International Journal of Thermal Sciences 49 (2010) 36–47

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Unsteady natural convection cooling of a water storage tank with an internal gas flue

I. Hmouda [a,*], I. Rodriguez [b], C. Bouden [a], A. Oliva [b]

[a] Research Group: Energy in Buildings and Solar Energy, National School of Engineers of Tunis, BP 37, Le Belvédère 1002 Tunis, Tunisia
[b] Heat and Mass Transfer Technological Centre (CTTC), Polytechnical University of Catalonia (UPC), Colom 11 08222, Terrassa, Barcelona, Spain

ARTICLE INFO

Article history:
Received 5 November 2008
Received in revised form
14 April 2009
Accepted 26 May 2009
Available online 10 July 2009

Keywords:
Natural convection
CFD simulations
Experimental set-up
Storage tanks
Model development

ABSTRACT

The cooling process by natural convection in cylindrical cavities is a phenomenon which takes place in several applications such as solar energy systems. In the present work a storage tank with an internal gas flue is studied experimentally and numerically during its long-term cooling process. The computational domain includes two fluids, i.e. water in the store and air in the chimney, and two external and internal layers of steel separated by polyurethane insulation material. In this paper, the numerical and the experimental analysis of the temperature field inside the tank submitted to an external convection cooling process with a constant convection heat transfer coefficient is presented. The air and the water temperature profiles along the vertical lines are obtained experimentally and numerically, for a cooling period of 90 h. The numerical analysis is carried out using a specific CFD code developed for the present work; an axisymmetric domain has been considered. Finally, a detailed description of the phenomena that occur inside the water part of the domain during the cooling process is also provided.

1. Introduction

A cylindrical tank of liquid water with a gas chimney which is crossing the tank from the bottom to the top is one of the practical configurations that has been widely used as a heating system. In order to enhance the thermal efficiency of such system, its configuration should be optimised. This effective optimisation requires an extensive knowledge of the thermal and fluid dynamic behaviour of the fluid in the tank, and its relationship with the air inside the gas flue and with the tank walls and thermal insulation.

However, on some situations, designs are based in simple mathematical models (e.g. one-dimensional models), that together with expensive trial-and-error experimental set-ups provide the empirical parameters necessary for those models. The main advantages of the one-dimensional models are their simple implementation and their reduced CPU time when long-term simulations are carried out. This kind of models is often used in commercial global codes such as TRNSYS [1]. In these models, the tank is modelled by means of energy balance in a fixed number of fluid volumes and the temperature of each volume is considered uniform [2,3]. Comparisons and results analysis of simulations performed using these models can be found in [2,4,5]. When a new element (such as a gas flue inside the tank) is introduced in the system, the one-dimensional model becomes inadequate. Thus, empirical parameters have to be introduced in order to model the global system.

In this sense, in the last decades, detailed numerical simulations of heat transfer and fluid flow using Computational Fluid Dynamics (CFD) codes have became a powerful tool for studying the complex phenomena taking place in these equipment. Several works describing the unsteady cooling or heating processes of a fluid inside an enclosure can be found in the literature. Among these studies, the work conducted by Hyun [6] can be cited. He studied the transient process of stratification of a fluid in a closed cylinder initially at rest, by imposing a temperature gradient in its lateral wall. His work provided earlier evidence of the mechanism of stratification and the influence of buoyancy forces in the circulation of the fluid. Cotter and Charles [7,8] investigated the transient cooling process of warm crude oil confined in a vertical cylinder. In the study, the influence in the heat losses of the aspect ratio, external heat loss coefficient, as well as the fluid viscosity were analysed. Furthermore, the numerical results were compared with experimental data and a time dependence correlation of Nusselt number for several oil viscosities was defined.

Papanicolaou and Belessiotis [9] studied numerically the natural convection in a vertical cylindrical enclosure heated from the sidewall with constant heat flux and at high Rayleigh numbers. From a numerical point of view, a good agreement between the

* Corresponding author. Tel.: +216 71 874 700; fax: +216 71 872 729.
E-mail address: imen.hmouda@enit.rnu.tn (I. Hmouda).

1290-0729/$ – see front matter © 2009 Elsevier Masson SAS. All rights reserved.
doi:10.1016/j.ijthermalsci.2009.05.011

Nomenclature

c_p	specific heat at constant pressure ($J\,kg^{-1}\,K^{-1}$)
n_{ins}	number of control volumes in insulation
D	tank diameter (m)
n_e	number of control volumes in external wall
D_C	chimney diameter (m)
$\overline{Nu}$	average Nusselt number
$\vec{g}$	acceleration of gravity vector ($m\,s^{-2}$)
p_d	dynamic pressure (Pa)
H	tank height (m)
Pr	Prandlt number
H_c	water store height (m)
S	inner area of the tank (m^2)
h_{ext}	external superficial heat transfer coefficient ($W\,m^{-2}\,K^{-1}$)
t	time (s)
k	thermal conductivity ($W\,m^{-1}\,K^{-1}$)
Δt	time increment (s)
k_e	thermal conductivity of the external solid wall ($W\,m^{-1}\,K^{-1}$)
$\overline{T}_f$	mean fluid temperature (K)
q''	heat flux per unit surface area (W/m^2)
$\overline{T}_w$	mean inner wall temperature (K)
Ra	Rayleigh number
T	temperature (K)
r	radial coordinate (m)
T_{ini}	initial temperature (K)
n_r	number of control volumes in radial direction
T_{amb}	ambient temperature (K)
n_z	number of control volumes in axial direction
T_{ref}	reference temperature $T_{ref} = (T_{ini} + T_{amb})/2$ (K)
n_c	number of control volumes in chimney
v_r	radial velocity ($m\,s^{-1}$)
n_s	number of control volumes in internal solid wall
v_z	axial velocity ($m\,s^{-1}$)
n_w	number of control volumes in water part
$\vec{v}$	velocity vector ($m\,s^{-1}$)
z	z-coordinate (m)

Greek letters

α	thermal diffusivity ($m^2\,s^{-1}$)
δ_{ins}	insulation thickness (m)
β	thermal expansion (K^{-1})
δ_{in}	internal wall thickness (m)
ρ	density ($kg\,m^{-3}$)
δ_{ex}	external wall thickness (m)
μ	dynamic viscosity (Pa s)
$\vec{\tau}$	stress tensor vector ($kg\,m\,s^{-2}$)
Ω	water volume (m^3)
ϵ	relative error (%)
θ	non-dimensional temperature

model predictions and the experimental data for both the laminar and the turbulent regimes was found. Moreover, one of the goals of their work was to study the performance of several turbulence models for their use in the unsteady simulation of cylindrical configurations, shedding some light about the fluid regime present in real working conditions.

Lin and Armfield [10] studied numerically the cooling process of an initially homogeneous fluid in a vertical cylindrical tank by natural convection. In their work, the top and the bottom walls were maintained adiabatic while the lateral wall was suddenly cooled. In a later paper, the same authors [11] analysed the long-term behaviour of cooling a fluid in a cylindrical tank considering two different situations: one with an imposed sidewall temperature and the other with fixed side and bottom wall temperatures while the other walls were maintained adiabatic. Using the numerical results, scaling relations for the mean fluid temperature and Nusselt number have been obtained for each situation. Although their study was in non-dimensional form, because of the imposed boundary conditions, the correlations they found can not be extended to the real situation of a cylindrical tank submitted to the unsteady natural convection due to heat losses to the environment.

The cooling process of a real prototype of storage tank was studied numerically and experimentally by Oliveski et al. [12]. They analysed the transient phenomena taking place due to the heat exchange with the environment. In their work, the influence of the aspect ratio, the volume and the insulation thickness on the thermal performance of the tank was studied. A good agreement between the numerical and experimental results was observed. Furthermore, a correlation for the Nusselt number for each tank volume, function of the previously mentioned parameters, was developed. Adopting the same methodology, the same authors [13], compared the results of a one-dimensional model with those obtained from a detailed model and to the experimental results. They have shown that the simplified model agreed with the experimental results only when several computational artifices were included.

Fernandez-Seara et al. [14], carried out an experimental analysis of the thermal performance of a commercial hot water storage tank under the static mode of operation. The main purpose of their work was to determine the thermal behaviour of the tank in order to characterise its performance, while at the same time determining its overall heat transfer coefficient. For the analysis, they used the experimental results for quantifying the degree of thermal stratification and the exergy efficiency.

More recently, Rodríguez et al. [15] carried out a scaling analysis and numerical simulation of the transient process of cooling-down of an initially homogeneous fluid in a vertical cylinder submitted to natural convection. The correlations obtained can be extrapolated to other situations as they are expressed in term of non-dimensional parameters governing the phenomena that occur inside the tank.

All the mentioned numerical and experimental studies have been carried out in standard cylindrical tanks. However, the results obtained in the aforementioned works can not be extrapolated to the configuration under study in the present paper, i.e. a water storage tank with an internal gas flue. As far as the author's knowledge is concerned, there are no previous works in the literature modelling in detail this configuration of storage tank. In fact, this configuration has only been studied by means of one-dimensional models [16–19]. In these works, the fluid inside the tank and the air inside the gas flue were modelled by means of a multinode approach. For the coupling of both media, the values of the superficial heat transfer coefficients must be supposed. Furthermore, obtaining realistic results required the inclusion of computational artifices (e.g. procedures that examine for each time step the distribution of the temperatures and, in the case of finding hotter water layers under colder ones, artificially mix or interchange the layers of water). Under these circumstances, these models are not capable of describing in detail the transient