International Journal of Heat and Mass Transfer 122 (2018) 574–586

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Numerical study of the thermal behaviour of a water heater tank with a corrugated coil

M.A. Gómez [a,*], J. Collazo [b], J. Porteiro [b], J.L. Míguez [b]

[a] Defense University Center, Spanish Naval Academy, Plaza de España s/n, 36900 Marín, Spain
[b] Industrial Engineering School, University of Vigo, Lagoas-Marcosende s/n, 36310 Vigo, Spain

ARTICLE INFO

Article history:
Received 22 October 2017
Received in revised form 5 January 2018
Accepted 30 January 2018

Keywords:
CFD simulation
Tank
Heat storage
Heat transfer

ABSTRACT

This work proposes a method based on CFD techniques for the study of thermal water heater tanks. The method is based on using the Navier-Stokes equations to calculate the internal movements of the fluid and the transport of energy through the flow. The Boussinesq approximation is applied to solve the fluid movements that occur by natural convection, which represents a great computational savings compared to other methods based on variable densities with very short time steps. Because the tank has a corrugated coil whose mesh poses major computational challenges, a fitting method is used to calculate the heat transfer of the corrugated surface in a smooth coil. The model is applied to a tank with 150 L of water, which is heated by a heat exchanger in the form of an outer jacket. The heat accumulated in the tank is exchanged with the drinking water by means of a corrugated coil located inside the tank. The model is tested by simulating several steady-state continuous consumption experiments and by conducting a transient test of heating and subsequent cooling. For the comparison of results, the data used include the flow rates through the water jacket and the inner coil, the inlet and outlet temperatures of these flows and various temperatures recorded by thermocouples located inside the tank at different heights. The results obtained from the simulations are reasonably close to the experimental observations. The experimental data show an aggressive temperature increase in all thermocouples, except the highest, at the beginning of heating and a softer increase at the end. A similar behavior occurs during cooling, aggressive at the beginning and softer as time progresses. This behavior is also obtained in the model predictions, especially in the lower thermocouples. The greatest discrepancy is found for the highest thermocouple during cooling. The experimental tests show a noticeably slower cooling than the model predictions.

1. Introduction

The supply of domestic hot water represents an important energy expenditure in the residential sector. A domestic hot water installation must have guaranteed availability throughout the period of occupancy of the dwelling and must be sufficient to satisfy the uses of the occupant. There are basically two types of systems for the generation of hot water: instantaneous heating systems and storage heating systems.

Instantaneous heating systems generate hot water on-demand. Their major limitation is that the flow of water that they are able to supply is limited by the installed heat power. This drawback is eliminated in hot water storage systems. In these systems, production and consumption are decoupled in time. This makes energy consumption more efficient because the boilers or heating systems can operate in their zones of optimal performance, minimizing the number of starts and stops. Tanks can be classified into two major types: direct heat and indirect heat. There are many configurations for energy exchange in indirect systems: coil or a series of tubes inside the tank, an external shell and a tube exchanger or an outer jacket that envelops the tank.

The stratification in these devices is key [1,2] because it improves the quality of the thermal energy stored in the tanks for subsequent use. The heated water decreases in density and rises to the top, while the bottom of the tank remains cold. This effect is highly desirable because it greatly increases the storage efficiency of the tank [3].

Ievers and Lin [4] studied the levels of stratification for different geometries using an exergetic analysis. They determined that stratification levels can be improved by increasing the height-to-diameter ratio, decreasing the inlet and outlet flow rates and

* Corresponding author.
E-mail address: miguelgr@uvigo.es (M.A. Gómez).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.01.128
0017-9310/

Nomenclature

D_ω	crossed diffusivity of ω
$\vec{F}$	volumetric force
$\vec{g}$	gravity acceleration
$\tilde{G}$	generation of term of k or ω due to the gradient of average velocities
h	total enthalpy of the fluid
h_{conv}	convective coefficient
k_f	thermal conductivity
k_{ext}	equivalent thermal conductivity in the coil external layer
k_{int}	equivalent thermal conductivity in the coil internal layer
k	turbulent kinetic energy
p	pressure
P_{atm}	atmospheric pressure
Q_v	volumetric heat sources
S	source term
T	temperature
T_0	reference temperature
t	time
v_n	normal component of velocity
v_t	tangential component of velocity
W^v	viscous work
Y_k	dissipation of k or ω due to turbulence
Greek	
β	thermal expansion coefficient
Γ	diffusivity
ρ	density
ρ_0	reference density
$\acute{\tau}$	stress tensor
ϕ	heat dissipation due to viscous forces
ω	specific dissipation rate

positioning the water inlet and outlet of the tank at the ends. Kang [5] studied the thermal mixing in a tank during heating process through experimental tests with different orientation of the heating pipes. Dragsted et al. [6] studied the performance improvement produced by thermal stratification by introducing the hot water through inlets at different heights. Natural convection plays an important role in thermal stratification, this was studied by Andersen and Furbo [7] to analyse the effect of different geometries.

It is very difficult to predict the behaviour of heat exchangers due to the difficulty of knowing the convection coefficients for the different geometries. Several authors [8–10] have used experimental methods to calculate these coefficients. Numerous works [11–13] on heat transfer in heat exchangers focus on improving the exchange process, which results in a reduction in the size of the heat exchanger with consequent cost savings. To improve the efficiency of the exchangers, we seek to promote turbulence near the exchange surface. This is done in passive exchangers using different geometries such as corrugated tubes, rough spiral tubes or helical tubes [14–22]. There are other geometries that yield very good results for heat transfer but cause large pressure losses. Therefore, they are not usually used industrially. Some authors find improvements of approximately 200% in the Nusselt number when using corrugated tubes instead of smooth tubes.

Conventional methods have traditionally been used for the design of heat exchangers, but these processes are time consuming and costly. In the design of heat exchangers, it is sought to maximize heat transfer, minimize pressure losses, increase effectiveness and minimize costs and appliance size [23]. Computational fluid dynamics (CFD) is an efficient tool for the design and optimization of heat exchangers [24–28]. Jiang et al. [24] studied the effects of heat transfer and pressure drop in spiral tubes compared to cylindrical tubes and considered the effect of corrugation depth. Ağra et al. [25] analysed different surface types in corrugated tubes, and the CFD results were validated with experimental data. With this work, they determined parameters such as the friction factor, temperature distributions, and heat flows in the tubes. Ozden and Tari [26] used these CFD techniques to study the temperature distribution and the fluid paths within a tubular heat exchanger. Karmo et al. [27] used CFD to study different distributions of tubes and fins in the design of heat exchangers. Chorak et al. [28] studied the effect of the passage length between corrugations in tubes. Bhutta et al. [29] presented a review of CFD applied to the design of heat exchangers. They analysed the different models used and the level of agreement among the results. Abdelhak et al. [30] used CFD modelling to study the stratification characteristics of a vertical and horizontal tank during the discharge phase. Bouhal et al. [31] simulated a standard water heater tank and studied different configurations of plates and water injections to analyse their effects on thermal stratification. Wang et al. [32] used CFD techniques to investigate the influence of the stratification factor on the storage of hot water to improve the stratification in a previous design of a tank by means of an equalizer that avoids the excessive mixing produced by the entry of cold water breaking the stratification. Experimental validation of the simulations yielded discrepancies of approximately 10% that can increase significantly in particular cases [32].

In this work, we will analyse the behaviour of a water heater storage tank in different operating regimes. In this case, an indirect heat system with two heat exchangers will be studied. The tank works as a storage system for energy, not for drinking water. The heat reaches the tank through an external jacket, and domestic hot water is generated by passing the mains water through a coil located inside the tank. The inner exchanger is constructed of corrugated pipe to improve the heat exchange of drinking water. This construction of the coil complicates the domain discretization, so an equivalent conductivity estimation method has been implemented to carry out the simulation. Two studies are presented: a steady-state case under conditions of continuous production and a transient case that allows us to analyse the process of loading and unloading the hot water tank. The results obtained were compared with data measured in an experimental plant.

2. Experimental equipment

Many hot water storage tanks use an interior coil as the primary exchanger, and the tank directly stores the water to be consumed. There are also Ref. [33] in which an outer jacket exchanger is used as the primary exchanger. In this case, the tank has two exchangers, as shown in Fig. 1. The primary exchanger is an outer jacket surrounding the cylindrical portion of the tank, and the secondary exchanger is a coil constructed of corrugated pipe that is housed inside.

For reasons of maintenance, safety and cleaning, the tank has gas and sludge purge and a port on top for checking and cleaning the inside of the tank. Both the tank and the exchangers are made of AISI 316 stainless steel to ensure durability and water hygiene during use. To improve its thermal behaviour, the tank has polyurethane insulation injected between the body and the shell, except