Experimental and Numerical Study of the Dynamic Behavior of an Air to Water Heat Pump

Nicolás Ablanque[a,*], Santiago Torras[a], Imanol Aizpurua-Udabe[a], Carles Oliet[a], Joaquim Rigola[a]

[a]Universitat Politècnica de Catalunya – Barcelona Tech (UPC), Heat and Mass Transfer Technological Center (CTTC), Terrassa, Barcelona, Spain

Abstract

This work is focused on studying the dynamic behavior of an air to water heat pump both experimentally and numerically. The system consists of a conventional heat pump cycle (compressor, condenser, capillary tube and evaporator) which transfers heat from the surrounding air into a storage tank with water. The transient heating process of the water has been analyzed under different external conditions (ambient temperature), operational settings (refrigerant mass charge) and geometric characteristics (capillary tube length). A numerical model has been used to predict the system response under all the aforementioned conditions using a pseudo-transient approach. The model has been validated with measurements obtained from a fully instrumented experimental facility. The results show relatively good agreement against measurements.

Keywords: air to water, water heater, transient, numerical model, experimental data

1. Introduction

Air-to-water heat pumps are systems that work under conventional vapor compression principles. Their role is to heat up flowing or stored water by extracting energy from the surrounding air. The obtained hot water is usually used for direct supply or heating purposes. These units are very interesting from the consumption point of view as their efficiency is much higher than conventional electric water heaters [1]. In fact, several technical studies - attempting to better understand and improve the performance of such systems - have been carried out and published in the open literature [2-5]. Among the possible approaches to analyze such units, computer simulations are widely used due to their reduced cost and execution time.

The ideal simulation tool for complex systems with several elements interacting with each other should have a generic approach and be numerically stable, robust, flexible, accurate and fast. These desirable characteristics may come into conflict between each other so that the model characteristics should be oriented to achieve the best compromise [6]. A comprehensive numerical analysis of a heat pump should include steady-state predictions, for performance analysis and unit design, but also transient simulations, for system control strategies and energy consumption calculations. In addition, numerical predictions should be compared against experimental data in order to appropriately validate any model.

The present work reflects the recent efforts done by the authors to develop a reliable and low-time consuming numerical tool enabling both steady and dynamic simulations of air to water heat pumps. For this purpose an in-house numerical platform has been adapted to simulate a heat pump during the transient process of heating an amount of water stored in a reservoir. A particular numerical pseudo-transient approach has been implemented in order to achieve relatively accurate transient predictions but with reduced execution time. This document represents a further step of the previous work presented in [7] where a very similar air to water heat pump has

* Corresponding author. Tel.: +34 93 739 81 92; fax: +34 93 739 89 20.
E-mail address: nicolas@cttc.upc.edu.

been studied at steady-state conditions.

The second Section of this paper is devoted to explain the system numerical model while the third Section is focused on the system components. In the fourth Section an experimental unit is presented together with relevant details of the cases used to validate the model. In the fifth Section the model predictions are compared against the experimental data but also additional studies are conducted to analyze the influence of different parameters (capillary tube length, air temperature, and refrigerant mass charge).

2. System Numerical Model

2.1. Numerical tool and transient approach

The calculation of the whole system has been carried out using the object-oriented in-house numerical tool called NEST. This platform is based on a modular approach where complex systems such as heat pumps are represented as a collection of different components connected between them. Each component represents a specific part of the system (e.g. heat exchangers, compressor, storage tank, capillary tube, oil separator, etc.) and can be independently solved from a given set of boundary conditions. The whole system is solved iteratively by calculating all its components and transferring information between them, until a converged solution is reached. The numerical model allows both steady and transient predictions and includes several features such as system layout adaptability, multi-level components, capacity to study control strategies, parallel computing, and system optimization techniques. The standard transient calculation scheme is depicted in Figure 1.

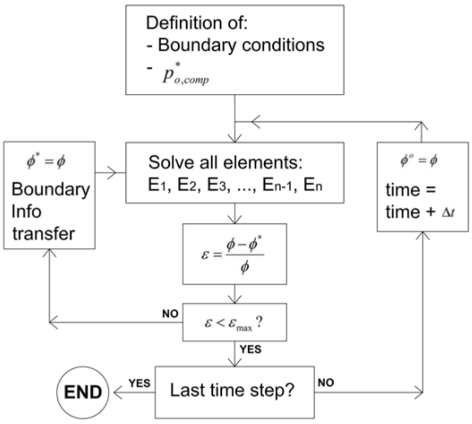


Fig 1. Simplified scheme of transient resolution.

2.2. Pseudo-transient approach

The simulation of a heat pump over a relatively extended period of time by means of a transitory resolution - where the accumulative terms of the components are calculated - is time consuming and difficult to achieve due to the number of components and circuits involved. Therefore a much more robust and faster alternative to model dynamic systems has been implemented: the pseudo-transient approach. It assumes that the relevant transient phenomena in the whole system are mostly related to the water circuit and the tank. This approach cannot be strictly accurate as the vapor compression cycle thermal and fluid-dynamic accumulative terms are not considered. However, the resulting negative effect on the system dynamics may be relatively low as the refrigerant cycle time constant is generally smaller than that of the water loop.

For instance, the studied heat pump is made of two sub-systems. On one side, the refrigerant cycle which includes a compressor, a condenser, a capillary tube, an evaporator, and other minor elements such as a mass flow meter or an oil separator. And on the other side, a water loop which includes a pump to drive the fluid and a reservoir to store the water. Both sub-systems interact at the condenser component where heat is transferred from the refrigerant cycle to the water loop. The simplified block diagrams of both sub-systems are shown in Figure 2 (left).

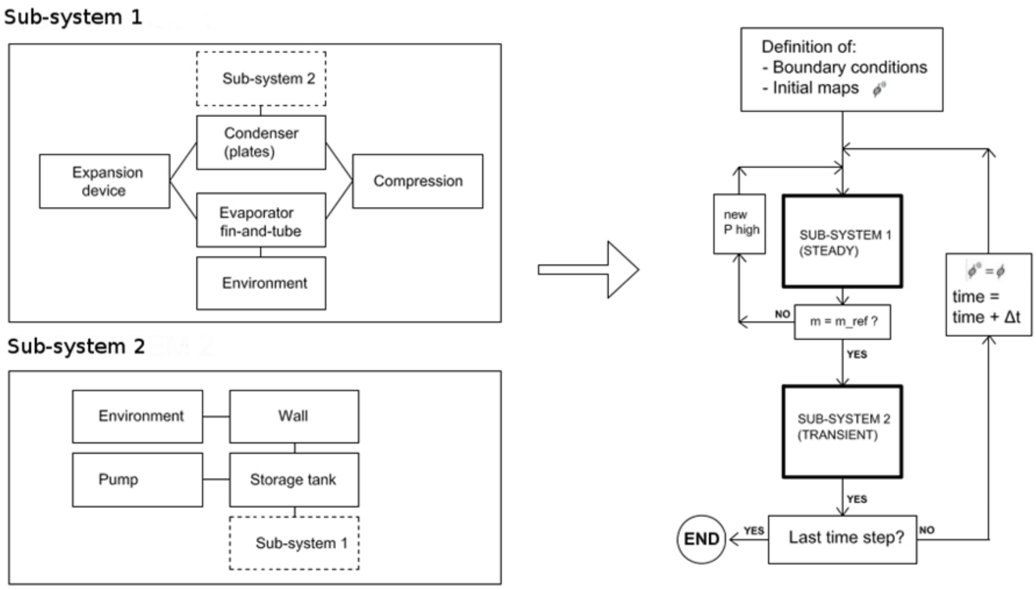


Fig 2. Sub-systems schemes (left) and combined resolution procedure (right).

The pseudo-transient simulation of the whole system is carried out by solving the refrigerant cycle at steady-state conditions but the water loop at transient conditions in a consecutive way as shown in Figure 2 (right). At each time step, the current storage tank water temperature is used as the condenser auxiliary flow inlet temperature when the refrigerant cycle is being solved, while the current heat flow calculated in the condenser is used as an input when the water loop has to be solved.

The water circuit is solved with the standard transient procedure presented in Figure 1. Instead, the refrigerant cycle is solved at steady-state conditions using the scheme presented in Figure 1 but setting a very large time step so that the accumulative terms in all components become negligible. This modification leads to an indeterminate set of equations as the mass flow rate value is common to all components. To overcome this issue, one of the refrigerant cycle variables is kept constant during the calculation of the refrigerant cycle (i.e. the compressor outlet pressure). Therefore, to achieve full closure, the refrigerant mass inside the whole system must be known and a new iterative loop must be added (see the resolution of sub-system 1 in Figure 2). In this way, after each refrigerant cycle converged solution at a specific compressor outlet pressure, the current refrigerant charge mass is calculated (by adding up the refrigerant mass from each component) and compared against the actual refrigerant charge. If the calculated mass does not match with the reference mass a new iteration starts with a corrected compressor outlet pressure.

The refrigerant circuit could be fully solved at each iteration step as shown in Figure 2. However, in order to further speed up the pseudo-transient resolution, the vapor compression cycle has been modeled as a set of parametric equations. For this purpose, a set of steady-state simulations of the refrigerant cycle, varying different parameters such as the condenser auxiliary temperature, are carried out prior to the pseudo-transient simulation. In this way, the whole transient resolution takes just few seconds to run as the refrigerant cycle response is extremely fast.

3. Numerical Models of Components

The main circuit of the studied heat pump is basically made up of the following elements: a commercial rotary compressor, a plate-type condenser used to heat up water flowing through an auxiliary circuit, a fin-and-tube evaporator used to extract heat from the surrounding air, a capillary tube used as the expansion device, and other minor elements such as mass flow meters, connecting tubes and oil separator, that may add some pressure losses to the circuit. The auxiliary loop includes a water storage reservoir, a pump to drive the water, and some connecting tubes. The simulation models used to predict the thermal and fluid-dynamic behavior of the aforementioned components are briefly explained in the following sections.

3.1. Compressor

The compressor model is based on the approach reported in [8]. The compressor consists of three main parts, namely, the shell inner volume, the compression chamber, and the discharge line. Both a mass and an energy balance equations are applied to the fluid inside the shell, while an energy balance over the compressor solid part is also considered. The model is fed with some empirical heat transfer coefficients and assumes the following hypotheses: oil effects are neglected, the suction pressure is equal to the pressure inside the shell, the suction and discharge mufflers influence is not taken into account, and the mixture inside the shell is considered thermally homogeneous. In addition to the model equations, the compressor is characterized by both the volumetric and isentropic efficiencies. These efficiencies are also expressed as variables depending on the compression pressure ratio [9] (the mathematical relations could be obtained from the data provided by the manufacturer, from experimental tests, or from detailed numerical simulations carried out previously).

3.2. Non-adiabatic connecting tubes

The thermal and fluid-dynamic behavior of fluid flow inside tubes is predicted with a distributed two-phase fluid flow model based on [10]. The fluid domain is represented by means of consecutive control volumes where the governing equations (continuity, momentum and energy) are applied and solved. The flow is evaluated on the basis of a step-by-step numerical implicit scheme where the wall temperature map acts as the boundary condition. The formulation requires the use of empirical correlations to evaluate the void fraction, the shear stress, and the convective heat transfer coefficient used to evaluate the heat transferred between the tube and the fluid.

3.3. Heat Exchangers

For both plate and fin-and-tube heat exchangers two levels of simulation could be considered. Firstly, the detailed model called CHESS [11] where the domain is divided into a set of control volumes (e.g. fin-and-tube blocks for the fin-and-tube evaporator). The model allows steady and transient analysis, flexible geometry and circuitry, working at dry or wet/frosting conditions. The inner refrigerant flow is solved with the two-phase flow model where non-uniform heat transfer coefficients can be considered in radial and axial directions. And secondly, a numerical model based on a lumped approach [12] where the heat exchanger is divided into different macro zones where semi-analytic methods (ε-NTU and F-ΔT_{lm}) are applied. For instance, the condenser is represented by three zones (super-heated gas, two-phase, and sub-cooled liquid) as well as the evaporator (two-phase, dry-out, and super-heated gas). Each of the defined zones has particular phenomenological characteristics and considerations.

The second method has a poorer level of detail as it considers a uniform behavior within each zone (i.e. constant physical properties and constant heat transfer coefficient). In addition, both the external heat losses and the axial heat transfer between zones are neglected. Only steady-state conditions are possible so that no ice accumulation could be considered (except by means of a specific parameter). However, if most of the simplifying hypotheses are fulfilled, and all the geometrical details are taken into account when the heat transfer is calculated, a great accuracy could be attained. In fact, its multi-zonal nature allows the model to adapt fairly well to heat exchanger sections having different global heat transfer coefficients.

The lumped model has been extensively validated in previous works and has been chosen for the present work because of its fast response and its reasonably good accuracy which are appropriate for the simulation of a whole system (e.g. heat pump water heaters).

3.4. Expansion Device

The expansion device used in the system was an adiabatic capillary tube which does not operate efficiently over a wide range of conditions compared to a thermostatic expansion valve. However, it is reliable, less expensive and performs nearly well.

The algorithm to simulate the capillary tube is based on the two-phase flow model mentioned in Section 1.2 and detailed in [13]. The resolution procedure is carried out in two steps. The first step consists in determining the capillary tube critical condition. The mass flow rate inside a capillary tube increases as the evaporating temperature decreases (lower discharge pressure) but only up to a critical value from which the mass flow rate remains constant. This critical limit occurs when the entropy generation equation is not accomplished anymore. It can be alternatively calculated when dp/dz approaches to infinity at the capillary tube discharge end. Thus, by comparing the current discharge pressure with the critical pressure (outlet pressure at critical conditions) it is possible to deduce if the capillary tube is operating at critical or non-critical conditions. The second step consists in performing the corresponding simulation according to the capillary tube critical or non-critical behavior. If the flow is non-critical, new iterations are carried out in order to find the refrigerant thermal and fluid-dynamic behavior taking into account the actual discharge pressure. However, if the flow is critical, an additional control volume is considered at the capillary tube outlet end. There, an energy balance is applied to calculate the capillary tube discharge enthalpy (heat transfer and transient terms are neglected).

3.5. Storage tank and other minor elements

The storage tank consists of a single control volume and its simulation is based on a global energy balance where heat losses through the walls are considered. In the numerical model, the storage tank is not represented as a single component but as a collection of components (e.g. liquid volume, wall, insulation, external environment etc.) that are linked together. A stratified approach for the tank model would be more accurate but for the aim of this work – which is focused on testing the whole system resolution – the present model should be good enough. In other minor elements such as the mass flow meter or the oil separator the pressure drop is calculated based on the manufacturer information.

4. Experimental Set-up

An experimental facility has been built to study the performance of air to water heat pumps. The unit is fully equipped with instruments and provides reliable measurements of the system thermal performance. The acquired experimental data has been used to validate the numerical predictions conducted with the numerical model previously presented. A schematic diagram of the experimental unit is shown in Figure 3.

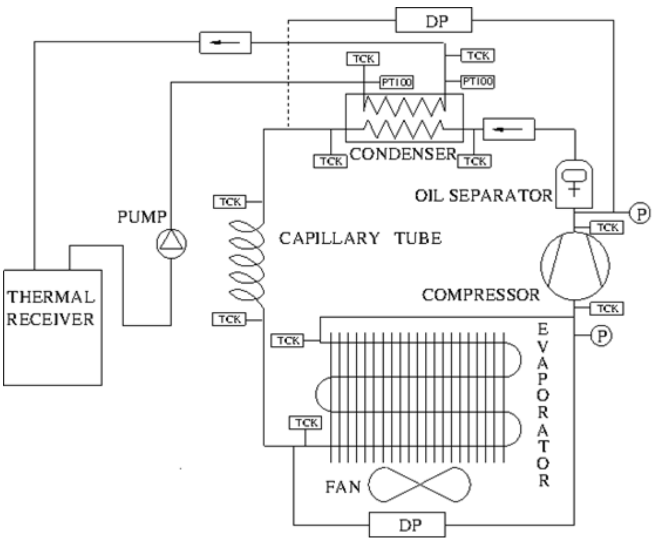


Fig 3. Simplified scheme of the experimental facility.

The unit main loop works with R134a and includes the following elements: a rotary compressor, an oil separator, a plate heat exchanger (condenser), a capillary tube as the expansion device, and a fin-and-tube heat exchanger (evaporator). Several measuring instruments are placed throughout the main loop path: type K thermocouples and/or PT100 thermoresistances to measure the refrigerant inlet and outlet temperatures of each element, a Coriolis mass flow meter located after the oil separator, two pressure transducers and two differential pressure sensors to measure the absolute high and low system pressures and the most relevant pressure drops, respectively.

The condenser secondary circuit works with water and includes the following elements: a water tank and a pump to drive the fluid. It also includes both K thermocouples and PT-100 thermoresistances at the inlet and outlet positions of the condenser together with a Coriolis mass flow meter located before the water tank.
The evaporator external fluid is air, driven by a fan. The air channel is equipped with a turbine speed sensor together with capacitive sensors of both temperature and relative humidity located at both the entrance and the exit of the evaporator.

All the elements of the experimental facility such as the tank and the connection pipes - except for the compressor - have been fully insulated to reduce heat losses. A commercial insulation has been used for this purpose. The main details of the heat pump components are described as follows:

- Compressor: Rotary R134a; N = 50 Hz; Vcl = 18 cm^3
- Condenser: Plate heat exchanger
- Capillary tube: L = 1.5 m; D = 1.5 mm
- Evaporator: Fin-and-tube heat exchanger, louvered fin, flat tube.

The Agilent 34970A data acquisition and control module was used coupled with a LabView data logging and control software application.

4.1. Experimental tests

A transient experimental test has been carried out in order to monitor the heating process of water. The test was performed in a climatic chamber in order to control the environmental conditions (i.e. the temperature of the chamber was kept constant during the experiments). The testing conditions are detailed in the following list:

- Evaporator air inlet temperature: 20 ºC
- Evaporator air mass flow rate: 0.15 kg/s
- Evaporator air relative humidity: 50 %
- Water initial temperature: 20 ºC
- Water mass flow rate: 8 l/min
- Water storage tank capacity: 16 l

The test process begins with the refrigerant cycle running at steady-state conditions with a constant water temperature of 20 ºC. This starting point is achieved by adding an auxiliary system that cools down the storage tank and compensates the energy gain due to the heat pump. At that point, the auxiliary system is removed so that the water starts to increase its temperature due to the energy received from the heat pump. The test process finishes when the water temperature inside the tank reaches 50 ºC.

5. Numerical Results

5.1 Experimental data vs. numerical prediction

The numerical predictions obtained with the simulation model presented in Sections 2 and 3 are compared against the experimental measurements conducted with the test facility presented in Section 4. The results are plotted in Figure 4 where the evolution of the most relevant refrigerant cycle parameters is shown together with the evolution of the water temperature.

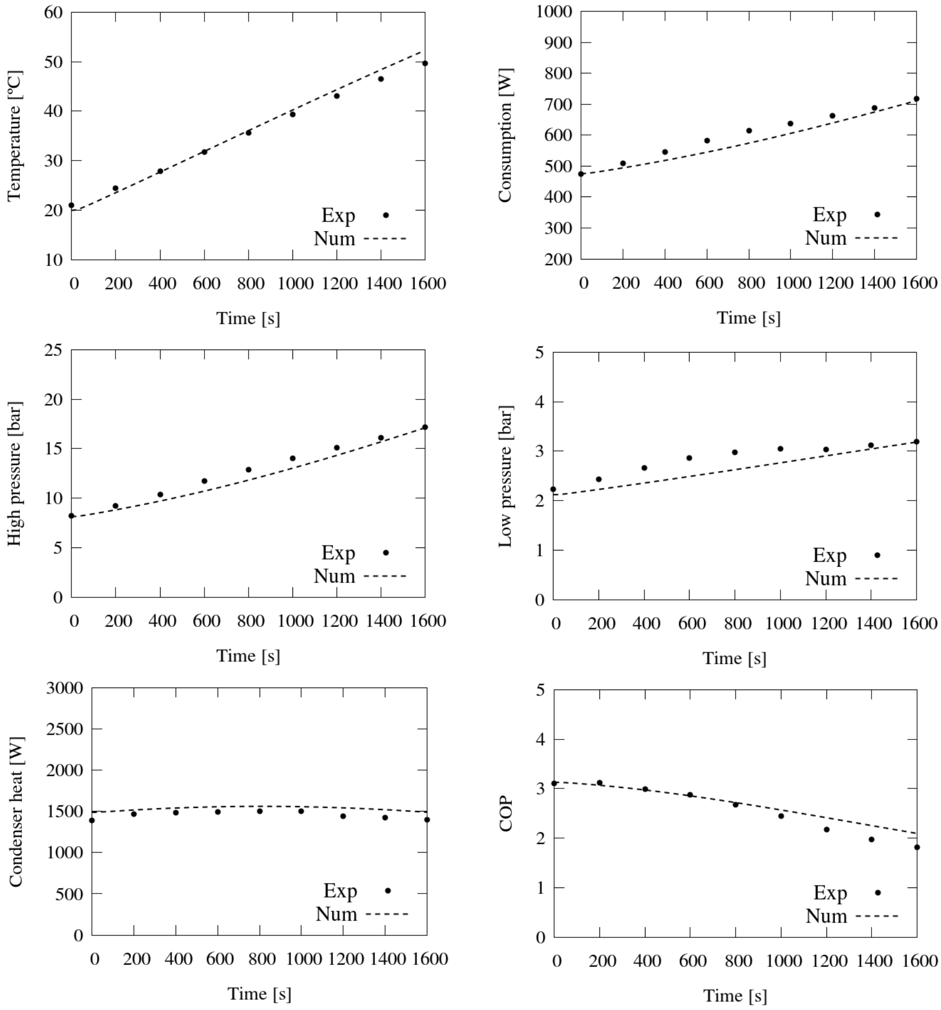


Fig 4. System relevant parameters for different water mass flow rates. Experimental vs. numerical comparison.

The time needed to heat up the water from 20 to 50 ºC is very well predicted with the numerical model as shown in the water temperature evolution graphic. Similarly, the predictions of the most relevant heat pump parameters present very similar trends and tendencies as the experimental data.

From Figure 4 it is observed how, as the water temperature of the tank increases, the system condensing pressure rises in order to have a sufficiently large temperature gradient so that the refrigerant could condensate. This process is constantly adjusted by the capillary tube behavior: an incomplete condensation process would result in a poor performance of the capillary tube, leading to a mass flow rate reduction, which in turn, leads to a better condensation process. As a result, the greater condensing pressure together with the slightly larger evaporating pressure requires a higher compressor energy consumption.

5.2. Numerical study: refrigerant charge

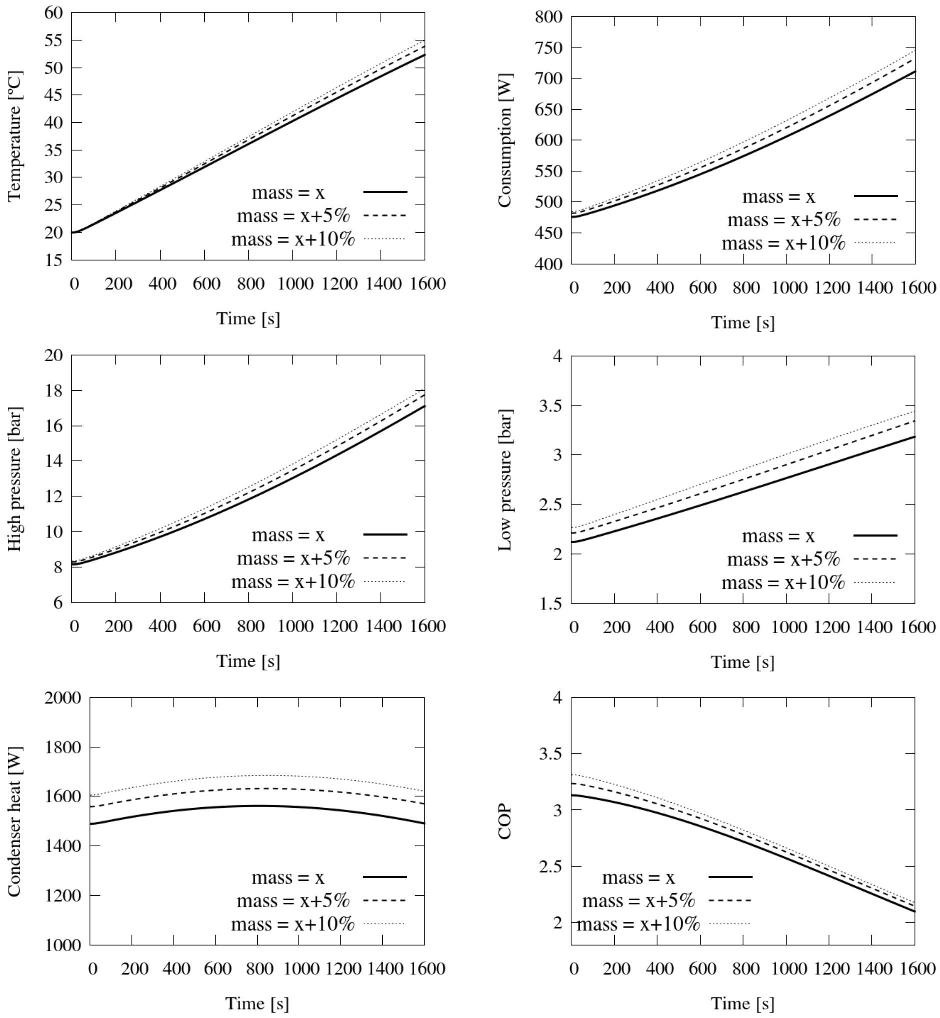
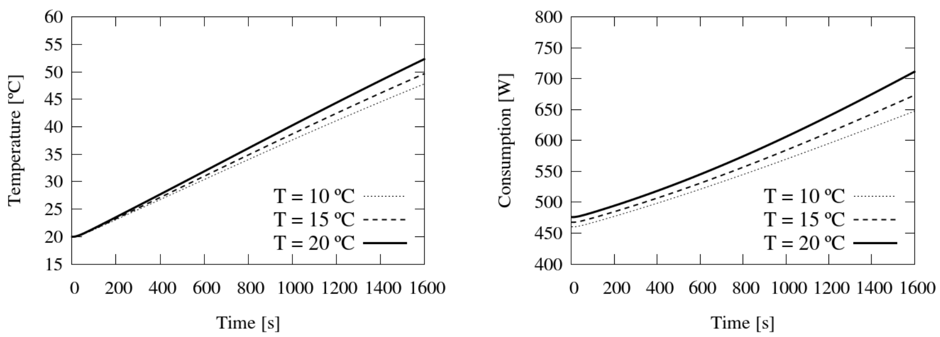


Fig 5. System relevant parameters for different refrigerant charges. Numerical results.

Figure 5 shows how the heating capacity and the system efficiency deteriorate as the refrigerant charge diminishes. Both, the discharge and the suction pressures become lower together with the compressor power consumption and the condenser delivered heat. The resulting COP reduction is due to the varying rate of the two aforementioned parameters. The COP also decreases with time because the sink temperature becomes larger as the water is heated.

5.3. Numerical study: ambient temperature

The effect of the ambient temperature (i.e. the heat pump source temperature) has been numerically studied and the results are plotted in Figure 6. Although power consumption increases as the source temperature rises, the heating process is carried out faster and with a better COP. This behavior is coherent with the maximum theoretical COP definition where its value should increase as the leap between the system source and sink temperatures becomes narrower.



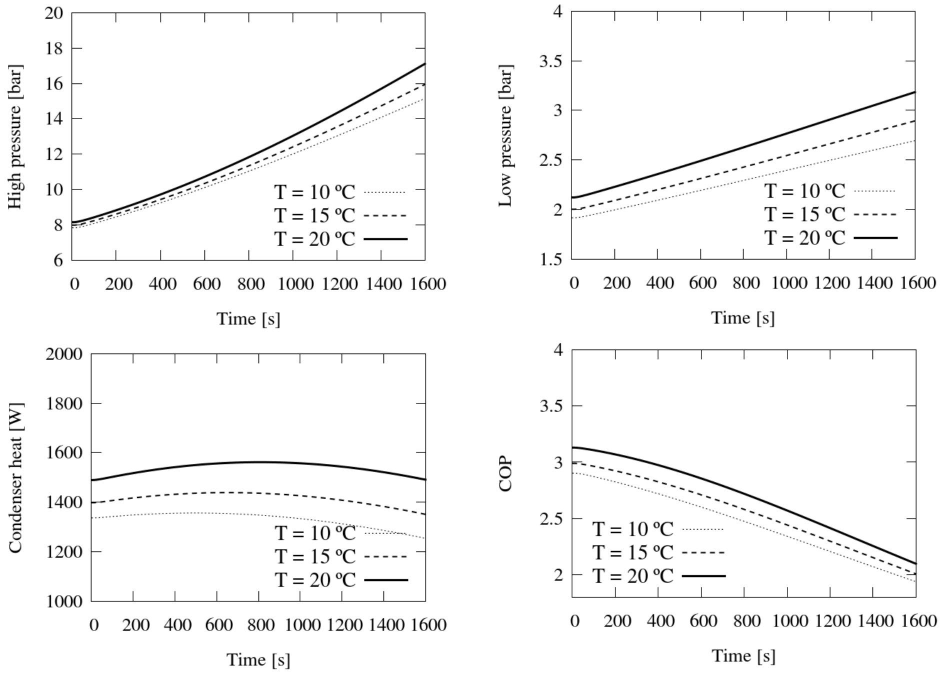
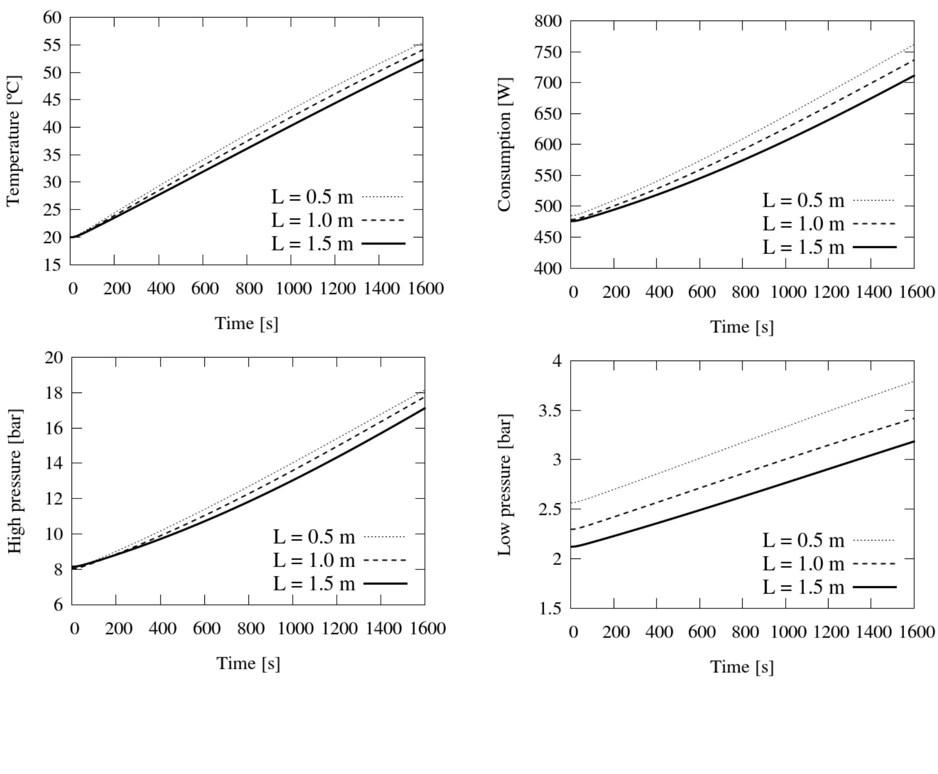


Fig 6. System relevant parameters for different ambient temperatures. Numerical results.

5.3. Numerical study: capillary tube length

The capillary tube length influence on the heating process has also been numerically investigated, the results are depicted in Figure 7. A longer capillary tube produces a lower mass flow rate as resistance to flow becomes greater. The lower refrigerant mass flow rate deteriorates the heat transfer capacity of heat exchangers (lower heat transfer coefficient). The system high pressure decreases so that the temperature gradient between fluids in the condenser is maximized at the new operation point of the refrigerant cycle.



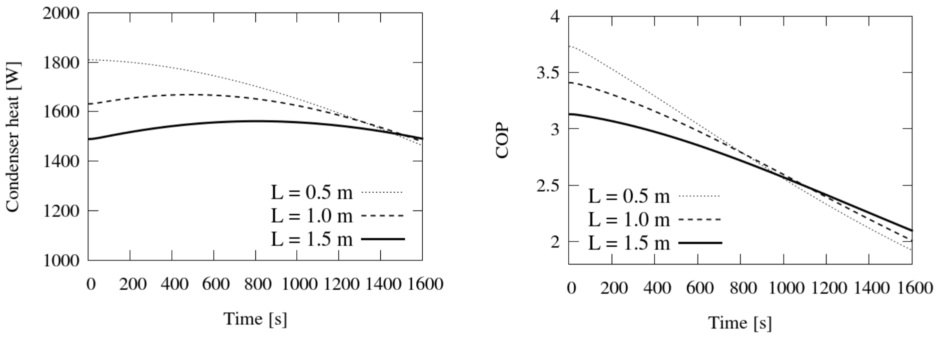


Fig 7. System relevant parameters for different capillary tube lengths. Numerical results.

6. Conclusions

A numerical model has been used to analyze an air to water heat pump at transient conditions. The main feature of the resolution consists in using a pseudo-transient approach (and also with the whole refrigerant cycle modeled with fitted curves) in the simulation of a transient heating process. The run time of simulations is very small (just few seconds for the cases studied in this work where a time step of 1 second was used) and the model accuracy is relatively acceptable when compared to experimental results. Further more, good qualitative agreement is observed as the numerical predictions follow very similar trends to the numerical ones even though the refrigerant cycle is being solved in steady-state conditions.

The numerical parametric studies show that an increase in the ambient temperature, a shortening of the capillary tube, as well as an increase of the refrigerant charge produce similar effects over the heating process and the refrigerant cycle response. For instance, the water is heated faster and although, both the compressor power consumption and the condenser heat increase, the COP value is enhanced.

A detailed study in order to find the most efficient combination of parameters (e.g. capillary tube length, refrigerant charge) at the operating conditions of interest (e.g. ambient temperature, heating need) is now possible with the developed code. In addition, the system component models can be modified, namely, the inclusion of a TEV as the expansion device or the use of a stratified model for the tank reservoir. A control strategy will also be implemented to study the long term operation performance of the system.

Acknowledgement

This work has been developed within the research projects “GREEN-MVC Prototipo de sistema autónomo de desalación de agua por compresión mecánica de vapor a partir de fuentes de energía 100 % renovables” (RTC-2014-2056-5) and “GALIFLOW – Multiscale modelling and Direct Numerical Simulation of gas-liquid multiphase flows in bubble, films and sprays” (ENE2015-70672-P) of the Spanish Government. It has been financially supported by an FPU doctoral grant awarded by the Spanish Government to I. Aizpurua-Udabe (FPU12/06265).

References

[1] Neksa P, Rekstad H, Zakeri R, Schiefloe PA. CO_2-heat pump water heater: characteristics, system design and experimental results. *International Journal of Refrigeration* 1998; 21: 172-179.
[2] Zhang J, Wang RZ, Wu JY. System optimization and experimental research on air source heat pump water heater. *Applied Thermal Engineering* 2007; 27: 1029-1035.
[3] Guo JJ, Wu JY, Wang RZ, LI S. Experimental research and operation optimization of an air-source heat pump water heater. *Applied Energy* 2011; 88: 4128-4138.
[4] Lei M, Zhang H, Wang F, You X. Experimental study of an instantaneous-heating air source heat pump water heater with a temperature stratified water tank . *Advances in Mechanical Engineering* 2016; 8: 1-9.

[5] Jing-Wei P, Hui L, Chun-Lu Z. Performance comparison of air-source heat pump water heater with different expansion devices. *Applied Thermal Engineering* 2016; 99: 1190-1200.
[6] Ding G. Recent developments in simulation techniques for vapour-compression refrigeration systems. *International Journal of Refrigeration* 2007; 30: 1119-1133.
[7] Oliet C, Sadurni A, Ablanque N, Rigola J, 2014. "Numerical and experimental analysis of a heat pump water heater", 11th Heat Pump Conference. Montréal, Canada.
[8] Ndiaye D, Bernier M. Dynamic model of hermetic reciprocating compressor in on-off cycling operation. *Applied Thermal Engineering* 2010; 30: 792-799.
[9] Wenhua L. Simplified steady-state modeling for variable speed compressor. *A.T.Eng.* 2013; 50: 318-326.
[10] García-Valladares O, Pérez-Segarra CD, Rigola J. Numerical Simulation of double-pipe condensers and evaporators. *International Journal of Refrigeration* 2004; 27: 656-670.
[11] Oliet C, Pérez-Segarra CD, Danov S, Oliva A, 2002. "Numerical simulation of dehumidifying fin-and-tube heat exchangers. Model strategies and experimental comparisons". International Refrigeration Engineering Conference. Purdue, U.S.
[12] Oliet C, Pérez-Segarra CD, Danov S, Oliva A. Numerical simulation of dehumidifying fin-and-tube heat exchangers: Semi-analytical modeling and experimental comparisons. *International Journal of Refrigeration* 2007; 30: 1266-1277.
[13] Ablanque N, Oliet C, Rigola J, Oliva A. Numerical simulation of non-adiabatic capillary tubes. Special emphasis on the near-saturation zone. *International Journal of Refrigeration* 2015; 55: 153-167.