## PAPER • OPEN ACCESS

# Thermo-hydraulic analysis of the condenser (helically-coiled type) of an air-water heat pump, using CFD numerical simulation

To cite this article: R López et al 2019 J. Phys.: Conf. Ser. 1221 012043

View the article online for updates and enhancements.

## You may also like

- <u>Managing momentum in climate</u> <u>negotiations</u> Stefano Carattini and Andreas Löschel
- <u>Condensate water as a compressor</u> discharge cooler to generate subcooling on the residential air conditioning using <u>R32 as refrigerant</u> K Sumeru, AS Margana and S Hidayat
- <u>Synthesis, Model and Stability of Helically</u> <u>Coiled Carbon Nanotubes</u> Dora Fejes, Zoran P. Popovi, Manuella Raffai et al.





DISCOVER how sustainability intersects with electrochemistry & solid state science research



This content was downloaded from IP address 18.221.187.121 on 28/04/2024 at 03:31

IOP Conf. Series: Journal of Physics: Conf. Series 1221 (2019) 012043 doi:10.1088/1742-6596/1221/1/012043

## Thermo-hydraulic analysis of the condenser (helically-coiled type) of an air-water heat pump, using CFD numerical simulation

**R López<sup>1</sup>, M Vaca, H Terres, A Lizardi, S Chávez, O Rocha, E. Meza**. Universidad Autónoma Metropolitana Azcapotzalco, Av. San Pablo No. 180, Col. Reynosa Tamaulipas, México, 02200.

E-mail: rlc@correo.azc.uam.mx

Abstract. This paper presents the results obtained from the modeling of the flow and heat transfer in the condenser of the heat pump used for heating the water of a pool, by means of CFD analysis. Using the k- $\epsilon$  turbulence model its real behavior was described; these data may be useful to propose the improvement of its thermal design. The distributions of temperature, pressure, and velocity, both for the water and the R-410A refrigerant were obtained. The value of the COP of the pump which was 7.6 was found through the results. The experimental value of this variable was 7.32, both values are lower than those offered by the manufacturer of this equipment.

#### 1. Introduction

The energy that must be provided to the water supply of a public pool was experimentally quantified and reported by López et al. [1]. That energy is delivered by a system of air/water- type heat pumps, which work alternately to meet the required demand, considerably saving energy, as compared to, for example, a gas water heater or an electric boiler. All the experimentation required for its assessment was performed and we found that the coefficient of performance (COP) of the pump ranged between 6.39 and 7.32, slightly lower values than those reported by the manufacturer.

An important element of these pumps is the condenser, which had some design defects. Its operation was then described by means of the numerical simulation for the subsequent proposal of redesign. The heat exchanger installed in the condenser is helically-coiled type, offering the advantage of allowing a greater heat transfer in a smaller area, thus rendering a device of smaller physical dimensions, but also with the disadvantage of direction change in coils resulting in a greater loss of energy than in the straight sections of piping.

There have been several analyses on experimentation and modeling of these heat exchangers, for instance, Purandare et al. [2] observed that helical spirals were efficient for low Reynolds number (Re) values. Jayakumar et al. [3] made some experimental fluid-to-fluid heat transfer studies through a helically coil tube using Computational Fluid Dynamics (CFD), and noted that CFD predictions reasonably corresponded to the experimental results for all operating conditions. Kumar and Karanth [4] analyzed and compared straight-tubular and helically coil-type heat exchangers using CFD under identical operating conditions and the same length; the results showed a significant increase in heat transfer rate, but pressure drop of the flow increased. Kumar et al. [5] developed the modeling of a helical tube, mainly focused on the presence of secondary flow and its possible consequences.

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI. Published under licence by IOP Publishing Ltd 1 The objective of this work is to present the analysis of the modeling of the flow and heat transfer in the condenser of the heat pump used for water heating of a public swimming pool, using CFD, based on the  $k - \epsilon$  turbulence model.

## 2. Background.

The water heating system of the Azcapotzalco-Aquatic Center pool consists of 18 heat pumps; they are all of the same model and are arranged under a combined work schedule, on average, 10 pumps are required to work at the same time. For the analysis of performance, one of them was instrumented with temperature and pressure gauges at the entrance and exit of the fluid, and the necessary equipment to measure the consumption of electric power from the compressor [1]. In addition, using the original instrumentation of the pool, inlet and outlet temperatures of the water supply, pressures of the recirculation pump and volumetric flow, were measured. The Olympic pool is 50 m long, 25 m wide and has an average depth of 1.50 m. Since its service is public, it must be provided from 6:00 to 21:00 hours, 6 days a week.

Registered average temperature values were: compressor outlet/condenser inlet, 88.0 °C; condenser outlet/expansion valve inlet, 37.0 °C; expansion valve outlet/ evaporator inlet, 3.5 °C; the refrigerant leaves the evaporator and enters the compressor almost with the same temperature value. The basic components of the heat pump are presented in Figure 1, namely 1, compressor, 2, condenser, 3, expansion valve, and 4, evaporator. The volumetric flow of R-410A refrigerant, provided in the pump data chart, is  $0.004 \ m^3/s$  and the transversal section of the pipe area is  $3.16 \times 10^{-6} \ m^2$ . The water to be reheated enters the condenser at 29 °C with a maximum flow of  $0.0063 \ m^3/s$ . These values correspond to turbulent flow for both streams and the critical Reynolds number calculated from the definition of helical tube given by Dean [2, 3] is turbulent in both cases. The internal diameter and length of the shell are 0.12 m and 1.00 m, respectively, the diameter of the connection of inlet and outlet pipes is  $0.012 \ m$ .

### 3. Numerical simulation.

The numerical model must solve the conservation of continuity, and momentum and energy quantity equations adapted to the  $k - \varepsilon$  model. It is one of turbulence models widely used in this type of applications, based on the following equations: the continuity equation,

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

the movement quantity equation,

$$\frac{\partial u_i u_j}{\partial x_i} = -\frac{1}{\rho} \frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_i} \left( (v + v_t) \left( \frac{\partial u_j}{\partial u_i} + \frac{\partial u_i}{\partial x_j} \right) \right)$$
(2)

and the energy equation.

$$\frac{\partial u_i T}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \left( \frac{v}{Pr} + \frac{v_t}{Pr_t} \right) \frac{\partial T}{\partial x_i} \right)$$
(3)

The model introduces two additional transport equations and two dependent variables: the turbulence kinetic energy, k, and the turbulent dissipation rate,  $\varepsilon$  [6]. The turbulent viscosity  $\mu_T$ , is modeled as:

$$\mu_T = \rho C_\mu \frac{k^2}{\varepsilon} \tag{4}$$

where  $C_{\mu}$  is a constant. The transport equation for k is:

$$\rho \frac{\partial k}{\partial t} + \rho \boldsymbol{u} \cdot \nabla k = \nabla \cdot \left( \left( \mu + \frac{\mu_T}{\sigma_k} \right) \nabla k \right) + P_k - \rho \varepsilon$$
(5)

IX International Congress of Physics Engineering

IOP Conf. Series: Journal of Physics: Conf. Series **1221** (2019) 012043 doi:10.1088/1742-6596/1221/1/012043

where  $P_k$  is the production term defined by:

$$P_{k} = \mu_{T} \left( \nabla \boldsymbol{u} : (\nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^{T}) - \frac{2}{3} (\nabla \cdot \boldsymbol{u})^{2} \right) - \frac{2}{3} \rho k \nabla \cdot \boldsymbol{u}$$
(6)

The transport equation for  $\varepsilon$  is:

$$\rho \frac{\partial \varepsilon}{\partial t} + \rho \boldsymbol{u} \cdot \nabla \varepsilon = \nabla \cdot \left( \left( \mu + \frac{\mu_T}{\sigma_{\varepsilon}} \right) \nabla \varepsilon \right) + C_{\varepsilon 1} \frac{\varepsilon}{k} P_k - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k}$$
(7)

The values of the constants used in equations (1), (2), (3), and (4) have been determined from experimental data, resulting in:  $C_{\mu} = 0.09$ ,  $C_{\varepsilon 1} = 1.44$ ,  $C_{\varepsilon 2} = 1.92$ ,  $\sigma_k = 1.0$  and  $\sigma_{\varepsilon} = 1.3$  [6]. The V 4.3 COMSOL Multi-physics software is used for the solution of these equations, the model should consider the flow of two fluids in turbulent regime that do not mix together, coupled with an energy transport through these fluids, with the conditions of non-isothermal flow and conjugate heat transfer. The selected mesh was fixed using thick-size elements, namely 1,393,388 domain elements for the 50 laps-geometry, Figure 2.



Figure 1. Components of the heat pump.



Figure. 2. Mesh for the shell and double-helix coil.

**IOP** Publishing

#### 4. Results and analysis.

The temperature distribution of the water contained in the interior of the shell indicates that water enters the condensate at a temperature equal to that enforced in the 29 °C boundary condition and the heating process starts immediately. As the water advances upwards, it warms from the coil towards the shell, the fluid that surrounds the coil has a higher temperature than the outside which becomes uniform towards the exit, reaching 32 °C; the temperature distribution is displayed in Figure 3. To better observe this temperature rising process, three sections are displayed in Figure 4. The first corresponds to the inlet, where the fluid starts to heat very slowly; in the central section of the condenser, the water between coil tubes is approximately at 34.5 °C and externally, it is very close to 31.0 °C. Finally, the water temperature evens to 32.0 °C in the upper section. In all three cases water that stands between the pipes of the two coils has a non-homogeneous distribution, since the fluid close to the outer tube is at lower temperature that the internal coil, this is due to heat transfer from the refrigerant circulating through them.

These results can be observed from a better perspective in Figure 5, where three sections are presented, the first one shows temperature uniformity at the inlet, due to start of the heating process; the middle section, at the center, has a yellow ring that correspond to fluid located between the input coils and output of the refrigerant and the right one is the overhead section, clearly showing that the fluid has increased its temperature.

IOP Conf. Series: Journal of Physics: Conf. Series **1221** (2019) 012043 doi:10.1088/1742-6596/1221/1/012043





**Figure 3.** Temperature distribution in the transverse plane of the shell, the condenser is shown in horizontal position; water enters through the left side and exits through the right extreme of the figure.

**Figure 4.** Three sections of the condenser, input, half and a higher one, which corresponds to the exit of the hot water.

Since the fluid does not change state and temperature rise does not affect the density, the velocity of water entering and exiting the condenser is virtually the same, 3 m/s. The distribution of velocity within the device is irregular, since the diameter is greater, the fluid velocity near the borders is less than 0.33 m/s, inside the coil it reaches a value of 2.5 m/s and towards the center a maximum velocity of 2.7 m/s is measured, this behavior is maintained all along the condenser, Figure 6.



Figure 5. Cross-sections of the condenser, inlet, intermediate, and exit of water.

Figure 6. Distribution of water velocity inside the condenser.

An illustration of the velocity vectors can be seen in Figure 7, depicting a slice from the center of the condenser to the far right for its middle section. The inner piping corresponds to the coil where the refrigerant enters; it exits through the external pipe. The velocity of water close to the wall of the device is the lowest and towards the center is maximum, between the coil tubes the fluid moves at 0.7 m/s.

The pressure distribution of water inside the condenser is 26 kPa, and 0.10 kPa at the exit, the difference originates from the loss of energy of water as it passes through the condenser, approximately 25.9 kPa; the results obtained with the software do not actually provide more information on this variable, as shown in Figure 8.

The condenser is formed by two coils of 25 laps each, the inlet is located at the internal spiral and the exit on the opposite one. This configuration is presented in Figure 9, R410A refrigerant circulates through it and software results indicate 87.0 °C at the inlet and 68.0 °C at the end of the inner helix, when it comes out through the outer helix, 37.0 °C were recorded. This process of energy loss may be appreciated by the color change. The fluid that surrounds these coils is benefited with this energy, reaching the aimed warming process.

IOP Conf. Series: Journal of Physics: Conf. Series 1221 (2019) 012043 doi:10.1088/1742-6596/1221/1/012043





Figure 7. Water velocity distribution, through vectors, inside the condenser.

Figure 8. Water pressure distribution at the inlet and outlet of the condenser.

Temperature and pressure of the refrigerant both depend on the existing thermodynamic state. At 87 °C for the inlet, software results indicated 5.48 MPa, a superheated steam state. At 68.0 °C, at the exit of the inner helix, the pressure is 3.0 MPa and the refrigerant is still in a state of overheated vapor. Within the coil; at 37.0 °C, at the exit of the condenser, pressure is 2.3 MPa and its state is saturated liquid. Details of the inlet and the bottom of the coil are presented in Figure 10.



inside the condenser.

Figure 9. Temperature distribution of refrigerant Figure 10. Bottom section and inlet of the condenser.

Since refrigerant changes of phase from superheated steam to saturated liquid, and according to the diagram of the working fluid, the input velocity is 44.04 m/s, and the output one is 2.8 m/s. This process for the bottom section and the condenser inlet can be observed in Figure 11.

With these temperature-pressure values and the R-410A Mollier diagram, enthalpies at the pump compressor inlet and outlet were calculated as 454.87 kJ/kg and 260.47 kJ/kg, respectively; for the inlet and outlet of the condenser, the same value was determined, 429.28 kJ/kg. Using the definition of COP [7], given as:

$$COP = \frac{Q_{cond}}{\dot{W}_p} \tag{8}$$

where  $Q_{cond}$  is the flow of heat in the condenser and  $\dot{W}_p$  is the drive power of the heat pump, the COP numerical value is 7.6. Compared to the experimentally retrieved COP of 7.32, both values are below the value offered by the manufacturer of these devices, 8.0. The objective in the following work should be to make the necessary modifications by simulation to achieve this value.

IOP Conf. Series: Journal of Physics: Conf. Series 1221 (2019) 012043 doi:10.1088/1742-6596/1221/1/012043



Figure 11.- R-410A Velocities distribution, bottom section and condenser inlet.

### 5. Conclusions

The flow and heat transfer in the condenser of the heat pump used for heating the water of a pool were modelled using CFD analysis, with the k- $\epsilon$  turbulence model. The aim was to obtain its actual behavior, which may be useful for the improvement of the thermal design. The distributions of temperature, pressure and velocity, both for water and refrigerant R-410A were described. The temperature increase for water was of 3.0 ° C with a pressure drop of 25.9 kPa. The velocity in the inlet and outlet pipe is 3.0 m/s and inside the heat exchanger, 2.5 m/s. For the refrigerant, with temperature and pressure data, and the Mollier diagram, the enthalpy was calculated at the input and output of the coil and the input and output to the pump compressor with which it was possible to get the value of the COP of the pump which was 7.6. The experimental value of this variable was 7.32, both values are lower than the offered by the manufacturer of this equipment. These data may be useful to improve the thermal design of the condenser through simulation.

#### 6. References

- [1] López R, Vaca M, Terres H, Lizardi A, Morales J and Chávez S 2017 Experimental evaluation of a heat pump for the water-supply heating of a public swimming pool *Journal of Physics: Conference Series 2017* **792** pp 1-7
- [2] Purandare P S, Lele M M and Gupta Purandare R 2012 Parametric Analysis of Helical Coil Heat Exchanger, *International Journal of Engineering Research and Technology* **1** 1-5
- [3] Jayakumar J S, Mahajani S M, Mandal J C, Iyer K N and Vijayan P K 2010. CFD analysis of single-phase flows inside helically coiled tubes *Computers and Chemical Engineering* 34 pp 430-46
- [4] Kumar S and Karanth V K 2013 Numerical Analysis of a Helical Coiled Heat Exchanger using CFD *International Journal of Thermal Technology* **3** pp 126-30
- [5] Kumar Reddy K, Sudheer Prem Kumar B, Ravi Gugulothu, Kakaraparthi Anuja P, Viajaya Rao 2017 CFD Analysis of a Helically Coiled Tube in Tube Heat Exchanger *Materials Today: Proceedings* 4 pp 2341–9
- [6] Comsol. 2013. COMSOL Multiphysics. CFD Module User's Guide. Version 4.3 COMSOL AB.
- [7] Cengel Y A and Boles M 2011 Thermodynamics (McGraw-Hill, USA)