Article

Modeling of the grain-shell mixture in a castor-cleaning machine (*Ricinus communis* L.)

Pedro Cruz Meza[1]
Gilberto de Jesús López Canteñs[1§]
Samuel García Silva[1]
Conrado Márquez Rosano[1]
Eugenio Romanchik Kriuchkova[1]
Omar Reséndiz Cantera[1]

[1]Universidad Autónoma Chapingo- Posgrado en Ingeniería Agrícola y Uso Integral del Agua. Carretera México-Texcoco km 38.5. Chapingo, Estado de México. CP. 56230. México. (samuelmx57@yahoo.com; cmar_ros59@yahoo.com.mx; eugenio.romantchik@gmail.com; caromam687300@yahoo.com.mx).

[§]Autor para correspondencia: alelopez10@hotmail.com.

Abstract

Currently the cleaning of the grains by means of vacuum is spreading, in the development of agricultural equipment, due to the absence of dust in the work area. A common problem is the accumulation of material in conflicting sectors of the cleaning system, such as in places of sudden changes of direction or section in the tubes. In particular, it is of special interest, the component that distributes the flow, since a correct design must ensure a homogeneous distribution of the flow of air and material, without clogging (accumulations). The objective of this work is to analyze the behavior of the grains and shells of the castor oil plant (*Ricinus communis* L.) in the cleaning system by means of the computational fluid dynamics (CFD) technique and the trajectory of the particle at the exit of the fan, to determine the proper minimum speed. In the simulation, the k-epsilon and DPM (discrete phase model) models of ANSYS FLUENT were used. A test bench and a metal structure was constructed and instrumented to compare the experimental and computational results. The comparison of the results showed a good qualitative and quantitative agreement (R^2= 82%) in the velocity profile. The average speed was 5.98 and 5.09 ms^{-1} for the estimated and experimental respectively. The standard deviation was 1.85 and 1.63 ms^{-1} for the estimated and experimental speed respectively. Finally, the model of the trajectory of the particle adequately predicts the distribution of the material at the outlet of the ventilator.

Keywords: CFD, grain cleaning, trajectory.

Reception date: january 2018
Acceptance date: march 2018

Introduction

In the state of Oaxaca, Mexico, the shelling of the capsules of the castor oil plant (*Ricinus communis* L.) is done manually. A complementary stage to the hulling is the cleaning of the grains, generally carried out in the field, by venting the air. Manual cleaning has proven to be an arduous and painful task, since it requires physical resistance and skill for its execution, as well as harming the health, by contaminating the air product of this process (Ponpesh and Giles, 2009; Ponpesh *et al.*, 2010). Currently the cleaning of the grains by means of vacuum is spreading, in the development of agricultural equipment, due to the absence of dust in the work area. A common problem is the accumulation of material in conflicting sectors of the cleaning system, such as in places of sudden changes of direction or section in the tubes.

In particular, it is of special interest, the component that distributes the flow, since a correct design must ensure a homogeneous distribution of the flow of air and material, without clogging (accumulations). The machines used in the processing of agricultural products in Mexico, are mostly designed from information on imported products, with characteristics different from the needs of the farmer. The construction of cleaning machines for agricultural products is mostly carried out by empirical methods. In this context, modeling and simulation techniques have proved to be a very useful tool in the design, analysis, optimization and improvement of agricultural machines (Alves *et al*., 2005).

Modeling in computational fluid dynamics (CFD) allows to develop and test the prototypes of the harvester machines, and obtain the essential design parameters in engineering without the need to build the physical model, since it takes more time and is relatively expensive (Ponpesh and Giles, 2008). Therefore, modeling in CFD allows to optimize the design of the cleaning system of the harvesters, and the physical process of separation of the particles in the air stream.

Modeling in CFD has been applied successfully in numerous cases. Some of the agricultural applications are: the modeling of the spray drag (Teske *et al.*, 2011), the design of aerial sprayers (Herrera *et al.*, 2006; Herrera *et al.*, 2010), the cleaning system of harvesters of grains (Gebrehiwot *et al.*, 2010 a; Gebrehiwot *et al.*, 2010 b; Du *et al.*, 2013; Ni *et al.*, 2013), collection system and cleaning of walnut harvesters (Ponpesh and Giles, 2008; Ponpesh and Giles, 2009; Ponpesh *et al.*, 2011 a; Ponpesh *et al.*, 2011 b), the collection and cleaning system for coffee harvesters (Alves *et al.*, 2005; Magalhães *et al.*, 2006 a; Magalhães *et al.*, 2006 b), in cyclones (Corrêa *et al.*, 2004; Cernecky and Plandorova, 2013), the design of ventilation systems for agricultural buildings (Sun *et al.*, 2004), among others. However, it has not been reported the use in the design of castor hulling machines.

The turbulent flow can be modeled through the use of different models, which seek to find the best results and that have been used in several jobs conducted in the development of agricultural equipment. Among the models available, Reynolds averaged navier- stokes equation (RANS) and large-eddy simulation (LES) are the most common. Despite the reliability, the LES models require a lot of time for resolution; therefore, they are rarely used, unless great precision is required. On the other hand, RANS models require less computing time, but some situations have problems of lack of precision. Thus, the RANS models such as the standard k-ε, two equation k-ε, realizable k-ε, renormalization group (RNG) k-ε and Reynolds stress model (RMS) are the most used for turbulent flow process simulation.

The objective of this study is to analyze the behavior of grains and shells of the castor oil plant (*Ricinus communis* L.) in the cleaning system using the CFD technique and the trajectory of the particle (shell) at the outlet of the fan, to determine the minimum proper speed of the hulling machine.

Materials and methods

In the Postgraduate course of Agricultural Engineering and Integral Use of Water (IAUIA, for its acronym in Spanish) of the Autonomous University of Chapingo, a prototype of a castor shelling machine was designed and built. Samples were taken in the cleaning system at the entrance and exit of the center of the duct and also to the exit of the fan of the flow and the speed of the air, using a hot wire anemometer brand Lutron model YK-2005 AH.

Methodology of computational fluid dynamics

Object of study. The computational simulation was carried out based on the information obtained from the cleaning system of a protuer of a hulling machine developed in the postgraduate course of IAUIA. In the Figure 1 shows the duct of the cleaning system of the husking machine (M D). The dimensions of the duct with respect to the fan are 0.152 and 0.1 m greater and smaller diameter respectively and total length of 1.94 m (section 1 horizontal 0.2 m, section 2 horizontal 0.26 m, section 3 vertical 0.58 m, section 4 horizontal 0.46 m and section 5 vertical 0.44 m). The vacuum box has a width of 0.51 m and a length of 0.515 m. The height of the box in the front and back has 0.375 and 0.17 m respectively.

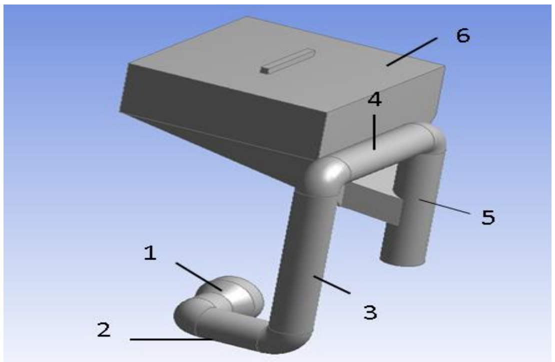


Figure 1. System for cleaning the castor-hulling machine. The components are 1- horizontal section 1, 2- horizontal section 2, 3- vertical section 3, 4- horizontal section 4, 5- vertical section 5, 6- vacuum box.

While the shell collector has a height of 0.52 m in the transversal trapezoidal section, in the lower base and greater 0.5 and 1.1 m respectively. The collector has a total length of 4 m; the dimensions of the fan outlet are 0.16 x 0.16 m x 0.305 m in length. The height in the center of the air outlet and the shell is located 0.335 m away from the lower base of the collector trapezoid.

Geometry and meshing

The 3D drawings were made in the Design Modeler of Ansys Fluent 14. To do this in order to optimize the use of computing resources and due to the symmetry, half of the M D cleaning system is built and modeled (Figure 1) and another of the exit of the material in the fan, for the analysis of the trajectory of the shell. The meshing of the cleaning duct and the collector of the shells was carried out, with a size of the center of fine relevance, using the function of advanced size in the curvatures and a high smoothing, as well as the automatic use of inflation by the controlled program.

Regarding the discretization of the domain, an unstructured mesh of 434 215 tetrahedral elements and 87 318 nodes was used. For the cleaning system, two inlets were established, one for the fluid (air) and another for the material (shells and grains), as well as the outlet and walls of the model. While for the model of the trajectory of the shell, a single entrance was defined for the air and the material, the exit, the walls and the symmetry.

Mathematical models

Model of gas flow. For the simulation, the models of the energy and viscous k - ε equation were first selected with an improved wall treatment.

Model of the flow of the particles. The discrete phase model (DPM) was selected, for the analysis of the trajectory of the grains and shells, which uses the Lagrange approach. The air-grain-shell mixture was analyzed as a flow in the diluted phase, less than 10% solids in the volume of the material. The type of particle selected in the DPM model was inert and injected on the surface of the upper entrance of the vacuum box. The discrete random-pitch mode was selected for the stochastic path of the distribution of the diameters of the material. Modeling of the granular and powder materials was done through the particle size distribution. In this case, the Rosin-Rammler function for computational simulation was selected.

Boundary conditions. All boundary conditions imposed were based on measurements made directly on the object of study. The main air properties used in the simulation are shown in Table 1.

Table 1. Main properties of air for computational simulation.

Properties	Value
Density, kgm^{-3}	1.225
Specific heat, $J\ kg^{-1}\ k^{-1}$	1006.43
Thermal conductivity, $W\ m^{-1}\ K^{-1}$	0.0242
Dynamic viscosity, $kg\ m^{-1}\ s^{-1}$	$1.7894\ x10^{-5}$

In Tables 2 and 3, the initial and boundary conditions used in the simulation for the viscous model k-ε are shown.

Table 2. Boundary conditions for the k-ε model of the castor hulling cleaning system.

Parameter	Input 1	Input 2	Output
Speed, ms^{-1}	6	1	
Pressure, Pa	0	0	0
Intensity of the flow or turbulent reflux, (%)	10	10	10
Hydraulic diameter, m	0.1	0.036	0.152

Table 3. Boundary conditions of the k-ε model for the trajectory of the castor shell.

Parameter	Input	Output
Speed, ms^{-1}	6	
Pressure, Pa	0	0
Intensity of the flow or turbulent reflux, (%)	10	10
Hydraulic diameter, m	0.16	0.59

In the Tables 4 and 5 show the main properties of the material to be cleaned in the simulation of the DPM model. The boundary conditions or boundary of the discrete phase are applied to the physical limits in order to determine the fate of the particle trajectories in the duct and the shell collector of the castor. For the entrance and the exit, the escape option was selected, while for all the walls and the collector, reflect and trap respectively.

Table 4. Properties for the grain and husk of the castor for the DPM model of the cleaning system.

Parameter	Grain	Shell
Speed, ms^{-1}	1	1
Mass flow, kgs^{-1}	0.00825	0.00675
Density, kgm^{-3}	556	125
Specific heat (C_p), kJ/kg K	3.32	3.32
Minimum dimension (D_{gmin}), mm	5	4.5
Maximum dimension (D_{gmax}), mm	20	25
Average dimension (D_{gprom}), mm	14	18
Number of diameters	30	30

Table 5. Properties for the castor shell for the DPM model of the particle trajectory.

Parameter	Shell
Speed, ms^{-1}	6
Mass flow, kgs^{-1}	0.00675
Density, kgm^{-3}	125
Specific heat (C_p), kJ/kg K	3.32
Minimum dimension (D_{gmin}), mm	2.25
Maximum dimension (D_{gmax}), mm	12.5
Average dimension (D_{gprom}), mm	9
Number of diameters	30

Numerical solution

The numerical solution was made using a separate solver and is linearized by the implicit scheme. In addition, due to the fact that in the system to be analyzed there are no mobile structures or sliding meshes, the formulation of absolute velocity was used.

In order to reduce the convergence time of the transient simulation. The CFD simulation was started under stationary conditions to simplify and save computational resources. To derive the centroid variables from the faces of the neighboring cells, the average was used in the Green-Gauss cell. The pressure-velocity coupling method was used to calculate the pressure from the equations of the moment and the continuity equation, through a semi-implicit method of the algorithm of the equations related to pressure (SIMPLE). In addition, of the discretization of the pressure by means of the standard algorithm (that is to say, the interpolation of the pressure for the values of the face); of other dependent variables (for example, momentum, turbulence kinetic energy, turbulent dissipation rate and Reynolds stress) were calculated using the second order upwind scheme.

Experimental analysis of the model. To compare the results estimated by the CFD model, against the experimental measurements, a test bench was built for the cleaning system and an inverted trapezoidal section structure (Figure 2) to collect the shell at the fan outlet. In the lower part, 16 polyethylene bags were placed to collect the shells and grains.

Figure 2. Test bench of the cleaning system and structure for the collection of the shell. Validation of air speed.

The speed was measured in different locations of the duct, the separation chamber and the vacuum box (Figure 3). To this end, Pitot tubes with a diameter of 5 mm were used connected to a differential "U" manometer that uses alcohol as a manometric liquid, obtaining the dynamic pressure for each point. In addition, a Lutron brand hot wire anemometer model YK-2005 AH was used to verify the air velocity in the pipeline. The test bench fan and the feeder rollers were calibrated at 1 500 and 86 rpm respectively, by means of a Lutron tachometer model DT 2230. The fan was driven by a 3 hp three-phase motor ABB brand and the speed was adjusted using the Danfoss model VLT frequency variator.

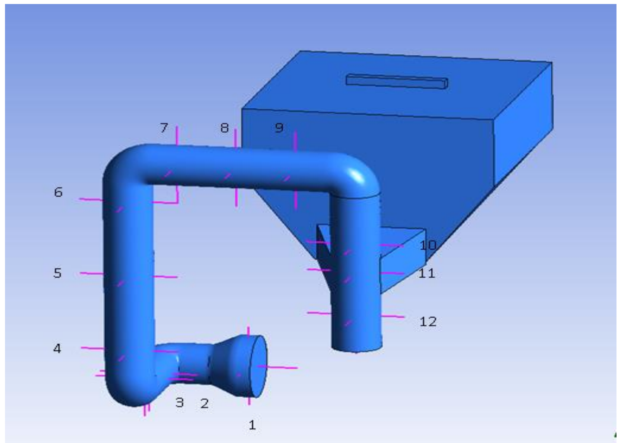


Figure 3. Reading points of the dynamic pressure.

The impurities of the castor capsules were manually separated and dehulled in a dehulling prototype designed and built in the Graduate Course in Agricultural Engineering and Integral Use of Water. The trial consisted of a sample of 3 kg of grains and castor hulls with three repetitions.

The air velocity of the different locations of the duct (Figure 3) was calculated by the following expression.

$$v = \left(\frac{\rho_m}{\rho_a} 2gh\right)^{1/2} \quad (1)$$

Where: v- air velocity, ms^{-1}; ρ_m - density of the manometric liquid, kgm^{-3}; ρ_a- air density, kgm^{-3}; g- acceleration of gravity, ms^{-2}; and h- gauge height or pressure, Pa.

Validation of particle flow

The structure for the collection of the castor hull was placed at the exit of the fan. The sample of the shells collected in each bag was separated by a set of screens of 12.7, 11.11, 9.53, 7.94 and 6.35 mm, with the purpose of classifying and quantifying the particles of the volume analyzed, in separate fractions, according to their dimensions. The total volume and the fractions of particles retained in each sieve were weighed and quantified the percentage by weight of the material under study. The experimental data obtained were statistically processed in the professional package of Microsoft Excel 2010, performing the descriptive analysis of the sample, the analysis of simple variance using a Student t test for paired samples.

Results and discussion

Field of air flow

The field of air velocities was obtained with the numerical models described above considering the grain and the castor shell (Figure 4).

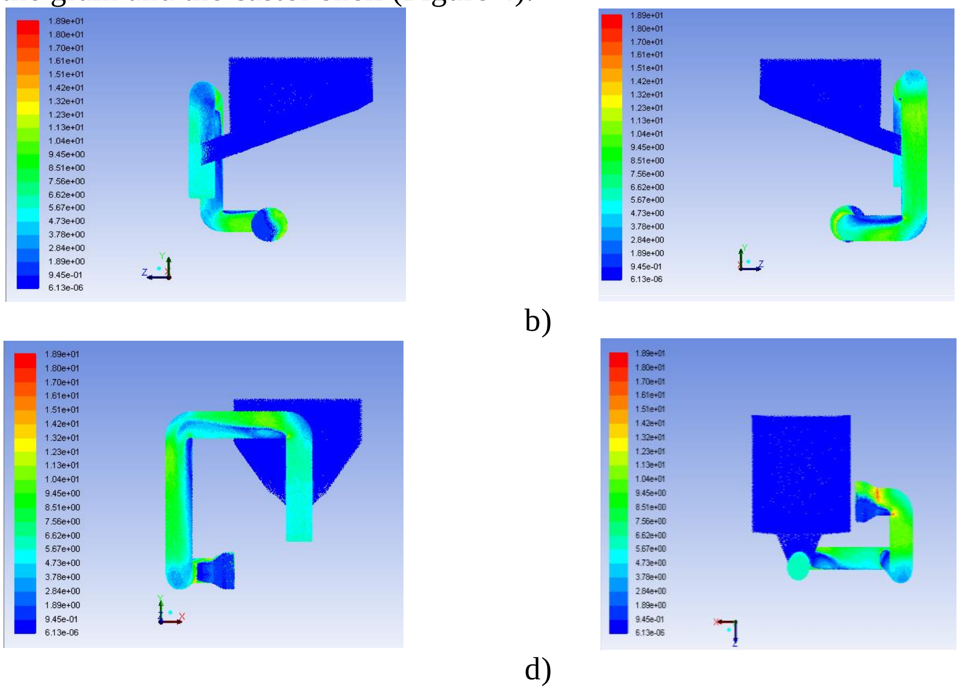


Figure 4. Speed vector. a) Right V.; b) Left V.; c) Frontal V. and d) Lower V.

The Figure 4 shows the different views of the vectorial diagram of the speed of the air flow in the cleaning system, which is colored by the magnitude of it. The speed reached a maximum of 18.9 ms^{-1} of the airflow in the reducer and the elbow of section 1 and a minimum of 6.13 x 10-3 ms^{-1} in the vacuum box.

The magnitude of the velocities obtained, present a considerable congruence with those previously reported by Alves *et al.* (2005), Magalhães *et al.* (2006a); Ponpesh *et al.* (2011a) for a coffee harvesting machine and nut harvester respectively.

Figure 5 shows the comparison of the speed estimated by the model and the experimental, in the different points of the cleaning pipe of the castor-shelling machine. While in Figure 6 a good correlation (R^2= 82%) between the estimated data of the speed and the experimental ones is shown. The standard error is 0.8176. The mean was 5.98 and 5.09 ms^{-1} for the estimated and experimental speed respectively. The standard deviation was 1.85 and 1.63 ms^{-1} for the estimated and experimental speed respectively.

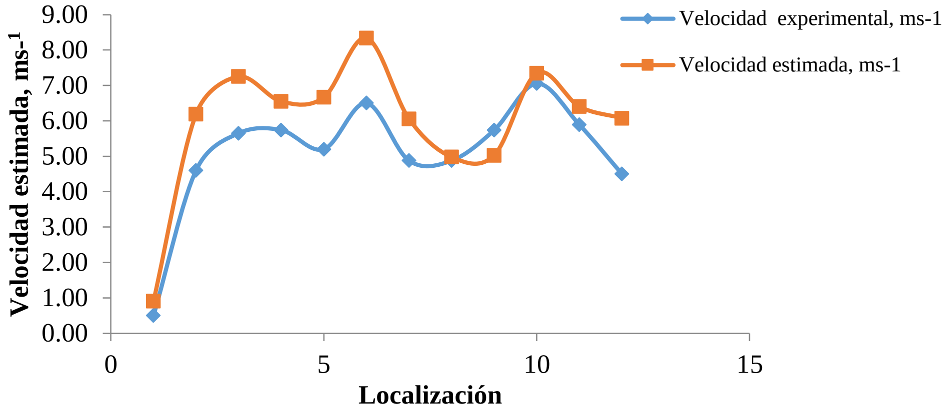


Figure 5. Comparison of the speed estimated by the model k-epsilon realizable and experimental data.

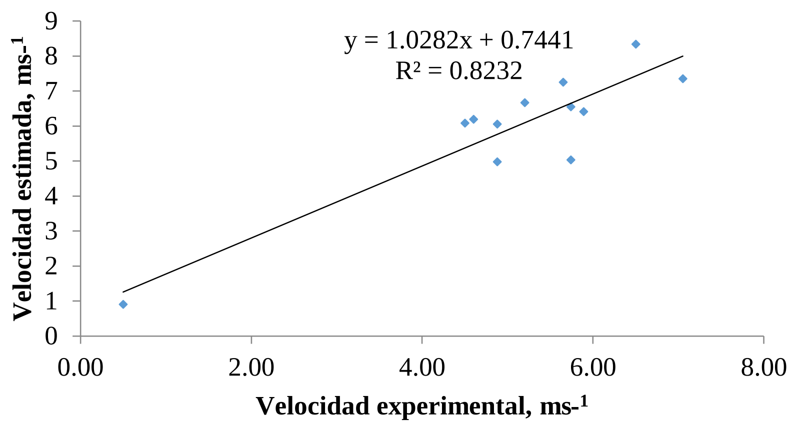


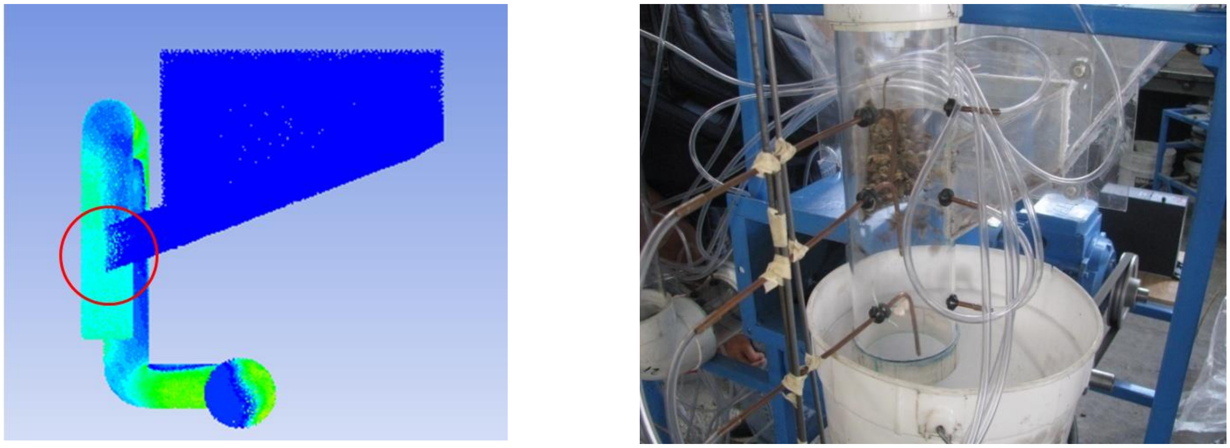
Figure 6. Estimated vs. experimental speed for the castor grains cleaner.

When the means of the estimated and measured speed were statistically compared, a ‘t’ test revealed that there was no significant difference between them at the 5% level of significance (Table 6).

Table 6. Validation parameters of the CFD model with the experimental data of the castor cleaner at the significance level *p*≤ 0.05.

Model	Validation parameters			Significance
	Value of R^2	Value “t_o” calculated	Value “t” of tables	
Speed	0.82	1.245	2.074	Not significant

In the experiment and in the simulation it was observed that there is accumulation of the material to be cleaned at the entrance of the separation chamber (Figure 7), which is why it is necessary to increase the slope angle of the vacuum box or hopper of the peeling chamber, for the construction of the new prototype.



a) b)

Figure 7. Detail of the accumulation of material at the entrance to the main cleaning duct. a) CFD model, b) experiment.

In practice, it was found that the problem is critical if the equipment is powered by an electric motor, in the case of an internal combustion engine it is not, due to the vibration produced by it.

According to Olaoye (2000) and Cruz *et al*. (2012) for metallic surfaces, the angle of repose of the separation chamber should be between 22 and 25°, to avoid obstructing the flow of the grain-shell mixture.

Path of the shell particle. Figure 8 shows the diagram of the contour of the velocity of the air flow and the shell at the exit of the cleaning system, which is colored by the magnitude thereof. The airflow velocity reached a maximum of 6.34 ms^{-1} at the fan outlet, and a minimum of 0 ms^{-1} at the end and at the walls of the shell collector. In modeling, the following assumptions were established: 1) all particles were modeled as equivalent spheres; 2) there is no interaction between particles; y 3) there is no transfer of heat or mass between the particles and the air.

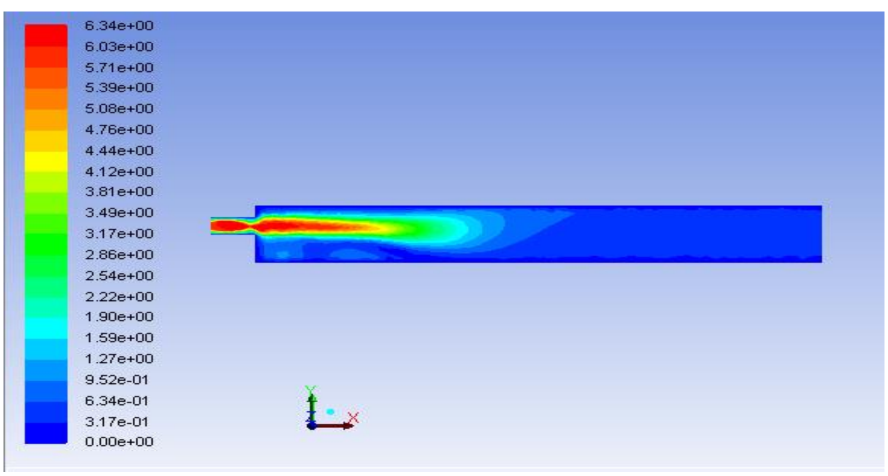


Figure 8. Contour of the velocity of the particle (shell).

The dimensions of the shell of the simulation of the flow of the velocity are half of the trajectory of the particle, since it is assumed that these are crushed when passing through the blades of the fan. The Figure 9 shows the profile distribution of air velocity and shells at a height Y= 0. 335 m, along the length of the collector. The speed reached a maximum of 6 ms^{-1} and a minimum of 0.8 ms^{-1}. The particles are deposited at a maximum distance of 1.5 m from the fan outlet. The length at which the shells are thrown is adequate, since at a certain time interval the machine has to be moved to a new station to peel the castor capsules or to clean the work area.

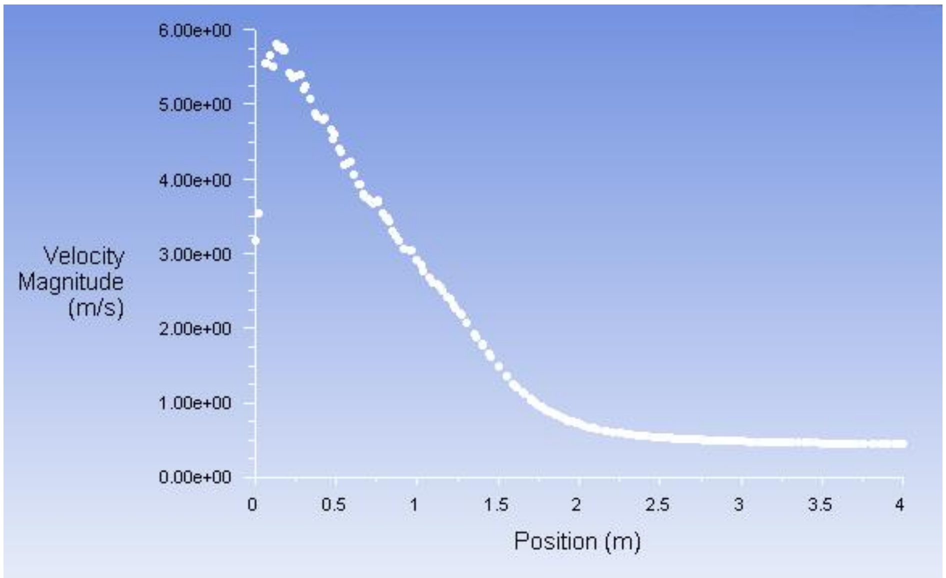


Figure 9. Distribution of air velocity and shells along the length of the collector.

In the Figure 10 shows the trajectory of the particles at the outlet of the fan as a function of diameter, most of these of different sizes fall at a distance between 0.75-1.5 m.

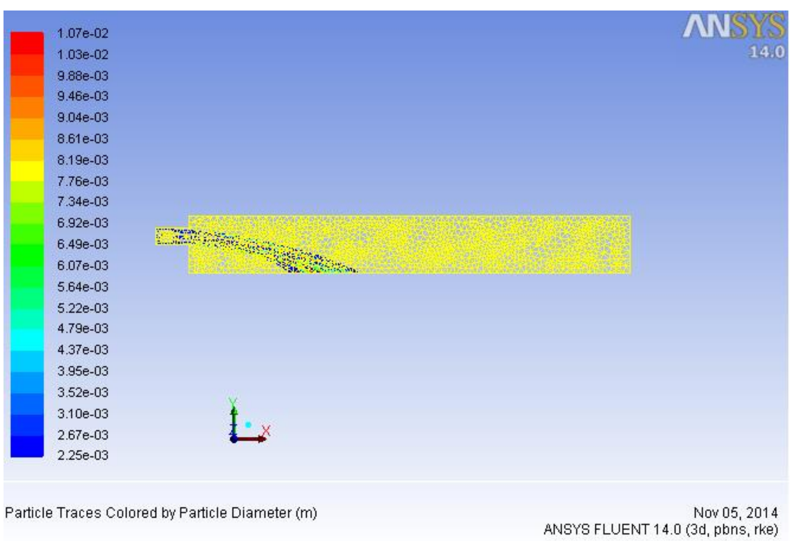


Figure 10. Simulated trajectory of the shell at the exit of the cleaning system.

Other particles are kept in suspension in the atmosphere. The equipment should be oriented in the opposite direction to the direction in which the air is blown to achieve greater quality in cleaning the grain and clean work area.

The Figure 11 shows the percentage by weight of the particles collected experimentally as a function of their diameter. In the 0.5, 0.75, 1, 1.25 and 1.5 m lengths, 9.92, 33.21, 22.98, 15.14 and 7.17% of the weight percentage of the particles are distributed. In the dimension of 0.75 m are the sizes of 12.7, 11.11, 9.53 and 7.94 mm of the shells in the percentages by weight of 45.78, 29.35, 40.70 and 33.82% respectively. While in the length of 1 m, the dimensions of 6.35 and less than 6.35 mm of the particles are distributed in the percentages by weight of 30.93 and 29.12%.

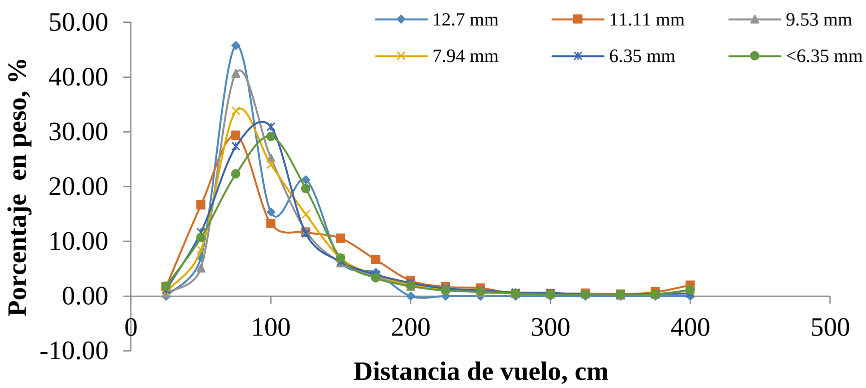


Figure 11. Collection efficiency and flight distance of the shell released by the fan.

The results obtained from the computational and experimental simulation of the particle transport model do not agree with those reported by Bourges *et al.* (2014) for the head of a soybean seeder machine because the material is confined in a chamber.

Conclusions

The comparison of the experimental and computational results showed a good qualitative and quantitative concordance in the velocity profile. The model of the trajectory of the particle adequately predicts the distribution of the material along the collector of castor hulls. It can be concluded that the numerical simulation in CFD using ANSYS FLUENT can correctly predict the quantitative flow pattern and is an alternative method for the study of the separation and cleaning of biological materials for the improvement of the operation and design parameters of the agricultural equipment.

Cited literature

Alves, C. M.; Marcal, Q. D.; Roberto, C. P. y Leyva, R. L. Z. 2005. Desarrollo y modelación de una máquina de pre-limpieza de café en el cultivo de montaña. Rev. Cienc. Téc. Agropec. 14(2):1-13.

Bourges, G.; Eliach, J. y Medina, M. 2014. Comparación entre resultados experimentales y numéricos en un cabezal distribuidor de sembradora “air drill”. Mecánica Computacional. 33(40):2555-2563.

Cernecky, J. and Plandorova, K. 2013. The effect of the introduction of an exit tube on the separation efficiency in a cyclone. Braz. J. Chem. Eng. 30(3):627-641.

Corrêa, J. L. G; Graminho, D. R.; Silva, M. A. and Nebra, S. A. 2004. The cyclonic dryer - a numerical and experimental analysis of the influence of geometry on average particle residence time. Braz. J. Chem. Eng. 21(1):103 - 112.

Cruz, Meza. P.; García, S. S.; Márquez, Rosano. C. y López, C. G. 2012. Determinación de las propiedades físico-mecánicas para el descascarado de las cápsulas de la higuerilla (*Ricinus communis* L.). Rev. Mex. Cienc. Agríc. (4):644-649.

Du, X.; Ni, K.; Chen, J.; Wu C. and Zhao Y. 2013. Numerical Simulation and Experiment of gas-solid two-phase flow in a cross-flow grain cleaning device. ASAE Paper No 131586235, at the 2013 ASABE Annual International Meeting. Kansas City, Missouri. 7 p.

Gebrehiwot, M. G.; De Baerdemaeker, J. and Baelmans, M. 2010 a. Effect of a cross-flow opening on the performance of a centrifugal fan in a combine harvester: computational and experimental study. Biosys. Eng. 105(2):247-256.

Gebrehiwot, M. G.; De Baerdemaeker, J. and Baelmans, M. 2010 b. Numerical and experimental study of a cross-flow fan for combine cleaning shoes. Biosys. Eng. 106(4):448-457.

Herrera, P. M. I.; García de la Figal, C. A. E; De las Cuevas, M. H y Martins, T. M. 2010. Modelación mediante la dinámica de los fluidos por computadora (CFD) de la corriente de aire de un pulverizador agrícola con ventilador centrífugo. Rev. Cienc. Téc. Agrop. 19(3):62 -68.

Herrera, P. M. I.; Martins, T. M.; Rodríguez, G. J. y De las Cuevas, M. H. 2006. Modelación mediante la dinámica de flujo computacional de la corriente de aire de un pulverizador. Rev. Cienc. Téc. Agrop. 15(1):12 -18.

Magalhães, C. A.; Martins, T. M. e Herrera, M. P. M. 2006. Construção e avaliação de uma máquina recolhedora de café emterreiro. Rev. Cienc. Téc. Agrop. 15(2):13-20.

Magalhães, C. A.; Teixeira, M. M.; Couto, M. S. e De Resende, C. R. 2006. Modelagem de máquina pneumática recolhedora de frutos de café em terreiro usando análise por elementos finitos. Engenharia Agrícola. 26(2):483-492.

Ni, K.; Du, X.; Chen, S.; Chen, J. and Zhao, Y. 2013. Study on grain distribution in a cross-flow grain separation chamber based on the discrete element method. ASAE Paper No 131586216, at the 2013 ASABE Annual International Meeting. Kansas City, Missouri. 8 p.

Olaoye, J. O. 2000. Some physical properties of castor nut relevant to the design of processing equipment. J. Agric. Eng. Res. 77(1):113-118.

Ponpesh, P; Giles, D. K. and Downey, D. 2011a. Modeling turbulent multiphase flow for design of nut harvesters with reduced dust emission and low power demand: swirl-tube design. Transactions of the ASABE 54(3):777-788.

Ponpesh, P.; Giles, D. K. and Downey, D. 2011b. Modeling turbulent multiphase flow in design of nut harvesters with reduced dust emission and low power demand. Transactions of the ASABE 54(4):1199-1211.

Ponpesh, P.; Giles, D. K. and Downey, D. 2010. Mitigation of in-orchard dust through modified harvester operation. Transactions of the ASABE 53(4):1037-1044.

Ponpesh, P. and Giles, D. K. 2009. Turbulent gas-particle flow modeling for the mitigation of dust emission from nut harvesters. Ann. Inter. Meeting 2009. ASABE. Reno, Nevada. 21 June- 24 June. 34 p.

Ponpesh, P.and Giles, D. K. 2008. Modeling turbulent flow for design of almond harvesters with low power demand and reduced emission of particulate material. Ann. Inter. Meeting 2008. ASABE. Providence, Rhode Island. 29 June-2 July. 17 p.

Sun, H.; Keener, H. M.; Deng, W. and Michel, F. C. 2004. Development and validation of 3-D CFD models to simulate airflow and ammonia distribution in a High-Rise™ hog building during summer and winter conditions. Agricultural engineering international: the CIGR. J. Sci. Res. Develop. 6:1-24.

Teske, M. E.; Thistle, H. W.; Schou, W. C.; Miller, P. C. H.; Strager, J. M.; Richardson, B.; Butler, E. M. C.; Barry, J. W.; Twardus, D. B. and Thompson, D. G. 2011. A review of computer models for pesticide deposition prediction. Transactions of the ASABE. 54(3):789-801.