J Braz. Soc. Mech. Sci. Eng. (2016) 38:2393–2401
DOI 10.1007/s40430-015-0348-y

TECHNICAL PAPER

Computational study of the air flow dynamics in an induced draft cooling tower

Juan S. Velandia[1] · Marie Chery[2] · Omar D. Lopez[1]

Received: 11 September 2014 / Accepted: 23 March 2015 / Published online: 2 April 2015

Abstract Cooling towers are one of the main solutions to reduce the water temperature used in industrial facilities. The air flow dynamics inside these devices is of especial interest, since it determines important operational variables such as power consumption. The present paper focuses on the air flow dynamics of two different geometrical configurations of induced draft cooling towers: a squared transverse area (referred as "FV25") and a circular transverse area (referred as "FV25c"). The complexity of the flow in some elements of the cooling tower such as the filmic fill and the drift eliminator was simplified by modeling them as porous media. The numerical parameters of the porous media models for these zones were based on detailed simulations of both elements in a wind tunnel-like configuration. Comparison of both configurations of the towers was made in terms of velocity and pressure contours, pressure drop and power consumption. It was found that the power used by the FV25c tower was only 60 % of the power used by the previous FV25 model due to complexity on the flow interaction and sudden transverse section changes. Computational fluid dynamics has proven to be a useful tool for the design process of cooling towers in the Colombian industry, as it allows understanding details of the air flow dynamics inside the cooling tower.

Keywords Computational fluid dynamics · Cooling tower · Air flow dynamics · Porous media modeling

Technical Editor: Francisco Ricardo Cunha.

✉ Juan S. Velandia
js.velandia157@uniandes.edu.co
URL: http://www.uniandes.edu.co/

Marie Chery
MChery@edospina.com.co
URL: http://www.edospina.com/

Omar D. Lopez
od.lopez20@uniandes.edu.co
URL: http://www.uniandes.edu.co/

1 Department of Mechanical Engineering, Universidad de los Andes, Cra 1 Este #19A-40, Bogotá, Colombia

2 Edospina, Cra 7 #24-89, Bogotá, Colombia

1 Introduction

One of the main problems that many industrial facilities face today is the amount of heat generated during their operating cycle. It is common to find facilities where, after cooling any process, hot water is thrown away, leading to environmental hazards for the community and increasing cooling costs in the production process. Heat exchange becomes an important issue for almost any application, from manufacturing to energy generation processes.

To improve heat waste management and looking for a "greener" industry, solutions as heat exchangers or cooling towers are being implemented. Both solutions reduce the problem by taking the heat generated (as hot liquid) during any process and converting it partially. Then, the "cooled" liquid returns to the system and restarts the cycle. This way, the industry saves resources in the cooling process and reduces its environmental impact. When the space available for these devices is reduced, cooling towers are a better option than heat exchangers. Different sizes and shapes in cooling tower are commonly used in many facilities such as nuclear power plants, manufacturing industries or air-conditioning systems. Not only sizes and shapes define the versatility of this solution, but also differences in the draft generation mechanism (natural [1, 2] or mechanical [3, 7]) or the cooling pads geometry [5, 7, 11] increase the chances to reach a custom-made solution. During the design process of such solutions, computational fluid dynamics (CFD) plays a major role, where experimental and manufacturing costs are significantly reduced.

Multiple complex physical phenomena, such as flow dynamics, heat and mass transfer take place simultaneously in a cooling tower. The present work focuses only on the air flow dynamics and does not consider any water dynamics, heat or mass transfer. The air flow dynamics in a cooling tower plays an important role in several aspects of the tower such as heat exchange (in the fill section) and power consumption. The geometry of the cooling tower should guarantee the uniformity of the air flow in the fill section aiming at an improvement on heat exchange [6]. Other factors as the geometry of the tower, its transverse section and the selection for the filmic fill are related to pressure loss of the air flow along the tower, which determines the power consumption and the operative cost over a year. The understanding of the air flow inside a cooling tower is the first step in its conceptive design.

In the present study, two configurations of cooling tower were analyzed, named “FV25” and “FV25c”; these towers are offered by a Colombian company called EDOSPINA in the Colombian market as a solution to air-conditioning systems. Both configurations are induced draft towers and count with a filmic fill and a drift eliminator section. The FV25 tower has a square transverse section, while the FV25c has a circular section; both were designed for similar operating conditions and used the same fill and drift eliminator geometry.

One of the limitations of CFD in the simulation of the air flow along a cooling tower is the different geometrical scales involved in the problem. Due to the presence of the fill and drift eliminator, the air has to flow through small channels [6], which means that an appropriate mesh would require a large number of elements implying an increment on the CPU time. Because of this limitation, a sub-model for the fill and drift eliminator was developed [5] and simulated to characterize both elements and simplify the full tower model by replacing them with porous media.

The main objective of this work is to compare the air flow on two different geometrical configurations of induced draft cooling tower that were designed for the same operational condition. This comparison should lead to the identification of problems on the air flow on both towers that must be addressed by the design group at EDOSPINA. This work represents the first step in the implementation of CFD as a tool in the cooling tower design and manufacturing business in Colombia. It is important to mention that most of the computational tools used in the design of cooling towers are based on semi-empirical models that are typically zero or one dimensional and do not take in account more complex flow dynamics and heat transfer phenomena that take place inside the cooling tower [1]. This work suggests a new approach where cooling tower design process does not depend on experimentation or technical reports.

2 Computational model

The present work uses computational models to predict the airflow inside a cooling tower. To do so, a detailed CAD model of filmic fill and drift eliminator was built; then, both elements were simulated in a wind tunnel-type configuration. This approach was taken to characterize the large-scale behavior of both elements to relate the pressure drop of the air passing through them with its velocity. Results from such characterization were used as an input in the whole tower model, where filmic fill and drift eliminator were replaced by porous media.

2.1 Fill and drift eliminator

Filmic fill is one of the main components in a cooling tower. Its role is to increase the heat transfer area by forcing the air and the water to pass through several independent channels. The filmic fill is built by packing thermoformed plastic sheets alternating its direction. A 3D CAD of the packing as well as the form of the sheet can be seen in Fig. 2a. In this case, the channels have a trapezoidal shape shown in Fig. 2b, where two adjacent channels are highlighted. The height of each channel is technically known as the “flute” of the filmic fill; in this case the flute is 12 mm. The advantage of such geometry is to increase the area where heat transfer occurs; nevertheless, a significant pressure drop in the air flow is induced by its presence. A sketch of the path the air should follow inside the fill is shown in Fig. 1c. The fill was modeled as a cubic section of 30 cm × 30 cm × 30 cm of a standard fill used for cooling tower applications with 12 mm flute height (see Fig. 1a).

On the other hand, the drift eliminator is used to reduce the amount of water that is lost in the cooling process by evaporation. Moist air is blown outside the tower by the draft generated; the role of the eliminator is to catch this air and condense most of the water evaporated during the heat and mass transfer process that occurs in the fill. Drift eliminator is also generated by packing thermoformed plastic sheets; however, this time, they all have the same direction but are translated a few centimeters. The array, composed of 24 plastic sheets, can be seen in Fig. 2a. Details of the flute is also shown in Fig. 2b, where the irregularity of the array can be appreciated, as all the channels are not of the same size. The path that should be followed by the air is also shown in Fig. 2c, where a similar shape to the one present in the pack (Fig. 2a) can be seen. The drift eliminator was modeled with a section of 24 cm × 30 cm × 30 cm

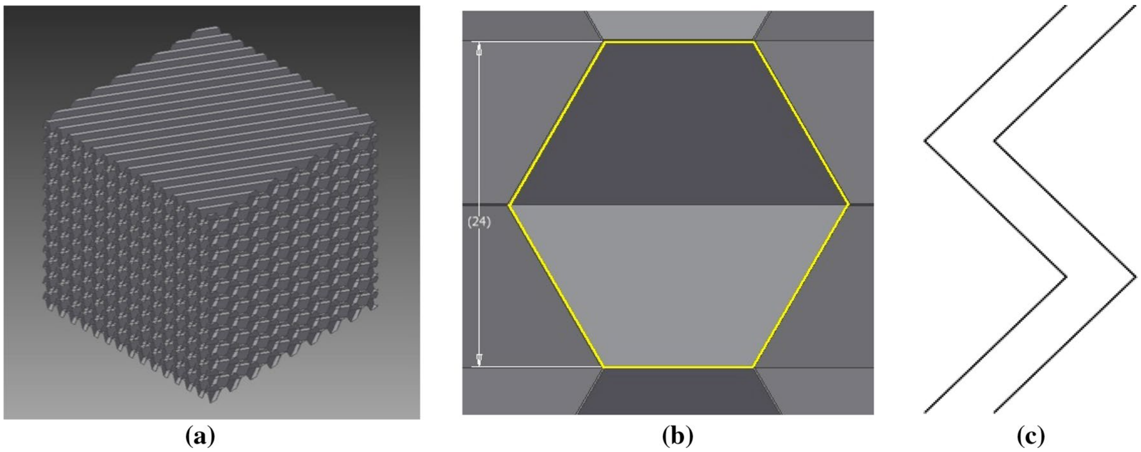


Fig. 1 Filmic fill packing (**a**), detailed flute (**b**) and independent channel (**c**)

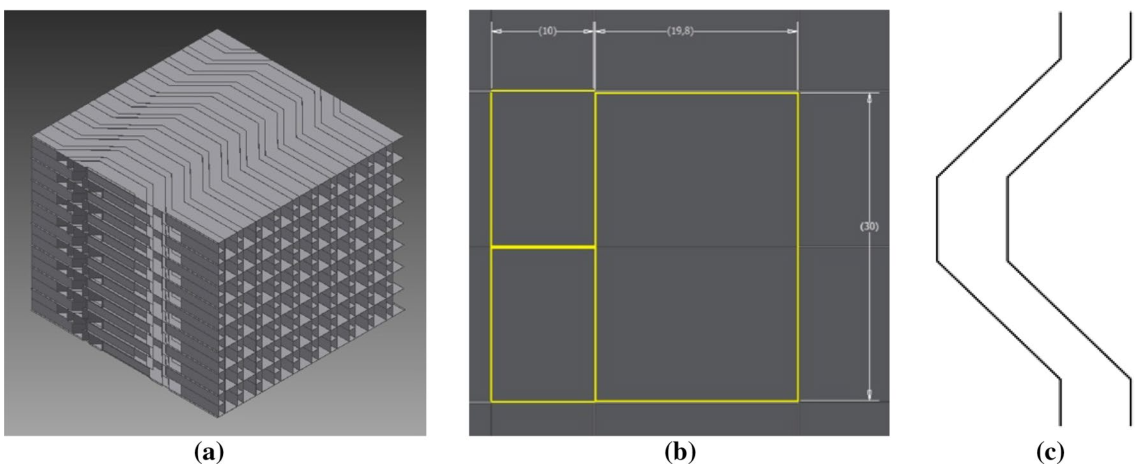
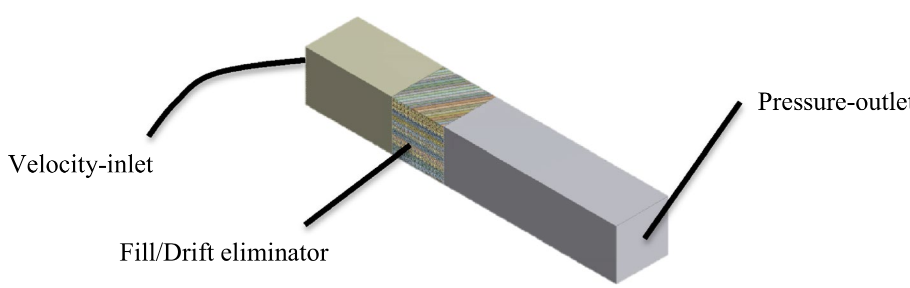


Fig. 2 Drift eliminator packing (**a**), detailed flute (**b**) and independent channel (**c**)

Fig. 3 Wind tunnel configuration



where the flute height is around 15 mm and its width varies between 10 and 20 mm.

Fill and drift eliminator geometries were placed in a wind tunnel-like computational domain, as shown in Fig. 3. The inlet section is located 0.5 m upstream of the device and the outlet section 1 m downstream. The length of the domain prevents any influence of flow disturbances on the total pressure drop. The inlet was modeled as a velocity-inlet condition, while the outlet was modeled as a pressure-outlet condition as shown in Fig. 3.

According to EDOSPINA, both the fill and the eliminator deal typically with flow velocities of around 3 m/s, but in the present study a range of velocities between 1 m/s and 6 m/s were tested. This way, it was possible to characterize the performance of the fill and the eliminator in terms of pressure drop against velocity in a wider range of operating conditions.

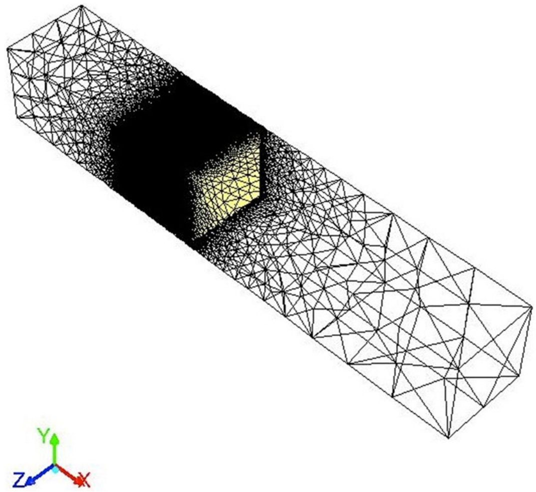


Fig. 4 Coarse mesh for filmic fill

Table 1 Mesh size for fill and drift eliminator in millions of elements

Geometry	Coarse	Medium	Fine
Fill	1.7	6	9
Drift eliminator	1.5	5	11

Table 2 General dimensions of both towers

	FV25	FV25c
Height (m)	3.04	2.41
Fill section diameter (m)	0.95	0.61
Air inlets windows	10	Continuous
Air inlet area (m^2)	0.849	1.169
Air outlet area (m^2)	0.196	0.3643

The computational domain was discretized with tetrahedrons as shown in Fig. 4. Three different meshes (coarse, medium and fine) were generated for the fill and the drift eliminator. Table 1 shows the size of the grids used in the present study in millions of elements. A convergence analysis showed that the medium mesh balanced CPU time and accuracy of results for both geometries. Therefore, any further simulation was made using the medium mesh.

The main objective of the detailed model of the flow dynamics of the fill and the eliminator is to obtain a simplified model of these devices; typically, this is done by matching the behavior of the pressure drop along the fill and the drift eliminator for different inlet velocities to a porous media [8–11]. This approach is based on the Forchheimer equation (See Eq. 1) where the pressure drop (ΔP) of the porous media depends on the averaged velocity of the flow (V), the length of the medium (L) and two numerical parameters: a viscous resistance ($1/\alpha$, where α represents the permeability of the medium) and an inertial resistance parameter (K).

$$\Delta P = \left(\left(\frac{1}{2} K \rho L \right) V^2 + \left(\frac{\mu}{\alpha} L \right) V \right). \quad (1)$$

Once the computational model is run for a given inlet velocity, the drop pressure is obtained by subtracting the mean pressure at the entrance and at the exit of the tested element. With (ΔP) computed at different averaged velocities (V), the viscous and inertial resistance are estimated for the fill as well as for the drift eliminator. This simplification saves considerable computing time and meshes refinement in both zones, reducing cost of simulations.

2.2 Cooling tower

Induced draft cooling towers are recognized by the presence of a fan on the top of the tower which generates vacuum inside the tower. This type of towers are typically installed in vertical position, so that the cool air flows into the tower at the bottom and the hot and moist air leaves the tower at the top. The interior of the cooling tower has several different zones, from top to bottom there are: fan (air exit), drift eliminator, spray zone, rain zone, filmic fill, air entry and the pool. The hot water that enters the tower below the drift eliminator is sprayed at the spray zone and drops due to gravity into the filmic fill. Once the mass and heat transfer process that occurs between the water and the air at the fill is done, the cold water is gathered at the bottom of the tower in a pool.

The geometry of both cooling towers was modeled by CAD. Geometry simplifications were made for each model, taking advantage of symmetries. General dimensions of both towers are shown in Table 2. One of the main differences between the two models is that FV25c has one continuous air inlet, while FV25 has only few windows where the air can enter the tower.

The tower FV25c (Fig. 5a) was modeled using its axial symmetry as can be seen in Fig. 5b. As the FV25 tower is square shaped, the geometry is considered symmetric by its longitudinal and lateral side, which means that a geometric model of a quarter of the tower (Fig. 5d) would be enough. The mesh for the FV25c (Fig. 5c) tower was generated with quadrilaterals, leading to a structured mesh. On the other hand, the mesh for the FV25 (Fig. 5e) tower is unstructured, composed of tetrahedra. The sizes of both grids in its coarse, medium and fine versions are shown in Table 3.

The computational domain of the geometric model of both towers is subdivided into regions according to the different zones inside the tower. Interfaces were set between the spray, rain and porous zones (fill and drift eliminator).

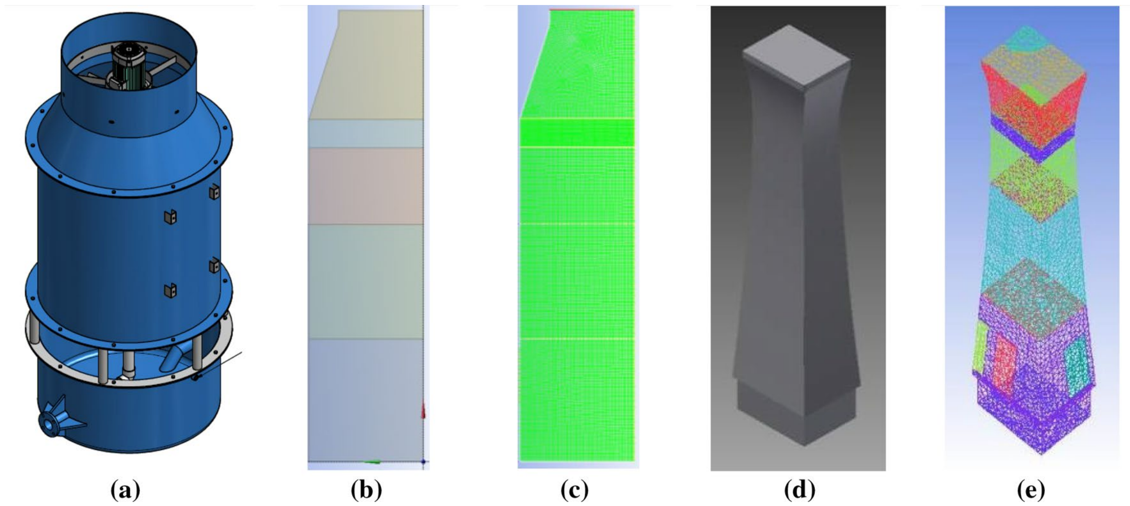


Fig. 5 Geometry and mesh used for the FV25c model (**a**, **b**, **c**) and FV25 model (**d**, **e**)

To simulate an induced draft effect, air outside the tower was modeled with a pressure-inlet condition at atmospheric pressure located far (one radius of the tower for the FV25c and one side of the tower for the FV25) from the inlet of the tower. The draft is then generated by an exhaust fan condition at the top of the tower, where a negative pressure was set. Figure 6 shows the mentioned boundary for the FV25c tower, and the remaining boundaries were set as walls (no-slip). Boundary conditions were the same for the FV25 tower.

The operating conditions are slightly different for each tower in order to match the design flow rate. Both towers were tested for atmospheric conditions corresponding to an altitude of 2600 m AMSL [75 kPa (absolute), $\rho = 0.9375$ km/m^3, $\mu = 1.7034\text{E}{-5}$ kg/m^{-s}]. The exhaust fan boundary condition was set to -180 Pa for the FV25 tower and to -85 Pa for the FV25c. The mentioned conditions led to air flow rates of 2.15 m^3/s in the FV25 tower and 3.87 m^3/s in the FV25c.

2.3 Solver setup

All cases were solved with ANSYS Fluent v.14.5; the air was assumed to be dry, Newtonian and incompressible and any gravitational effects were ignored. An estimation of the Reynolds number showed that all cases were in the turbulent regimen. Shear–Stress-Tensor-ke (SST-ke) [13] was used to model turbulence due to its well-known accuracy in fully turbulent internal flow applications. The coupled solver used was the semi-implicit method for pressure-linked equation (SIMPLE) scheme. Space discretization for velocity gradient was least squares cell based, second-order upwind for momentum and first-order upwind for turbulent kinetic energy and specific dissipation rate.

Table 3 Mesh size for both tower configurations in millions of elements

Geometry	Coarse	Medium	Fine
Tower “circular”	0.02	0.075	0.3
Tower “FV25”	0.3	1.3	2.3

3 Results

3.1 Porous media parameters

Pressure drop through fill and drift eliminator for different averaged inlet velocities were found based on the numerical results of the detailed models of the fill and drift eliminator. Figures 7 and 8 show the performance curve (pressure drop versus averaged inlet velocity) for each element including a fitting curve. Viscous resistance and inertial resistance parameters were estimated from the fitted curve in both elements and the model shown in Eq. 1 (see Table 4). It is clear that the drift eliminator has smaller (around 1 order of magnitude) parameters than filmic fill, which suggests that the pressure drop caused by the eliminator is almost negligible when compared with the drop in the filmic fill.

3.2 Flow visualization

Pressure and velocity contours in the axisymmetric model and in the symmetry planes of the 3D model are shown in Fig. 9.

Pressure contours evidenced the pressure drop through the fill in the midsection of both towers. For the FV25 tower, an important pressure jump is shown in the upper

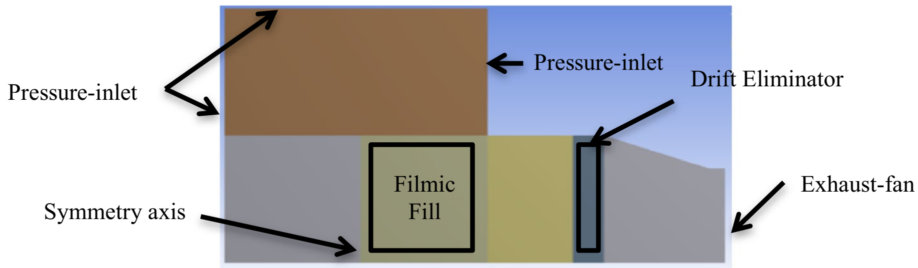


Fig. 6 Boundary conditions for the circular tower

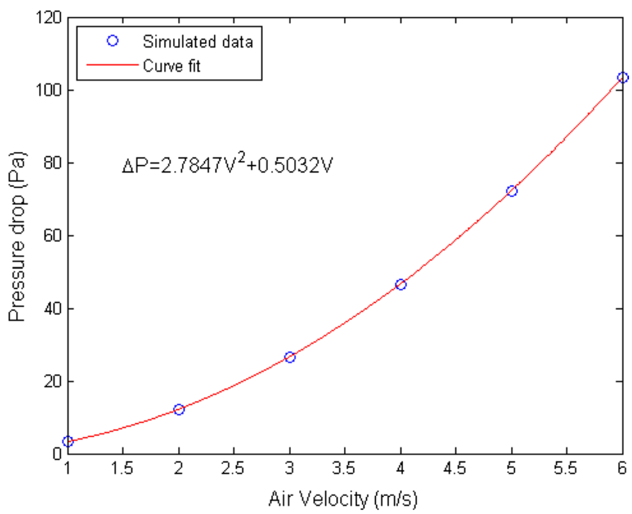


Fig. 7 Pressure drop in the filmic fill

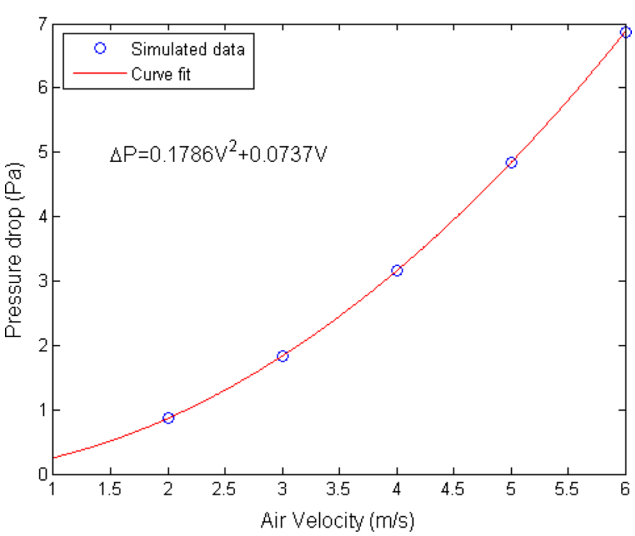


Fig. 8 Pressure drop in the drift eliminator

Table 4 Porous media parameters

Device	Viscous resistance ($1/m^2$)	Inertial resistance (1/m)
Filmic fill	98,469.7	19.8
Drift eliminator	14,422.1	1.27

section. Velocity contours showed the flow acceleration near the air inlet of the tower as well as a recirculation zone in the lower section for both towers.

Figures 10 and 11 show the details of the velocity field near the air inlet in both cases, where vortices of different size were observed.

FV25c tower presented one main recirculation below the air inlet, while the FV25 tower showed a more complex behavior, where not only one but at least three different recirculation regions can be observed (Fig. 11). Interactions between the perpendicular air flows entering the squared (FV25) tower increases the need of power of the fan.

3.3 Pressure drop and power consumption

The total pressure drop along both cooling tower models was estimated by the change of pressure along the symmetry axis for the FV25c tower and by the pressure along a line generated by the intersection of the symmetry planes for the FV25 tower. Figures 12 and 13 show the static pressure along the height of the tower.

Pressure drop for the FV25c tower shows that the biggest drop appears in the filmic fill section. On the other hand, for the FV25 tower, the biggest drop in pressure does not occur in the fill section but in the upper zone of the tower. This pressure drop in the upper zone of the FV25 tower is explained by a sudden reduction in the transverse area where it changes from square shaped to circular shaped to fit the fan circumference:

$$P_{\mathrm{h}} = \mathrm{Q} \times (\Delta P + \mathrm{VP}). \quad (2)$$

Power consumption in the cooling tower is estimated with the total pressure. Equation 2 shows how power consumption (P_{h}) was estimated: ΔP is the static pressure jump at the fan, VP is the velocity pressure at the fan inlet and Q is the air flow. Results of the estimation of the different parameter of Eq. 2 are shown in Table 5. Under the same atmospheric conditions, FV25c has a higher air flow and consumes only 61 % of the power consumed by FV25. This reduction in the power needed to move the air through the tower means considerable savings in an all-year operating cost.

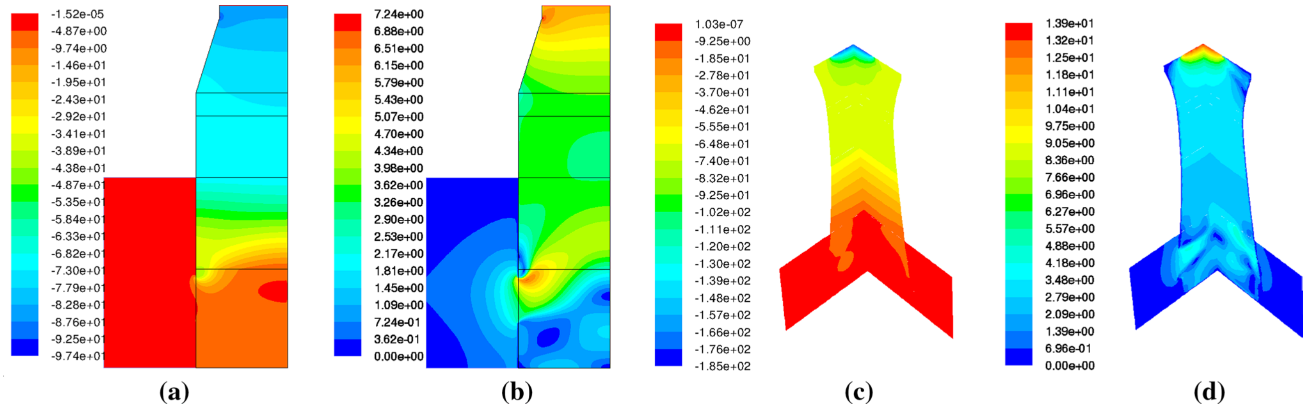


Fig. 9 Pressure (**a**) and velocity (**b**) contours in the circular tower. Pressure (**c**) and velocity (**d**) contours in the FV25 tower

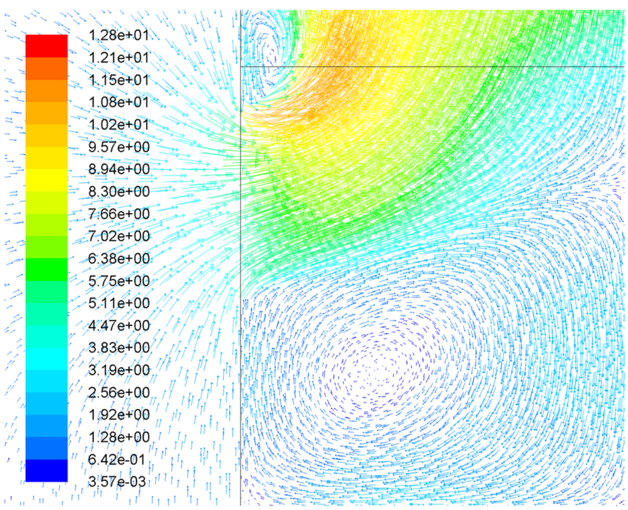


Fig. 10 Recirculation region at the inlet of the FV25c tower (vectors colored by velocity magnitude)

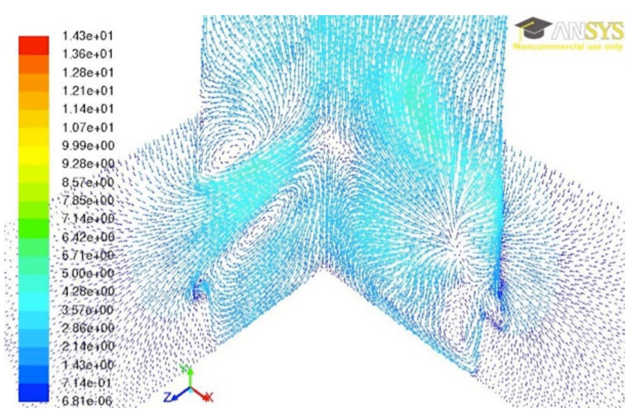


Fig. 11 Recirculation region at the inlet of the FV25 tower (vectors colored by velocity magnitude)

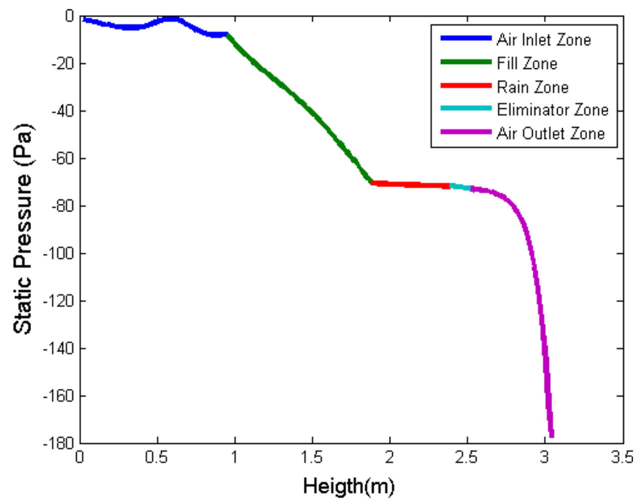


Fig. 12 Pressure drop per zone on FV25

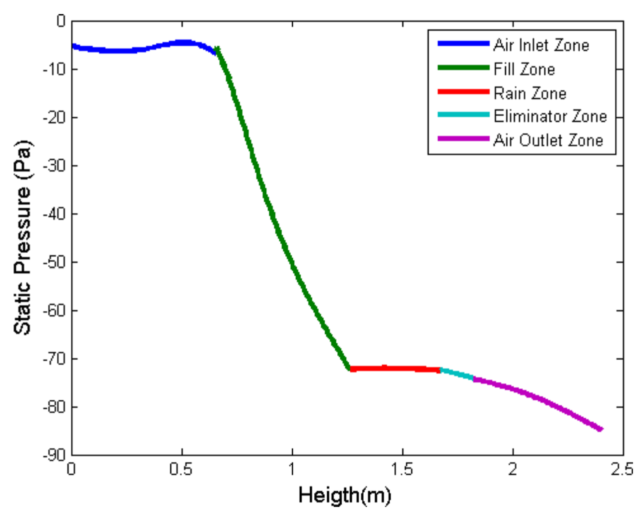


Fig. 13 Pressure drop per zone on FV25c

Table 5 Power consumption estimation

Tower	ΔP (Pa)	VP (Pa)	Q (m^3/s)	P_h (HP) (kW)
FV25c	85	18	3.87	0.53 (0.40)
FV25	180	120	2.15	0.86 (0.64)

Table 6 Pressure drop comparison for the FV25c tower

Zones	IDCF (Pa)	Simulated (Pa)	Difference (%)
Fill	54.3	56.6	4.23
Drift eliminator	7.72	1.7	77.9
Others zones	34.13	26.5	22.3
Total	96.15	84.4	12.2

Table 7 Pressure drop comparison for the FV25 tower

Zones	IDCF (Pa)	Simulated (Pa)	Difference (%)
Air inlet	–	6.9	–
Fill	64	62.1	3 %
Rain zone	–	2	–
Drift eliminator	6.5	1.1	83 %
Top section	80	104.5	30 %
VP fan	105	120	14 %
Total	137	176.6	29 %

3.4 Results verification

Verification of the numerical results was made comparing pressure drop obtained by CFD against pressures estimated by the manufacturer's software design. This software (called IDCF) is based on a one-dimensional Merkel model and empirical correlations for the fill and drift eliminator [14]. As shown in Tables 6 and 7, estimations made by the CFD model are in good agreement with the design software ICDF. The difference in the predicted pressure drop for the fill section is below 5 % in both cases; this value confirms that the use of a porous media and its characterization lead to an accurate prediction.

An important difference in the prediction of the pressure drop between both approaches was found in the drift eliminator. Nevertheless, drift eliminator is the section that produces the lowest pressure drop (around 5–6 % of the total drop) on the tower. Such differences could be explained by the difference in the geometries used in the CFD and IDCF and the pressure drop in IDCF is computed from the empirical equation of 2D S-shaped channels (see Fig. 2c), while CFD takes in account the exact 3D shape of the channels (see Fig. 2a, b).

3.5 Computational costs

All simulations were run on the server master-hpc-mox.uniandes.edu.co from Universidad de los Andes. This cluster has seven DELL PowerEdge M915 blade servers; each one has four 16-core processors AMD Opteron 6282 SE @ 2.6 GHz. Each blade has 128 GB of RAM and 500 GB of hard disk. The blades are interconnected through a high-speed switch with InfiniBand. Estimations of computational costs were made from the number of hours needed to run 1000 iterations per case (Table 8).

The porous media simulations were more expensive that the other cases because their geometry was modeled in detail. The FV25c tower simulations were more efficient in terms of computational cost because of the reduced number of elements on the grid (less than 300,000).

4 Conclusions

A CFD model of two different cooling towers was presented and discussed. Filmic fill and drift eliminator were successfully modeled and replaced by porous media in the full tower configuration. Simplifications on geometry as symmetries were used for both towers. The FV25c tower was modeled as axisymmetric and for the FV25 a quarter of the 3D body was analyzed.

Numerical results show that the airflow on the FV25 tower has more complex dynamics than the one on the FV25c. Comparison of both models lead to a difference of 0.33 HP; the mean circular tower consumes around 60 % of the power consumed by the square-shaped tower. This could be explained by complex flow interaction at the bottom of the FV25 tower and by its sudden change in traverse area at the top of the device.

Agreement in numerical predictions between CFD and IDCF proves that the approach proposed suits the design process requirements for induced draft cooling towers.

Table 8 Computational cost for different simulations for coarse (C), medium (M) and fine (F) meshes

Geometry	Porous medium			FV25c tower			FV25 tower		
Mesh	C	M	F	C	M	F	C	M	F
RAM memory (GB)	132	132	132	132	132	132	132	132	132
Number of processors	32	32	32	4	4	4	4	12	20
Estimated time (h)	2	6	14	1.5	2.5	4	1.25	20	22.5

Furthermore, CFD analysis offers more information than the zero-dimensional model used so far. The versatility of the technique proposed would allow the design team to vary operating conditions or to analyze only a new filmic fill to include in the whole model tower without relying on the manufacturer's technical report.

As shown, the air dynamics model was verified, and inclusion of heat exchange estimations implemented in the axisymmetric model to make it a completely functional design tool. This would be the first complete CFD model used as a design tool by the Colombian industry.

Acknowledgments The authors acknowledge the support of EDOSPINA for financing the presented work.

References

1. Klimanek A (2013) Numerical modelling of natural draft wet-cooling towers. Arch Comput Methods Eng 20:61–109
2. Tanimizu K, Hooman K (2013) Natural draft dry cooling tower modelling. Heat Mass Transfer 49:155–161
3. Fisenko SP, Brin AA, Petruchik AI (2004) Evaporative cooling of water in a mechanical draft cooling tower. Int J Heat Mass Transf 47:165–177
4. Heidarinejad G, Karami M, Delfani S (2008) Numerical simulation of counter-flow wet-cooling towers. Int J Refrig 32:996–1002
5. Franco A, Valera DL, Peña A, Pérez AM (2011) Aerodynamic analysis and CFD simulation of several cellulose evaporative cooling pads used in mediterranean greenhouses. Comput Electron Agric 76:218–230
6. Milosavljevic N, Heikkilä P (2001) A comprehensive approach to cooling tower design. Appl Therm Eng 21:899–915
7. Gharagheizi F, Hayati R, Fatemi S (2006) Experimental study on the performance of mechanical cooling tower with two types of film packing. Energy Convers Manag 48:277–280
8. Al-Waked R, Behnia M (2006) CFD simulation of wet cooling towers. Appl Therm Eng 26:382–395
9. Kloppers JC (2003) A critical evaluation and refinement of the performance prediction of wet-cooling towers, University of Stellenbosch
10. Gan G, Riffat SB, Shao L, Doherty P (2001) Application of CFD to closed-wet cooling towers. Appl Therm Eng 21:79–92
11. Goshayshi HR, Missenden JF (2000) The investigation of cooling tower packing in various arrangements. Appl Therm Eng 20:69–80
12. Whitaker S (1996) The Forchheimer equation: a theoretical development. Transp Porous Media 25:27–61
13. Launder BE, Spalding DB (1974) The numerical computation of turbulent flow. Comput Methods Appl Mech Eng 3:269–289
14. Kloppers J, Kröger D (2005) Cooling tower performance evaluation: merkel, poppe, and e-NTU methods of analysis. J Eng Gas Turbines Power 127:1–7