

**COB-2023-1167**

## **STUDY OF INORGANIC FOULING DEPOSITION IN A HYDROCYCLONE: COMPARISON BETWEEN NUMERICAL SIMULATION AND EXPERIMENTAL RESULTS**

**Andrei Hünemeyer Dullius \***

**Juliana Ferreira Gonçalves**

**Andressa Amorim Daás**

**Mariana Fernanda Rio Dominguez**

**Juliana Braga Rodrigues Loureiro**

Interdisciplinary Centre for Fluid Dynamics (NIDF)/ COPPE / Federal University of Rio de Janeiro (UFRJ), Rua Moniz Aragão, 360 – Bloco 2, Cidade Universitária, Rio de Janeiro.

\*andreidullius@mecanica.coppe.ufrj.br

**Fabricio Soares da Silva**

PETROBRAS – CENPES/PDIP/ESUP/TPMF, Avenida Horácio Macedo, 950 – Cidade Universitária, Rio de Janeiro.

fabriciosoares@petrobras.com.br

**Abstract** Calcium carbonate fouling formation is a great challenge faced by the oil and gas industry in Brazil, occurring throughout the production process. The extreme conditions of production, the high content of salinity and CO<sub>2</sub> in the formation water, along with the types of rock reservoir found in the pre-salt layer, highly favor the occurrence of this phenomenon. Notably, de-oiling hydrocyclones, employed in the treatment of wastewater during the later stages of production, have their efficiency highly compromised due to fouling formation, possibly leading to increased pressure drop and complete blockage. Accurate prediction of the fouling phenomenon in hydrocyclones requires a multitude of approaches, given the complexity of the flow within the equipment and the difficulty in describing the fouling process itself. The flow inside the equipment with its characteristic opposing inner and outer vortices may direct the fouling particles formed in the bulk flow towards the wall, thus favoring the phenomenon of particle adhesion. The comparison between experimental results and numerical simulations is explored in this work, with the goal of providing a practical description of the phenomenon of fouling in hydrocyclones. In this study, we adopt the well-established Ning and Thornton model for adhesive particles in the numerical simulation of wall-deposit formation in a hydrocyclone, in order to investigate the dynamics of fouling in a complex turbulent flow. The velocity field is provided by a Reynolds-averaged Navier-Stokes (RANS) computation calculated in OpenFoam. The geometry of the model, the input for pressure and velocity in the simulations, as well as the number and size distribution of particles are all taken from the experiments carried out in the laboratory with a hydrocyclone prototype under conditions very similar to real field applications. The experiments consist of inducing calcium carbonate fouling formation through the mixture of sodium bicarbonate and calcium chloride solutions in a pipe circuit, leading to an encapsulated hydrocyclone. The results obtained by the simulations provide further understanding of the results found in the experiments, giving a clearer picture of the fouling process in the hydrocyclone. The results show a higher concentration of particles in the underflow, where the denser phase exits the equipment.

**Keywords:** Calcium carbonate, fouling, hydrocyclone, experiments, numerical simulations.

### **1. INTRODUCTION**

Fouling formation can be defined as the buildup of unwanted material on the surface of an equipment or component, resulting in compromised functionality and efficiency. This phenomenon is seen in a variety of technological applications, ranging from heat exchangers (Bott, 1995), to wastewater treatment (Le-Clech et al., 2006) and in hulls of ships (Farrapeira, 2007). Specifically in the oil and gas industry, this phenomenon is observed throughout the production process, leading to obstructions in the production line and frequent stops for maintenance. The mixing of incompatible waters with a high concentration of low solubility salts, the types of rock reservoir found in the pre-salt layer of Brazilian offshore production and the high content of CO<sub>2</sub> highly favor inorganic fouling formation.

During the later stages of production, de-oiling hydrocyclones are employed in the treatment of wastewater, usually separating a mixture with a low concentration of oil. A hydrocyclone is part of a general class of cyclonic equipment used to separate two phases based on a density difference. A hydrocyclone is used in the separation of two fluid phases, making use of the centrifugal effect as a result of the flow within the equipment. A hydrocyclone is composed of two tangential inlets connected to a cylindrical section, followed by a conic section and a straight section. The equipment has two outlets,

called the underflow and overflow, collecting each of the separated phases. The mixture enters the equipment through the tangential inlets inducing a rotational motion of the fluid. Due to an inertial effect, the heavier phase is directed towards the wall, while the lighter phase stays in the center. The conic section maintains the rotational motion of the flow and imposes an important restriction to the flow, as it reduces the cross sectional area of the flow redirecting part of the flow to the opposing outlet, thus separating the two phases (Svarovski, 2000).

The phenomenon of fouling formation in a hydrocyclone leads to the blockage of the inlets and outlets, increasing the pressure drop and eventually making the flow bypass the hydrocyclones.

Numerical simulations of the fouling phenomenon can be a useful tool in the prediction of fouling in systems and equipment, providing useful information to improve geometries and processes. Borello et al. (2012), applied the Ning and Thornton model to simulate fouling formation in a subsonic compressor, and compared the results with experiments. The authors verified that the Ning and Thornton model applied to this problem is able to predict qualitatively the preferential sites at which the onset of fouling is given. Similarly, other studies made use of the Ning and Thornton model applied to the fouling problem with satisfactory results (Abd-elhady et al., 2004, Borello et al., 2009).

In this study, we evaluate the phenomenon of calcium carbonate formation in a hydrocyclone through the conjunction of experimental and numerical results with the aim of validating the Ning and Thornton model for the stick/bounce behavior of particles when applied to this problem.

## 2. EXPERIMENTS

In this section, we describe the experimental procedure and the apparatus used. All of these experiments were carried out at the Interdisciplinary Center for Fluid Dynamics (NIDF). The basic idea of the experiment is to induce calcium carbonate fouling through the mixture of sodium bicarbonate and calcium chloride solutions in a pipe circuit preceding an encapsulated hydrocyclone.

The apparatus used is shown in Fig. 1. The saline solutions are prepared and stored in two 2.8 m<sup>3</sup> tanks with a concentration of 0.00735 kg/m<sup>3</sup> and 0.01226 kg/m<sup>3</sup>, for calcium chloride and sodium bicarbonate, respectively. Two Progressive Cavity Pumps (PCP) feed the pipe circuit, promoting the mixture of the saline solutions in a specially designed mixer, resulting in calcium carbonate formation in the line. Three pressure transducers and four flow meters are placed along the line for data collection. Removable tube sections are placed along the line, so that once the experiment is over, solid samples can be collected and further analyzed. Liquid samples can also be taken during experiment, through withdrawal valves located along the line. Two valves are placed on either outlet of the capsule, with the goal of inducing the flow rate and pressure the initial conditions.

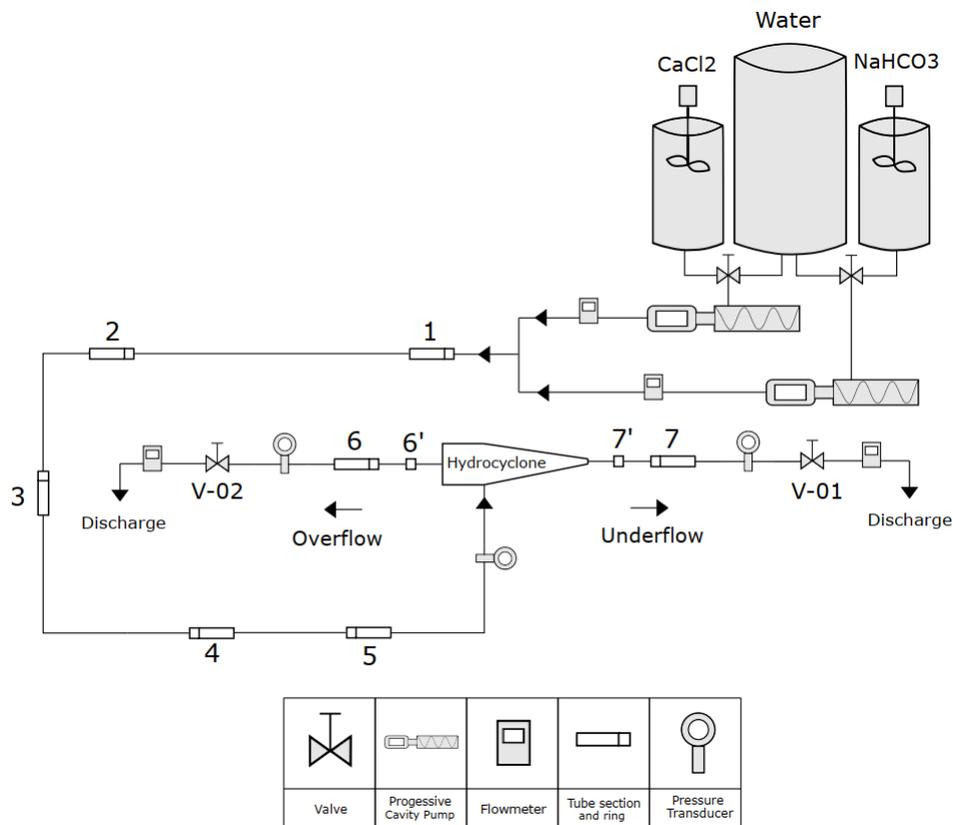


Figure 1 – Schematic diagram of the experimental setup.

The initial conditions for the experiments are chosen to replicate the real field conditions of operation of a hydrocyclone in the production platform. The initial conditions are obtained through the simultaneous manipulation of the valves. A pressure of  $8 \times 10^5 Pa$  is imposed in the inlet of the capsule, and a split of 2% is imposed for the flow rate in the overflow of the hydrocyclone. The flow rate in each PCP is maintained at  $0.000278 \text{ m}^3/\text{s}$ . With that, the flow rate at the overflow is kept at  $0.0000111 \text{ m}^3/\text{s}$ . The experiment is over once a pressure of 20 bar is reached in the inlet, which is the limiting pressure of the setup. Once the experiment is over, the hydrocyclone, valves and tube sections are removed from the setup, dried and weighed.

The hydrocyclone employed in this study was separated into four pieces, making it possible to evaluate the deposition inside the equipment. Three additional removable tube sections were placed inside the hydrocyclone, as shown in Figure 2, along with the measures of the equipment in Table 1, making it possible to evaluate the morphology of the formation through Scanning Electron Microscopy (SEM). Fig. 3 shows the hydrocyclone disassembled.

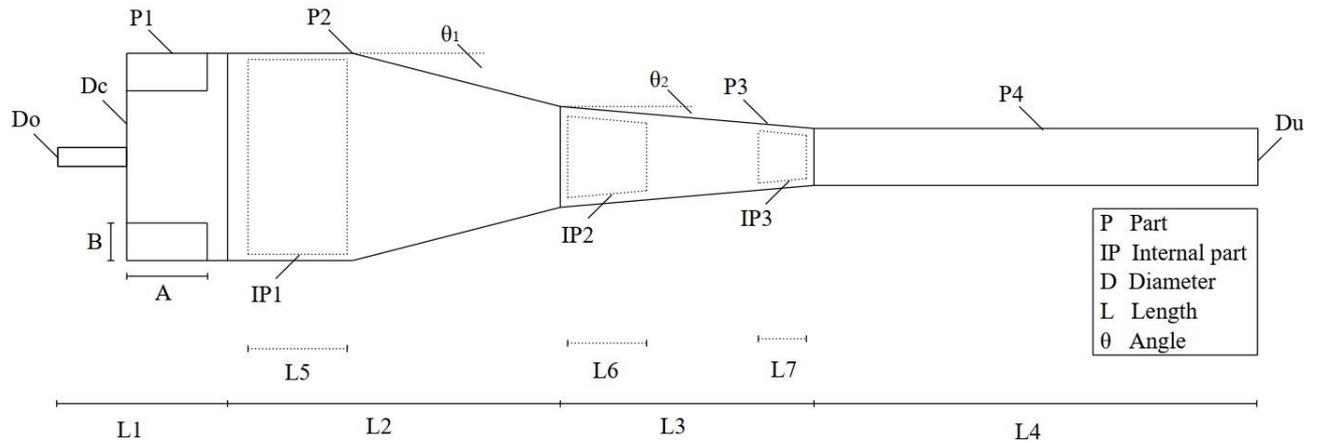


Figure 2 – Test sections of the hydrocyclone.

Table 1 – Characteristics of the investigated hydrocyclone.

Component	Symbol	Value
Length of part 1 [m]	L1	0.042
Length of part 2 [m]	L2	0.200
Length of part 3 [m]	L3	0.136
Length of part 4 [m]	L4	0.560
Length of internal part 1 [m]	L5	0.026
Length of internal part 2 [m]	L6	0.017
Length of internal part 3 [m]	L7	0.018
Overflow diameter [m]	Do	0.002
Underflow diameter [m]	Du	0.0116
Cylinder diameter [m]	Dc	0.040
Greater angle [°]	$\theta_1$	15
Smaller angle [°]	$\theta_2$	1
Inlet height	A	0.013
Inlet width	B	0.0059

### 3. NUMERICAL SIMULATIONS

The numerical simulations were carried out in OpenFOAM and later compared to the experimental results in order to validate the hypothesis and simplifications made in its execution. In this study, we have chosen a  $\kappa$ - $\omega$  SST model with

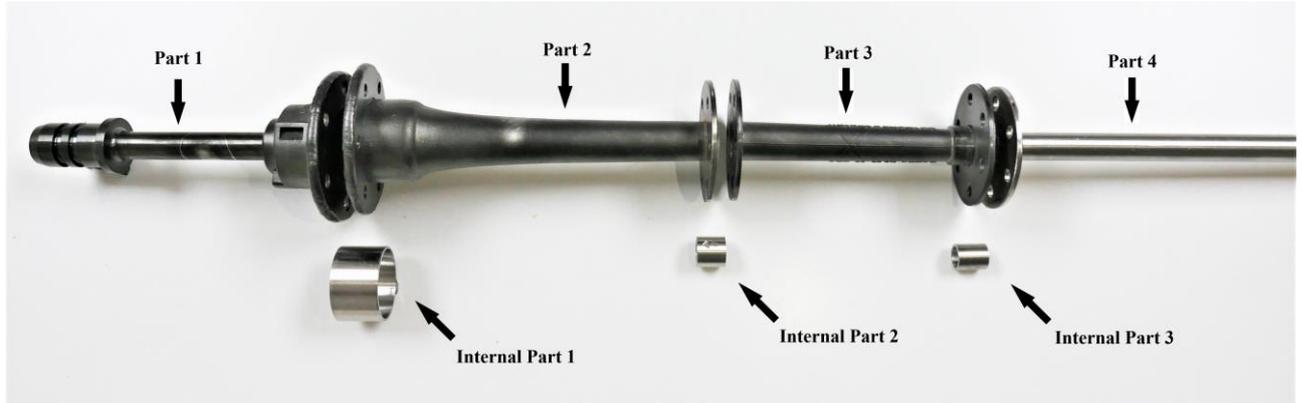


Figure 3 – Photo of the hydrocyclone

no curvature correction to calculate the velocity field. The particle dynamics is solved through a one-way coupling to the continuous field using a Lagrangian approach.

The mesh was generated using the meshing tool snappyHexMesh, available in OpenFOAM. The final mesh is composed of approximately 1.5 million hexahedral elements.

For the Eulerian phase, the solver pimpleFOAM was employed. The boundary conditions for pressure and velocity are summarized in Table 2. The velocity condition for the inlets was chosen to match the condition of flow rate in the experiments. The pressure conditions on the outlets are chosen to match the initial pressure drop imposed in the hydrocyclone during the experiments.

Table 2 – Boundary conditions for the numerical simulations.

Position	Pressure	Velocity
Inlets	Zero gradient	3.6 m/s
Overflow	$4.65 \times 10^5$ Pa	Zero gradient
Underflow	$6.05 \times 10^5$ Pa	Zero gradient
Walls	Zero gradient	No slip

For the Lagrangian phase, the solver particleFOAM was employed. The deposition model of Ning and Thornton was adopted in this study. The authors proposed a model based on the coefficient of restitution ( $e$ ), defined as the ratio between the rebound ( $v_{r,n}$ ) and impact ( $v_{i,n}$ ) normal velocities, shown in Equation 1.

$$e = \frac{v_{r,n}}{v_{i,n}} \quad (1)$$

This ratio can be rewritten in terms of a more explicit form for the kinetic energy ( $\Delta K$ ) before and after the impact. When the coefficient of restitution is zero, the kinetic energy is completely dissipated during impact, leading to the particle sticking to the surface, and defining as a result a sticking velocity ( $v_s$ ), i.e. a velocity below which the particle always sticks to the surface.

$$v_s = 1.84 \left[ \frac{(\Gamma)^5}{\rho_p^2 E^{*2}} \right]^{\frac{1}{6}} \quad (2)$$

The sticking velocity ( $v_s$ ) is a function of the adhesion energy ( $\Gamma$ ), the radius ( $r_p$ ), density ( $\rho_p$ ) and equivalent Young modulus ( $E^*$ ) of the particle.

The Equations for the model are shown below (Equations 3,4 and 5), and are written in terms of the normalized sticking ( $\omega_s$ ) and yielding ( $\omega_y$ ) velocities defined as  $\omega_s = v_s/v_{i,n}$  and  $\omega_y = v_y/v_{i,n}$

$$e = 0 \text{ for } \omega_s \geq 1 \quad (3)$$

$$e = [1 - \omega_s^2]^{\frac{1}{2}} \text{ for } \omega_s < 1 \leq \omega_y \quad (4)$$

$$\left\{ \left( \frac{6\sqrt{3}}{5} \right) \left[ 1 - \frac{1}{6} \omega_y^2 \right] \left[ \frac{\omega_y}{\omega_y + 2\sqrt{\frac{6}{5} - \frac{1}{5} \omega_y^2}}} \right]^{\frac{1}{2}} - \omega_s^2 \right\}^{\frac{1}{2}} \text{ for } \omega_y < 1 \quad (5)$$

So according to this model, when a particle interacts with the walls, it can either stick to the surface, as a result of its impact velocity being lower than the critical velocity, otherwise it rebounds. This model considers only normal impacts on the surface.

#### 4. RESULTS

In this section, we present the results obtained. The particle size distribution obtained for the inlet of the capsule through the analyses of liquid samples taken from 10, 30 and 50 minutes of the experiments are shown in Fig. 4. The liquid samples analyzed to determine particle sizes distribution were taken from withdrawal valves located just prior to the inlet of the capsule, the samples were then analyzed using a commercial Particle Counter.

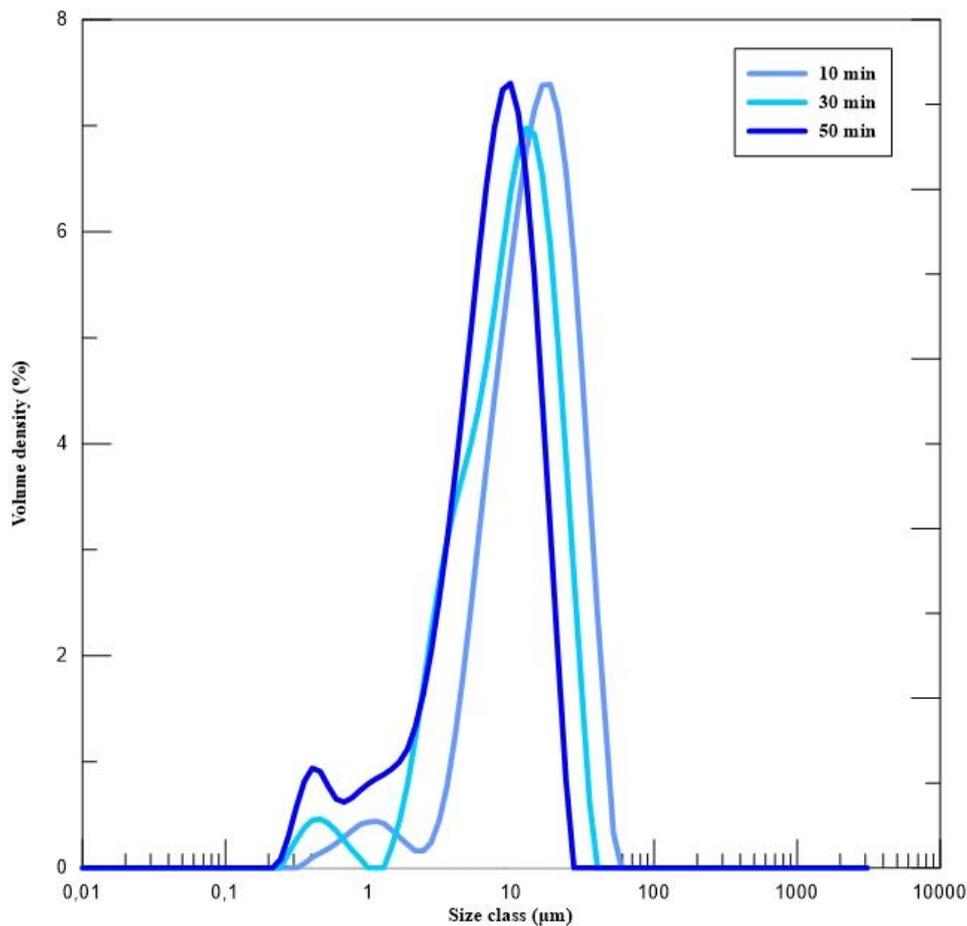


Figure 4 - Size distribution of particles during the experiment.

As observed above, the size distribution does not vary significantly throughout the experiment. The solid concentration for 10, 30 and 50 minutes are 379, 495 and 451 ppm, respectively. With this data, an estimate of the total mass of particles introduced in the system can be obtained. This result will prove valuable in the comparison between the experimental and numerical results, as it is used to normalize the deposited mass.

In Fig. 5, pressure in the inlet of the capsule is shown as a function of time for the two experiments performed. Pressure increases as a result of blockage related to fouling formation until the limiting pressure of  $20 \times 10^5$  Pa. The three abrupt variations in pressure presents in this graph represent the moments at which the liquid samples were collected.

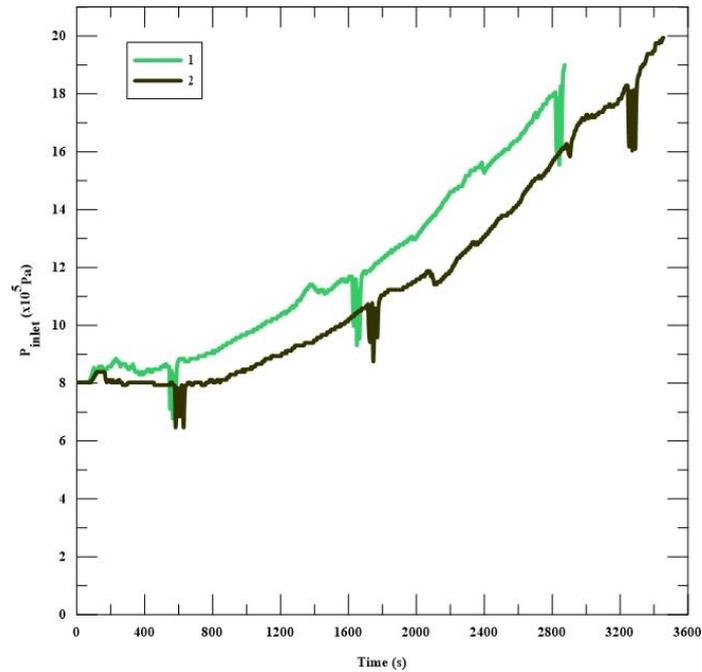


Figure 5- Pressure in the inlet of the capsule as a function of time.

The SEM images obtained for the tube sections in the inlet, underflow, overflow and internal parts of the hydrocyclone are shown in Fig. 6.

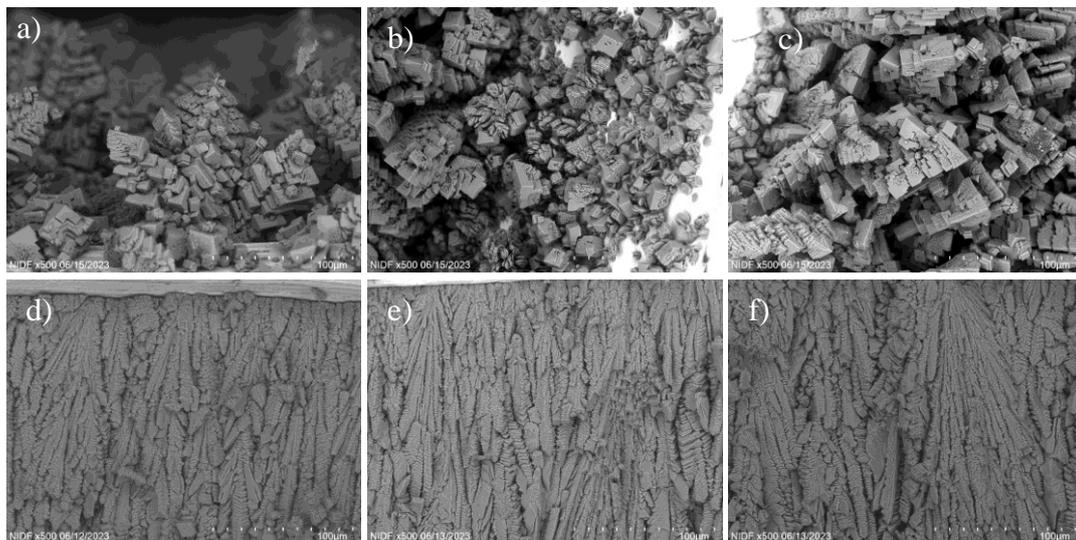


Figure 6 – SEM images. (a) inlet, (b) underflow, (c) overflow, (d) – (e) internal parts 1 – 3.

For the numerical simulations, the velocity profile for tangential and axial velocity in several points along the length of the hydrocyclone is presented. The positions chosen are shown in Fig. 7.

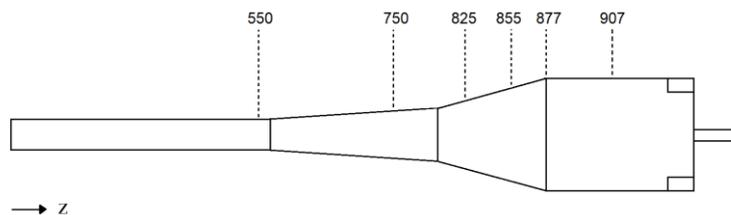


Figure 7 – Positions for the velocity profiles in the hydrocyclone in mm

The axial and tangential velocity profiles for these positions are shown in Fig. 8 and 9.

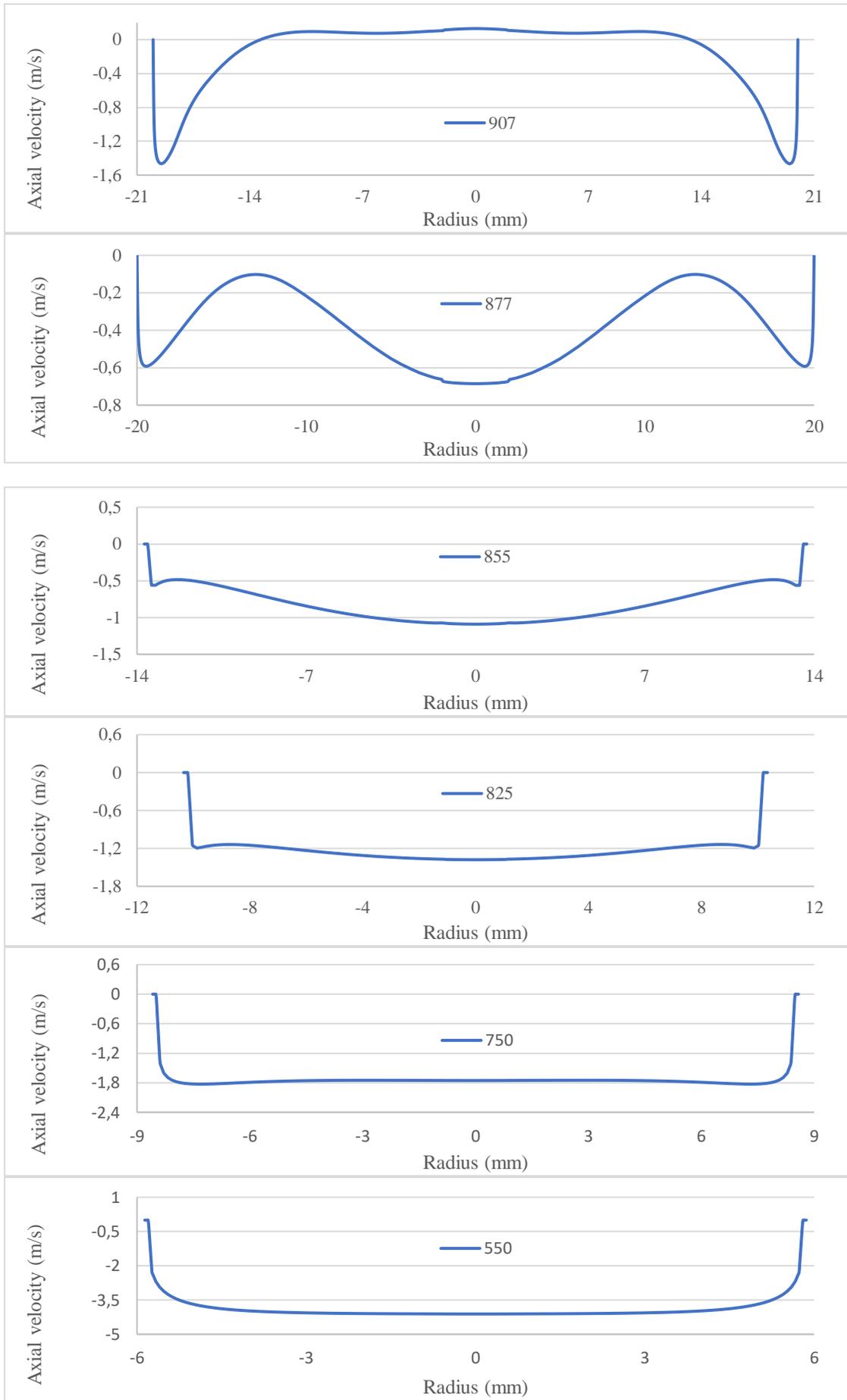


Figure 8 – Axial velocity profile

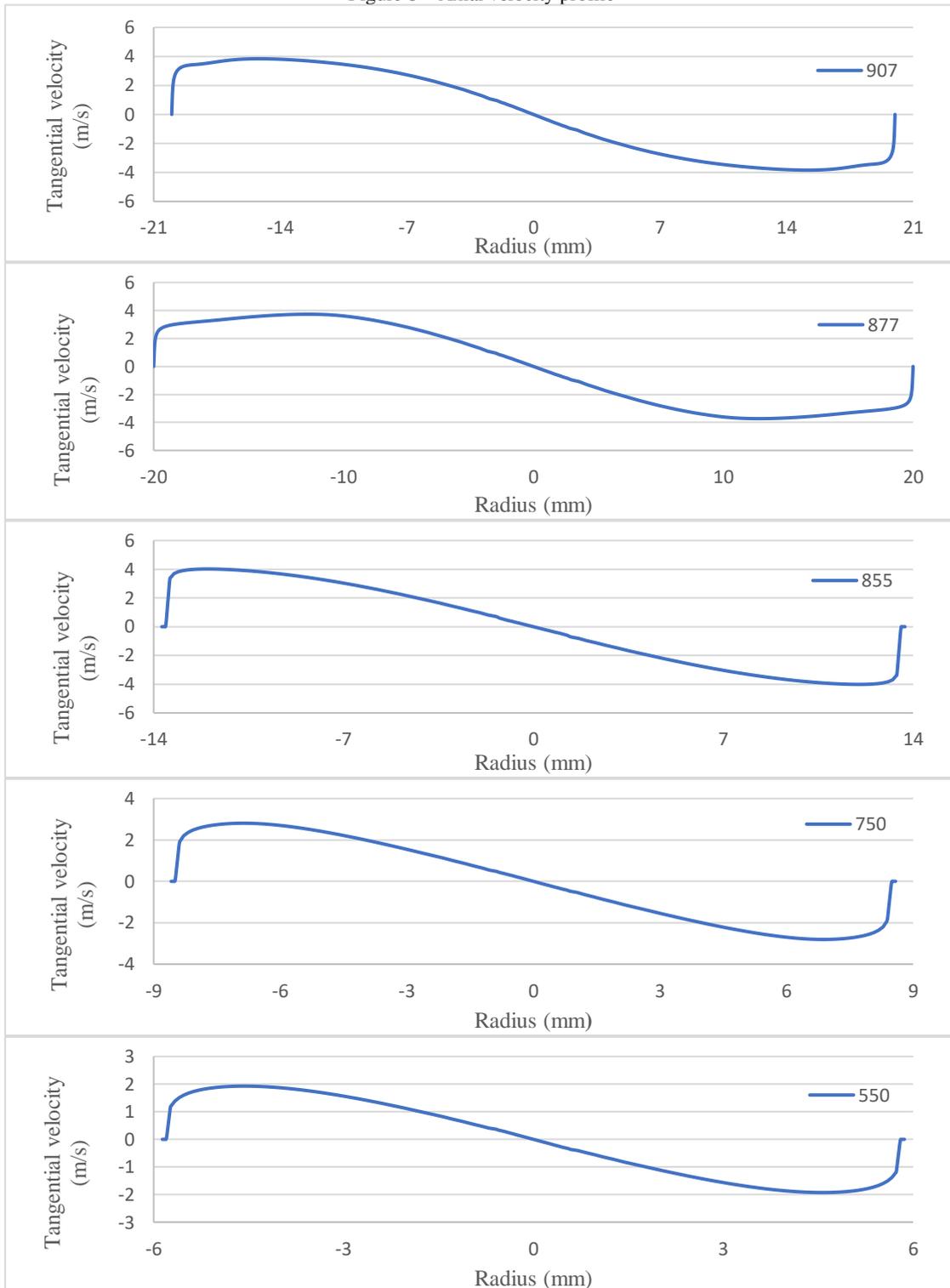


Figure 9 – Tangential velocity profile.

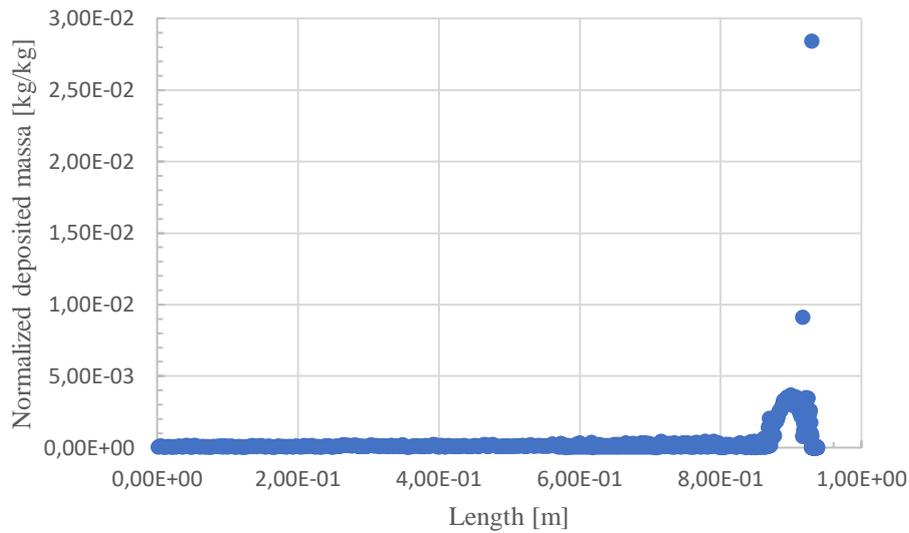


Figure 10 – Normalized deposition along the hydrocyclone.

Table 4 offers a numerical comparison between the results obtained with the experiments and the numerical simulations. The deposited mass in the experiments and the numerical simulations were obtained by dividing the deposited mass by the introduced mass. For the experiments, the introduced mass was obtained through the particle concentration results.

Table 4 – Comparison between the experiments and the numerical simulations.

Part	Experiments – Normalized mass [kg/kg]	Simulations - Normalized mass [kg/kg]
P1	$1.40 \cdot 10^{-3}$	$9.81 \cdot 10^{-2}$
P2	$2.11 \cdot 10^{-3}$	$1.06 \cdot 10^{-1}$
P3	$9.23 \cdot 10^{-4}$	$7.41 \cdot 10^{-3}$
P4	$3.68 \cdot 10^{-3}$	$2.19 \cdot 10^{-2}$
IP1	$5.70 \cdot 10^{-4}$	$2.31 \cdot 10^{-3}$
IP2	$1.52 \cdot 10^{-4}$	$2.21 \cdot 10^{-3}$
.IP3	$1.39 \cdot 10^{-4}$	$1.31 \cdot 10^{-3}$

In all cases, the Ning and Thornton model numerically over predicted deposition by at least one order of magnitude. This could be resulting from a number of factors: the model does not take into account resuspension of particles and given the amount of fouling deposited in the experiments, that is likely to occur; the restitution coefficient  $e$ , chosen as the common value of 0.1 in this study might not reflect the reality of the phenomenon. A higher value of  $e$  would mean particles would bounce back with a higher normal velocity, thus deposition would decrease; the adhesion energy estimated for the calcium carbonate has not been directly measured, thus the estimated value might be incorrect.

Qualitatively the model predicted a high deposition closer to the inlets and almost no deposition on the overflow, which is verified by the experimental results as shown in Fig. 11.

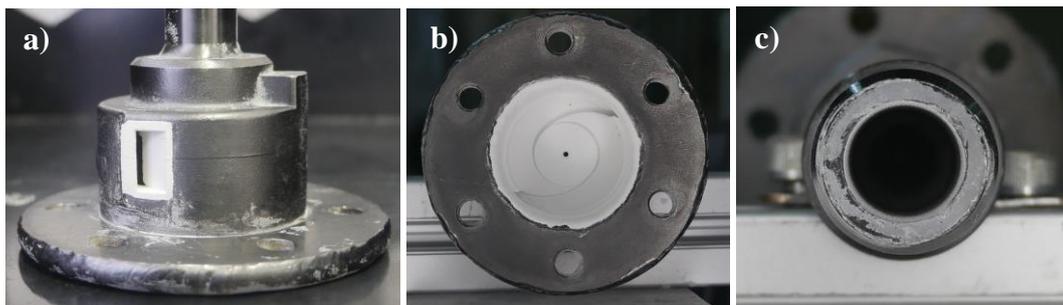


Figure 11 – Photos of the results (a) inlet, (b) part 1, (c) overflow.

## 5. CONCLUSIONS

In this study, we have performed experiments under real field conditions to evaluate calcium carbonate fouling formation in a hydrocyclone. The parameters obtained were used to conduct numerical simulations in OpenFOAM using the Ning and Thornton model to compute the bounce/stick behavior of particles. The results show that the model over predicts fouling in the hydrocyclone, which could be rooted in the limitations of the model or incorrect parameters. This type of study leads to more reliable data as to the behavior of models for fouling formation. Numerical simulations on fouling formation can be a valuable tool in optimizing geometries and processes during the development phase.

## 6. ACKNOWLEDGMENTS

We would like to thank PETROBRAS for all the technical and financial support for carrying out this research. We also thank the Interdisciplinary Center for Fluid Dynamics (NIDF) for providing all the necessary structure for this study to be carried out.

## 7. REFERENCES

- Abd-elhady, M.S., Rindt, C.C.M., Wijers, J.G. , Van Steenhoven, A.A., Bramer, E. A., Van Der Meer, T. H., 2004. "Minimum gas speed in heat exchangers to avoid particulate fouling" *International Journal of Heat and Mass Transfer*. v. 47, pp. 39433955
- Borello, D., Rispoli, F., Venturini, P., 2012. "An Integrated ParticleTracking Impact/Adhesion Model for the Prediction of Fouling in a Subsonic Compressor" *Journal of Engineering for Gas Turbines and Power*. v. 134, pp. 092002(7).
- Borello, D., Rispoli, F., Hanjalic C., Venturini, P., 2009. "A study of wall-deposit formation using LES generated velocity field" 9th International Conference on Turbulence, Heat and Mass Transfer. Roma, Italy.
- Bott, T. R., 1995. *Fouling of Heat Exchangers*. Birmingham, UK, Elsevier Science.
- Farrapeira, C. M. R. Melo, A. V. O. M., Barbosa, D. F., Silva, K. M. E. 2007. "Ship hull fouling in the port of Recife, Pernambuco." *Brazilian Journal of Oceanography*. v. 55, pp. 207-221.
- Le-Clech, P., Chen, V., Fane, T. A. G., 2006. "Fouling in membrane bioreactors used in wastewater treatment." *Journal of membrane science*. v. 28, pp. 17-53
- Svarovsky, L., 2000. *Solid-liquid Separation*. Czech Republic, ButterworthHeinemann
- Thornton, C., Ning, Z., 1998. "A theoretical model for the stick/bounce behavior of adhesive, elastic-plastic spheres" *Powder Technology*. v. 99 pp. 154-162.