Development of a Zonal Model

to Analyze a Thermal Storage Tank with an Electric Heater

Seung Hyo Baek,[1] Hyun Min Nam,[1] Myoung Souk Yeo,[2] and Kwang Woo Kim.[2]

[1] Department of Architecture and Architectural Engineering, Graduate School of Seoul National University, Seoul, Korea
[2] Department of Architecture and Architectural Engineering, College of Engineering, Seoul National University, Seoul, Korea

ABSTRACT

A thermal storage tank with an electric heater is a common type of storage for solar domestic hot water system, and is operated to supply remainder energy, which is not provided by the solar collector. In order to evaluate the energy performance of solar domestic hot water systems, it is necessary to analyze water flow and the heat transfer of water contained in the storage tank. In general, it is possible to analyze water flow and the heat transfer of the storage by using CFD. However, using CFD is a very complex and time-consuming process.

In this paper, we propose a zonal model for thermal storage with an electric heater. Computer simulation by using a zonal model is conducted, and we find that the developed analysis model is appropriate to predict fluid motion and the water temperature of the thermal storage tank, based on simulation results.

INTRODUCTION

In Korea, solar energy systems have mainly been applied in buildings for the purpose of supplying hot water. Most solar domestic hot water (SDHW) systems are meant to furnish 20% to 80% of the annual hot water demand; the remainder should be provided by auxiliary heaters, such as electric heater and boilers. In the case of an immersed electric heater installed in a thermal storage tank, thermal stratification in the storage tank will be destroyed, due to the operation of the auxiliary heater. The performance of a SDHW is affected by the temperature distribution in the storage tank. Therefore, accurate modelling of a SDHW system requires an accounting for the thermal stratification in the storage tank.

Dynamic energy simulation programs, such as TRNSYS and EnergyPlus, are in general use for evaluating the thermal performance of SDHW systems. These tools adopt a one-dimensional model to predict water flow and the temperature distribution in a storage tank. The one-dimensional model is simple and efficient in terms of calculation. Therefore this model is particularly suitable for implementation into an overall energy simulation program. However, this model is not appropriate for predicting fluid motion in a storage tank, because water circulation caused by the operation of a heater cannot be evaluated. CFD models can accurately predict fluid motion and water temperature. However, the accuracy of the CFD calculation results strongly depends on the assumptions, such as boundary conditions and simplification. Therefore, the users should have good theoretical knowledge. Furthermore, CFD models require large computational resources and computing time. Consequently, they are not applicable to evaluate the long-term thermal performance of SDHW systems.

In order to predict fluid motion and water temperature of a thermal storage tank with an electric heater, this paper proposes a simplified zonal model. This research is carried out in three main steps. Firstly, existing analysis methods, such as the CFD method and simplified models, are reviewed. Secondly, a simplified zonal model is proposed, that includes a heat balance model and mass balance model. Lastly, a fluid analysis program is developed, and the proposed analysis model is validated through simulation.

ANALYSIS OF THE EXISTING ANALYSIS METHODS

CFD methods

CFD (Computational Fluid Dynamics) methods have been widely used to predict fluid motion and water temperature inside a thermal storage tank. The governing equations for fluid are written as follows:

Mass conservation: $$\nabla \cdot (\rho \vec{u}) = \rho \nabla \cdot \vec{u} = \nabla \cdot \vec{u} = 0 \quad (1)$$

Momentum conservation: $$\frac{D\vec{u}}{Dt} = -\frac{\nabla P}{\rho_0} + v\nabla^2 \vec{u} - \frac{\Delta \rho g}{\rho_0} \quad (2)$$

Energy conservation: $$\frac{\partial T}{\partial t} + \vec{u} \cdot \nabla T = \alpha \nabla^2 T \quad (3)$$

where, ρ is the density (kg/m^3), $\vec{u}$ is the velocity vector (m/s), t is the time (seconds), T is the water temperature (℃), P is the pressure (N/m^2), and g is the gravitational acceleration (m/s^2).

In general, CFD analysis is preferred, because it is more accurate than other analysis methods. However, it requires many input data, and using CFD is a very complex and time-consuming process. CFD is also suitable for the prediction of fluid motion, but not prediction of the energy performance of a solar domestic hot water system.

Simplified fluid motion and water temperature prediction method

One-dimensional stratified model

This model is valid only in the case that the temperature gradient in the radial direction is small, and that stratification in the tank is well formed. It is assumed that the water flow is one-directional, and the water in the tank is divided into multiple nodes of equal volume. Differential equations governing the energy balances on the nodes are solved simultaneously, using the numerical method. Inversion mixing occurs when the node below is warmer than the node above. This buoyancy force causes the nodes to mix. Since inversion mixing occurs very rapidly, the flow rate caused by inversion mixing is assumed to be the maximum value that will provide a stable solution, given the node mass and the time step.

Several one-dimensional methods are suggested in previous research, and are implemented in dynamic energy simulation tools, such as EnergyPlus and TRNSYS.

Plume entrainment model

If the incoming fluid from the heat source is cooler than the fluid at the top of the tank, the hot fluid in the tank will be entrained in the falling plume. Thus, the incoming stream is heated up, and it will fall down to the position in the tank where its density, and therefore temperature, matches that of the tank. The mathematical model of plume entrainment is based on energy and mass balance. The storage tank is modelled as having two separate sections: the plume region and the tank region. The energy balance equation of the stream region is written as:

$$C_p \frac{\partial(\dot{m}_S T_S)}{\partial x} = C_p T_T \frac{\partial \dot{m}_S}{\partial x} \tag{4}$$

The energy equation of the tank region is written as:

$$\rho_f A C_f \frac{\partial T_T}{\partial t} = -C_f \frac{\partial(\dot{m}_T T_T)}{\partial x} + C_f T_T \frac{\partial \dot{m}_T}{\partial x} + kA\frac{\partial^2 T_T}{\partial x^2} - UA_{loss}(T_T - T_{amb}) \tag{5}$$

where, $\dot{m}_S$ is the mass flow rate of the stream (kg/s), T_S is the temperature of the stream (℃), $\dot{m}_T$ is the mass flow rate of the tank region (kg/s), and T_T is the temperature of the tank region (℃).

MATHEMATIC MODEL OF THERMAL STORAGE WITH AN ELECTRIC HEATER

Heat transfer of a thermal storage tank node

In this research we assumed that fluid motion in the thermal storage is two-dimensional. It is considered that the temperature of the upper region of the heater is higher than that when a heater is not installed. The x-axis is divided into two parts.

The storage tank heat transfer model accounts for heater heat input, heat gain from the heat exchanger, heat gain from the heat source, conduction between adjacent nodes, mixing between adjacent nodes, thermal losses to the environment, and heat release to the load. The governing equations for fluid are written as follows:

$$\begin{aligned} m_{i,j} \cdot c_p \cdot \frac{dT_{i,j}}{dt} &= \frac{k \cdot A_x}{dy} \cdot (T_{i+1,j} - T_{i,j}) + \frac{k \cdot A_x}{dy} \cdot (T_{i-1,j} - T_{i,j}) \\ &+ \frac{k \cdot A_y}{dy} \cdot (T_{i,j+1} - T_{i,j}) + \frac{k \cdot A_y}{dy} \cdot (T_{i,j-1} - T_{i,j}) + \dot{m}_{i+1,j} \cdot (T_{i+1,j} - T_{i,j}) \\ &+ \dot{m}_{i-1,j} \cdot (T_{i-1,j} - T_{i,j}) + \dot{m}_{i,j+1} \cdot (T_{i,j+1} - T_{i,j}) + \dot{m}_{i,j-1} \cdot (T_{i,j-1} - T_{i,j}) \\ &+ \dot{m}_{source} \cdot (T_{source} - T_{i,j}) + \dot{m}_{use} \cdot (T_{use} - T_{i,j}) + q_{aux} + q_{HX} + UA_{loss} \cdot (T_{out} - T_{i,j}) \end{aligned} \tag{6}$$

where, m is the mass of a node (kg), C_p is the specific heat (J/kg·℃), k is the thermal conductivity of the water (W/m·K), A is the surface area of the node (m^2), dy is the distance between the centers of nodes (m), T is the temperature of the water (℃), $\dot{m}_S$ is the mass flow rate from node to adjacent node, q_{aux} is the heat transfer from the electric heater (J), q_{HX} is the heat transfer from the heat exchanger

(J), UA_{loss} is the heat loss coefficient of the storage (W/m2·K), T_{out} is the ambient temperature (℃), subscript i is the i^{th} node on the x-axis, and subscript j is the j^{th} node on the y-axis.

The differential equations governing the energy balances on the nodes are solved simultaneously, using the numerical method.

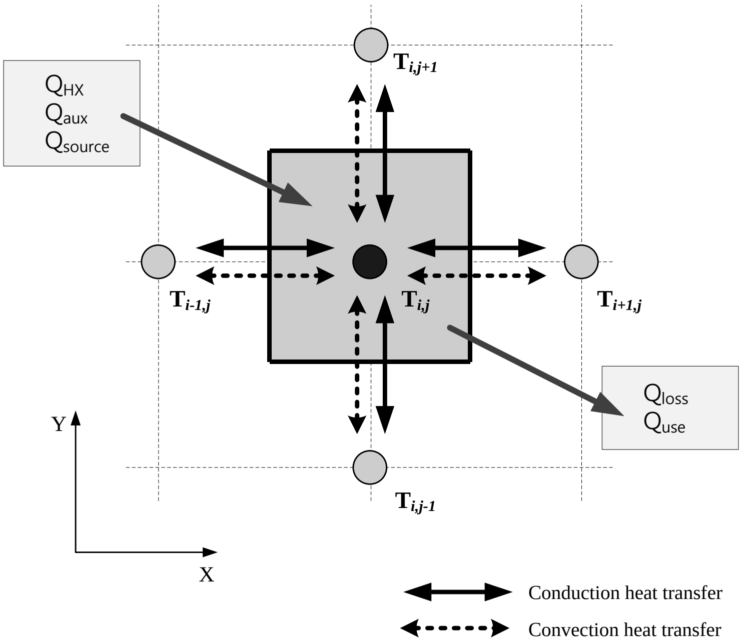

Figure 1. *Heat transfer between the water node and adjacent nodes*

Mass transfer of the thermal storage tank node

Mass flow is introduced into the node, and then some water, equivalent to the amount of incoming water, should be extracted from the node, to maintain mass balance. For each of the nodes, mass flow between vertical adjacent nodes, mass flow between horizontal adjacent nodes, mass flow from the source side, and mass flow from the load side will all contribute to its mass balance. The mass balance equation of each water node is written as below:

$$\Sigma \dot{m}_{in} + \dot{m}_{out} = 0 \tag{7}$$

where, $\dot{m}_{in}$ is the introduced water flow from an adjacent node (kg/s), and $\dot{m}_{in}$ is the extracted water flow to an adjacent node (kg/s).

To calculate the mass flow rate, the momentum conservation and mass conservation of each node should be solved simultaneously, and grid discretization is required. We propose using a simplified method to calculate the mass flow rate of each node. If the temperature of the node below is higher than that of the node above, mass flow from the node below to the node above occurs. It is assumed that the density differences between adjacent nodes are mainly governed by variations in the driving flow.

To simplify the calculation, the natural convection flow was modelled by Boussinesq approximation, with the shape of a water node assumed to be rectangular.

A two-dimensional model is also assumed; the vertical direction is assumed as the y-axis, and the radial direction is assumed as the x-axis. Since mass flow is mainly caused by buoyancy force, it is assumed that buoyant energy is transformed kinetic energy. The buoyancy force is expressed as a function of temperature:

$$\frac{1}{2}\rho \cdot v^2 = \rho \cdot g \cdot \beta \cdot (T - T_0) \cdot dy \tag{8}$$

where, β is the thermal expansion coefficient, and T_0 is the operating temperature (℃).

As a general rule, we express the mass flow rate that crosses a vertical and horizontal border between two current nodes by the following expression:

$$\dot{m} = \rho \cdot Cd \cdot A \cdot v = \rho \cdot Cd \cdot A \cdot \sqrt{2g \cdot \beta \cdot (T - T_0) \cdot dy} \tag{9}$$

where, $\dot{m}$ is the mass flow rate due to the buoyancy force (kg/s), and Cd is the discharge coefficient.

The discharge coefficient Cd is an empirical coefficient that includes viscous effects and the local contraction of streamlines when the flow crosses an opening. This coefficient has to be determined experimentally, by the ratio between the actual mass flow and the theoretical flow.

Analysis model evaluation

An analysis model is developed using Microsoft Visual Studio 2010. The analysis program consists of main two parts: data input and calculation. The thermal property of materials, system design data and weather data is treated in the data input part. The mass balance equations and heat balance equations are solved in the calculation part. The calculation procedure is as follows:

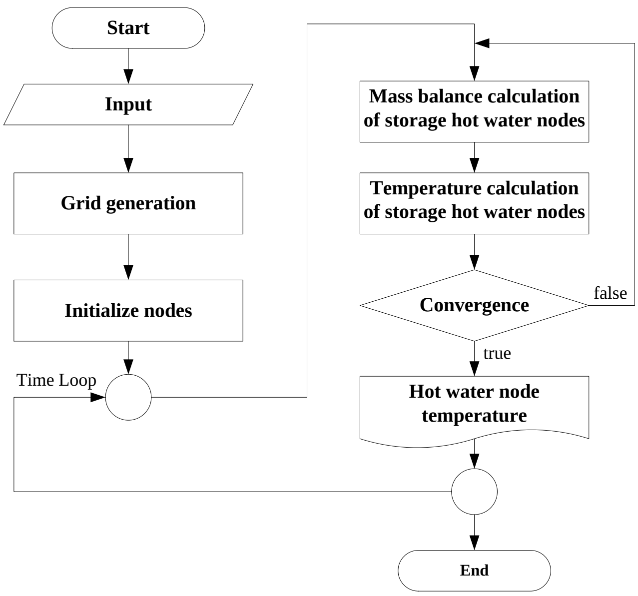

Figure 2*.* *Analysis process of the model*

SIMULATION OF THE THERMAL STORAGE TANK ZONAL MODEL

Simulation Conditions

In order to validate the zonal model for a thermal storage tank, simulation is conducted for a cylindrical thermal storage with an electric heater. Fluid motion and the water temperature distribution are evaluated during heater operation, and without water flow from solar collectors and water flow to buildings. Table 1 summarizes the geometry of the thermal storage tank with the operational conditions.

Table 1. *Input data for simulation*

Parameter	Value
Storage outside diameter	1.2m
Storage height	2.4m
Storage capacity	1,730liter
Electric heater capacity	20kW
Initial temperature of the water	25℃ (bottom ~ 1.2m) 30℃ (1.2m ~ 1.8m) 40℃ (1.8m ~ top)
Electric heater operation	1hour (continuously)
Flow rate from source side	0lpm
Flow rate to building	0lpm

Results

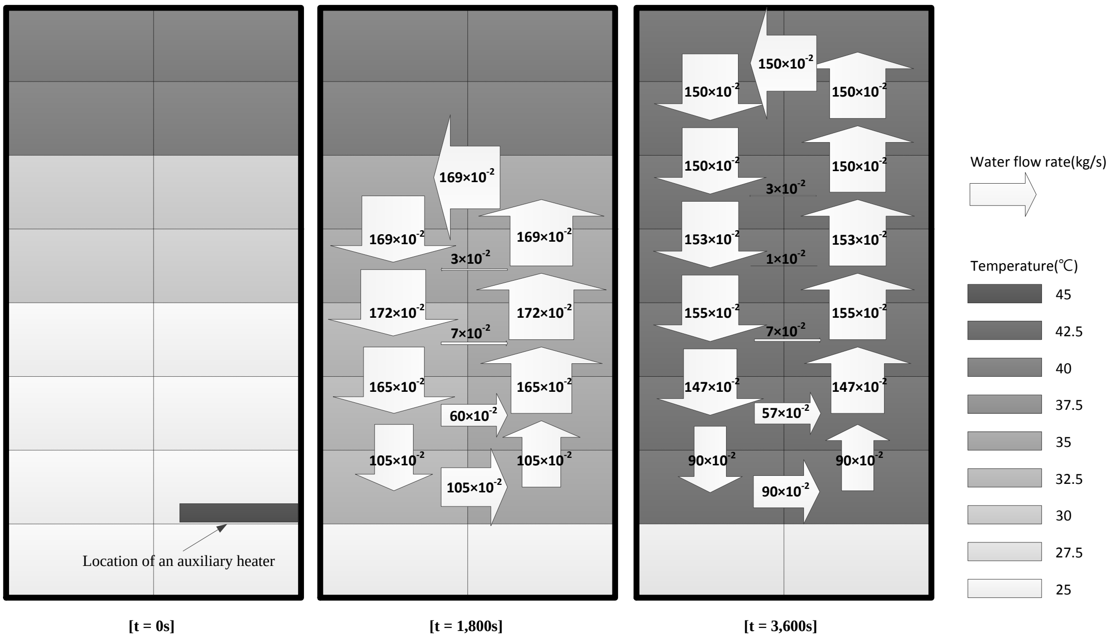

Figure 3. *Simulation result: temperature distribution and water flow rate at time=0s, time=1800s, and time=3600s*

Figure 3 presents the temperature distribution in the thermal storage tank and mass flow rate from node to adjacent node, after 30 minutes and 1 hour. Stratification in the thermal tank has been maintained prior to the electric heater operation. If the heater starts operation, stratification is destroyed, due to the rising of warm water. As can be seen from the water flow rate in the figures, the rising water flow does not penetrate into the warm fluid above, until they have reached the same temperature.

After 1 hour, the temperature of each water node is almost the same, due to mixing. Warm water is circulated in the upper region of the operation of an electric heater, therefore warm water does not reach the lower region of the electric heater.

CONCLUSION

A zonal model method for thermal storage with an electric heater is suggested, to predict fluid motion and the water temperature of the tank. The zonal model method includes a 2-dimensional unsteady state heat transfer algorithm, and mass balance algorithm. In order to analyze water flow due to buoyancy force, a mass flow analysis equation is suggested.

The results from computer simulation using the zonal model show the water flow between water nodes and temperature distribution due to the warm water circulation, under the condition of the auxiliary heater operation. Based on the simulation results, the zonal model method can be incorporated as an energy simulation tool to evaluate the annual thermal performance of SDHW systems. We intend to complete further work to validate the developed analysis model through experiments.

ACKNOWLEDGMENTS

This research was supported by the Basic Science Research Program through the National Research Foundation of Korea (NRF), and funded by the Ministry of Education, Science and Technology (2011-0004093) of the Korean Government.

REFERECNES

ASHRAE. 2007. ASHRAE Handbook HVAC Applications, Atlanta.

J. A. Duffie and W. A. Beckman Solar Engineering of Thermal Processes. John Wiley and Sons, New York. 2006.

E. M. Kleinbach. “Performance study of one-dimensional models for stratified thermal storage tanks”. Master’s thesis, University of Wisconsin-Madison, Madison, Wisconsin, United States, 1990.

K. G. T. Hollands, and M. F. Lightstone, “A review of low-flow stratified- tank solar water heating systems” . Solar Energy, 1989, V.43, No.2, pp. 97-105

N.M. Al-Najem, A.M. Al-Marafie, K.Y. Ezuddin, Analytical and experimental investigation of thermal stratification in storage tanks, Int. J. Energy Res. 1993. 17 77.88.

Mi-Soo Shin et al. “Numerical and experimental study on the design of a stratified thermal storage system”, Applied Thermal Engineering, 2003. V. 24, pp. 17-27.