ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil
Copyright © 2013 by ABCM

NUMERICAL EVALUATION OF THE PRE-COLLECTOR OF A HOT MIX ASPHALT PLANT

Julian Marcell Enzveiler Marques
Universidade Feevale RS 239
julian.marcell@hotmail.com

Angela Moura
Universidade Feevale RS 239
angelab@feevale.br

Anderson Braun
Universidade Feevale RS 239
braun@feevale.br

Abstract. *The market of road building equipment is very competitive and is in constant evolution in the application of new technologies; that requires dedicated research for this segment. It is well known that Brazil requires massive investment in infrastructure, especially in road construction. For this propose there are hot mix asphalt plants which produces the HMA (hot mix asphalt) necessary for that. Among the systems that compound an asphalt plant is the pre-collector of particles, component that makes part of the filtering system and is composed by a fixed set of fins. The objective of this work is to evaluate the pre-collector through numerical simulation. For this propose the angle of sweep and the output angle are verified in order to obtain the flow velocity and total pressure. With the axial and tangential velocities resulting from fins it is possible to calculate the Swirl and assess the particulate collection efficiency. The use of CFX Software, commercialized by Ansys, Inc. is essential to develop the CFD (Computational Fluid Dynamics) simulations of the flow. The solution will be presented to different angles of sweep and output that make up the fins and the particulate collection location, containing the total pressure, axial and tangential velocities and Swirl. The results showed the possibility of improving the efficiency of the system analyzed.*

Keywords: *asphalt plant, Swirl, numerical simulation, total pressure*

1. INTRODUCTION

The quality of the pavement is important on the roads in order to not affect the comfort and guarantee the security of the users. Because of this, the market of pavement's equipment is in constantly evoluting, developing new technologies with high and optimized performance. Among the main road building equipments there are the hot mix asphalt plants. A high-tech plant is constituted by several sets, each one executes a specific task in the production of the asphalt.

The aggregate minerals are deposited in one silo, separated according to the granulometry. These silos are composed by belts responsible for the dosage of the material. A second conveyor belt carries the material to the rotative dryer, which dries the aggregate minerals that are exposed to the flame of the burner. Apart from the stack gas, the entrainment of the dried material, that is being transporteded by the dryer, also occurs. These gases are sucked by an exhaustion system that contains a pre-collector of particulates. Fig. 1.

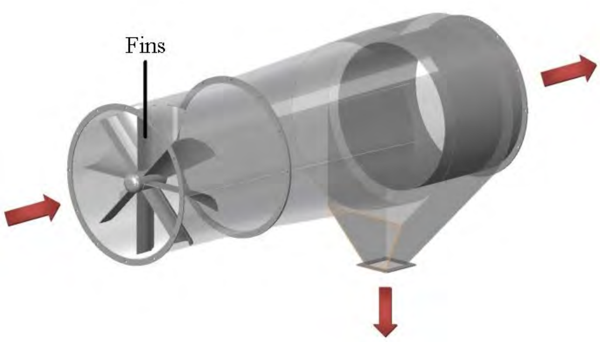


Figure 1. Pre-collector of particulates.

ISSN 2176-5480

The function of the pre-collector is to separate the biggest particulates, in an inertial way, through a fix set of fins. Thus, the granulometry separation of the materials suspended in the gases occurs. This detachment provides a preservation of the secondary collector, known as filter of sleeve, because the most abrasive and the largest material is removed. These particulates return to the asphalt mix compound, what correspond to the fraction of 2.1% of the nominal asphalt plant, approximately, according to Ciber Equipamentos Rodoviários Ltda (2007).

Thus, the general objective of this study is to evaluate the pre-collector of an asphalt plant through numerical simulations.

As specific objectives there are:

- Investigate the sweep angles and output angle of the fins in flow velocities;
- Investigate the charge drop resulting from these angles;
- Investigate energy consumption;

The evaluation of these flows, in the fluid mechanics involves solving a set of coupled equations (Navier-Stokes equations, conservation of mass and energy) that due to the complexity can be obtained by numerical simulation.

In this study supported by the concepts of Computational Fluid Dynamics (CFD) is used the Element based Finite Volume Method for discretization of the equations in the domain of the space and time. It was used the software Ansys CFX, Inc.

The Computational Fluid Dynamics (CFD) is a computational tool to simulate the behavior of systems that involve fluid flow, heat transfer and other physical processes related. It works by solving the equations of fluid flow in a special way on a region of interest, with known conditions specified in the limit of the region (ANSYS, INC., 2011).

Techniques of Computational Fluid Dynamics are essential for this type of study, since experimentally there would be a large demand of time and high cost to furnish the large number of geometries needed.

2. METHODOLOGY

The method used to evaluate the pre-collector was the total pressure or charge loss also obtained with numerical simulations carried out in Ansys CFX Software, Inc. In addition, the efficiency of the particulates collection was analyzed through Swirl, and through the tangential and axial velocities.

The use of computational fluid dynamics is essential, as well as the geometry modeling, the mesh of finite volume based on elements, the boundary conditions, the turbulence model and the stopping conditions for the simulations, as well as velocities profiles.

The input boundary conditions used are said to be ideal. This estimate is quite important to mark the results from the simulations that are shown in Tab. 1.

Table 1. Input boundary conditions.

Fluid	Air
Heat transfer	Isothermal
Temperature	20 (ºC)
Input velocity	18.750 (m/s)
Domain pressure	101.300 (kPa)
Air density	1.205 (kg/m³)
Dynamic viscosity of the air	1.820 x 10^{-5} (Pa s)
Reynolds Number	10,104,517.580

The control volume is defined in the region of interest of the pre-collector, Fig. 2. As the fluid flows through the volume of control, Eulerian description method was adopted for this study, since they are also involved in the transport equations for each phase of the flow.

The problem was classified as tridimensional flow with velocity dependent on the three coordinates x, y and z.

The asphalt plant has a window of cold air that regulates the temperature of the gases. With this, it was considered that the heat transfer was worthless in the simulations.

ISSN 2176-5480

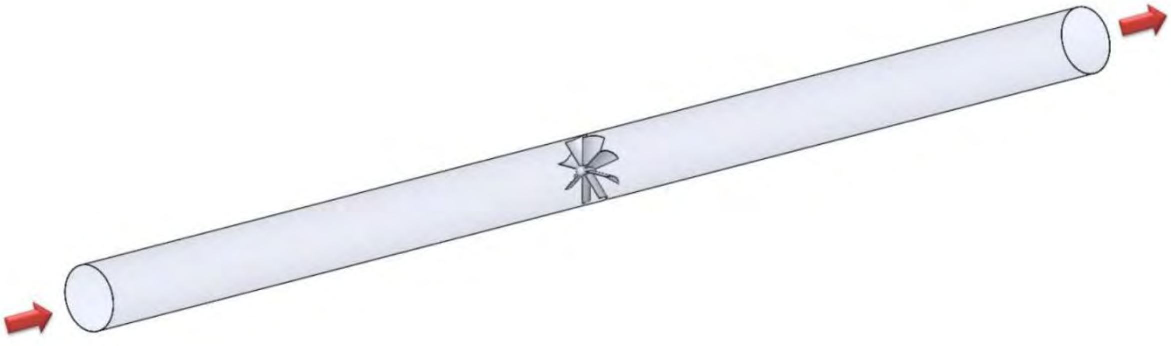


Figure 2. Control volume.

2.1 Geometry modeling

The geometry was modeled in the Dassault Systèmes Software, called SolidWorks 2011. Later, it was imported into Ansys Workbench 14.0.

The variations in sweep angles and output angles (see Fig. 3) were investigated.

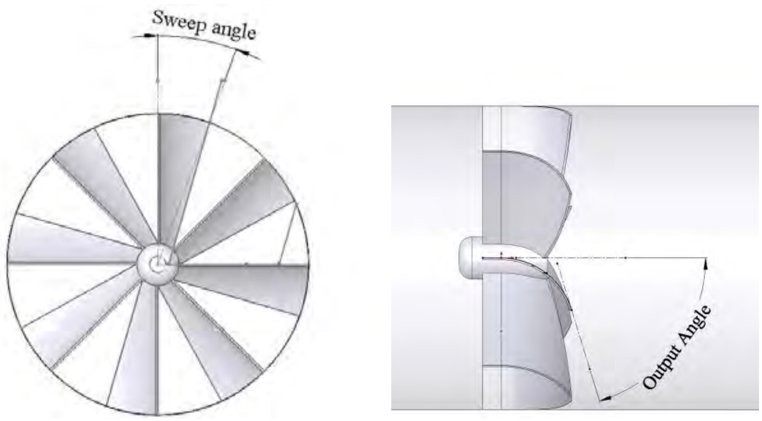


Figure 3. Details that show sweep angle and output angle.

It was analysed 13 combinations among output angles and sweep angles for investigation, as shown in Tab. 2. The definition of these compounds was based on a previous study by Hobbs (2004) and the combined 10 s erved as a reference due to the knowledge acquired in physical tests.

Table 2. Combination of the output angles and sweep angles studied in these work.

Combination N°	Output angle	Sweep angle
1	73°	45°
2	60°	45°
3	50°	45°
4	73°	35°
5	60°	35°
6	50°	35°
7	73°	25°
8	60°	25°
9	50°	25°
10	73°	20°
11	73°	17°
12	60°	17°
13	50°	17°

ISSN 2176-5480

2.2 Element based Finite volume method

The CFX uses the Element based Finite Volume Method (EbFVM). In this technique, the region of interest is divided into small sub-regions, called control volumes. The equations are discretized and solved iteratively for each control volume. As a result, an approximation of the value of each variable at specific points throughout the domain can be obtained (ANSYS, INC., 2011).

This method is based on the formal integration of the transport equations that manage the fluid flow in all control volumes obtained by the discretization of the domain. Thus, the equations are solved regarding to the partial derivative based on the resolution of mass balances, energy and movement quantity, in a determinated volume of continuous medium (PINTO; LAGE, 2001). In addition to method continues anchored in the premises of finite volumes, that is, performs balance in elementary volumes, however, uses the geometric representation via shape functions which are typically used in the finite element method (MALISKA, 2004).

2.3 Computacional meshes

The most used meshes in the area of CFD are the tetrahedral, hexahedral, prismatic and polyhedral ones. Mixed meshes are also very used for constructing complex systems. In most cases, the unstructured grids allow more freedom and are characterized by a lack of regularity in the spatial distribution of the points.

As the result of the simulations is very dependent on the mesh that is being used, some criteria must be used for selecting the optimal mesh. Thus, a test with coarse mesh was performed in Ansys Meshing, but the physic of the model had already been defined as the CFD.

Later, it was set up the mesh comprised of tetrahedral elements, containing layers of prismatic elements close to the pipe walls (Inflated) always aiming ratios of smooth aspects for the elements. These prismatic elements are important to the numerical solution be more accurate in the region where the largest gradients of velocitiess occur (near the walls). The mesh was also refined with prismatic elements on the surfaces of the fins and the Hub, Fig. 4.

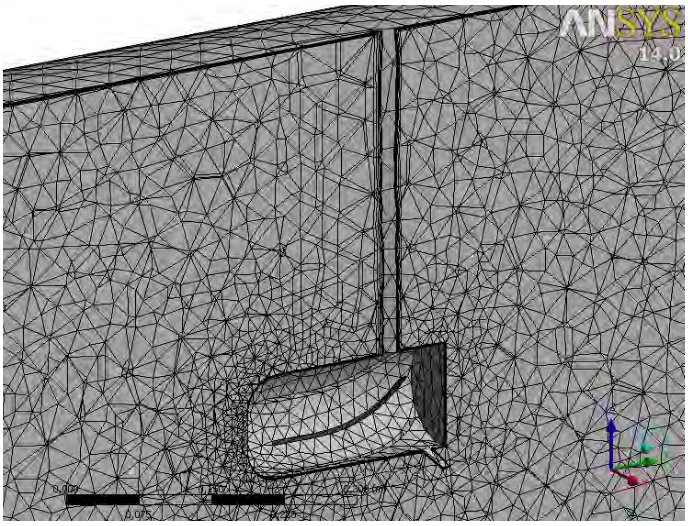


Figure 4. Detail of Mesh used in these work.

2.4 Boundary conditions

For these simulations was defined as working fluid the air of the matter’s library that belong to the own software as fluid in the domain, the temperature of 20 °C, isothermal and pressure 101.3 kPa. In this work, the fluid was considered as an approximation of the real model, and thus particles contained in the gas of the tubulation were not considered.

At the entrance (Inlet) fluid velocity was 18.75 m/s, which corresponds to the average velocity of industrial dust transport for internal flow indicated by Sobrinho (2009) and adopted in the equipment. Sobrinho’s indication (2009) can be seen in Tab. 3.

ISSN 2176-5480

Table 3. Velocity of industrial dust transport.

Nature of the contaminant	Examples	Transport velocity (m/s)
gas, vapor and smoke	All types	From 5.0 to 6.0
Chromic acid mist	Hard chrome and decorative	12.5
Fume	Zinc and aluminum	From 7.0 to 10.0
Light and very fine dust	Cotton and wooden dust	From 10.0 to 12.5
Dry dyst	Scraps of leather and rubber	From 12.5 to 17.5
General industrial dust	Cement dust/ceramic, amianthus fibers and foundry sand	From 17.5 to 20.0
Heavy dust	Iron powder and lead	From 20.0 to 22.5
Heavy and moist dust	Lead, cement and amianthus	22.5 or more

2.5 Turbulence model

The set of equations describing the dynamics processes, heat and mass transfer is known as the Navier-Stokes equations. These equations plus the continuity equation form a set of four nonlinear partial differential equations. Analytical solutions to these equations have been obtained for many special cases, but only for geometry and initial conditions or simpler outline, for which many of the terms in the equations can be considered equal to zero (FOX; MCDONALD, PRITCHARD 2006).

The turbulence phenomenon is very complex and difficult to evaluate in numerical simulations through Navier-Stokes original equations because of the possibilities of the current processing. Due to this restriction, approximate solutions of the effects of turbulence are sought. The most known and most widespread approach is the numerical simulation via Reynolds-Averaged Navier-Stokes equations (RANS), can also be called Classical Modeling of Turbulence.

Reynolds-Averaged Navier-Stokes equations are based on the concept of turbulent viscosity and on the transport equations of Reynolds tensor, where they are evaluated considering the average of averages over large time intervals for turbulence.

An important aspect of this average is that most turbulent flows of interest are stationary and so, in these cases, the numerial simulation can be executed for a single instant of time.

In the numerical simulation via Reynolds-Averaged Navier-Stokes equations, it is used model of turbulence for describing the fluctuation product, called Reynolds tensor.

Among the most used turbulence models is the $k - \varepsilon$,based on the transport of scalars largeness, the k is the fluid kinetic energy and the ε is the dissipation of kinetic energy. This model is deficient in regions where the pressure gradients are very large, as it occurs near the walls. Despide the chosen turbulence model is the $k - \varepsilon$, due to its satisfactory robustness demands less hardware capacity compared to the other existing models and it is the most used in the industry.

To improve the accuracy of the model were developed models and Speziale-Sarkar-Gatski (SSG).

The model $k - \omega$ was elaborated for the calculation of flows at low Reynolds numbers, what occurs near the wall, being more suitable for aerodynamic problems (flow with detachment regions). The model doesn't involve the non-linear functions needed for the model $k - \varepsilon$, becoming, therefore, more accurate and robust (CARVALHO, 2008). The model SSG inserts only small improvement in the flow evaluation over blunt bodies, despite the remarkable complexity embedded in the formlation (PICCOLI, 2009).

2.6 Stopping conditions for the simulations

A CFD problem is solved through the following form:

- The differential equations are integrated throughout all volumes of control in the interest region. This is equivalent to apply a basic conservation law for each volume of control.
- These integral equations are converted into an algebraic equation system, engendering a set of approximation for the terms of the integral equations.
- The algebraic equations are solved iteratively.

An iterative approach is necessary due to the non-linear nature of the equations, and as the solution approximates to the accurate solution, it is said that they converge. For each iteration, a mistake or residual is reported as measure of general conservation of the flow properties (ANSYS, INC., 2011a, our translation).

Control criteria are fundamental to the simulation stopping conditions. In this study, it was used a minimum of 1 iteration and a maximum of 200 iterations.

ISSN 2176-5480

2.7 Lines for velocity profiles

The velocities were evaluated in six lines within the pre-collector arranged among distances of 0.4 m from the top of the fins. The length of these lines is equal to the diameter of the tubulation. The position of each line is shown in Fig. 5.

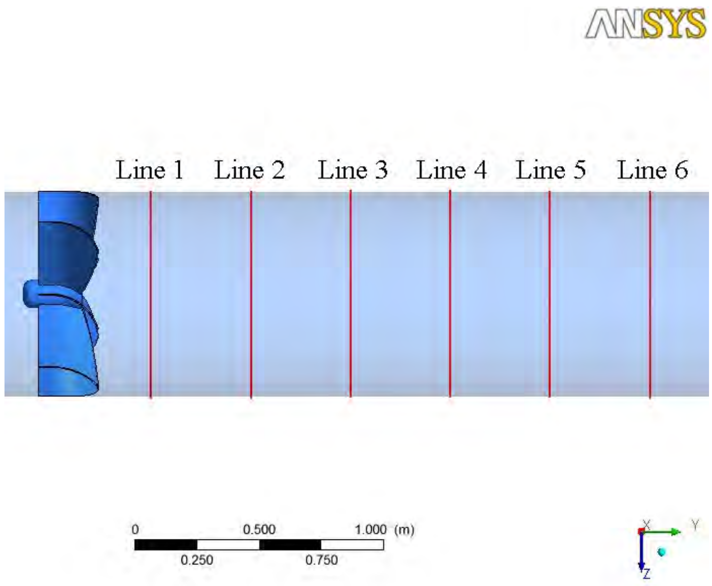


Figure 5. Position of the lines for velocity profiles

The objective of these evaluations is to obtain the profile of the axial velocity and the tangential velocity profile, to caculate the Swirl. Thus, it is possible to analyse and qualify the efficiency of the pre-collector collection.

3. RESULTS

It is important to notice that in this work, in oder to preserve the company project's parameters, the results were nondimensionalized . This technique is widely used in order to keep secret the results of the project, however it is possible to perform the analysis of the simulated results.

3.1 Quality of the mesh

The creation of the mesh for the simulations needs a processor with considerable RAM memory (Random Access Memory). The CFX used about 4 kbytes of memory per knot.

The mesh used in the simulations engendered 397,083 knots and 1,584,667 elements. For this the processor needed about 1.5 Gbytes of RAM memory.

3.2 Total pressure

The sweep angles that were investigated were 17, 20, 25, 35 and 45 degrees. The sweep angle determines the percentage of the área of the perpendicular fins to the axial axle. The output angle can have more influence in the Swirl, and 3 angles were investigated, 50, 60 and 73 degrees.

The result of the total pressure of the gas flow in the pre-collector is detailed for each angle combination shown in the Fig. 6.

ISSN 2176-5480

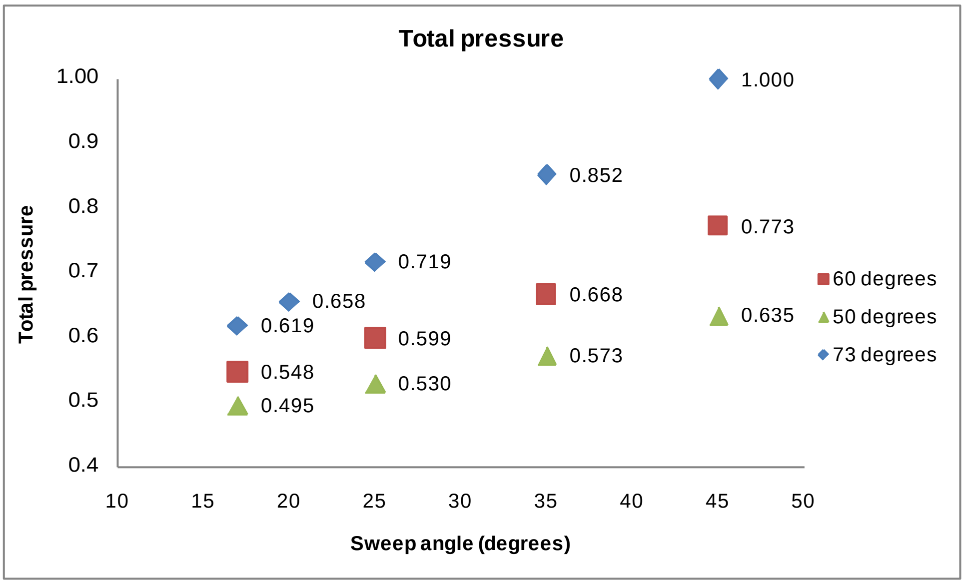


Figure 6. Graph of the dimensionless total pressure for the investigated combinations.

The combination of the output angle of 73 degrees and the sweep angle of 20 degrees (combination number 10 of the Tab. 2), resulted in dimensionless total pressure of 0.658.

In the Fig. 6, it is observed that with the smaller the output and sweep angles, the smaller the total pressure. For the seven combinations (3, 6, 8, 9, 11, 12 e 13) the dimensionless total pressure was below 0.658.

In an asphalt plant, the total pressure of the pre-collector is usually the second largest of the filtering and exhaustion system. Among the variables of the exhauster selection (centrifugal fan with blades facing backward) is the total pressure. Thus, the smaller the total pressure, the smaller the size of the exhauster, the smaller the power required for the electric motor and, consequently, the smaller the energy consumption.

The total pressure is more susceptible to the output angle than the sweep angle. This is observed when 20.0% of the output angle is increased, combination 13 to 12, resulting a increasing of 10.7% of the total pressure. Although in the sweep angle, the increase of 47.0%, combination 13 to 9, the pressure increases 7.0%.

Aiming to diminish the total pressure of the system, the study progressed without the combinations above of the value of dimensionless total pressure of 0.658.

3.3 Plans of velocity

The gases composed of particles are highly abrasive and compromise the structure of the pre-collector. The wear out, mainly, of the fins and tubulation is due to the high velocities of the particles in a turbulent flow.

The axial and tangential velocities plans have as characteristic to indicate where the largest particles move, because of the higher values of velocity on the walls when compared to the center of the tubulation. These plans are two-dimensional and were placed in the center of the control volume, parallel to the axial axle of the geometry.

The comparison between the combination 3 and 10 is the most important, because the next section demonstrates that the combination 3 shows the largest Swirl and the Fig. 6 shows lower total pressure when confronted to the combination 10. For this reason, Fig. 7 shows the plans of tangential and axial dimensionless velocities of these two combination with the same scale of colors. In these figure, the upper velocities plans are the results of the combination 3 and lower velocities plans are the results of combination 10.

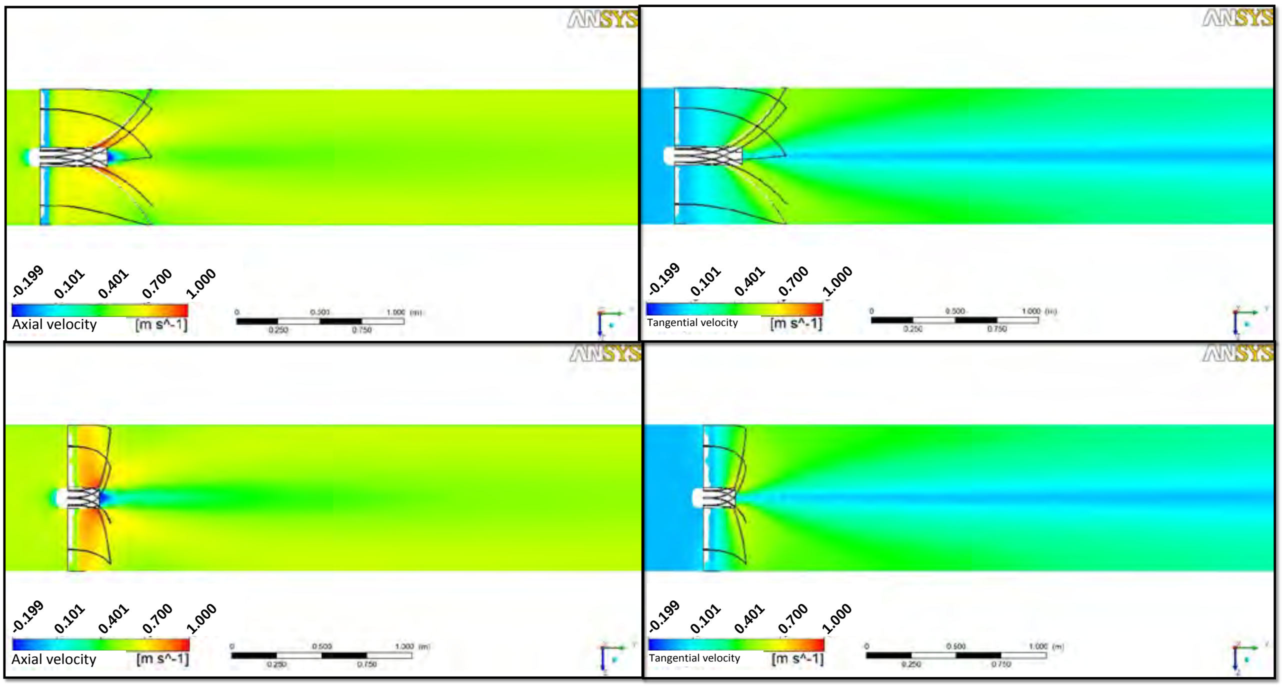


Figure 7. Axial and tangential dimensionless velocities of the combinations number 3 and 10 of Tab. 2.

Despite the velocities are similar owning to small variations in color, the combination 3 has the lower velocities that are distributed in the region of the fins near the Hub. In this situation it is possible to reduce wear out and increase the useful life of the pre-collector.

3.4 Swirl in the lines

The turbulent flow can be assessed Swirl, *S*. This index is defined in Eq 1 and represents the ratio of the axial component of the moment of of the tangential motion quantity, I_ϕ, and the amount of axial movement, I_x, multiplied by the radius of the geometry, r_o, (INSTITUTO DE PESQUISAS TECNOLÓGICAS, 1999).

$$S = \frac{I_\emptyset}{I_x r_o} \tag{1}$$

The values I_ϕ and I_x of the components described previously can be obtained from Equation 2 and Equation 3.

$$I_\emptyset = 2\pi \int_0^{r_o} \rho(uw)r^2 dr \tag{2}$$

$$I_x = 2\pi \int_0^{r_o} \rho u^2 r dr + 2\pi \int_0^{r_o} p r dr \tag{3}$$

Where ρ is the density of the fluid, *u* and *w* are, respectively, components of the velocity field in function of the spatial coordinates *x* and *z*, and *p* is the output pressure.

The calculation of the Swirl is based on the tangential and axial velocity profiles in the 6 lines of the proposals, in the path of the pre-collector, posterior to the fins. To determine the velocity of each profile, 50 points are integrated to approximation. This value is proportional to the tubing's diameter, in this case, sufficient for the study.

The Swirl equation is applied in each row of each combination, and the results can be observed Tab. 4.

ISSN 2176-5480

Table 4. values of the Swirl.

Combination	Line 1	Line 2	Line 3	Line 4	Line 5	Line 6
3 (50° and 45°)	0.406	0.525	0.473	0.429	0.391	0.356
6 (50° and 35°)	0.452	0.459	0.417	0.378	0.345	0.315
8 (60° and 25°)	0.497	0.455	0.413	0.377	0.343	0.314
9 (50° and 25°)	0.438	0.403	0.366	0.334	0.306	0.280
10 (73° and 20°)	0.515	0.467	0.425	0.388	0.356	0.326
11 (73° and 17°)	0.470	0.430	0.392	0.359	0.328	0.302
12 (60° and 17°)	0.426	0.390	0.356	0.327	0.299	0.274
13 (50° and 17°)	0.382	0.349	0.320	0.292	0.268	0.246

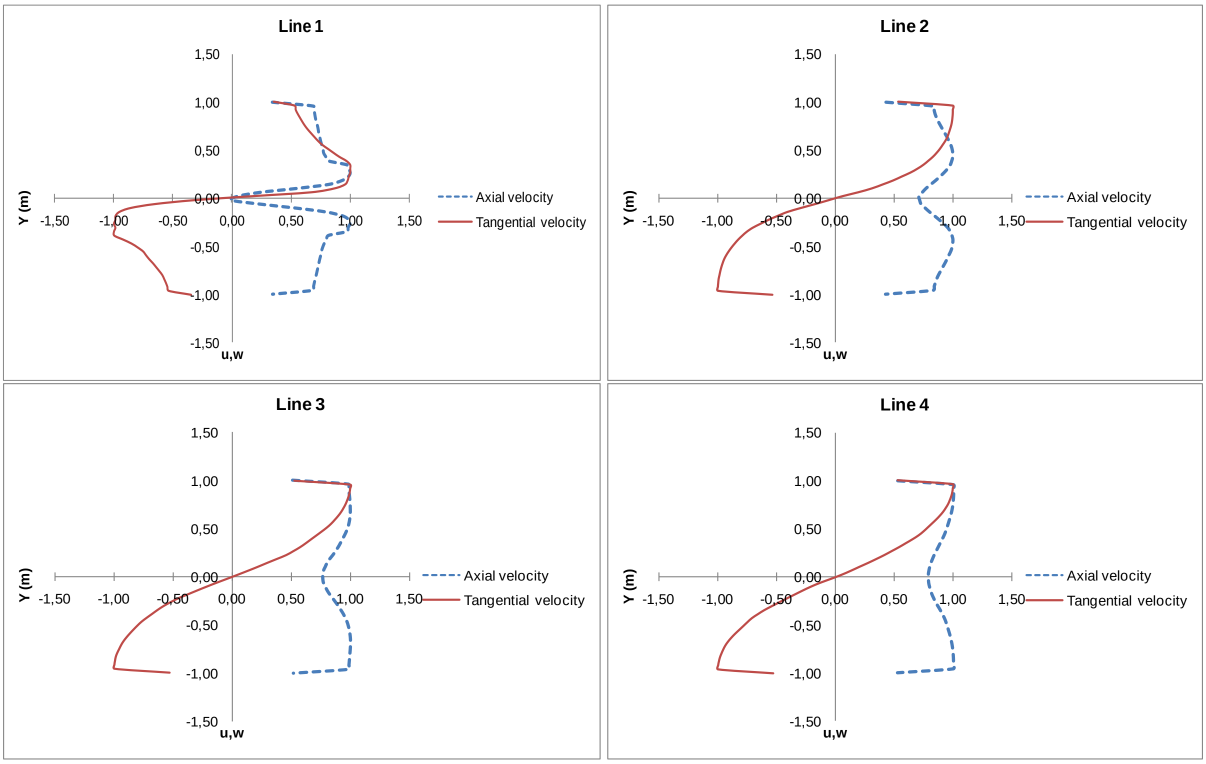
Assessing the results shown in Tab. 4, the highest values of the Swirl are in lines 1 and 2, and the lowest values are in the line 6. This indicates that in every combination of the Tab. 4, the Swirl decreases according to the velocities. Despite being close Swirl values between combinations, it can be observed the relationship of total pressure with the value of Swirl.

Another important finding withdrawn from the numerical simulations is that the increase of the output angle provides highest values of Swirl in line 1. The larger this angle, the larger the Swirl near the fins will be. The opposite occurs with the sweep angles, they provide higher values of Swirl in lines 2 and 3.

It is verified the largest Swirl (0.525) in line 2 of the combination 3, which presents increase of 2.0% in the Swirl and reduction of 3.5% in the total pressure related to the combination 10. This result is essential to define the particulate's collection point.

According to the Swirl's classification by Eddings (2012), the high turbulence appears with the highest values of 0.6. Due to the low axial force, the gas flow splits and doesn't allow the infiltration in the recirculation zone.

Therefore, the difference among the profiles of axial and tangential velocitiess for every line can be verified through Fig. 8. Thus, it is possible to assess, with more accuracy, the characteristics regarding to the particulates. Note that in this figure, the values multiplied by 1000.



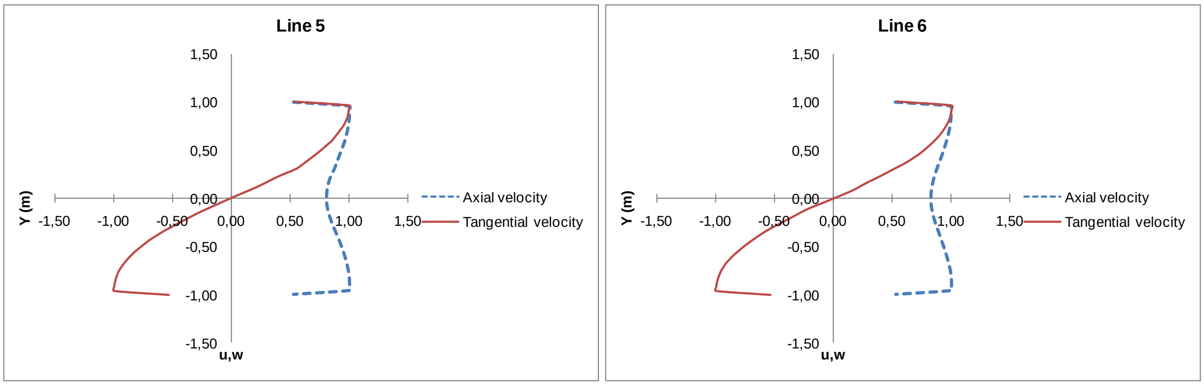


Figure 8. Profile of tangential and axial dimensionless velocities in the lines.

The diameter of tubulation corresponds to the line u, w, and it was nondimensionalised by its highest value. The axial and tangential velocities were nondimensionalised from the highest value of each other, respectively.

As it is possible to observe, the velocities profiles in lines 3, 4, 5 and 6 don't alter very much, however, it is still possible to verify that there are velocities profiles that are not totaly developed.

For the Swirl, velocities profiles with axial velocities lower than the tangential velocities are preferred, because the axial velocity doesn't have the characteristic of moving the particles to the tabulation's wall as the tangential velocity does.

This is noted through the tangential velocity difference between the walls and the tubulation center. Despite the velocitiess are nondimensioned singly, the highest value of Swirl is in line 2, due to the axial velocities is lower than the tangential.

4. CONCLUSIONS

Only the numerical simulations do not validate the study. It is often necessary experiments in scales or reals, known as physical prototypes. However, the significance of the numerical simulations to analyse this study demonstrates to be broad, as well as the research's evlotuion and the representation of the computational models.

In the geometry's modeling many difficulties were found. Parameterizing the geometry for, later, solving the problem in the CFX is something recent for the industry. Perhaps, in the CFX are the largest difficulties, to create the suitable mesh, select the correct turbulence model and obtain the results in the pos-processor demand time, study and knowledge.

Analyzing the results of the numerical simulations that were executed in this study, it was concluded that the combination 3, Tab. 2, showed the highest value of the Swirl, providing an improvement to the collection of the particulates in 2.0% if compared to the combination 10. The total pressure is smaller than (3.5%), for this combination it is possible to diminish the size of the exhauster, the power required to the electric motor and even the energy consumption.

The research does not characterise the particulates contained in the gases. Thus, the results considering the air as fluid, can disguise the analysis and obtain approximation. The particulates contained in gases have different granulometries, which influence in the collection.

A posterior study about the most suitable turbulence model can be conducted to improve the values of the velocities and the total pressure, mainly, near the fins and the limit layer. New domains can be studied considering the pre-collector's particulates, and use the CFX's Particle Tracking.

5. REFERENCES

ANSYS, INC. ANSYS CFX: Introduction. Canonsburg, USA: SAS IP, Inc., 2011.

CIBER EQUIPAMENTOS RODOVIÁRIOS LTDA. Especificação Comercial do Produto: Usina de Asfalto Advanced. Porto Alegre, RS, 2012.

EDDINGS, E. G. Flame and Burner Aerodynamics. Salt Lake City, USA: University of Utah, 2012

FOX, R. W.; MCDONALD, A. T.; PRITCHARD, P. J. Introdução à Mecânica dos Fluidos. 6. ed. Rio de Janeiro, RJ: LTC, 2006.

HOBBS, A. Design and Optimization of a Vortex Particle Separator for a Hot Mix Asphalt Plant. Chattanooga, USA: Astec Industries, 2004.

INSTITUTO DE PESQUISAS TECNOLÓGICAS. Curso Combustão Industrial. São Paulo, SP, 1999. Cap. 2.

MALISKA, C. R. Transferência de Calor e Mecânica dos Fluidos Computacional. 2. ed. Rio de Janeiro, RJ: LTC, 2004.

ISSN 2176-5480

PICCOLI, G. L. Análise Numérica na Engenharia do Vento Computacional Empregando Computação de Alto Desempenho e Simulação de Grandes Escala. Departamento de Engenharia Mecânica, Universidade Federal do Rio Grande do Sul, Porto Alegre, RS, 2009.

PINTO, J. C.; LAGE, P. L. C. Métodos Numéricos em Problemas de Engenharia Química. Rio de Janeiro, RJ: E-papers Serviços Editoriais Ltda., 2001.

SOBRINHO, F. V. et al. Ventilação Local Exaustora em Galvanoplastia. São Paulo, SP: FUNDACENTRO, 1996.

6. RESPONSIBILITY NOTICE

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