Control of solid tobacco emissions in industrial fact ories applying CDF tools

Geanette Polanco[1], Alfonso Quiroga[2], Nathaly Moreno[3], Joaquín Santos[4] and Luis Cortes[5]
[1]Departamento de Mecánica, Universidad Simón Bolívar, Caracas, Venezuela, gpolanco@usb.ve
[2]Departamento de Mecánica, Universidad Simón Bolívar, Caracas, Venezuela, aquiroga@usb.ve
[3]Departamento de Conversión y Transporte de Energía, Universidad Simón Bolívar, Caracas, Venezuela, nmoreno@usb.ve
[4]Departamento de Procesos y Sistemas, Universidad Simón Bolívar, Caracas, Venezuela, jsantos@usb.ve
[5]Universidad Simón Bolívar, Caracas, Venezuela, luis_cortes1000@yahoo.com

ABSTRACT
The emission of light solid aromatic particles from any tobacco industry affects the surrounding inhabitants, commonly causing allergies and eye irritation and, of course, uncomfortable odours, therefore, these emissions to the air must be regulated. An increasing in production must be considered in the sizing of mechanisms used to achieve the precipitation and final filtration, before discharging to the atmosphere. A numerical tool was applied to study the internal behaviour of low velocity precipitation tunnel and discharge chimney of the refuses treatment system. The characterization of the two-phase flow streamlines allows determining the velocity gradient profiles across the whole tunnel; which is intimately related with the particle concentration, and deposition zones locations. The application of CFD techniques gives the bases to find new design parameters to improve the precipitation tunnel behaviour capability to manage the increment of the mass flow of particles, due to changes in mass cigarette production.

Key words: solid-gas two-phase flow, tobacco industry emissions, CFD for designing

1. INTRODUCTION

The expansion of residential constructions to immediate surroundings areas that originally were devoted to different purposes, such as industrial uses, is not a strange situation. Tobacco factories do not escape this situation. In this scenario the control of solid emissions from the tobacco factory to environment must be revised even more carefully and stronger restrictions most be applied, to cope the proximity of human activities.

The analysis of solid emissions is considered as a fluid mechanics problem, specifically, a two-phase flow problem, due to the fact, that air is the transport agent of sediments. System design for treatment of solid refuses can nowadays been treated using computational tools like CFD to solve the governing flow equations (Navier Stokes and continuity equations), throughout a computational domain, which represents the geometric characteristics of the physical space of the problem studied, that allows the researcher to obtain flow fields relevant to the phenomenon, as for instance velocity, pressure, temperature, turbulence fields, etc. Therefore, with such a means it is possible to achieve the characterization of different designs in relatively low cost and time.

2. REVIEW

2.1 EFFECT OF PARTICLES ON TURBULENCE STRUCTURES

The presence of particles in a flow is not always clearly understood. The three-dimensional characteristics of the turbulent structures formed behind those spheres are complicated, and due to the large number of particles, they can interfere with each other and with structures already created by others particles generating a situation that is even more complex. However, it is clear, from the conservation of energy law, that the vortex created as result of a chain of events must contain less energy than the first one, so, for a flow through a large number of particles the energy will be partially dissipated as a function of displacement.

For low Reynolds numbers in a pure undisturbed flow the effects of the presence of spheres particles in the flow induce alterations in the main flow stream lines around their bodies. This deformation will depend on the relative movement between the droplets and the mean flow. For a single droplet moving opposite to the main flow, with a relative low Reynolds number, the flow will follow its body and not vortex shedding will be created after the particle, meanwhile, if the droplet has the same direction of the mean flow, the droplet surface curves the streamlines of the flow close its body, as shown in Figure 1 and Figure 2, [1].

Although the above explanation helps to draw a brief image about what happens when particles are introduced into a flow. The problem solution in our case is more complex.

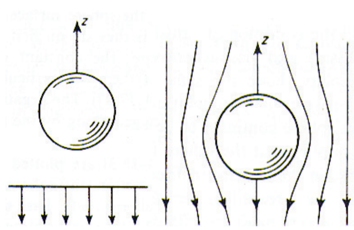

Figure 1. Streamlines for streaming flow past a sphere

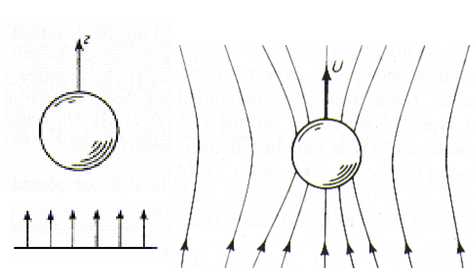

Figure 2. Streamlines for a moving sphere

2.2 MODELLING OF PARTICLES TRANSPORT

The Droplet Discrete Model (DDM) was selected to perform the simulation presented in this paper. In this particular case, the particles in the fluid flow were taken as of spherical shape. DDM solves equations for the continuous phase as an Eulerian flow field. The movement of the disperse phase through the calculated Eulerian flow field is performed separately using a Lagrangian frame of reference that permits the tracking of particles. The solutions of both phases are coupled by introducing appropriate source term in the continuous phase, allowing the inclusion of the effect of the discrete phase on the continuous phase and vice versa, and it permits alternate calculations of the continuous phase and discrete phase equations until a converged coupled solution is achieved.

Trajectory calculation of the particle was carried out by integrating a general force balance per unit particle mass, which includes in addition to the drag and gravity effects, the force required to accelerate the fluid surrounding the particle, called virtual mass force, which becomes important if the density of the gas is larger than the density of the particle. Also are included, force due to the influence of pressure gradient. The general force balance equation can be written as follows:

$$\frac{du}{dt} = \underbrace{\frac{18\mu}{\rho_p d_p^2}\frac{C_D \operatorname{Re}}{24}\left(u-u_p\right)}_{Drag\ Force} + \underbrace{\frac{g\left(\rho_p-\rho\right)}{\rho_p}}_{Gravity\ Force} + \underbrace{\frac{1}{2}\frac{\rho}{\rho_p}\frac{d\left(u-u_p\right)}{dt}}_{Virtual\ mass\ force} + \underbrace{\frac{\rho}{\rho_p}u_p\frac{\partial u}{\partial x}}_{Force\ due\ to\ pressure\ gradient} \qquad (1)$$

Where relative Reynolds number is defined as:

$$\operatorname{Re} = \frac{\rho d_p\left|u_p-u\right|}{\mu} \qquad (2)$$

C_D is a function of Re and it normally can be written as:

$$C_D = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \qquad (3)$$

Where a_1, a_2 and a_3 are constants that apply for smooth spherical particles over several ranges of Reynolds given by Morsi and Alexander [2], or

$$C_D = \frac{24}{Re}\left(1 + b_1 Re^{b_2}\right) + \frac{b_3 Re}{b_4 + Re} \qquad (4)$$

Where the variables b_1, b_2, b_3 and b_4 are functions of the shape factor, which is defined as the surface area of a sphere having the same volume as the particle divided by the actual surface area of the particle [3].

2.3 TWO-PHASE FLOW K-ε APPLICABILITY

Even though, the dynamic involved in the flow of a mixture of air and solid refuses in a tunnel seems to be very complex, the treatment of two-phase flow is based on the application of continuity, momentum and energy equations for each phase. The application of the single phase developed κ-ε turbulence model for predicting the turbulent behaviour of the jet and the droplets inside rest on the assumption of the interactions between the phases can be introduced in the equations corresponding using the one way or two ways coupling methods. Different authors have studied the applicability of the k-e turbulent model to two-phase flow cases, as a natural extension of the single phase model, in which considerations about the generation or the dissipation of turbulence due to the presence of a second phase are incorporated [4]. For one way coupling model, it is assumed the presence of the particle phases has a negligible effects on the properties of the carrier phase; this assumption is normally valid for small particle-fluid concentration ratios or high Stokes numbers (the particles motion is unaffected by the carrier flow field). The two ways coupled numerical method includes the effects of the particle in the carrier phase.

The experimental results from the study of turbulence on the bubbly flow confirm that the effect of the interfacial interactions on the turbulence of the liquid phase is important and it is assumed to have the same importance for any other disperse particle flow. However, the dynamic of the bubbles is far away from the dynamics of the particles or the droplets flow, because of the pliant surfaces, the significance of buoyancy forces presented, and the differences in the dominant physics on the particle motion based on the concentration of the disperse phase in the flow. Then, the determination of the type of flow as dilute or dense becomes also important. In the dilute flow the particle motion is controlled by the surface and the body forces on the particle, whilst in the dense particle flows, the trajectory is controlled by the particle-particle collision or interactions [5].

For particles transport by the flow, the diameter size and it comparison with the size of turbulent structure, known as eddies, could be significant in the determination of the turbulence generation or dissipation.

3. PROBLEM DEFINITION

The original system for treatment of solid refuses that is going to be studied in this work, consist in a horizontal tunnel that induces the sedimentation of the particles before the tunnel ends. The original design parameters, dimensions and shape, correspond to a tobacco production level smaller than the current one; therefore the quantity of solid particles that the system has to manage currently overcomes the designed value. This situation produces that some solid particles can get out of the system over the sanitary permitted limit generating some serious problems.

So, the current engineering challenge is focused in to identify possible improvements in the horizontal settling chamber to increase efficiency for current production conditions and to assure the legal regulations about the solid emissions, as it can be seen in different scenarios of the industry application [6].

The precipitation process of solid refuses system is based on the physical interaction between the solid particle and the gas that transports it. This interaction can be classified as a case of two-phase flow transport phenomenon. Regardless the complexity of this kind of problem, the application of numerical modelling tools to test different alternatives of solutions is proposed. The efficiency of the chamber will be evaluated qualitatively in term of the chamber affectivity.

Initially, a measurements campaign was settled to get the dimensions of the horizontal chamber as well the working flows of the installed blowers. This campaign also included properties of tobacco as the solid transported, shown in Table 1.

Table 1. Tobacco properties

PROPERTIES	VALORES
Tobacco density	550 kg/m^3(*)
Particle size	50 μm (**)
Tobacco mass	0,0025 kg/s (***)

(*) Average density obtained from the average of a measurement performed by the Laboratory of Coal and Waste Oil from USB, based on a sample collected in the settling chamber, and the value provided by Bigott.
(**) Average size of particles according to results of analysis performed by the Laboratory of Coal and Oil Waste USB.
(***) Calculated based on the information provided in plant, whereas 25 sediment sacks of 35 kg, 16 hours of operation, 5 days a week.

4. MODELLING PROCESS

Although the original problem is a three-dimensional tunnel, as other cases studied before by Thinglas et al [7], the modelling process can be simplified using some feasible arguments. For instance, based on the analysis of the dominant physical sediment transport phenomenon occurred in the axial direction, it is possible to conduct the evaluation of proposed options in qualitatively by a two-dimensional modelling covering the longitudinal geometry of the tunnel, about 55 m long. It is also possible to ignore in a first approach the steps or any other obstacle present within the chamber. In term of the tobacco, tobacco particles can be assumed as spherical uniformly distributed with constant properties equal to the ones determined in the laboratory.

A simplified problem simulated as 2D problem allows the evaluation of different options for improvement of the horizontal chamber, even when information as the presence of vortices is lost. Therefore, the flow distribution generates a pattern of particles, which does not correspond exactly with the real problem. However, it can be interpreted in a way that important parameters can be estimated to make a decision over the applicability of some real modifications of the tunnel. Additionally, 2D simulations require less time and computing resources that full tri-dimensional ones.

So, the analysis of the numerical results will allow to describe qualitatively the behaviour of the improvements proposed in the horizontal settling chamber to increase efficiency, as well as, the calculation of the chamber effective for each case allows to use a quantitative parameter to characterize the analysis.

According the theory the solid deposition is favoured by slow motion and low velocity. Under these conditions the gravity forces overcome the viscous forces drawing the particles to the floor. In this particular case, large amount of deposition increase the efficiency of the system and decreases the amount of refuses that goes out of the system. Therefore, any modification introduced to the system, will be made first at all, looking to slow down the gas velocity and the particles velocity. It is known that for a fixed flow increase the area will reduce the velocity, even if it is two-phase flow, however, it is also known that the streamlines are directly affected by the geometry changes, slopes, corners, etc. Precisely, this last point is the main reason for what CFD tools add great value to the analysis presented in this work.

The velocity and pressure profiles of two different scenarios were obtained and analysed. These two scenarios are described as follow:

1. Evaluation of extension of the output section of the settling chamber with a modified corner radius.
2. Evaluation of increased cross section of the chamber by digging a trench with an increment in the output section.

4.1 MESH AND BOUNDARY CONDITIONS

Due to the characteristics of the computational domain a structures mesh was used along the all domain. Three different meshes were generated: 73600, 294400 and 1177600 elements, respectively. A study of the sensitivity of results to the number of elements of the mesh used, concluded that, the mesh of 294400 elements provided the best balance between simulation time and accuracy of the obtained result.

The real flow conditions must be also represented in the simulations; to do that some boundary conditions must be imposed at the borders of the domain. The boundary conditions used to reproduce the actual conditions at the chamber are constant velocity for the gas and the particles at the inlet in combination with ambient pressure at the exit of the domain, wall condition for the laterals borders. The selection of these conditions is based on the fact that the domain is sufficient to achieve a complete developed flow condition at the back of the tunnel.

4.2 CHAMBER EFFECTIVENESS

In order to determine how much refuse dust is precipitated inside the chamber it is required to determine the reduction of the overall mass flow at the chamber discharge. To do it precisely will require further research using a 3D model. Nevertheless, the authors decided to do a comparison of the EFI parameter value in 2D and 3D simulation models, to determine the chamber effectiveness. The chamber effectiveness is defined as the ratio of mass flow at the exit of the chamber and the mass flow at the inlet of the chamber.

The vortexes in the air discharge has an unfavourable effect on the flow, particularly in the output section, due to the reduction of section caused by the structural mesh muffler in the discharge outlet, that causes an increased flow velocity. The static pressure distribution is uniform in the horizontal run of the sedimentation chamber, which is in agreement with the experimental measurements made, although they differ in the magnitude, because the simulation did not include the muffler mesh at the output section. It is also evident that the presence of the muffler in the output section increases the static pressure loss.

5. RESULTS

Due to the addition of new polluted air discharges associated with new areas of production, the flow driven by the sedimentation chamber has been increasing over time. Consequently the sedimentation rate has been declining gradually increasing amounts of contaminated air discharged into the atmosphere. i.e., the cleaning performance of the sedimentation chamber has been reduced. The most economical solution of the problem, apparently, is to study alternatives to reduce the average speed in the sedimentation chamber where possible, and promote sedimentation by installing devices to facilitate the performance of the cleaning process of the residual air from production processes.

Hereafter for the original geometry (without changes) will be called 2D basic case. For this case the computed EFI value is 0.73, which demonstrates the existence of a rate of precipitation inside the chamber responsible for the mass difference between the inlet and the outlet. However, the reported contamination in the surrounding areas indicates that the precipitation current precipitation rate must be increased.

The figures 3 and 4 show the velocity and the pressure profiles of the basic case. It is clear that for the current flow the magnitude of the acceleration in the sector close to the corner, and its corresponding descends of pressure in the exit area which actually induces an extraction effect of the flow with particles towards the ambient, which is opposite to the desired effect.

Figure 3. Velocity field for the basic case

Figure 4. Pressure field for the basic case

The flow driven by the sedimentation chamber has been increasing over time due to the addition of new air pollution discharge associated with new areas of production and consequently the sedimentation rate has been declining gradually increasing amounts of air contaminated discharged into the atmosphere. The only economical solution of the problem, apparently, is to study alternatives to reduce the average speed in the sedimentation chamber where possible, and promote sedimentation by installing devices to facilitate the performance of the cleaning process of the residual air from production processes.

5.1 EVALUATION OF EXTENSION OF THE DISCHARGE SECTION OF THE SEDIMENTATION CHAMBER AND A MODIFIED CORNER RADIUS

The enlargement of the outlet section of the sedimentation chamber, by adding a new mufflers module could bring advantages not only from the standpoint of improving the deposition of dust, but by reducing noise pollution due to the reduction of the exit velocity. Figure 5 shows the velocity profile in a side view of the sedimentation chamber. It can be seen that the velocity profile, at the discharge is more uniform and of smaller magnitude than in the base case (a change from 1.8 m/s to 0.8 m/s). Also it can be seem the modified corner radius of 1 m, actually it prevents the pressure drop by generation or swirl effect in corner vortex, together with the smoother effect on the stream lines of the flow, generating a more uniform profile at the exit. The EFI parameter calculated Chamber with output section expanded twice and radius of curvature at the vertex is 0,69.

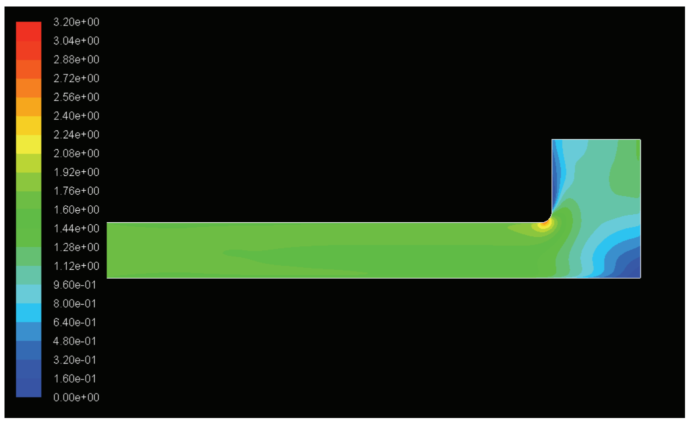

Figure 5. Velocity field for output section expanded to double and curvature radius of 1 meter

Figure 6. Pressure field for output section expanded to double and curvature radius of 1 meter

Figure 6 shows the static pressure profile by increasing the area of the discharge section, plus the effect of the radius of curvature at the upper vertex. The result is a remarkable uniformity of the static pressure, not only in the output section but also inside the sedimentation chamber, which can have a positive impact on the power consumption of the fans, although it should be clear that the major fans load is before suction connection not at discharge.

The effect on sedimentation of particles cannot be quantified; nevertheless, the stagnation area created by the right angle connector can promote the sedimentation of the particles.

5.2 EVALUATION OF INCREASED CROSS SECTION OF THE CHAMBER BY DIGGING A TRENCH

Additionally to the extension of the discharge section different increments of the cross section of the chamber were also tested. The increment in this cross section is proposed to be achieved by digging a trench; however, due to current safety regulations the angle of the ramp inside the chamber cannot exceed 5%. To fulfil these regulations it is proposed three scenarios identified as case 1, 2 and 3, respectively. Due to the discharges installation are located mainly at the beginning of the chamber, for cases 1 and 2 the ramp starts after some distance inside the chamber, meanwhile for case 3 the ramp starts at the beginning of the chamber other ways the ramp inclination will be higher that the allowed value.

Case 1: Trench of the 1 meter deep

Figure 7 shows the velocity field of the particles. When compared with the particles velocity of the basic case no major differences are found at the average speed variation, so, as can be established that only one meter lower the depth of the chamber would not have significant effects.

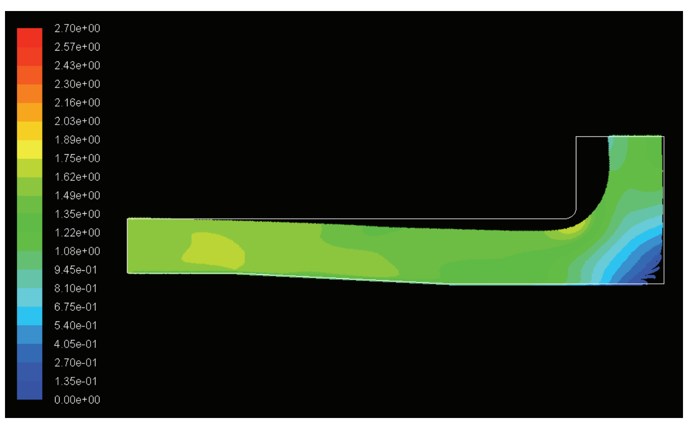

Figure 7. Particles velocity field for modified chamber with a trench 1 meter deep and output section expanded to double and curvature radius of 1 meter

Case 2: Trench of 2 meters deep

In Figure 8 it can be observed the velocity field in the chamber. When compared with the basic case (Figure 3) it can be seem that an enlarged exit section as twice the current section produces an important decrement in speed (from approximately 1.5 m / s to 1 m / s on average), which significantly favours the deposition of the particles, combined with the beneficial effects of increasing the output section.

Figure 8. Velocity field for modified chamber with a trench 2 meter deep and output section expanded to double and curvature radius of 1 meter

Case 3: Trench of 3 meters deep

In this case, making a trench depth of 3 m in total length available (55 m) would be violated safety regulations in force, since the slope would have a value of 6.8%.

Figure 9 shows the flow velocity field, which shows that the average velocity of the chamber would be above 1 m / s, and only at the bottom vertex's outlet, a lower speed is produced, resulting in improved particle settling.

Figure 9. Velocity field for modified chamber with a trench 3 meter deep and output section expanded to double and curvature radius of 1 meter

The calculated value of EFI is equal to 0.57, a decrease of 16% over the current configuration and, an improvement of 5% over the trench of 2 meters deep, which is significant, qualitatively speaking. Among the options evaluated so far, this last value is the best results and, it has the additional advantage that could provide a safety margin, which is not possible to estimate at the moment, in case that new fans are added to the settling chamber.

By comparing the values of EFI showed in Table 2, it can be seem that the increment of the vertical section (double size) plus the radius at the corner only keeping the original cross section has a minor influence on the average velocity inside the chamber and therefore on the EFI value obtained (EFI = 0.69). Also, it can be observed that increases the depth of the trench, increases the efficiency of deposition of particles by lowering the average speed in the section. The EFI value varies from 0.66 to 0.57 for trench of 1 meter deep and 3 meters deep, respectively.

Tabla 2. Summary of the values of EFI.

CASE 2D	**EFI**
Original camera	0.73
Camera output section and expanded to twice the radius of curvature in the upper vertex	0.69
Camera output section expanded and trench1 m deep	0.66
Camera output section expanded and trench 2 m deep	0.62
Camera output section expanded and trench 3 m deep	0.57

6. CONCLUSIONS

Analysing of the results of the study, it can be concluded the following:

The parameter EFI obtained from CFD simulations can be taken as a "qualitative" parameter to compare the numerical simulation outcomes, and to make the respective interpretation of realistically operation cases of the sedimentation chamber.

The understanding of the two phase particles transport and the use of CFD techniques and tools can be successful combined to solve industrial applications.

The possibility of knowing flow fields and streamlines give invaluable information to argue any alternative of modification without great investment of money and time.

According to this study, the opening of a pit at least 2 meters depth, combined with the increase of the output section of the chamber, correspond to the best operation conditions adapted to the current production level.

ACKNOWLEDGMENTS

To the Simón Bolívar University Fluid Mechanics Laboratory of for the uses of its spaces and resources.

REFERENCES

[1] Happel J. and Brenner H. *Low Reynolds number hydrodynamics*. The Hague, Martinus Nijhoff Publishers. 1983

[2] Morsi, S. A. and Alexander A. J. An investigation of particle trajectories in two-phase flow systems. *The Journal of Fluid Mechanics*, 1972 (55) Issue (2): pp. 193-208.

[3] Haider and Levenspiel O. Drag coefficient and terminal velocity of spherical and nonspherical particles. *Power Technology*, 1989 (58), pp. 63-70.

[4] Lee, S. L., Lahey R. T., et al. The prediction of two-phase turbulence and phase distribution phenomena using a k-e model. *J. Multiphase Flow*, 1989 (3), pp. 335-368.

[5] Crowe C.T., Troutt T.R. and Chung J.N. Numerical models for two-phase turbulent flows. *Annual Reviews*, 1996 (28), pp. 11-43.

[6] Inthavong K., Tian Z.F and Tu J.Y., Effect of ventilation design on removal of particles in woodturning workstations. *Building and environment*. 2009 (44), Issue 1, pp. 125-136.

[7] Thinglas, T., and Kaushal <mailto:kaushal@civil.iitd.ac.in> , D.R. Three Dimensional CFD Modeling for Optimization of Invert Trap Configuration to be used in Sewer Solid Management, *Particulate Science and Technology*, 2008 (26) Issue 5, pp. 507-519.