International Journal of Scientific Engineering and Technology ISSN : 2277-1581
Volume No. 6, Special Issue 1, PP : 23-25 22-23 Dec. 2017

A Review of CFD Methodology used for Solar Devices

Himanshu Pandya
Dept. of Mechanical Engg, Techno India NJR Institute of Technology, Udaipur, Rajasthan, India
Corresponding Author Email: erhimanshupandya@gmail.com

Abstract : Tremendous need of renewable energy development is very much felt in every part of the globe and sun energy is a prime source of renewable energy, many different techniques and devices are created to harness this vast amount of clean energy source as an alternative to the fossil fuels. Experiments on the physical models and prototypes has been done to create a higher efficiency device but they are time-consuming and costly processes and with the development in the field of computer, scientist and inventors are equipped with the powerful technique of numerical or computational fluid dynamics (CFD) simulation. With the help of numerical or CFD simulation various parameters and effects are check prior to building a physical system with a good accuracy. This article discusses the computational approach used by various researchers in developing various solar systems such as solar water heater, solar air heater and solar still. And also, about the advantages and limitations of the computational approach. In this review it is found out that CFD results are validates with the experimental results and various parametric study can be done more efficiently.CFD is a powerful tool of the analysis of the physical problem.

Keywords: Solar energy, solar water heater, solar air heater, solar still, CFD.

1. Introduction

The sun is a major source of renewable free energy (i.e. solar energy) for our planet Earth. With the modernization new technologies are being employed to generate energy from harvested solar energy. These approaches have already been proven and are widely practiced throughout the globe as renewable alternatives to conventional nonrenewable energy sources[1].Also use of solar energy for domestic and industrial heating purposes has also increased. With the increase in demand in solar energy due to the following reasons: -

1. Solar energy is free and available for most of the year for the major part of the globe.
2. It is pollution free and also helps in carbon reduction in the world [2].
3. It is Available in abundance such that it can full fill all the world demand if its harvesting and supplying technologies are readily available [3].

It's now a great challenge for engineers, researchers, scientist and inventors to create such devices which can easily and efficiently harness, store, and utilize this immense source of pollution free energy. This required great amount to research has to be done, which is also happing in more advance ways then it was before. The analysis of solar devices was carried out in the literature using three approaches as stated below

1. Experimental
2. Theoretical (mathematical)
3. Computational approach.

In this paper, main objective is to highlight the latest work done in Computational approach for solar devices with brief introduction and comparison with Experimental approach.

2.Methods or approaches for solar devices analysis

A solar device involves the physics of fluid and heat flow. For analyzing the solar devices its thermal and hydraulic performance has to be predicted.To evaluate its thermal and hydraulic performance, solar devices can be analyzed using experimental, theoretical or analytical and numerical or computational approach (i.e. CFD).

2.1 Experimental approach

In experimental approach, a prototype (actual dimensions or scaled model) of a solar device is manufactured and on it experiments are performed. Thermal and hydraulic parameters like temperature, pressure, flow etc are measured to evaluate the performance of the device. Various factors like cost of manufacturing the device, time required for research, experimental facility and measurement devices, apart from these human error, measurement error and atmospheric condition play a critical role in the accuracy of the research data from the experimental method [4].

2.2 Analytical approach

In theoretical or analytical approach, mathematical equations which are generally partial differential equations represent the governing equation of the physics are used. These are solved using various mathematical analytical approach but the solving a higher order complex equation is a difficult task. To solve these complex equations researcher, use various assumption such that equation becomes easy to solve and the results obtained from it matches with the experimental results [5].

2.3 Computational approach

Computational approach is the latest approaches for analysis, it uses numerical solutions to the mathematical governing equations with the help of powerful computational software like ANSYS FLUENT, ANSYS CFX, CFD ++, Open FOAM, GASP CFL 3D, TYPHON etc. In CFD governing equations which are in the form of integrals or the partial derivatives are converted into discretized algebraic which gives the solutions at discrete points. The main three elements of every CFD codes are (i) a pre- processor, (ii) a solver and (iii) a post-processor

2.3.1. Pre-processing

In Pre-processing a 3D model of the object of interest is created, and then this domain is divided into number of smaller domain which is known as grid generation or meshing. Smaller element thus formed from meshing is also known as cell and solution to the governing equation are defined at nodes inside each cell. As the number of cell increases accuracy and cost of computational also increases. Fluid properties and boundary conditions are also stated in this step.

2.3.2 Solver

Governing equations are converted into discrete system of algebraic equation using suitable discretization procedure such as finite difference method (FDM), finite element method (FEM) or finite volume method (FVM). In CFD codes FVM is mostly preferred. Now these algebraic equations are solved by an iterative method.

2.3.3 Post-processing

Post processing is use to examine the results, it includes the visualization tools like contour plots, vectorplots, line and shaded plots, 2D and 3D surface plots etc. and also numerical reporting tools to examine various properties like pressure, temperature, heat transfer coefficient etc. of the system.

2.3.4 Validation

Table 1 Difference between experimental and computational approach [6]

Experimental	**Computational**
Experiments are expensive to setup	Except initial software cost, it is less costly
These are time consuming	Less time consuming as compare to experimental
Modification in setup is difficult	Modification can be done easily
Large number of measuring instruments required	Various tools are available for calculating
Not limited to the complexity of the problem	Availability of mathematical model sets limit to the complex problem
Limited number of experimental data and at limited time period	No such limitation of time and space
But gives real solution to the problem	Solution accuracy depends upon approximation taken during simulations

CFD solution of physical problem must be validated with the experimental data to ensure sufficient accurate description of the reality. In many conditions no experimental data is available; simulation process is carried out with scale model for which experimental results are available and then it is extended for full scale model. Table 1 shows the difference between experimental and computational

3. CFD methodology for solar devices

Bouhal et al.[7] have simulate two different configuration of solar water heater's storage tank, one with the charge inlet and outlet in horizontal direction (Configuration A) and other having charge inlet and outlet in vertical direction (Configuration B). In both the configurations effect of flat plate insidethe tank as an obstacle to flow has been studied. The flat plateis located horizontally in different position in Configuration A and tilted in various angle in Configuration B has been simulated. The following conclusions were drawn from their analysis: (1) CFD framework has been done to evaluate thermal stratification in vertical solar storage. (2) CFD codes are validated with the experimental work done before and agood agreement between them has been found out. (3) The optimum case for configuration A is when two horizontal plates located at the Middle and Top in the tank. (4) The optimum case for configuration B is when plate is tilted 30°.

D.G. Gunjo et al. [8] have carried out experiment and CFD analysis on the novel type solar collector.CFD analysis of a single bent riser tube attached to an absorber plate has been done.

Their results reveal that: (1) CFD results are validated with the experimental setup with low deviation in the results. (2) For the investigated solar collector with 60°C outlet water temperature maximum thermal efficiency of 71% was obtained. (3) The simulation model of the collector gives the outlet water temperature, energy efficiency, absorber plate temperature, and overall heat loss coefficient with maximum error of 9%.

Both experiment and CFD analysis have been carried out by Facao [9] to analyze the flow distribution in solar collectors which is done first time on the varying diameter solar collector which are generally of constant diameter. The main outcomes were: (1)There is good agreement between CFD and experimental results. (2) The header manifold dimension of the outlet must be greater than that of the inlet for better results.

ANSYS CFX was used by H.N. Panchal et al. [10] to simulate the solar still model and validate with the experimental setup. Their results show that: (1) the average deviations from CFD and experimental values for production rate and water temperature are 6.0% and 10.25% respectively. (2) ANSYS CFX is very powerful tool for design, difficulty removal in solar still construction and parameter analysis.

A computational analysis using ANSYS FLUENT has been carried out by Khare et al. [11] on simple solar still The main observations from their study were: (1) Simulation results were found to be in good agreement with the experimental data within the scope of their study. (2) Thermal efficiency of the Solar Still is higher from 16:00 to 17:00 hrs. (3) Rubber is found tobe one of the best basin materials to improve absorption, storage, and evaporation effects. (4)The Solar Still for low water depth has more productivity.

Panchal et al. [12] performed a CFD simulation and experimental analysis of hemispherical solar still. The conclusions drawn from their analysis were: (1) Distillate water errors of 12 % while comparing with actual experimental results. (2) Also, good agreement with experimental data and average error of water temperature is 8 %.

Jin et al. [13] Conducted 3D numerical investigation of solar air heater having multi V-shaped ribs on the absorber plate to study the effect of heat transfer and fluid flow characteristics. The major findings of theirs tudies are listed here – (1) Multi V-shaped ribs promote the fluid mixing between the colder upper channel fluid and the warmer near-bottom-wall fluid by generating the streamwisehelicalvortex flows. (2)Heat transfer wasgreatlyimprovedwhencomparedtoasmoothwall channel.

(3)The maximum values of both the thermo hydraulic performance parameter and average Nusselt number was for Angle of attack of 45°, and an angle of attack of 60° gave the highest value of the friction factor.

Yadav et al. [14] simulated a 3D computational domain of roughened solar air heater with non-uniform mesh using V-shaped perforated blocks as artificial roughness on the absorber plate. Their study revealed following conclusions: (1) for perforated V-shaped blockages average enchantment in the Nusselt number was found to be 33% higher than solid blockages. (2) Friction factor blockages were decreased by 32% of the value as observed in solid blockages.

Singh et al. [15] investigates a CFD using 3D computational domain of solar air heater to study effect of non-uniform and uniform cross section transverse rib on the friction factor and Nusselt number of roughened solar air heater. The major findings of the study were: (1) For Reynolds number above 7000 the Nusselt number for non-uniform cross-section saw-tooth rib was more than uniform cross-section ribs. (2) For the range of Reynolds number investigated the Nusselt number for trapezoidal rib was found to be highest followed by square rib and circular rib. (3) CFD results also found in good agreement with the experimental data.

4.Conclusion

The conclusions which can be derived from this article are presented here:

(1) CFD is a powerful tool for the analysis of the physical problems
(2) With the advance in technology and research more number of analysis with CFD are happing all around the world
(3) With number of advantages over experimental approach many researchers are performing CFD analysis
(4) With growing need of free and clean energy CFD analysis in the field of solar energy is a vast field and lot many work has to be done.
(5) Still there is limited number of research available on CFD analysis on solar devices
(6) For obtaining results with greater accuracy in CFD; mathematical model, boundary conditions, and assumption made during simulation play a great role
(7) But for the final validation of the CFD results experimental data are required.

References

i. *Solar energy: Potential and future prospects EhsanulKabira, Pawan Kumarb, Sandeep Kumarc, Adedeji A. Adelodund, Ki-Hyun Kime,Renewable and Sustainable EnergyReviews 82 (2018) 894–900*

ii. *International Energy Agency. 2DS-hiRen Scenario, Energy Technology Perspectives; 2012.*

iii. *Blaschke T, Biberacher M, Gadocha S, Schardinger I. Energy landscapes: meetingenergy demands and human aspirations. Biomass-Bioenergy 2013;55:3–16.*

iv. *GawandeVB,DhobleAS,ZodpeDB.Effectofroughnessgeometrie sonheat transfer enhancementinsolarthermalsystems—a review.RenewSustain Energy Rev2014;32:347–78.*

v. *Shamardan MM,Norouzi M,KayhaniMH, DeloueiAA.Anexactanalytical solution forconvectiveheattransferinrectangularducts.ZhejiangUniv-SciA 2012; 13(10):768–81*

vi. *A reviewofCFDmethodologyusedinliteratureforpredicting thermo-hydraulicperformanceofaroughenedsolarairheater Vipin B.Gawande , A.S.Dhoble , D.B.Zodpe , SunilChamoli RenewableandSustainableEnergyReviews54(2016)550–605.*

vii. *Numerical modeling and optimization of thermal stratification in solar hotwater storage tanks for domestic applications: CFD studyT. Bouhala, S. Fertahib, Y. Agrouaza,, T. El Rhafiki, T. Kousksou, A. Jamil ,Solar Energy 157 (2017) 441–455.*

viii. *Exergy and energy analysis of a novel type solar collector under steady state condition: Experimental and CFD analysis DawitGudetaGunjo, Pinakeswar Mahanta, PuthuveettilSreedharanRobi* Renewable Energy 114 (2017) 655-669.*

ix. *J. Facao, Optimization of flow distribution in flat plate solar thermal collectorswith riser and header arrangements, Sol. Energy 120 (2015) 104-112.*

x. *Hitesh N Panchal, P. K. (2011). Modelling and verification of single slope solar still using ANSYS CFX. International Journal of Energy and Environment, 2(6), 985-998.*

xi. *Modeling and Performance Enhancement of Single Slope SolarStill using CFD Vaibhav Rai Kharea, Abhay Pratap Singhb*, Hemant Kumarc, Rahul Khatrib Energy Procedia 109 (2017) 447 – 455.*

xii. *Experimental and ANSYS CFD simulation analysis of hemispherical solar still, Hitesh N Panchal, P.K. Shah.International Journal of Renewable Energy, Vol. 8, No. 1, January – June 2013*

xiii. *Dongxu Jin,Manman Zhang, PingWang, ShashaXu. Numerical investigation of heat transfer and fluid flow in a solar air heater duct with multi V-shaped ribs on the absorber plate.Energy2015;89:178–90.*

xiv. *YadavAS,SamantTS,VarshneyL.ACFDbased analysis of solar air heater having V-shaped per forated blocks on absorber plate.IntResJEngTechnol 2015;2(2):822–9.*

xv. *Singh S,SinghB,HansVS,GillRS.CFD(computational fluid dynamics) investigation on Nusseltnumber and friction factor of solarairheaterduct roughened withnon-uniformcross-sectiontransverserib.Energy 2015; 84:509–17.*