THE HEAT AND FLUID FLOW ANALYSIS FOR WATER HEATER

by

Chien-Nan LIN a*, Cheng-Chi WANG a, and Yi-Pin KUO b

a Department of Mechanical Engineering, Far East University, Hsin-Shih, Tainan, Taiwan
b Department of Electronic Engineering, Far East University, Hsin-Shih, Tainan, Taiwan

In this paper, the heat transfer and fluid flow are studied for the water heater of RV cars, in which the hot water is heated by the combustion energy of liquefied petroleum gases. Three types of combustion tubes are performed in this investigation, which are circular tube, elliptic tube and elliptic tube with screwed wire inserted. The heat transfer performances of numerical simulation results are compared with those of the experimental works: they are in good trend agreement. The elliptic combustion tube performs better than the circular one, which indicates the average 7% energy saving for the elliptic combustion tube and 12% energy saving for the elliptic combustion tube with screwed wire under static heating.

Key words: numerical simulation, water heater, combustion

Introduction

Water heaters are widely used in various industrial and domestic applications for hot water. The heating energy may come from different kinds of resources, such as petroleum fuel, natural gas, electrical power or solar energy. And combustion is still the convenient way to produce thermal energy for outdoor uses. The recreation vehicle is one example, in which the water heater adopts liquefied petroleum gases or liquefied natural gas as fuel. The combustion reaction occurs in the combustion chamber, which is enclosed by the water tank filling with cold water. The thermal energy is then transferred from the produced gases through the chamber walls to the cold water within the tank. The structure of the water heater is shown in fig. 1. The fuel is injected from a nozzle and flows through a Ventura tube to induce the first inlet fresh air. The mixed gases are led to the combustion chamber through the gas tube, and the second inlet air is also supplemented near the outlet of the mixed gas tube to make up the proper air to fuel ratio.

It is difficult to measure the temperatures and gases flow distributions in the combustion chamber by experimental work due to the high temperatures, which are normally above 2000 K. Therefore, the numerous numerical simulations were widely used to analyze the heat and fluid flow in these years [1-4]. Saade and Kozinski’s research [5] presented and