27th INTERNATIONAL CONGRESS OF MECHANICAL ENGENEERING
COBEM 2023
Brazilian Society of Mechanical Sciences and Engineering
27th ABCM International Congress of Mechanical Engineering
December 04-08, 2023, Florianópolis-SC, Brazil

COB-2023-0771
THREE-DIMENSIONAL NUMERICAL SIMULATION OF EQUIPMENT WITH A CIRCULATING FLUIDIZED BED AND AXIAL CYCLONE IN SEMI INDUSTRIAL SCALE

Guilherme Maciel
Karina Silva
Cassius Ferreira
Valério Luiz Borges
Solidônio Carvalho
Marcelo Braga dos Santos
Universidade Federal de Uberlândia, Av. João Naves de Ávila, 2121- Santa Mônica, Uberlândia-MG, 38408-100
guilherme.maciel@ufu.br, karinadepaula@ufu.br, cassiusferreira@ufu.br, valerioluizborges@ufu.br, solidonio@ufu.br, marcelo.bragadossantos@ufu.br

Abstract. *This study is motivated by the production of synthesis gas (syngas) from gasification of refused-derived fuel (RDF) in thermochemical reactors. For this purpose, an equipment with a circulating fluidized bed transforms RDF into synthesis gas at temperatures above 800°C. The syngas produced inside is in a mixture containing fuel gas (such as: H_2, CO, CxHy, CO_2, O_2, N_2, among other elements) added of solid particles (sand and ashes). An axial cyclone was positioned inside the equipment to remove particulates dragged along the bed. Due to the size and density of the solid particles and its residence the cyclone must be designed to clean the gas, improving its quality and reducing costs with scrubbers at the same time it promotes the chemical reactions that takes long time to complete. Therefore, in this study, a vertical cylindrical with 10.56 m height and 0.95 m diameter was modeled computationally, with an internal axial cyclone measuring 9.3 m in height and 0.6 m in the largest diameter region. To simplify the problem, in this work the model does not consider thermochemical reactions. For the solid phase, 0.425 m³ of 100 mesh sand particles with a constant density equal to 1500 kg/m³ were considered. For the gas phase an air flow of 800 kg/h at 500°C was adopted. Numerical-computational models were solved using the ANSYS software, based on the classical equations of mass conservation, momentum and energy. The k-omega SST turbulence model has been applied. From the analysis of the pressure gradient, velocity profiles and the particulate removal rate, the cyclone efficiency and the pressure drop were determined. It was demonstrated that the CFD simulation of industrial cyclone can be used effectively to design it. The obtained results allowed analyze the distribution of particles inside the equipment and their passage in the axial cyclone. Based on such data, future studies will be focused on thermochemical reactions, separation efficiency, increasing the quality of syngas produced by the equipment.*

Keywords: *Ansys, gasification, cyclone, syngas, waste to energy*.

1. INTRODUCTION

The management of municipal solid waste (MSW) is a problem concerning government across the world, since its production is inevitable, hence it is imperative to seek sustainable routes to its disposal. An interesting alternative to solve this problem is the gasification process, which consists of converting MSW in various gaseous components at a high temperature generating the so-called synthesis gas (syngas), which is odorless, colorless and flammable gas that can be used directly in the generation of electrical or thermal energy (Sajid et al, 2022).

The syngas is produced in an equipment called thermochemical reactor. A fluidized bed was chosen to be studied, as they have high operational flexibility, being able to work with a great heterogeneity of Refused-Derived Fuel (RDF), at different temperatures, gas residence times, reagents, catalysts and fluidizing agents (Arena et al, 2010). The ascendent flow during the gasification process carries small solid particles, which must be removed by cyclones to produce a clean gas.

In this research the cyclone is positioned inside the equipment. Therefore, the process of removing solid particles occurs at high temperatures (above 800ºC), which promotes chemical reactions, thus preventing the formation of tar from condensable gases, increasing syngas lower heating value.

The cyclone separator is a device that uses the action of centrifugal forces to separate the dense phase of a multiphase flow (Karagoz, 2005), taking advantage of the inertial forces of the fluid and a cylindrical-conical geometry, it creates an ascending vortex that carries the less dense part of the mixture out of the device, removing the dense part that remains at the bottom of the cyclone.

Based on the fluid dynamic analysis of high temperature air flowing through an equipment with fluidized bed, this work aims to analyze the efficiency of the axial cyclone in the removal of solid particles carried by syngas. However, in this work, as a simplification the thermochemical reactions inside the equipment were not considered.

2. METHODOLOGY

It was done a transient simulation of the fluidized bed, on a semi-industrial scale, and the interaction between air and sand particles inside the equipment to evaluate the fluid dynamics and to determine the efficiency of the axial cyclone in removing solid sand particles carried by the hot air. The chemical reactions were not considered in this simulation, as the objective is to verify the cyclone quality in cleaning the gas with low pressure drop, therefore simplifying the setup and the calculations.

2.1 The semi-industrial equipment

Figure 1 depicts the geometry of the simulated equipment. It consists in a cylinder with a stationary axial cyclone in the center.

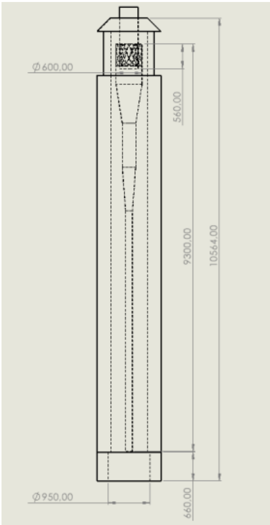

Figure 1: Thermochemical equipment geometry for the simulation.

In the Table 1 the dimensions of the equipment on a semi-industrial scale are presented.

Table 1. Dimensions of the equipment.

Equipment Geometric Specifications	Size Value
Total height, mm	10564
Internal diameter, mm	950
Axial cyclone height, mm	9300
Axial cyclone diameter, mm	600
Height of cyclone blades, mm	560
Fluidized bed height, mm	660

One of the fundamental characteristics of the chosen thermochemical equipment is to have a high height so that, while the gas and its particles rise, the chemical reactions of catalytic breaking of the carbonic chains take place, which is extremely important for tar reduction and increase syngas heating value, despite the reactions not being considered in the simulation the geometry must match the real equipment. The fluidized bed and the end of the cyclone are positioned at the lower part of the equipment (Fig. 2). Flowing through the bed the air lifts the particles that react with the hot atmosphere producing syngas.

The cyclone improves the quality of the gas produced in the equipment by separating the solid particles, at the same time it is a good option as it requires less constructive efforts and maintenance, in addition saving space as it is inside the equipment itself. To gain even more space, instead of using conventional (radial) cyclones, an axial one was chosen, as it presents an inlet area that will not require complex modifications in the equipment geometry. The upper part of the equipment in Fig. 3 demonstrates how the axial cyclone will be positioned.

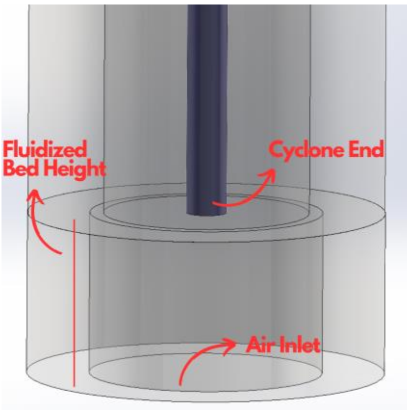

Figure 2: Lower part of the equipment.

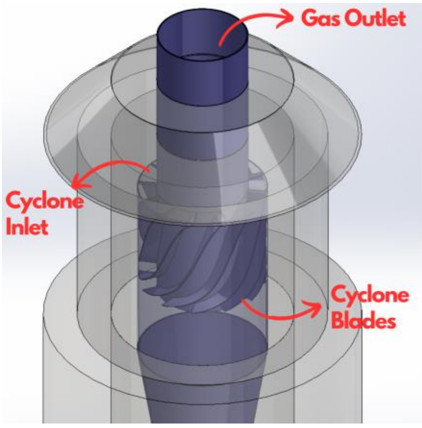

Figure 3: Upper part of the equipment.

There is a tradeoff in the cyclone design concerning the centrifugal acceleration and the pressure drop, the first one guarantee the separation of the mixture that flows into the cyclone and the second is an unwanted collateral effect. To solve this tradeoff twelve blades were designed to develop the vortex, these blades have a helical shape with increasing helix angle welded to the cylindrical wall. This blade shape is responsible for the rotation of the gas stream that enters the cyclone, causing a centrifugal force that expels the particles (the densest part of the mixture) causing them to fall into its lower part, while the rest of the fluid escapes through the top.

2.2 The fluid dynamic model

To develop a computational model for the designed equipment, it was first necessary to extract the volume of the geometry, where the flow will be simulated, since the structural reactions are not relevant for this study. Then, using the interface ANSYS® Fluent (version 2023 R1), a volume mesh was modeled. For this model, curvature and proximity refining functions were applied, as the cyclone blades required a well refined mesh to produce acceptable results. However, the remaining parts of the equipment could have a thicker mesh, since the fluid and particle behavior in those zones does not require a precise visualization.

Then, the k-ω-SST turbulence model (Menter, 1994), which is the combination of the k- ω model (Wilcox, 1998) and the k-ε model (Launder e Spalding, 1974), was applied to the simulation. According to Silveira Neto (2020), this formulation allows damping functions to be neglected for the solution of flows close to walls, which is convenient for the case of an internal flow. The governing equations that support the model are specified by Menter (1994), in addition to the principle of mass conservation, which ensures that the model is working properly.

As boundary conditions of the problem, an inlet velocity of 0.16 m/s was adopted, which represents an air mass flow rate of 500 kg/h, in addition to a null outlet pressure, allowing the calculations of the necessary pressure that the fan must be able to deliver for the specified flow rate. Furthermore, a mass of 0.279 kg of sand, with density equal to 1600 kg/m³, was added at 10000 points within a volume represented by 0.5 m of height, starting from the bottom of the equipment, uniformly filled with particles distributed by a Rosin-Rammler distribution pattern, where the diameters are ranging from 300 µm to 10 µm, with a mean diameter of 150 µm.

For this mass to be simulated, the Discrete Phase Model (DPM) was used. It presents a Eulerian-Lagrangian formulation, which works with the independence of the thermophysical properties between the two phases of the flow, where the Lagrangian frame of reference is used to track particles' motion, while the Eulerian formulation is used for tracking the continuous phase of the flow (Zahari et al, 2018).

The choice of this method promotes a simplified analysis of the expected results in the real process, since the real fluidized bed will present a much larger number of particles, however, it would require an excessive computational effort to enable its reproduction with DPM.

Understanding the principles of the simulation, it only remains to apply them in the equipment geometry, changing the temperature, density and velocity of the air entering the lower part to 800°C, 0.3246 kg/m³ and 0.6036 m/s, respectively. Furthermore, the temperature of the entire system must be defined as constant.

In the simulation process, the numerical convergence of the 3D model was obtained considering the following parameters:

Table 2: Simulation Parameters.

Parameter	Value
Type of numerical mesh	Tetrahedral
Total number of cells	10,541,682
Time step, s	0.02
Total time simulated, s	40
Iterations per time step	5

For the axial cyclone efficiency calculation, the inlet and outlet mass values in the cyclone were considered, with a 100% efficiency meaning that every particle that has entered the cyclone managed to be filtered (zero escaped mass). The described inlet mass can be calculated using a command in Fluent when all the particles in the simulation are inside the cyclone.

3. RESULTS

Based on Table 2, Figure 4 shows the air velocity streamlines inside the equipment and axial cyclone, with a maximum velocity of 9.975 m/s, which is approximately 24 times the input velocity of the system. Also, an inlet pressure of 54.09 Pa was calculated as well as the pressure contour as depicted in Figure 5, furthermore the pressure in the axial cyclone head described an expected behavior, since it had negative pressure in the center of the cylinder, signaling that the flow should ascend in that region. To verify the validity of the data acquired a mass flow balance between the inlet and the outlet of the equipment was calculated, finding a negligible error percentage as shown in the table below:

Table 3: Mass Flow Balance.

Parameter	Value
Inlet	0.13880302
Outlet	-0.13880401
Net	-9.8805691e-07

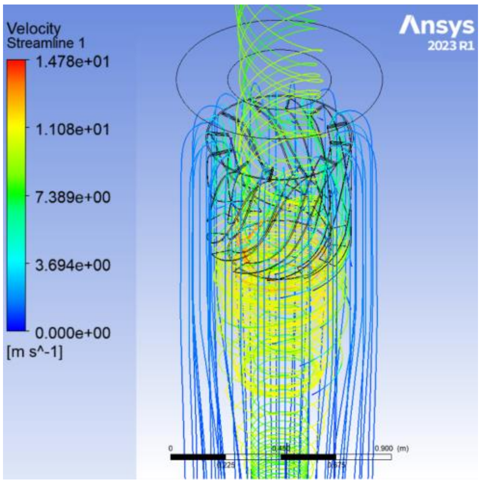

Figure 4: Streamlines according to the air velocity inside the equipment and axial cyclone.

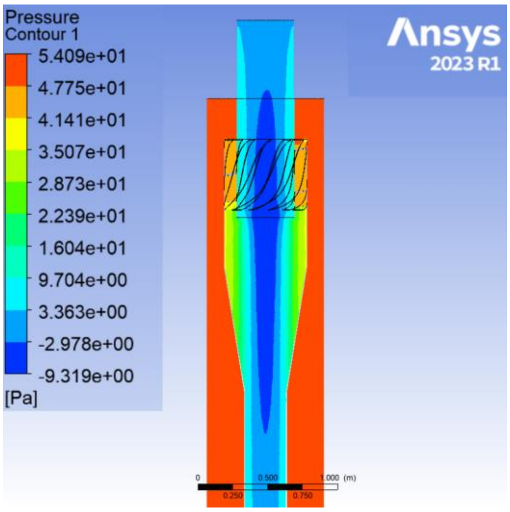

Figure 5: Pressure contour at the axial cyclone head.

In the tenth second, the smaller particles reach the cyclone inlet, and their filtering process begins, as shown in Figure 6. The largest particles being considered in the simulation, with diameters larger than 107 µm, remain at the base of the fluidized bed, and stop being graphically represented, to consume less computational resources.

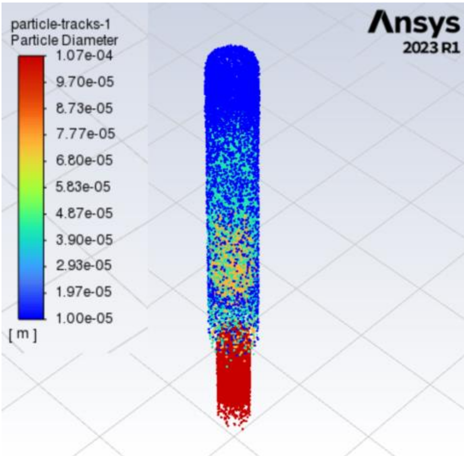

Figure 6: Behavior of the particles in the instant before entering the cyclone.

Figure 6 depicts the particle mass flow leaving the cyclone.

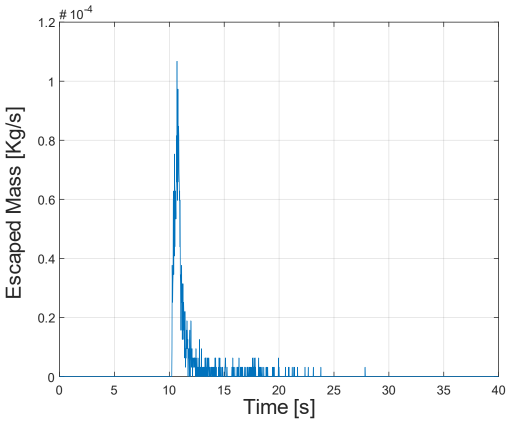

Figure 6: Mass of sand leaving the cyclone as a function of the physical time.

Applying an integral in Fig. 6, it is possible to verify that 72 mg of particles left with the air during 40 seconds of simulation. Heavier particles remain inside the equipment, returning to the base of the fluidized bed after passing through the cyclone. The total mass of sand that came out of the upper part of the cyclone represents 0.026% of the initial mass inside the equipment, proving the effectiveness of the axial cyclone in cleaning the air.

In Figure 7 it is possible to visualize, in the end of the simulation, the particles which remained inside the equipment. They are continuously filtered by the cyclone, descending in a rotational movement tangent to the cyclone wall.

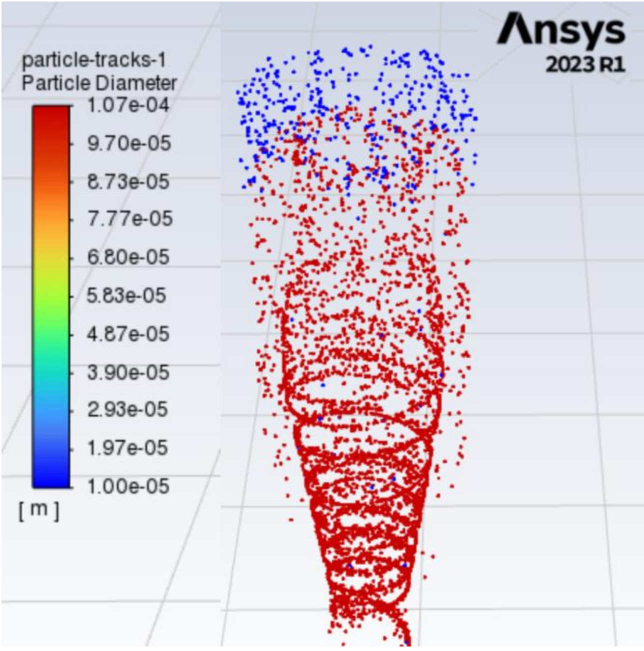

Figure 7: Particle behavior during filtering of larger diameter particles.

At the end of the simulation, the particle summary described that a total mass of 0.11 kg was computed, with 72 mg not being filtered by the cyclone. Therefore, the cyclone has an efficiency of 99.93 % for particles with more than 10 µm of diameter, i.e. it was able to effectively filter most of the particles and a clean syngas was produced even in the presence of the fluidized bed inside the equipment. Therefore, the main goal to clean the gas with low pressure drop was obtained.

4. CONCLUSION

This work presents a preliminary study of the computational fluid dynamics of air and sand particles inside a equipment in semi-industrial scale with fluidized bed.

The mathematical and computational model demonstrated an excellent characterization of the proposed physical problem. Considering the mixture of air and sand inside the equipment, there was an efficiency of 99.83% in the cleaning process carried out by the axial cyclone.

A Dell Precision T5820 workstation, Xeon W-2245, Ram 32GB, Quadro T600, SSD 512GB was required to carry out the simulations. The computational cost was 18 hours.

The next step consists of inserting the thermochemical reactions inside the equipment to simulate the gasification process of refuse derived fuel.

For future validation of the model, experiments will be carried out in a pilot plant at the company Carbogas Energia in Mauá - SP. In addition, a equipment on a scale equivalent to the one simulated in this work is under construction at the Federal University of Uberlândia. The experiments are scheduled for December 2025.

5. ACKNOWLEGMENTS

The authors would like to thank the Financing Agency for Studies and Projects (Finep), the Research Support Foundation of the State of Minas Gerais (FAPEMIG), the State Secretariat for Economic Development, Science, Technology and Higher Education of Minas Gerais (SEDE), the Coordination for the Improvement of Higher Education Personnel (CAPES) and the National Council for Scientific and Technological Development (CNPq) for financial support.

6. REFERENCES

Arena, U., Zaccariello, L., Mastellone, M. L., 2010. "Fluidized bed gasification of waste derived fuels". *Waste Management*, v. 30, n. 7, p. 1212–1219.

Karagoz, I. e Avci, A., 2005. "Modelling of the Pressure Drop in Tangential Inlet Cyclone Separators". Aerosol Science and Technology, Vol. 39, p. 857-865.

Launder, B.E. e Spalding, D.B., 1974. "The numerical computation of turbulent flows". Computer Methods in Applied Mechanics and Engineering, Vol. 3, p. 269-289.

Menter, F.R., 1994. "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications". AIAA Journal, Vol. 32, p. 1598-1605.

Sajid M., Raheem A., Ullah N., Asim M., Rehman M.S.U., Ali N., 2022. "Gasification of municipal solid waste: Progress, challenges, and prospects". *Renewable and Sustainable Energy Reviews*, Vol. 168, 112815.

Silveira Neto, A., 2020. *Escoamentos Turbulentos Análise Física e Modelagem Teórica*. Composer, Uberlândia.

Wilcox, D.C., 1998. *Turbulence Modeling for CFD*. DCW Industries, Inc., La Canada, California.

Zahari, N.M., Zawawi M.H., Sidek L.M., Daud Mohamad, Zarina Itam, Ramli M.Z., Agusril Syamsir, Aizat Abas and Rashid M., 2018. "Introduction of discrete phase model (DPM) in fluid flow: A review". *AIP Conference Proceedings 2030, 020234.* Published Online: 12 November 2018.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.