Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2014,26(1):104-113
DOI: 10.1016/S1001-6058(14)60012-3
www.sciencedirect.com/
science/journal/l0016058

Simulation of buoyancy-induced turbulent flow from a hot horizontal jet*

El-AMIN M. F., SUN Shuyu, SALAM Amgad
Computational Transport Phenomena Laboratory (CTPL), Division of Physical Sciences and Engineering, King Abdullah University of Science and Technology, Thuwal 23955-6900, Kingdom of Saudi Arabia,
E-mail: mohamed.elamin@kaust.edu.sa

(Received September 14, 2013, Revised November 11, 2013)

Abstract: Experimental visualizations and numerical simulations of a horizontal hot water jet entering cold water into a rectangular storage tank are described. Three different temperature differences and their corresponding Reynolds numbers are considered. Both experimental visualization and numerical computations are carried out for the same flow and thermal conditions. The realizable $k-\varepsilon$ model is used for modeling the turbulent flow while the buoyancy is modeled using the Boussinesq approximation. Polynomial approximations of the water properties are used to compare with the Boussinesq approximation. Numerical solutions are obtained for unsteady flow while pressure, velocity, temperature and turbulence distributions inside the water tank as well as the Froude number are analyzed. The experimental visualizations are performed at intervals of five seconds for all different cases. The simulated results are compared with the visualized results, and both of them show the stratification phenomena and buoyancy force effects due to temperature difference and density variation. After certain times, depending on the case condition, the flow tends to reach a steady state.

Key words: turbulent flow, realizable $k-\varepsilon$ turbulence model, heat transfer, jet, heat storage

Introduction

Thermal stratification in the storage tanks has a strong influence on the thermal performance of solar heating systems. The water entering the solar water storage as a jet causes mixing and destroys the stratifications. CFD-software has been used widely to simulate the components of solar heating systems. Knudsen et al.[1] studied the effect of thermal stratification phenomena in the storage tank by means of experiments and numerical computation. They concluded that the effect of thermal stratification in storage tanks is very important for the thermal performance. Shah and Furbo[2] presented a numerical and experimental analysis of water jets entering a solar storage tank. Three inlet designs with different inlet flow rates were simulated out to illustrate the varying behavior of the thermal conditions in a solar store. Their results showed how the inlet design influences the flow patterns in the tank and how the energy flux in a hot water tank is reduced with a poor inlet design. Velocity and temperature fields around a cold water inlet device of small solar domestic hot water tanks were investigated by Jordan and Furbo[3] using the CFD tool Fluent. The simulation results were compared with temperature measurements inside a commercial storage tank. Knudsen et al.[4] analyzed the flow structure and heat transfer in a vertical mantle heat exchanger. The flow structure and velocities in the inner tank and in the mantle were measured using a particle image velocimetry (PIV) system. In turbulent flow, the Reynolds-averaged Navier-Stokes (RANS) tewchnique is usually adopted in order to make the system amenable to solution. The problem of using RANS approach, however, is that till now, there is no unifying set of equations to model all kinds of turbulent flows and heat transfer scenarios. Therefore, it is important to choose the model which suites the case under investigation and even to calibrate its coefficients in order to fit experimental results. The flow structure of a water jet entering into a rectangular storage tank was investigated by experimental visualizations as well as numerical CFD calculations for the problem have been introduced by El-Amin et al.[5]. El-Amin et al.[6] have pre-

* **Biography:** EL-AMIN M. F. (1971-), Male, Ph. D., Research Scientist

sented the 2-D upward, axisymmetric turbulent confined jet and developed several models to describe flow patterns using the realizable $k-\varepsilon$ turbulence model. Both realizable and RNG $k-\varepsilon$ turbulence models were later calibrated[7]. Several experimental works were conducted to highlight the interesting patterns and the governing parameters pertinent to this kind of flows[8,9]. For example, Arakeri et al.[10] introduced the problem of bifurcation in a buoyant horizontal laminar jet. O'Hern et al.[11] performed experimental work on a turbulent buoyant helium plume. El-Amin and Kanayama[12,13] studied buoyant jet resulting from hydrogen leakage. They developed the similarity formulation and solutions of the centerline quantities such as velocity and concentration. Moreover, El-Amin[14] performed a numerical investigation of a vertical axisymmetric non-Boussinesq buoyant jet resulting from hydrogen leakage in air as an example of injecting a low-density gas into high-density ambient. On the other hand, the mechanics of buoyant jet flows issuing with a general three-dimensional geometry into an unbounded ambient environment with uniform density or stable density stratification and under stagnant or steady sheared current conditions is investigated by Jirka[15,16] extended this work to also encounter plane buoyant jet dynamics resulting from the interaction of multiple buoyant jet effluxes spaced along a diffuser line.

This paper introduces an analysis for three temperature differences with the three corresponding Reynolds numbers of horizontal hot water jets entering a rectangular solar storage tank filled with cold water. In order to investigate the stratification phenomena inside the tank, an experimental visualization was performed for the different cases in sequential time steps, followed by numerical investigations under the same conditions with wide-ranging of analyses for fields of pressure, velocity, temperature and turbulence inside the water store.

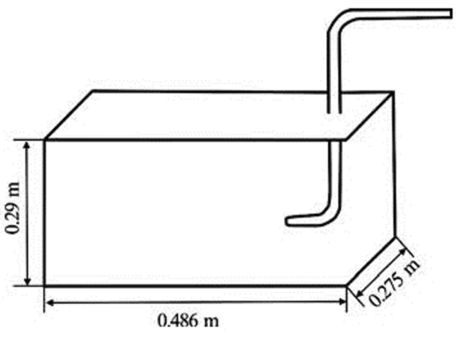

Fig.1. Sketch of the water tank

1. Experimental flow visualizations

Visualization has been used to observe the flow patterns and thermal effects in a storage tank with dimensions in meter $X \times Y \times Z = 0.486$ m×0.275 m×0.29 m. The upper boundary is open with no cover for the tank, thus, the outflow of the tank is on the top. It is worth mentioning that, initially the tank was filled by water and this is useful for describing the upper boundary condition as pressure outlet with avoiding open surface boundary condition. So, this simplification will help in the CFD simulation such that the upper boundary will be considered as outflow boundary condition. In Fig.1 a systematic diagram for the problem is drawn. Red dye is applied to visualize the charging process of the tank. In this experiment the density of the water coloring is slightly different from the water density. So, the amount of change of density will be negligibly small. The colored hot water enters the tank through a horizontal inlet nozzle that has a diameter of 0.007 m. The sketch in Fig.1 indicates that the entering pipe has a converging nozzle, i.e., a nozzle diameter less than the pipe itself. The desired Reynolds number (Re) is determined by flow rate and temperature of the nozzle. In this study we need different flow rates high or low. So, the choice of this design of the nozzle can provide a quite large flow rate that may be not obtained by the normal pipe.

Table 1 Overview of the experiments

Case	Re	T_{cold}	T_{in}	ΔT (K)	t (s)
1	2 100	293.15	293.15	0	95
2	3 200	292.15	313.15	21	85
3	4 400	294.15	333.15	39	105

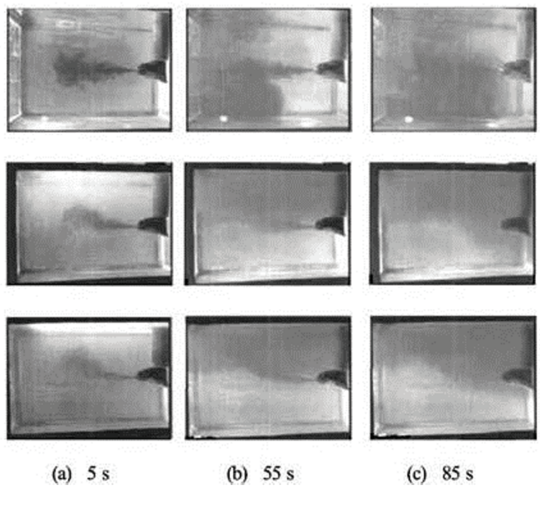

Fig.2 Visualization of flow structures for $\Delta T = 0$ K (1st row), $\Delta T = 20$ K (2nd row) and $\Delta T = 40$ K (3rd row)

The parameters used for the three cases of studies are listed in Table 1, where Re is the Reynolds num-