UMA COMPARAÇÃO QUALITATIVA NUMÉRICA ENTRE CONDENSADORES EVAPORATIVOS DE GRANDE E PEQUENA ESCALA

IVONI CARLOS ACUNHA JR.[1], MARCELO MORAES GALARÇA[1], ANDERSON FAVERO PORTE[1], ELIZALDO DOMINGUES DOS SANTOS[2], LIÉRCIO ANDRÉ ISOLDI[2], PAULO SMITH SCHNEIDER[3]

RESUMO

Este artigo tem por objetivo avaliar qualitativamente os resultados numéricos obtidos utilizando-se CFD para modelagem de condensadores evaporativos. Compara-se resultados da literatura com aqueles obtidos a partir de um condensador evaporativo de pequena escala, para testes em laboratório. Os campos de pressão e velocidade foram avaliados. Os resultados mostraram-se coerentes com aqueles apresentados na literatura. Dois ângulos de entrada de ar foram testados: 0° and 45° em direção ao reservatório de água. Foi observado uma pequena influência do ângulo de entrada de ar sobre o fluxo mássico de ar na saída. Estudos futuros podem ser realizados no sentido de melhorar a distribuição de ar dentro do equipamento a fim de aumentar a sua eficiência térmica.

PALAVRAS-CHAVES: Condensador evaporativo. Modelagem numérica. Análise qualitativa.

NUMERICAL QUALITATIVE COMPARISON BETWEEN A LARGE AND SMALL SCALE EVAPORATIVE CONDENSER

ABSTRACT

This paper aims to evaluate qualitatively the numerical results obtained by using a commercial CFD code for modeling of an evaporative condensers. It is the comparison between results presented in the literature, referring to a major equipment, and those obtained from an evaporative condenser on a smaller scale, for laboratory tests. Pressure and velocity fields are evaluated. The results show a good agreement with those reported in the literature. Two inlet air angles are performed: 0° and 45° toward the water sump. It is observed a little influence on the air inlet angle over the mass flow of air at the exit. Future studies can be done with the aim of improving the air distribution inside the equipment in order to improve the thermal efficiency.

KEYWORDS: Evaporative Condenser. Numerical Modeling. Qualitative Analysis.

[1]Instituto Federal de Educação, Ciência e Tecnologia do Rio Grande do Sul, ivoni.acunha@riogrande.ifrs.edu.br., marcelo.galarca@riogrande.ifrs.edu.br. anderson.porte@riogrande.ifrs.edu.br.
[2] Escola de Engenharia – FURG, elizaldosantos@furg.br., liercioisoldi@furg.br.
[3] Departamento de Engenharia Mecânica – UFRGS, pss@mecanica.ufrgs.br..

1. INTRODUCTION

During the last decades, the demand for cooling has increased significantly because, mainly, to the global economic growth and new parameters perceived by the lifestyle of the population. This factor directly produces a greater dependence on water and energy, once the most amounts of energy in the world today is from sources that use non-renewable resources like fossil fuels. Increasing the efficiency of the equipment present in refrigeration cycles contributes to a reduction in energy consumption. Evaporative condensers have as its main features the high efficiency of operation and the most economical utilization of materials and physical space, which ultimately generates a lower investment cost and operation. They have been widely used to increase heat transfer and improve the performance of large size and high thermal capacity refrigeration systems.

The correct modeling of this equipment is a challenge to researchers and is a very important tool in the industry, allowing an improvement of thermal performance with a cost lower than that involved in experimental processes of trial and error. In this sense, one major hurdle is to find good models which represent as close as possible the heat and mass transfer phenomena in evaporative condensers.

Mathematical modeling of evaporative condensers has been studied by many authors over time. A model based on the overall heat and mass transfer coefficients was proposed by Parker and Treybal [14] and then a new model made up of four equations with the respective boundary conditions and other auxiliary equations has been proposed by Zalewski [19]. It was found that the wet bulb temperature had a greater influence than in the model tests. Armbruster and Mitrovic [5] investigated the temperature distribution of cooling water along the sequence of condenser tubes and found that for certain conditions of inlet air condenser, different temperature values of the water due to increased air velocity and change in relative humidity were observed. Ettouney et al. [6] made an analysis of evaporative condensers as function of the ratio of air-mass flow rate passing through the equipment. It was also performed a comparison of condensers and cooling towers, where the efficiency of the equipment was 88% and 97% respectively. The data showed that the system efficiency increases with low air-water ratios and higher inlet temperatures of the superheated water vapor. Hwang et al. [11] compared the performance of an evaporative condenser with an air-cooled condenser. Both were mounted in a test chamber using the fluid R22. The results showed that the equipment had a capacity from 1.8 to 8.1% greater than the air-cooled condenser, and the coefficient of performance (COP) from 11.1 to 21.6%. Smrekar et al. [17] presented a study on how the natural drag can improve heat transfer in cooling towers. The work shows the influence of the air and water flow in the heat exchanger effectiveness, stating that two thirds of the heat rejected by evaporation and the remainder occurs by convection. It has been shown that the effectiveness increases by about 8% with a decrease in water flow rate of 0.51 kg/s. Qureshi and Zubair [15] presented an evaluation of performance by applying the Second Law in the evaporative condensers and cooling towers. It was observed that the Second Law efficiency falls as the increase in condensing temperature and is also lower for smaller flow rates of refrigerant. Thus, one can done an analogy between the phenomena that occurs in evaporative condensers and cooling towers, especially those operating in closed circuit which experiences the same functional structure that evaporative condensers (FIGURE 1) having a fluid flows internally to the tubes (responsible for the rate heat transferred to the external environment) and externally, the spray water flowing in counterflow with the air. The identification of possible inefficiencies related to the flow of air or water can be revealed by computer simulation, making mathematical modeling and rendering application of CFD (computational fluid dynamics) important tools in this field.

A numerical investigation of the heat transfer characteristics of an evaporatively-cooled condenser was done by Jahangeer et al. [12], where a detailed model is developed and numerical simulations are carried out using finite difference techniques. The numerical results were compared with available experimental and theoretical work and the agreement was found satisfactory.

Heyns and Kröger [10] perfomed experimental test on an evaporative cooler and the results show that the water film heat transfer coefficient is a function of the air mass velocity, deluge water mass velocity as well as the deluge water temperature, while the air–water mass transfer coefficient is a function of the air mass velocity and the deluge water mass velocity.

The study carried out by Al-Waked and Behnia [4] presents the results of the application of CFD in a three-dimensional numerical simulation of the operation of a cooling natural draft. The CFD model has utilized the standard $k - \varepsilon$ turbulence model in the FLUENT software through discrete phase model, where the air flow was solved as a continuous phase using the Eulerian approach. The water droplets trajectory was simulated as a dispersed phase, using the Lagrangian approach. The heat transfer from the water droplets in to surrounding air inside cooling tower consists of both convective and evaporative heat transfer, whereas heat transfer due to radiation is not significant, and has been neglected.

Another simulation of the cooling tower developed by Williamson et al. [18] presents a two-dimensional and axisymmetric biphasic simulation of heat transfer and mass, where the water droplets in the spray and inside the cooling tower were represented with trajectories of drops also written in Lagrangian approach using a computational model in FLUENT package, that account the coupling of heat and mass transfer between droplets and air.

The model was developed to solve the problem in a steady state through the Reynolds Averaged Navier-Stokes Equations (RANS) using $k - \varepsilon$ standard turbulence model with buoyancy terms included in the transport equations of k and ε. A semi-implicit method for the coupled velocity-pressure fields was used through the method of discretization with second order upwind scheme for the advective terms.

The results of these simulations showed good agreement with data from experimental measurements.

The characterization and selection of the model being used to simulate the distribution of diameter and evaporation prove to be quite important in the work done revealing the influence of particle diameter and relative humidity on the phenomenon of evaporation.

A hybrid cooling tower in closed circuit simulation through commercial software (ANSYS™ Fluent™) was made by Sarker et al. [16]. The research showed good agreement with experimental data. The standard $k - \varepsilon$ turbulence model was used, assuming a steady state incompressible flow. The coupling pressure-temperature field was made by the SIMPLEC algorithm. The problem was solved in a two-dimensional, adiabatic simulation, with no heat transfer between the equipment and the surrounding. For further simplification, an area that includes half of the tower was used, due the symmetry problem. The tubes were represented by cylinders with internal heat generation. A comparison of the performance of a prototype of a cooling tower in closed circuit from experimental measurements with those obtained through analysis and computer modeling done by Hasan and Sirén [9]. It revealed the importance of parameters such as pitch tubes, number and rows of pipes, flow of air and water, air velocity and pressure loss in the ability to reject heat from the tower and the power absorbed by it. This way, it can define the tower coefficient of performance (COP) which was the objective of optimization. The behavior of air, cooling water and the spray of water was also reported, showing how these parameters evolve in a vertical center line in the cooling tower from the first contact until the last tube.

Acunha Jr. and Schneider [2] performed an analysis of an evaporative condenser using CFD, based on the Kaiser et al. [13] modelling. It was used the commercial code FLUENT™. A multiphase modeling was adopted. The air, considered as a continuous phase using the Eulerian approach, consisted of a mixture of air and water vapor, while the water droplets were modeled as dispersed phase through the Lagrangian approach. A qualitative analysis showed that the velocity, pressure and temperature fields present a good agreement, in a general view, to the results presented in literature and according to those presented by the equipment manufacturer.

The present work aims to evaluate qualitatively a small scale evaporative condenser by using a CFD commercial code (FLUENT by ANSYS™ [7]) when compared to that presented on a previous work by Acunha and Schneider [2]. Since the velocity and pressure fields present a good topology agreement, one can track the main parameters which could be considered for a future quantitative and optimization analysis. Besides the size of equipment concerned, it presents a different design layout from that measured on geometry provided by the manufacturer Acunha and Schneider [2].

2. COMPUTATIONAL MODELLING

The computational modeling of the evaporative condenser is done by using a CFD commercial code, FLUENT™. The code is based on the element based finite volume method (EbFVM). It is used to simulate the air and water flows outside the tubes. The air consists of a mixture of air-water vapor and is modeled as a continuous phase using the Eulerian approach. The water droplets are modeled as dispersed phase through the Lagrangian approach. The model is solved according to the implicit transient formulation. The Realizable $k - \varepsilon$ turbulence model is adopted. The effects of buoyancy and standard wall function are considered. The SIMPLE algorithm is used for the coupling pressure-velocity fields. Because the length of the condenser is much larger than the height and width, the flow can be approximated as a two-dimensional problem. It is considered also adiabatic, with no heat transfer between the equipment and the surrounding. Further simplification, a field considering only half of the evaporative condenser is not used, due to the non symmetry presented by the problem. The schematic geometry which describes the problem can be seen in FIGURE 1.

The geometry of evaporative condenser under study considers a geometric similarity towards an evaporative condenser commercially manufactured to operate with R-717 in industrial refrigeration plants. The scale factor used is equal to four. The equipment has a cross section of 0.25 m wide by 0.51 m long. The coil is considered made of copper tubing with an outside diameter of 6.35 mm (corresponding to the commercial diameter ¼") and is composed of 35 columns, each with six tubes in a staggered arrangement, connected to a distributor at the top and a sink at the bottom, with a total heat exchange area equal to 2.032 m^2. Just above the coil is the water distributor. Above the water distributor is a drift eliminator to the air passages of 1.8 mm wide. The shape of the eliminator makes the air flow path has a 45 ° vertical direction to one side and then turn 90 ° in the other direction.

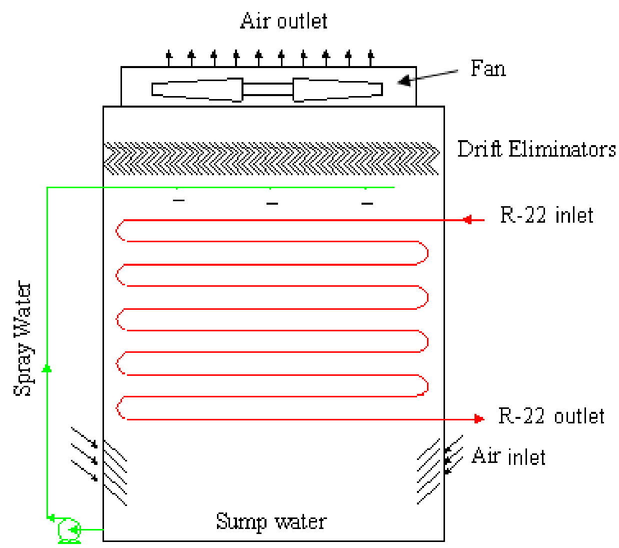

Figure 1: Schematic geometry for the evaporative condenser.

2.1 Governing Equations for the Air

The modeling for the turbulent flow of the continuous phase (air), including heat and mass transfer between this and the dispersed phase consists of conservation equations of mass, momentum energy and species. Equation (1) represents the equation of mass conservation.

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \vec{v}) = S_m \tag{1}$$

where t is time (s) ρ is the density (kg/m^3), v is velocity (m/s) and S_m is a source term that includes the addition of mass by the discrete phase due to vaporization of water droplets.

Equation (2) represents the conservation of momentum which includes the source term ($\vec{F}$) due to interaction with the discrete phase and the tension tensor ($\nabla(\bar{\tau})$) introduced, it is detailed in Equation (3).

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla(\rho \vec{v}\vec{v}) = -\nabla p + \nabla(\bar{\tau}) + \rho \vec{g} + \vec{F} \tag{2}$$

$$\bar{\tau} = \mu\left[\left(\nabla\vec{v} + \nabla\vec{v}^{T}\right) - \frac{2}{3}\nabla\vec{v}I\right] \tag{3}$$

where µ is viscosity molecular (kg/m.s) and *I* is the unit tensor.

The energy is calculated with the aid of Equation (4).

$$\frac{\partial}{\partial t}(\rho E) + \nabla\left[\vec{v}(\rho E + p)\right] = \nabla\left[k_{eff}\nabla T + \sum_{j} h_j \vec{J}_j + \left(\overline{\tau_{eff}}\vec{v}\right)\right] + S_k \tag{4}$$

where k_{eff} is the effective thermal conductivity (W/(mK)), corresponding to k+k_t where k_t corresponds to turbulent thermal conductivity, $\vec{J}_j$ is the mass diffusion flux of species *j* (kg/m².s). The first two terms on the right side of the equation refer to energy transferred by heat diffusion and mass diffusion, respectively. The third term corresponds to viscous dissipation, which was neglected. The energy *E* shown in Equation (4) is given by:

$$E = h - \frac{p}{\rho} + \frac{v^2}{2} \tag{5}$$

The species are conserved according to Equation (6).

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla(\rho\vec{v}Y_i) = -\nabla\vec{J}_i + R_i + S_i \tag{6}$$

In this equation, Y_i is mass fraction of each species *i*, R_i is the production of species by chemical reaction and S_i is the rate of creation by the addition of the dispersed phase. The production of species by chemical reaction was also neglected. The flow of mass diffusion equation used in the energy and species conservation equation is calculated by Equation (7).

$$\vec{J}_i = -\left(\rho D_{i,m} + \frac{\mu_t}{S_{Ct}}\right)\nabla Y_i \tag{7}$$

where S_{Ct} is the turbulent Schmidt number which is given by $\mu_t/(\rho D_t)$, μ_t being the turbulent viscosity and D_t turbulent diffusivity.

2.2 Governing Equations for Water

The dispersed phase consists of spherical particles of water being injected at six points per injection plane. The trajectory of the discrete phase particles can be determined by making the integration of the balance of forces on particles using Lagrangian reference. This force balance equates the particle inertia forces with the forces acting on the particles (Gan Et al. [8]) and can be written in the form of force per unit mass as shown in Equation (8).

$$\frac{dv_p}{dt} = F_D\left(v - v_p\right) + \frac{g\left(\rho_p - \rho\right)}{\rho_p} + \frac{\rho}{\rho_p} v_p \frac{\partial v}{\partial r_p} + \frac{1}{2}\frac{\rho}{\rho_p}\frac{d}{dt}\left(v - v_p\right) \tag{8}$$

where r_p is the trajectory of each particle and, v and v_p are the velocities (m/s) of the continuous phase and particles, respectively. The second term on the right side of the equation represents the gravitational force per unit mass, and the third, also on the right side of this equation, represents an additional force per unit mass that occurs due to the pressure gradient in the fluid. The last term is due to acceleration caused in the fluid surrounding the particle, which is important in cases where $\rho > \rho_p$ [8]. The first term refers to the drag force per unit mass experienced by the particles, it is given by:

$$F_D = \frac{18\mu}{\rho_p d_p^2}\frac{C_D \, \mathrm{Re}}{24} \tag{9}$$

where d_p is the diameter (m) of the particles. The Reynolds number (Re) is calculated by the relative velocity between particles and continuous phase, defined as:

$$\mathrm{Re} = \frac{\rho d_p \left|v_p - v\right|}{\mu} \tag{10}$$

The drag coefficient C_D is calculated with the aid of Equation (11). The constants a_1, a_2 and a_3 are constants given by Morsi and Alexander for spherical particles under different ranges of Reynolds numbers [8].

$$C_D = a_1 + \frac{a_2}{\mathrm{Re}} + \frac{a3}{\mathrm{Re}^2} \tag{11}$$

2.3 Boundary Conditions

The simulation done by Acunha and Schneider [2] of the condenser is made with the injection of water droplets through holes of 8 mm in diameter. The water is injected on small plates that promote the spreading of droplets on the tube bank. As the simulation is done in 2D, it is considered the unitary length, the flow of water and air used for simulation corresponds to 1/6 of the air and water flow from the real condenser as it has 6 m in length. The symmetry condition of the problem allowed simulating of only a half of the geometry, thus establishing a vertical line of symmetry in the center of the condenser.

The spray water temperature was 302 K, and is sprayed on the plates spread at a rate of 6 kg/s.

The air is admitted into the condenser under atmospheric pressure at an angle of 45º with the horizontal toward the condenser water sump at wet bulb temperature ($T_{WB,in}$) of 293.45 K and dry bulb temperature ($T_{DB,in}$) of 302.6 K.

At the air outlet a total manometric pressure of -370 Pa was considered at the ventilator intake, with $T_{WB,out}$ = 302.04 K and $T_{DB,out}$ = 303.4 K. These conditions ensure an air flow rate of approximately 7.4 kg/s.

The present work, which presents small scale equipment, is considered a Dirichlet boundary condition for the tube bundle, T_{tube} = 299.18 K. Although the tube bank can be divided on three distinct zones: dessuperheating, condensation and sub cooling, there was observed by thermography a homogeneous temperature distribution over the tubes, as presented by FIGURE 2 (Acunha Jr. and Schneider [1 and Acunha Jr. and Schneider [3]). In fact, this behavior can be explained by the fact that the regions of dessuperheating and sub cooling are very small, when compared to the condensation region. In the present work the symmetry condition is not used once there are design differences, such as the drift eliminator and the number of fans, when compared to that presented by Acunha and Schneider [2]. More details about the small scale evaporative condenser can be seen at Acunha and Schneider [3]. The spray water temperature is 297.75 K, which is sprayed on a rate of 0.12 kg/s. The inlet pressure for the air is considered as atmospheric. The air is admitted into the condenser under atmospheric pressure at wet bulb temperature ($T_{WB,in}$) of 292.93 K and dry bulb temperature ($T_{DB,in}$) of 296.56 K. At the air outlet a total manometric pressure of -47 Pa is considered with $T_{WB,out}$ = 296.39 K and $T_{DB,out}$ = 297.55 K.

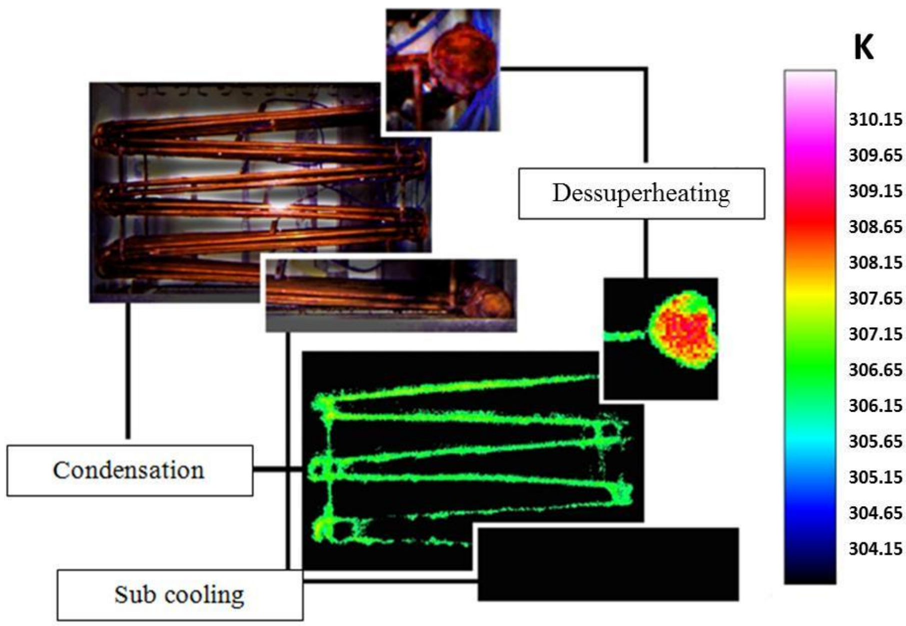

Figure 2: Tube bank thermography [1].

3. **RESULTS AND DISCUSSION**

Two simulations are performed in order to evaluate the improvement of the air flows conditions. The first case is done considering an inlet air with an angle of 45^o. The second one considers the inlet air horizontally.

FIGURE 3a shows the pressure field while FIGURE 3b presents the velocity field obtained by Acunha and Schneider [2] and the results obtained by the present work are depicted in the FIGURE 4a and 4b for the pressure and velocity fields, respectively, considering an inlet air angle of 45°.

The topology of two cases shows that the small scale evaporative condenser is very similar to those results presented by Acunha and Schneider [2] with respect to the velocity and pressures fields. Some differences are verified because the present computational simulation is performed considering the whole geometry domain, while for the study performed in Acunha and Schneider [2], only a half of the computational domain was simulated. The symmetry condition was not used once there are design differences, such as the drift eliminator and the number of fans.

The pressure drop of air flow in an evaporative condenser is a factor of great importance because it directly influences the costs of manufacturing and operating for such equipment. The intensity of the pressure drop is mainly due to factors such as slope of deflectors in the air inlet openings, tubes arrangement, tube spacing of the bank in relation to the walls, geometry and spacing of the drift eliminators, and position of the ventilator with respect to the condenser. The velocity (and therefore the flow) of air is strongly influenced by the resistance offered by the condenser air flow. FIGURE 5a shows the pressure field obtained from the simulation with inclination of 0^o at inlet air zone. The pressure in the condenser is reduced proportionally to its height, being smaller the closer it is to the exhaust air, as expected. When the air passes through the tube bank it suffers a slight pressure drop, which intensifies when it is passing by the drift eliminators. These also contribute to the formation of a non-homogeneous distribution of air in the fan intake, providing a stream of air with significant changes of direction. The drift eliminator of small scale evaporative condenser has a *V* shape providing non symmetric flows.

This behavior can also be observed in the velocity field shown in FIGURE 5b, noting the outlines formed by different velocities experienced from input to output.

At the bottom of the tube bank there are no more vortexes. The vortexes disappear around of these tubes due to the direction of air flow at inlet air region. Thus, a better air distribution for this zone can be achieved.

In this condenser, the highest speeds also are at the top of the entrance and the sides of the tube bank, where there is a spacing which offers less flow resistance than the spacing between tubes, providing a preferential outlet area, but inert to the heat transferred directly by the tubes.

Due to the lower pressure drop inside condenser, a greater air mass flow rate should be expected, but the increasing in this value has no significance as well. The air mass flow rate increased from 0.176 kg/s to 0.18 kg/s.

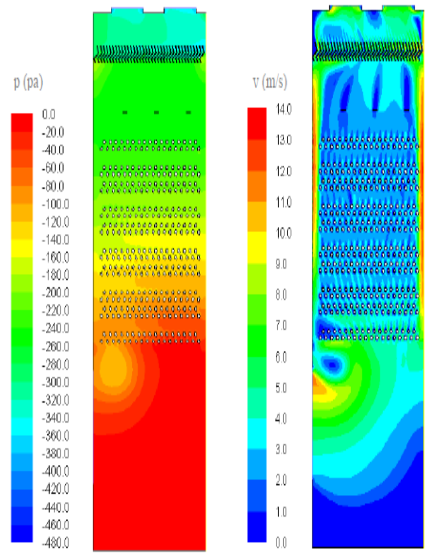

Figure 3: a) pressure field, Pa; b) Velocity field, m/s [8].

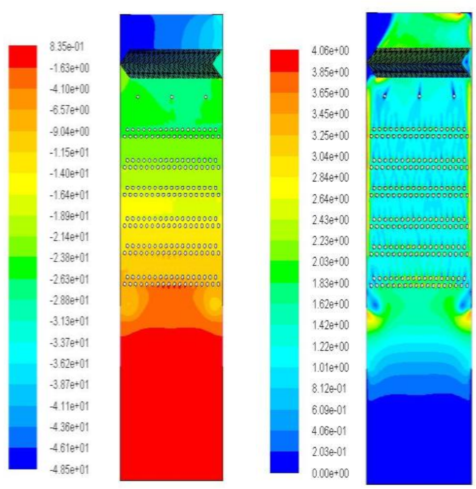

Figure 4: Topologies obtained in the present work: a) pressure field, Pa; b) Velocity field, m/s

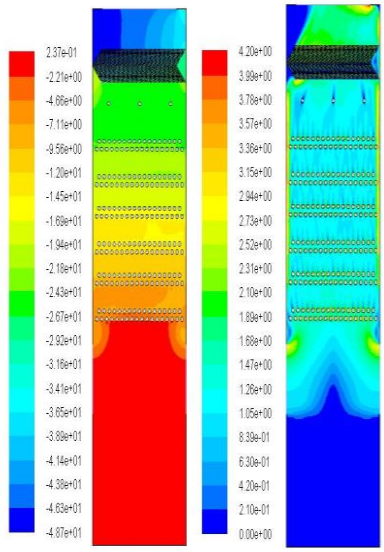

Figure 5: Topologies for an evaporative condenser with inclination of 0^o at inlet air zone: a) Pressure field, Pa; b) Velocity field, m/s.

4. CONCLUSIONS

The present work was proposed to evaluate the computational modeling for an evaporative condenser on a small scale. It was used the same model previously applied by Acunha and Schneider [2] in a major equipment. As can be observed, a good agreement was found between the cases. Since the model showed a good ability to describe the heat and mass transfer phenomena inside the condenser, it is possible to infer that for future design changes, when the tubes distribution, drift eliminator, temperature conditions and inlet air angle, this modeling can be used. Based on these findings, further work include the evaluation and optimization of the following: inlet air angle; tubes distribution; geometry of the tubes; geometry of drift eliminator; form of spray water and overall dimensions of the equipment in order to find the optimum air distribution inside the condenser, increasing its thermal efficiency. Thus, from a qualitative analysis presented in this paper in order to validate the numerical model, others quantitative assessments are being developed.

REFERENCES

[1] ACUNHA Jr. I. C.; WALTER F. M.; SCHNEIDER P. S.; BEYER P. O. Experimental analysis of and evaporative condenser. **Proceedings of 13th Brazilian Congress of Thermal Sciences and Engineering – ENCIT2010**, Uberlândia, Brazil, 2010.

[2] ACUNHA Jr. I. C.; SCHNEIDER P. S. Evaluation of air-water flow in an evaporative condenser. **IEEE Computer Society proceedings 3rd Southern Conference on Computational Modeling**, Rio Grande, Brazil, p. 49-54, 2010.

[3] ACUNHA Jr. I. C.; SCHNEIDER, P. S. Experimental Assessment of Heat Transfer Correlations for a Small Scale Evaporative Condenser. **Heat Transfer Engineering**, v. 34, p. 532 – 540, 2013.

[4] AL-WAKED R.; BEHNIA M. Enhancing performance of wet cooling towers. **Energy Conversion & Management**. v. 48, p. 2638-2648, 2007.

[5] ARMBRUSTER R.; MITROVIC J. Evaporative cooling of a falling water film on horizontal tubes. **Experimental Thermal and Fluid Science**, v. 18 (3), p. 183 – 194, 1998.

[6] ETTOUNEY H. M.; EL-DESSOUKY H. T. BOUHAMRA W., AL-AZMI B. Performance of evaporative condenser. **Heat Transfer Engineering**, v. 22 (4), p. 41-55, 2001.

[7] **FLUENT: User Guide**, Fluent Inc., (2012). < http://www.fluent.com>

[8] GAN G.; RIFFAT S. B.; SHAO L.; DOHERTY P. Applications of CFD to closed-wet cooling towers. **Applied Thermal Engineering**. v. 21, p. 79-92, 2001.

[9]HASAN A.;, SIRÉN K. Theoretical and computational analysis of closed wet cooling wet cooling towers and its application in cooling of building. **Energy and Buildings**, v. 34, p. 477-486, 2002.

[10] HEYNS J. A.; KRÖGER D. G. Experimental investigation into the thermal-flow performance characteristics of an evaporative cooler. **Applied Thermal Engineering**, v. 30, p. 492-498, 2010

[11] HWANG Y.; RADERMACHER R.; KOPKO W. An experimental evaluation of a residential sized evaporatively cooled condenser. **International Journal of Refrigeration**. Vol. 24, pp. 238-249, 2001

[12] JAHANGEER K. A.; ANDREW A. O.; RAISUL I. Md. Numerical investigation of transfer coefficients of an evaporatively-cooled condenser. **Applied Thermal Engineering**, v. 31 (10), p.1655-1663, 2011.

[13] KAISER A. S.; LUCAS M.; VIEDMA A.; ZAMORA B. Numerical model of evaporative cooling processes in a new Type of Cooling Tower. **International Journal of Heat and Mass Transfer**, v. 48. p. 986-999, 2005.

[14] PARKER R. O.; TREYBAL R. E. **The heat, mass transfer characteristics of evaporative coolers**. New York University. Source: American Doctoral Dissertations, Sorce code X1959., page:127, 1959

[15] QURESHI B. A.; ZUBAIR S. M. A Comprehensive design and rating study of evaporative coolers and condensers: part I. Performance evaluation. **International Journal of Refrigeration**, v. 29, p. 645-658, 2006.

[16] SARKER M. A.; KIM E.; MOON C. G.; YON J. I. Performance characteristics of the hybrid closed cooling tower. **Energy and Buildings**, v. 40, p. 1529-35, 2008.

[17] SMREKAR W. F.; OMAN J.;SIROK B. Improving the efficiency of natural draft cooling towers. **Energy Conversion and Management**, v. 47, p. 1086-1100, 2006.

[18]] WILLIAMSON N.; ARMFIELD S.; BEHNIA M. Numerical simulations of flow in a natural draft wet cooling tower – the effect of radial thermofluid fields. **Applied Thermal Engineering**. v. 28, p. 178-189, 2008.

[19] ZALEWSKI W. Mathematical model of heat and mass transfer processes in evaporative condensers. **International Journal of Refrigeration**, v. 16 (1), p. 23-30, 1993.