International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 2 (2018) pp. 891-895

Transitory Analysis of Fluid Behaviour Inside Hollow Cone Sprinklers Using CFD

Jorge Andrés Sierra de Rio[1], Aldemar Hernandez[1], Leonardo Mesa[2], Diego Hincapié-Zuluaga[1]

[1]*Facultad de Ingeniería, Departamento de Mecatrónica y Electromecánica, Instituto Tecnológico Metropolitano (ITM), Calle 73 No. 76A – 354, Medellín, Colombia.*

[2]*Centro de Tecnología de la Manufactura Avanzada, SENA, Regional Antioquia. Diagonal 104 # 69-120, Medellín, Colombia.*

Abstract

Hollow cone sprinklers are devices widely used in the industry to atomize liquids such like water in a gas such as air, to achieve a proper atomization process, the sprinkler efficiency must be as high as possible, otherwise it will be necessary to perform the processes again and the costs increase. A parameter for the measurement of efficiency of these devices is the aspersion distributed by volume of water, and this is achieved with the analysis of the fluid at constant pressures. This article presents the evaluation via CFD simulation of a hollow cone sprinkler WT400 in the CFX® module of the Ansys Workbench V17.0® program. The evaluation analyses the performance of the sprinkler when there is a 29.7% change in the exit angle of the sprinkler hollow cone, compared to its initial configuration. Computational analysis of volume fraction, velocities in symmetry planes and flow lines are carried out; Experimentally, spray distribution of the configurations are evaluated.

Keywords: Two-phase CFD, sprinkler, hollow cone, cone angle.

INTRODUCTION

Hollow cone sprinklers and their use in different fields of industry have led to different dispersion studies. This is the case of pesticides used in agriculture, which are generally applied by means of sprinklers that determine the quality of distribution of said pesticide for the preservation of plants [1], where the efficiency of the distribution of pesticides in tobacco plants is studied with a hollow cone sprinkler, evaluating the dispersion coverage. This coverage makes these nozzles an important component in the atomization of the fluid for applications dust elimination with collectors [2], and diverse applications like the dry cooling towers of natural draft, where the conservation of the water is sought due that the use of these towers in arid zones and power plants requires pre-cooling systems with water spray to achieve rapid evaporation [3].

A hollow cone sprinkler consists of a distributor that has a hydrocyclonic form on its upper part, a cylinder in the middle and an adjacent conical part at the bottom area, as shown in figure 1.

Figura 1. Operation of a hollow cone nozzle. [4]

The distributor generates a rotational flow that accelerates the fluid while it is approaching the central zone as the distributor diameter decreases towards it. If the operating conditions are adequate in the intermediate zone the fluid descends in contact with the surface having a high angular momentum, thus, forming an air core in the center. In the conical part of the sprinkler the fluid forms a film due to the coanda effect [5] which decreases its thickness until it falls off forming small drops.

The output geometry [4] is one of the main characteristics in the performance of the nozzle. However, the variation of conditions and type of flow can make the hollow cone nozzle one of the most suitable devices for performing various spray applications such as gasoline spraying [6], or in nuclear applications [7], where can perform experimentation under operating conditions determined by the manufacturer and variation of the injection angles [8], leading to develop previous models in 3D simulations that can deliver results from the behaviour of the characterization of said model [9]. Other methods such as image processing experimentation for the study of the spray in a hollow cone nozzle [10] can analyse behaviour patterns of a biphasic transient fluid in its output distribution [11]. However, this type of experimentation has a long process of generating cyclical information to obtain results.

In this paper we present the modelling of a commercially available BETE WT400 hollow cone nozzle in Ansys Workbench V17, which represents one of the most suitable computational tools [12] for the characterization of this device to make improvements in design [13] [14] by optimizing the geometry determined by the dispersion distribution size and the mass flow conservation [14]. This is achieved by making a computational analysis of fluid dynamics (CFD) [15] to obtain simulations that are close to reality, a method that has been explored for the evaluation of theoretical models [16] [17]. Finally, the validation of the best geometric configuration at the same operating conditions will be determined through the manufacture of the improved nozzle for the evaluation of both performances.

METHODS AND MATERIALS

In order to improve the angle of exit of the sprinkler, it starts from a design of a commercial hollow cone sprinkler BETE® WT 40080 (straight nozzle) figure 2.

Figure 2. Aspersor de cono hueco comercial de referencia (Boquilla recta a derecha-Copa Distribuidor espiral izquierda)

For the CFD analysis of the sprinklers, initially we proceeded to the configuration of the geometric model of a straight nozzle WT-400 as shown in Figure 3 and 4, this was done using the CAD tool Autodesk Inventor 2016.

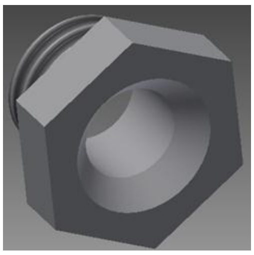

Figure 3. CAD model of straight studio nozzle.

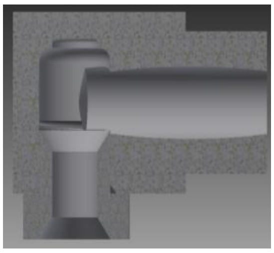

Figure 4. CAD model of spiral dispenser.

Once obtained the volume of fluid occupied inside each of the 2 assemblies (cup and nozzle) and the spherical outlet chamber, in which the fluid injection is made, meshing was carried out by means of the meshing module Meshing® of the Ansys Workbench V17 for both geometries, adapting the mesh parameters (obliquity, aspect ratio, expansion factor, Figure 4) according to the software requirements. A total of 1.4x106 elements were required to guarantee the independence of the mesh in each of the simulations, which allows to guarantee the reduction of the error by discretization of the domain.

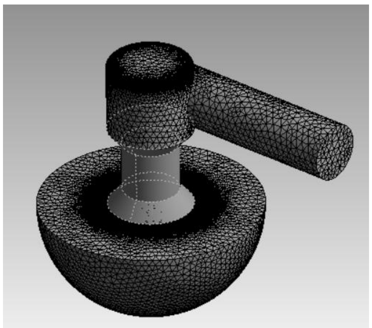

Figure 4. Meshing of the control volume of the cup-straight nozzle assembly and injection chamber.

The boundary conditions were established according to the operating conditions of the sprinkler, which on the recommendation of the manufacturer, 2 scenarios were established varying the supply pressure of 10 and 30 Psi and the flow of 2.0 and 3.5 GPM respectively, wall conditions without slippage in order to estimate the friction of the sprinkler with the fluid and pressure gauge opening conditions 0 Pa for the injection chamber.

The resolution of the equations of conservation of mass and momentum for a fluid in rotation were made using the CFX® module of the Ansys Workbench V17.0. With transient simulations, with step times of 0.002 s, 25 iterations per step time for a total simulation time of 1 s, were performed for each of the geometries. The configuration of the fluid in the simulation corresponds to a diphasic fluid made up of water and air, both with properties at 25° C, the selected turbulence model corresponds to the kw due to the high vorticity that occurs in the fluid inside and at the outlet of the hollow cone nozzle.

From the simulation obtained, the geometric modification of the straight nozzle is made by expanding the angle of inclination of the exit walls of the nozzle, resulting in a conical pattern, without modifying the generation chamber of tangential rotation to water atomization, as shows in Figures 5 and 6.

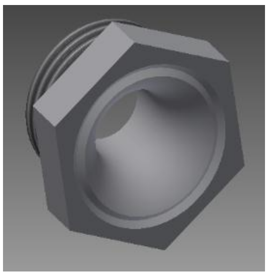

Figure 5. CAD model of hollow cone nozzle with geometry modification.

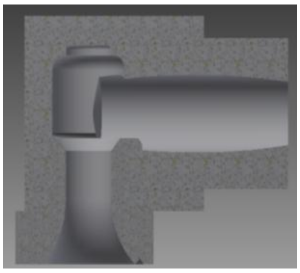

Figure 6. CAD model of spiral distributor with geometry modification.

Figures 7 and 8 show a comparison of the volumetric fractions (100% water red and 100% air blue) between the hollow cone nozzle sprinkler for the operating conditions of 30 Psi and 3.5 GPM. From this we can highlight the increase in the opening angle of the injection cone, going from 81.4° for the straight nozzle and 121° for the curved nozzle.

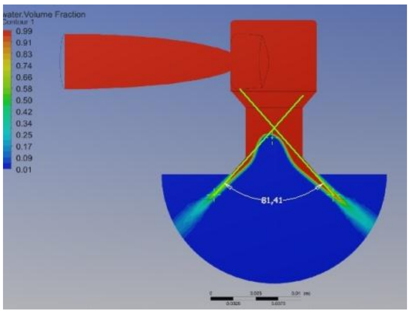

Figure 7. Contours of volumetric fraction in plane of symmetry of the straight nozzle sprinkler**.**

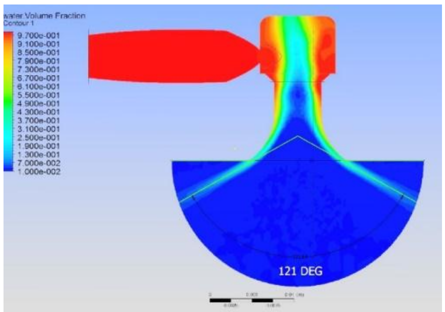

Figure 8. CAD model of spiral distributor with modification of hollow cone nozzle geometry**.**

The experimentation phase to test the obtained design uses a 795 mm diameter tank and a height of 980 mm with a pumping system, the mass flow measurement is taken by means of a rotameter that will allocate the equivalence with the mass flow from 0.22 kg/s to a pressure of 30 PSI according to the operation values of the manufacturer BETE [18]. Then, the distribution of the circular pattern is measured by dividing the tank into an octagon, to section the circular spray pattern into eight equal parts and to measure its distributed volume as shown in Figure 9.

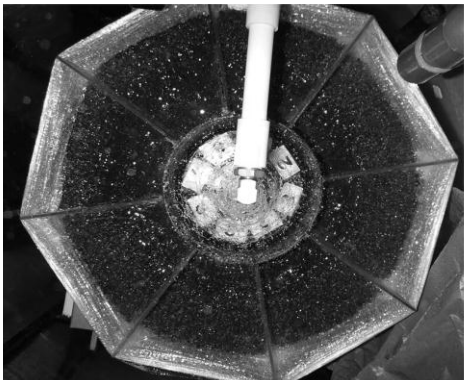

Figure 9. Distribution octagon

RESULTS AND DISCUSSION

In the experimental results an increase in the dispersion due to the direction of exit of the water with the new geometry and the increase in the speed of rotation of the vortex was evidenced, which can be evidenced by the volumetric fraction of air that enters the sprinkler as it is seen in Figure 6, from this one can infer a high speed of rotation of the water, which generates an exit angle greater than that provided by the geometry of the straight nozzle.

Due to the octagonal division of the tank, the volumetric distributions deposited in each of the cells are obtained. These volumes differ both in the experimentation with the straight nozzle and the hollow cone nozzle, since the entrance of the flow is in the direction of the position 5 of the octagon Figure 7. This causes the distribution of the volumes to be unbalanced with respect to the position 1, as evidenced in Figure 10 and Figure 11.

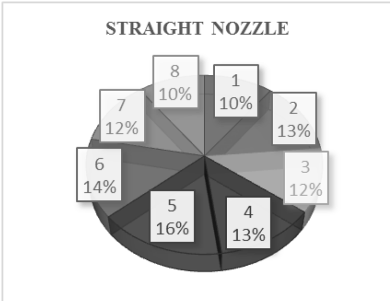

Figure 10. Volume distribution in straight nozzle.

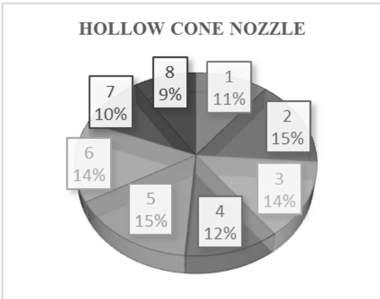

Figure 11. Volume distribution in hollow cone nozzle.

From these distributions of volumes between both nozzles, an increase in the distribution of volume more uniform in the nozzle verified by the weight of the water contained in each of the cells of the octagon, as shown in Figure 12, is presented.

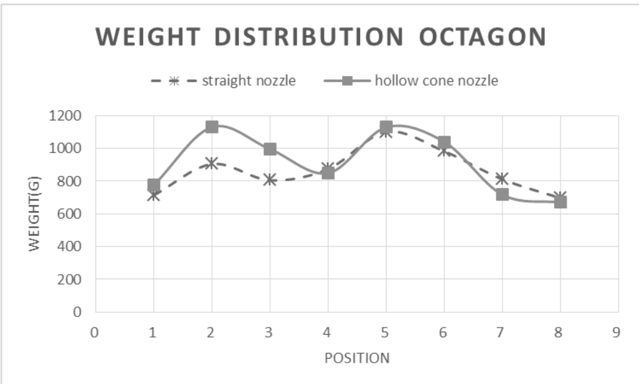

Figure 12. Weight distribution in the octagon.

To visualize the performance of the opening angle in the straight nozzle, one can compare the results of the CFD simulation in contrast to the actual nozzle at a pressure of 30 psi, as shown in Figures 13 and 14.

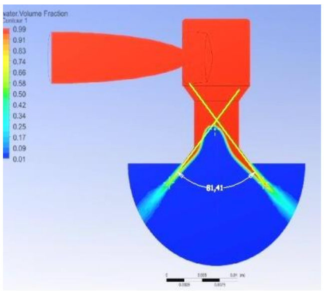

Figure 13. Spray angle straight nozzle CFD simulation.

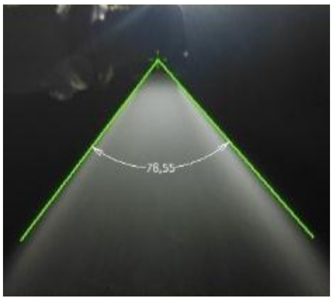

Figure 14. Spray angle straight nozzle experimental test.

In this comparison there is a reduction in the spray angle of 3.5%, similarly between the model and the experimental test in the hollow cone nozzle, the reduction is 15.7%, Figures 15 and 16.

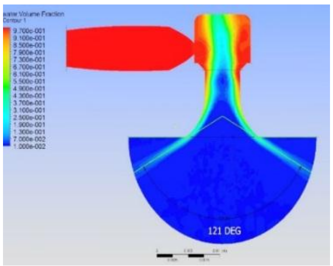

Figure 15. Spray angle hollow cone nozzle CFD simulation.

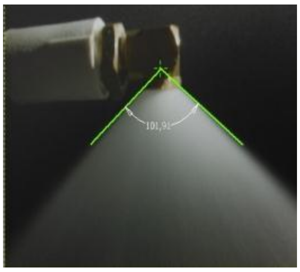

Figure 16. Spray angle hollow cone nozzle CFD simulation.

To improve the modification of the straight geometry in the opening of the nozzle, with a modification made by ANSYS, there is an increase of 29.7%, when going from an angle of 78.55 ° to 101.91 °.

Table 1 shows the results for the 2 simulation scenarios, from which it can be highlighted that with the curved (improved) nozzle a greater cone angle can be obtained, regardless of the operating conditions, guaranteeing a saving in fluid consumption of work.

Table 1: Cone angle for 2 operating conditions.

Nozzle	Preasure [Psi]	Flow [GPM]	Cone angle [degrees]
Straight	30	3.5	81.41
Curve			121
Straight	10	2.0	77.05
Curve			101.9

CONCLUSIONS

Fluid-CFD computer simulation allows the designer to parameterize the design of a sprinkler, allowing him to make both geometric and operational variations with shorter verification times compared to the experimental process.

Increases in the opening angle in the injection cone were obtained for the hollow cone nozzle, this without changing the operating conditions with respect to the straight commercial nozzle, this indicates that the consumption of working fluid can be reduced, requiring less sprinklers by area unit, which improves industrial processes and guarantees the protection of the environment in industrial processes in which water is used as a working fluid.

Despite the reliability of the computational results, it is required in the future to perform experimental validations of the computational model developed in this work.

REFERENCES

[1] C. Feng, C. Zhang, F. Kong, and J. Wang, "Effects of spray height and spray angle on spray deposition in tobacco plants," *Proc. 2011 6th IEEE Conf. Ind. Electron. Appl. ICIEA 2011*, pp. 2390–2393, 2011.

[2] W. Cheng, Y. Ma, J. Yang, and B. Sun, "Effects of atomization parameters of dust removal nozzles on the de-dusting results for different dust sources," *Int. J. Min. Sci. Technol.*, 2016.

[3] Y. Sun, Z. Guan, H. Gurgenci, X. Li, and K. Hooman, "A study on multi-nozzle arrangement for spray cooling system in natural draft dry cooling tower," *Appl. Therm. Eng.*, vol. 124, pp. 795–814, 2017.

[4] S. Nonnenmacher and M. Piesche, "Design of hollow cone pressure swirl nozzles to atomize Newtonian fluids," *Chem. Eng. Sci.*, vol. 55, no. 19, pp. 4339–4348, 2000.

[5] M. Trancossi, J. Stewart, M. Subhash, and D. Angeli, "Mathematical model of a constructal coanda effect nozzle," *J. Appl. Fluid Mech.*, vol. 9, no. 6, pp. 2813–2822, 2016.

[6] Y. S. Shim, G. M. Choi, and D. J. Kim, "Numerical and experimental study on effect of wall geometry on wall impingement process of hollow-cone fuel spray under various ambient conditions," *Int. J. Multiph. Flow*, vol. 35, no. 10, pp. 885–895, 2009.

[7] A. Belhadef, A. Vallet, M. Amielh, and F. Anselmet, "Pressure-swirl atomization: Modeling and experimental approaches," *Int. J. Multiph. Flow*, vol. 39, pp. 13–20, Mar. 2012.

[8] K. M. Bade, W. Kalata, and R. J. Schick, "Experimental and Computational Study of a Spray at Multiple Injection Angles Spray Analysis and Research Services Spraying Systems Co .," *Am. 22nd Annu. Conf. Liq. At. Spray Syst.*, no. May, pp. 2010–2010, 2010.

[9] J. Malet, a Foissac, C. Vayaboury, T. Gélain, and S. Mimouni, "Numerical study on the influence of

simplified spray boundary conditions for the characterization of large industrial safety spray systems used in nuclear reactors Institut de Radioprotection et de Sûreté Nucléaire , BP 68 Gif-sur-Yvette , France," pp. 2–9, 2012.

[10] H. Zhang, B. Bai, L. Liu, H. Sun, and J. Yan, "Droplet dispersion characteristics of the hollow cone sprays in crossflow," *Exp. Therm. Fluid Sci.*, vol. 45, pp. 25–33, Feb. 2013.

[11] J. Hu, D. Liu, and L. Fang, "Image-based analysis of atomization characteristics of a hollow cone nozzle," *Proc. - 2013 5th Conf. Meas. Technol. Mechatronics Autom. ICMTMA 2013*, pp. 610–614, 2013.

[12] M. Jain, B. John, K. N. Iyer, and S. V. Prabhu, "Characterization of the full cone pressure swirl spray nozzles for the nuclear reactor containment spray system," *Nucl. Eng. Des.*, vol. 273, pp. 131–142, Jul. 2014.

[13] F. Li, K. J. Brown, W. Kalata, and R. J. Schick, "Analysis and Optimization of Spray Tower in WFGD," 2014.

[14] R. Schick, *ANSYS and Spraying Systems: Innovating in a Rapidly Changing Environment with Simulation*. 2014.

[15] J. W. Ding, G. X. Li, Y. S. Yu, and H. M. Li, "Numerical Investigation on Primary Atomization Mechanism of Hollow Cone Swirling Sprays," *Int. J. Rotating Mach.*, 2016.

[16] X. Liu, R. Xue, Y. Ruan, L. Chen, X. Zhang, and Y. Hou, "Flow characteristics of liquid nitrogen through solid-cone pressure swirl nozzles," *Appl. Therm. Eng.*, vol. 110, pp. 290–297, 2017.

[17] A. Alkhedhair, I. Jahn, H. Gurgenci, Z. Guan, S. He, and Y. Lu, "Numerical simulation of water spray in natural draft dry cooling towers with a new nozzle representation approach," *Appl. Therm. Eng.*, vol. 98, pp. 924–935, 2016.

[18] T. Fidalgo, "Bete Catalogue Nozzles For Industry, Pollution Control, And Fire Protection." BETE fog nozzle, p. 127, 2003.