Numerical investigation of flow and heat transfer performance of solar water heater with elliptical collector tube

Kaichun Li\textsuperscript{a}, Tong Li\textsuperscript{b}, Hanzhong Tao\textsuperscript{a,b, *}, Yuanxue Pan\textsuperscript{a}, Jingshan Zhang\textsuperscript{a}

\textsuperscript{a}Sunrain Group Co., Ltd, Lianyungang, 222243, China
\textsuperscript{b}College of Energy, Nanjing Tech University, Nanjing 211816, China

Abstract

The flow and heat transfer performance of solar water heaters for different initial temperatures (273K, 283K, 293K, 303K and 313K) with elliptical collector tubes are evaluated using numerical simulation. The predicted results match fairly well with experimental data. Results indicate that, the temperature distributions of all the tube cross sections are alike, but the velocity profiles of them are much dissimilar. The fluid velocity near the wall decrease with decrease in the ratio of the cross section major and minor axis, which induces the reduction in the circulation rate through the collector tubes, which is not conducive to heat transfer. The mean Nusselt number of solar water heater with $b/a=1$ (b and a here are respectively major and minor axis of collector tube cross section) are respectively 59\% and 19\% larger than the solar water heaters with $b/a=0.6$ and 0.8 for the temperature ranging from 273K to 313K.

Keywords: Numerical simulation; Solar water heater; Heat transfer;

1. Introduction

Solar water heater is widely used all over the world due to its simplicity, technological feasibility, economical and commercial viability. Extensive and systematic researches have been carried out over the years to evaluate the performance of the solar water heater [1-4]. Jaisankar et al. [5] experimentally investigated the friction factor and heat transfer characteristics of thermosyphon solar water heater with full length Left-Right twist, twist fitted with rod and spacer at the trailing edge, results showed that the solar water heaters with full length twists are better for heat...
transfer and with lower flow resistance than the other two. Morrison et al. [6] developed a numerical model of the heat transfer and flow inside a single-ended evacuated tube to evaluate the characteristics of water-in-glass evacuated tube solar water heater, results showed that, the natural convection flow rate in the tube is high enough to disturb the tank’s stratification and that the tank temperature strongly affects the circulation flow rate through the tubes. Budihardjo et al. [7] undertake experimental and numerical investigations to develop a correlation for natural circulation flow rate through single-ended water-in-glass evacuated tubes mounted over a diffuse reflector. The circulation flow rate was correlated in terms of solar input, tank temperature, collector inclination and tube aspect ratio. The sensitivity of the flow rate correlation to the variation in circumferential heat flux distribution was also investigated. Yan et al. [8, 9] numerically and experimentally studied the effect of irradiance, tilt angle and the guide plate on the heat transfer performance, based on the results of the study, they proposed many suggestions for optimization of the solar water heater.

With the popularity of wall hanging solar water heater, there will be security threat because of the large weight of the collector. The potential safety hazard may be reduced if the circular collector tube changed into elliptical pipe, which make the weight of the collector tube lower. In this paper, the flow and heat transfer performance of solar water heaters with elliptical collector tubes are evaluated and compared with the normal one using CFD.

**Nomenclature**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( D )</td>
<td>circular tube diameter (m)</td>
</tr>
<tr>
<td>( a )</td>
<td>minor axis of elliptical cross section (m)</td>
</tr>
<tr>
<td>( b )</td>
<td>major axis of elliptical cross section (m)</td>
</tr>
<tr>
<td>( L )</td>
<td>length of collector tube (m)</td>
</tr>
<tr>
<td>( D_i )</td>
<td>inner diameter of tank (m)</td>
</tr>
<tr>
<td>( T )</td>
<td>fluid temperature (K)</td>
</tr>
<tr>
<td>( c_p )</td>
<td>specific heat of fluid (J/kg · K)</td>
</tr>
<tr>
<td>( k )</td>
<td>thermal conductivity (W/m · K)</td>
</tr>
<tr>
<td>( m )</td>
<td>mass flow rate (kg/s)</td>
</tr>
<tr>
<td>( m_i )</td>
<td>inlet temperature of tank (K)</td>
</tr>
<tr>
<td>( m_o )</td>
<td>outlet temperature of tank (K)</td>
</tr>
<tr>
<td>( \rho )</td>
<td>density (kg/m³)</td>
</tr>
<tr>
<td>( \mu )</td>
<td>dynamic viscosity (kg/m · s)</td>
</tr>
</tbody>
</table>

\( h \) heat transfer coefficient (W/m²·K)  
\( A \) heat transfer surface area (m²)  
\( T_w \) average wall temperature (K)  
\( T_b \) average bulk temperature (K)  
\( Nu \) Nusselt number  
\( d \) equivalent diameter of collector tube (m)  
\( q_w \) average heat input into the tube (W/m²)  
\( \alpha \) tilt angle of the solar water heater (°)  

**2. Numerical method**

**2.1. Physical model**

The inner tube of the all glass evacuated solar collector tube used in this paper was elliptical tube, and the outer one was circular tube. The general structure of the inner tube cross section of the collector tube was shown in Fig.1. The major axis of the all tubes were 0.047m, the minor axis of the tubes were respectively 0.047m, 0.0376m and 0.0282m. In the other words, the ratios of the minor axis and major axis were respectively 1, 0.8 and 0.6. The length of the collector \( L \) was 1.8m, the inner diameter of tank was 0.36m, and the tilt angle of the solar collector was 45°. The thermophysical property of the working fluid, water, was shown in Table 1. The properties between them get from line interpolation.

<table>
<thead>
<tr>
<th>( T ) (K)</th>
<th>( \rho ) (kg/m³)</th>
<th>( c_p ) (J/kg · K)</th>
<th>( k ) (W/m · K)</th>
<th>( \mu ) (kg/m · s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>273</td>
<td>999.9</td>
<td>4212</td>
<td>0.551</td>
<td>0.001788</td>
</tr>
<tr>
<td>283</td>
<td>999.7</td>
<td>4191</td>
<td>0.574</td>
<td>0.001306</td>
</tr>
<tr>
<td>293</td>
<td>998.2</td>
<td>4183</td>
<td>0.599</td>
<td>0.001004</td>
</tr>
</tbody>
</table>
2.2. Data reduction

The data is reduced in the following procedure. Firstly, the total heat transfer rate can be obtained by

\[ Q = m c_p (T_{out} - T_{in}) \]  

(1)

\( T_{in} \) and \( T_{out} \) are calculated using

\[ T = \frac{\sum_{i=1}^{n} T_i \rho_i \vec{u}_i \cdot \vec{A}_i}{\sum_{i=1}^{n} \rho_i \vec{u}_i \cdot \vec{A}_i} \]

(2)

where \( T_i, \rho_i, \vec{u}_i, \vec{A}_i \) are respectively temperature, density, velocity vector and area vector of area element \( i \) which on the inlet or outlet of the tank.

The heat transfer coefficient is determined by

\[ h = \frac{Q}{A(T_w - T_b)} \]

(3)

\[ T_b = \frac{T_{in} + T_{out}}{2} \]

(4)

The \( Nu \) is calculated from

\[ Nu = \frac{hd}{k} \]

(5)
2.3. Numerical model

Some assumptions are used to simplify the problem: (1) the radiation intensity is almost uniform on the top half of the tube, and the bottom half is thermally insulated, (2) the tank has good heat-retaining performance, the heat loss through the insulation layer is zero, (3) the tank is positioned horizontally and all the walls are hydraulically, (4) there is no heat exchange between the evacuated collector tube and outside surrounding due to its good vacuum quality, (5) the tank is full of water and no air, (6) the flow in the whole computational domain is laminar. The numerical model used in this paper is a single tube connected to tank, the whole computational domain as shown in Fig. 2. For simplicity, the model considered the inner tube and the outer tube was neglected. In addition, the heat transformed from the radiation replaced by the uniform heat input.

2.4. Mesh design

Grid generation is a key issue in numerical simulation as it governs the stability, economy and accuracy of the predictions. The meshed geometry of the tube cross sections are shown in Fig. 3. Considering the steeper gradient near the wall, finer mesh was meshed near the wall. A mesh independence test was carried out by comparison of $Nu$ for different mesh sizes. Three grid systems with about 870000, 1510000 and 2690000 nodes are adopted to calculate a baseline case in which $b/a=1$, $T=313K$. The Nusselt numbers of the baseline case are shown in Fig. 4, which demonstrates that the difference between the calculated results of 1510000 and 2690000 nodes is very small. Therefore, the grid system with 1510000 nodes is adopted for the following calculations.
2.5. Governing equations

The problem under consideration is assumed to be three-dimensional, laminar and steady. Equations of continuity, momentum and energy for the fluid flow are given below in a tensor form,

Continuity equation:
\[
\frac{\partial \left( \rho u_i \right)}{\partial x_i} = 0
\]  

(6)

Momentum equation:
\[
\frac{\partial \left( \rho u_i u_j \right)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) - \frac{\partial p}{\partial x_i} + \rho g_i
\]

(7)

Energy equation:
\[
\frac{\partial}{\partial x_i} \left( \rho u_i C_p T - k \frac{\partial T}{\partial x_i} \right) = 0
\]

(8)

2.6. Boundary conditions and solution scheme

At the computational domain of inlet and outlet, periodical condition is imposed, which indicates that the velocity, pressure and temperature gradient repeat themselves in the domain as follows:

\[
u_i (\tilde{r}) = u_i (\tilde{r} + \tilde{L}) = u_i (\tilde{r} + 2\tilde{L}) = ...
\]

(9)

\[
\Delta p = p(\tilde{r}) - p(\tilde{r} + \tilde{L}) = p(\tilde{r} + \tilde{L}) - p(\tilde{r} + 2\tilde{L}) = ...
\]

(10)

\[
\sigma = \frac{\frac{T(\tilde{r} + \tilde{L}) - T(\tilde{r})}{L}}{\frac{T(\tilde{r} + 2\tilde{L}) - T(\tilde{r} + \tilde{L})}{L}} = ...
\]

(11)

where \( \tilde{r}, \tilde{L}, \Delta p \) and \( \sigma \) are respectively position vector, length vector of a period, pressure drop of a period and temperature of a period. The mass flow rate through the inlet and outlet is 0.02kg/s.

No slip condition is applied on all the walls, that is, the velocity magnitude near the wall is zero:

\[
u_i = 0
\]

(12)

Uniform heat flux condition is imposed on the top half of the tube wall:

\[
q = q_w
\]

(13)

where \( q_w = 750W \/ \text{m}^2 \). All the other walls are thermally insulated.

In the present study, the governing equations were solved with the finite volume approach. The second order
upwind scheme was used to discrete momentum and energy equation. A Body Force Weighted interpolation method was imposed for pressure. The coupling between the pressure and velocity field was established using SIMPLE algorithm. The solution convergence is achieved by a criterion for the normalized residuals below $10^{-6}$ for energy and $10^{-3}$ for other equations.

To validate the accuracy of the numerical solutions, the predicted results in this paper are compared with the experimental and numerical results of Morrison [6] under similar condition. From Fig. 5 it is clearly seen that the deviation between the results is very limited. Therefore, the present numerical simulations have reasonable accuracy.

3. Results and discussions

3.1. Flow structure

![Fig. 6 (a) Velocity counters and (b) temperature counters of the cross sections](image)

Fig. 6 shows the velocity and temperature counters of the tube cross sections which $L/3$ away from the tanks (the initial temperature is 293K). It can be seen that, as heated by the top half of the tube wall, the water near the top half of the tube wall flow toward the tank driven by the body force due to the density decrease. At the same time, the water near the bottom half of the tube wall flow downward driven by gravity due to its larger density. For this reason, the velocity of the water near the tube wall is higher than other regions. Furthermore, the velocity of the water near the top half of the tube wall is higher than the water near the bottom half of the tube wall due to its higher temperature. Since the water near the top and bottom half of the tube wall flow in opposite directions, a shear layer is formed between them. And the shear layer has the smallest velocity. It can also be seen that all the cross sections have similar temperature distributions. However, the velocity distributions of the cross sections are different from each other. From Fig. 6 (a) it can be seen that the velocity decrease with the decrease in the ratio of major axis and minor axis of the elliptical cross section, it will induce the reduction of circulation rate which is not conducive to heat transfer.

Fig. 7 depicts the variation of mean velocity magnitudes of eighteen cross sections, namely, $L'/L = 0, 0.06, 0.11, 0.17, 0.22$ until $0.94$, where $L'$ is the distance of the cross section away from the tank. From Fig. 7, it can be seen that the mean velocity magnitude near the tank is smaller than the region range from $0.06L$ to $0.4L$. The explanation for this phenomenon is that, the upward hot water is cooled by the downward cold water and it will lead the hot
water to flow in the opposite direction due to the smaller density, at the same time, the cold water is heated by the hot water, and the cold water will flow upward due to the increment in density, all these can lead to the generation of short current flow. The velocity magnitude in the domain range from 0.06$L$ to 0.4$L$ is larger because of the higher temperature. The mean velocity magnitude is become smaller from 0.4$L$ to the bottom of the tube. This is because the cold water is heated in the process of flowing to the bottom of the tube, and it is carried off by the upward hot flow before reaching the tube bottom. Another feature can be seen from Fig. 7 is that the mean velocity magnitude increase with increment in the ratio of major axis and minor axis of the tube cross section, which indicates excellent agreement with Fig. 6.

3.2. Heat transfer

Fig. 8 presents the variation in the Nusselt number of the three solar water heaters at different initial temperatures. It can be seen that the value of Nusselt number increase with increment in initial temperature. This is because the higher temperature leads to a higher circulation rate, which makes heat transfer better. The Nusselt numbers of the solar water heaters with $b/a=1$, 0.8 and 0.6 are respectively increase by 2.21, 1.98 and 1.80 times for the initial temperature ranging from 273K to 313K. Moreover, it can also be seen that, the larger the value of $b/a$, the better heat transfer is, which is agree with Fig. 6 and 7. Over the range investigated, the mean Nusselt number of the solar water heater with $b/a=1$ is respectively 59% and 19% larger than the solar water heaters with $b/a=0.6$ and 0.8. Consequently, although the solar water heater with elliptical collector tube is ahead of the normal solar water heater in some ways, but, as to flow and heat transfer performance, the normal solar water heater do better than it.

4. Conclusion

The flow and heat transfer performance of solar water heaters for different initial temperatures (ranging from 273K to 313K) with elliptical collector tubes are evaluated and compared with normal solar water heater using numerical simulation. The predicted results match fairly well with experimental data. Results indicate that, the temperature distributions of all the tube cross sections are similar, but the velocity profiles of them are much dissimilar. The fluid velocity near the wall and decrease with decrease in the ratio of the cross section major and minor axis, which induces the reduction in the circulation rate through the collector tubes, which is not conducive to heat transfer. The value of Nusselt number increase with increment in initial temperature. The larger the value of $b/a$, the better heat transfer is. Over the range investigated, the mean Nusselt number of solar water heater with $b/a=1$are
respectively 59% and 19% larger than the solar water heaters with \( h/a = 0.6 \) and 0.8 for the temperature ranging from 273K to 313K.

References


