CFD SIMULATION OF A HYDROCYCLONE USED AS A MIXER

Isabelly Pereira da Silva, bellynhaaa@msn.com
Gabrielly Pereira da Silva, gabrielly@ufs.br
Mikele Cândida Sousa de Sant'Anna, mikelecandida@gmail.com
Sheila Cristina de Souza Nascimento, shcrisnascimento@yahoo.com.br
Ana Eleonora Almeida Paixão, aepaixão@ufs.br
Gabriel Francisco da Silva, gabriel@ufs.br
Universidade Federal de Sergipe, Laboratório de Tecnologias Alternativas. Av. Marechal Rondon s/n. Bairro Rosa Elze. São Cristóvão/ SE.

Abstract. *The computational fluid dynamics - CFD can be applied in many engineering problems, especially in fluid flow. Simulation of water-oil separation in hydrocyclones, particle separation in cyclones, multiphase separators, multiphase flow, wind turbine, multiphase pumps, valves, washers, mixers, heat exchangers, reactors and bioreactors are the most typical cases. The sectors of interest are the chemical process, petroleum, gas, biofuels, energy and environmental issues, among others. The aim of this study was to develop simulations in a hydrocyclone to change their role from separator to mixer. The originality of this project is to develop an optimal geometry for these systems of flow, with the objective to be used more effectively in the processes of production of biodiesel. The results of this study were the motivation to the development of a series experiments and simulations in order to obtain better equipment for biodiesel production through the continuous route.*

Keywords*: CFD, hidrocyclone, mixer*

1. INTRODUCTION

The computational fluid dynamics (CFD) is a powerful tool for solving important engineering problems. It is able to predict behavior of fluid flow, heat and mass transfer, chemical reactions and related phenomena by solving the mathematical equations that govern these processes from a numerical algorithm (Maliska, 1995). CFD can be applied in many problems, especially in fluid flow. Simulation of water-oil separation in hydrocyclones, particle separation in cyclones, multiphase separators, multiphase flow, wind turbine, multiphase pumps, valves, washers, mixers, heat exchangers, reactors and bioreactors are the most typical cases.

The aim of this study was to develop simulations in a hydrocyclone to change their role from separator to mixer. To carry out this activity was a necessary tool to perform CFD simulations and a comprehensive study of hydrocyclone and its operating principle.

The hydrocyclone is a device for separating liquid-solid, liquid-liquid or liquid-gas systems. The principle of separation is the same as the centrifuge's one, where the particles are subjected to a centrifugal field which causes the separation of fluids.

In the hydrocyclone separation, suspension is fed into a tangential inlet located in the cylindrical part of the equipment. At the top of the region there is a cylindrical tube concentric called Vortex Finder, where the diluted stream is removed, called Overflow. There is also a hole in the lower conical part, responsible for directing the concentrated suspension (Underflow) filled with larger particles (ALMEIDA, C.A.K., 2008).

From knowing the mechanisms of separation from a cyclone, the idea is to convert its functions to make it a mixer. Well, it is known that inside the disposal occurs in a spiral motion and that there may be a mixture of substances modifying only inputs and outputs with respect to the separator. The components were mixed in the simulation were the substances of soybean oil and ethanol, a ratio of 1: 3.

There were two simulations. In the first, oil supply was simulated by the tangential inlet and feeding the Overflow by alcohol, the mixture was withdrawn by Underflow. In the second case the oil supply was in the overflow of alcohol and food was simulated in the tangential inlet.

The CFD tool used was CFX computational package which consists basically of three modules (Pre-Processor, Processor, Post-Processor) with four programs: the CFX-Build, used to construct the geometry and numerical grid; the CFX Pre, used to define the physical and boundary conditions; the CFX-Solver, used to solve the system of equations involving the problem and CFX-Post used for the analysis of results.

2 METHODOLOGY

2.1 Software and Hardware

The ANSYS CFX software package, version 12.0, was used to perform simulations on a computer with basic configuration of processor cores 4 2.83 GHz Intel (Core 2 Quad) and 4 Gb RAM
We made a plan with the aim of analyzing the efficiency of a hydrocyclone for the mixing of fluids of different density.

2.2 Data Hydrocyclone

The hydrocyclone used in this work was built at the Laboratory of Alternative Technologies at the Federal University of Sergipe and has the dimensions shown in Fig. 1. It will be used for mixing alcohol and oil for biodiesel production in a 3:1 ratio as shown in eq. (1).

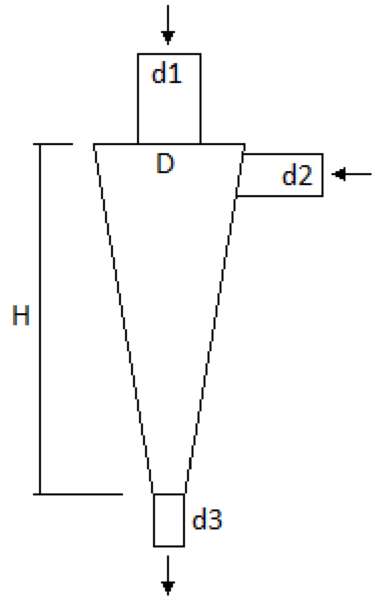


Figure 1: Hydrocyclone

D = 4.75cm

H = 38 cm

d1 = 2.67 cm

d2 = 1.52 cm

d3 =1.57 cm

$$3.\frac{m_{oil}}{MM_{oil}} = \frac{m_{alcohol}}{MM_{alcohol}} \quad (1)$$

Where:

$m_{óleo}$: mass of oil;
$MM_{óleo}$: molecular weight of oil;
$m_{álcool}$: mass of alcohol;
$MM_{álcool}$: molecular weight of alcohol.

For simulation purposes, was used ethanol as alcohol and soybean oil.as oil. The data of ethanol are provided by the ANSYS database. Table 1 contains the important properties of these substances.

Table 1 – Properties of substances

	Soybean oil	**Ethanol**
Molar mass(g/mol)	873(1)	46.070
Density (kg/m³)	919(2)	789

Viscosity (Pa.s)	0.059(1)	0.001197

(1) DANTAS *et al*, 2007
(2) JASPER *et al*, 2010

To determine the inlet velocity of the oil, Eq (2) was used. Considering the turbulent flow, Re = 3000 was attributed. With the value of the inlet diameter, the tangential velocity of the oil was calculated. From this velocity, volume flow and the mass flow rate were found by Eq (3) and Eq. (4).

$$Re = \frac{\rho V D}{\mu} \tag{2}$$

$$Q_v = V \cdot A = V.\frac{\pi D^2}{4} \tag{3}$$

$$Q_m = \rho \; Q_v \tag{4}$$

Where:
- Re : Reynolds number;
- ρ : Density of the substance;
- V : Tangenty velocity;
- D : Diameter input;
- μ : Viscosity of the substance;
- Q_v : Flow volume;
- A : Input area;
- Q_m: Mass flow.

2.3 Independence of the mesh

We used a mesh of tetrahedral structure with prisms on the walls of the domain. The test the independence of the mesh was performed using three meshes with increasing levels of refinement. It is said that independence is when the mesh refinement with numerical results do not differ significantly from the results of a further refined mesh. Figure 2 shows the refinement of the mesh. Statistics of refined meshes generated in the simulator are shown in Table 2.

The acquisition of refining the mesh was due to the use of boundary features such as inflation, which refines the mesh near the body wall and spacing that increases the number of elements in the hydrocyclone.

Table 2. Statistics of the meshes

Mesh	Control volumes	Tetrahedron	Pyramid	Prism	Total number of elements
1	21143	31937	516	27576	59959
2	49491	88730	439	61415	150584
3	134618	333007	29	140345	473381

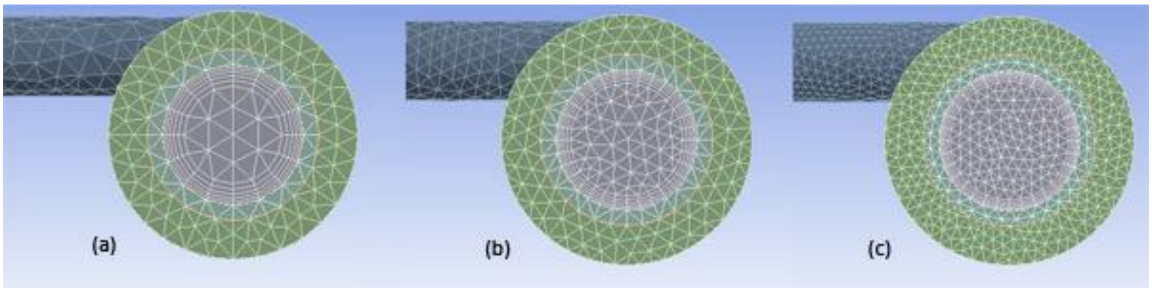


Figure 2.Drawing demo refining the mesh: (a) mesh 1; (b) mesh 2; (c) mesh 3.

To analyze better the mesh, the tangential velocity profile of soybean oil in the middle of the tube along the radial direction was observed, since this determines the velocity centrifugal field inside the hydrocyclone, ie the mixing power equipment.

Fig. 3. illustrates the graph of the tangential velocity as a function of radius of the hydrocyclone for different meshes tested. The curves of tests 2 and 3 are very close, almost identical. It was decided, then, for the second loop because it presents a smaller number of elements.

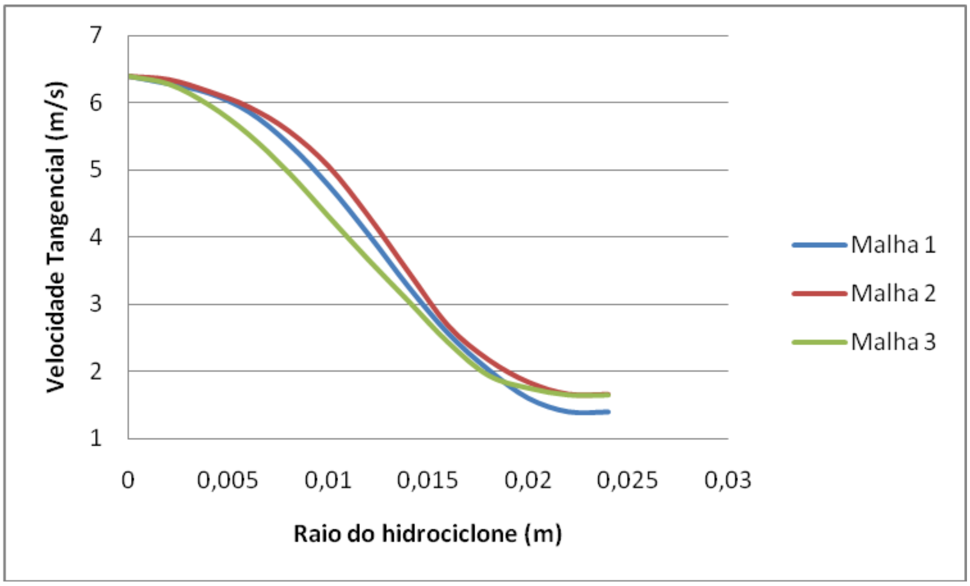


Figure 3. Graph of the tangential velocity of oil is a function of radius.

2.4 Numerical Simulations

The simulations were performed to determine the best installation position of the mixer, i.e. the situations will be simulated by the cyclone in the normal, inverted and horizontal positions. The best positions for the oil and alcohol inlet were also studied. Data from simulations are described in Tab. 3.

Tabela 3 - Simulations

	Inlet		**Gravity**		
	Oil	**Alcohol**	**Axis x**	**Axis y**	**Axis z**
1	overflow	axial	0	g -	0
2	overflow	axial	0	g +	0
3	overflow	axial	g +	0	0
4	axial	overflow	0	g -	0
5	axial	overflow	0	g +	0
6	axial	overflow	g +	0	0

All simulations were performed under the same conditions of regime and same criterion of convergence, so they can be compared with each other. These data are in Tab. 4.

Table 4 - Conditions of the Simulations

Configuration	**Valor**
Scheme	Turbulent
Turbulence model	K-Epsilon
Residue	10^-4
Oil velocity (m/s)	7,21351 (when the overflow) 12,67110 (when axial)
Alcohol velocity (m/s)	4,10436 (when axial) 0,75726 (when the overflow)

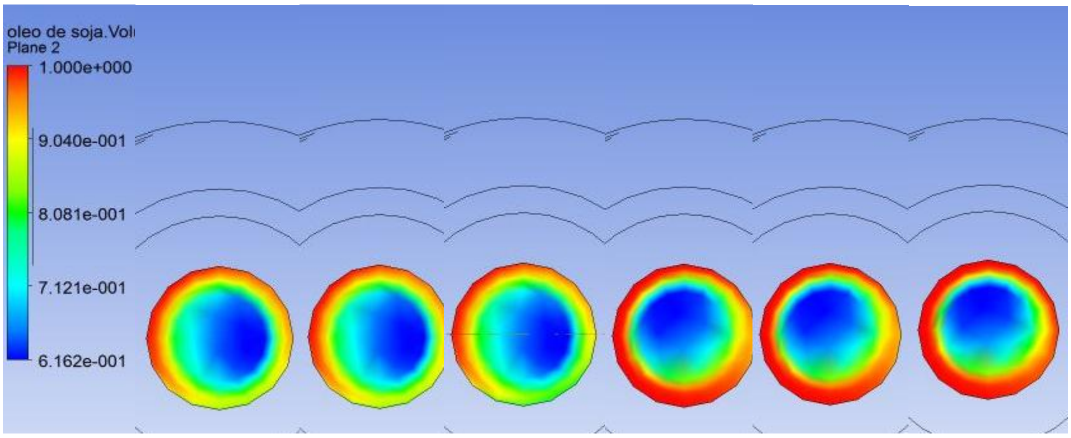
3. RESULTS AND DISCUSSION

The six simulations provided the pressure and velocity output, mass fraction of oil and alcohol in the output of the equipment. Based on these data, it was observed that the position of the hydrocyclone did not influence these values. The Table 5 shows the results of pressure, mass fraction of oil and fluid velocity at the outlet.

Table 5: Results obtained in the simulations

Experiment	inlet		gravidade					
	oil	alcohol	x	y	z	Pressure	Fraction Oil	Velocity
1	overflow	axial	0	g -	0	1956,69	0,856507	25,0335
2	overflow	axial	0	g +	0	1956,35	0,856447	25,0330
3	overflow	axial	g +	0	0	1958,16	0,856514	25,0352
4	axial	overflow	0	g -	0	789,337	0,819668	14,7131
5	axial	overflow	0	g +	0	788,899	0,819217	14,7107
6	axial	overflow	g +	0	0	792,198	0,828978	14,7414

The distribution of oil and alcohol was not completely homogeneous at the outlet of the tube, especially in cases 4, 5 and 6, as seen in Figure 4. In these simulations, the oil (higher density) was fed into the tangential inlet, and so, oil is stuck to the walls as seen in Figure 5. A region of stagnant oil has occurred. In the center of the hydrocyclone, as observed in all simulated cases, we obtained a mixture with the approximate proportions of both alcohol and oil.



(1) (2) (3) (4) (5) (6)

Figure 4. Concentration profile of soybean oil at the outlet of the hydrocyclone.

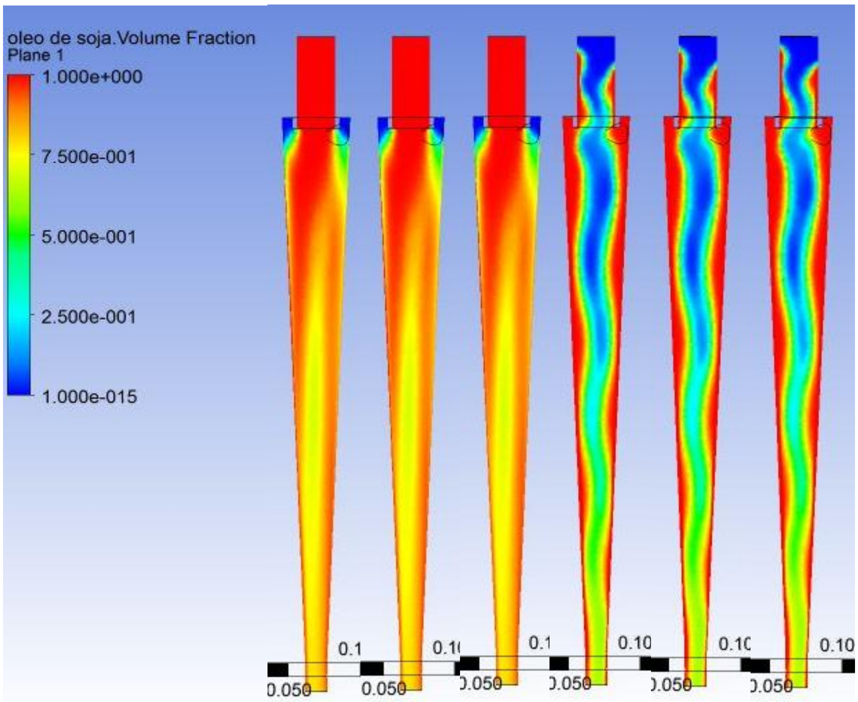


Figure 5. Concentration profile of the oil is in the axial plane of the hydrocyclone.

Streamlines inside the hydrocyclone in simulations 4, 5 and 6 confirm the region of stagnant oil in the pipe walls, as seen in Fig. 6. In cases 1, 2 and 3 there was more interaction between oil and alcohol than in cases 4, 5 and 6, where alcohol is concentrated in the center of the flow and where the mixture start to occur only on the half of the hydrocyclone. This can be confirmed through observation of fig. (5) and (6).

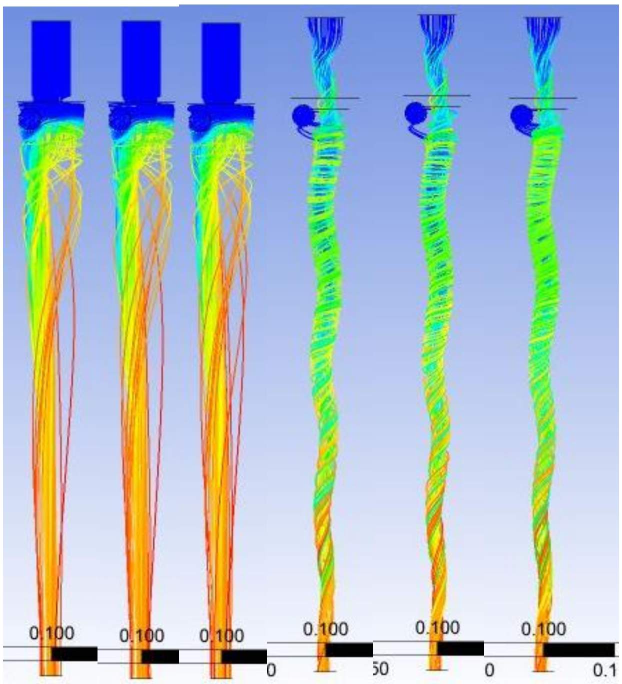


Figure 6. Current Lines

In Fig. 7 is illustrated the areas of alcohol in the hydrocyclone, and we can confirm once again that the hydrocyclone with better interaction between the particles of oil and alcohol form the simulated cases 1, 2 and 3.

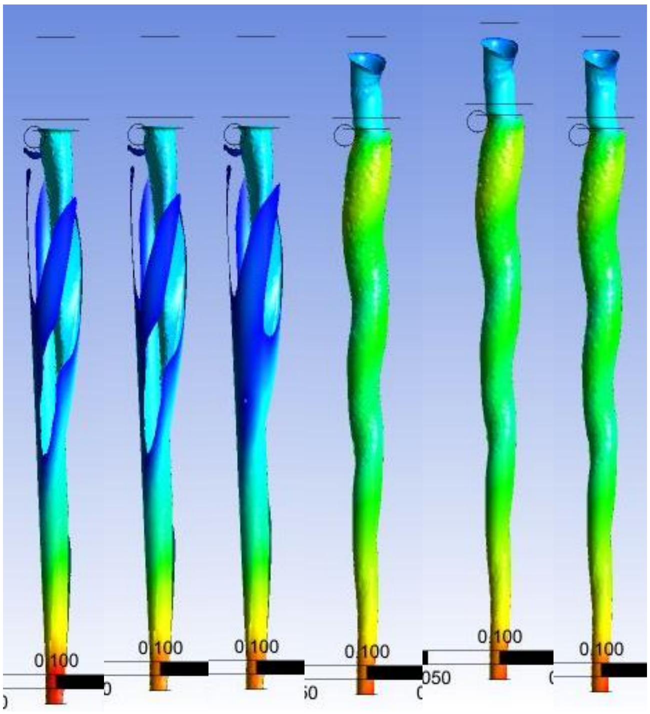


Figure 7. Surface of alcohol inside the hydrocyclone

4. CONCLUSION

This work has a fundamental importance for understanding the mechanism of operation of hydrocyclones. In this study we observed that the simulations to change the function of the hydrocyclone were satisfactorily good. We also observed that, among the six simulations presented, three showed a better profile of the mixture and these simulations were the 1, 2 and 3.

It was found that the best position for the oil inlet was to the overflow position, illustrated in the simulations 1, 2 and 3. In the simulations (4, 5 and 6), where the oil was fed in the axial position, the profiles were not good because the oil remained on the walls of the hydrocyclone.

About the changes in the positioning of the hydrocyclones, differences in the profiles of the mixture were not observed.

5. REFERENCES

Dantas, J. A.T.A. and Rocha, M. G. C., 2007., “Produção de Biodiesel: Cálculo de Massas Moleculares de Óleos Vegetais e Quantidades de Reagentes para Diferentes Razões Molares (óleo:álcool) em Reações de Transesterificação” Universidade Federal da Bahia, Instituto de Química, Departamento de Físico-Quimica, Ondina, Salvador-Ba.

Jasper,S. P., Biaggioni, M. A. M., Silva , P. R. A., 2010. “Comparação do custo de produção do crambe (crambe abyssinica hochst) com outras culturas oleaginosas em sistema de plantio direto” revista energia na agricultura. botucatu, vol. 25, n.4, p.141-153.

MALISKA,C.R., “Transferência de calor e mecânica dos fluidos computacional - Fundamentos e coordenadas gereralizadas”, LTC – Livros Técnicos e Científicos, Editora S.A., Rio de Janeiro, 1995.

6. RESPONSIBILITY NOTICE

The authors are solely responsible for the content of the printed material included in their work.