CHEMICAL ENGINEERING TRANSACTIONS

VOL. 98, 2023

A publication of AIDIC

The Italian Association of Chemical Engineering
Online at www.cetjournal.it

Guest Editors: Sauro Pierucci, Carlo Pirola

ISBN 978-88-95608-97-6; **ISSN** 2283-9216

DOI: 10.3303/CET2398002

Modeling, Simulation and Optimization of Hydro-cyclones

Fatima Ez-zahra El Hamra[a], Sara Tajidi[a], Lhachmi Khamar[a,b], Saad Benjelloun[a*], Safae Elmisaoui[a,c]

F LGCE, Mohammed V University, Rabat, Morocco
saad.benjelloun@um6p.ma

Cyclones are widely used in chemical and pharmaceutical industries to classify, separate and sort particles in fluid suspensions. They constitute the main device in many industrial and water treatment plants. Therefore, many mathematical studies were concerned about optimizing the required classification energy and maximizing the efficiency, as well as avoiding the rejection of harmful elements into nature through the overflow of the hydro-cyclones. This work presents numerical techniques to solve different mathematic and phenomenological models describing the physics inside a hydro-cyclone and conducts a parametric optimization of this unit operation. The ANSYS suit is used to achieve these objectives. To validate the CFD results, a semi-empirical model has been adopted (Plitt model). Many useful insights were obtained, for instance, the most influencing operating parameter is the inlet solid fraction for the cyclone. This parameter widely affects its efficiency, and for some range of values, the particles may go through the cyclone's overflow instead of its underflow.

1. Introduction

Due to their high flow rate, low maintenance needs, space saving and other advantages, cyclones are highly used in diverse industries (Hoffmann and Stein, 2008). Recently, many new cyclone conceptions were proposed but only the conic and the cylindrical ones are widely used in the industry. The feed pulp is tangentially injected into the hydro cyclone at a relatively high rate, producing a large centrifugal force field with strong vortex flow. The separation in this apparatus is the result of the swirling flow and the displacement of the particles with respect to the fluid having a centrifugal force, gravitational force and drag force (Pericleous, 1987).
In the hydro cyclone, the coarser and / or heavier particles move towards the wall and move simultaneously downwards, while the smallest and / or lighter particles are carried at the top by a fluid, reversed in the Axial direction, spiral upwards, then driven by overflow.
The efficiency of the hydro cyclones can be measured by several parameters. The construction of the partition curve is the first step to assess the performance of classification. The distribution of a particle size class i is defined by the ratio of the flow rate of the particles of the class i in the underflow to the flow of the same class i in the feed (Davailles, 2011). The present work demonstrates the feasibility of using a software package, based on computational fluid dynamics (CFD) codes to simulate two-phase (liquid–solid) flows in an industrial equipment, taking the case of a conical cyclone with a tangential inlet. There are many advantages of using CFD, such as its flexibility and the possibility to modify dimensions and the geometry of a designed device, useful in the case of sizing or optimizing equipment such as cyclones. Also CFD has applications in modeling multiphase flows, which are encountered for cyclones (Wojtowicz & Wolak, 2016), and most of all has an universality of application. Many papers have been published in topics using CFD codes; however, not many works have been established on cyclones parametric optimization. Previous authors used theoretical approach based on cyclone geometry and fluid properties (Pericleous, 1987). The present CFD model is based on a system of equations, including conservation of mass, conservation of momentum, and equations modelling the turbulence inside the cyclone. On the other hand, the simulation of solid particles can be done using the DPM (Discrete Phase Model) approach, which is the case for our study, or the Eulerian approach, where the solid phase is considered as a continuous one.

Paper Received: 11 November 2022; Revised: 20 February 2023; Accepted: 27 March 2023
Please cite this article as: El Hamra F., Tajidi S., Lhachmi K., Benjelloun S., Safae E., 2023, Modeling, Simulation and Optimization of Hydrocyclones, Chemical Engineering Transactions, 98, 9-14 DOI:10.3303/CET2398002

2. Cyclone description and spatial discretization

2.1 Geometry

The equipment used in this study is a conic cyclone, with a height of 1500mm. Its cylindrical part has a radius of 250mm, the inlet and the underflow radius are both 70mm, while the overflow has a radius of 100mm (see Figure 1).

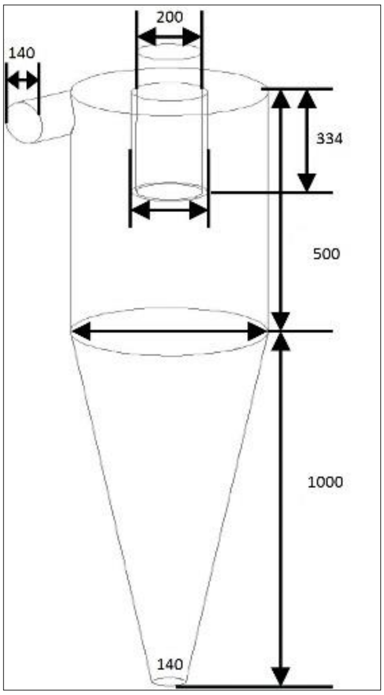


Figure 1: The hydrocyclone geometry.

2.2 Operating parameters

The considered fluid is a slurry composed of water and solid particles. The main hydro-cyclone operating parameters are summarized in the table 1 below.

Table 1: Operating parameters

Parameter	Value
Density	2940 kg/m3
Feed flow rate	1202 m3/h
Feed pressure	91 KPa
Solide concentration in the feed stream	1%wt to 40%wt
Feed size solid particle	10-100 µm

2.3 Domain discretization

Due to the geometrical complexity of this system and the absence of a privileged direction of flow, the structured, conforming and purely tetrahedral mesh constitutes the best choice for the spatial discretization (see Figure 2).

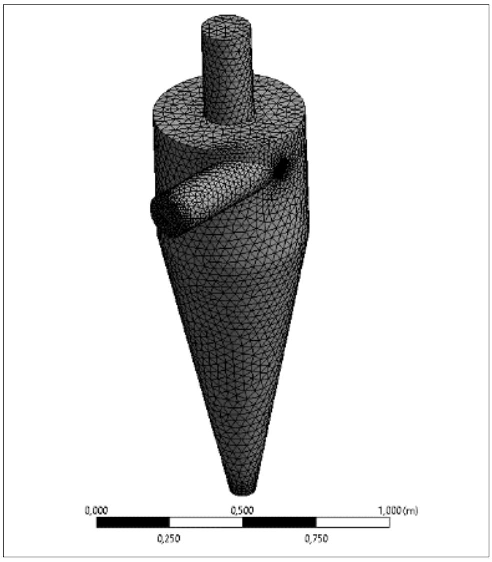


Figure 2: The hydrocyclone mesh.

2.4 Phenomenological modeling

To study and diagnose the flow within the hydro-cyclone, several equations were solved simultaneously, namely continuity equation, Navier-Stokes equations, and turbulence model equations which are added for the closure of the system. The K-Ɛ, realizable model is used. For solid particles flow, the DDPM approach (Dense Discrete phase Model) is used (Surmen et al, 2011; Shahbazi et al, 2009).
As simplifying hypothesis, the flow is considered incompressible, and the system is isothermal, i.e. the temperature is constant and homogeneous throughout the fluid sheath of the cyclone.

3. Results and discussion

Based on the finite volume method, the CFD approach is used to simulate and to analyze the hydrodynamics flow inside the cyclone, its performances and efficiency under different operating parameters, aiming to optimize the solid-liquid separation and particle size classification.

3.1 Effect of hydro-cyclone inlet velocity

Among the most interesting characteristics for the diagnostic and the assessment of proper functioning of hydro-cyclones is the determination of the velocity field inside the fluid sheath, in particular at the level of the connection between the supply line and the cylindrical section. Indeed, at this point, the velocity must be maximal and should have a uniform distribution over the cylindrical section of the hydro-cyclone. Otherwise, the presence of very high or very low speed zones leads directly to a bad solid liquid separation.

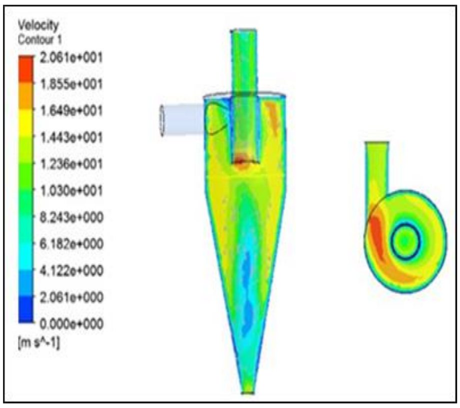


Figure 3: Contours of inlet velocity V=10m/s.

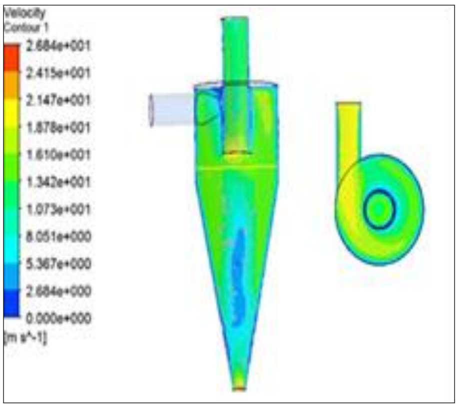


Figure 4: Contours of inlet velocity V=20m/s.

As seen in Figures 3 and 4, showing the velocity fields with 15 wt% inlet solid fraction, the velocity profile in the cylindrical section exhibits poor distributions for input velocity inlet less than 10m/s. However, when the velocity values exceed 15 m/s, the tangential velocity distribution becomes uniform, which ensures a best separation and a regular classification of the hydro-cyclone. This phenomenon has been observed at different inlet solid fractions in the considered range (1 to 40 wt%).

3.2 Effects of solid content in the feed stream

To inspect the effect of the solid content (Ts) on the separation acuity, we tested several mass percentages of the solid at the inlet, ranging from 1%wt to 40%wt. The results of the simulations showed that one could distinguish between two cases, a feed of solid less than 10%wt and another one higher than this value.

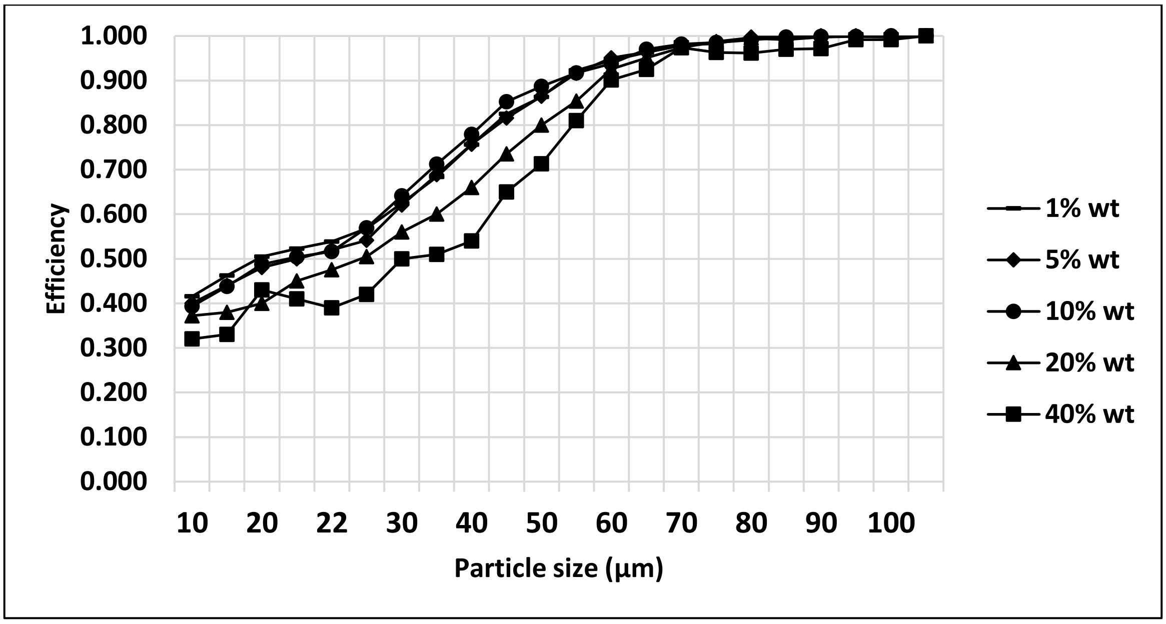
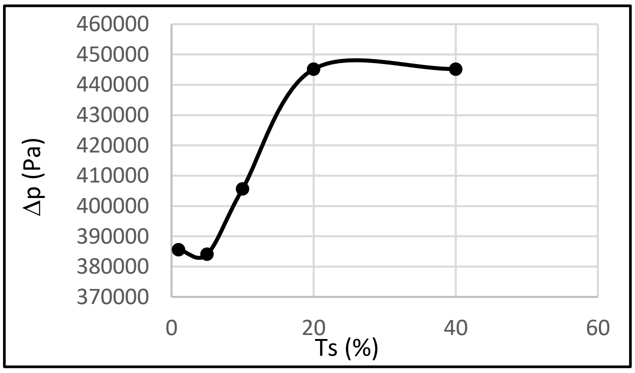


Figure 5: Effect of the inlet solid rate on the classification efficiency.

In the first case, the partition curves remain practically insensitive to the increase of the solid concentration in the pulp feed of the hydro-cyclone. While in the second case, we can notice that it substantially affects the classification efficiency.
For the hydro-cyclone cut size, the CFD results show that the median diameter dp50 for the underflow increases linearly with the increase in the solid content in the pulp feeding the hydro-cyclone. However, pressure losses remain virtually constant for solid levels below 5wt% and then begin to increase linearly until the solid content reaches 20% and then stabilizes again and remain insensitive to the solid rate increasing (Figures 6, 7).



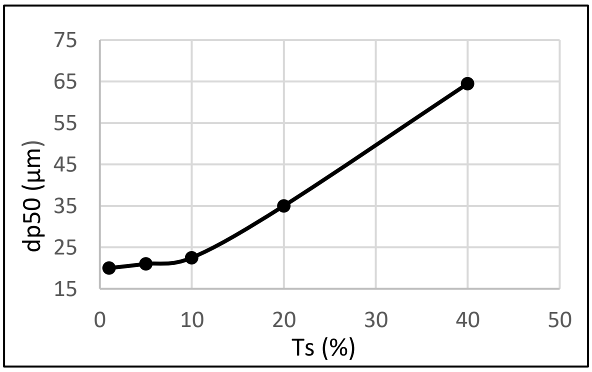
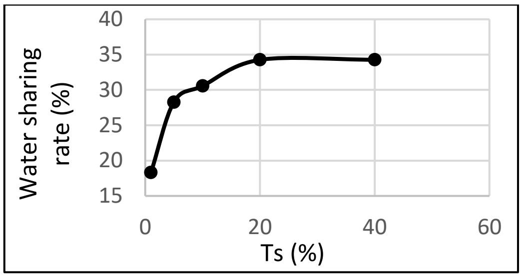


Figure 6: Effect of the inlet solid content on the pressure drop. *Figure 7: Effect of the inlet solid content on cut size*

In terms of water sharing and short-circuiting, the simulation results show that the first parameter increases until the feed solid fraction (Ts) reaches 20% wt, then it remains virtually constant and insensitive to solid fraction. On the other hand, the rate of short-circuit particle decreases in a quasi-linear manner with the increase of Ts.



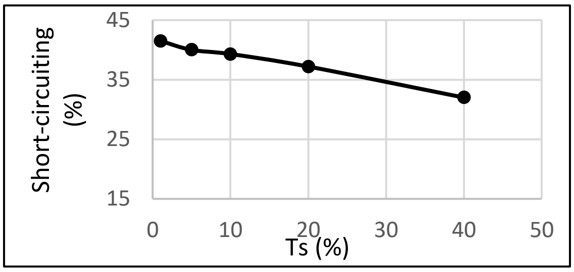


Figure 8: Effect of Ts in the inlet on the water sharing

Figure 9: Effect of Ts in the inlet on the short-circuiting

3.3 Effect of the feed solid rate on the residence time

As shown in the figures 10 and 11 below, the residence time is also highly affected by the feed solid fraction, and it increases by increasing Ts. We explain this by the fact that more the number of particles is important in the cylindrical section, more they need time to get through the overflow, so they keep spinning in that part before finally leaving it.

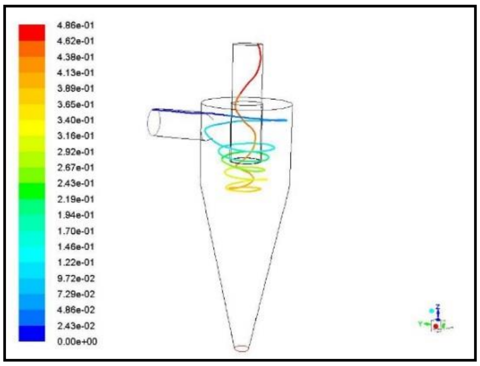


Figure10: Ts= 1 %wt, d=10 µm, ts=0.462 s

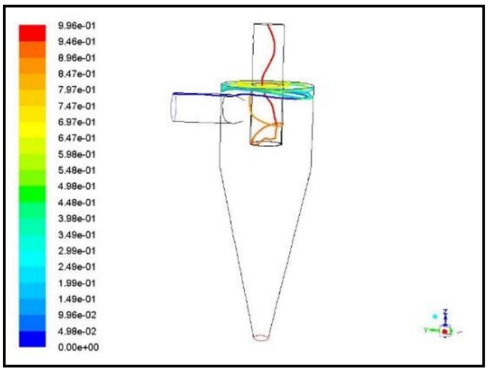


Figure11: Ts=40 %wt, d=10 µm, ts=0.996 s

3.4 CFD predictions versus empirical models

To confirm the CFD results, Plitt model was used, enabling us to calculate the cyclone cut size and its pressure drop (Nageswararao et al, 2011). The table 2 shows that both the CFD approach and the Plitt model give similar results.

Table 2: Models comparison

Ts (%wt)	Dp50 (µm)		ΔP (Pa)	
	CFD	Plitt	CFD	Plitt
1	20	20.86	95096.53	90751.53
5	21	22.01	96099.83	91449.11
10	25.5	25.68	97102.83	92382.23

4. Conclusion

In this work, the CFD approach is used to study and to diagnose the hydrodynamic flow inside an industrial cyclone used for liquid-solid separation. The CFD results reported in this paper revealed that the classification efficiency of hydro-cyclones is highly affected by the operating parameters, especially velocity and solid concentration in the stream feed. Indeed, the results of parametric sensitivity studies show that with feed velocity values greater than or equal to 15 m / s the hydro-cyclone ensures good separation. For the solid content in the feed stream, it has been found that for values below 10 wt%, the latter remains without any noticeable effect on the performance of this separation and particle size classification equipment. Exceeding this value of Ts, the hydro-cyclone becomes very sensitive.

References

Davailles, A., 2011, Effet de la concentration en solide sur les performances de séparation d'un hydrocyclone (simulations numériques et expériences de références). Toulouse INP, Phd Thesis.

Hoffmann A.C., Stein L.E., 2008, Gas cyclones and swirl tubes. Principles, design and operation, Springer - Verlag, Berlin.

Nageswararao K., Wiseman D.M., Napier-Munn T.J., 2004, Two empirical hydrocyclone models revisited. Minerals Engineering 17, 671–687.

Pericleous, K.A. ,1987, Mathematical simulation of hydrocyclones. Applied Mathematical Modelling, 11(4), 242-255.

Wojtowicz, R., & Wolak, P., 2016, An example of the use of computational-fluid-dynamics analysis for simulation of two-phase flow in a cyclone with a tangential inlet. Environment Protection Engineering, 42(3).

Shahbazi, B., Rezai, B., & Koleini, S. M. J., 2009, The effect of hydrodynamic parameters on probability of bubble – particle collision and attachment. Minerals Engineering, 22(1), 57–63.

Surmen A., Avci A., Karamangil M.I., 2011, Prediction of the maximum-efficiency cyclone length for a cy-clone with a tangential entry, Powder Technol., 207, 1.