REVISTA MATÉRIA V.30
ISSN 1517-7076 articles e20240587, 2025

Analysis of fluid flow in Lapple cyclone using analytical method and computer modelling

Givaldo Leopoldo de Oliveira Junior[1], Flavio Maldonado Bentes[1]

[1]Centro Universitário Augusto Motta. Rio de Janeiro, RJ, Brasil.
e-mail: givaldoone@icloud.com, flavio.bentes@gmail.com

ABSTRACT

Cyclones are industrial equipment widely used to induce the separation of suspended solid particles based on a driving force related to the terminal velocity in fluid flow. They are applicable to both gaseous and liquid fluids (hydrocyclones), enabling separation between the solid-fluid physical states. The physical principle behind the separation and operation phenomenon is inertia, utilizing centrifugal force to displace air, consequently facilitating the removal of particulate matter present in the stream. One of its main functions is gas cleaning in industrial processes, due to its low acquisition, operation, and maintenance costs, along with the ability to handle streams at high pressures and temperatures. The primary objective of this study is to simulate the flow and disaggregation profiles in a cyclonic separator using computational fluid dynamics (CFD) via finite volumes, where a control volume is subdivided into discrete elements aimed at referencing points within the continuous domain. This approach enables the application of constitutive equations, converting partial differential equations into systems of linear equations. This study applies the method to a Lapple-type cyclone, validating the numerical results obtained with those available in the scientific literature under the same operating conditions. The comparative parameter used to estimate the relative error was the pressure drop. As a secondary objective, the applicability of the cyclone for neutralizing the hazardous chemical agent ammonia was evaluated. This was achieved through its chemical reaction with acetic acid, enabling a realistic hypothetical leakage study to investigate the possibility of formulating emergency plans. In the event of an industrial accident involving ammonia dispersion, this system could be activated. For this purpose, a multiphase plug flow reactor (PFR) was designed, estimating the conversion, reaction time, and dynamic concentration profiles for the synthesis of ammonium acetate, a chemical agent with lower toxicity compared to ammonia.

Keywords: CFD; Lapple cyclone; Turbulence; Numerical analysis; Separation process.

1. INTRODUCTION

Cyclones are gas-solid separation equipment arranged in various process industries; the high applicability refers to the function of separating suspended solid particles from a gas stream. It is found vastly in industrial chemical, environmental, catalytic recoverers, and bioprocesses [1]. An ideal cyclonic device can be analyzed analytically or via CFD (computational fluid dynamics). Understanding via analytical routes is limited due to the need to make simplifying assumptions during modeling and simplifications regarding the phenomenology. Numerical methods are preferred because they allow for the most feasible reproducibility regarding separation [2], and can analyze heat and mass transfer, turbulent flow, boundary layers, and the driving force for separation. Currently there are several commercially available software programs predominantly designed to solve volume and finite element simulations.

In addition to the study of the applicability of cyclones, evaluations of flow fields, turbulence, definitions of dimensionless numbers and vorticity are still widely evaluated in the scientific literature. In [3], two main flows are evaluated: lip flow and secondary radial inward flow, relating them to the dimensionless Reynolds number and identifying the equality between the magnitudes. In order to evaluate and quantify the interrelationship between the physical properties of the fluid and the flow variables, [4] simulates a two-level cyclone using computational fluid dynamics and compares the air velocity inside the equipment with experimental data. New designs and geometric arrangements are also studied to induce separation and improve performance. In [5] three types of arrangement are evaluated experimentally and numerically considering the cyclone inlet,

Corresponding Author: Givaldo Leopoldo de Oliveira Junior Received on 05/09/2024 Accepted on 16/12/2024

DOI: https://doi.org/10.1590/1517-7076-RMAT-2024-0587

changing the vorticity, changing the inlet diameter and inlet angle, based on the comparative variable Q (vortex identification criterion).

1.1. Theoretical reference

According to [6], cyclones are devices applied for removal of solid or liquid particles with diameter up to 15 µm, which can be achieved collection efficiencies above 99% [7] with particle diameters between 1 and 5 µm, consequently an inversely proportional relationship between diameter and separation efficiency is evaluated [8]. A cyclone is a centrifugal separator in which the particles, due to their inertial mass, suffer the action of a tangential centrifugal force having as response a displacement towards the inner surfaces, resulting in an inelastic collision with the wall [9].

By understanding the conservation of momentum, this provides for the formation of a gradient to promote separation [10, 11]. The inlet air is geometrically influenced to adopt a spiral motion of accelerated rotation called a double vortex. This motion consists of an outward flow in a spiral shape and vertical upward direction concurrently with an inward flow with the same typology but in a downward direction [12]. In the interchange between the two streams, there is transfer of the gaseous current between the adjacent streams [13]. The airborne particles are flushed to the outer edges and leave the separator through a collection device installed at the bottom of the cyclone separator. The operational air velocity in the cyclone is between 10 and 20 m/s, with an optimized value of 16 m/s. Among the disturbances in the separation, velocity fluctuations (values lower than 10 m/s), reduce the separation efficiency to a great extent, by decreasing the tangential force in line with the collision energy.

1.2. Cyclone – Lapple model

The families of cyclones were based on practical development, in this study the Lapple (1984) theory was used, the method used consists of varying the diameter of the gas outlet duct as a function of efficiency, from obtaining the optimal value between both variables [14].

The set of equations illustrated below refers to the Lapple type cyclone model. The efficiency (E) is previously described as inversely proportional to particle diameter (dp) and cyclone cut (dc). The estimate for cut diameter relates to physical properties and flow. The number of turns (N), head loss (ΔP) and Head Loss Coefficient (NH) are calculated for the complete description and sizing. Importantly, such relationships are obtained from empirical correlations based on the transport phenomena occurring inside the separator [15]. Table 1 details each parameter with its physical description and unit of measurement (International System – SI).

$$E = \frac{1}{\left[1 + \left(\frac{d_c}{d_p}\right)^2\right]} \tag{1}$$

Table 1: Physical parameters applied in Lapple modeling.

PARAMETERS	PHYSICAL DESCRIPTION	UNIT
d_p	Particle diameter	[m]
d_c	Cyclone cut diameter	[m]
μ	Gas viscosity	[kg/m.s]
W	Inlet Width – Cyclone	[m]
N	Number of Turns – Streamlines	[–]
V	Inlet gas velocity	[m/s]
ρ_p	Specific mass – particle	[kg/m^3]
ρ_g	Specific mass – gas	[kg/m^3]
H	Inlet cyclone height	[m]
L_b	Body length – cyclone	[m]
L_c	Cone length	[m]

$$d_c = \sqrt{\frac{9\mu W}{2\pi NV\left(\rho_p - \rho_g\right)}} \tag{2}$$

$$N = \frac{1}{H}\left(L_b + \frac{L_c}{2}\right) \tag{3}$$

$$\Delta P = \frac{\rho_g}{2} V^2 N_H \tag{4}$$

$$N_H = 16\left(\frac{WH}{D_e^2}\right) \tag{5}$$

1.3. Computational fluid dynamics

Computational Fluid Dynamics (CFD) is a tool based on numerical methods that simulates the behavior of real systems involving fluid flow, heat transfer, chemical reactions, turbulence, porous media, and others. CFD solves the partial differential equations of fluid flow (Navier Stokes equation) over a bounded region, applying the specified boundary conditions at the system boundaries in order to stipulate the resulting integration constants during the solution [16].

Regarding the operational steps, initially a geometry is prepared in CAD, which will be used to study the equations of conservation of momentum. Subsequently, the calculation mesh is prepared, in other words, discretization of the continuous medium, whose geometry is divided into small cells to which the numerical methods will be applied to solve the problem. The next step is to solve the system of algebraic equations in which the flow profiles are obtained as a function of pressure, velocity, turbulent kinetic energy, temperature and others, facilitating the observation of the phenomena inside the device.

1.4. Equation for conservation of momentum

For the numerical evaluation of a physical phenomenon, it is necessary to translate the physical aspect into mathematical expressions that are based on conservation principles. Among the most used in engineering are: conservation of momentum, energy, mass, electric charge, chemical species. As for fluid flow, the equation that describes the respective velocity, pressure, and turbulence profiles is the conservation of momentum, referred to as the Navier - Stokes Equation in a control volume [17].

Equation (7) mentions the conservation, where each term, respectively, means: accumulation term that expresses the fluid compressibility along the flow; convective term, where inertial forces contribute; pressure loss; tension between fluid layers plus surface effects and gravitational force.

$$\frac{\partial U_i}{\partial t} + \frac{\sum_i U_j \partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \sum_j \frac{1}{\rho}\frac{\partial \tau_{ij}}{\partial x} + g_i \tag{6}$$

1.5. Turbulence model: k-ε two RNG equations

The class of $\kappa - \varepsilon$ – two equation models is the basis for several other turbulence models, which in this work applied the RNG type. The formulation via RNG is based on the statistical approach called group theory, in which it incorporates calculation constants for the purpose of improving the accuracy of the numerical method. This model was selected because the physical characteristic of air is estimated were better description by group theory formulation, in addition the phenomena required a description of kinetic energy between wall and inlet conditions, it was possible to get through RNG-model. Equations (8) and (9) calculate the turbulent kinetic energy and the dissipative factor, respectively [18]. Table 2 conceptualizes the model parameters for turbulence.

Table 2: Turbulence model parameters.

PARAMETER	PHYSICAL MEANING
$\kappa - \varepsilon$	Turbulent kinetic energy and dissipation rate
G_k	Generation of turbulent kinetic energy
G_b	Turbulent kinetic energy generation due to buoyancy effects
$\alpha_k - \alpha_\varepsilon$	Inverse of Prandtl number, relating hydrodynamic and thermal boundary layers
S_ε	Source term

$$\frac{\partial(\rho\kappa)}{\partial t}+\frac{\partial(\rho\kappa u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left(\alpha_k\mu_{eff}\frac{\partial k}{\partial x_j}\right)+G_k+G_b-\rho\varepsilon-Y_M+S_k \tag{7}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial\rho\varepsilon u_i}{\partial x_i}=\frac{\partial}{\partial x_j}\left(\alpha_\varepsilon\mu_{eff}\frac{\partial\varepsilon}{\partial x_j}\right)+C_{1\varepsilon}\frac{\varepsilon}{\kappa}\left(G_k+C_{3\varepsilon}G_b\right)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}-R_\varepsilon+S_\varepsilon \tag{8}$$

1.6. Chemical reaction on neutralization ammonia

The CFD simulation of the hydrocyclone validated the information in the literature on pressure loss, as well as the profiles of velocity, pressure, turbulent intensity, and drift properties. Currently, the industrial application of this equipment is centered on the separation of particulate materials in gaseous streams. However, by geometrically evaluating the current lines formed during operation together with the residence time that the gas stream remains inside the equipment, we investigated the possibility of using the hydro cyclone in addition to separating particulate material, the technological applicability for chemical reactions, expanding the functionality of the hydro cyclone to chemical reactors [19].

This observation has been seen structurally in that the number of turns that the fluid travels to induce separation is close to the behavior of a tubular reactor, or Plug Flow Reactor (PFR), in which the reaction and mass transfer phenomena occur as a function of the length of the reactor, which would be correlated with the current example, the number of turns of the hydro cyclone. Tubular reactors are widely applied for gaseous reactions in which the concentration profile is predominantly in the axial direction [20].

After calculating the pressure loss, this value was incorporated into the dynamic simulation of a PFR reactor in which the geometric dimensions were obtained as a function of the diameter, number of turns and height of the hydro cyclone. The pressure loss is due to surface friction together with the volumetric variation throughout the reaction, as the gaseous ammonia reacts with the liquid acetic acid, producing ammonium acetate in solution [21, 22]. The reaction is described below:

$$NH_{3_{(g)}}+CH_3COOH_{(l)}\rightarrow NH_4CH_3COO_{(l)} \tag{9}$$

It can be concluded from Equation (6), which explains the neutralization reaction, that the phenomenon of mass transfer from ammonia to the liquid stream (acetic acid) and consequently the formation of ammonium acetate, occurs with a reduction in the volume flow in the reactor, both due to diffusion and mass convection, as well as the pressure loss determined by the simulation.

2. MATERIALS AND METHODS

For CFD analysis of fluids flow in a Lapple-type cyclone, the commercial software ANSYS Fluent was used. The elaboration of the geometry or CAD drawing of the separator with the relative measurements are available in Table 3. Figure 1 points out the resulting geometry on the left side while the discretized mesh on the right, in isometric view.

It can be seen that the resulting mesh is classified as structured, composed of hexahedral elements with the red and blue arrows indicating the output and input, respectively. The parameters for the mesh fit

Table 3: Dimensions of the cyclone separator.

GEOMETRIC MEASUREMENTS DATA	SIZE	DATA	SIZE
a	1.21 m	H	9.68 m
b	0.605 m	h	4.84 m
De	1.21 m	S	1.5 m
B	0.605 m		

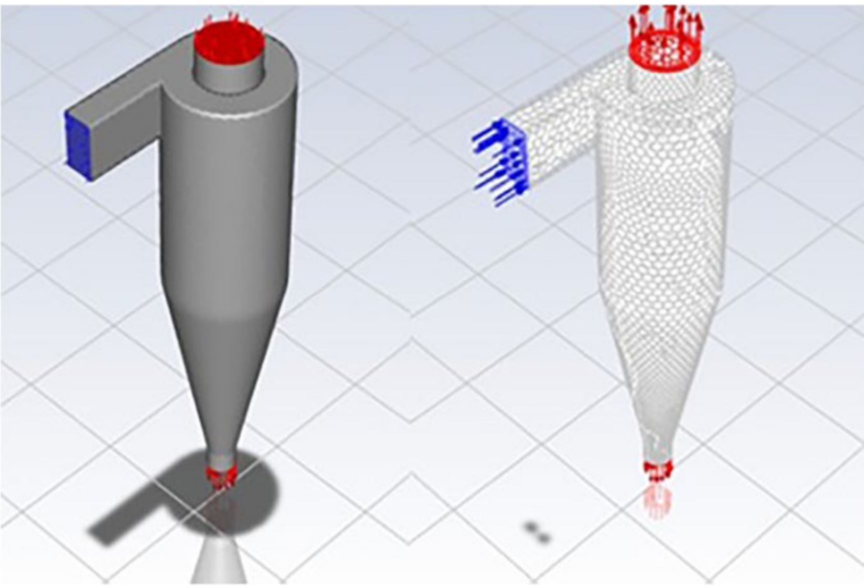

Figure 1: Cyclone geometry and mesh.

are: element size between 0.005 and 0.2 meters; growth rate at 1.2; normal angle to curve of 1.2 and high transitions and smoothing. The simulation set was based on pressure boundary (1000 < ΔP < 2000) measured in Pascal, with direct observance with separation efficiency, between 70 to 80%. The operating conditions of the cyclone are: flow rate (Q = 6.135 m^3/s), specific mass of the gas (ρg = 1.2 kg/m^3) and main diameter (D = 2.42 m).

2.1. Boundary conditions

The following boundary conditions describe the phenomenon of separation of suspended particles in the gas stream, considering the operation conditions at real cyclone installed at Rio de Janeiro – Brazil.

- Inlet: Specific velocity of the gas inlet inside the equipment at 8.24 m/s;
- Out_overflow: Outlet pressure of the gas stream, setting the gauge pressure to 0 Pascal assigning the distribution only in the radial direction of the system, considering the axial and angular negligible and specifying the turbulence in relation to the hydraulic diameter of backflow at the value of 1.21 meters and intensity of 5%;
- Out_underflow: Particle inlet pressure, considering null gauge pressure and assuming the value of backflow hydraulic diameter at 0.605 meters at the same turbulence intensity;
- Wall condition, i.e. the velocity is zero at this position.

Table 4: Tunable parameters for the turbulence model.

PARAMETER	VALUE
C2 – ε	1,9
TKE Prandlt Number	1
TDR Prandlt Number	1,2
Turbulence Intensity	5%
Turbulent Viscosity Ratio	10

2.2. Corresponding models

The momentum conservation equation is solved predominantly in fluid flow analysis, and can be expressed according to fluid compressibility, geometry typology and coupled in several differential models. For the Lapple-type cyclone, the following criteria were employed for the pressure, velocity, transient, and force balance terms:

- Model solved in Pressure class – Based by the characteristic of the fluid dynamics;
- Air velocity evaluated in an absolute way;
- Steady state simulation, with spatial discretization only;
- The gravitational force was inserted in the −Y axis, with a value of 9.8 m/s^2
- The specific mass and kinetic viscosity for air are: $\rho = 1.2$ kg/m^3 e $\mu = 1.7894 \times 10^{-5}$ Pa.s, casting to the assumptions of incompressible flow and Newtonian fluid.

To obtain information about the turbulence, we used the $\kappa - \varepsilon$ model with two equations, with insertions of the respective tunable parameters listed in Table 4.

2.3. Dynamic model to plug flow reactor

The dynamic model obtained for the reactor (hydro cyclone) began with the mass balance with a transient chemical reaction, listing the following hypotheses:

1. Isothermal reactor – Effects of energy release or consumption in the system were disregarded, as the enthalpies of formation of the products were not significant.
2. Perfect mixing reactor – It was assumed that the concentration of the chemical species changes as a function of time, and that they are in a steady state throughout the reactor.
3. Elementary reaction – An elementary reaction is defined as one in which the stoichiometric coeficient is equal to the concentration exponent in the reaction rate, derived from the thermodynamic concept of chemical activity.
4. The equimolar supply of reactants.
5. Considering the gas phase, there is a change in volumetric flow throughout the reactor, which is accounted for by pressure loss and volume variation, unified by the parameter (ε).

$$\left[L + C_{ao}\varepsilon\frac{V_o}{V}\right]\frac{dX}{dt} = C_{ao}\frac{Q_o}{V} - C_{ao}\frac{Q}{V}\frac{(1-X)}{(1+\varepsilon X)} - kC_{ao}^2\frac{(1-X)^2}{(1+\varepsilon X)^2} \quad (10)$$

3. RESULTS AND DISCUSSIONS

After complete convergence and minimization of the residuals in the components evaluated in the simulation, the profile graphs for each targeted physics are obtained.

3.1. Velocity

It is possible to observe initially the blue line in the central radial axis: facing the symmetry of the separator the boundary condition with zero variation in the center is identified in the separation, predominantly by the descending current that when receiving the action of tangential force accelerates the particles in suspension to the extremes, while in the center the velocity tends to a value close to zero. The profile also identifies the highest velocities near the walls, a condition which is visualized by the reaction of the tangential force inducing acceleration and consequent increase in kinetic energy in the collision, as can be observed on Figure 2. The entrance regions of the gas stream with velocities around 17 m/s are the highest observed, due to the presence of the intermediate plate to support the equipment. The gradient at axial direction it is the principle forces that acting on volume control to promote the separation of gas/solids solutions.

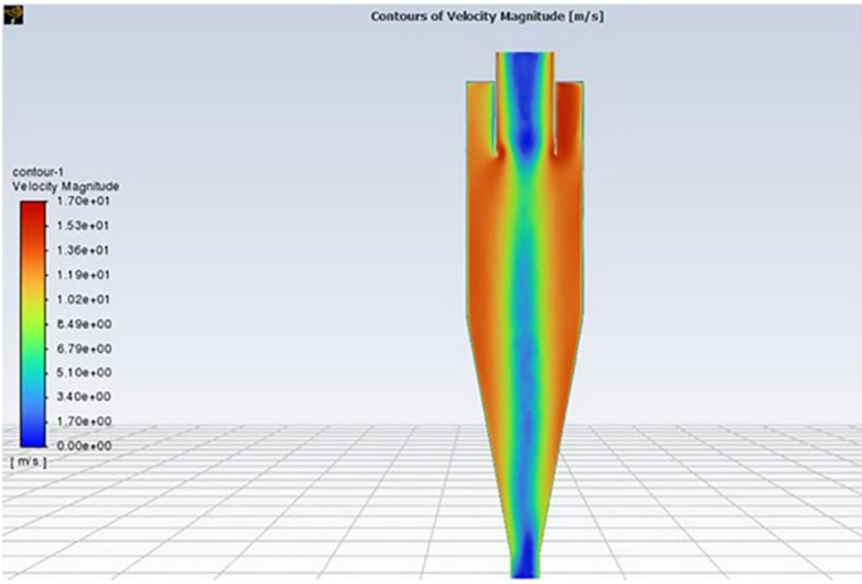

Figure 2: Contour velocity.

3.2. Pressure

The pressure distribution profile is compared in different perspectives along five views, available in Figure 3. Inserting the comparison reference from left to right, the pressure lines (analogously to the current lines) seen in view (1) and view (4) the radial pressure distribution with higher values near the wall. The energy present in the elastic collision is transferred as pressure by the abrupt stagnation of velocity, for view (5) at the same figure, this magnitude is seen as higher values at the walls and lower in the center. Contextualizing with the solution of partial differential equations (PDE), the PDE is characterized as parabolic, justifying the calculated profile.

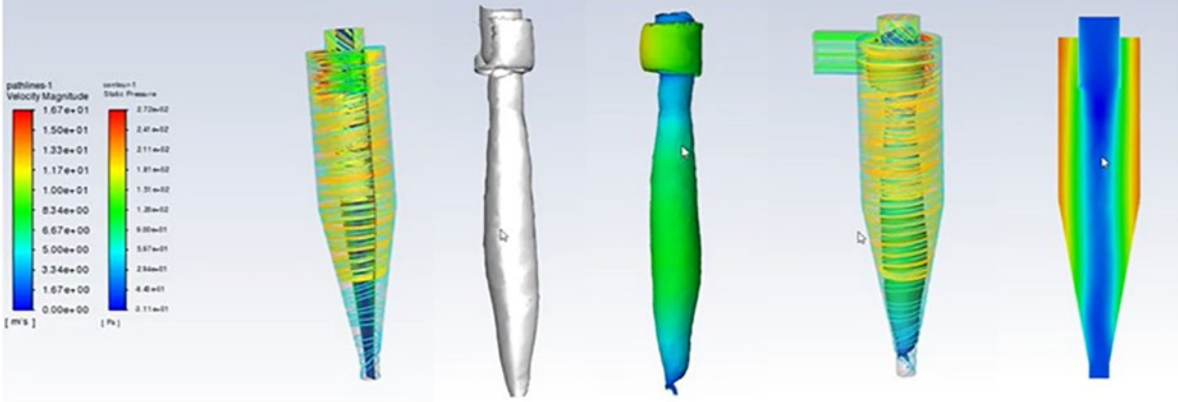

Figure 3: Pressure profile.

The pressure at the cyclone outlet is null, the pressure drop is known to be given by the value of the pressure at the inlet, the value of the pressure drop is 324.76 Pa, available in literature. Comparing the simulated value (ΔP_s) and the one stipulated by literature (ΔP_l), a relative error of 0.44% is obtained, illustrated in equation 4 ratifying the potentiality of the numerical method for the solution of the equations of conservation of quantity of movement and turbulence, the error is determined by the expression below:

$$E_r = \frac{\Delta P_l - \Delta P_s}{\Delta P_s} \times 100 = \frac{326.21 - 324.76}{324.76} \times 100 = 0.44\% \tag{11}$$

3.3. Turbulency

Figure 4(a) shows the turbulence intensity profile, measured in %, along the equipment. This parameter is important to analyze since the goal of the process is to improve the separation of the particles. It is possible to observe the vortices in the inlet current with the formation of two turbulent hydrodynamic layers with symmetrical values in mirror images, an expected result when dealing with circular geometries, with dissipation at the base of the cone. A critical point is also observed, represented in red, which is the location of structural monitoring of the cyclone over time, because with frequent use, the region with the highest concentration can induce structural stresses with consequent damage.

Figure 4(b) depicts the turbulent Reynolds number profile. Larger values are observed in the central axis of the cyclone with vertical profile approaching both boundary layers with average values, represented in green in contrast to the increase of turbulence in the initial cyclone part. The decrease is expected to promote particle separation.

3.4. Technology application – dynamic modeling of neutralization ammonia reaction on the hydrocyclone

After modeling the PFR reactor considering hypothesis listed, the follow step it was calculated the parameters to run the dynamic model. Final and initial volumes it was selected by practical experience on equipment available at industrial sales. The ammonia concentration was determined by units' conversion considering a mole fraction at commercial fluid. Expansion factor is estimated by chemical reaction, thought product of mole fraction (0.5 because the equimolar hypothesis) and the variations by stoichiometry (+1−1−1 = −1), get ε = 0.5. Flow rate interval took into account the operation curve of pump available. The Table 5 resume the parameters and variables used at dynamic model to PFR reactor. On Figure 5 shows the conversion profile of reaction. It is possible to observed that a 92% of ammonia was neutralized by acid acetic reaction, taking 3.5 minutes to get these results. The Figure 6 shown the dynamic structured at Simulink™, where each block represents a function, came as: products, sum, integral and plot.

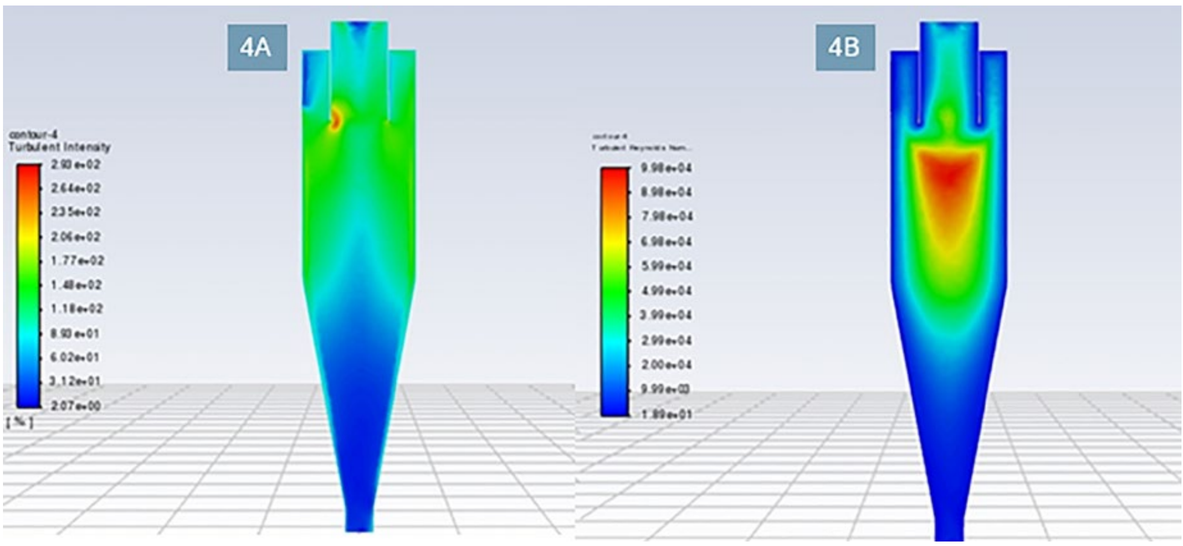

Figure 4: (a) Turbulent intensity, (b) turbulent Reynolds number.

Table 5. Dynamic model parameters.

PARAMETER	VALUE	UNIT	PHYSICAL MEANING
L	$-2\varepsilon - 2\varepsilon^2$	[–]	Flow rate variation
C_{ao}	48,71	[mol/m^3]	Concentration NH_3
ε	–0.5	[–]	Expansion fator
V_o	35	[m^3]	Initial Volume
X	–	[–]	Conversion
Q_o	4.3	[m^3/min]	Initial flow rate
Q	6.135	[m^3/min]	Final flow rate
V	40	m^3	Final Volume
t	–	[min]	Time

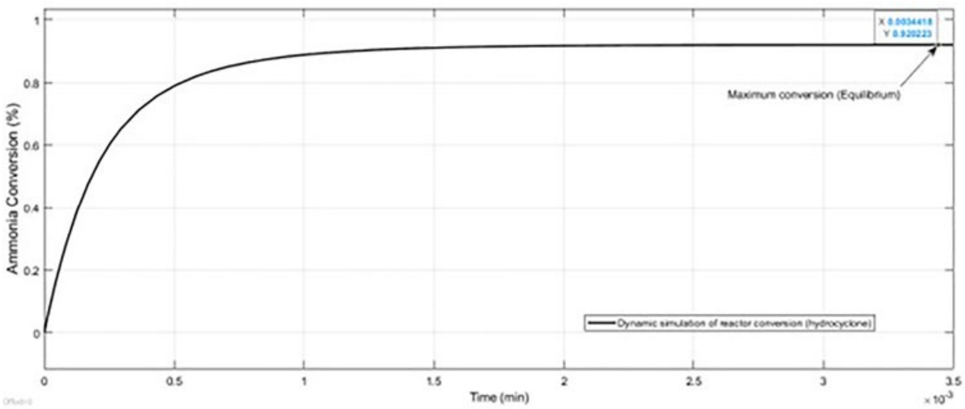

Figure 5: Conversion profile of reaction on the reactor.

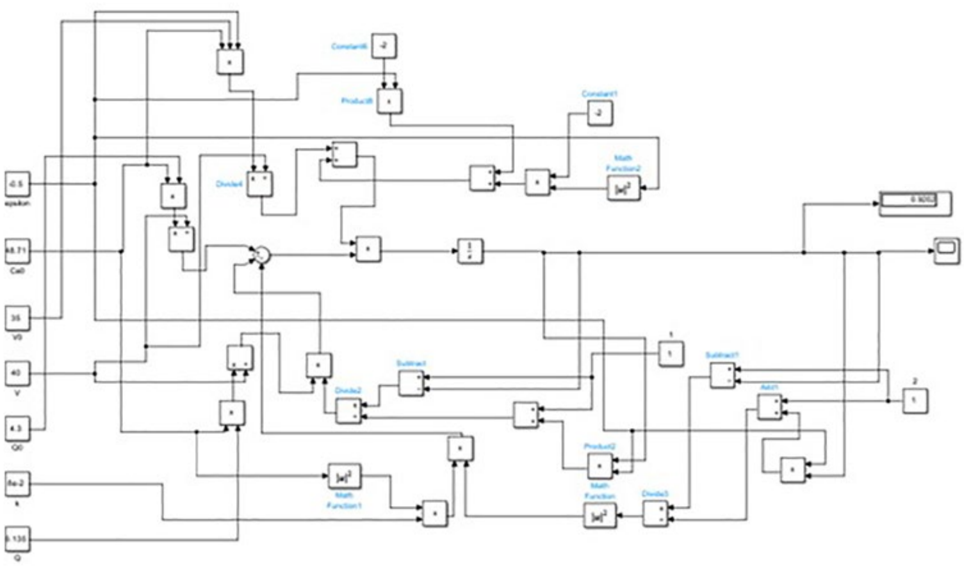

Figure 6: Block diagram to dynamic reactor.

4. CONCLUSIONS

The aim of this study was to simulate fluid flow in a Lapple cyclone and compare the results with experimental data from the literature, using the dimensions in Table 4 as a reference for evaluating flow characteristics such as pressure, velocity, and turbulent kinetic energy. According to Equation 11, the calculated pressure drop was 324.76 Pa, with a relative percentage error of 0.44%. The application of CFD (Computational Fluid Dynamics) techniques for analyzing fluid flow in the Lapple-type cyclone yielded results that were highly consistent with empirical data from the literature. This made it possible to investigate flow behaviors that are difficult to measure experimentally. After validating the pressure loss data, the study explored the feasibility of using a hydrocyclone for processing a chemical neutralization reaction. The motivation for this investigation was safety concerns in industrial plants that use ammonia as a refrigerant, particularly in the food industry, where safety systems are designed to direct toxic ammonia gas into a reaction with acetic acid for neutralization. To support this, the study reviewed literature on the kinetics of the ammonia-acetic acid neutralization reaction and the availability of relevant kinetic constants, and dynamic modeling was applied using concentrated parameters. By considering the relevant assumptions, Equation (10) was derived and numerically solved using Simulink™. The dynamic evaluation showed that the hydrocyclone is a plausible system for carrying out chemical reactions, with a high conversion rate of approximately 92%, despite challenges such as pressure drop, mass transfer resistances between phases, and variations in flow and volumetric phenomena. This study presents a novel concept for the use of hydrocyclones in chemical processes, which could be tested on a prototype for further investigation.

5. ACKNOWLEDGEMENTS

Those involved.

6. BIBLIOGRAPHY

[1] HAN, S., YANG, J., ZHANG, R., *et al*., "Forming and breaking the ceiling of inlet gas velocity regarding to separation efficiency of cyclone", *Particuology*, v. 79, pp. 85–94, 2023. doi: http://doi.org/10.1016/j.partic.2022.11.004.

[2] WANG, Z., SUN, G., SONG, Z., *et al*., "Effect of inlet volute wrap angle on the flow field and performance of double inlet gas cyclones", *Particuology*, v. 77, pp. 29–36, 2023. doi: http://doi.org/10.1016/j.partic.2022.08.006.

[3] MISIULIA, D., LIDÉN, G., ANTONYUK, S., "Secondary lip flow in a cyclone separator", *Flow, Turbulence and Combustion*, v. 110, n. 3, pp. 581–600, 2023. doi: http://doi.org/10.1007/s10494-023-00395-5.

[4] BALTRĖNAS, P., CHLEBNIKOVAS, A., "Numerical study of the aerodynamic parameters in a two-level multichannel cyclone", *Separation Science and Technology*, v. 51, n. 12, pp. 2105–2113, 2016. doi: http://doi.org/10.1080/01496395.2016.1201112.

[5] GAO, Z., WANG, J., LIU, Z., *et al*., "Effects of different inlet structures on the flow field of cyclone separators", *Powder Technology*, v. 372, pp. 519–531, 2020. doi: http://doi.org/10.1016/j.powtec.2020.06.014.

[6] SONG, J., HE, Y., MENG, F., *et al*., "Effects of gas in-leakage on separation performance and gas-phase flow field of a FCC cyclone", *Powder Technology*, v. 415, pp. 118201, 2023. doi: http://doi.org/10.1016/j.powtec.2022.118201.

[7] ZHANG, L., FAN, J., ZHANG, P., *et al*., "Effect of local erosion on the flow field and separation performance of the cyclone separator", *Powder Technology*, v. 413, pp. 118007, 2023. doi: http://doi.org/10.1016/j.powtec.2022.118007.

[8] LI, W., HUANG, Z., LI, G., "Improvement of the cyclone separator performance by the wedge-shaped roof: A multi-objective optimization study", *Chemical Engineering Science*, v. 268, pp. 118404, 2023. doi: http://doi.org/10.1016/j.ces.2022.118404.

[9] CHAGHAKABOODI, H.A., SAIDI, M., "Numerical study of gas-solid flow in a square cyclone separator with different vortex finders", *Chemical Engineering Research & Design*, v. 194, pp. 621–635, 2023. doi: http://doi.org/10.1016/j.cherd.2023.05.001.

[10] LI, T., SUN, Z., GENG, K., *et al*., "Numerical analysis of a novel cascading gas-liquid cyclone separator", *Chemical Engineering Science*, v. 270, pp. 118518, 2023. doi: http://doi.org/10.1016/j.ces.2023.118518.

[11] ZHU, L., WANG, S., RU, Y., *et al*., "Numerical investigation on dynamic characteristics of flow field in cyclone separators with different dust hopper structures", *Particuology*, v. 82, pp. 134–145, 2023. doi: http://doi.org/10.1016/j.partic.2023.01.016.

[12] LE, D.K., YOON, J.Y., "A hybrid CFD-Deep learning methodology for improving the accuracy of pressure drop prediction in cyclone separators", *Chemical Engineering Research & Design*, v. 190, pp. 296–311, 2023. doi: http://doi.org/10.1016/j.cherd.2022.12.035.

[13] BUMRUNGTHAICHAICHAN, E., "A note of caution on numerical scheme selection: evidence from cyclone separator CFD simulations with appropriate near-wall grid sizes", *Powder Technology*, v. 427, pp. 118713, 2023. doi: http://doi.org/10.1016/j.powtec.2023.118713.

[14] LEITH, D., MEHTA, D., "Cyclone performance and design", *Atmospheric Environment*, v. 7, n. 5, pp. 527–549, 1973. doi: http://doi.org/10.1016/0004-6981(73)90006-1.

[15] WANG, B., XU, D.L., CHU, K.W., *et al*., "Numerical study of gas-solid flow in a cyclone separator", *Applied Mathematical Modelling*, v. 30, n. 11, pp. 1326–1342, 2006. doi: http://doi.org/10.1016/j.apm.2006.03.011.

[16] FLETCHER, D.F., "The future of computational fluid dynamics (CFD) simulation in the chemical process industries", *Chemical Engineering Research & Design*, v. 187, pp. 299–305, 2022. doi: http://doi.org/10.1016/j.cherd.2022.09.021.

[17] SHENG, W., "A revisit of Navier-Stokes equation", *European Journal of Mechanics - B/Fluids*, v. 80, pp. 60–71, 2020. doi: http://doi.org/10.1016/j.euromechflu.2019.12.005.

[18] WANG, X., HSIEH, Y., TANG, P., *et al*., "A review of the research on productivity in the construction industry", *Advanced Engineering Informatics*, v. 35, pp. 92–108, 2017.

[19] KANG, S.K., KWON, T.W., KIM, S.D., "Hydrodynamic characteristics of cyclone reactors", *Powder Technology*, v. 58, n. 3, pp. 211–220, 1989. doi: http://doi.org/10.1016/0032-5910(89)80116-0.

[20] ZHANG, B., YANG, J., ZHENG, Q., *et al*., "Centrifugal force caused high-density rotating downward quasi-plug flow in cyclone reactors", *Chemical Engineering Science: X*, v. 11, pp. 100101, 2021.

[21] SOMEYA, S., YOSHIDA, S., TABATA, T., *et al*., "The effect of chemical reaction on the mixing flow between aqueous solutions of acetic acid and ammonia", *International Journal of Heat and Mass Transfer*, v. 52, n. 19-20, pp. 4236–4243, 2009. doi: http://doi.org/10.1016/j.ijheatmasstransfer.2009.04.005.

[22] ALOPAEUS, V., LAAVI, H., AITTAMAA, J., "A dynamic model for plug flow reactor state profiles", *Computers & Chemical Engineering*, v. 32, n. 7, pp. 1494–1506, 2008. doi: http://doi.org/10.1016/j.compchemeng.2007.06.025.