Renewable Energy 122 (2018) 1–16

Contents lists available at ScienceDirect

Renewable Energy

journal homepage: www.elsevier.com/locate/renene

Application of nanofluid to improve the thermal performance of horizontal spiral coil utilized in solar ponds: Geometric study

Erfan Khodabandeh [a], Mohammad Reza Safaei [b, c, *], Soheil Akbari [d], Omid Ali Akbari [e], Abdullah A.A.A. Alrashed [f]

[a] *Mechanical Engineering Department, Amirkabir University of Technology (Tehran Polytechnic), 424 Hafez Avenue, P.O. Box 15875-4413, Tehran, Iran*
[b] *Division of Computational Physics, Institute for Computational Science, Ton Duc Thang University, Ho Chi Minh City, Vietnam*
[c] *Faculty of Electrical and Electronics Engineering, Ton Duc Thang University, Ho Chi Minh City, Vietnam*
[d] *Department of Chemical Engineering, Iran University of Science and Technology (IUST), Iran Narmak, 16846-13114, Tehran, Iran*
[e] *Young Researchers and Elite Club, Khomeinishahr Branch, Islamic Azad University, Khomeinishahr, Iran*
[f] *Department of Automotive and Marine Engineering Technology, College of Technological Studies, The Public Authority for Applied Education and Training, Kuwait*

A R T I C L E I N F O

Article history:
Received 30 September 2017
Received in revised form
7 January 2018
Accepted 9 January 2018

Keywords:
Solar pond
Spiral coil
Water - graphene nanoplatelet/platinum hybrid nanofluid
CFD analysis

A B S T R A C T

Today, one of the most common methods for heat extraction from the solar ponds is using spiral piping system. In this study, effects of nanofluid concentrations and different cross-sections of tube on thermal performance of horizontal spiral-coil in laminar fluid flow are investigated numerically. Water-graphene nanoplatelet/platinum hybrid nanofluid with 0.02, 0.06 and 0.10% volume concentration has used as working fluid. Simulations are performed for different mass flow rates between 0.0005 and 0.005 kg/s. Different shapes including rectangle, elliptic, trapezoid and circle are selected as tube cross-sections. Uniform temperature and velocity distributions with several mass flow rates are applied to geometry at inlet and constant wall temperature as boundary conditions. The results show that variations of average Nusselt number in lower mass flow rates is not dependent to the shape of flow cross-section. By increasing the nanoparticles concentration, the highest Nusselt number belongs to tube with elliptical cross-section. Surfaces with angular corners create greater velocity variations in comparison with surfaces with curved corners and this behavior leads to higher pressure loss as well as more pumping power. Also, by increasing mass flow rate, the heat transfer between hot surfaces and cooling fluid is enhanced. The highest and lowest values of outlet temperature are reported for fluid with highest solid nanoparticle volume concentration and distilled water, respectively. Among the considered mass flow rates in $\varphi = 0.06$ and 0.10%, mass flow rate of 0.002 kg/s has the highest thermo-fluid efficiency.

1. Introduction

By increasing the concerns about environmental problems such as water and air pollution, climate change and most importantly global warming, there is an essential need to replace the fossil fuels with renewable energy source in the long term [1]. According to the amount of energy consumption, fossil fuels including natural gas, coal and oil will be evacuated in most countries and also generate high carbon dioxide emissions. Therefore, using renewable energy source will be played a significant role in the upcoming decades [2]. Among the different options, solar energy seems to gain most popularity, due to its cleanliness and especially abundance compared to other renewable energy sources [3].

Solar pond is a typical thermal energy storage system which is consisted of three layers: the upper convective zone (UCZ), the non-convective zone (NCZ), and the lower convective zone (LCZ) (see Fig. 1(a)) [4]. In order to accumulate the heat in the body of solar pond, salt is mixed with water to generate salinity gradient. The existing heat in the solar pond has been conventionally extracted from LCZ using two methods: withdrawing the hot brine and extracting its thermal energy in external heat exchanger [5] or extracting thermal energy by circulating cold fluid in a tube placed in bottom and lateral of solar pond, especially in LCZ [6]. The temperature in three layers has been distributed differently because of slat concentration; where the LCZ and UCZ have been

* Corresponding author. Division of Computational Physics, Institute for Computational Science, Ton Duc Thang University, Ho Chi Minh City, Vietnam.
E-mail address: cfd_safaei@tdt.edu.vn (M.R. Safaei).

https://doi.org/10.1016/j.renene.2018.01.023
0960-1481/

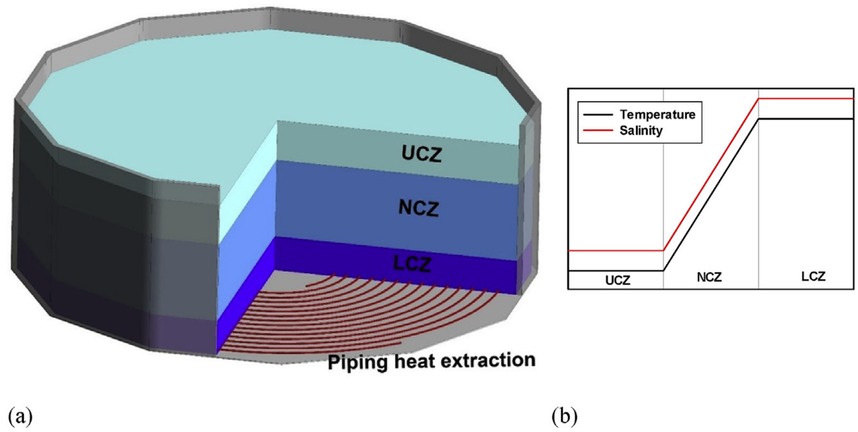

Fig. 1. (a) proposed solar pond regions & (b) behavior of temperature and salinity concentration field in these regions [4].

experienced most and least temperature, respectively (see Fig. 1(b)). In this regard, piping in LCZ or withdrawing the hot brine from this zone seems to be the best choice for maximum performance of heat extraction. In first heat extraction method, the high cost of repair and maintenance is the most important disadvantage because of high tendency of brine-sediment in external heat exchangers. For this reason, piping system for circulating cold water in bottom and lateral of solar pond becomes a common tool.

Coil tubes including spiral and helical tubes are widely used in solar pond heat extraction system, due to creating secondary flow which is obtained by centrifugal force and causes heat-mass transfer and cross-sectional mixing improvement. A comprehensive review of heat-mass transfer and flow characteristics in spiral coil is presented by Naphon and Wongwises [7]. Based on their report, most of previous researches in this area have been performed through circular pipes. A few studies have been carried out on non-circular cross section pipes using CFD methods. Sasmito et al. [8] conducted the numerical simulation of laminar heat transfer and fluid flow in spiral ducts with rectangular, square, triangular, trapezoidal, circular and half circular cross sections, under constant heat flux as well as wall temperature. Their study showed that the triangular and rectangular cross sections have higher performance from the point of heat transfer rate and drop pressure, compared to other cross sections. Effects of other geometry parameters such as curvature ratios, cone angle and helical pitch on hydrodynamic and thermal characteristic of spiral tubes have been numerically studied by Ke et al. [9].

Recently, due to thermo-physical properties limitations of conventional fluids, researchers have been focused on new fluids by dispersing solid particles in nano-dimension to carrying fluids to enhance heat transfer capability. Such fluids are called nanofluids [10–13].

Several scholars have concluded that using nanofluids instead of pure fluid can effectively enhance the thermal conductivity of fluid which consequently enhances the heat transfer performance, especially in equipment with low efficiency [14–16]. For instance, Sasmito et al. [17] studied the laminar nanofluid flow in square cross section of in-plane spiral, helical spiral, conical spiral and straight tubes. Their results demonstrated an impressive effect of nanoparticles usage on heat transfer performance. However, other parameters such as pumping power and pressure drop tend to deteriorate thermal performance. A list of recent studies on using nanofluid in spiral and helical heat exchangers are tabulated in Table 1 with details [18–23].

A review on the relevant literature shows that although some scholars have studied on nanofluid flow in spiral tubes, there is no investigation on the effect of geometrical parameters. In the current study, the hydrothermal attributes and energy efficacy of the water– graphene nanoplatelet/platinum hybrid nanofluid flow have been numerically studied in a horizontal spiral tube with four cross-sections used in bottom of solar ponds. The effect of geometry, Reynolds number and nanoparticles concentration are studied and discussed. The outcomes of this study can be useful for engineers whom work on design more efficient solar ponds.

2. Problem definition and mathematical method

2.1. Geometry

Solar pond technology is one of the relatively simple techniques for using thermal energy of the Sun. Based on Fig. 1(a), in these ponds; water is divided in three regions with different salinities. In the surface layer (UCZ), the fluid has the highest convection heat transfer with the environment while temperature and salinity is the lowest in this region, according to the (Fig. 1(b)).

In the middle region (NCZ), density of water increases with depth in pseudo-linear until reaching its maximum value. Due to increased density, there is not any possibility of convective motion in this region. Similar to NCZ zone in Fig. 1(b), heat transfer is happened through conduction which is a slow process. In the lower layer (LCZ) which is the densest zone of solar pond, the saline water density is almost uniform and close to saturation. Therefore, in this region, heat extraction happens only through convective motion in a weak manner. It is noticed that in LCZ zone, the heat of water is stored in the lower layer and if significant thermal loss does not happen through pond walls, temperature of lower region of pond gradually increases. The heat extracting is carried out by circulating cold fluid in a tube placed in bottom and lateral of solar pond (see Fig. 2(a)). In this paper and based on Fig. 2(b), three concentric spiral tubes are used to extract heat and utilize the absorbed heat from the last layer (LCZ). Regarding this, the total length of coil is constant and placed in the horizontal plane. Temperature of outer surface of the coil is considered equal to 308 K. The inlet section of coil has the constant inlet temperature of 293 K. In this study, simulations are done for four different mass flow rates of 0.0005, 0.002, 0.0035, 0.005 kg/s. To increase thermal efficiency of cooling fluid in this study, nanofluid comprised of hybrid nanoparticles composed of graphene nanoplatelet/platinum as soluble solid nanoparticles in the base fluid are used. Nanofluids are studied in volume concentration of 0.00, 0.02 and 0.10%. Four different geometries including circular, elliptical, trapezoidal and rectangular are studied as tube cross sections shapes (see Fig. 2(c)). In this study, the width of sections for all four geometries is equal. Geometrical dimensions of considered sections are presented in