18th IAHR International Conference on Cooling Tower and Air Cooled Heat Exchangers,
16-20 October 2017, Lyon, France

NUMERICAL OPTIMIZATION OF A NEW DESIGN OF COOLING TOWER

Miguel F.CABALLERO[1], Mónica HERNÁNDEZ[1], Iván AROCAS[1], Antonio VIEDMA[1], Francisco SÁNCHEZ[1], Javier RUIZ[2], Manuel LUCAS[2] and Antonio S. KAISER[1]

[1]Universidad Politécnica de Cartagena, Cartagena, Spain
[2]Universidad Miguel Hernández, Elche, Spain

Contact person: antonio.kaiser@upct.es

Keywords: cooling tower, droplet emission, Legionella, porous media, CFD, CT (Tower Characteristic), ε (Thermal efficiency), capture efficiency (%), DPM (Discrete Phase Model), Number of Transference Units (NTU).

ABSTRACT:

Cooling towers are one of the most efficient components dissipating heat from power plants, water-cooled refrigeration and air conditioning systems in buildings and industrial processes. However, during the last years, various outbreaks of Legionella caused by the drift from cooling towers in urban areas occurred in southern Spain and other regions. Due to this fact, local public health authorities have introduced policies to replace the cooling towers with other less efficient systems,contributing to increase energy consumption and thusclimate change effects.

In this work, the numerical optimisationand analysis of a new design of cooling tower withinverted flow are presented. The aim is to limit the drift and to reach energy efficiency similar to commercial mechanical draft cooling towers. For the optimised design, a thermo dynamical study of the tower performance under different conditions of air and water flow is carried out, assessing its drift, tower characteristics (CT) and efficiency. Particular emphasis has been placed on the numerical model of the filling and its interaction with water droplets.

Numerical results show that the new design presents CT values slightly higher than commercial cooling towers for different conditions of air and water flow. No significant droplet emission has been detected in the numerical simulations.

1. INTRODUCTION

Cooling towers are widely used to remove heat from industrial processes and from refrigeration systems. They present several advantages over traditional systems:

- Lower space and energy consumption requirements.
- Higher energy efficiency.
- Decrease in the sound emission levels.

However, there are some issues to be resolved; the most important consist in the droplet emission of this kind of refrigeration systems, aphenomenon called drift.

This emission must be reduced for several reasons (Lewis B.G., 1974), the most important is that water droplets may contain microorganisms, which once released to the atmosphere can lead to a high probability of the Legionella's bacteria proliferation.

The most common tools used to eliminate this effect are thedrift eliminators, which avoid the droplets emission to the atmosphere through anabrupt change of sections. However recent studies, (J.López, et al., 2015), conclude that drift eliminators can have a reverse effect acting as droplet generators, although velocity drift limits may be found to preventit.

Another problem is the highly restrictive legislation surrounding cooling towers applied in Spain, due to the high number of Legionella's outbreaks. The risinguse of less efficient dry coolers motivates the necessity of more research into an alternative design of cooling tower to take advantage ofits higher efficiency (Pugh M.D., 2005).

In this work, a new concept of thecounter-flow cooling tower is analysed with the same operationprinciple, consisting in hot water sprayed into an air, decreasing the water enthalpy while that of air increases.Detailed derivation of the governing equations of evaporative cooling in wet-cooling towers is given in Kloppers & Kröger (2005).

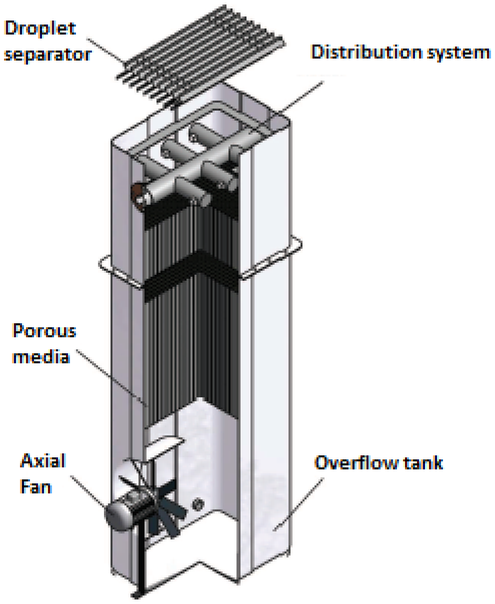
Mohiuddin and Kant (1996) describe part of the detailed methodology for the thermal design of wet counter flow cooling towers. The thermo-dynamical performance of the tower is characterisedby the tower characteristic ratio (CT) or the number of transfer units (NTU).

Some authors, as N.W. Kelly & L.K. Swenson (1956),correlated the tower characteristic with the water/air mass flow ratio and concluded that the factors affecting this value were the water-to-air ratio, the packed height, the deck geometry and the hot water temperature (slightly relevant).

To increase the cooling rate, the interface air-water is increased by providing packed beds. There are three types of packing in use, namely, film (S.V. Bedekar, P. Nithiarasu, & K.N. Seetharamu, 1998), splash and film-grid packing.

To obtain reliable results about this effect through CFD analysis, a real geometry computationally viable of the porous media is essential. The main problem about it is the inexistence of actual interaction between porous media and droplets in popular CFD softwarethat only simulates pressure and velocity drops through inertial and viscous resistances.

2. NEW TOWER GEOMETRY



The new design tries to maintain the main figures of the conventional designed cooling tower of the University Miguel Hernández, located in Elche, Spain, see Figure 1, what has been deeply studied bythe group both numerically and experimentally, in order to achieve a meaningful comparison.

This tower has a nominal power of 30kW and an air flow of 5200 L/h, in this case, the air flows upwards meanwhile the water is pulverised from the upside producing a counter flow (traditional counter-flow cooling tower). This prototype is analysed in Ramírez(2014).

Figure 1: Conventional cooling tower in U. Miguel Hernández, Elche, Spain

The novelty of the new design and the reason why it is called inverted-flow cooling tower is the downward movement of the air flow, entering at the top of the tower due to the depression caused by a mechanical fan. At mid-height, the hot water is pulverised upwards producing a counter flow. The air leaves the tower at its lower heights, after a change in direction and a decrease in velocity to avoid droplet drift, see Figure 2.

Porous media and drift separators are incorporated to increase the air-to-water interfacial contact area and to reduce the droplet emission.

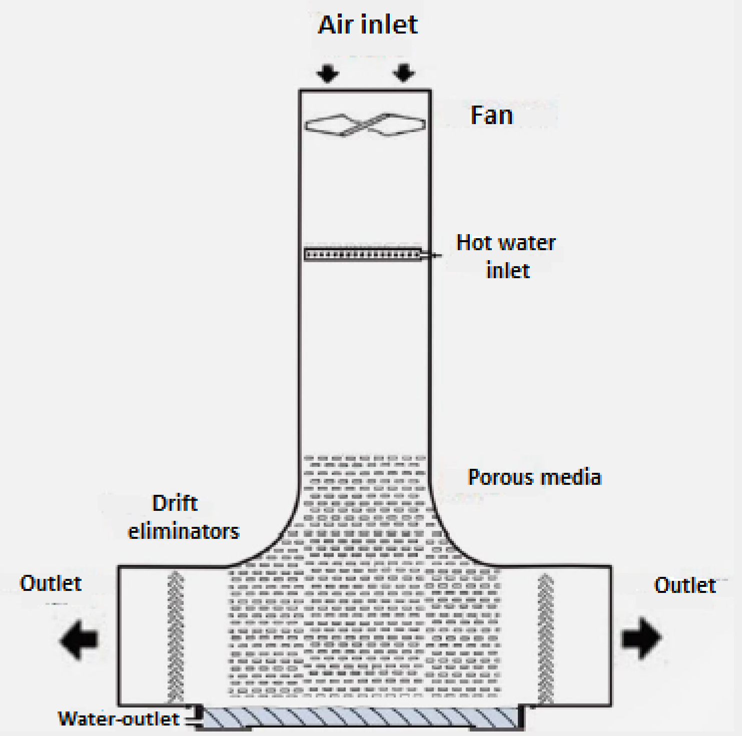


Figure 2: New design geometry

The main objective of this design is to reduce droplet emission by a decrease in the outlet velocity. This is achieved by the maximisation of the outlet-inlet area ratio. The reduction in the velocity also leads to an increase in the droplets residence time and the energy efficiency.
Numerical simulations are carried out on different designs to simplify and make viablefrom a constructive point of view the new concept.Meanwhile thermo-dynamical and fluid-dynamics characteristic are maintained.

3. NUMERICAL MODEL

Analytic and experimental criteria are being considered based on the bibliography and previous works. In an inverted counter flow cooling tower, the process consists of a gas phase flowing downwards, a liquid phase (water film) flowing upwards, and a great interface between these two phases.

All variables obey the conservation principle defined with a differential equation. Any conservation equation of Fluid Mechanics can be expressed as:

$$\boldsymbol{\frac{\partial}{\partial t}(\rho\phi) + \nabla\cdot(\rho\vec{u}\phi) = \nabla\cdot(\Gamma grad\phi) + S_{\phi}}$$

The first term refers to the transitory, second to convective, third to diffusion coefficient ($\boldsymbol{\Gamma}$) and fourth the source term, being $\boldsymbol{\phi}$ the scalar defined in each equation, $\boldsymbol{\rho}$ density, and $\boldsymbol{\vec{u}}$ velocity. This equation is applied to each equation (mass, movement and energy), being combined with the finite volume method discretization.

Second order UPWIND scheme is used for themoment, energy, and dissipation equations for both phases. Once discretized the equations are solved using the SIMPLE algorithm, through an iterative method.

The general boundary conditions employed in this work can be divided intofour types of surface, as shown in Table 1, including parameters of the Discrete Phase Model (DPM), and resistances for the porous media obtained following the method described in (Ramírez, 2014). All surfaces described apart from "Wall" are considered as evacuation (Escape), however for the walls is considered the Wall-Film model, depending on the energy of impact and some parameters the droplet could be in 4 different situations, as it is explained in (ANSYS Inc., 2011).

Inlet				**Outlet**		**Wall**	**Floor**	**Porous media**	
V	*T*^{a}	*Species*	*DPM*	Pressure	DPM	DPM	DPM	Viscous resistance	Inertial resistance
4 m/s	*303 K*	*0.0133115*	*Escape*	0	Escape	Wall-Film	Escape	1500000 m^{-2}	32 m^{-1}

Table 1: Boundary conditions

Meshing can be mainly done using tetrahedral and hexahedral elements; hexahedral elements offer less computational cost due to their simplicity. Meanwhile tetrahedral is the easiest solution for complex geometries although they considerably increase the computational cost. The combinations of both element types usually give the best results in complex meshes.

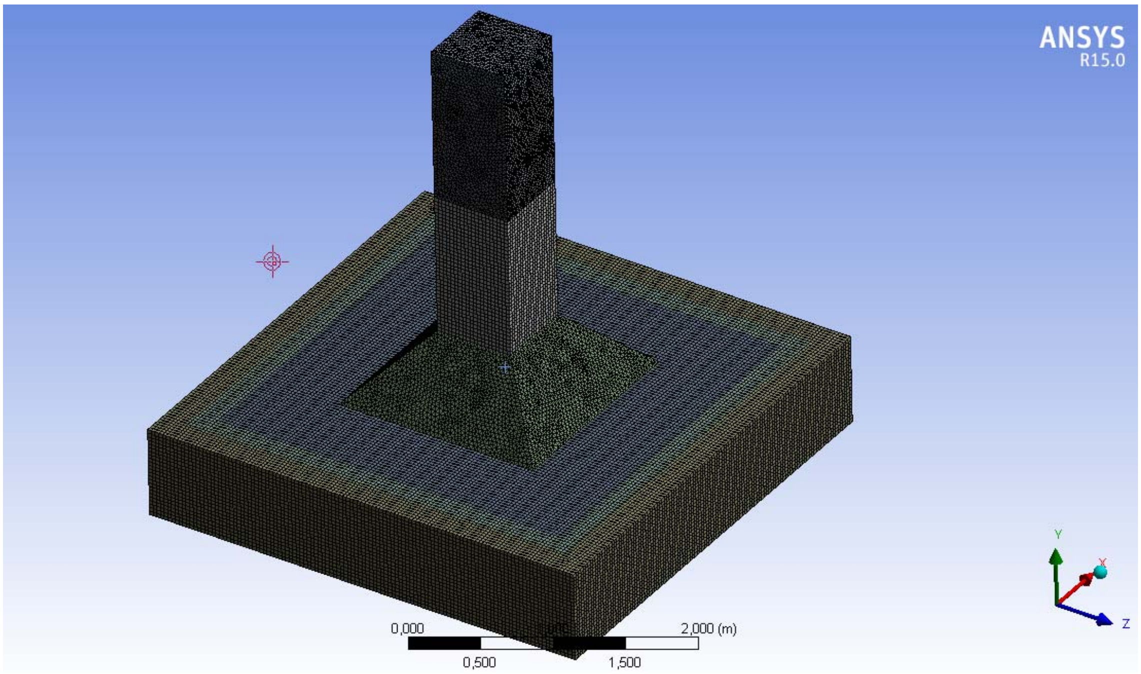


Figure 3: Cooling tower mesh

The meshing was built for to 2 different geometries: cooling tower's fluid volume and porous media. The objective was to maximise the number of hexahedral elements to decrease the computational cost.

For the first geometry, Figure 3, a mesh of 967114 elements was obtained, mixing hexahedral and tetrahedral elements. Most of thetetrahedral elements were gathered in the section change, improving convergence in the simulations.

The porous media geometry, Figure 4, consists of a 12x12x12cm cube, formed of crossed tubes simulating the effect of a real porous media. The references used for the mesh are based in J.López, et al. (2015), considering the wet area of the commercial catalogue as a key parameter.

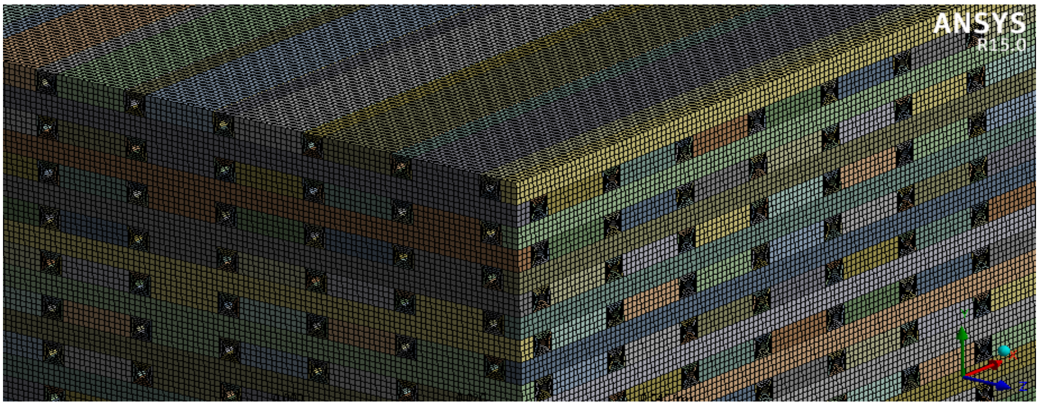


Figure 4: Mesh of the reference volume

4. GEOMETRY NUMERICAL OPTIMIZATION

The starting point of the new geometry optimization is the design presented in Arocas(2015):a circular based geometry, obtained with an iterative process. Here the focus was seton the fluid dynamical aspect, trying to obtainthe best geometry to homogenise the flow, but with the difficulty of its constructive complexity as it was based in circular figures.

Figure 5 presents the geometry evolution introducing different constructive simplifications, maintaining the premise of straight lines and square elements. The analysis is donebasedoneight parameters: outlet velocity, %mass water escaped, %droplets escaped and characteristic diameters of the droplets.

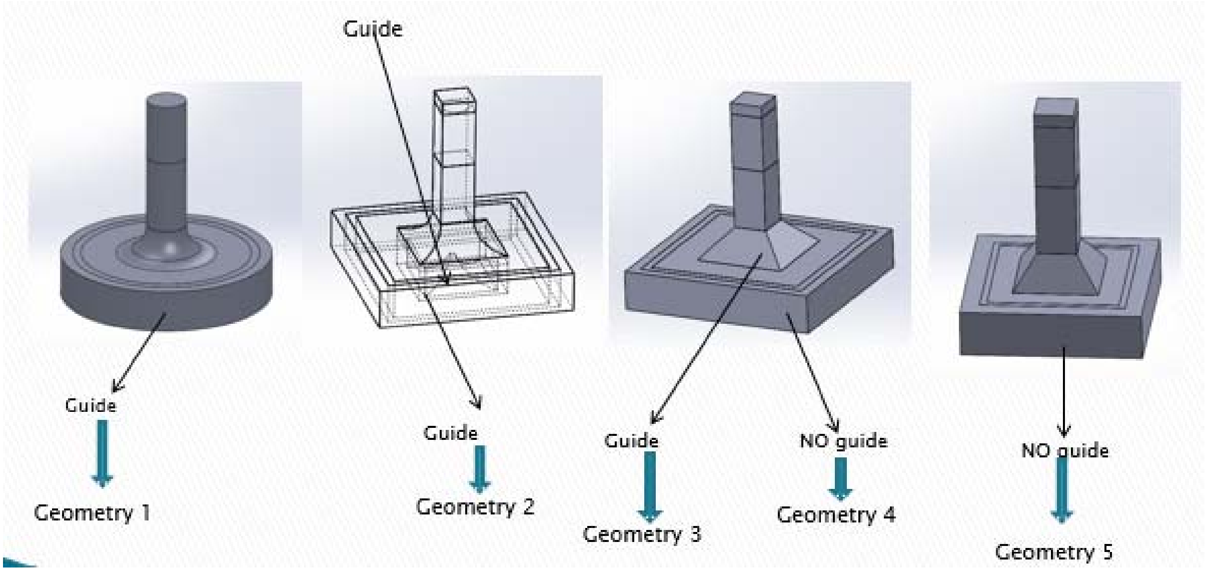


Figure 5: Geometry evolution.

The simplifications implemented are basically to replace curved elements by straight ones and to remove the flow guide placed downwards. Furthermore, finally a reduced sized geometry is analysed as Geometry 5.

Geometry	Deviation (m/s)	Minimum velocity (m/s)	Maximum velocity (m/s)	Meanvelocity (m/s)	%mass	%droplet	D_{32}(m)	D_{10}(m)
1	0.0074	0.166	0.1975	0.18	0.077	4.54	3.96E-05	2.82E-05
2	0.0821	0.0082	0.812	0.1562	0.076	7.3	9.29E-04	9.23E-04
3	0.0573	0.0085	0.235	0.1549	0.0004	2.34	5.76E-05	4.90E-05
4	0.0545	0.0075	0.2226	0.155	0.0004	2.03	5.63E-05	4.78E-05
5	0.0955	0.0118	0.3563	0.1961	0.114	3.46	6.27E-05	7.32E-05

Table 2: Comparison of geometries

The results are shown in Table 2.As expected, the most homogenous outlet contour is in the Geometry 1, as recirculations are avoided. At the square section cases, recirculations are produced at the corners with velocities of almost 0 m/s, reducing the actual outlet section.

The true objective of slowing the outlet velocity is to minimisethe droplet emission and its size, leading to no significant effects in the recirculation produced in the corners of square section geometries.

No significant result is obtained about the droplet’s size as no interaction with the surface was considered, reality should show smaller sizes.

The most homogeneous contour of outlet velocities and the smallest percentage of mass and droplets escaped are obtained in the Geometry 4, therefore it was chosen as the ideal based on these parameters and its constructive simplicity over Geometry.

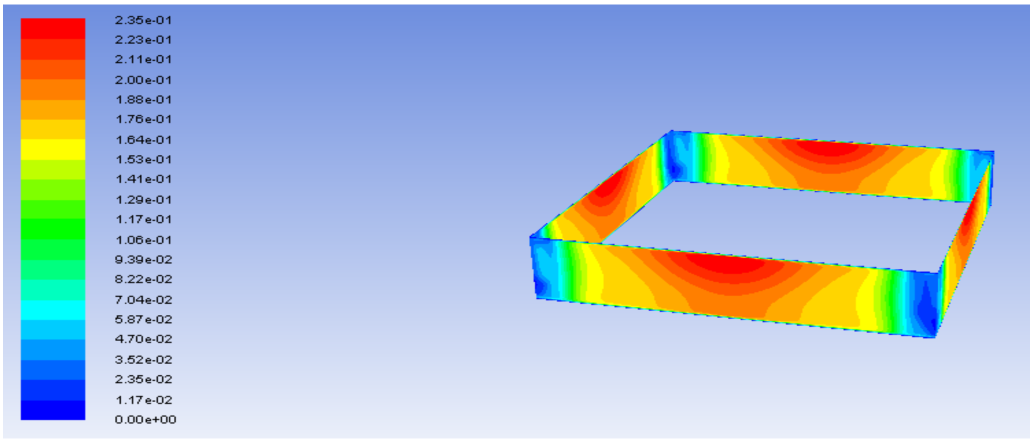


Figure 6: Outlet contour

5. NUMERICAL CHARACTERIZATION OF THE FINAL GEOMETRY

5.1. EVALUATION OF CT

The scheme ofFigure 7Figure 6is included to show the thermo-dynamical analysis of counter flow cooling towers. The complete development of the conservation equations governing the processes of heat and mass transfer between air and water currents is described in Pinazo (1995). The following hypotheses are considered:

- Heat and mass transference only goes in the proper direction of the flow
- Heat and mass transference with the exterior walls is disregarded.
- Specific heat of air and water considered constant.
- Lewis number considered constant all along the cooling tower.
- Dragging water losses considered null.
- Transversal section considered constant.

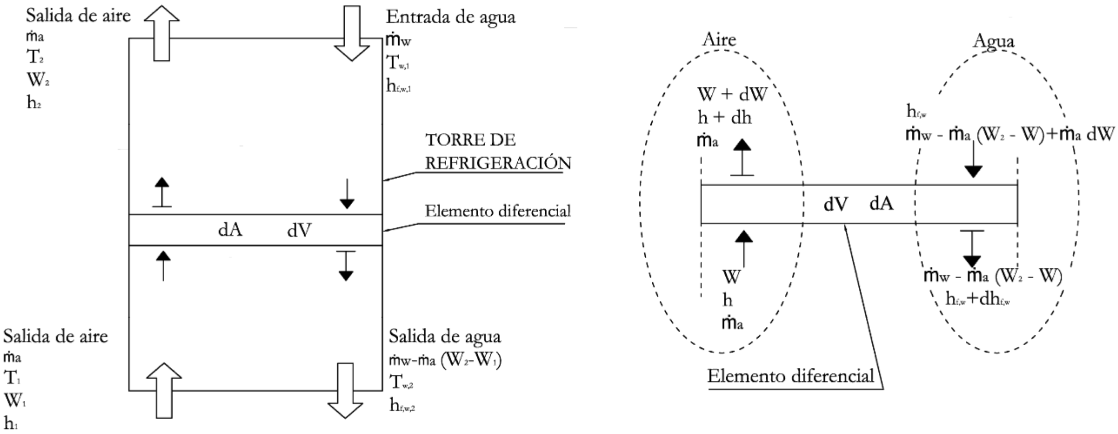


Figure 7: Counter-flow cooling tower scheme

The number of transference units is defined as:

$$NTU = \frac{1}{\dot{m}_a} \int_{ent}^{sal} h_D \cdot A_V \cdot dV = \int_{ent}^{sal} \frac{dW}{\left(W_{s,w} - W\right)}$$

Also, another of the most used parameters for characterising cooling towers is the CT, thermo-dynamical characteristic, defined as:

$$CT = \frac{1}{\dot{m}_w} \int_{ent}^{sal} h_D \cdot A_V \cdot dV = \int_{ent}^{sal} \frac{-dh_{f,w}}{\left(h_{s,w} - h\right)} = c_{p,w} \frac{T_{W,1} - T_{W,2}}{2} \sum_{j=1}^{4} \frac{1}{\left(h_{s,w} - h\right)_j}$$

The analytic resolution of this equations results impracticable, as there are no integrable equations for the enthalpy of the air. In this case, Tchebyshev Method is applied following the standards ATC-CTI 105 [2000] and BS4485 [1988]. Mohiuddin & Kant(1996), following the method, describe the details of the integration procedure.

To obtain a complete range of results in the simulations were used the following water and air flow: 5200,4000 and 3000 L/h (water), and 50,37.5 and 25 Hz for the fan's frequency controller. The parameters of the fan are based on thecommercial catalogue.

Combiningthese values into a parameter defined as the ratio of water-air flow, the following results were obtained:

$\dot{m}_w$(L/hr)	5200	5200	5200	4000	4000	4000	3000	3000	3000
Freq(Hz)	50	37,5	25	50	37,5	25	50	37,5	25
$\dot{m}_w / \dot{m}_a$	1,297	0,935	0,770	0,973	0,701	0,578	0,750	0,541	0,445
NTU	1,261	0,981	0,853	1,024	0,798	0,689	0,854	0,669	0,574

Table 3: NTU results

The CT for different operation conditions is defined as:

$$CT = \frac{h_D \cdot A_V \cdot dV}{\dot{m}_w} = c \left(\frac{\dot{m}_w}{\dot{m}_a}\right)^{-n}$$

The constants c and n, according to ASHRAE, regarding water-air flow ratio are defined as:

c	**n**
1,04	**0.365**

Table 4: Constants of CT

The results show a decrement in the CT as $\frac{\dot{m}_w}{\dot{m}_a}$ is increased. The following graph compare the CT with the University of Elche’s cooling tower, showing betters values in the prototype studied in this case.

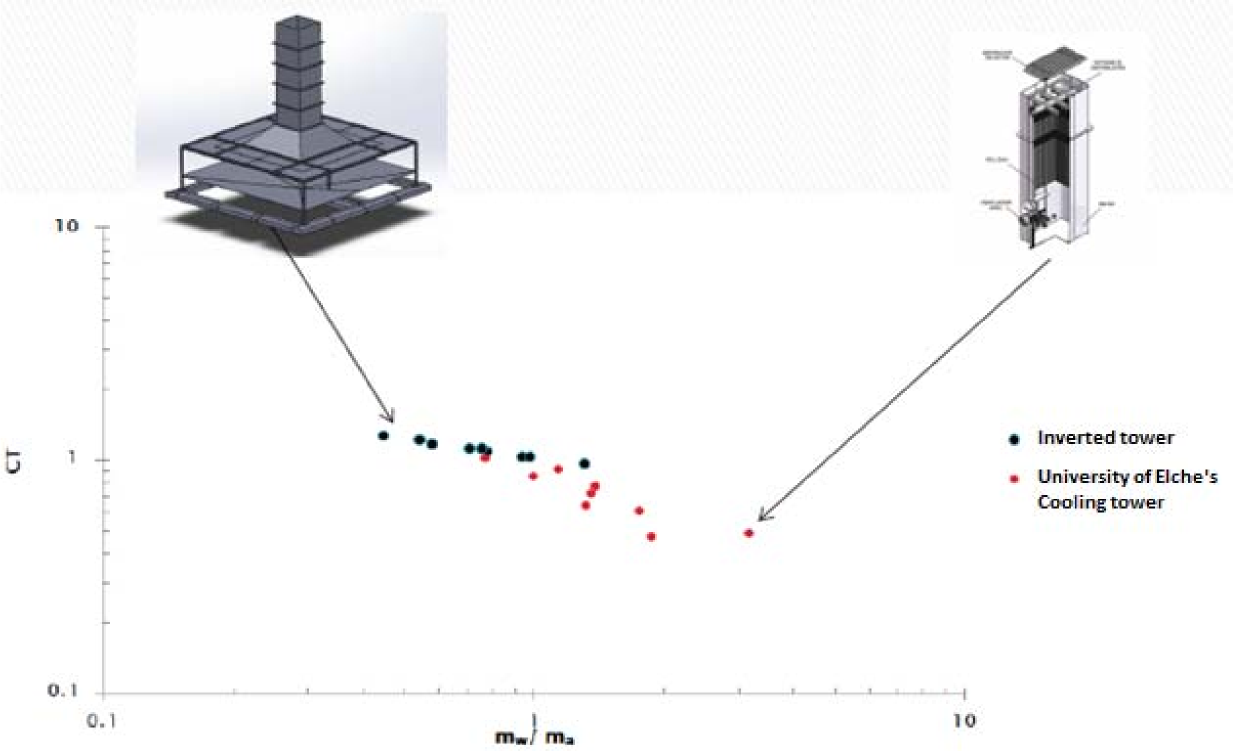


Figure 8: CT comparison between the new design and Elche’s experimental model

5.2. EVALUATION OF DRIFT

The analysis of porous media with commonly used software implies no interaction between porous media and droplets. Simulation is based on Darcy’s equations, considering only pressure and velocity drops.

In this case, to pursue more reliable results and to get a real vision of the interaction between droplets and porous media, a modular analysis of the porous media is done.

The biggest issue encountered is the complexity and size of the porous media considered in the cooling tower, leading to a huge number of elements (meshing) and an enormous computational cost.

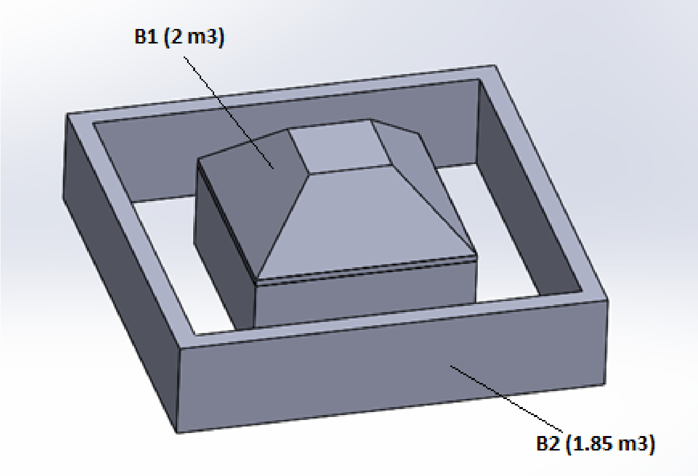


Figure 9: Porous media volumes

The solution used in this work lies in using a 12x12x12cm cube as reference geometry, Figure 10, limiting the number of elements to 7.3million. The geometry chosen is adapted tothe conditions of the porous media, introducing different boundary conditionsto simulate the area increase and droplet distribution.

The main idea of this analysis is to combine the reference geometry and a characteristic streamline of the flow obtained in simulations, using its results for velocity and gravity parameters. The streamline represents the reference trajectory of droplets through de porous media.

The line considered has a parallel trajectory to XY plane, the parameters taken into accountfor the analysis are: distance, time residence, and average velocity.

The streamline is divided into 13 stretches, as shown in Figure 2, each one presents certain conditions to simulate the real situation in the porous media.

The procedure includes the following steps:

1. Defining the cube's surfaces: Wall1 and wall2 are considered as outlet surfaces to simulate the area increase (Figure 1), introducing values of normal and tangential velocity. In the other twosurfaces, the flow is supposed to be parallel to the direction of the streamline.

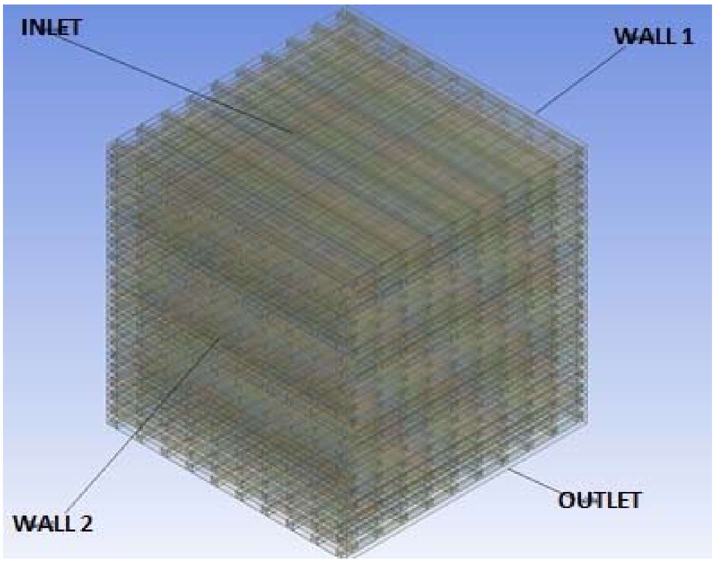


Figure 10: Reference volume domain

2. The inlet droplet distribution is obtained using an approximated Rosin-Rammler function by the minimum square method. Values of droplets are exported from the results of the numerical simulations at the inlet surface of the porous media.

V_Y(m/s)	m(kg/s)	D_{MEAN}(m)	T(K)	Nº of diameters	Spread Param.
-3.65	0.057	5e-4	306.3	100	3.94

Table 5: Inlet droplet parameters

3. Outlet conditions are obtained for each stretch.

Stretch	V_{OUTLET}(m/s)	V_Z $_{LATERAL}$(m/s)	V_Y $_{LATERAL}$(m/s)	g_x(m/s^2)	g_y(m/s^2)
1	1.277	0.15	1.27	-1.41	-9.71
2	0.936	0.17	0.93	-2.57	-9.46
3	0.69	0.123	0.68	-2.87	-9.38
4	0.5	0.094	0.5	-3.1	-9.31
5	0.355	0.074	0.35	-3.65	-9.11
6	0.24	0.056	0.23	-4.9	-8.5
7	0.192	0.026	0.19	-6.73	-7.14
8	0.176	0.008	0.17	-8.41	-5.1
9	0.173	0.0014	0.16	-9.4	-2.84
10	0.173	0	0.17	-9.76	-0.985
11	0.09	0	0.09	-9.81	-0.11

Table 6: Outlet conditions for each stretch

4. For the first stretch, the values or humidity and temperature considered are 85% and 30ºC respectively. The outlet droplet distribution of each stretch is introduced as the inlet droplet distribution for the next stretch, combining these values with the ones shown inTable 6. Once the saturation point is reached the droplets will be considered as inert.

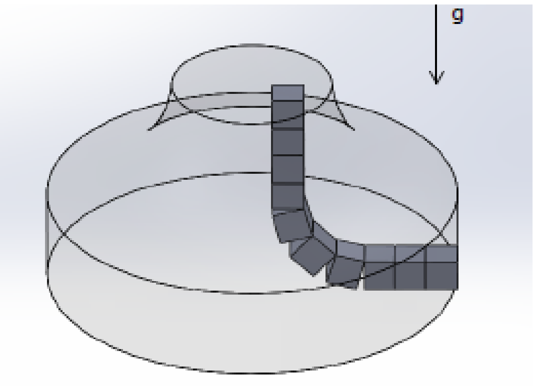


Figure 2: Evolution of the geometry of reference through the volume of control

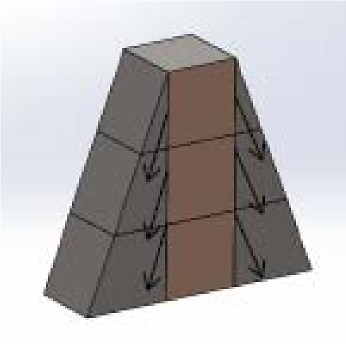


Figure 1: Augment of area in each stretch

Figure 3 shows the comparison between the results obtained considering theinteraction between droplets and drift, and the commercial software's simulation of the porous media. The capture efficiency values are appreciably higher considering interaction, obtaining values of 100% for droplets with a diameter up to 28 µm.

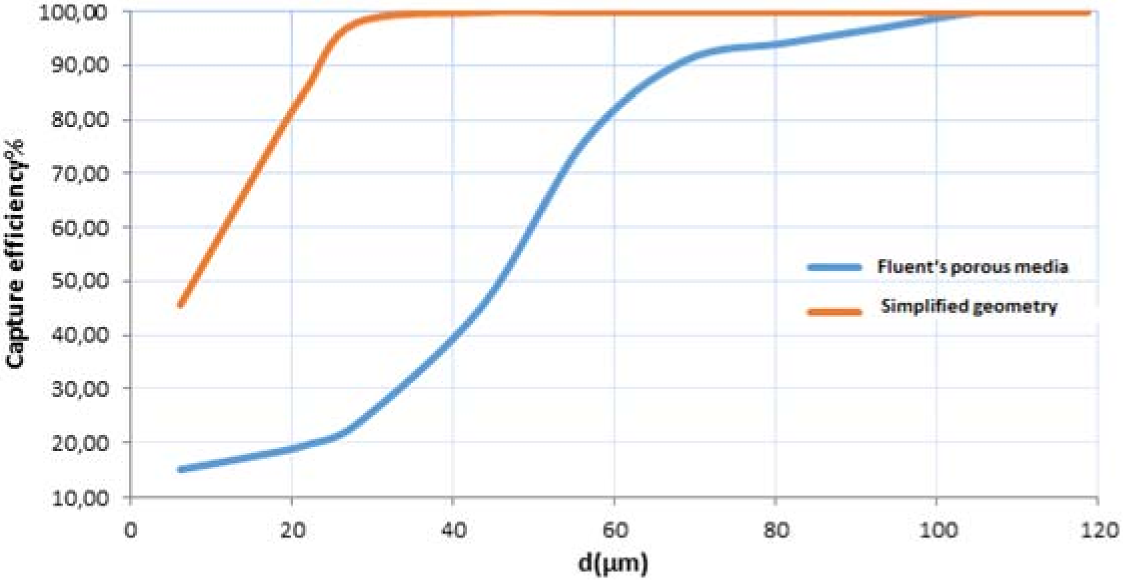


Figure 3: Comparison of capture efficiency between the model and Fluent approximation

The results inFigure 4 show the percentage of mass flow represented for each diameter, as the flow advances in the geometry the droplets with lower diameter represent a higher percentage of the total mass flow.

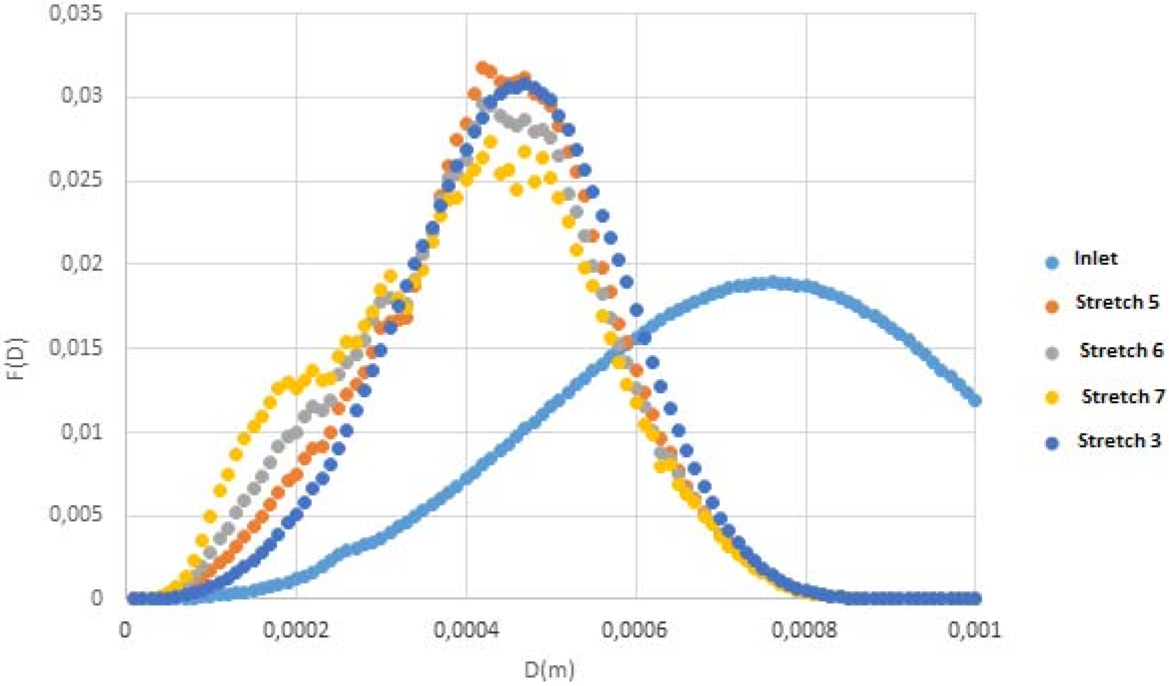


Figure 4:Percentage of mass flow for each range of diameters

6. CONCLUSIONS

The most relevant conclusions obtained are the following:

- Once compared the different design alternatives, square Geometry 4, with straight lines and squared forms and no flow guidesis chosen as the optimal, offering the most homogeneous contour of velocity in comparison with the circular geometry analysed in Arocas (2015).
- The thermo-dynamical analysis of the cooling tower leads us to conclude that an increase in the air flow produces an increase in CT. It is remarked as a huge advantage over the commonly used cooling towers, the fact that margin of growth in the air flow using this prototype of cooling towers in much higher in comparison with traditional ones.

  Commonly used cooling towers are limited to the increase in the air flow due to its constant section in the geometry, what leads to great impact on the droplet emission.
- The modular analysis is doneon the porous media integrated into the cooling tower using a simplified real geometry.

  Graphs of the capture efficiency depending on droplet diameter show how escaped particles have sizes around 3-25 μm, representing around de 0.00016% of the initial mass flow. Along withthese results, the poor reliability of the results obtained with porous media simulations of common software is confirmed.

  The effect of gravity and velocity over the capture efficiency is remarked, obtaining better values when the gravity is perpendicular to the flow. Once reached a limit in the droplet size particles show trajectories avoiding drift's tubes. However, these particles are destined to be lost in long trajectories due to the gravity effect.

REFERENCES

ANSYS Inc. (2011). *ANSYS FLUENT THEORY GUIDE.*

Arocas, I. (2015). *Optimización de una torre de refrigeración de tiro mecánico mediante simulación numérica para evitar la emisión de gotas de agua al ambiente.* Cartagena: Trabajo Fin de Grado, Universidad Politécnica de Cartagena.

F.Caballero Miguel. (2016). *Análisis y mejora del diseño de una torre de refrigeración de tiro mecánico planteada para la reducción de emisión de gotas al ambiente mediante técnicas de mecánica de fluidos computacional (CFD).* Cartagena: Trabajo Fin de Grado, Universidad Politécnica de Cartagena.

J.López, A.S.Kaiser, B.Zamora, M.Lucas, J.Ruiz, & A.Viedma. (2015). Numerical study and experimental validation of the water films and the detachment of drops on drift eliminators. *Computational Thermal Sciences: An International Journal*, 167-180.

Kloppers, J., & Kröger , D. (2005). A critical investigation in to the heat and mass transfer analysis of counterflow wet-cooling towers. *International Journal of Heat and Mass Transfer 48*, 765-777.

Lewis B.G. (1974). On the question of airborne transmission of pathogenic organisms in cooling tower drift. *Cooling Tower Institute*, Technical paper-T-124A.

Mohiuddin , A., & Kant, K. (1996). Knowledge base for the systematic design of wet cooling towers. *International Journal of Refrigeration, Volume 19, Issue 1*, 43-51.

N.W. Kelly, & L.K. Swenson . (1956). Comparative performance of cooling tower packing arrangements. *Chemical Engineering Procces, 25-7:263.*

Pinazo, J. (1995). *Manual de climatización. Tomo1 transformaciones psicrométricas.* Servidio de publicación Universidad Politécnica de Valencia.

Pugh M.D. (2005). Benefits of water-cooled systems vs air-cooled systems for air-conditioning applications. Orlando, USA: ASHRAE Expo.

Ramírez, J. R. (2014). *Caracterización experimental y modelización del binomio sistema de distribución-separadar de gotas en torres de refrigeración.* Cartagena: Tesis Doctoral, Departamento de Ingeniería Térmica y Fluidos, Universidad Politécnica de Cartagena.

S.V. Bedekar, P. Nithiarasu, & K.N. Seetharamu. (1998). Experimental investigation of the performance of a counter-flow, packed-bed mechanical cooling tower. *Energy Volume 23, Issue 11*, 943-947.