Proceedings of the ASME 2012 Summer Heat Transfer Conference
HT2012
July 8-12, 2012, Puerto Rico, USA

HT2012-58201

Numerical Simulation of the Temperature Field and the Flow Field inside the Domestic Cooling and Heating Unit's Water Tank

Chao Zhu
College of Energy and Power Engineering
University of Shanghai for Science and Technology
Shanghai, China

Mo Yang*
College of Energy and Power Engineering
University of Shanghai for Science and Technology
Shanghai, China

Yuwen Zhang
Department of Mechanical and Aerospace Engineering
University of Missouri
Columbia, Missouri, USA

Jinlong Wang
College of Energy and Power Engineering
University of Shanghai for Science and Technology
Shanghai, China

ABSTRACT

A water tank of the domestic cooling and heating unit, which has a helix coil, is used to recover the waste heat of the unit. The temperature field and the flow field in the water tank have great effects on the variation of the water temperature in it. In order to obtain the temperature distribution, the flow distribution and the influencing factors, and then obtain the changing situation of the water temperature, the temperature field and the flow field of the water tank are simulated by using Fluent. The results showed that the water temperature will change with different coil decorate. The numerical model which is created by Fluent is appropriate and could be used to improve the layout of the coil in the water tank and speed up heating.

INTRODUCTION

Household multifunctional units are widely used in residential buildings nowadays. In a conventional residential heat pump air conditioner, the waste condensation heat is exhausted to the outdoor surroundings vainly in the summer, which not only engenders energy waste but also yields thermal pollution to the outdoor environment. To a modern urban family, domestic hot water is necessary for daily life and is usually provided by a domestic water heater making use of electricity or gas, which is known for its great energy expenditure. As we know, the condensing temperature of a heat pump is about 70°C, and the temperature of domestic hot water is about 45°C or so. So, proper recovery of waste condensing heat to produce domestic hot water is a kind of efficient means of comprehensive energy utilization [1]. Therefore, the domestic cooling and heating unit with waste heat recovery is of great potential to save energy and decrease thermal pollution.

Shao et al. [2] studied a new scheme of an inverter air cooling heat pump system with domestic hot water. This new design is able to reduce energy consumption by 31.1% and decrease thermal pollution to the environment. Baek et al. [3] designed a heat pump heating system using wastewater as a heat source. The result is that the yearly mean COP of heat pump is about 4.8 and heat pump can supply 100% of hot water load except weekend of winter season. Rousseau [4] proposed an improved system integration methodology aimed at enhancing the potential impact of (heat pump water heater) HPWHs. Jie et al. [5] introduced a multi-functional domestic heat-pump system. The result indicated that the new system can save energy through multi-duties, and can work stably under prolonged operation in regions having mild-winter temperatures. Gorzelnik and Greyvenstein [6] studied the recovery of energy in the heat of compression from air conditioning, refrigeration, or heat-pump equipment. In the study of closed natural convection in a square cavity, many scholars had concluded the results [7-10].

The objective of this paper is to study natural convection in the water tank of the domestic cooling and heating unit with software-FLUENT under different conditions. The study is based on the solution of the steady 2D incompressible Navier-Stokes equations by using finite volume method. The control equation is solved with SIMPLE algorithm. The grids are structured non-uniform staggered under Cartesian coordinate system, and the turbulence model chooses the RNG k-ε model. The results discussed in this paper include the velocity and temperature distribution in the water tank, the velocity vector inside. Finally the performance of the tank with different arrangement coil is discussed.

PROBLEM AND MATHEMATICAL MODEL

Problem description

The 2D section graph of the domestic cooling and heating unit's water tank is shown in Fig.1. The water tank includes 2 import and 2 export, which has internal spiral condensing coil. The height of the tank is 800mm, and the bottom radius is 250mm. The internal condensing coil diameter is 8mm. The Coil spacing is 30mm. The distance between Coil and the underside is 50mm.

Governing equations

The study is based on the solution of the steady 2D incompressible Navier-Stokes equations by using finite volume method. The control equation is solved with SIMPLE algorithm. The grids are structured non-uniform staggered under Cartesian coordinate system. Because the Rayleigh is about 10^7, the flow model is turbulence. The turbulence model chooses the RNG k-ε model.

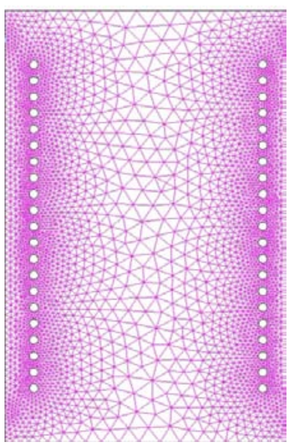

Fig. 1 The water tank's 2D section graph

As the research is convenient, this paper made the assumptions:

(1) The 3-d water tank is simplified as 2-D shown in the Fig. 1. The spiral diameter of the coil is larger than the coil spacing, and the coil geometry is very complex, so the tank is simplified into a 2-d model.
(2) Assuming the wall thickness of the coil as 0.
(3) Assuming the density of water is constant.
(4) Import and export of the water tank closed during heating.
(5) Ignoring water tank walls leak heat.
(6) Water is treated as incompressible fluid, and the flow model is turbulence.
(7) During the heating, all the refrigerant which flow through the coil are two phase. Those can be shown general equation, which are expressed as:

$$\frac{\partial}{\partial x}(\rho u\phi)+\frac{\partial}{\partial y}(\rho u\phi)=\frac{\partial}{\partial x}(\Gamma_{\phi}\frac{\partial \phi}{\partial x})+\frac{\partial}{\partial y}(\Gamma_{\phi}\frac{\partial \phi}{\partial y})+S_{\phi}$$

$RNGk-\varepsilon$ model equations：

(8)

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}(\alpha_k \mu_{eff}\frac{\partial k}{\partial x_j})$$
$$+G_k+G_b-\rho\varepsilon-Y_M+S_k$$
$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_j}(\alpha_\varepsilon \mu_{eff}\frac{\partial \varepsilon}{\partial x_j})$$
$$+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}-R_\varepsilon+S_\varepsilon$$

The thermophysical properties of water are

T(k)	c_p(kJ/kg·K)	λ(W/m·K)	ρ(kg/m³) ×10^{-3}	ν (m²/s) ×10^6
288	4.186	0.591	0.999	1.132
298	4.180	0.608	0.997	0.889
308	4.178	0.622	0.994	0.721
318	4.180	0.635	0.990	0.601
328	4.183	0.646	0.986	0.511
338	4.187	0.655	0.981	0.442

The parameters for turbulent models are:

c_{mu}	C1-Epsilon	C_2-Epsilon	Wall Prandtl Number
0.0845	1.42	1.68	0.85

Boundary conditions

The computational domain is the area which includes the fluid and the tank's boundary. The grid is structured trig mesh. The condition of water wall uses no-slip condition. The flow of outlet can be seen fully developed flow. The pressure is standard atmospheric pressure. The import of the tank uses velocity-inlet.

RESULTS AND DISCUSSION

In this paper, the simulation results of the temperature and the velocity are shown. The initial temperature is 280k, 290k, 300k. The velocity of import is 0.05m/s, 0.1m/s, 0.5m/s.

Velocity distribution under different initial temperature

(1) The velocity of import is 0.05m/s.

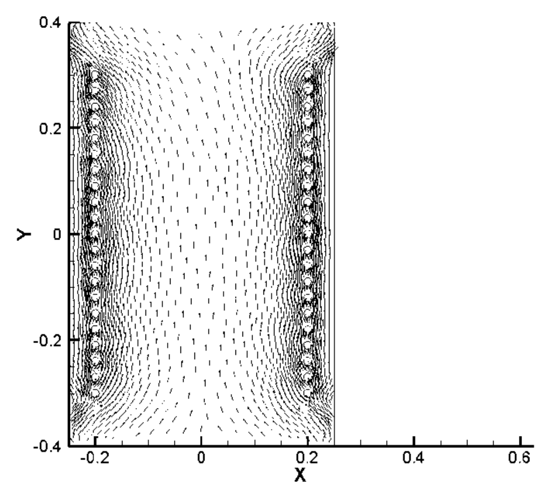

Fig. 2 Deflection of 280K

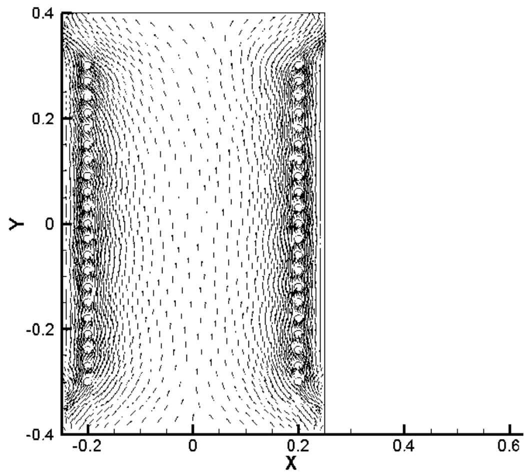

Fig. 3 Deflection of 290K

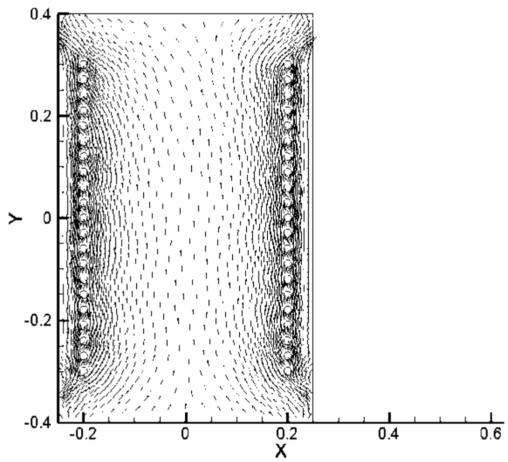

Fig. 4 Deflection of 300K

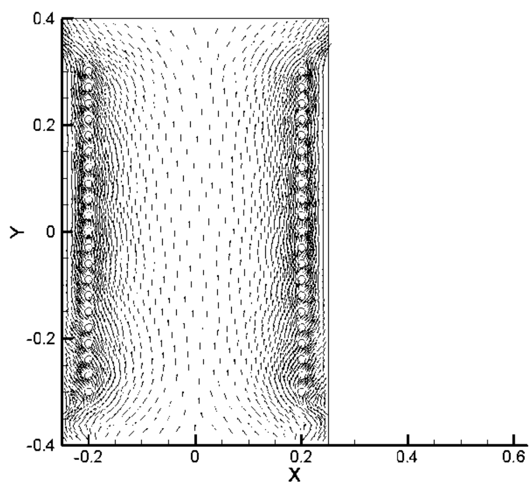

Fig. 5 Deflection of 280k

(2) The velocity of import is 0.1m/s.

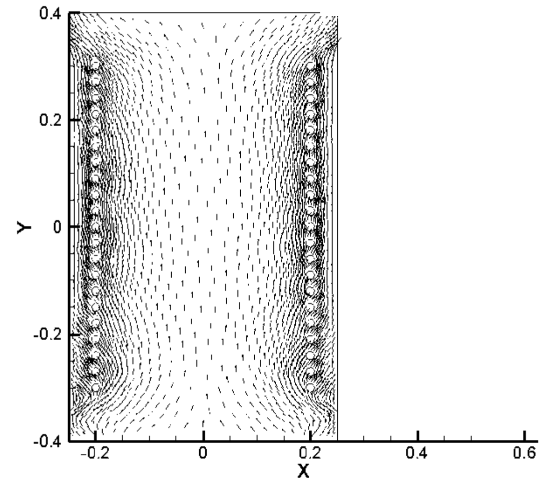

Fig. 6 Deflection of 290K

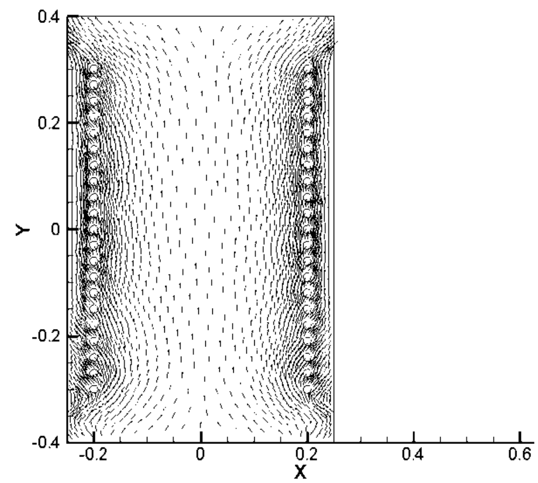

Fig. 7 Deflection of 300K

(3) The velocity of import is 0.5m/s.

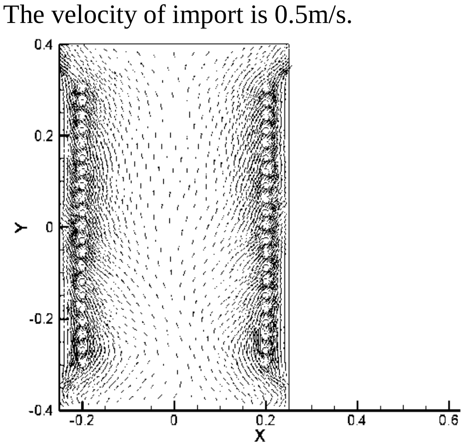

Fig. 8 Deflection of 280k

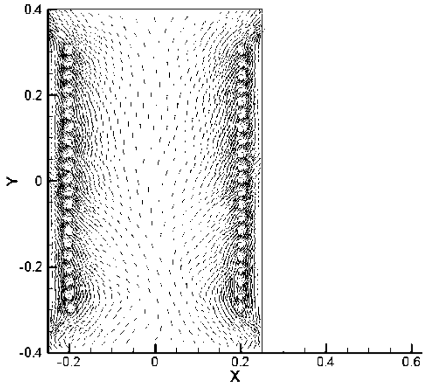

Fig. 9 Deflection of 290K

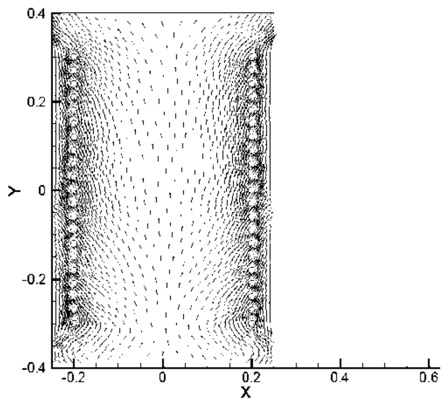

Fig. 10 Deflection of 300K

It shows velocity distribution of different initial temperature and different import velocity from fig.2 to fig.10. When the velocity of import is constant, and the initial temperature from 280k to 300k, the flow of the inside tank is more intense. When the initial temperature is constant, and the velocity of import from 0.05m/s to 0.5m/s, the flow of the inside tank is more intense.

Temperature distribution under different initial temperature

(1) The velocity of import is 0.05m/s.

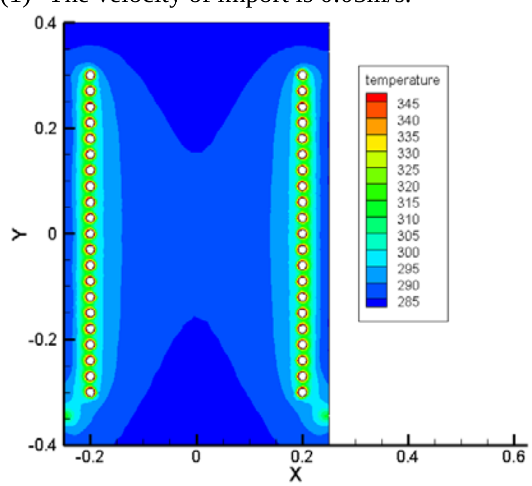

Fig. 11 Deflection of 280K

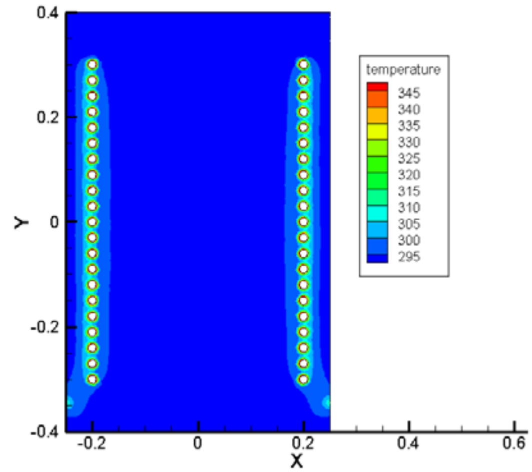

Fig. 12 Deflection of 290K

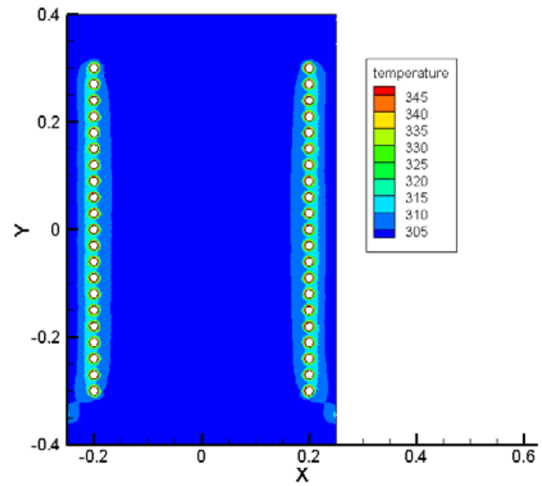

Fig. 13 Deflection of 300k

(2) The velocity of import is 0.1m/s.

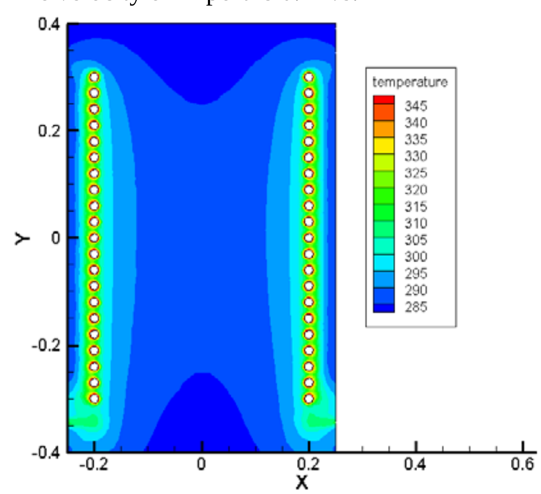

Fig. 14 Deflection of 280k

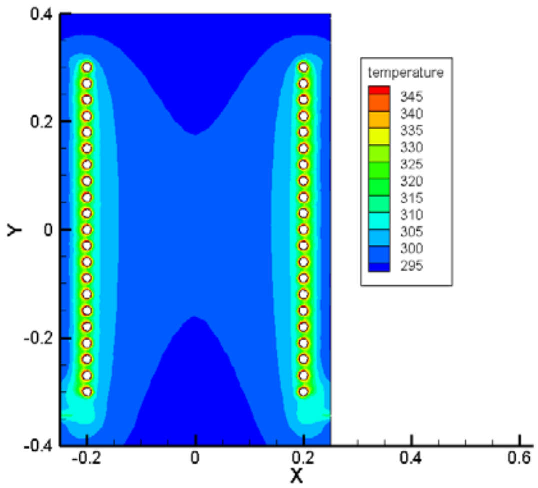

Fig. 15 Deflection of 290K

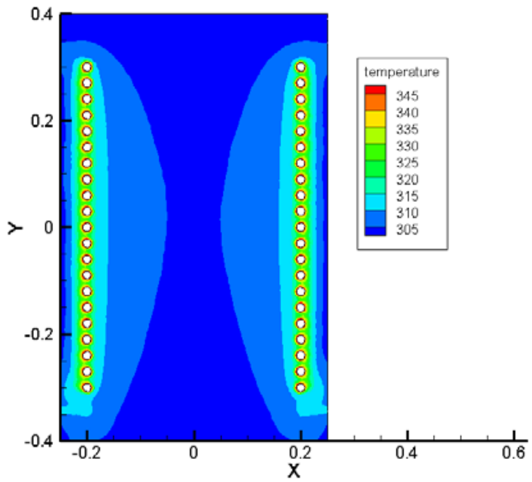

Fig. 16 Deflection of 300k

(3) The velocity of import is 0.5m/s.

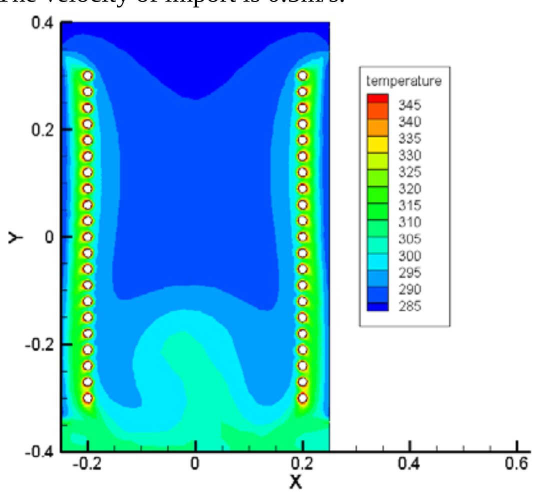

Fig. 17 Deflection of 280k

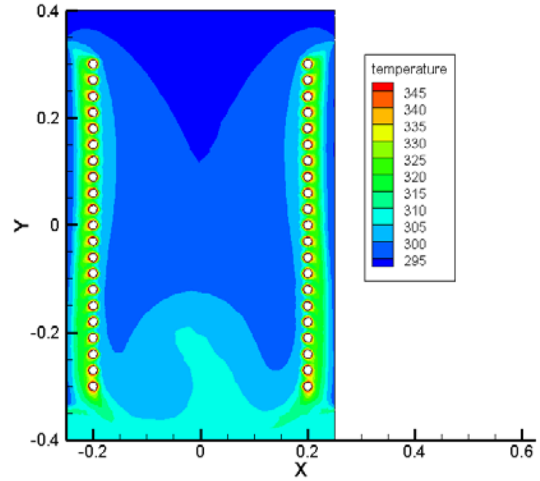

Fig. 18 Deflection of 290k

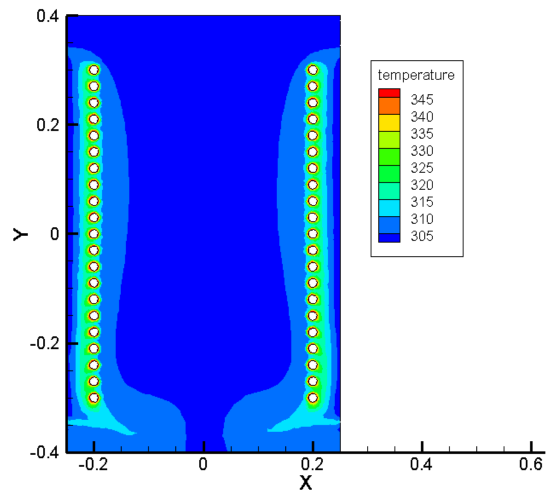

Fig. 19 Deflection of 300k

It shows temperature distribution of different initial temperature and different import velocity from fig.11 to fig.19. When the velocity of import is constant, and the initial temperature from 280k to 300k, the flow of the inside tank is more intense. When the initial temperature is constant, and the velocity of import from 0.05m/s to 0.5m/s, the the flow of the inside tank is more intense.

CONCLUSIONS

When the initial temperature and the import velocity change, the velocity and temperature deflection of the inside water tank are changing too. When the initial temperature drops and the import velocity increases, the flow is more intense. The initial temperature drop and the import velocity increase are conducive to Rayleigh and Reynolds. So it can be seen that the results is reliability.

ACKNOWLEDGMENTS

The present work is supported by the National Natural Science Foundation of China (No. 51076105) and the National Natural Science Foundation of China (No. 51129602).

REFERENCES

[1] Mason, R.S., Bierenbaum, H.S, 1977, "Energy conservation through heat recovery water heating," ASHRAE J, 19, pp. 36-40.

[2] Shao, S.Q., Shi, W.X., Li, X.T., Ma, J., 2004, "A new inverter heat pump operated all year round with domestic hot water," Energy Conversion and Management, 45, pp. 2255-2268.

[3] Baek, N.C., Shin, U.C., Yoon, J.H., 2005, "A study on the design and analysis of a heat pump heating system using wastewater as a heat source," Solar Energy, 78, pp. 427-440.

[4] Rousseau, P.G., and Greyvenstein, G.P., 2000, "Enhancing the impact of heat-pump water heaters in the South African commercial sector," Energy, 25, pp. 51-70.

[5] Jie, J., Gang, P., Tin-tai, C., Wei, H., Ai-feng Z., 2005, "Performance of multi-functional domestic heat-pump system," Applied Energy, 80, pp. 307-326.

[6] Gorzelnik, E.F., 1977, "Heat water with your air-conditioner," Electrical World, 188, pp. 54-55.

[7] Vahl, D., Davis, G., 1983, "Natural convection of air in a square cavity," Int J Numer Methods Fluids, 3, pp. 249-264.

[8] Saitoh, T., Hirose, K., 1989, "High accuracy benchmark solutions to natural convections in a square cavity," Comput Mech, 4, pp. 417-427.

[9] Hortman, M., Peric, M., 1990, "Finite volume multigrid predition of laminar natural convection: benchmark solutions," Int J Numer Methods Fluids, 11, pp. 189-207.

[10] Barakos, G., Mitsoulis, E., 1994, "Natural convection flow in a square cavity revisited: laminar and turbulent models with wall function," Int J Numer Methods Fluids, 18, pp. 695-719.

