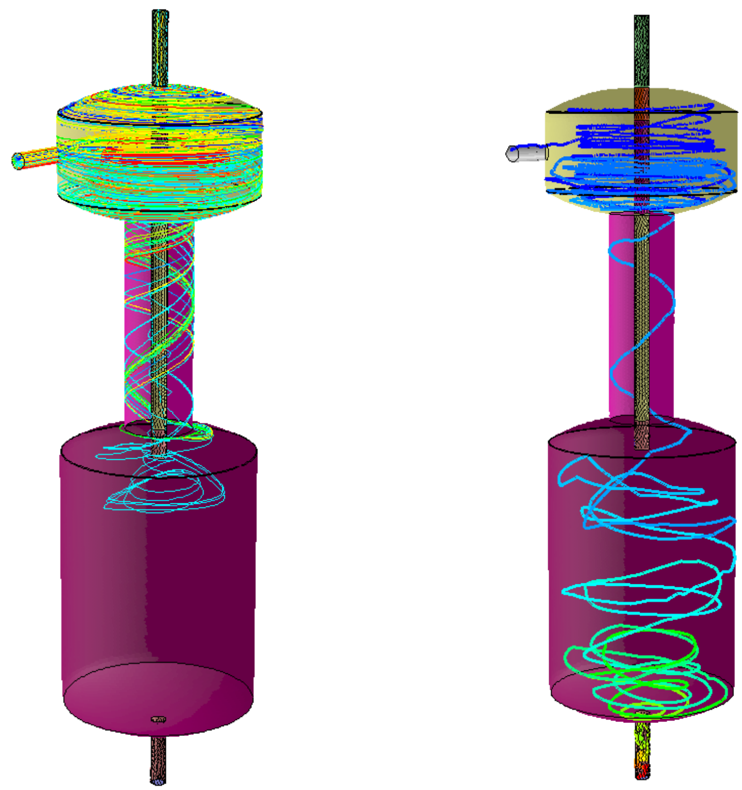


Fig. 2 - Representation of the must speed profile **(a)** and the Re number profile in the hydrocyclone **(b)** at a inlet flow rate of 0.531 kg/s

At the input maximum flow of the must in the hydrocyclone, the distribution of the field of speed (fig. 2a) show a different speed of 0 in the sedimentary body, on the vertical sedimentation pipe direction.

Moreover, it can be seen that, through the representation of the speed field, it shows an increase of must speed in upper body where the centrifugal forces prevail, and as the fluid forward towards the bottom, the field speed intensity drops by allowing the particles to settle.

The turbulence field given by the Reynolds number (fig. 2b) is higher in the upper body of the hydrocyclone, where centrifugal force predominates. Here, it can be observe an increase of the turbulence in the upper part of the separation chamber and its decrease in the particles deposit on the drain pipe.



a **b**

Fig. 3 - Representation of the current lines (**a**) and the trajectory of a particle with dp = 100 μm in the hydrocyclone (**b**) at a inlet rate of 0.531 kg/s

The current lines obtained by CFD simulation (fig. 3a), after the stabilization of the must flow, at a maximum must rate entry, don't have trajectories to reach the bottom wall of the sedimentation bod. Thus, it can be say that, the most many particles with de diameter of 100 μm will settle on the drain pipe.

In figure 3b it can be seen that the representation of a sinuous trajectory of a particle in the hydrocyclone, due to high turbulence, it is finally deposited on the drain pipe.

CONCLUSIONS

1. The mathematical modeling and the CFD simulation (Computational Fluid Dynamic) of the mixture flowing in the proposed hydrocyclone, show us the flow regime from inside as well as the trajectories of the fluid current lines and of the solid particles.

2. The advantages of using CFD simulation on a hydrocyclone are saving time and material resources, when using performance programs and hardware (FLUENT, TYAN workstation).

3. By using the CFD simulation, which are based on experimental data, we can design new types of hydrocyclone, in order to obtain optimal variants to increase the efficiency of separation

REFERENCES

1. **Băetu M.M., Țenu I., Cârlescu P., Pahoni C., Barcan (Băetu) Alina Loredana, 2012** - *Research on experiment the hydrocyclon SLV02 for the partial separation of the must,* Lucrări științifice, USAMV Iasi, Seria Horticultura, vol. 55, nr.1, p. 537-540.
2. **Chen W., Zydek N., Parma F., 2000 -** *Evaluation of hydrocyclone models for practical applications*, Chem. Eng. J., 80, , p. 295-303.
3. **Dai G.Q., Chen W.M., Li J.M., Chu L.Y., 1999** – *Experimental study of solid-liquid two-phase flow in a hydrocyclone.* Chem. Eng. J., 74, p. 211-216.
4. **Hsieh K.T., Rajamani R.K., 1991** – *Mathematical-model of the hydrocyclone based on physics of fluid-flow,* AIChE J., 37, p. 735-746.
5. **Knowles S.R., Woods D.R., Feuerste I.A. 1973** – *Velocity distribution within a hydrocyclone operating without an air core,* Can. J. Chem. Eng., 51, p. 263-271.
6. **Nageswararao K., Wiseman D.M., Napier-Munn T.J., 2004** – *Two empirical hydrocyclone models revisited,* Miner. Eng., 17, p. 671-687.
7. **Plitt L.R., 1976** – *A mathematical model of the hydrocyclone classifier*, CIM Bull., 69, p. 114-123.
8. **Wang B., Yu A.B., 2008** – *Numerical Study of the Gas-Liquid-Solid Flow in Hydrocyclones with Different Configuration of Vortex Finder*, Chem Eng J.,vol. 135, p. 33-42.
9. **Xu P., Wu Z., Mujumdar A.S., Yu A.B., 2009** - *Innovative Hydrocyclone Inlet Designs to Reduce Erosion-InducedWear in Mineral Dewatering Processes*, Drying Tech., Vol. 27, p. 201-211.
10. *****,** 2010 – *Ansys-Fluent – User Guide*