Desalination and Water Treatment
www.deswater.com

doi:10.5004/dwt.2019.24150

159 (2019) 200–213
August

Computational fluid dynamic as a tool for solar still performance analysis and design development: a review

A.E. Kabeel[a,*], Emad M.S. El-Said[b], Mohamed Abdulaziz[c]

[a]*Mechanical Power Engineering Department, Faculty of Engineering, Tanta University, Tanta, Egypt, email: kabeel6@yahoo.com, kabeel6@hotmail.com (A.E. Kabeel)*
[b]*Mechanical Engineering Department, Faculty of Engineering, Fayoum University, Fayoum, Egypt*
[c]*Institute of Ship Technology, Ocean Engineering and Transport Systems (ISMT), University of Duisburg-Essen, Duisburg, Germany*

Received 15 September 2018; Accepted 17 March 2019

ABSTRACT

Computational fluid dynamic (CFD) is a simulation tool which utilizes the applied and computational mathematics for fluid flow regimes modeling to predict the heat, mass and momentum transfer behavior. In recent years, the solution's accuracy of the CFD simulations became within the acceptable range demonstrating that CFD is a valid tool for performance analysis and design development for many thermal-hydraulic systems. This encourages researchers to use it in the modeling of evaporation and condensation phenomena or two-phase flow with a vapor-liquid phase change process and mass and heat transports through the interface simulation which is one of the urgent difficulties in the solar stills development and analysis. The aim of this is to introduce a comprehensive review work of the highlights that have been made through the recent years for most important studies about the CFD approach as a tool for performance prediction and analysis in addition to design improvement and development for solar stills. This study is supported with a compact synopsis in the form of, computational domain (two or three dimensional), CFD software utilized, operating and geometrical parameter range, simplified assumptions and the better CFD results.

Keywords: Solarstill; CFD; Development; Performance

1. Introduction

Solar still (SS) been considered as an alternative desalination device for utilizing solar thermal energy to provide the isolated or remote areas by fresh water [1,2]. SS, due to its design simplicity, is inexpensive compared with conventional desalination methods, which are rare and costly, which make it more generally utilized distillation devices of saline water can give fresh water to these isolated areas at small expenses. SS is an ecologically agreeable energy process that utilizes a renewable energy source to produce fresh water from the salty water by distillation process. A SS system has many favorable features over other desalination systems such as simple design,low fixed and running costs [3].

In a SS, the system is loaded with saline water, which heated using thermal solar power to increase the water temperature to evaporation point. The resultant water vapor is isolated from salt and any contaminants or impurities and then cooled and condensed on the inclined glass cover. Water droplets stream toward a trough accumulation channel and are put away [4].

Many reviews gave profitable summaries with broad reference indices of the historical design, development and performance augmentation of a SS a critical desalination system [3–6]. Edalatpour et al. [7] exhibited the most recent numerical investigations on different types of SSs, including single slope, double slope, multi-effect, tubular. Elango et al. [8] performed a critical review of thermal models carried out for different kinds of SSs and modifications done to augment their performance throughout the years. Sharshir et al. [9] talked about various theoretical methodologies

---

*Corresponding author.

which have been utilized to assess the thermal performance and exergy analysis of SSs. Tsilingiris [10] introduced a considerable measure of consolidated theoretical and experimental studies done towards better comprehension of the interrelated heat and mass transport mechanisms in SSs.

As seen in the above literature, there are many experimental and theoretical studies have been done to assess the SS performance, however, very few efforts have been made about the investigation of the CFD approach yet which may be due to flow regime complication and computation process restrictions. With the advancement of the physical parts of the computer system in addition to numerical strategy, utilizations of CFD become able to do important studies in the SS field.

Nevertheless, in this paper, we will present a comprehensive review of the most important studies made through the recent years about the CFD approach as a tool for performance analysis and development for solar stills. The essential goals of this paper are:

- Introduce a survey of the CFD utilization in the studies related to solar stills.
- Assess the CFD ability to simulate the mass and heat transports through the two-phase interface inside the solar still cavity.

2. Prediction and analysis methods

A SS involves the physics of fluid flow, heat and mass transfer, which can be solved and analyzed utilizing experimental, theoretical or analytical and numerical or computational approach (i.e. CFD).

2.1. Experimental approach

The generality of reliable data about a physical system is regularly given by practical measurements. An experimental approach, involving full-scale model can be utilized to predict how the actual model in distinguishable reduplicates would implement under similar conditions. Due to the restrictive costly and regularly unbelievable, an alternative then is to carry out the experiments on small scale models. Moreover, the small-scale models don't generally simulate all the characteristics of the full-scale one; habitually, important characteristics. This also decrements the suitability of the test results. In the last, it must be known that there are a number of serious difficulties in the measurement process in many situations, and that the measuring devices are not without errors [11].

2.2. Theoretical approach

A theoretical modeling can produce results as opposed to those of a real actual mode. For the physical processes of intrigue, the mathematical model fundamentally comprises of an arrangement of differential equations. If the methods of classical mathematics were to be utilized for solving these equations, there would be little hope of predicting many phenomena of practical interest. The simplifying assumptions are utilized in the theoretical modeling to make the problems easier to solve [12].

2.3. Computational approach

Computational fluid dynamics or CFD is the analysis of systems, including fluid flow, heat transfer and related phenomena by utilizing the computer simulation and solving mathematical equations with the assistance of numerical investigation. The governing equations of the fluid flow are the continuity (conservation of mass), the momentum, and the energy (conservation of energy) equations which form a system of non-linear partial differential equations.

Due to the presence of non-linear terms, CFD is a solving method used discretization process which carried out by using a set of algebraic equations instead the differential equations governing the fluid flow. The commonly utilized discretization methods in CFD method are:

- Finite difference method (FDM),
- Finite volume method (FVM),
- Finite element method,
- Boundary element method.

The CFD technique has many unparalleled merits over experimental approaches to the design of various fluid systems such as:

- Essential decreasing in required time and cost,
- Capability for investigating systems when experiments are hard or unattainable to carry out (e.g., huge systems)
- Capability for investigating systems under dangerous conditions (e.g., studies for safety and incident purposes)
- Capability for giving practical boundless results details.

So, CFD is considered a useful tool in a SS design as well as in the optimization. CFD uses a very simple technique to perform the numerical solutions such as pressure distribution, temperature variations, flow parameters in a short time with low cost in comparison with experimental work [13]. There are a number of software in view of CFD codes have been carried out, few of them are: CFX, FLUENT, PHOENICS, FLOVENT, CFDRC-Esi and STAR-CD.

3. Numerical modeling assumptions

There are many of simplifying assumptions were successively considered on the basis of the just-described detailed model. These assumptions help in saving the calculation time or/and reduce the solution complexity level of the physical problem. The simplifying assumptions impacted in the whole way of the numerical solution such as:

- **Domain description:** the description of the numerical domain starts by deciding the domain geometry and dimensions which related to the still type and design in addition to studied model scale according to the experimental or other scales related to some considerations [11].
- **Mathematical formulation:** the mathematical formulation and governing equations in this section are introduced in view of the essential physical equations.

For the thermo-fluid processes of the SS numerical domain, there are some assumption such as:

- Domain dimensions dependency; two-dimensional or three-dimensional.

- Flow behavior; laminar or turbulent.
- Time dependency consideration; steady or unsteady.
- Fluid idealization behavior; ideal gas or real gas.
- Thermodynamic and physical property values for gas, liquid and solid media; constant or fixed.
- Viscous dissipation or hydraulic friction; ignored or considered.
- Gas and water mixture homogeneity in the still cavity; homogeneous or heterogeneous (two phases).

4. Discussion about evaporation and condensation modeling

While CFD additionally holds an incredible guarantee for multi-phase flows, getting accurate solutions is significantly more difficult, not on account of each of the phases must be dealt with independently, but, in addition, various new and hard factors are presented [14]. In this section we will present a comprehensive discussion in view of a current literature aim to abuse past published numerical studies which were done under an extensive variety of operating conditions and number of simplified assumptions towards better molding of the coupled heat and mass transport mechanisms in SSs.

Dunkle [15] was the pioneer to introduce an entire mathematical formularization and a basic theoretical model for the forecast of heat and mass transfer processes in SSs. He assumed that the working fluid was nearby to saturation conditions, the appropriate thermo-physical properties associated with the pertinent heat and mass transport processes in SSs. This analysis depended on the description of the natural convection heat transfer in the SS cavity in view of the familiar dimensionless correlation $\mathrm{Nu} = 0.075\ \mathrm{Ra}^{1/3}$, for upward heat flow in horizontal spaces [16]. However, Dunkle's model has some constraints [15]: (1) It is autonomous of cavity volume (2) It was proposed in view of experimental data for temperature range between 55 and 70°C (3) The mean temperature contrast amongst water and glass is 11°C, (4) The slope of the glass cover was small (10°) (5) It was originally carried out for natural convection of air without evaporation [17]. Shawaqfeh and Farid [17] demonstrated that the Dunkle's model overpredicted the evaporation rate of water. They proposed two empirical relationships, in view of bulk motion and Chilton–Colburn analogy and demonstrated that, with a precise appreciation of plate absorptance, glass transmittance, and wind loss, the precision of their model is superior to Dunkle's model. Rheinlander [18] numerically built up a substitutional model for the solution of the heat, mass and momentum transfer equations in the SS cavity and obtained results which were fruitful contrasted and before work by Cooper [19] and Kumar and Tiwari [20]. An ensuing simplified analysis, which shows up more than once in the literature was also introduced by Malik et al. [21], in view of similar simplified assumptions by Dunkle, prompting fundamental heat and mass transfer relationships, which were likewise then obtained utilizing Lewis relation. A generous number of theoretical and experimental studies also was done during the last years, which have unequivocally added to the best comprehension of the different coupled heat and mass transfer processes in SSs. The idea of solar fractionation was introduced and the investigation of the saline water layer depth impacts on heat and mass transfer conditions happening in passive and active SSs has likewise been studied [22–25].

As of late, there have been many studies gave to numerical simulations in view of the volume of fluid (VOF) technique in different conditions [26]. The VOF model is a surface-tracking method applied to a fixed Eulerian mesh. This strategy is published firstly by Hirt and Nichols [27] and later on by Rider et al. [28]. It is intended for at least two immiscible fluids where the position of the interface between the fluids is of interest. In the VOF model, a single set of momentum equations is shared by the fluids, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. As the VOF strategy provides the possibility of tracking immiscible interfaces. In a VOF strategy, the volume and fractions are utilized to reconstruct a geometrical approximation to the interface with the single line interface construction method, Noh and Woodward [29] or the piecewise linear interface construction with the upwind scheme [30].

In this study, many parameters describing the conduct of the water vapor inside SSs cavity were analyzed in light of numerical simulations utilizing VOF strategy, for example, the condensation process, flow shapes, rising trajectory, and humid air velocity.

5. Numerical models and methods

5.1. Models in view of developmental codes

Rheinlander [31] utilized the FDM to solve the governing equations in two dimensional model of a single basin, double slope (SBDS) type SS in unsteady state condition. Also, he utilized the k-ε model to solve the transport equation for the kinematic eddy (turbulent) viscosity of the fluid flow. He compared the computed values of the mass-transfer rate with those, measured by Cooper [32] in an experimental model of a SS with the same dimensions. He showed a good agreement between the numerical and experimental results with maximum error of 25%. Djebedjian and Abou Rayan [33] introduced a numerical investigation in view of the FDM on the performance prediction and augmentation in the single basin single slope (SBSS) type SS design utilizing a mathematical model in view of the time-averaged Navier-Stokes equations in steady state, two-dimensional. They have taken into consideration the impact of the variable fluid properties by utilizing a mixture of air and vapor in the SS. They utilized discretization schema with FDM. Their results unmistakably indicated different circulation zones with reverse velocity inside SS cavity. They additionally demonstrate the need to embrace a numerical investigation before SS sizing. Rahbar and Esfahani [34,35] studied the free convection impact in a two dimensional SBSS type SS under steady state, laminar, and incompressible ideal gas conditions utilizing developed CFD code. They acquired a relationship to determine the HTC by convection in view of their numerical results. Their results demonstrated that, for a given aspect ratio, *Ra* directly affects *Nu*. On the other hand, at constant *Ra*, the value of *Nu* increments when the aspect ratio increments. Besides, they found that the greatest HTC is in the zone where flow directed downward from glass to water. They performed a comparison between the numerical, experimental results and the Chilton–Colburn analogy reported by Shawaqfeh and Farid [17]. Juárez et al. [36] investigated numerically the double-diffusive free convection and surface thermal radiation in an inclined space that simulates a SS. They utilized the FVM to solve the governing equations in two dimensional steady state conditions. They displayed the streamlines, isotherms, iso-lines of vapor and water mass flow rate in addition to average Nusselt (*Nu*) and Sherwood (*Sh*) numbers for various angles of the glass cover. Their results demonstrated that surface thermal radiation changes the fluid flow from one-cell to multi-cellular pattern because of the surface thermal radiation increments the velocity close to the walls, as a consequence the average convective Nusselt number, the total *Nu* and the *Sh* were incremented around 25%, 175% and 15%, respectively. The mass flow rate of distillate incremented as aspect ratio and cover angle increment. The results of the average Nu with different buoyancy ratios are compared with that introduced by Béghein et al. [37] and Sezai and Mohamad [38] with maximum error of 2.1%. Rashidi et al. [39] built up a CFD code to optimize the position and size of the segment inside a SBSS type SS. The segment was introduced independently at base surface and glass cover of the still to augment the performance and give adequate pathways to heat exchange and increment the still efficiency. Their optimization procedure was performed to decide the greatest value of the Nu as a response. Their code solved two-dimensional steady equations with laminar assumption utilizing a FVM. They demonstrated that the real optimized parameters for the maximum normalized *Nu* of base introduced segments are X′ = 0.23 and Y′ = 0.18. The results of *Nu* of the present study are compared with the published data by Rahbar et al. [34] with maximum error of 1.87%. Rashidi et al. [40] studied experimentally and numerically the impacts of segments in SS on performance recovery. They utilized their numerical simulation in view of the SIMPLE algorithm and various contours in two dimensional utilizing FVM. They found that the still with segments work at significantly higher temperature contrast amongst the water and the condensing zone particularly for the afternoon hours. In addition, the water production rate increments by introducing the segments in the still cavity. Rashidi and Esfahani [41] utilized the CFD simulation to determine entropy generation locally for the design improvement of a SBSS type SS. They utilized a numerical approach in view of the SIMPLE algorithm to simulate the double diffusive natural convection in the SS and solve the mass, momentum, energy, and concentration equations in two dimensional utilizing FVM. They studied the impacts of aspect ratio and temperatures of glass cover and water for various types of entropy generation containing frictional, diffusive, and thermal entropy generations. Their results demonstrated that an increment in various kinds of entropy generation was happening with an increment in the glass and water temperatures. In addition, the still with higher aspect ratio makes higher values of entropy generation than that of still with smaller aspect ratio. The results of Nu of the present study are compared with the published data by Rahbar et al. [35] with maximum error of 1.87%.

Table 1 presents a summary of the previous computational studies in view of developmental CFD codes.

5.2. Models in view of CFD commercial codes

Palacio and Fernandez [42] carried out a numerical analysis to assess the technical feasibility of a single stage SS with a cover slope with a varied angle from 24° to 60°

Table 1

Summary of previous computational studies based on developmental CFD codes

Author	Two phase modelling method	Computational domain	Model description	Discretization schema
Rheinlander [31]	Heat and mass transfer by molecular diffusion and buoyant convection, and the transport of momentum, heat and vapor in cavity, where the buoyant driving force is dominant.	SBDS type SS	Two dimensional, unsteady state, turbulent model.	FDM
Djebedjian and AbouRayan [33]	Evaporation of the water is not included. Mixture of air and water vapour (humid air) exists in the still.	SBSS type SS	Two dimensional, steady state, laminar model.	FDM
Rahbar and Esfahani [34], [35]	Based on solution of mass, momentum, energy, and concentration equations.	SBSS type SS with partition blades	Two dimensional, steady state, laminar model.	FVM
Juárez et al. [36]	Based on solution of mass, momentum, energy, and concentration equations.	Inclined type SS with steady state, laminar model.	Two dimensional, steady state, laminar model.	FVM
Rashidi et al. [39]	Based on solution of mass, momentum, energy, and concentration equations.	SBSS type SS with partition blades	Two dimensional, steady state, laminar model.	FVM
Rashidi et al. [40]	Based on solution of mass, momentum, energy, and concentration equations.	SBSS type SS with partition blades	Two dimensional, steady state, laminar model.	FVM
Rashidi and Esfahani [41]	Based on solution of mass, momentum, energy, and concentration equations.	SBSS type SS	Two dimensional, steady state, laminar model.	FVM

utilizing PHOENICS code. They utilize two dimensional numerical models in steady state with laminar and turbulent conditions of mixture flow. The point of their research was to calculate the still productivity and the relative significance of the heat and mass transfer mechanisms when water diffusion is insignificant and convection overwhelms. They showed that the incorporation of the turbulence equations in the mathematical model had its significant impacts for the cases where the tilt angle took the biggest values. Under these conditions, the computed heat flux was roughly 5% bigger than that calculated for laminar flow conditions. Likewise, their computed results were matched to the measured heat flux at a Prandtl number between 0.01 and 0.03, however, underestimate the heat flux by about 30% at $Pr = 0.1$. Abakr and Ismail [43] utilized FLUENT software package to simulate the condensation process inside one stage of the multi-stage evacuated SS desalination with two dimensional mixture model in transient conditions. They utilized the CFD simulation to consider the impact of the stage height on the still water production. They found that the impact of the characteristic height variation in the still estimated productivity was very strong on the productivity; as the height increments the productivity decrements essentially. Setoodeh and Rahimi [44] carried out a three-dimensional, two-phase model for in SS by utilizing ANSYS-CFX software to simulate evaporation and condensation processes in view of VOF method. They determined the convective and evaporative heat transfer coefficients in view of Dunkle's correlation and Kumar and Tiwari model. They also found that by utilizing new HTCs evaluated for experimental results in CFD simulations, the rate of freshwater production did not vary significantly, but it affected water temperature results and decrements its error in comparison with previous simulation results. Panchal and Shah [45] made a three dimensional, two phase model for evaporation as well as the condensation process in a single slope traditional SS by utilizing ANSYS-CFX software. They compared between the experimental and simulation results with time variation which indicated that the deviations from the experimental measurements are 6% as well as 10.25% for production rate and water temperature respectively. Panchal and Shah [46] introduced a three-dimensional three-phase model of hemispherical SS for studying the evaporation and condensation process utilizing ANSYS CFD. They compared the water temperature and water productivity calculated from numerical solution with the experimental measurements with average errors for production rate and water temperature are 12% as well as 8% respectively. LeFevre [47] and LeFevre et al. [48] modeled and optimized pentahedron-shaped covers for application on a passive SS utilizing three geometries: single slope still with full vertical back and side walls and half-pyramid shaped cover with vertical back wall; cover tilt angle (φ) and (θ) were taken 15°, 30° and 45°. They utilized FLUENT to simulate two dimensional model for Grashof numbers from 4.0×10^3 to 1.0×10^9 (Grashof number range depends on the tilt angle) and predict the convection heat transfer correlations inside for a different operating conditions of a SS. Badusha and Arjunan [49] carried out numerically the condensation and evaporation process in view of liquid water volume fraction method in single slope SS. They solved a three dimensional, two phases, quasi steady-state condition, laminar flow model with FVM by utilizing ANSYS-CFX. The results of water productivity, water temperature, evaporation and convective heat transfer coefficients of the present study are compared with the experimental measurements with average errors for water productivity, water temperature, evaporation and convective are 15%, 2.57%, 5.5% and 3.01% respectively. Gokilavani et al. [50] modeled a conventional solar distillation utilizing three dimensional, two phases, quasi steady-state condition, laminar flow model. Temperature distribution of water and over the glass and salt water evaporation were simulated utilizing ANSYS-CFX. They introduced that the simulation results were similar to experimental results. Shakaib and Khan [51] carried out numerically an analysis of velocity and temperature trends and distribution of shear stress and heat transfer coefficient (HTC) in SBSS utilizing ANSYS-FLUENT in three dimensions with two phases, quasi steady state condition, laminar flow model. The results of Nu of the present study are compared with Nu correlation of Dunkle [15] with 30% error. Kumar [52] studied a multi-phase three dimensional CFD model of a single slope SS. His model simulated the temperatures at different positions inside the still cavity utilizing ANSYS-FLUENT. He utilized the mixture model as a multi-phase model to simulate the evaporation and condensation phenomena in the closed domain inside the SS. Also, he utilized the k-ε model with standard wall function in order to simulate the transport equation for the kinematic eddy (turbulent) viscosity of the fluid flow. The results of water productivity, water, glass cover and vapor temperatures of the present study are compared to the experimental measurements with maximum errors 13.33%, 1.5%, 1.87% and 2.8% respectively. Rahbar et al. [53] and Bafghi et al. [54] studied the ability of a two dimensional CFD simulation in computation of heat and mass transfer in a tubular SS utilizing ANSYS-FLUENT. They demonstrated that the CFD simulation introduced a recirculating area with a clockwise direction inside the still cavity. They performed new relations to estimate water productivity, HTCs and MTCs in the tubular SS which help to estimate water-productivity in different operational conditions. Their results also showed the inverse impact of glass temperature, and direct impact of water temperature on the performance of a tubular SS. Maheswari et al. [55] modeled a single basin, double slope SS utilizing solid works and CFD analysis utilizing ANSYS-CFX. They illustrated that the modeling of phase change and temperature distribution was because evaporation. Their results of the temperature of water calculated and the production rate were compared with experimental results with about 2.63% average error. Taamneh [56] carried out a numerical simulation focused on CFD analysis validation with experimental results utilizing a model of phase change interaction (evaporation-condensation model) inside two identical pyramid-shaped SSs utilizing FLUENT. One was filled with Jordanian zeolite-seawater and the second was filled with seawater only. They utilized a VOF model to simulate the inter phase change through evaporation-condensation between zeolite-water and water vapor inside the two SSs. They investigated the impact of the volume fraction of the zeolite particles ($0 \le \varphi \le 0.05$) on the heat and distillate yield inside the SS. Their study established the utility of utilizing the VOF two phase flow model to provide a reasonable solution for the complicated inter phase mass transfer in a SS. Their results of water productivity were compared with the experimental measurements with average errors 7%. Bait and Ameur [57] undertaken a numerical analysis regarding geometrical dimensions of a multi-stage SS to track multi-physics evaporation and condensation mechanisms indoor the distillation tower in three dimensional utilizing ANSYS-FLUENT software. Their finality was to optimize definitely its thermal performance for the purpose of a local manufacturing. Malaiyappan and Elumalai [58] carried out a three dimensional numerical analysis in view of CFD computations for the experiments conducted with three different single slope SS basin materials (glass, aluminium and galvanized iron) utilizing ANSYS-CFX solver. Their model captured the phase change interactions between the water evaporating into vapor at the liquid-gas interface and the condensing vapor into water droplets on the top glass surface. The comparisons of the experimental and their computational results for the same boundary conditions in CFD as that of the experiment which indicated that the average errors 5.26%. Arya and Sreenath [59] compared the performance of the SS with different temperature for feed water by considering with and without preheating of water in view of three dimensional numerical models utilizing ANSYS-CFX. They also determined instantaneous efficiency of the still in view of numerical results. Panchal and Patel [60] carried out a numerical model in ANSYS-CFX to simulate condensation as well as the evaporation process in SS by making models of the actual dimensions of experimental one. They also compare parameters affected performance and productivity. The results of productivity, outer glass cover, basin water, inner glass cover temperatures of the present study are compared with the experimental measurements with maximum errors for water productivity, outer glass cover, basin water, inner glass cover temperatures are 10%, 9.52%, 9.5%, 10.1% and 8.33% respectively. Khare et al. [61] carried out a multi-phase three-dimensional CFD model of a simple SS for simulation by utilizing ANSYS-FLUENT. Their simulation had been done for transient state to validate the results with experimental data for climate conditions. Their results of basin water and glass temperatures of the present study were compared with the experimental measurements with maximum errors for basin water and glass temperatures are 4.35% and 2.87% respectively. Rashidi et al. [62] proposed a VOF model by utilizing ANSYS-FLUENT investigate the potential of Al_2O_3-water nano-fluid to improve the productivity of a single slope SS. They utilized VOF model to simulate the evaporation and condensation phenomena in the SS for two dimensional model utilizing FVM. In view of their numerical results, the entropy generation was performed to evaluate the system from the second law of thermodynamics viewpoint. They also examined the impacts of the solid volume fraction of nano-fluid on the productivity and entropy generation in the SS. Their numerical results showed that the productivity of SS increased with an increase in the solid volume fraction of nano-particles. The productivity increases about 25% as the solid volume fraction increases in the range of 0%–5% because an augmentation in the average *Nu*. The results of water productivity are compared with the experimental measurements with maximum errors 16%.

Table 2 presents a summary of the aforementioned previous computational studies in view of commercial CFD codes.

6. Discussion about appropriate selection of turbulence model

The literature survey reveals that the SS cavity is thermo-hydraulically system operates under the impact of viscosity. The correct simulation and analysis of mass and heat transfer inside the SS cavity utilizing a CFD approach needs an appropriate selection of turbulence model which is the biggest challenge in CFD modeling. No single turbulence model can be universally applied to all situations. Certain considerations must be taken into account while choosing an appropriate turbulence model. Important considerations in this regard are physically involved in the flow problem, desired level of accuracy and the availability of the computational resources [63]. Selection of a turbulence model depends upon the complexity of the problem and the desired accuracy of the result. Turbulence modeling is the structure and utilization of a model to predict the effects of turbulence by application a computational step. There are number of turbulence models in order to simulate the turbulent flows. Most commonly utilized turbulence models are: Standard k–ε turbulence model, Realizable k–ε turbulence model, Renormalization-group (RNG) k–ε turbulence model, Standard k–ε turbulence model, and Shear Stress Transport (SST) k–ω turbulence model. Wilcox [64] introduced the various strategies and models to simulate turbulence and their applications. So, keeping in your mind the complexity of your problem and the available time for simulations to select the suitable turbulence model for the simulation.

Table 3 presents a summary of turbulence models utilized in numerical modeling of previous computational studies. It is clear from this table that most studies have applied k–ε modelling of turbulence for a mainly buoyant flow is explained elsewhere as well as the numerical treatment of the time dependent iteration process [65].

7. Validation and verification of CFD models

Users and developers of computational simulations today face a critical issue: The confidence in modeling and simulation because different simplifying assumptions and numerical grid generation quality, and analysis of the influence of each one of the predicted results. Verification and validation of computational simulations are the primary methods for qualitative and quantitative comparison of CFD results with data from experiment, analytical solutions, and direct numerical simulations. Briefly, verification is the evaluation of the solution accuracy of a computational model by comparison with known solutions. Validation is the evaluation of the computational simulation accuracy by comparison with experimental results performed only for validation modeling or experimental and analytical results of similar case (in design, geometry and operating conditions) with the numerical domain, available

Table 2
Summary of previous computational studies based on commercial CFD codes

Author	Two phase modelling method	Computational domain	Dimensional approach	Discretization schema
Palacio and Fernandez [42]	Heat and mass transfer by molecular diffusion and buoyant convection, and the transport of momentum, heat and vapor in enclosures, where the buoyant driving force is dominant.	SBDS type SS	Two dimensional, steady state, laminar and turbulent flow model	FVM
Abakr and Ismail [43]	Based on liquid water volume fraction method	Multi-stage, SBSS type SS	Two dimensional, unsteady state, laminar flow model	FVM
Setoodeh and Rahimi [44]	Based on liquid water volume fraction method	SBSS type SS	Three dimensional, two phase, quasi steady state condition, laminar flow model	FVM
Panchal and Shah [45]	Heat and mass transfer by molecular diffusion and buoyant convection, and the transport of momentum, heat and vapor in cavity, where the buoyant driving force is dominant.	SBSS type SS	Three dimensional, two phase, quasi steady state condition, laminar flow model	FVM
Panchal and Shah [46]	Heat and mass transfer by molecular diffusion and buoyant convection, and the transport of momentum, heat and vapor in cavity, where the buoyant driving force is dominant.	Hemispherical SS	Three dimensional, three-phase model, quasi steady state condition, laminar flow model	FVM
LeFevre [47] and LeFevre et al. [48]	Heat and mass transfer by molecular diffusion and buoyant convection, and the transport of momentum, heat and vapor in cavity, where the buoyant driving force is dominant.	Pentahedron-shaped covers SS with three geometries	Two dimensional, two phase, unsteady state, turbulent flow model.	FVM
Badusha and Arjunan [49]	Based on liquid water volume fraction method	SBSS type SS	Three dimensional, two phase, quasi steady state condition, laminar flow model	FVM
Gokilavani et al. [50]	Heat and mass transfer by molecular diffusion and buoyant convection, and the transport of momentum, heat and vapor in cavity, where the buoyant driving force is dominant.	SBSS type SS	Three dimensional, two phase, quasi steady state condition, laminar flow model	FVM
Shakaib and Khan [51]	Heat and mass transfer by molecular diffusion and buoyant convection, and the transport of momentum, heat and vapor in cavity, where the buoyant driving force is dominant.	SBSS type SS	Three dimensional, two phase, quasi steady state condition, laminar flow model	FVM
Kumar [52]	Heat and mass transfer by molecular diffusion and buoyant convection, and the transport of momentum, heat and vapor in cavity, where the buoyant driving force is dominant.	SBSS type SS	Three dimensional, three phase, quasi steady state condition, turbulent flow model	FVM
Rahbar et al. [53] and Bafghi et al. [54]	Heat and mass transfer by molecular diffusion and buoyant convection, and the transport of momentum, heat and vapor in cavity, where the buoyant driving force is dominant.	Single basin tubular SS	Two dimensional, three phase, steady state condition, laminar flow model	FVM
Maheswari et al. [55]	Based on liquid water volume fraction method	SBDS type SS	Three dimensional, two phase, quasi steady state condition, laminar flow model	FVM

(*Continued*)

Table 2 (*Continued*)

Author	Two phase modelling method	Computational domain	Dimensional approach	Discretization schema
Taamneh [56]	Based on liquid water volume fraction method	Pyramid-shaped SS	Three dimensional, two phase, unsteady state condition, laminar flow model	FVM
Bait and Ameur [57]	Not specified	Multi-stage SS	Three dimensional, quasi steady state condition, laminar flow model	FVM
Malaiyappan and Elumalai [58]	Based on liquid water volume fraction method	SBSS type SS	Three dimensional, two phase, quasi steady state condition, laminar flow model	FVM
Arya and Sreenath [59]	Not specified	SBDS type SS	Three dimensional, quasi steady state condition, laminar flow model	FVM
Panchal and Patel [60]	Not specified	SBSS type SS	Three dimensional, two phase, quasi steady state condition, laminar flow model	FVM
Khare et al. [61]	The mixture model for air, liquid water and water vapor system at transient state condition, solves for the mixture momentum equation and prescribes relative velocities to describe the dispersed phases.	SBSS type SS	Three dimensional, three phase quasi steady state condition, turbulent flow model	FVM
Rashidi et al. [62]	Based on liquid water volume fraction method	SBSS type SS	Two dimensional, two phase, steady state, laminar model.	FVM

Table 3
Summary of turbulence models used in numerical modeling of previous computational studies

Author	Turbulence model	Notable CFD analaysis results
Rheinlander [31]	K-ε Model	Accepted results with maximum error of 25%.
Palacio and Fernandez [42]	K-ε Model	Accepted results with maximum error of 30%.
LeFevre [47]	Not specified	Accepted results with less than 37% error
Kumar [52]	K-ε Model	Accepted results with maximum error of 13.33%.
Khare et al. [61]	Not specified	Accepted results with maximum error of 4.35%.

in the literatures. In verification, the relationship of the simulation to the real world is not an issue. Invalidation, the relationship between computation and the experimental data is the issue. Stated differently, verification is primarily a mathematical issue; validation is primarily a physics issue [66]. So, the corresponding validity for the simplified modeling of the heat and mass transport phenomena is a very important effort. In this section we will present most valuable studies which may help us to more trust on computational and numerical tools utilized in SS modeling. It can be seen from the literature that the results achieved by numerical modeling had an agreement with experimental data with maximum error varied from 16% to 17%. There is always cause a discrepancy between the numerical and experimental results because the following reasons:

The discrepancy is created by some experimental factors containing calibrating equipment for lab measurements, experiment accuracy, human errors, missing out some processes, etc.

1. The discrepancy is created by some numerical errors. These numerical errors are created by considering some simplifying assumptions such as:
2. Two dimensional modeling.
3. Adiabatic conditions of SS side walls (zero heat loss) assumption. However, they were not completely adiabatic in the experiments,particularly in high operating conditions.
4. Laminar behavior of the mixture flow. However, the turbulence intensity is affected the SS productivity [42].
5. Table 4 presents a summary of confidence in numerical modeling and simulation methods of previous computational studies.

Table 4
Summary of confidence in numerical modeling and simulation methods of previous computational studies

Author	Confident method	Assessment method	Details
Rheinlander [31]	Validation	Quantitatively comparison	Computed results of the mass-transfer rate are compared with those, measured by Cooper [32] in a laboratory model of a still using similarity laws with maximum error of 25%.
Djebedjian and AbouRayan [33]	Not specified	Not specified	Not specified
Rahbar and Esfahani [34,35]	Validation	Quantitatively comparison	The solution obtained of Nu and water productivity has been compared with the experimental data studied by Shawaqfeh and Farid [17] of a still with the same dimensions with maximum error of 4.8% and 5.25% for Nu and water productivity respectively.
Juárez et al. [36]	Verification	Qualitatively and quantitatively comparison	The results of average Nu with different buoyancy ratios are compared with that presented by Béghein et al. [37] and Sezai and Mohamad [38] with maximum error of 2.1%.
Rashidi et al. [39]	Validation	Quantitatively comparison	The results of Nu of the present study are compared with the published data by Rahbar et al. [34] with maximum error of 1.87%.
Rashidi et al. [40]	Not specified	Not specified	Not specified
Rashidi and Esfahani [41]	Validation	Quantitatively comparison	The results of Nu of the present study are compared with the published data by Rahbar and Esfahani [35] with maximum error of 1.87%.
Palacio and Fernandez [42]	Validation	Quantitatively comparison	The heat flux from numerical solution is compared with the experimental measurements with maximum error of 30% according to Prandtl number value.
Abakr and Ismail [43]	Validation	Quantitatively comparison	The temperature difference in the SS and the water productivity calculated from numerical solution are compared with the experimental measurements with maximum error of 17.4% and 7.14% for temperature difference and water productivity respectively.
Setoodeh and Rahimi [44]	Validation	Quantitatively comparison	The water temperature calculated from numerical solution are compared with the experimental measurements with average error 9.98%.
Panchal and Shah [45]	Validation	Quantitatively comparison	The water temperature, glass cover temperature and water productivity calculated from numerical solution are compared with the experimental measurements with average errors for production rate and water temperature are 6.0% as well as 10.25% respectively.
Panchal and Shah [46]	Validation	Quantitatively comparison	The water temperature and water productivity calculated from numerical solution are compared with the experimental measurements with average errors for production rate and water temperature are 12% as well as 8% respectively.
LeFevre [47] and LeFevre et al. [48]	Validation	Quantitatively comparison	The results of Nu of the present study are compared with various existing Nu correlations such as Dunkle [15], Clark [68], Farid and Shawaqfeh [17], Kumar et al. [69] and Shruti [70] with less than 37% error.
Badusha and Arjunan [49]	Validation	Quantitatively comparison	The results of water productivity, water temperature, evaporation and convective heat transfer coefficients of the present study are compared with the experimental measurements with average errors for water productivity, water temperature, evaporation and convective are 15%, 2.57%, 5.5% and 3.01% respectively.
Gokilavani et al. [50]	Validation	Quantitatively comparison	The results of water productivity and water temperature of the present study were similar to experimental results.
Shakaib and Khan [51]	Validation	Quantitatively comparison	The results of Nu of there study are compared with *Nu* correlation of Dunkle [15] with 30% error.

(*Continued*)

Table 4 (*Continued*)

Author	Confident method	Assessment method	Details
Kumar [52]	Validation	Quantitatively comparison	The results of water productivity, water, glass cover and vapor temperatures of his study are compared to the experimental measurements with maximum errors 13.33%, 1.5%, 1.87% and 2.8% respectively.
Rahbar et al. [53] and Bafghi et al. [54]	Verification	Quantitatively comparison	The water productivity obtained from numerical solution has been compared with the experimental data for a given geometry reported by Islam and Fukuhara [71]. The results show that CFD simulation can predict mass productivity within the range of 15%.
Maheswari et al. [55]	Validation	Quantitatively comparison	The results of water productivity is compared with the experimental measurements with average errors 2.63%.
Taamneh [56]	Validation	Quantitatively comparison	The results of water productivity is compared with the experimental measurements with average errors 7%.
Bait and Ameur [57]	Validation	Quantitatively comparison	The maximum water temperature of multi-stage distillation unit which reached 80.96°C is compared to that reported from Reddy et al. [72] and Shatat and Mahkamov [73] which they have found high temperature between 80°C and 100 °C.
Malaiyappan and Elumalai [58]	Validation	Quantitatively comparison	The results of water productivity is compared with the experimental measurements with average errors 5.26%.
Arya and Sreenath [59]	Not specified	Not specified	Not specified
Panchal and Patel [60]	Validation	Quantitatively comparison	The results of productivity, outer glass cover, vapor, basin water, inner glass cover temperatures of the present study are compared with the experimental measurements with maximum errors for water productivity, outer glass cover, vapor, basin water, inner glass cover temperatures are 10%, 9.52%, 9.5%, 10.1% and 8.33% respectively.
Khare et al. [61]	Validation	Quantitatively comparison	The results of basin water and vapor temperatures of the present study are compared with the experimental measurements with maximum errors for basin water and vapor temperatures are 4.35% and 2.87% respectively.
Rashidi et al. [62]	Validation	Quantitatively comparison	The hourly productivities of the their study are compared with the published data by Rashidi et al. [67] with maximum error of 16%.

8. Studies related to SS thermo-fluid modeling

In this section we review some of the important studies and research that has been accomplished during the past few years and was directly related to the numerical modelling and analysis of thermo-fluid domain in SS and may help us to understand as well as control it. The gas injection method has a direct impact on the boundary layer flow in the vicinity of the surface which led to a significant impact on the heat transfer rate [74]. Hamed et al. [75] studied numerically the augmentation of heat and mass transfer between a gas and liquid phase by injection of various gases through water bed such as air, carbon dioxide and helium. They found that the coefficients of mass transfer improve with the increment of injected gas molecular weight. Ezzat et al. [76] introduced experimental and numerical studies of the impact of injected air bubbles on the HTC through the water flow in a vertical pipe under the influence of uniform heat flux. Their results showed that the impact created by air bubbles plays an important role on heat transfer improvement and temperature trends. They found that averaged Nu augmentation was 33.3% and 23% in numerical and experimental, respectively. El-Said and Abdulaziz [77] studied numerically a solar-based thermo-electric generator utilization for dry regions. Their method was in view of air humidification by injection through saline water bed utilizing two dimensional model and CFDRC commercial CFD code.

Surface vibration will be able to augment the heat transfer rate. This method is very useful in natural convection since it changes the heat rejection mode to forced convection. Also, fluid vibration useful in the heat transfer rate augmentation and more practical than surface vibration since it has no negative impact on the structures of heat exchange devices. This method is very useful for low Reynolds number forced convection flows [78]. Duan et al. [79] numerically investigated the heat transfer augmentation mechanism of planar elastic tube bundle by flow-induced vibration in view of a two-way fluid structure interaction unsteady and three-dimensional model. Results show that the oscillating relative velocity was a crucial factor for heat transfer augmentation by 11.43%. Talebi et al. [80] investigated numerically the impact of upper surface oscillation on natural convection heat transfer in a cylindrical enclosure filled with air is by the FDM. They showed that augmentation of HTC can be up to 175% by inducing ultrasonic waves.

9. Conclusions

This paper provides a comprehensive, up-to-date review in a chronological order on the research progress made in SS performance prediction and analysis in addition to design improvement and development by utilizing numerical simulation as a tool. Finally, some suggestions for future work are introduced. Therefore, the present study cannot only be utilized as the starting point for the researcher interested in the SS cavity numerical simulation, but it also includes recommendations for future studies on this important subject. In view of the review and discussions, the following could be concluded;

1. CFD is a powerful tool to solve the complex fluid, but have a critical drawback related to its solution's accuracy, which only high for the physical models on which they are based.
2. Recent numerical studies on SS have opened a new research direction which complements the traditional experimental approach. Direct numerical simulation of these phenomena requires water-vapor interface tracking and accurate prediction of interface transport processes.
3. The impacts of adding nano-particles to the basin water of SSs need more attention because his significant role in the SS performance augmentation.
4. Usage of CFD method has ability to determine the local entropy generation in the domain.
5. The inclusion of the turbulence equations in the numerical modeling had its major impacts for the cases where the tilt angle of the glass cover took a value about 60°.
6. The turbulence models used in numerical modeling of previous computational studies introduced an accepted result with maximum error from 4.35% to less than 37%.

10. Recommendations for future studies

This study will address new points include some of the suggestions of future research directions, which will be expected to be the research focus in the coming years. Analysis and prediction of SS performance can be done utilizing the following:

1. Further investigation is required in order to study the impact of the different parameters such as the inclination angle of the glass cover, absorptivity of the basin water, gap distance between the glass cover and bottom.
2. In order to generalize the impact of various parameters on the performance of SS, various types of performance correlations can be generated by utilizing the CFD model of SS.
3. Geometry optimization of the SS can be performed by the CFD modeling of SS.
4. CFD analysis of a SS consisting of a separate condenser section can be conducted in order to augment the productivity SS.
5. Complex designs of SS such as still having corrugated basin, wick type SS and still having water flowing over the glass cover can also be modeled by utilizing thin film flows in CFD tools.
6. More improved models can be utilized to accurately simulate the thermo-fluid model in a SS cavity such as Euler's model for multi-phase would provide highly accurate results.
7. A more studies and quantitative model for cases such as phase change materials (PCM) heat storage integration with SS are needed, which provides insight into the temperature, the velocity and the phase distribution and to find a more accurate correlation of its impacts.

8. Impact of salinity on the productivity of SSs should be carried out.
9. Recently, the impacts of air or other gases to the basin water of SSs have been investigated experimentally and theoretically many researchers [81–87]. CFD simulations of different types of SSs where different gases have been injected in the basin water could be studied. This field of study will play an important role in future works, since gas injection applications in thermal systems are developing day by day.
10. The performance augmentation of a SS coupled with vibratory excited have been investigated experimentally and theoretically by Eldalil [88] and [89]. This field of study will have a good potential for future works, since the vibration impact on thermal applications in thermal systems are in continuous evolution.
11. CFD studies should be done on the impacts of internal and external reflectors on the SSs performance.
12. The performance augmentation of a modified still with porous fins has been investigated experimentally and theoretically by Srivastava and Agrawal [90]. This field of study will have a significant effect on SS development of future research activities.
13. A comparative study about the impact of different turbulence model for various operating conditions are needed for more accurate and practical numerical solutions.
14. Further investigations are necessary to explain and minimize the inconsistencies between the computational codes and to identify the best models.

Symbols

Ra — Rayleigh number, *dimensionless*
Nu — Nusselt number, *dimensionless*
Sh — Sherwood number, *dimensionless*

Greek

ϕ — Nano-material fraction, *dimensionless*

Abbreviations

CFD — Computational fluid dynamics
SS — Solar still
VOF — Volume of fluid
RNG — renormalization-group
SST — Shear stress transport
FDM — Finite difference method
FVM — Finite volume method
HTC — Heat transfer coefficient
MTC — Mass transfer coefficient

References

[1] A.E. Kabeel, E.M.S. El-Said, Technological aspects of advancement in low capacity solar thermal desalination units, Int. J. Sustain. Energy, 32(5) (2013) 315–332.

[2] A.E. Kabeel, E.M. S. El-Said, Development strategies and solar thermal energy utilization for water desalination systems in remote regions: a review, Desal. Water Treat., 52(22–24) (2014) 4053–4070.

[3] A.E. Kabeel, Z.M. Omara, F.A. Essa, A.S. Abdullah, Solar still with condenser – A detailed review, Renew. Sustain. Energy Rev., 59 (2016) 839–857.

[4] D.D.W. Rufuss, S. Iniyan, L. Suganthi, P.A. Davies, Solar stills: A comprehensive review of designs, performance and material advances, Renew. Sustain. Energy Rev., 63 (2016) 464–496.

[5] S.W. Sharshir, N. Yang, G. Peng, A.E. Kabeel, Factors affecting solar stills productivity and improvement techniques: a detailed review, Appl. Thermal Eng., 100 (2016) 267–284.

[6] H. Panchal, I. Mohan, Various methods applied to solar still for augmentation of distillate output, Desalination, 415 (2017) 76–89.

[7] M. Edalatpour, K. Aryana, A. Kianifar, G.N. Tiwari, O. Mahian, S. Wongwises, Solar stills: A review of the latest developments in numerical simulations, Solar Energy, 135 (2016) 897–922.

[8] C. Elango, N. Gunasekaran, K. Sampathkumar, Thermal models of solar still—A comprehensive review, Renew. Sustain. Energy Rev., 47 (2015) 856–911.

[9] S.W. Sharshir, A.H. Elsheikh, G. Peng, N. Yang, M.O.A. El-Samadony, A.E. Kabeel, Thermal performance and exergy analysis of solar stills – A review, Renew. Sustain. Energy Rev., 73 (2017) 521–544.

[10] P.T. Tsilingiris, Analysis of the heat and mass transfer processes in solar stills – The validation of a model, Solar Energy, 83 (2009) 420–431.

[11] S.V. Patankar, Numerical Heat Transfer and Fluid Flow. Hemisphere, New York, 1980.

[12] R.H. Pletcher, J.C. Tannehill, D. Anderson, Computational Fluid Mechanics and Heat Transfer, 3rd ed. (Series in Computational and Physical Processes in Mechanics and Thermal Sciences), CRC press, 2013.

[13] H.K. Versteeg, W. Malalasekera, An introduction to computational fluid dynamics: The finite volume method, Longman Scientific & Technical, 1995.

[14] J.R. Grace, F. Taghipour, Verification and validation of CFD models and dynamic similarity for fluidized beds, Powder Technol., 139 (2004) 99–110.

[15] W.H. McAdams, Heat Transmission, 3rd ed. McGraw-Hill, 1958.

[16] R.V. Dunkle, Solar water distillation: the roof type still and a multiple impact diffusion still. In: ASME Proc. Int. Heat Transfer Conf. Part V, Int. Develop. Heat Transfer, University of Colorado, Boulder Colorado, 1961.

[17] A.T. Shawaqfeh, M.M. Farid, New development in the theory of heat and mass transfer in solar stills, Sol. Energy, 55 (1995) 527–535.

[18] J. Rheinlander, Numerical calculation of heat and mass transfer in solar stills, Solar Energy, 28(2) (1982) 173–179.

[19] P.I. Cooper, Heat and mass transfer within a single impact solar still envelope. Introduced at the First Australian Conference on Heat and Mass Transfer, Melbourne, 1973.

[20] S. Kumar, G.N. Tiwari, Estimation of convective mass transfer in solar distillation systems, Solar Energy, 57(6) (1996) 459–464.

[21] M.A.S. Malik, G.N. Tiwari, A. Kumar, M.S. Sodha, Solar Distillation. Pergamon Press, 1982, Oxford, NY.

[22] R. Tripathi, G.N. Tiwari, Effect of water depth on internal heat and mass transfer for active solar distillation, Desalination, 173 (2005) 187–200.

[23] R. Tripathy, G.N. Tiwari, Performance evaluation of a solar still by using the concept of solar fractionation, Desalination, 169 (2004) 69–80.

[24] R. Tripathy, G.N. Tiwari, Thermal modelling of passive and active solar stills for different depths of water by using the concept of solar fraction, Solar Energy, 80 (2006) 956–967.

[25] A.K. Tiwari, G.N. Tiwari, Effect of water depths on heat and mass transfer in a passive solar still: in summer climatic condition, Desalination, 195 (2006) 78–94.

[26] N.H. Mostafa, B. Djebedjian, E.M.S. El-Said, M. Abou Rayan, Experimental and numerical study of spoiler impact on ship stability: effect of spoiler inclination angle, CFD Lett., 1(1) (2009) 29–42.

[27] C.W. Hirt, B.D. Nichols, Volume of fluid (VOF) method for the dynamics of natural boundaries, J. Comput. Phys., 39 (1981) 201–225.

[28] W.J. Rider, D.B. Kothe, S.J. Mosso, J.H. Cerrutti, J.I. Hochstein, Accurate solution algorithms for incompressible multiphase fluid flows, AIAA Paper 95-0699, 1995.

[29] W.F. Noh, P.R. Woodward, SLIC (Simple Line Interface Method), in A.I. van de Vooren and P.J. Zandbergen, (Editors), Lecture Notes in Physics, 59 (1976) 330–340.

[30] D.B. Kothe, W.J. Rider, S.J. Mosso, J.S. Brock, Volume tracking of interfaces having surface tension in two and three dimensions, AIAA Paper 96-0859, 1996.

[31] J. Rheinlander, Numerical calculation of heat and mass transfer in solar stills, Sol. Energy, 28 (1982) 173–179.

[32] P.I. Cooper, Heat and mass transfer within a single-impact solar still envelope-I. Austral. Conf. on Heat and Mass Transfer, Melbourne, 1973.

[33] B. Djebedjian, M. Abou Rayan, Theoretical investigation on the performance prediction of solar still, Desalination, 128 (2000) 139–145.

[34] N. Rahbar, J.A. Esfahani, Estimation of convective heat transfer coefficient in a single-slope solar still: a numerical study, Desal. Water Treat., 50 (2012) 387–396.

[35] N. Rahbar, J.A. Esfahani, Productivity estimation of a single-slope solar still: Theoretical and numerical analysis, Energy, 49 (2013) 289–297.

[36] R.A. Juárez, G. Álvarez, J. Xamán, I.H. López, Numerical study of conjugate heat and mass transfer in a solar still device, Desalination, 325 (2013) 84 –94.

[37] C. Béghein, F. Haghighat, F. Allard, Numerical study of double-diffusive natural convection in a square cavity, Int. J. Heat Mass Trans., 35 (1992) 833–846.

[38] I. Sezai, A.A. Mohamad, Double diffusive convection in a cubic enclosure with opposing temperature and concentration gradients, Phys. Fluids, 12 (2000) 2210–2223.

[39] S. Rashidi, M. Bovand, J.A. Esfahani, Optimization of partitioning inside a single slope solar still for performance improvement, Desalination, 395 (2016) 79–91.

[40] S. Rashidi, J.A. Esfahani, N. Rahbar, Partitioning of solar still for performance recovery: Experimental and numerical investigations with cost analysis, Solar Energy, 153 (2017) 41–50.

[41] S. Rashidi, J.A. Esfahani, Spatial entropy generation analysis for the design improvement of a single slope solar still, Environ. Progress Sustain. Energy, (2017) 1– 9.

[42] A. Palacio, J.L. Fernandez, Numerical analysis of greenhouse-type solar stills with high inclination, Solar Energy, 50(6) (1993) 469–476.

[43] Y.A. Abakr, A.F. Ismail, Theoretical and experimental investigation of a novel multistage evacuated solar still, J. Solar Energy Eng., 2005 (127) 381–385.

[44] N. Setoodeh, R. Rahimi, A. Ameri, Modeling and determination of heat transfer coefficient in a basin solar still using CFD, Desalination, 268 (2011) 103–110.

[45] H.N. Panchal, P.K. Shah, Modelling and verification of single slope solar still using ANSYS-CFX, Int. J. Energy Environ., 2(6) (2011) 985–998.

[46] H.N. Panchal, P.K. Shah, Experimental and ANSYS CFD simulation analysis of hemispherical solar still, Int. J. Renew. Energy, 8(1) (2013) 1–14.

[47] J.D. LeFevre, Modeling of complex pentahedron solar still covers to optimize distillate, Master thesis, Brigham Young University, USA, 2012.

[48] J. LeFevre, W.J. Bowman, M.R. Jones, Numerical simulation of convection in triangular cavities to predict SS performance, J. Thermophys. Heat Trans., 27(3) (2013) 482–488.

[49] A.R. Badusha, T.V. Arjunan, Performance analysis of single slope solar still, Int. J. Mech. Eng. Rob. Res., (2013) 74–81.

[50] N. Gokilavani, D. Prabhakaran, T. Kannadasan, Experimental studies and CFD modeling on solar distillation system, Int. J. Innov. Res. Sci. Eng. Technol., 3(9) (2014) 15818–15822.

[51] M. Shakaib, M.A. Khan, Modeling of fluid flow and temperature profiles in SSs using CFD, Proceedings of 2015 International Conference on Chemical, Metallurgy and Environmental Engineering (ICMAEE-15), Istanbul (Turkey), (2015) 272–276.

[52] A. Kumar, CFD modeling and validation of a single slope solar still, M.Sc. thesis, Malaviya National Institute of Technology, India, 2015.

[53] N. Rahbar, J.A. Esfahani, E.F. Bafghi, Estimation of convective heat transfer coefficient and water-productivity in a tubular solar still – CFD simulation and theoretical analysis, Solar Energy, 113 (2015) 313–323.

[54] E.F. Baggy, N. Rahbar, J.A. Esfahani, Productivity improvement of a tubular solar still, using CFD simulation, J. Model. Eng., 10 (2013) 37–48.

[55] C.U. Maheswari, B.V. Reddy, A.N. Sree, A.V. Reddy, A.S.P. Reddy, C.R.R. Prasad, B.H.K. Varma, CFD analysis of single basin double slope solar still, Invent. J. Res. Technol. Eng. Manage. (IJRTEM), 1(2) (2016) 01–05.

[56] Y. Taamneh, Influence of Jordanian zeolite on the performance of a solar still: experiments and CFD simulation studies, Water Sci. Technol.: Water Supply, 16(6) (2016) 1700–1709.

[57] O. Bait, M.S. Ameur, Numerical investigation of a multi-stage solar still under Batna climatic conditions: Impact of radiation term on mass and heat energy balances, Energy, 98 (2016) 308–323.

[58] P. Malaiyappan, N. Elumalai, Numerical investigations: basin materials of a single-basin and single-slope solar still, Desal. Water Treat., 57(45) (2016) 21211–21233.

[59] J. Arya, B. Sreenath, Performance study of a solar still using ANSYS, Int. J. Scient. Res. Devel., 4(4) (2016) 1397–1399.

[60] H.N. Panchal, N. Patel, ANSYS CFD and experimental comparison of various parameters of solar still, Int. J. Ambient Energy, (2017) 1–7.

[61] V.R. Khare, A.P. Singh, H. Kumar, R. Khatri, Modelling and performance augmentation of single slope solar still using CFD, Energy Procedia., 109 (2017) 447–455.

[62] S. Rashidi, S. Akara, M. Bovand, R. Ellahi, Volume of fluid model to simulate the nanofluid flow and entropy generation in a single slope solar still, Renew. Energy, 115 (2018) 400–410.

[63] Fluent, Ansys. "Ansys Fluent Theory Guide." ANSYS Inc., USA, 2011.

[64] D.C. Wilcox, Turbulence Modeling for CFD, 2nd ed., 1998.

[65] M.S. Houssain, W. Rodi, Equations for turbulent buoyant flows and their modelling. Rep. SFB 80/T/46, Univ. Karlsruhe.

[66] P.J. Roache, Verification and validation in computational science and engineering, Hermosa Publishers, Albuquerque, NM, 1998.

[67] S. Rashidi, J.A. Esfahani, N. Rahbar, Enhancement of solar still by reticular porous media: Experimental investigation with exergy and economic analysis, Applied Thermal Engineering, in press, 2017.

[68] J.A. Clark, The steady-state performance of a solar still, J. Solar Energy, 44(1) (1990) 43.

[69] S. Kumar, G.N. Tiwari, H.N. Singh, Estimation of convective mass transfer in solar distillation system, Solar Energy, 57(6) (1996) 459.

[70] A. Shruti, Computer Based Thermal Modeling of an Advanced Solar Distillation System: An Experimental Study. Diss. IIT Delhi, New Delhi, 1999.

[71] K. Islam, T. Fukuhara, Production analysis of a tubular solar still, Doboku Gakkai Ronbunshuu B, 63 (2007) 108–119.

[72] K.S. Reddy, K.R. Kumar, T.S. O'Donovan, T.K. Mallick, Performance analysis of an evacuated multiestage solar water desalination system, Desalination, 288 (2012) 80–92.

[73] M.I.M. Shatat, K. Mahkamov, Determination of rational design parameters of a multiestage solar water desalination still using transient mathematical modelling, Renew Energy, 35 (2010) 52–61.

[74] A.E. Bergles, Recent developments in enhanced heat transfer, Heat Mass Trans., 47 (2011) 1001–1008.

[75] M.H. Hamed, A.E. Kabeel, E.M.S. El-Said, Enhancement of heat and mass transfer performance on humidification tower using injection of different carrier gases into water bed, Appl. Thermal Eng., 111 (2017) 455–476.

[76] A.W. Ezzat, N.N. Abdullah, S.L. Ghashim, Effect of air bubbles on heat transfer coefficient in turbulent convection flow, J. Eng., 23(1) (2017) 8–28.

[77] E.M.S. El-Said, M. Abdulaziz, Thermo-electric solar-based freshwater generator for drinking needs in dry regions: A numerical study, ARWADEX11, 11th Water Desalination Conference in Arab Countries, Cairo, Egypt, 2017.

[78] D.D Ganji, A. Malvandi, Heat Transfer Augmentation Using Nanofluid Flow in Microchannels – Simulation of Heat and Mass Transfer, Elsevier, 2016.

[79] D. Duan, P. Ge, W. Bi, J. Ji, Numerical investigation on the heat transfer augmentation mechanism of planar elastic tube bundle by flow-induced vibration, Int. J. Thermal Sci., 112(16) (2017) 450–459.

[80] M. Talebi, M. Setareh, R.H. Abardeh, M.S. Avval, Numerical investigation of natural convection heat transfer in a cylindrical enclosure due to ultrasonic vibrations, Ultrasonics, 76 (2017) 52–62.

[81] G.P. Narayan, R.K. McGovern, J.H. Lienhard, S.M. Zubair, Helium as a carrier gas in humidification dehumidification desalination systems, Proc. ASME 2011 International Mechanical Engineering Congress & Exposition, Denver, Colorado, USA, IMECE2011-62875.

[82] S.A. El-Agouza, M. Abugderah, Experimental analysis of humidification process by air passing through seawater, Energy Convers. Manage., 49 (2008) 3698–3703.

[83] M.K. Abu Arabi, K.V. Reddy, Performance evaluation of desalination processes in view of the humidification/dehumidification cycle with different carrier gases, Desalination, 156 (2003) 281–293.

[84] H.M. Abd-ur-Rehman, F.A. Al-Sulaiman Mathematical Modeling of Bubbler Humidifier for Humidification-Dehumidification (HDH) Water Desalination System, Proceedings of the 1st International Conference on Mechanical and Transportation Engineering, Kuala Lumpur, Malaysia, 2015.

[85] A. Khalil, S.A. El-Agouz, Y.A.F. El-Samadony, Ahmed Abdo, Solar water desalination using an air bubble column humidifier, Desalination, 372 (2015) 7–16.

[86] W. Yuanxin, B. Chen One, M.H. Al-Dahhan, Predictions of radial gas hold up profiles in bubble column reactors, Chem. Eng. Sci., 56 (2001) 1207–1210.

[87] B. Moshtari, E.G. Babakhani, J.S. Moghaddas, Experimental study of gas hold-up and bubble behavior in gas-liquid bubble column, Petrol. Coal, 51(1) (2009) 22–28.

[88] K.M.S. Eldalil, Improving the performance of SS using vibratory harmonic effect, Desalination, 251 (2010) 3–11.

[89] K.M.S. Eldalil, New concept for improving SS performance by using vibratory harmonic effect theoretical analysis, Part-2, Thirteenth International Water Technology Conference, IWTC 13 2009, Hurghada, Egypt.

[90] P.K. Srivastava, S.K. Agrawal, Winter and summer performance of single sloped basin type SS integrated with extended porous fins, Desalination, 319 (2013) 73–78.