Available online at www.sciencedirect.com
ScienceDirect
Renewable Energy 33 (2008) 383–387
RENEWABLE ENERGY
www.elsevier.com/locate/renene

# Validation of CFD simulation for flat plate solar energy collector

Mohamed Selmi, Mohammed J. Al-Khawaja*, Abdulhamid Marafia

*Department of Mechanical Engineering, University of Qatar, P.O. Box 2713, Doha, State of Qatar, Qatar*

Received 30 December 2006; accepted 3 February 2007
Available online 26 April 2007

**Abstract**

The problem of flat plate solar energy collector with water flow is simulated and analyzed using computational fluid dynamics (CFD) software. The considered case includes the CFD modeling of solar irradiation and the modes of mixed convection and radiation heat transfer between tube surface, glass cover, side walls, and insulating base of the collector as well as the mixed convective heat transfer in the circulating water inside the tube and conduction between the base and tube material. The collector performance, after obtaining 3-D temperature distribution over the volume of the body of the collector, was studied with and without circulating water flow. An experimental model was built and experiments were performed to validate the CFD model. The outlet temperature of water is compared with experimental results and there is a good agreement.


*Keywords:* Solar energy; Solar radiation; Solar collector; CFD simulation

## 1. Introduction

The solar energy collection as a renewable energy topic has been the primary interests of many engineers and researchers for the last two centuries due to its wide applications such as domestic water heating systems. Today, solar water heating systems are being used for single family houses, apartment buildings, schools, car washes, hospitals, restaurants, agricultural farms and different industries. Solar water heating can reduce domestic water heating costs by as much as 70%. Owners of these buildings have found that solar water heating systems are cost-effective in meeting their hot water needs all over the year. A more intensive attention was given to this topic from 1970s of the last century, particularly, when the worldwide crisis of 1973 has taken place. Since then, the efficiency of solar heating systems and collectors has improved. The efficiencies can be attributed to the use of low iron, tempered glass for glazing (low-iron glass allows the transmission of more solar energy than conventional glass), improved insulation, and the development of durable selective coatings. Thus due to its importance, a flat plate solar collector with and without cover glass was analyzed using computational fluid dynamics (CFD) software and simulated without water flow in Refs. [1,2]. Here, the water flow is added and the problem is examined both computationally using CFD software as well as experimentally and the results are compared to each other for validation purposes. The CFD package is from Computational Fluid Dynamics Research Corporation (CFDRC) and has been successfully used to simulate water currents and heat transfer inside a water-intake lagoon along a coastal zone near a power plant in Doha, Qatar by Raouf and Selmi [3]. The software proved to be powerful and flexible in modeling a wide range of practical problems. More on the CFDRC package is found in Ref. [4]. The CFD analysis of the flow and heat transfer in flat plate solar collectors is computationally quite difficult and the number of research works on this subject is quite low. We refer the reader to the paper by Morrision et al. [5] for further details.

## 2. Experimental model

A simple model for the solar collector was built (See Fig. 1), which is used for implementing the required experiment, that can be helpful in gaining real outputs that

*Corresponding author. Tel.: +974 4852109; fax: +974 4670421.
*E-mail address:* khawaja@qu.edu.qa (M.J. Al-Khawaja).

0960-1481/$ - see front matter 
doi:10.1016/j.renene.2007.02.003

**Nomenclature**

| | |
|---|---|
| $h$ | heat transfer coefficient, W/m$^2$ °C |
| $n$ | number of iterations |
| $p$ | pressure, N/m$^2$ |
| $q$ | solar heat flux, W/m$^2$ |
| $R$ | residuals |
| $T$ | temperature, °C and K |
| $T_{in}$ | water inlet temperature, °C and K |
| $T_{out,\,exp}$ | experimental water outlet temperature, °C and K |
| $T_{out,\,sim}$ | simulation water outlet temperature, °C and K |
| $u$ | velocity component in $x$-direction, m/s |
| $v$ | velocity component in $y$-direction, m/s |
| $w$ | velocity component in $z$-direction, m/s |

*Greek letters*

| | |
|---|---|
| $\rho$ | fluid density, kg/m$^3$ |

are compared to those simulated by the CFD package described later in this work. Due to the high insulation efficiency and easy forming, the housing frame was made of wood. The project model consists of a wooden box of 1.5 m long; 166 mm wide and 70 mm high, covered with a transparent glass. The box contains an aluminum plate used as an absorber, fixed to it from the top a copper pipe of $\frac{1}{2}$-in diameter, and a polyurethane insulation from the bottom. Both plate and pipe are fixed at a certain level inside the wooden box. The inner walls of the box, absorber plate, and pipe are covered by black mate paint, while the outside box is painted with white color. The pipe extends slightly on both sides outside the housing frame, in order to connect the water inlet and outlet fittings easily.

A number of thermocouples are attached to the absorber plate, pipe at some selected points, and outside of the solar collector, to measure the collector inside, ambient, water inlet and outlet temperatures. The model pipe is connected to water source through manual control valve at one end, and a drain hose at the other. The water flow is controlled by means of a manual control valve, and a calibrated pot. The project model is oriented to face the sun rays normally, and tilted from time to time to keep tracing the sun position for most of the experiment time, thus the maximum energy can be gathered.

There are two types of experiments which are implemented by the built up project model: The first experiment is performed by still water, i.e. no water flow, and the second is performed by providing a controlled flow of water, while all other parameters are kept the same.

The following measurements are required: (1) water inlet temperature, (2) water outlet temperature, (3) absorber plate temperature, (4) pipe temperature, (5) ambient temperature, (6) solar radiation intensity, and (7) water flow rate. Measurements starts, upon complete set up of all components and steady-state conditions are achieved. The readings are automatically monitored and plotted, by means of a set of measuring devices connected to the absorber plate and pipe, through the thermocouples.

## 3. CFD simulation

The CFD simulations are done by the CFD package from the CFDRC. To gain simulated results using CFD software, a simulation procedure has to be followed. The procedure requires setting the boundary and volume conditions of the simulated module. Assuming that the collector is a simple flat plate solar collector, boundaries should have both convection and radiation heat transfer mechanisms, except for the face and bottom surfaces. Aluminum plate is set to have both emissivity and absorptivity of 1, since it is painted with black mate paint. In addition, aluminum plate has a density of 2770 kg/m$^3$ and conductivity of 177 W/°C m. The copper pipe has the same radiation characteristics to be an ideal emissive and absorptive surface; it has a density of 8800 kg/m$^3$ and conductivity of 401 W/°C m. All these properties, beside others for other materials are given in Table 1.

The procedure starts by drawing the module first, in accordance to the proposed geometries (see Fig. 2). This stage is done with the CFDRC geometric modeling module, CFD-Geom, where module volumes, boundaries, interfaces, and grids are created. The combination of these geometries forms the module body; but without the status

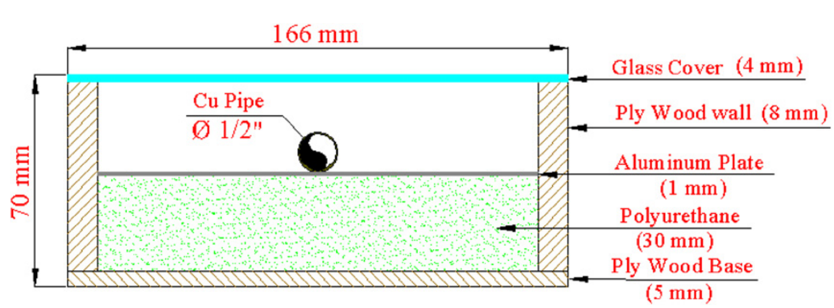


Fig. 1. Model geometry.